Altair

HyperWorks

Altair HyperMesh 2019

User's Guide

altairhyperworks.com

Contents

Intellectual Property Rights Notice

Copyrights, Trademarks, Trade Secrets, Patents & Third Party Software Licenses

Altair HyperMesh 2019 Copyright 1990-2019

The Platform for Innovation™

Note: Pre-release versions of Altair software are provided 'as is', without warranty of any kind. Usage of pre-release versions is strictly limited to non-production purposes.

Altair HyperWorks™ - The Platform for Innovation™

Altair Accelerator™ ©1995-2019; (formerly **NetworkComputer**)

Altair Accelerator Plus™©1995-2019; (formerly WorkloadXelerator)

Altair FlowTracer™ ©1995-2019; (formerly **FlowTracer**)

Altair Allocator™ ©1995-2019; (formerly **LicenseAllocator**)

Altair Monitor™ ©1995-2019; (formerly **LicenseMonitor**)

Altair Hero™ ©1995-2019; (formerly **HERO**)

Altair Software Asset Optimization™ (SAO) ©2007-2019

Note:

Compute Manager™ ©2012-2017 is now part of **Altair Access**

Display Manager™ ©2013-2017 is now part of **Altair Access**

PBS Application Services™ ©2008-2017 is now part of **Altair Access**

PBS Analytics™ ©2008-2017 is now part of **Altair Control**

PBS Desktop™ ©2008-2012 is now part of **Altair Access**, specifically **Altair Access desktop**, which also has **Altair Access web** and **Altair Access mobile**

e-Compute™ ©2000-2010 was replaced by **"Compute Manager"** which is now **Altair Access**

Altair SmartWorks™ - Innovation Intelligence®

Altair SmartCore™ ©2011-2019

Altair SmartEdge™ ©2010-2019

Altair SmartSight™ ©2014-2019

Altair intellectual property rights are protected under U.S. and international laws and treaties. Additionally, Altair software is protected under patent #6,859,792 and other patents pending. All other marks are the property of their respective owners.

ALTAIR ENGINEERING INC. Proprietary and Confidential. Contains Trade Secret Information.

Not for use or disclosure outside of Altair and its licensed clients. Information contained in Altair software shall not be decompiled, disassembled, "unlocked", reverse translated, reverse engineered, or publicly displayed or publicly performed in any manner. Usage of the software is only as explicitly permitted in the end user software license agreement. Copyright notice does not imply publication.

Third party software licenses

AcuConsole contains material licensed from Intelligent Light (www.ilight.com) and used by permission.

Software Security Measures:

Altair Engineering Inc. and its subsidiaries and affiliates reserve the right to embed software security mechanisms in the Software for the purpose of detecting the installation and/or use of illegal copies of the Software. The Software may collect and transmit non-proprietary data about those illegal copies. Data collected will not include any customer data created by or used in connection with the Software and will not be provided to any third party, except as may be required by law or legal process or to enforce our rights with respect to the use of any illegal copies of the Software. By using the Software, each user consents to such detection and collection of data, as well as its transmission and use if an illegal copy of the Software is detected. No steps may be taken to avoid or detect the purpose of any such security mechanisms.

Technical Support

Altair provides comprehensive software support via web FAQs, tutorials, training classes, telephone and e-mail.

Altair Support on the World Wide Web

The Altair web site is a valuable online companion to Altair software. Visit www.altairhyperworks.com for tips and tricks, training course schedules, training/tutorial videos, and other useful information.

Altair Training Classes

Altair training courses provide a hands-on introduction to our products, focusing on overall functionality. Courses are conducted at our main and regional offices or at your facility. If you are interested in training at your facility, please contact your account manager for more details. If you do not know who your account manager is, please send an e-mail to training@altair.com and your account manager will contact you.

Telephone and E-mail

When contacting Altair support, please specify the product and version number you are using along with a detailed description of the problem. Many times, it is very beneficial for the support engineer to know what type of workstation, operating system, RAM, and graphics board you have, so please have that information ready. If you send an e-mail, please specify the workstation type, operating system, RAM, and graphics board information in the e-mail.

To contact an Altair support representative, reference the following table or the information available on the HyperWorks website: www.altairhyperworks.com/ClientCenterHWSupportProduct.aspx.

Location	Telephone	E-mail
Australia	64.9.413.7981	anzsupport@altair.com
Brazil	55.11.3884.0414	br_support@altair.com
Canada	416.447.6463	support@altairengineering.ca
China	86.400.619.6186	support@altair.com.cn
France	33.1.4133.0992	francesupport@altair.com
Germany	49.7031.6208.22	hwsupport@altair.de
India	91.80.6629.4500 1.800.425.0234 (toll free)	support@india.altair.com
Israel		israelsupport@altair.com
Italy	39.800.905.595	support@altairengineering.it

Location	Telephone	E-mail
Japan	81.3.6225.5830	support@altairjp.co.jp
Malaysia		aseansupport@altair.com
Mexico	55.56.58.68.08	mx-support@altair.com
South Africa	27 21 8311500	support@altair.co.za
South Korea	82.70.4050.9200	support@altair.co.kr
Spain	34 910 810 080	support-spain@altair.com
Sweden	46.46.460.2828	support@altair.se
United Kingdom	01926.468.600	support@uk.altair.com
United States	248.614.2425	hwsupport@altair.com

For questions or comments about this help system, send an email to connect@altair.com.

In addition, the following countries have resellers for Altair Engineering: Colombia, Czech Republic, Ecuador, Israel, Russia, Netherlands, Turkey, Poland, Singapore, Vietnam, Indonesia

Official offices with resellers: Canada, China, France, Germany, India, Malaysia, Italy, Japan, Korea, Spain, Taiwan, United Kingdom, USA

See www.altair.com for complete contact information.

Get Started

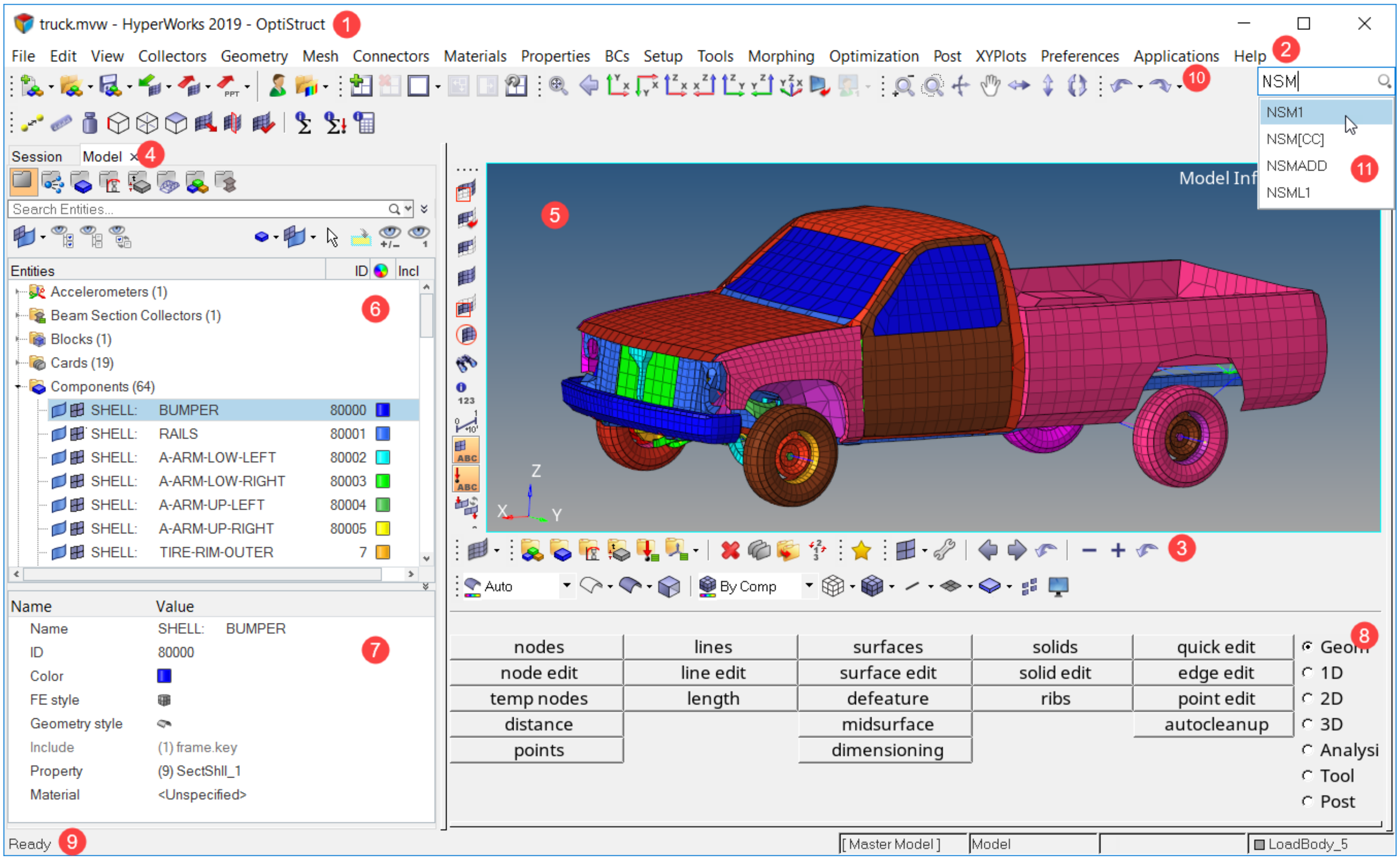
1

Learn the basics and discover the workspace.

This chapter covers the following:

User Interface

Explore the HyperMesh user interface.



1. Title Bar
2. Menu Bar
3. Toolbars
4. Tab Area
5. Modeling Window
6. Browsers
7. Entity Editor
8. Panels
9. Status Bar
10. Undo-Redo
11. Search and Find HyperMesh Tools

1. Title Bar. Indicates which product is active. Because you can switch between clients, clicking on and activating a different window may change the information in the title bar. If the client you have opened supports user profiles, the name of the active loaded user profile is also shown.
2. Menu Bar. Contains pull-down menus that provide access to standard functions such as New, Open, Save and Undo/Redo. This is also where you can access system preferences, and the help.
3. Toolbars. Group of icon buttons that provide access to common tools.
4. Tab Area. Contains browsers, Utility menus for the solver interfacing user profiles, and other functionality not shown in the panel area.
5. Modeling Window. Displays your model, geometry, and plots, and allows you to select entities and manipulate the view of the model in the modeling window.
6. Browsers. Displays view-related functionality in HyperWorks Desktop by listing the parts of a model in a tabular and/or tree-based format, and provides controls inside the table that allow you to alter the display of model parts.
7. Entity Editor. Opens when you select an entity in a browser, and allows you to view and edit entities in a model and correctly setup solver information.
8. Panels. Displays pre-processing and post-processing tools.
9. Status Bar. Displays information pertaining to the currently loaded model as well as descriptions of the pages and panels.
10. Undo-Redo. Undo and redo actions
11. Quick Access Tool. Quickly find and open the tools, panels, and browsers that are available from the menu bar pull-downs or from the Utility Browser, as well as create solver cards.

Title Bar

The title bar displays the name of the Session file (`*.mvw`) that is currently opened, along with the active product and user profile.

The title bar is located at the top, left of the application.

Because you can switch between clients, clicking on and activating a different window may change the information in the title bar. If the client you have opened supports user profiles, then the name of the active loaded user profile is also shown.

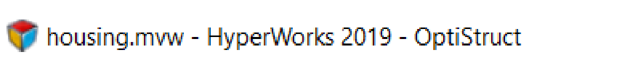


Figure 1: Status Bar

Menu Bar

The menu bar contains pull-down menus that provide access to standard functions such as file management operations, system preferences, and help.

Toolbars

Each toolbar contains a group of icon buttons that provide access to common HyperMesh tools.

Toolbars are dockable, meaning they can be moved and either floated or pinned to a location, allowing you to configure the workspace according to your preferences.

Turn toolbars on and off from the **View** > **Toolbars** menu.

Checks

The Checks toolbar provides access to checks and calculations tools that are commonly used in the model building process.

Turn the Checks toolbar on and off from the **View** > **Toolbars** > **HyperMesh** > **Checks** menu.

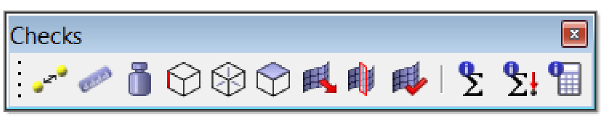


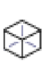
Figure 2:

Opens the Distance Panel.

Opens the Length Panel.

Opens the Mass Calc Panel.

Opens the Edges Panel.



Opens the Features Panel.

Opens the Faces Panel.

Opens the Normals Panel.

Opens the Penetration Panel.

Opens the Check Elems Panel.

Opens the Summary Panel.

Opens the Loads Summary Tool.

Opens the Count Panel.

Collectors

The Collectors toolbar provides access to tools that allow you to create, edit, delete, card edit, organize, and renumber HyperMesh collectors.

Turn the Collectors toolbar on and off from the **View** > **Toolbars** > **HyperMesh** > **Collectors** menu.

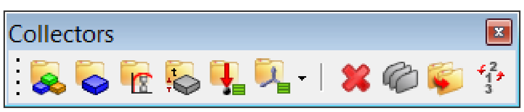


Figure 3:

Opens the Assemblies Panel.

Opens the Component Collectors Panel.

Opens the Material Collectors Panel.

Opens the Property Collectors Panel.

Opens the Load Collectors Panel.

Opens the System Collectors Panel.

Opens the Vector Collectors Panel.

Opens the Beamsection Collectors Panel.

Opens the Multibody Collectors Panel.

Opens the Delete Panel.

Opens the Card Editor Panel.

Opens the Organize Panel.

Opens the Renumber Panel.

Display

The Display toolbar controls what entities HyperMesh displays in the graphics area, primarily by masking entities to hide or display.

Turn the Display toolbar on and off from the **View** > **Toolbars** > **HyperMesh** > **Display** menu.

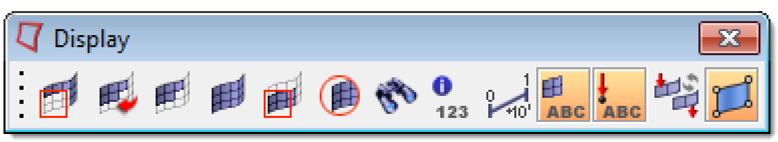


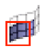
Figure 4:

Opens the Mask Panel.

Left-click: Reverses the mask state of all elements in currently displayed collectors.
Right-click: Reverses the mask state of all entities (elements, loads, and so on) in currently displayed collectors.

Unmasks the row of elements adjacent to the currently displayed ones. If some of the unmasked elements reside in components that are currently not displayed, those components will also be unmasked.

Unmasks all entities (elements, loads, and so on) in the currently displayed collectors.



Left-click: Masks all entities (elements, loads, and so on) located outside of the graphics area but in currently displayed collectors.
Right-click: Unmask all entities (elements, loads, and so on) located outside of the graphics area but in currently displayed collectors.

Opens the Spherical Clipping Panel.

Opens the Find Panel.

Opens the Numbers Panel.

Displays a scale in the lower, right-hand corner of the graphics area, which you can use to measure different parts of your model. The numbers on the scale are dependent upon the dimension of the model and the zoom factor you are currently using in the graphics area.

Switches the display of element handles on/off.

Switches the display of load handles on/off.

Points the load vector toward the load application point (tip), or away from the load application point (tail) when the tip or the tail of the load vector is attached to the load application point.

> **Note:** The direction of the vector does not change when you select this option.

Switches the display of fixed points on/off.

Favorites

The Favorites toolbar allows you to save and access a menu that lists your favorite panels. HyperMesh saves the list of favorite panels and restores it accordingly when you start a new session.

Turn the Favorites toolbar on and off from the **View** > **Toolbars** > **HyperMesh** > **Checks** menu.

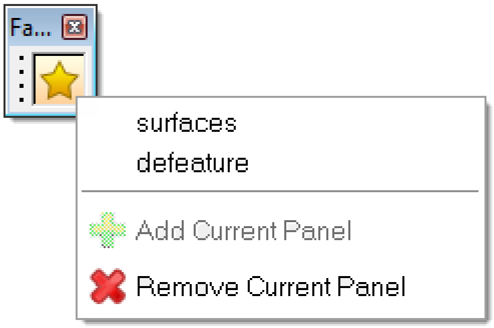


Figure 5:

Patch Checker

The Patch Checker toolbar contains a group of icon buttons that you can use to review quality results, sliver surfaces, elements attached to selected nodes, and so on.

Entities placed on the user mark are used as input. The user mark is populated by selecting the **save** option from advanced entity selections, specific panels that have the save button (such as Check Elems), or via Tcl script using `*marktousermark`.

This tool creates "patches", or local regions, from each input entity. A patch includes only displayed entities. Patches are not created for any input entities that are not displayed. A spherical clipping is then calculated and applied for each patch, with the input entity highlighted and the adjacent entities low lighted. In order to keep the performance high, only the first 500 entities on the user mark are considered.

Turn the Patch Checker toolbar on and off from the **View** > **Toolbars** > **HyperMesh** > **Patch Checker** menu.

Figure 6:

Select the elements entity type.

Select the surfaces entity type.

Select the nodes entity type.

Turn the tool on or off.

Go to the previous patch.

Go to the next patch.

Go back to the first patch.

Decrease the size of the spherical clip.

Increase the size of the spherical clip.

Reset the spherical clip back to its default.

Undo-Redo

The Undo-Redo toolbar provides access undo and redo functionality.

Turn the Undo-Redo toolbar on and off from the **View** > **Toolbars** > **HyperMesh** > **Undo-Redo** menu.

Figure 7:

Visualization

The Visualization toolbar contains a group of icons that you can use to control the display of entities in the graphics area.

Turn the Visualization toolbar on and off from the **View** > **Toolbars** > **HyperMesh** > **Visualization** menu.

Figure 8:

Geometry - Color mode options

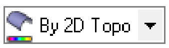
Automatically select a color mode based on the active panel.
You can change display colors in the Options panel, Colors subpanel.

All surfaces are colored based on the assemblies they belong to. Each assembly receives a different color (although models with many assemblies may have colors repeated for more than one assembly). Any surfaces that do not belong to an assembly are colored gray.

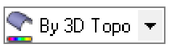
All surfaces are colored based on the parts they belong to. Any surfaces that do not belong to a part receive the color assigned to the master model.

Changes the color of all surfaces and solid faces to the color assigned to the component in which that geometry resides. All surface edges and solid face edges are colored black.

Surfaces are colored gray (2D faces (topo) with surface edges colored by topology: red (free edges), green (shared edges), yellow (t-junctions), or blue (suppressed edges). Solid faces and face edges are colored transparent green (bounding faces) with internal faces colored yellow (full partition faces).



Surfaces are colored gray (2D faces (topo) with surface edges colored by topology: red (free edges), green (shared edges), yellow (t-junctions), or blue (suppressed edges). Solid faces and face edges are colored blue, ignoring solid topology.



Surfaces and surface edges are colored blue, ignoring surface topology. Solid faces and face edges are colored transparent green (bounding faces) with internal faces colored yellow (full partition faces).

Surfaces are colored by component with surface edges colored by topology. Solid faces are colored by component with solid face edges colored by topology.

Surfaces display in wireframe mode, with surface edges colored blue (ignoring topology). Solid faces are colored by mappability: red (not mappable), yellow (1d mappable), or green (3d mappable). Solid face edges are colored by topology.

Geometry - Shade options

Set geometry mode to shaded with surface edges.

Set geometry mode to shaded.

Geometry - Wireframe options

Set geometry to wireframe with surface lines.

Set geometry to wireframe mode.

Opens the Transparency panel.

Mesh - Color mode options

All elements are colored based on the parts they belong to. Any elements that do not belong to a Part receive the color assigned to the master model.

All elements are colored by the color assigned to the component in which that element resides.

All elements are colored by the property assigned to that element. Properties are assigned to elements directly or indirectly. Properties are assigned directly to the element by using the Property > Assign panel. Indirect element properties are inherited from the component in which the element resides; component properties are assigned in the Component > Assign panel. Directly assigned properties override indirect ones. Solvers in group #1 (Radioss (Bulk Data), OptiStruct, Nastran) can support both direct and indirect element property assignment. Solvers in group #2 (Radioss (Block), LS-DYNA) only support indirect element property assignments. Any element without a property is colored gray.

All elements are colored by the material assigned to that element. Materials are assigned to elements differently for solver group #1 and solver group #2; Solver Group #1 (Radioss (Bulk Data), OptiStruct, Nastran) assign materials to properties, and then properties to elements (either directly or indirectly as discussed in Color by Property). Elements with both direct and indirect property assignments use the material associated with the direct element property assignment. Solver group #2 (Radioss (Block), LS-DYNA) assigns materials to elements indirect by assigning materials to the component in which the element resides using the Component > Assign panel. Any element which does not have a material assigned to it, directly or indirectly, will be colored gray.

All elements are colored based on the assemblies they belong to. Each assembly receives a different color (although models with many assemblies may have colors repeated for more than one assembly). Any elements that do not belong to an assembly are colored gray.

All elements are colored by their topology: green (1D), blue (2D), and red (3D).

All elements are colored by their element configuration (mass, reb2, spring, bar, rod, gap tria3, quad4, tetra4, and so on).

Opens the Thickness View, and colors shell elements according to their thickness values. Both element as well as node thicknesses are supported. A thickness legend is shown in the upper-left corner of the graphics area.
Thickness coloring can be combined with 2D Detailed Element Representation

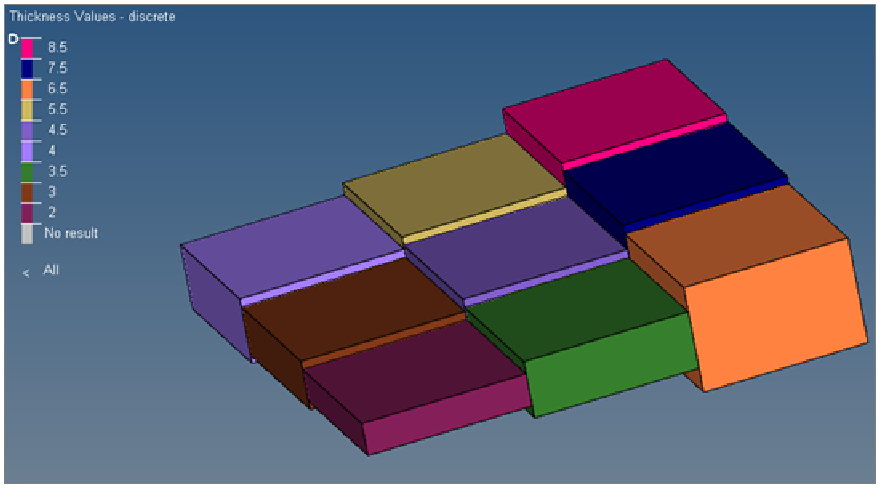


Figure 9: Element Thickness with 2D Detailed Representation

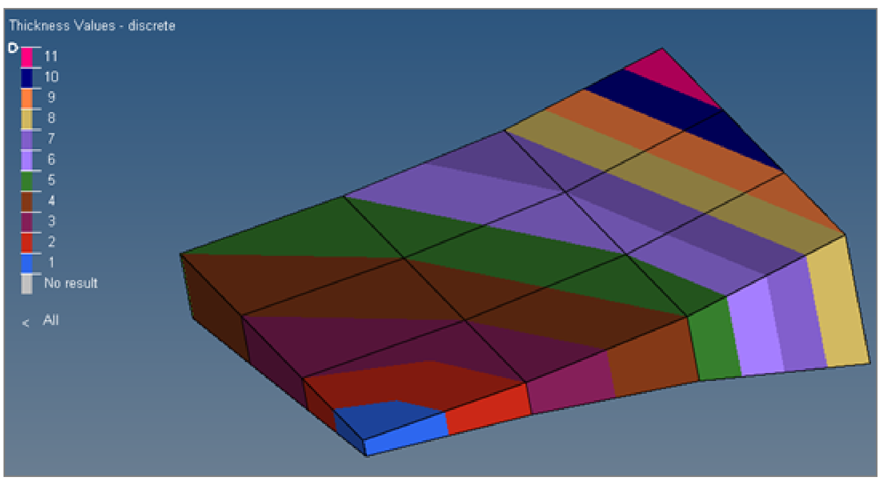


Figure 10: Nodal Thickness with 2D Detailed Representation

Opens the Element Quality View mode in the graphics area.
This permanent mode serves as a useful tool to investigate each specific element criteria, as well as evaluate the overall quality of a mesh.

All elements are colored based on the domains they belong to. A domain is a morphing entity which enables design changes to an existing FE topology. Each domain receives a different color. Any elements that do not belong to a domain are colored gray.

Elements - Shaded options

Set current element visual mode to shaded with mesh lines. Elements are shaded, and surface mesh lines display.

Set current element visual mode to shaded with feature lines. Elements are shaded but have no mesh lines, while feature lines display.

Set current element visual mode to shaded. Elements are shaded, but no lines display.

Elements - Wireframe options

Set the current element visual mode to wireframe (skin only). Internal mesh lines will not display.

Set the current element visual mode to wireframe. Internal and surface mesh lines display.

Set current element visual mode to **transparent with elements and feature lines**. Elements are shaded but transparent, no mesh lines display, but feature lines do.

1D - Element options

Display a more detailed, shaped-based representation for 1D beam elements.

Display both the simple and detailed representations for 1D beam elements.

2D - Element options

Display a simple representation for 2D shell elements.

Display a more detailed, shaped-based representation for 2D shell elements.

Display both the simple and detailed representations for 2D shell elements.

Ply/Composite options

Ply layers are not displayed.

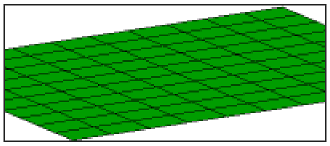


Figure 11:

Plies in a composite material are displayed.

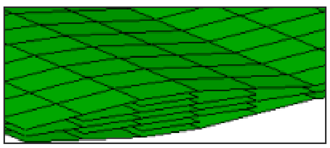


Figure 12:

The exact nature of the display depends on the 2D Element Representation button. See Element and Ply Visualization for details.
For continuum shells the display can be overlaid with the transparent representation of the original continuum shell elements, if **2D Traditional Element Representation** ( ) is turned on.

Display layers with vectors indicating their appropriate ply orientation. Corrected fiber directions are shown if the drape data is available on every element of the ply.

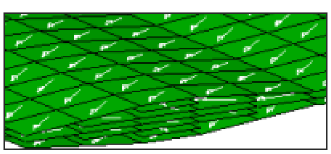


Figure 13:

The exact nature of the display depends on the 2D/3D element visualization button. See Element and Ply Visualization for details.
For continuum shells the display can be overlaid with the transparent representation of the original continuum shell elements, if **2D Traditional Element Representation** ( ) is turned on.

Enables the ply lay-up or stack boundaries to be visualized, which provides an easy way to view ply drop-off. When the stack topology shape is changed, the visualization of the edges is automatically updated. Ply layer geometry edges are always outlined in white, where as FE edges are always outlined in the same color as the ply. FE edges are always outlined with a thicker line compared to geometry edges.

Toggle on/off shrink elements by shrink factor.
Shrink factor can be set from the Options panel, Graphics subpanel.

Opens Visualization Controls tab.

Element and Ply Visualization

The exact visualization of ply layers in a composite material, requires the use of both the Composite Visualization and Element (complexity) Visualization options.

These options work in tandem to determine exactly how composite layers will display in HyperMesh.

Access these options from the Visualization toolbar.

- (1D/2D Element Representation)
- (Composite Visualization)

2D Traditional Element Representation ()

HyperMesh represents composite layers, when visible, as 2D shells:

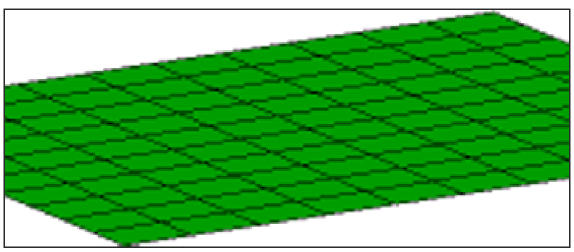


Figure 14: Layers Off

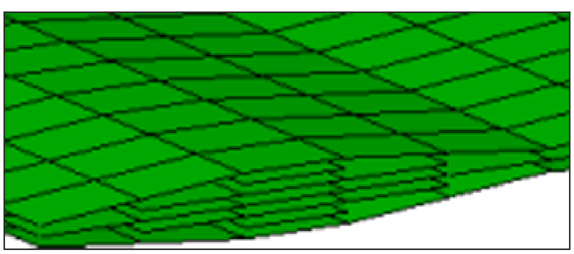


Figure 15: Composite Layers

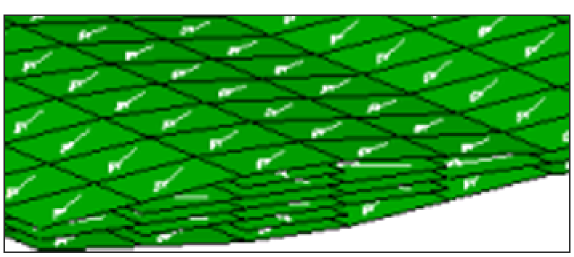


Figure 16: Layers with Fiber Direction

2D Detailed Element Representation ( )

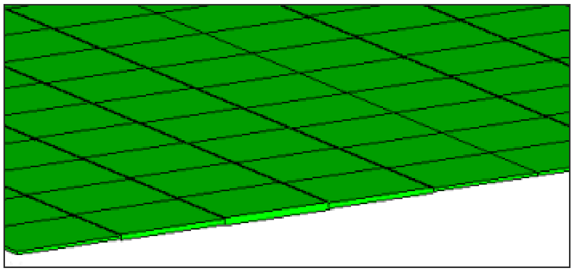


Figure 17: Layers Off

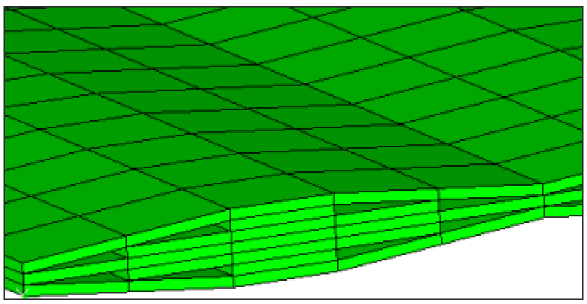


Figure 18: Composite Layers

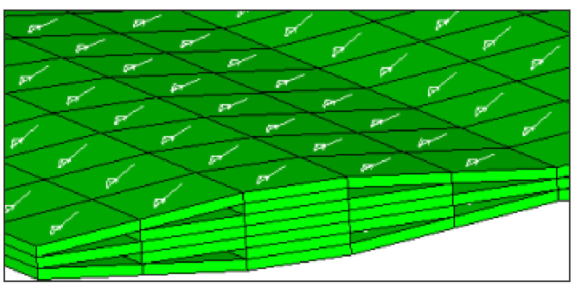


Figure 19: Layers with Fiber Direction

2D Traditional and Detailed Element Representation ( )

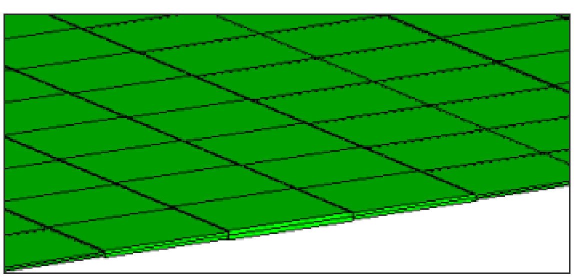


Figure 20: Layers Off

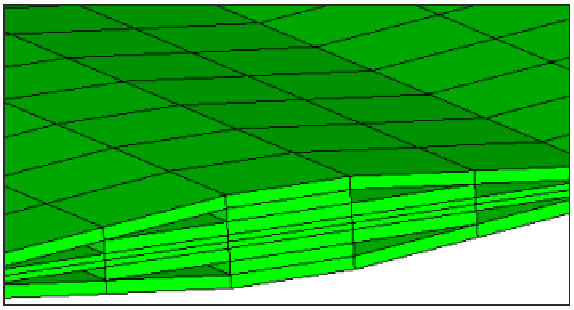


Figure 21: Composite layers

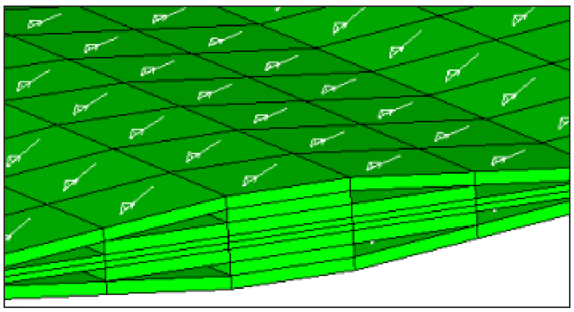


Figure 22: Layers with Fiber Direction

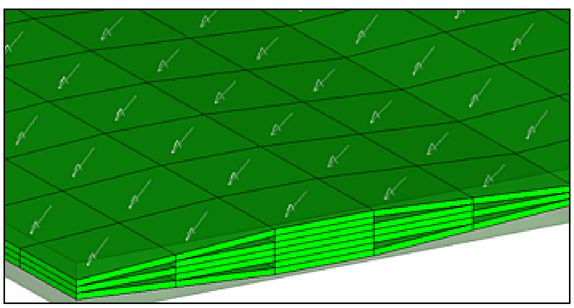


Figure 23: Layers with Fiber Direction for Continuum Shells

Original element geometry is shown in transparent mode, so that differences in elemental versus property thickness can be easily seen.

Tab Area

The tab area organizes browsers, Utility menus for the solver interfacing user profiles, and other functionality not shown in the panel area in different tabs.

The tab area can be turned on or off from the View menu. A check mark indicates that the tab area is currently displayed.

The tab area can be positioned on either the left or right side of the modeling window, both, or turned off completely by selecting **Left**, **Right**, **Expand Both**, or **Collapse Both** from the **View** > **Tab Area** menu.

Click a tab heading to bring it to the forefront.

Modeling Window

The modeling window displays your model, geometry and plots.

The modeling window occupies the middle portion of the application.

You can select entities and manipulate the view of the model in the modeling window.

The global axes, located in the bottom, left of the modeling window, indicates the location of the displayed model. The active background has a thin blue line surrounding it to indicate which window is currently active.

You can customize the color of the global axes and backgrounds in the modeling window using the Color Preferences.

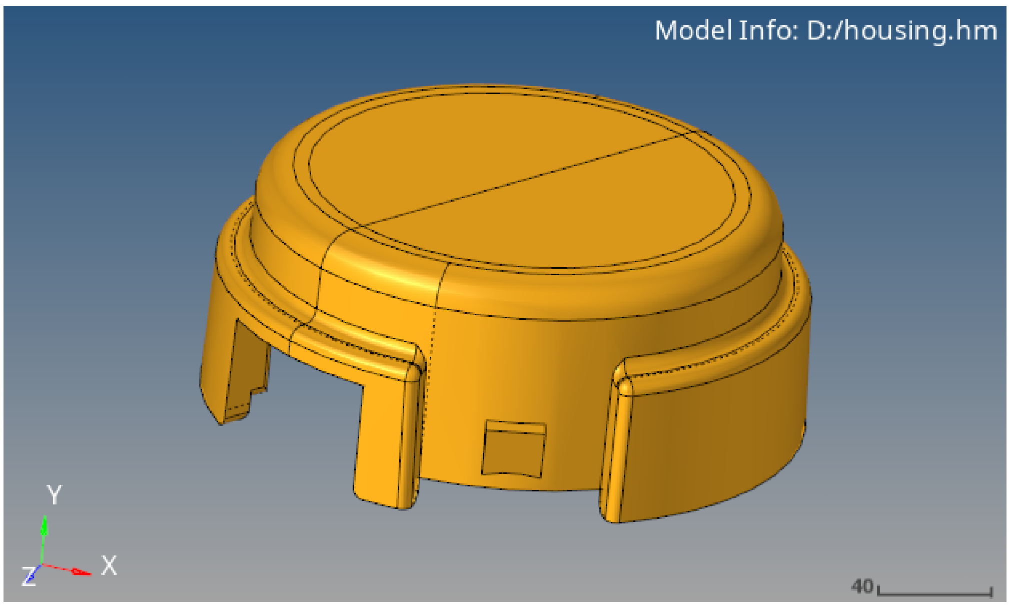


Figure 24: Modeling Window

Status Bar

The status bar displays information pertaining to the currently loaded model as well as descriptions of the pages and panels.

The status bar, located along the very bottom of the application, can be turned on or off from the View menu. A check mark indicates that a toolbar is currently displayed.

Messages

The left side of the status bar displays messages and descriptions of active features in the User Interface.

When you are on one of the main menu pages, not within a panel, a description of the active menu page name, for example, Geometry, displays.

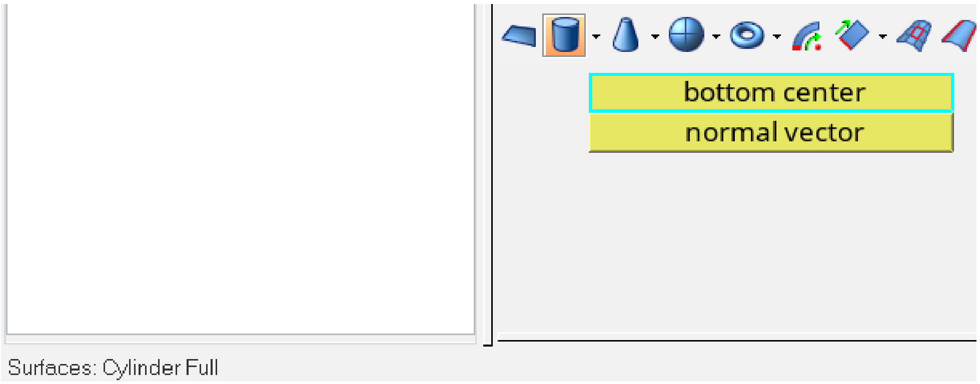


Figure 25: Surfaces Panel, Cylinder Full Subpanel

Panel and subpanel name is displayed in the status bar.

When you are in a panel, the panel title and subpanel title display.

> **Tip:** Holding down the left mouse button on the panel button displays a description of the panel.

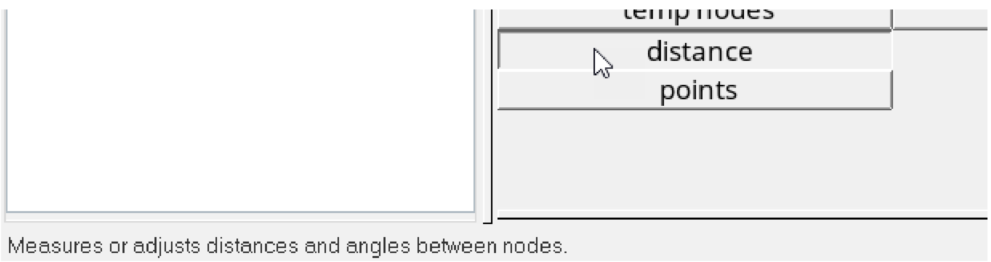


Figure 26: Distance Panel

Description of Distance panel is displayed in the status bar.

Errors messages and miscellaneous messages such as status updates or completed operations also display, and temporarily override the title and status information.

Messages are color-coded.

Red
: Error message.

Green/Gray
: Miscellaneous messages, such as status updates or completed operations. These messages appear in green when using HyperMesh classic dark menu colors and they appear in gray when using Windows light menu colors.

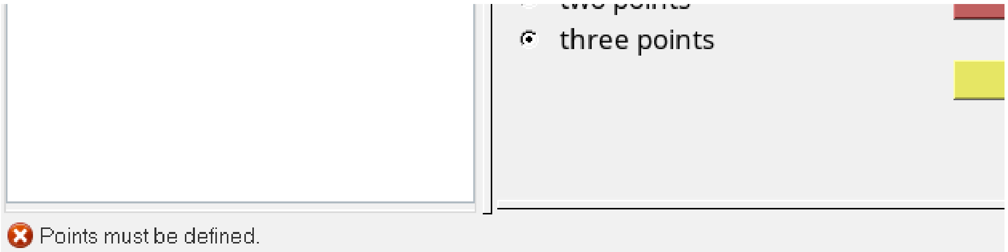


Figure 27: Error Message

Click a mouse button to remove a message from the status bar.

Current Collector

The right side of the status bar displays the current include, component and load collector.

> **Tip:** Quickly change the active collector by clicking the name of the collector in the corresponding box on the status bar to open the appropriate panel which lists the available collectors.

Figure 28: Current Collectors

Undo-Redo

Undo and redo actions performed in HyperMesh.

Not all the actions performed in HyperMesh can be undone or redone. If an operation does not support undo/redo, the undo/redo functionality will be disabled once the action is completed and any previous undo/redo history recorded will be reset. The undo/redo history will also be reset every time a macro or other tcl application is invoked.

> **Note:** The undo/redo functionality is supported by most HyperMesh panels, tools and user interface functionalities.

By default, the number of undo actions is set to 100 and the maximum undo memory (MB) is set to 2500. You can modify both of these settings in the Undo/Redo Settings dialog accessed from the **Preferences** > **Undo/Redo Settings** menu.

Undo/redo history may use a large amount of memory. By default, a threshold of 80% of the total memory limit will not be exceeded. Once the memory threshold is exceeded, all older actions will be automatically purged.

- Undo an action by clicking from the Undo-Redo toolbar, or pressing **Ctrl**+**Z**.

- Redo an action by clicking  from the Undo-Redo toolbar, or pressing **Ctrl**+**Y**.

> **Tip:** Simultaneous actions can be undone/redone by selecting them from the Undo/Redo drop-down menus on the Undo-Redo toolbar.
>
> 
>
>
> *Figure 29:*

Search and Find HyperMesh Tools

Quickly find and open the tools, panels, and browsers that are available from the menu bar pull-downs or from the Utility Browser, as well as create solver cards.

You can only create solver cards with the Quick Access tool in the Radioss, OptiStruct, Abaqus, LS-DYNA, PAM-CRASH 2G and Permas user profiles.

1. Enable the Quick Access tool by pressing **Ctrl** + **F**.

   Once enabled, a search bar opens in the top, right of the application.

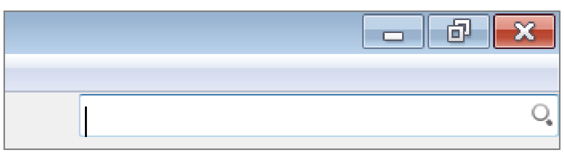
   

   Figure 30:

2. Enter a search string.
   Predicted results appear in a list.
3. Select a search result from the list, or press **Enter** to select the highlighted search result.
4. Exit the Quick Access tool by pressing **Esc** or clicking anywhere outside of the tool.

Selecting a tool, panel, or browser from the list opens the respective tool.

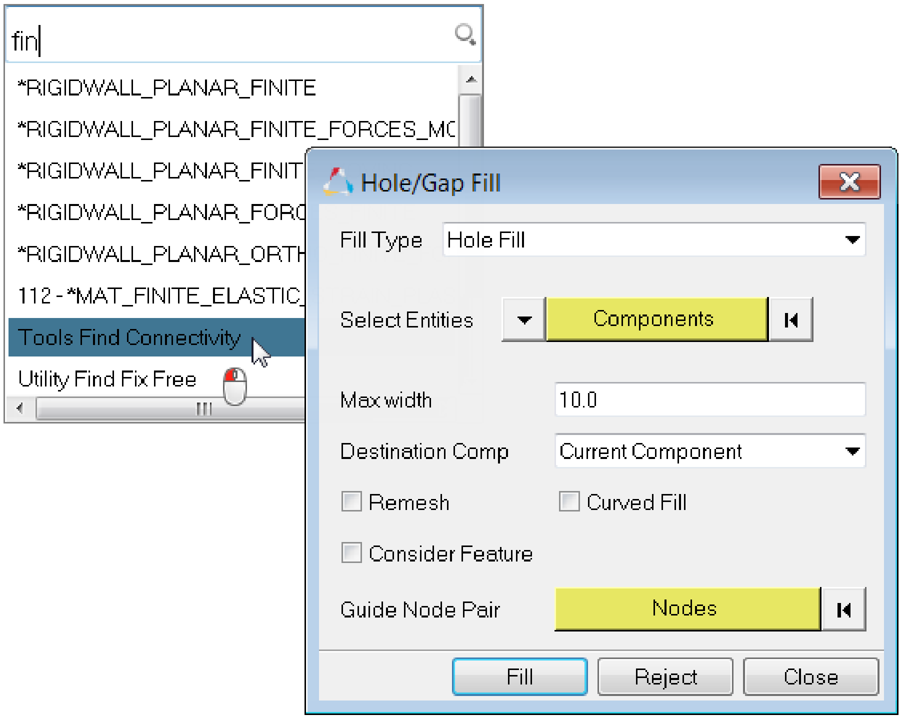


Figure 31:

Selecting a solver card from the list creates a new card.

> **Tip:** When creating solver cards with the Quick Access tool, set the Model Browser as the current tab so that you can easily recognize where newly created cards are being stored upon creation.

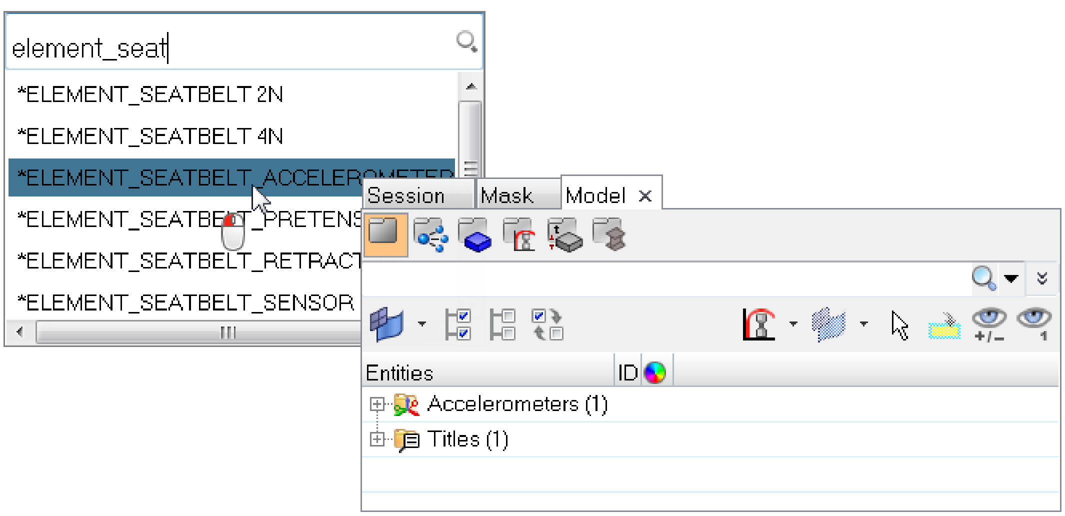


Figure 32:

Preferences

Configure the view of HyperMesh.

Check Element Settings

Check Element settings control which version of each element check to use.

Access Check Element settings by selecting **Preferences** > **Check Element Settings** from the menu bar.

Click the 2D and 3D tabs to set check element settings for the different types of elements.

Solver Settings

- Choose **All set to** to set all checks to the same solver's methods.
- Choose **Set individually** to set each check method individually.

Checks/Solvers

For each check, select the desired solver.

Checks/Method

For each check, select the method used to calculate the check.

Color Settings

Color settings control the display color of the user interface, as well as the display color of geometry and mesh entities.

Access Color settings by selecting **Preferences** > **Colors** from the menu bar.

General

Access color settings that control the user interface in the General tab.

Background 1/Background 2

Select any colors for Background 1 and Background 2, which control the color of the background gradient in the graphics area.

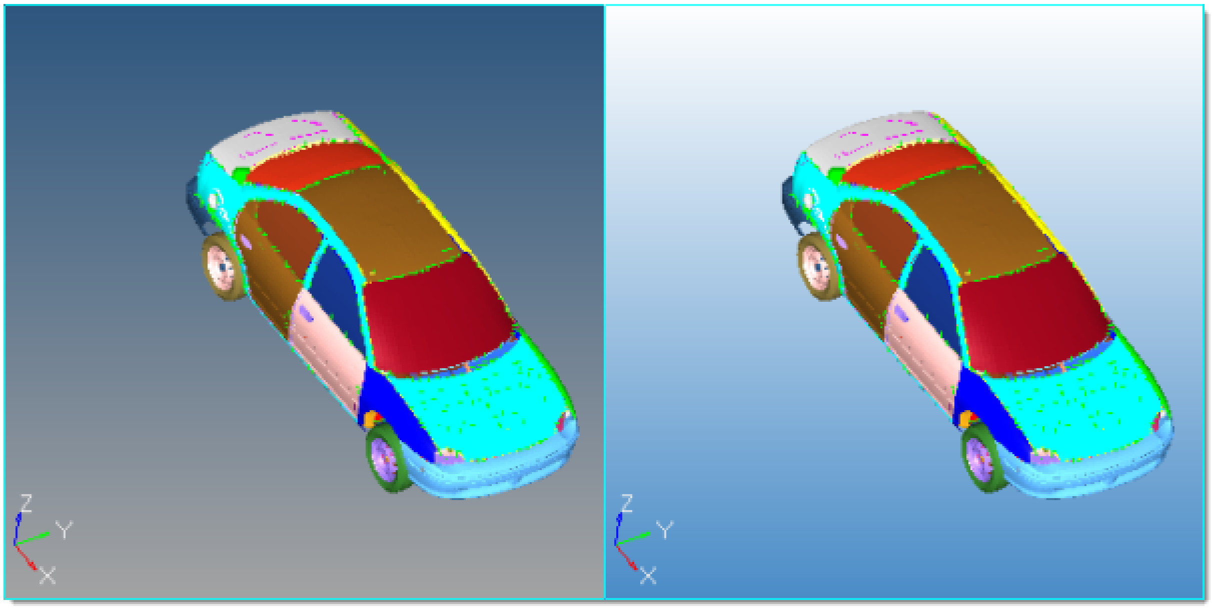


Figure 33:

Gradient Direction

Select one of the gradient boxes to change the direction and style of the gradient in the graphics area. The boxes themselves illustrate the gradient pattern that they apply, but not the current colors.

Show Global Axes

Show the global axes in the graphics area.

Global X axis / Global Y axis / Global Z axis / Axis label

Change the colors of the X, Y, and Z global axis vectors. By default, the X axis is red, the y axis is green, and the Z axis is blue.
Change the color of the axis labels. By default, the axis labels are white.

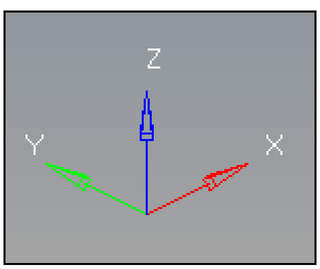


Figure 34:

Text and Line Color

Change the color of all the text and lines that displays in the graphics area to be black. By default, the text is white.

Geometry

Access color settings that control the geometric entities in the Geometry tab.

Different types of geometric features are broken down first by dimensionality (2D surfaces, 3D solids), each having no influence on the geometry of the other type. A third category, By mappable display mode (solids), applies to qualities of solids rather than parts of them. These colors apply specifically to how many possible directions solids can be mapped in, and are specific to the mappable geometry display mode. They will not show in any other display mode, even if the model contains solid entities.

Surface data

Free edges
: Edges of surfaces that do not connect to any other surfaces.

Shared edges
: Edges of surfaces that connect to one other surface.

Suppressed edges
: Shared edges that have been manually suppressed so that the automesher treats the shared surfaces as if they were one surface, allowing elements to cross the edge as if it were not there at all.

T-junctions
: Edges shared by 3 or more surfaces.

3D Solids

Fin faces
: Surfaces that split a 3D solid entity, but only partway through. They do not actually extend through the entire entity.

Bounding faces
: The outer faces of solid entities.

Full partition faces
: The faces of adjoined solids.

2D faces (topo)
: When using the by 2D topo visualization mode, this is the color of 2D faces that are not part of a solid.

Ignored (topo)
: The color of 2D faces when using the by 2D topo visualization mode.

Edges (comp)
: Mesh edges when coloring mesh with the by comp visualization mode.

By mappable display control (solids)

1 dir. map
: Visualization for solids that can be mapped (for 3D meshing) in one direction.

3 dir. map
: Visualization for solids that can be mapped (for 3D meshing) in three directions.

Not mappable
: Visualization for solids that have been edited, but still require further partitioning to create mappable solids.

Ignored map

Default visualization for solids that require partitioning to become mappable.

Mesh

Access color settings that control the FE entities in the Mesh tab.

Mesh line

Specify the color of the lines on a mesh that indicate the edges of mesh elements. By default, mesh line is set to **auto**, which assigns the component color as the mesh line color. To select a custom mesh line color for the entire mesh, click the toggle.

This option only works when the element color mode is set to **By Comp**. Other element color modes render mesh lines black or use the defaults selected in the Options panel, Color subpanel or in the **Colors** dialog, Mesh tab, accessed from the Preferences menu.

These settings are not applicable to 2nd order elements and clipped elements of section cuts, or when you are in Automesh panel mode.

Used in conjunction with the Mesh Appearance settings.

Elems, no prop/mat

Specify the color for elements that do not currently have any properties or materials assigned to them, either directly or inherited from the collectors that they belong to.

Mesh/Geometry Appearance Settings

Mesh/Geometry Appearance settings control the display of mesh lines, geometry lines, and application level anti-aliasing.

Access Mesh/Geometry Appearance settings by selecting **Preferences** > **Mesh/Geom Appearance** from the menu bar.

Meshing

Detail: Mesh Line Settings

Moving the Detail slider from 1 to 10 allows the displaying of mesh lines when you are viewing the model at further zoomed out levels regardless of the mesh coarseness.

These settings are not applicable to 2nd order elements and clipped elements of section cuts, or when you are in the Automesh panel.

These settings are used in conjunction with the mesh line option in the Options panel, Colors subpanel.

Transparency: Finite Element Settings

Moving the Transparency slider from 2.0 to 9.5 adjusts the transparency levels of finite element models from more opaque to more transparent.

Transparent with mesh lines

Check this box on to display mesh lines when finite elements are transparent.

Geometry

Detail: Geometry Line Settings

Moving the Detail slider from 1 to 10 allows the displaying of geometry lines when you are viewing the model at further zoomed out levels.

Solid edge width: Geometry Solid Edge Line Width Setting

Moving the Solid edge width slider from 1.0 to 5.0 allows the displaying of geometry solid edge line width thickness.

Auto edge adjustment

Check this box on to change the geometry line widths dynamically when zooming in and out.

Anti-aliasing

Anti-aliasing

Check this box on to have all graphic lines be anti-aliased, that is drawn smoother.

'

Lighting Settings

Lighting settings control the location, direction and intensity (high/low) of the specular lighting.

Access Lighting settings by selecting **Preferences** > **Lighting** from the menu bar.

One light

Manually control the direction of lighting, and adjust the specular lighting by moving the slider.

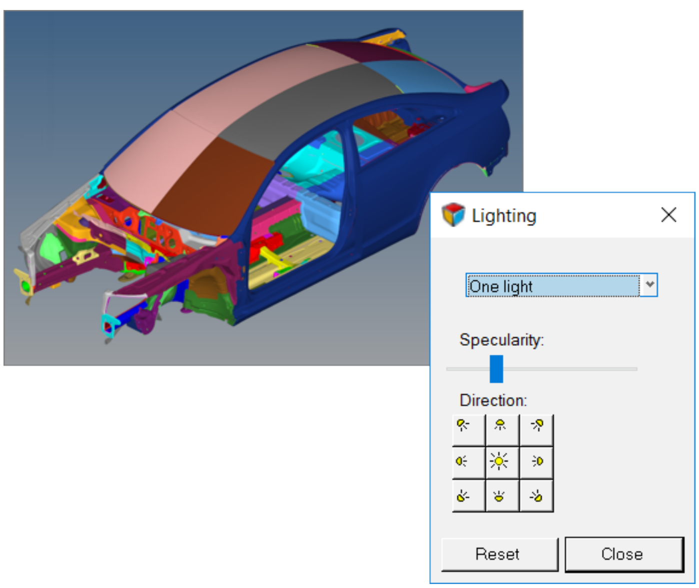


Figure 35:

Multi-lights (default)

Use the standard placement of the multiple lights, and adjust the specular lighting by moving the slider.

This setting does not allow for directional control.

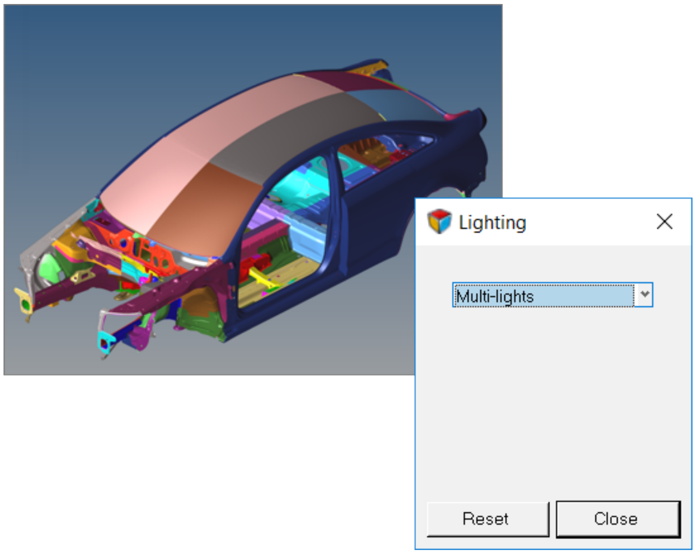


Figure 36:

Scale Settings

Scale settings control the scale display in the graphics area.

Access Scale settings by selecting **Preferences** > **Scale** from the menu bar.

Scale Position
: Select a location to display the scale.

Scale Size
: Select a display size for the scale.

Scale Color
: Pick a color to display the scale.

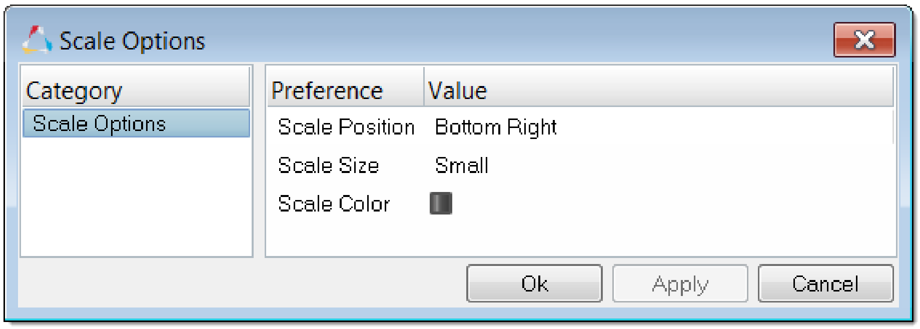


Figure 37:

Script Performance Settings

Script Performance settings control the APIs used to improve script performance when the original script may not have cleaned itself up correctly due to poor logic, unhandled errors or unexpected behavior.

Access Script Performance settings by selecting **Preferences** > **Script Performance** from the menu bar.

If you experience issues with entity selection not highlighting, messages not posting to the status bar, command files not populating or graphics not redrawing, reset the settings in this dialog.

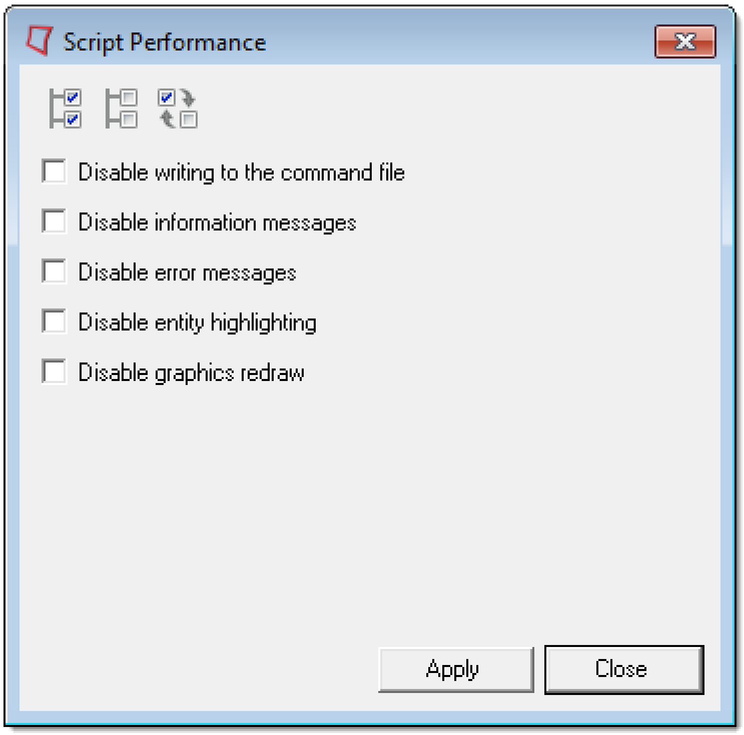


Figure 38:

Manage Favorite Panels

Save and access a list of your favorite panels. HyperMesh saves the list of favorite panels and restores it accordingly when you start a new session.

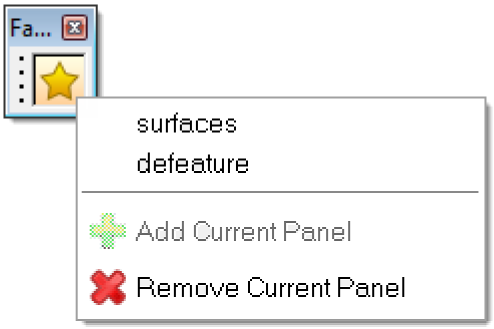


Figure 39:

Add Favorites

1. Open the panel you wish to save to the favorite panels menu.
2. From the Favorites toolbar, click and select **Add Current Panel**.

Remove Favorites

1. Open the panel you wish to remove from the favorite panels menu.
2. From the Favorites toolbar, click and select **Remove Current Panel**.

Change Solver Interfaces

Load a different solver interface in HyperMesh.

1. From the Standard toolbar, click .
2. In the **User Profiles** dialog, select a solver interface and click **OK**.

Interface with External Products

2

Solver interfaces supported in HyperMesh.

This chapter covers the following:

Current Supported Versions

Currently supported solver interface versions.

> **Note:** CAD translator support can be found in the *CAD Reader Support* and *CAD Writer Support* topics of the *Geometry* chapter.

Solver	Compatible Version	Model Files	Result File	Input Translator	Template
Abaqus	2019	`*.INP` (ASCII)	`*.FIN` (ASCII) `*.FIL` (binary) `*.ODB`	`abaqus`	Standard 2D Standard 3D Explicit
Actran	2004		`*.RES`		`actran/actran`
ANSYS	2019 R1	`*.CDB` (ASCII)	`*.RST` (binary) `*.RTH` (binary) `*.RMG` (binary)	`ansys`	`ansys/ansys.tpl`

EXODUS	6.09	`*.ex`
		`*.exo`
		`*.ex2`
			NA	`exodus`	`exodus/exodus.tpl`

LS-DYNA	971R6.1 – 13.0
	971R7.0 – 13.0.110
	971R7.1 – 14.0.110 (R7.0 renamed to R7.1)
	971R8.0 – 14.0.110
		`*.k`
		`*.key`
		`*.dyn`
		`*.dat`
		`*.bdf`
		`*.inc`
		`dynain` (ASCII)
			`d3plot`, `intfor`, `d3eigv`, `d3int`, `ptf`
			`binout` (binary)
			All ASCII time history files
			FEMZIP (up to v8.73)
				`dynakey`	`ls-dyna/dyna.key`

Solver	**Compatible Version**	**Model Files**	**Result File**	**Input Translator**	**Template**
	971R9.0 – 2019		d3plot, intfor		
MADYMO	7.0, 6.4, 6.3	*.xml (ASCII)	KIN3 (ASCII) FEMANI (ASCII)	madymo	madymo/ madymo70 madymo/ madymo63
Marc	Version 12 2008 r1 release	.dat (ASCII)	.t16 (binary)	marc.exe	marc/Stress2D.tpl marc/Stress3D.tpl
Nastran	MSC 2015 NX 9	*.dat*.bdf *.nas	*.f06 (ASCII) *.pch (ASCII) *.op2 (binary) *.xdb (binary) FEMZIP Nastran (v1.3.6)	nastran	nastran/ general nastran/ generallf
PAM-CRASH 2G	VPS 2012 – 14.0 VPS 2013 – 14.0.110 VPS 2014 – 14.0.110 VPS 2015 – 14.0.130 VPS 2016 – 2017.2 VPS 2017 – 2019	*.pc *.dat *.inc *.hma (ASCII)	*.DSY *.THP (binary) *.ERHF5 (binary) FEMZIP DSY (v8.73) VPS 2012 – 12.0.110 VPS 2013 – 13.0.110 VPS 2014 – 14.0 VPS 2015 – 14.0.130	pamcrash2g	pamcrash2g/ general

Solver	Compatible Version	Model Files	Result File	Input Translator	Template
			VPS 2016 – 2017.2 VPS 2017 – 2017.2.3 Modular ERF - 2019		
Permas	15	`*.dat` (`.uci` not supported)	`*.h3d` (binary) – no translator necessary	`permas`	`permas/permas`
OptiStruct	OptiStruct 2019	`*.fem` `*.dat`	`*.out` (ASCII) `*.pch` (ASCII) `*.h3d` (binary) `*.op2` (binary) `*.res` (binary)	`optistruct`	`optistruct/optistruct` `optistruct/optistructlf`
Radioss (Fixed Format)	4.1	`*.d00` (ASCII)	`animate.A00` (binary) `history.T0*` (binary) Engine file `*D01` not supported	`radiossfix`	`radioss/radioss31.fix` `radioss/radioss41.fix`

Radioss	Radioss2018 - 2019	`*.d00` (ASCII),`*_0000.r`
		Engine file `*D01`, `*_001.rad`
			`animation *A`
			`*#A00*` (binary)
			`Time history *T0*, *_000*.thy, *@T0*, *#T0*`
				`radiossblk`	`radioss/radioss41.blk`
					`radioss/radioss42.blk`
					`radioss/radioss44.blk`
					`radioss/radioss51.blk`
					`radioss/radioss90.blk`

Solver	Compatible Version	Model Files	Result File	Input Translator	Template
					radioss/radioss100.blk radioss/radioss110.blk radioss/radioss120.blk radioss/radioss130.blk radioss/radioss140.blk radioss/radioss2017.blk radioss/radioss2018.blk
Samcef	17.1	.dat	n/a	n/a	samcef/general

Change Solver Interfaces

Load a different solver interface in HyperMesh.

1. From the Standard toolbar, click .
2. In the **User Profiles** dialog, select a solver interface and click **OK**.

Types of Interfacing

HyperMesh uses three types of external interfacing.

These include:

- Importing CAD-generated geometry or finite element model information
- Exporting CAD geometry or finite element information for specific analysis codes
- Translating analysis results information to HyperMesh binary results format

CAD-generated geometry and finite element model information are input using the **Import** tab. HyperMesh internally supports PDGS and DXF formats. External interfaces are supplied for finite element analysis codes Abaqus, ANSYS, EXODUS, LS-DYNA KEY, LS-DYNA SEQUENTIAL, MADYMO, Marc, MoldFlow, Nastran, OptiStruct, PAM-CRASH, Radioss, Radioss {Fixed Format}, and Samcef) and for crash codes (LS-DYNA3D, PAM-CRASH, and Radioss), as well as for CAD codes, (CATIA, DXF, IGES, PDGS, PRO/E, STEP and UG). There is an external interface for the universal IGES format, as well. You can also create your own translation package.

The **Export** tab allows you to write information from a HyperMesh database to many finite element formats. Geometry data can be written in IGES format.

HyperMesh uses templates to create the analysis input decks for finite element solvers. You can modify the existing templates to support a desired feature or create a new template to support another analysis code. The HyperMesh templates can be used to create model summaries and perform some analysis calculations, such as center of gravity. You can also use the templates to perform complex editing or data manipulation tasks.

HyperMesh provides external translators that convert analysis-specific results into the HyperMesh results format. The results subpanel allows you to specify the results files for use in post-processing.

> **Note:** For more information, refer to the individual translators on the Results Translation page.
>
> HyperMesh uses both internal and external translators. You can write new external translators to support CAD or FEA formats by using the functions provided in hminlib.

> **Note:** The HyperMesh Programmer's Guide provides information about working with:
>
> hmlib
>
> hminlib
>
> hmmodlib
>
> hmreslib

See FE Input Readers for more information about writing your own translators.

When you open the **Export** tab, HyperMesh sets the **Template:** field to the default analysis template directory and the first template selected is designated the global template.

Export Options

There are several options available for exporting model data. For example, the **all/displayed** option allows you to export files based on a subset of the existing HyperMesh database. If you select **all**, the entire model is written, regardless of the current display settings. When you select **displayed** in the **Export** tab, only a portion of the model displayed on the screen is exported.

Additional information about export options for geometry and solvers is included in the section that pertains to the specific format.

Abaqus Interface

Overview of the Abaqus Interface.

Supported Abaqus Profiles

Standard.2d

Generates a deck for two-dimensional models containing planar or axisymmetric elements for use with Abaqus Standard.

Standard.3d

Generates a deck for three-dimensional models containing bar, shell, and solid elements for use with Abaqus Standard.

Explicit

Generates a deck for use with Abaqus Explicit.

Import and Export

- HyperMesh places all elements into separate components, based on sectional property. Each component is written out from HyperMesh as an element set in the Abaqus input deck.
- Although HyperMesh supports 160 characters in entity names, Abaqus output files are truncated at 80 characters to match the Abaqus name support level.
- Loads and constraints are organized under load collectors in HyperMesh and added to a load step for the *STEP card in the Abaqus history definition. If loads are applied to node or element sets, you can resolve these sets to individual nodes or elements by enabling the **Expand loads on sets** option in the Import panel under Import options.
- Output options are organized under output blocks in HyperMesh. These output blocks also need to be added to load steps.
- Resolve sets into nodes or elements, which are defined using the *GENERATE* parameter, using the Solver options in the Import Browser. This is useful when nodes/elements are renumbered due to ID conflicts during import.

Syntax

- HyperMesh supports some abbreviated key words and parameters.
- All Abaqus keywords and parameters supported in HyperMesh are not case insensitive.
- HyperMesh ignores spaces in keyword lines.
- HyperMesh supports quotation marks around component (ELSET with sectional properties) names, which is especially useful for names that begin with a number.

HyperMesh Operations

- Warnings and error messages are written to a file named `abaqus.msg`. Unrecognized lines are written to a `*.hmx` file. These files are created in the same directory from where HyperMesh is launched.
- Step time calculations can differ between HyperMesh and Abaqus (Explicit analyses), therefore you may find differences between reported values in the Abaqus status (`.sta`) file and the Time

subpanel of the Check Elems panel. The values reported by HyperMesh are close estimates of the step time; refer to the Abaqus documentation to learn about the factors that Abaqus/Explicit uses to reach the final result.

Degenerated Brick Element

Elements that are defined in one configuration can be degenerated or collapsed to create other configurations, while still preserving the number of nodes and element type in the original configuration.

For example, the HyperMesh Qaud4 element can degenerate into a Tria3 element, but when exported, the four nodes and quadrilateral element information is written in the solver deck.

Degenerated brick elements are either Hex8 or Hex20 elements, collapsed into a wedge element (Penta6 or Pet15). Penta6 (Element configuration 206) is a 3D, 1st order, triangular prism shape element with a 6 noded order as shown inFigure 40. Hex8 (Element configuration 208) is a 3D, 1st order, brick shape element with a 8 noded order as shown in Figure 41. The Degenerated Brick (Element configuration 206) option is a 3D, 1st order, triangular prism shape element with a 8 noded order as shown in Figure 42.

Standard node order for a 1st order, penta6 element will be 1, 2, 3, 4, 5, 6. In cases of degenerated brick elements, the node order will be 1, 2, 2, 4, 5, 6, 6, 8, with nodes 2 and 6 being repeated during export. The same node repetition method is used for a 2nd order, degenerated brick element, but nodes 2 and 6 will be repeated twice and node 18 will be repeated once (1, 2, 2, 4, 5, 6, 6, 8, 9, 10, 2, 12, 6, 14, 15, 16, 17, 18, 18, 20).

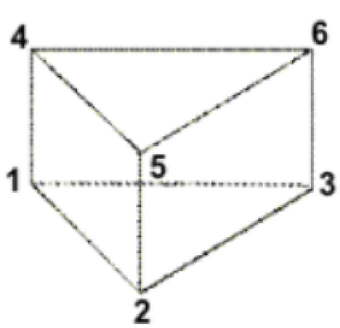


Figure 40: Element Configuration 206, 6-Noded Penta

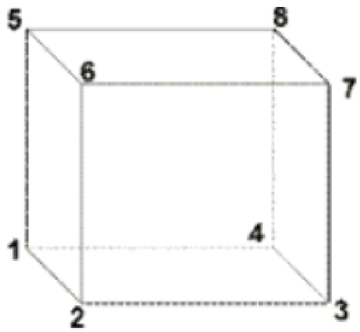


Figure 41: Element Configuration 208, 8-Noded Hexa

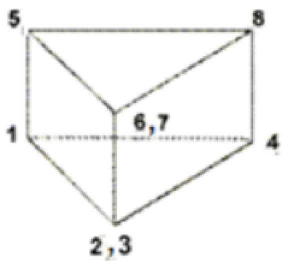


Figure 42: Element Configuration 206, 8-Noded Penta

Since the 8-noded, hexa element is degenerated into a 8-noded penta element, the 1st order brick element option is added under the 6-noded penta (Element configuration 206) and the 2nd order brick element option is added under the 15-noded penta (Element configuration 215).

Actran Interface

Overview of the Actran interface.

Define Actran Models

Define an Actran model in HyperMesh.

1. Create materials.

   Four types of materials can be defined: fluid, visco-elastic, porous, and viscothermal fluid.
2. Create components.
3. Create mesh on the appropriate components.
4. Define boundary conditions, both node-related and element face-related.
5. Define the control card properties.
6. Create the Actran input deck file.

AcuSolve Interface

Overview of the AcuSolve interface.

Supported AcuSolve Entities

Entities used to set up a case to run a CFD analysis using AcuSolve.

Global
Contains the physical description of the case, for example, laminar/turbulent and numerical parameters, such as convergence tolerance.

Output
Define the frequency of different outputs: Derived Quantity Output, Nodal Output and Restart Output.

Volumes
Components with Fluid and Solid card images are grouped in this folder.

Surfaces
Components with Inflow, Outflow, Wall, Slip, Symmetry and Far Field card images are grouped in this folder.

Components With No Card Image
Components with no card image will not be exported with the solver deck. By default, all of the components will not have a card image. You have to choose the appropriate card image for the component and the component will be moved to the Surfaces or Volumes folder accordingly.

Nodal Boundary Condition
Define boundary conditions on nodes apart from surface boundary conditions.

Materials
Contains material definitions.

Body Force
Define gravity and heat source for volume components.

Periodic Boundary Condition
Define periodic or axisymmetric boundaries along with the transformation definition.

Reference Frame
Define rotational reference frame for the model.

Mesh Motion
Define translation and rotation for the mesh.

Emissivity Model
Define emissivity model for radiation surface.

Multiplier Function

Define multiplier functions (Constant, Piecewise Linear, Cubic Spline, Piecewise Log Linear) and utilize multiplier functions to ramp up supported parameters, for example angular velocities of reference frame.

Parameters

Parameters can be utilized to run solver parameter based studies.
For example, if you want to run a series of simulations with inlet velocity values of 0.5 m/s, 0.75 m/s, 1 m/s respectively, you can define a parameter on inlet velocity with the desired values.
Using the **HyperStudy Job Launcher** DOE studies can be performed.

Solver Job Launcher

The Solver Job Launcher enables you to execute AcuSolve jobs directly from Engineering Solutions.

The tool integrates the mesh export and the AcuSolve job run. The submitted job can also be controlled from Engineering Solutions.

Invoke Solver Job Launcher

Click the icon in the CFD toolbar.

The **Job Launcher** dialog opens.

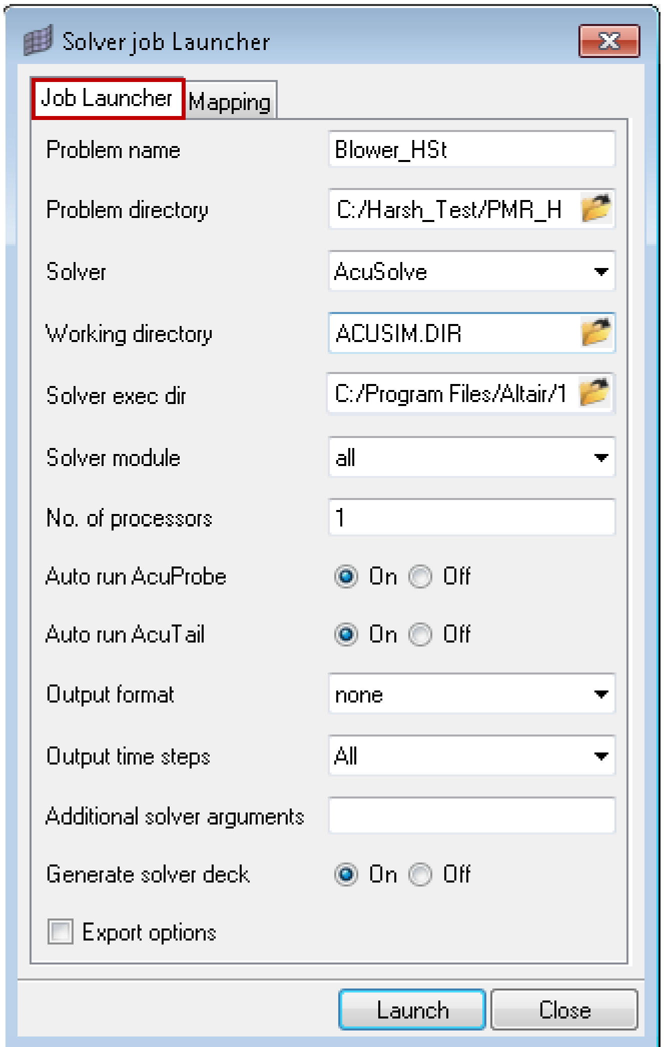


Table 1:

Problem name	Specifies the name of the input deck without the extension `*.inp`. Do not use spaces in the name.
Problem directory	Directory where the input deck is located. The solver run will be started in the problem directory.
Working directory	The result files of the AcuSolve run will be saved into the working directory.
Solver exec dir	Define the AcuSolve executable path. The path will be auto selected if the solver is available in the same installation path. If it is not available, the path will be blank. The path needs to point to the

	AcuSolve bin folder (`/acusolve/win64/bin` or `/acusolve/linux64/bin`).
Solver module	A selection of several solver modules. **prep** Processes and prepares the input data for the solver run. **view** View factor computation for problems including radiation. **solve** Runs the solver. **prep-solve** **all**
No. of processors	The number of processors used for this run.
Auto run AcuProbe	Automatically opens the plotting utility AcuProbe to monitor run data of the current run, for example residual.
Auto run AcuTail	Automatically opens AcuTail to view the log file of the current run.
Output format	Converts the results of the final time step into the specified format.
Output time steps	**All** Creates the output file in a user defined format with all of the time steps. This option is useful for a transient run. **Final** Creates an output file in a user defined format with only the final time step. This option should be utilized for a steady state run.
Additional solver arguments	Additional solver options can be specified, for example number of threads.
Generate solver deck	**On** A new `.inp` file will be created. **Off** An `.inp` file will not be generated and the program will look for the existing file.

Export options	**Always two layers for interfaces** Creates two interface layers between two volumes, if one or both are missing. **Create exterior faces** Creates outer shell surface of volume if missing. **Always add elem type to comp names** Append element configuration at the component names.

Extract Output from AcuSolve Run

Use this utility to extract specific output from an AcuSolve run.

The extracted results will be in a `.csv` file. From these results you can create a field, which can be utilized to map CFD results to other structural mesh. This utility requires a finished AcuSolve run. If the mapping field is defined before the solver run, the tool will generate output in a `.csv` file format. This utility can be utilized any time after the solver run to extract results.

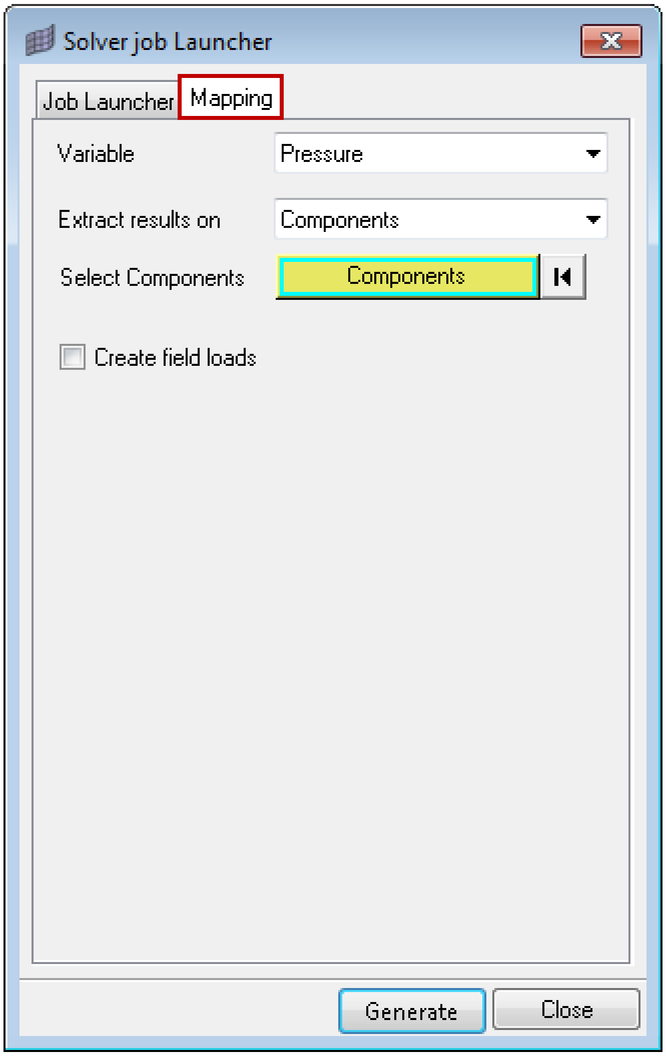


Figure 43:

Table 2:

Variable	You can select the desired output. The options are Pressure, Velocity and Eddy viscosity. The selection list is dependent on the solver options selected (Turbulence model).
Extract results on	You can extract results on all of the components that are selected.
Create field loads	If this checkbox is activated, a field will be created automatically, which can be utilized to map the results on a structure mesh.

Generate	This button will be activated to click if any result data is present in the working directory.

Launch Job

1. Click **Launch** in the Solver Job Launcher to open the **AcuSolve Control** dialog.

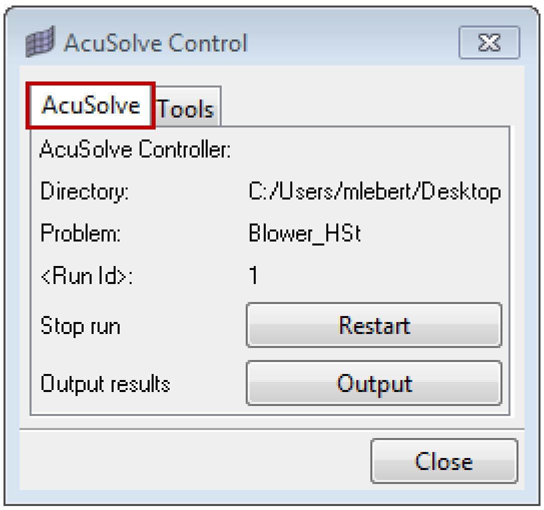


Figure 44:

2. Select the AcuSolve tab.
   The AcuSolve tab contains options to stop the run, restart the run, and output results.
3. Click **Stop**.
   The Restart option appears.
   Stop: stops the run as soon as the current iteration is finished.
   Restart: restarts the job from the previous time step.
   Output: the existing iteration result is output and you can visualize the results later on.
4. Select the Tools tab.

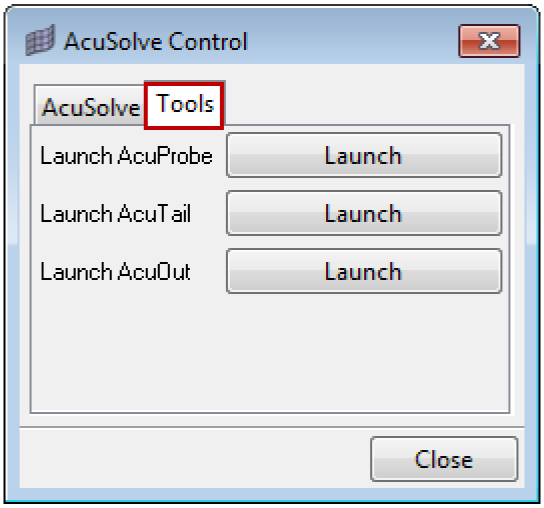


Figure 45:

The Tools tab contains options to launch or relaunch AcuProbe, AcuTail and AcuOut.

AcuSolve Set Up Features

The following AcuSolve set up features are supported in Engineering Solutions via the Solver Browser.

Problem Description

- Title
- Analysis type (Steady State or Transient)
- Flow equation (Navier Stokes)
- Abs. pressure offset
- Temperature equation (Advective Diffusive or None)
- Abs. temperature offset
- Radiation equation (Enclosure or None)
- Turbulence model (Laminar, Spalart Allmaras, SST, K Omega, Detached Eddy Simulation, SST DES, Dynamic LES, Classical LES)
- Mesh type (Fixed or Fully Specified)

Solver Settings

- Max time steps
- Final time
- Initial time increment
- Auto time increment
- Multiplier Function
- Convergence tolerance
- Min stagger iterations

- Max stagger iterations
- Relaxation factor
- Flow
- Temperature
- Temperature flow
- Enclosure radiation
- Turbulence

For full support of all AcuSolve features, use AcuConsole. A model can be transferred from Engineering Solutions to AcuConsole by clicking the icon in the CFD toolbar.

Set Up an AcuSolve Input Deck

The following section describes the set up of an AcuSolve input deck within Engineering Solutions.

> **Note:** All of the data in Engineering Solutions is dimensionless, meaning it is your responsibility to use a consistent unit system throughout the modeling process. All units are assumed to be SI units

Import an Existing AcuSolve Model

Click the icon on the Standard toolbar. If you are importing an AcuSolve solver deck (`.inp` file) the defined parameters in the solver deck will also be imported in the model.

If any parameters defined in the solver deck are not supported, they will be collected in Unsupported Entries under each folder category of the Solver Browser. You can manually edit those entries and the modified values will be exported.

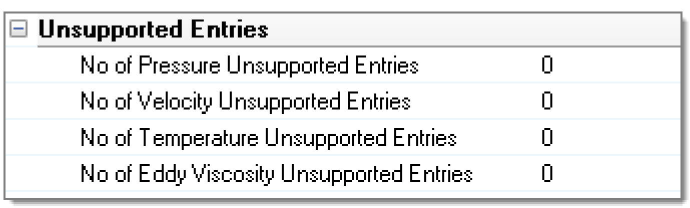


Figure 46:

Note: If you defined any fields in the Unsupported Entries category that are not recognized by AcuSolve, this will result in an error.

Load the CFD User Profile and Open the Solver Browser

1. Click the icon on the Standard toolbar.
2. Select **Engineering Solutions** > **CFD** > **AcuSolve** in the User Profiles dialog.
3. Click **View** > **Solver Browser**.

Create CFD Components

1. Click **Mesh** > **Components** > **CFD** to open the **Create CFD Components** dialog.

   You can change the name and color of the components, add new components to the list or change the quantities of the components.
2. Using the Organize panel, you can move elements to the appropriate components.

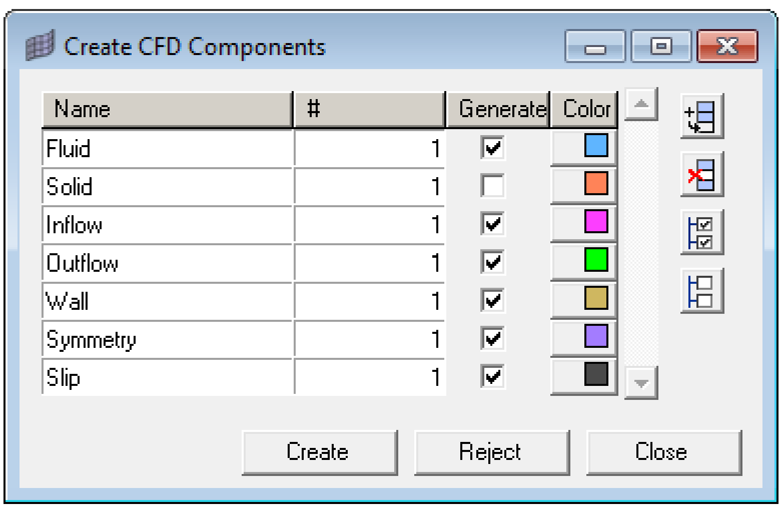


Figure 47:

Set Up the Model in the Solver Browser

The Solver Browser can be used to set up a case to run CFD analysis using AcuSolve.

It is useful for setting up problems with simple boundary conditions. For more complex problems, export the model and set it up in AcuConsole.

The Solver Browser contains the following main folders:

Table 3:

Global	This folder is used to define problems and set solver parameters for that problem. It contains the following sub folders: Nodal Initial Condition, Problem Description, Radiation Parameters and Solver Settings. The Problem Description folder contains utilities to set up the analysis type and solver models. Use the Solver Settings folder to define parameters related to selected analysis types and solver models. Initial condition can be defined in the Nodal Initial Condition folder. The radiation parameters for the model can be defined in the Radiation Parameters folder.
Output	Define the frequency of different outputs: Derived Quantity Output, Nodal Output and Restart Output.
Volumes	Components with Fluid and Solid card images are grouped in this folder.
Surfaces	Components with Inflow, Outflow, Wall, Slip, Symmetry and Far Field card images are grouped in this folder.
Components With No Card Image	Components with no card image will not be exported with the solver deck. By default, all of the components will not have a card image. You have to choose the appropriate card image for the component and the component will be moved to the Surfaces or Volumes folder accordingly.
Nodal Boundary Condition	Define boundary conditions on nodes apart from surface boundary conditions.
Materials	Define/edit the material model for CFD analysis.
Body Force	Define gravity and heat source for volume components.
Periodic Boundary Condition	Define periodic or axisymmetric boundaries along with the transformation definition.
Reference Frame	Define reference frames.
Mesh Motion	Define mesh motion.

Emissivity Model	Define emissivity model for radiation.
Multiplier Function	Define multiplier functions to ramp up supported parameters.
Parameter	Parameters can be utilized to run solver parameter based studies. For example, if you want to run a series of simulations with inlet velocity values of 0.5 m/s, 0.75 m/s, 1 m/s respectively, you can define a parameter on inlet velocity with the desired values. Using the HyperStudy Job Launcher DOE studies can be performed.

Clicking any of these entities in one of the main folders opens the Entity Editor at the bottom of the browser, where parameters for this particular entity can be entered.

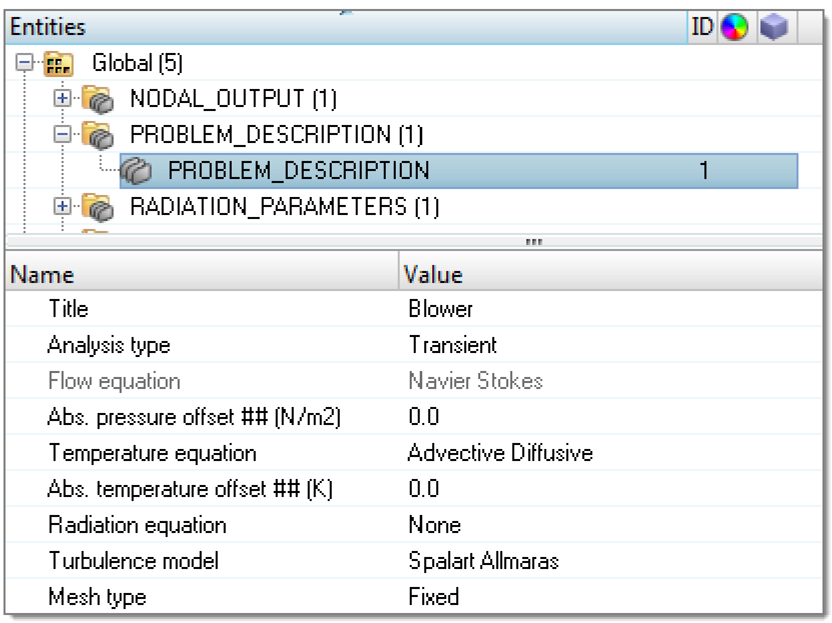


Figure 48:

Define Component Type / Properties

1. Load an existing mesh file into Engineering Solutions.

   The different components will appear in the browser.
2. Click the component and edit it in the Entity Editor.

   The elements are grouped into components of a specific type. By default, all components will have None card image and will appear in the Components With No Card Image folder. You have to choose the appropriate card image for the component and the component will be moved to the Surface or Volume folder accordingly. There are eight types of components: FLUID, SOLID, INFLOW, OUTFLOW, WALL, SLIP, SYMMETRY and Far Field. Each component in the model needs to be assigned to one of those types. For INFLOW, OUTFLOW, WALL and SYMMETRY there are four categories: Simple Boundary Condition, Radiation Surface, Interface Surface and Surface Output. Each category can be turned on or off.

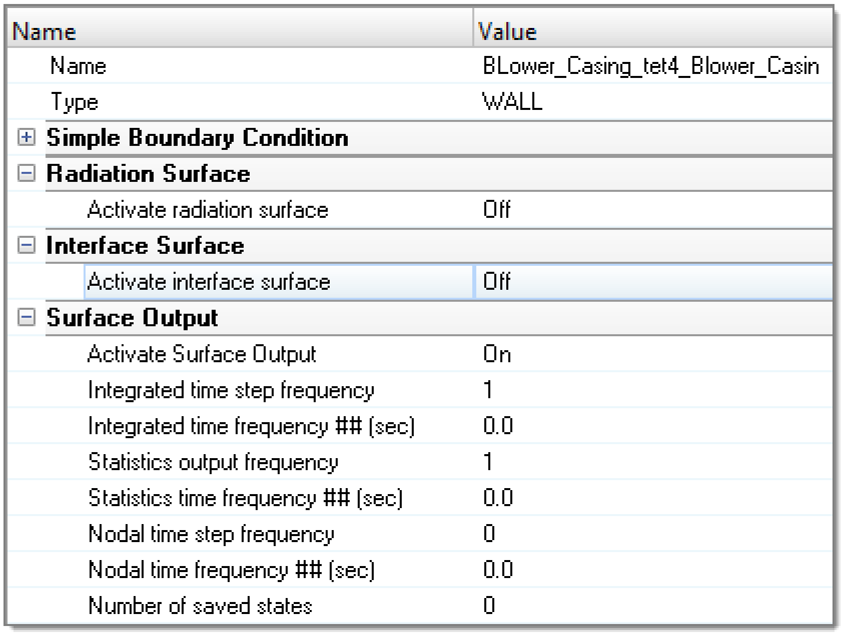


Figure 49:

Define Nodal Boundary Condition

This folder can be used to define boundary conditions on nodes apart from simple boundary conditions.

1. Right-click Nodal_Boundary_Condition and select **Create**.

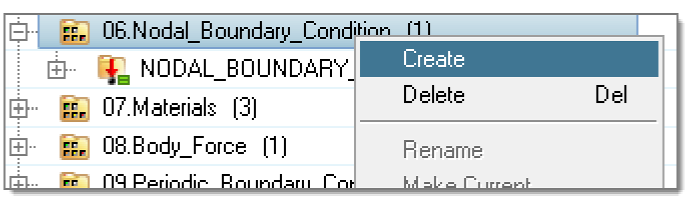


Figure 50:

2. Select Nodal component or Nodal BC file (`.nbc` file) as the input.

   If you select Nodal component select a component whose nodes will be assigned a specific nodal boundary condition. You have to make sure that the defined source and target are mappable, or else the solver run will have an error. If you select Nodal BC file and you already have the file from a previous run specify the file path.

3. Select a Boundary condition variable.
4. Select an Active type.
5. Select Multiplier function and Reference frame, if applicable.

Create Material Model

The Materials entity is used to define/edit the material model for CFD analysis.

The materials created by default when you open the Solver Browser are:

- Aluminum
- Air
- Water

1. Right-click Materials and select **Create** > **Material(Fluid)** or **Material(Solid)**.

Figure 51:

2. Define/edit the material properties.
   You can define Porosity when you select Material (Fluid).
3. Assign material to the volume components.

Create Body Force

This folder can be used to define gravity and heat source for volume components.

1. Right-click Body_Force and select **Create**.

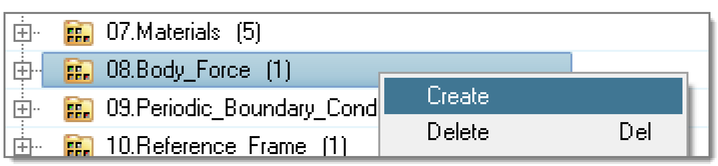


Figure 52:

2. Define the properties.
3. Assign newly created body force to volume components.

Create Periodic Boundary Condition

This folder can be used to define periodic boundary condition.

1. Right-click Periodic_Boundary_Condition and select **Create**.

Figure 53:

2. Select Periodic components or Periodic BC file as the input.

   If you select Periodic components you need to define the Source and Target components. You have to make sure that the defined source and target are mappable, or else the solver will have an error. Select Periodic BC file, if you already created a `.pbc` file using the acuPbc utility or have it from a previous run.
3. Select the Periodic BC type, which can be Periodic or Axisymmetric.

   If you select Axisymmetric, define the Rotation axis with two node definitions.

Create Reference Frame

This folder can be used to define reference frames.

1. Right-click Reference_Frame and select **Create**.

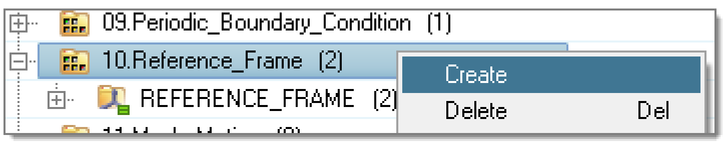


Figure 54:

2. Define the Rotation center, Angular velocity and set force.
3. Assign reference frame to volume components and/or surface components.

Create Mesh Motion

This folder can be used to define mesh motion.

1. Right-click Mesh_Motion and select **Create**.

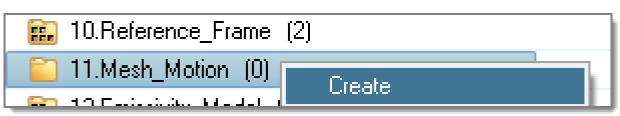


Figure 55:

2. Define the translation, rotation or translation parameters for mesh.

3. Assign reference frame to volume components and/or surface components.

Create Emissivity Model

This folder can be used to define emissivity model for radiation.

1. Right-click Emissivity_Model and select **Create**.

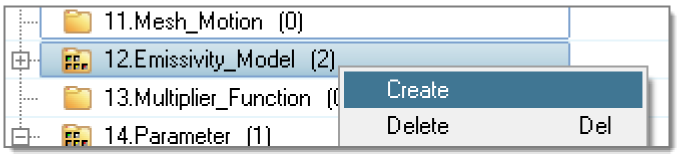
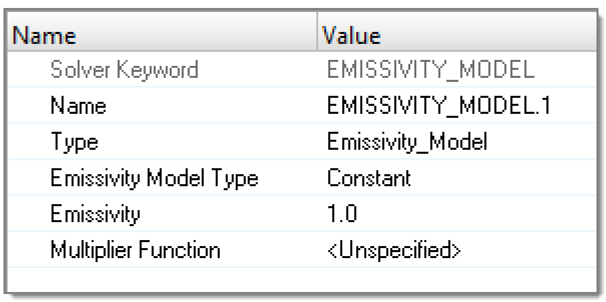


Figure 56:

2. Define emissivity.
3. To assign it to surface, activate radiation in the desired surface.
4. Assign emissivity to the model.



Name	Value
Solver Keyword	EMISSIVITY_MODEL
Name	EMISSIVITY_MODEL.1
Type	Emissivity_Model
Emissivity Model Type	Constant
Emissivity	1.0
Multiplier Function	<Unspecified>

Figure 57:

Create Multiplier Function

1. Right-click Multiplier_Function and select **Create**.

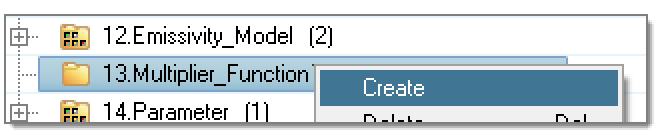


Figure 58:

2. Define the Multiplier Function Type as Constant, Piecewise Linear, Cubic Spline or Piecewise Log Linear.

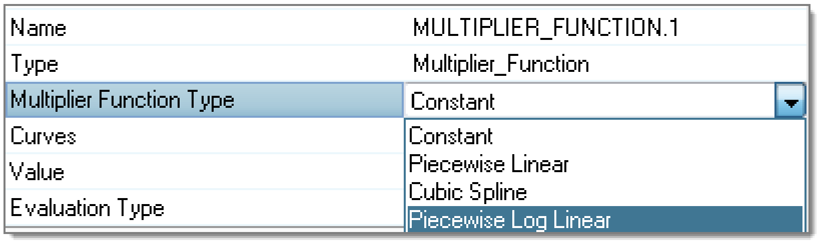


Figure 59:

3. If you select Piecewise Linear, Cubic Spline or Piecewise Log Linear, right-click on **Curves** > **Create** to create a new plot for the curve definition.

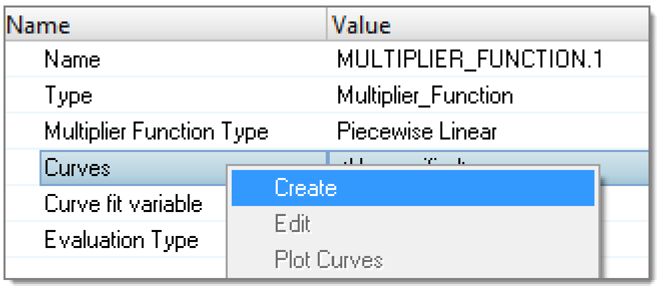


Figure 60:

4. Right-click on **Curves** > **Plot Curves**.

The Curve editor opens.

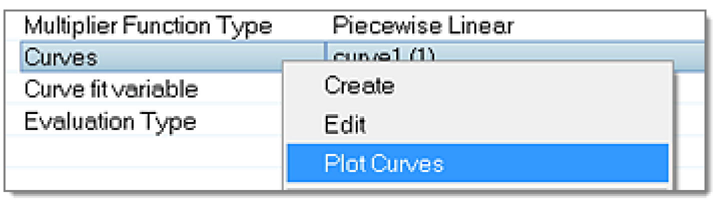


Figure 61:

5. Edit the plot values, click **Update** to visualize the plot and then close the Curve Editor.

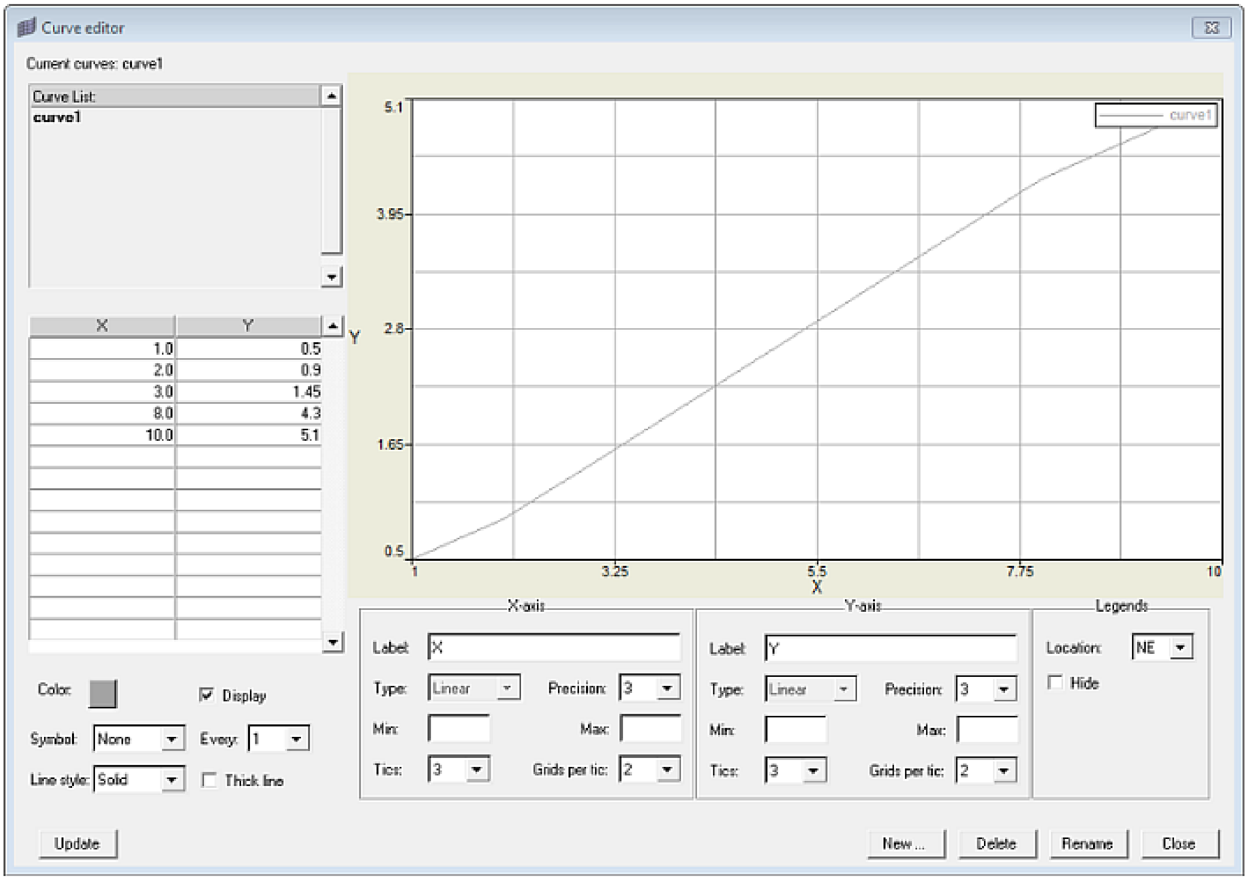


Figure 62:

6. Define the Curve fit variable and Evaluation Type.
7. Assign the multiplier function to one of the supported fields: reference frame, mesh motion, emissivity model or material properties (viscosity and conductivity).

Name	Value
Name	REFERENCE_FRAME.1
Centrifugal force	On
Coriolis force	On
Angular acceleration force	On
Rotation center	<Unspecified>
X-coordinate ## (m)	0.0072447808039115
Y-coordinate ## (m)	0.00012479068865341
Z-coordinate ## (m)	0.0625
Angular velocity - X ## (rad/sec)	0.0
Angular velocity - Y ## (rad/sec)	0.0
Angular velocity - Z ## (rad/sec)	2000.0
Multiplier Function	MULTIPLIER_FUNCTION.1 (2)

Figure 63:

Create Parameter

This folder can be used to define parameters for Solver parameter based studies.

1. Right-click Parameter and select **Create**.

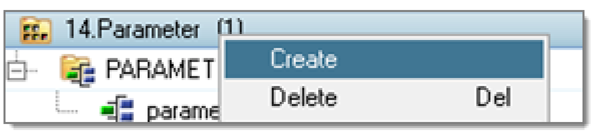


Figure 64:

2. Define the Parameter mode and the Parameter type. Two parameter modes are available: Discrete and Continuous. Discrete refers to the input values list. You have to first define how many counts you want to define, and then click on the table to define the values.

Figure 65:

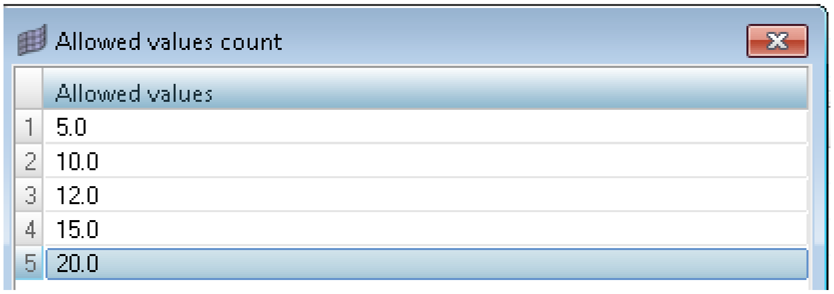


Figure 66:

Continuous refers to incremental values defined by Lower bound, Upper bound and No of levels.

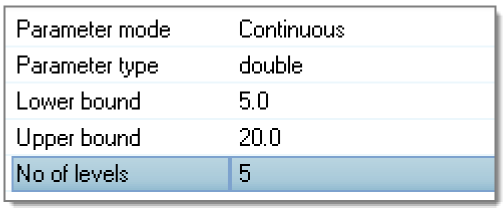


Figure 67:

3. Assign the defined parameter to the solver variable input by right clicking **Solver** > **variable input** > **Select Parameter**.

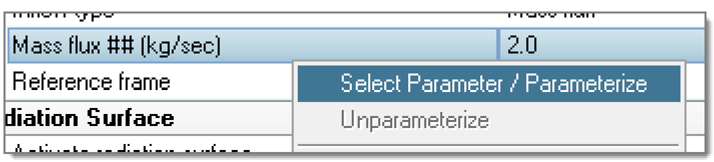


Figure 68:

> **Note:** If the solver variable accepts double values only parameters with double type will be filtered out for selection.

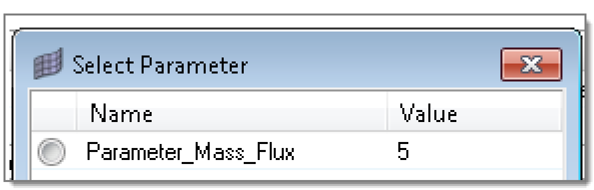


Figure 69:

4. Open the HyperStudy Job Launcher to run a sequential run with parametric values.

Launch the Solver Run

1. Invoke the Solver job Launcher by clicking the icon in the CFD toolbar. In the dialog, define the Problem name and specify the Problem directory, which contains the solver input file.
2. Clicking **Launch** starts the Solver run.

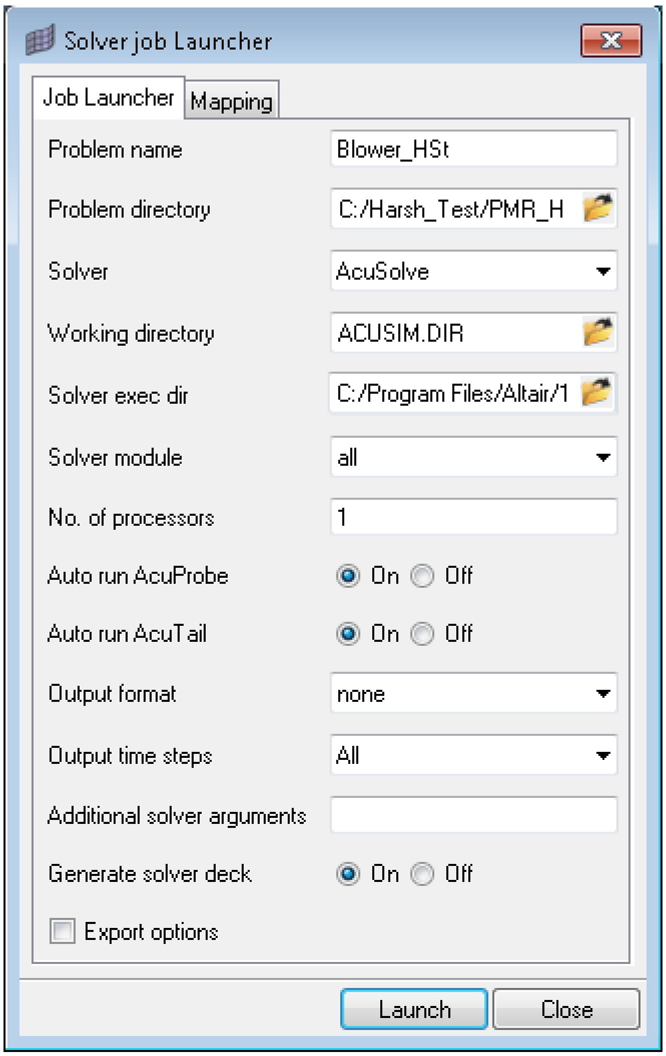


Figure 70:

By default Auto run AcuProbe and Auto run AcuTail are activated, which will invoke AcuTail to monitor the `.log` file and AcuProbe to monitor the residuals of the current run. Both utilities access the `.log` file of a solver run.

```
acuSolve:    GMRES    No iterations =        11 (1.10)
acuSolve:    GMRES     0/1/n norms = 2.392051e+004 3.919199e+004
2.334940e+003
acuSolve:    pressure    sol ratio = -0.000000e+000
acuSolve:    velocity    sol ratio = 9.173892e-001
acuSolve:   Turbulence stagger "turbulence": FORM-LHS
acuSolve:    eddy-visc.   res ratio = 6.626819e-001
acuSolve:    GMRES    No iterations =        11 (0.28)
acuSolve:    GMRES     0/1/n norms = 1.538764e+000 1.538764e+000
1.321114e-002
acuSolve:    eddy-visc.   sol ratio = 1.522780e+000
acuSolve:   Flow stagger "flow": FORM-LHS
acuSolve:    pressure    res ratio = 8.166296e-002
acuSolve:    velocity    res ratio = 3.613098e-003
acuSolve:    CGP      No iterations =        85
acuSolve:    CGP      0/1/n norms = 9.831736e+003 9.139119e+003 9.704266e+001
acuSolve:    GMRES    No iterations =        10 (1.00)
acuSolve:    GMRES     0/1/n norms = 1.859903e+004 1.452951e+004
9.061639e+002
acuSolve:    pressure    sol ratio = 2.692770e+000
acuSolve:    velocity    sol ratio = 3.092784e-001
acuSolve:   Turbulence stagger "turbulence": FORM-LHS
acuSolve:    eddy-visc.   res ratio = 3.936169e-001
acuSolve:   User signaled to stop
acuSolve:    GMRES    No iterations =        19 (0.47)
acuSolve:    GMRES     0/1/n norms = 3.225057e-001 3.225057e-001 2.707993e-003
acuSolve:    eddy-visc.   sol ratio = 4.539075e-001
acuSolve:   CFL timeInc            = 2.082624e-006
acuSolve:   Step CPU/Elapse time      = 1.101300e+001 1.104500e+001 Sec
acuSolve: ------------------- End Time Step -------------------
acuSolve: Input      CPU/Elapse time= 6.870000e-001 4.078000e+000 Sec
```

Figure 71:

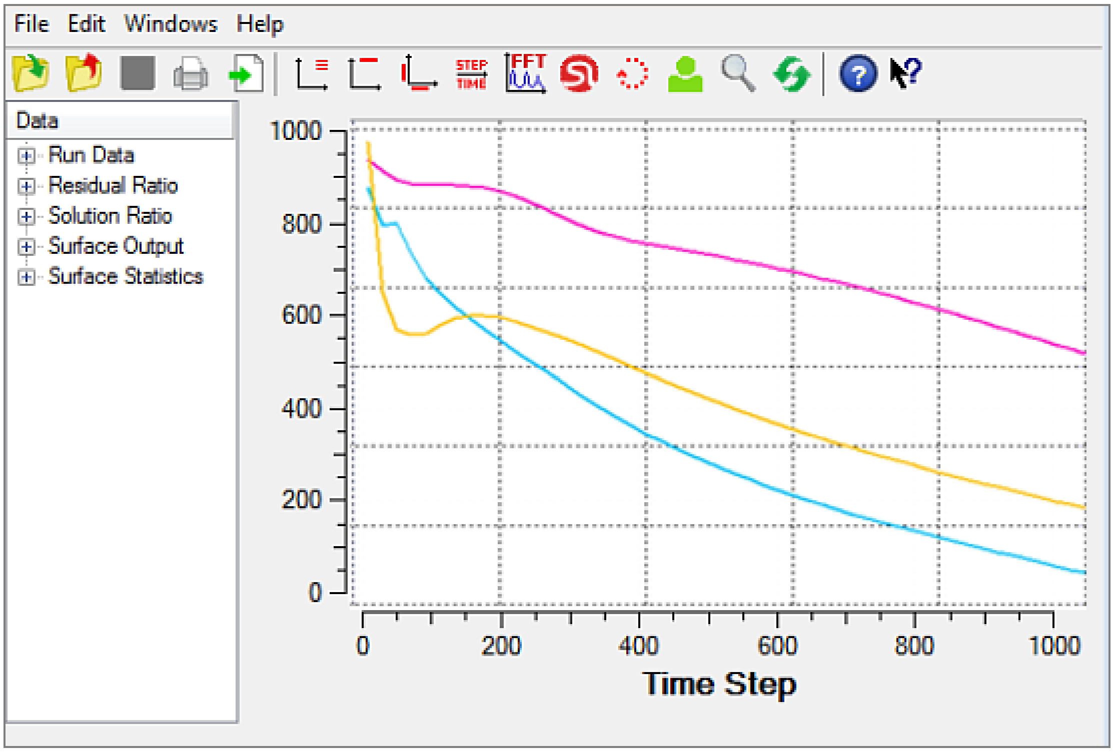

File Edit Windows Help
STEP TIME
FFT
Data
Run Data
Residual Ratio
Solution Ratio
Surface Output
Surface Statistics
1000
800
600
400
200
0
0
200
400
600
800
1000
Time Step

Figure 72:

ANSYS Interface

Overview of the ANSYS interface.

The HyperMesh ANSYS interface allows you to:

- Read an ANSYS ASCII database
- Create and edit an ANSYS ASCII database from within HyperMesh
- Preview and edit ANSYS cards after they are created
- Summarize ANSYS elements, loads, and properties
- Output an ANSYS ASCII database
- Convert an ANSYS binary results file into a HyperMesh binary results file that can be read into HyperMesh for post-processing
- Assign macro commands to modify HyperMesh's default ANSYS output

HyperMesh treats ANSYS as a card image code.

HyperMesh supports ANSYS element types for STATIC STRUCTURAL, THERMAL, and MODAL analyses using ANSYS versions 15.0, and earlier version. ANSYS 19.0 and earlier reads `.rst`, `.rmg` and `.rth` files in HyperView. HyperMesh can read `.rst`, `.rmg` and `.rth` files from ANSYS 8.1 or earlier versions. To read result files later than 8.1, use HyperView. Warnings and error messages are written to a file called `ansys.msg`. Unrecognized lines are written to a `*.hmx` file. These files are created in the directory from which HyperMesh is launched.

The ANSYS interface is only available once you have loaded the ANSYS user profile.

RBE3 Elements

In ANSYS, RBE3 card distribute the force/moment applied at the master node to a set of slave nodes, taking into account the geometry of the slave nodes as well as weighting factors.

In HyperMesh, RBE3 is an element that includes multiple 1D elements, which are defined between a single dependent (master) node and one or more independent (slave) nodes. RBE3 cards are created using the RBE3 panel (1D page).

RBE3 elements are different from Nastran RBE3 elements. In ANSYS, the RBE3 card is used to distribute the force/moments from the master node to a set of slave nodes. Whereas in Nastran, these elements define the rigid body motion. In HyperMesh, the RBE3 card is mapped as a RBE3 element. Therefore, when creating the RBE3 elements in HyperMesh for the template, the following points should be noted:

- When a RBE3 element is created in HyperMesh, the DOFs are defined for the master as well as for the slave nodes. However, since the slave node DOFs are not applicable in , they are not exported.
- By default, when a RBE3 element is created in HyperMesh, the same weight factor that is applied at the master node is also applied on all the slave nodes. The Update subpanel in the RBE3 panel allows you to modify the weight factors for the individual slave nodes. Since the master node weight factor is not applicable in ANSYS, it is not exported.

- The RBE3 elements created in HyperMesh using other solver templates, such as Nastran, are exported according to template terminology.

Parameter Arrays Exported for RBE3 Elements

Two parameter arrays are exported for each RBE3 element. The first array contains slave node IDs and the second array contains slave node weights. These arrays appears in the beginning of the exported file.

Exportation of the SLAVE# Array:

```
*DIM,SLAVE#,ARRAY,      NS,       1,        1
*SET,SLAVE#(       1,        1,        1),   ID_1
*SET,SLAVE#(       2,        1,        1),   ID_2
*SET,SLAVE#(       3,        1,        1),   ID_3
.
.
.
*SET,SLAVE#(      NS,        1,        1),   ID_NS
```

Where,

#
: HyperMesh element ID of the RBE3

NS
: Number of slave nodes within the RBE3

ID_1, ID_2, ID_3,…,ID_NS
: IDs of the slave nodes

Exportation of the WEIGHT# Array:

```
*DIM,WEIGHT#,ARRAY,      NS,       1,        1
*SET,WEIGHT#(       1,        1,        1),   WF_1
*SET,WEIGHT#(       2,        1,        1),   WF_2
*SET,WEIGHT#(       3,        1,        1),   WF_3
.
.
.
*SET,WEIGHT#(      NS,        1,        1),   WF_NS
```

Where,

#
: HyperMesh element ID of the RBE3

NS
: Number of slave nodes within the RBE3

WF_1, WF_2, WF_3,…,WF_NS
: Weighting factors corresponding to each slave node

Each RBE3 element is written using a RBE3 card with the following format:

```
RBE3, Master_ID, DOF, SLAVE#, WEIGHT#
```

Where,

Master_ID
ID of the master node

DOF
Degrees of freedom checked in the RBE3 panel for RBE3 elements with ID #. DOF is exported as follows:

U
If ux is checked

UY
If uy is checked

UZ
If uz is checked

ROTX
If rotx is checked

ROTY
If roty is checked

ROTZ
If rotz is checked

UXYZ
If ux, uy, and uz are checked

RXYZ
If rotx, roty, and rotz are checked

ALL
If all are checked

If a combination of DOFs not listed above are checked, several RBE3 cards will be exported for the same element. For example, if ux, uy, and roty are checked, three RBE3 cards will be exported.

```
RBE3, Master_ID, UX, SLAVE#, WEIGHT#
RBE3, Master_ID, UY, SLAVE#, WEIGHT#
RBE3, Master_ID, ROTY, SLAVE#, WEIGHT#
```

SLAVE#
Name of the slave node ID array

WEIGHT#
Name of the weighting factor array

Where, # is the RBE3 element ID.

The exportation location is the CP or CERIG elements.

FE Input Translator Support for RBE3

Importing the exported files

In ANSYS, RBE3 is not an element; it is created by a command called RBE3. ANSYS does not export these RBE3s as an RBE3 card, but rather as a set of constraint equations (CE). The

HyperMesh- FE input translator imports CE equations as CE equations only, and not as RBE3 elements.

Importing the HyperMesh exported files

HyperMesh writes out RBE3 elements using RBE3 cards, therefore the H_ FE input translator imports the HyperMesh exported RBE3 cards as RBE3 elements.

Tips and Techniques

Tip and techniques for working with ANSYS in HyperMesh

- The operations performed in HyperMesh can also be saved as a compact HyperMesh binary file using the **File** > **Save As** option.
- Bolt simulation configuration files are available in the ANSYS profile. In the Bolt panel, you can create general bolt, bolt with one or two washers, bolt with spider option and clips.
- Use the Mass Calc panel (Tool page) to determine the mass of your finite element model. Select the `ansys.tpl` template file through the Import tab. The mass calculation is supported for the following element type numbers: 1, 2, 3, 4, 5, 8, 10, 13, 16, 18, 20, 21, 23, 24, 25, 28, 31, 32, 33, 41, 42, 43, 44, 45, 46, 51, 53, 54, 55, 57, 58, 60, 61, 62, 63, 64, 67, 69, 70, 71, 75, 77, 78, 80, 82, 83, 87, 90, 91, 92, 93, 95, 96, 97, 98, 99, 107, 116, 117, 118, 119, 120, 121, 143, 145, 147, 148, 150, 152, 153, 154, 157, 162, 163, 164, 173, 174, 180, 181, 182, 183, 185, 186, 187, 191
- Each operation in HyperMesh results in an ASCII command that is added to a `command.cmf` file. This file is useful if your computer shuts down while working in HyperMesh. You can restore your previous work by reading this file with the command function, which can by found by clicking **File** > **Run** > **Command File**. You can delete this file if it requires too much disk space.

  The `command.cmf` file can also be used to store repetitive HyperMesh operations. If you deal with the same set of geometrical and material properties, you can record the creation of these collectors, such as steel plate, foam solids and aluminum rods, into series of command files and read them as needed.
- The ANSYS template is associated with the Composites panel. Using this panel for laminated elements, such as SHELL99, enables the following:

  Element orientation by two methods:

  - By system assigns a system to the element (set ESYS for the element).
  - By vector projects a vector on the element plane. A coordinate system, in which the x-axis is parallel to the defined vector, is created and assigned to the element as ESYS.
  - The by system axis and by angle options are not available for this template.
  - Visualization of the element orientation
  - Visualization of the ply direction on a layer-by-layer basis. Visualizes the angles defined in the R card with respect to the element orientation.
- If you change KEYOPT3 for MASS21 and KEYOPT6 for BEAM23 elements, make sure you do so on the component as well as on property collector.
- When you click the **card image** button while creating a property collector, the GENERAL card is shown at the end of the list. This card is used to card edit the properties that are not used by any element.

- The ANSYS connectivity for a second order (3-node) line element (such as BEAM189 or SURF153), is as follows: End-node-I, End-node-J and Mid-node-K. However, for creation of a bar3 element in HyperMesh (using the bar panel/bar3 subpanel), pick node A, node B, and node C in the following order: End-node-I, Mid-node-K and End-node-J, respectively. The ANSYS template will export the bar3 element using the ANSYS connectivity.
- Apply loads in local coordinate systems (ANSYS NROTAT card) using HyperMesh with the following methods:
    - The Forces panel and the Moments panel allow you to apply loads in a local coordinate system by using the local system option.
    - The set analysis option in the Systems panel, Assign subpanel can be used to assign a node to a coordinate system. A load applied to that node would be displayed correctly after exportation and importation of the ANSYS deck.

  If both options of applying loads in local coordinate systems are used for the same node, the first method (via the Forces and Moments panels) takes precedence.

Pressure Load on Beam Elements

HyperMesh supports pressure load on bar and rod elements, which means you can import the pressure load [SFBEAM] applied on ANSYS BEAM elements.

The following items are supported under this feature:

- Import pressure load applied on BEAM3, BEAM4, BEAM23, BEAM44 elements. Import restrictions are explained below.
- Use the Pressure panel to apply SFBEAM load on beam elements.
- Export pressure load applied on BEAM elements in HyperMesh as SFBEAM load in ANSYS solver deck. The Export feature is explained in detail below.

Pressure Load Import

SFBEAM is the keyword command in ANSYS decks to apply pressure load on BEAM elements. This command line has following arguments:

```
SFBEAM,ELEM,LKEY,Lab,R5.0,DIOFFST,DJOFFST
```

ELEM

Element ID and Lab is always pres(pressure).

LKEY

Load key number which determines the face of the beam where the pressure load is applied.

Refer to the ANSYS solver documentation for information on the face of the beam and corresponding LKEY number as it changes from BEAM to BEAM.

Restrictions

- The HyperMesh-ANSYS interface presently supports the following LKEY values for each BEAM element. You can import SFBEAM load with these LKEYs.

Table 4:

ELEMENT TYPE	SUPPORTED LKEYs
BEAM3	1, 3, 4
BEAM4	1, 2, 4, 5
BEAM23	1, 3, 4
BEAM24	1, 2, 4, 5
BEAM44	1, 2, 4, 5
BEAM54	1, 3, 4
BEAM188	1, 2, 4, 5
BEAM189	1, 2, 4, 5

- DIOFFST, DJOFFST, which define offset values for pressure value at node I and J, are not supported in HyperMesh.
- SFLOAD, which applies a pressure load that varies across the length of the beam, is not supported in HyperMesh 9.0. Only pressure load with constant value can be imported.

Pressure Applied on BEAM Elements

- You can apply a pressure load in HyperMesh on any bar2, bar3 and rod, elements. Only the pressure loads which are applied on supported BEAM element types can be exported.
- If no option for direction is selected, pressure is applied on 'element – y' direction by default. You can use the options available in the Pressure Load panel to specify the direction of the load.
- The ANSYS solver allows pressure on beam elements in only specified faces of the elements, as mentioned by LKEYS. In other words, pressure on beam elements is applied in element co-ordinate system. Keep this in mind before determining the direction of the pressure.
- You cannot apply a pressure load that varies across the length of the BEAM even though it is allowed in the ANSYS Solver.
- Load offsets at node I and J are not supported.

Export Pressure Load on BEAM Elements

- Pressure on beams is exported as SFBEAM cards.

- Pressure only applied on BEAM3, BEAM4, BEAM23, BEAM44, BEAM54, BEAM188, BEAM189 elements will be exported.
- Pressure applied in any arbitrary direction is resolved in to corresponding element systems, and relevant LKEYs will be assigned while exporting. When you import the deck back to HyperMesh, you will see resolved loads.

EXODUS Interface

Overview of the EXODUS interface.

The EXODUS interface enables you to store a large amount of topological and result data in a binary format for finite element analysis.

In HyperMesh, the import and export of EXODUS files is supported in the EXODUS interface. An EXODUS file (`*.ex`, `*.exo`, `*.ex2`) can be read using the Import tab, and the finished model can be written using the Export tab.

EXODUS supports the following types of entities, which can be created from the Solver Browser:

- Blocks (Component)
- Control Cards
- Coordinate Frames (System)
- Functions (Curve)
- Loads
- Materials
- Plots
- Sets
- TiedJoints (Group)

Feko Interface

The Feko interface provides a pre-processing environment for preparing and editing Feko mesh element and material data.

In HyperMesh you can import and export Feko mesh element and material data. To import, create, modify or export Feko mesh and material data you have to load the Feko user profile in HyperMesh.

Import and Export Guidelines

Guidelines for importing and exporting mesh and material data in the Feko interface.

- The Feko interface can import Feko mesh elements and material settings and assignments from Feko-generated `*.fhm` and `*.inc` files.

  > **Note:** When mesh data is exported from Feko into the `*.fhm` file format, an `*.inc` file with the same file name and containing the material data is created automatically.

  During import of such an `*.fhm` file, HyperMesh automatically imports the `*.inc` file with the same file name if it is found in the same directory.
- The Feko interface can export Feko mesh elements and materials settings and assignments into `*.fhm` and `*.inc` files that can be imported into a CADFEKO model (using the mesh import menu) or an EDITFEKO `*.pre` file (using the IN card). The Feko components would automatically import the materials data from the `*.inc` file if it is found in the same directory as the provided `*.fhm` file.
- The Feko interface supports Feko wires segments, triangles and tetrahedra. Straight and curvilinear wire segments in Feko are represented as Bar2 and Bar3 elements respectively in HyperMesh. Flat and curvilinear triangles in Feko are represented as Tria3 and Tria6 elements respectively in HyperMesh. Feko tetrahedra are represented as Tetra4 elements in HyperMesh.
- Each individual mesh label (Wire segments, Face, or Region) is represented as a unique Component in HyperMesh. HyperMesh Properties are used to assign Materials to these Components.
- Supported Feko media include (anisotropic) dielectric media (with optional magnetic properties) and metallic media. Feko's default Free space, Perfect electric conductor, and Perfect magnetic conductor media are also supported.

LS-DYNA Interface

Overview of the LS-DYNA interface.

HyperMesh provides a complete pre-processing environment for preparing LS-DYNA data decks for analysis.

HyperMesh can read existing LS-DYNA decks, create a model, display and edit LS-DYNA cards as they will look in the deck, and write a deck for analysis.

To create LS-DYNA decks in HyperMesh, you must load the LS-DYNA user profile with the appropriate template to access the full pre-processing capability.

Import and Export

- HyperMesh support LS-DYNA solver versions till 971_R9.0.
- Solver specific import options are available during import in the Solver Options tab.
- HyperMesh supports LS-DYNA Dummy models with the Primer and LSTC dummy information format. HyperMesh writes out the dummy information on Primer format.
- Most ID's in the solver deck are preserved in HyperMesh. If a keyword is not supported in a dedicated HyperMesh entity to ensure its unique ID-Pool, then HyperMesh renumbers those keywords when ID conflicts are detected. The new ID's are posted during the import process.
- The LS-DYNA interface supports a smart, reliable FE input reader that warns you when your input deck contains unsupported fields and unsupported data lines.
- HyperMesh supports parameterized IDs for Components, Materials, Properties, and Curves.
- HyperMesh supports undefined entities, meaning, entity IDs are referenced in keywords (for example a Material ID in a *PART) but not defined in the deck. In this case, HyperMesh creates a default card (for example a material of type *MAT_ELASTIC is then created) in order to preserve the ID. This keyword has the **Defined** checkbox toggled off and is automatically not exported.

Duplicate ID's

- Several LS-DYNA keywords are mapped to one HyperMesh entity in some instances. By default the LS-DYNA interface doesn't allow duplicate ID within the same HyperMesh entity with exception of elements while LS-DYNA allows duplicate ID's across cards mapped to one HyperMesh entity. In HyperMesh ID flexibility similar to LS-DYNA can be enabled by switching on Duplicate ID option in Preferences menu.
- Duplicate ID's are supported for the following HyperMesh entities in the LS-DYNA user profile: elements, properties, entity sets, sensors, Load collectors and control volumes.

Mass Calculations

- Mass supplied by *PART_INTERIA card is used instead of calculating the mass based on the individual elements. Also, mass calculations include the mass supplied on the *CONSTRAINED_NODAL_RIGID_BODY_INERTIA cards.
- Shell element thickness for volume calculation is one of the following:
    - Thickness on the first node for uniform thickness shells
    - Average thickness at three or four nodes for non-uniform thickness shells

- The thickness values come from the *SECTION_SHELL card, unless a *ELEMENT_SHELL_THICKNESS card is defined for an element. If an *ELEMENT_SHELL_THICKNESS card is defined, its thickness values override the thickness values from the *SECTION_SHELL.
- Integrated beams have an area equal to the average of the two end areas. Resultant beams use the area entered on the *SECTION_BEAM card. The volume is calculated by multiplying the length of the beam with the *SECTION_BEAM card area. Discrete beams use the volume supplied by the *SECTION_BEAM card. In all cases, if an *ELEMENT_BEAM_THICKNESS card is defined for an element, then the element values override the *SECTION_BEAM values.
- Only element masses are considered. Other mass specifications, such as on a rigid wall card, are ignored.

Recommended Process

Editing an LS-DYNA Model to Add Cards not Supported

Use unsupported cards with the LS-DYNA model by adding them in HyperMesh. There is no need to use a text editor. Select **unsupp_cards** in the Control Cards panel. You can then enter the cards in the pop-up text editor. Use caution regarding formatting and card validity. Care should also be taken if any of the cards point to entities, such as cards pointing to sets and parts. These cards are stored as text and pointers are not considered. When importing an LS-DYNA mode, any cards that are encountered that are not supported are written in this section, therefore they are exported along with the remaining model.

Blanks

In the Card Editor all of the attribute fields are supported as Blanks. You must click the field and input the value.

LS-DYNA Mass Calculation

Mass calculation for LS-DYNA is accessible from the Summary panel (Post page).

The mass reported is not simply calculated by Density x Volume for each part. It follows the many LS-DYNA requirements to handle rigid body mass, non-structural mass, and lumped mass.

Contributing Total Mass factors: totalmass = structuralmass + lumpedmass + nonstructuralmass + rigidbodymass + transferredmass + distributedmass

Structural Mass

Volume x density; except in case of *PART_INERTIA in which it is also the total mass.

Lumped Mass

Accounts for contributions from *ELEMENT_MASS, *ELEMENT_MASS_NODE_SET, and *ELEMENT_INERTIA. This does not take into account the transfer of lumped mass to rigids.

Non structural Mass (NSM)

Accounts for contributions from ELEMENT_MASS_PART, ELEMENT_MASS_PART_SET, and NSM in *SECTION. This does not take into account the transfer of lumped mass to rigids.

RigidBodyMass Mass

Mass of *CONSTRAINED_NODAL_RIGID_BODIES.

Transferred mass

Mass transferred from deformable nodes to rigid materials. This includes lumped mass transferred from rigid or deformable nodes to the rigid materials.

- For rigid material, this is the mass gained from deformable (+).
- For deformable parts, this is the mass lost to rigid material (-).

Distributed Mass

Mass distributed from nodal rigid bodies to free nodes.

Engineering Mass

Mass of the part that most closely matches its real engineering meaning. The engineering mass is the most useful for possible mass adjustments. Engineering mass is the sum of structural, non-structural, and lumped mass.
Engineering mass exceptions:

- For PART with PART_INERTIA:
    - Slave CRB of this part should have 0 mass.
    - Mass of the part should be equal to TM (if Iflag=1 => transfer mass from slave to master).
- For *CONSTRAINED_NODAL_RIGID_BODY_INERTIA:
    - The mass of the slave nodes on the connected part should not be taken into account and excluded.

LS-DYNA Part Mass

The mass listed in d3hsp, where you also have COG information.
Total mass is also obtained with the following calculations:
Total mass = LS-DYNA part mass + lumped mass (for deformable parts as rigid part already include lumped mass) + CNRB mass.
Center of Gravity (CG) is computed from the total mass (for each part); the inertia are computed from the total mass.

Deck Export

Supported LS-DYNA files that can be exported.

- LS-DYNA v971_R7.0, v971_R6.1, v971, v970 and v960 input files in Keyword format.
- By default, the LS-DYNA user profile outputs v971_R6.1 `.key` files.
- Two templates are also provided to output the defined curves in the database:
    - To output curves in Keyword format, use the `curves.key` template
    - To output curves in Structured format, use the `curves.seq` template

MADYMO Interface

Overview of the MADYMO interface.

The MADYMO interface supports both the import and export of MADYMO files. A MADYMO `.xml` file can be read using the Import tab. The model will be displayed. The model can be altered and the finished model can be written using the Export tab.

User Profiles

The MADYMO interface has two sub-profiles.

- MADYMO63
- MADYMO70

Importing a MADYMO Model

The MADYMO input file is based on XML and must be converted to be used in HyperMesh. If HyperMesh supports the equivalent MADYMO entities, the mapping is straightforward. If HyperMesh does not provide the entity, a transformation rule is used.

Several MADYMO entities are mapped on the same type of HyperMesh entities; for example ACTUATOR, CONTROLLER, OPERATOR, SENSOR, SIGNAL, STATE and SWITCH are all mapped on sensor. This puts a restriction on the use of names for these entities; all names of the MADYMO entities mapped on the same HyperMesh entity must be unique, for example, a MADYMO SENSOR cannot have the same name as a MADYMO CONTROLLER. During the FE import process HyperMesh will rename the entities as required to avoid duplicate names.

Exporting MADYMO Models

During export only the nodes that are in use according to MADYMO are exported. HyperMesh temporary nodes are not exported even if they are used in a node set or a MADYMO node LIST. It is advised to remove all temporary nodes before exporting the model.

In general, the IDs of entities in the HyperMesh model correspond with the IDs in the MADYMO `.xml` file after exporting the model. An exception to this rule is when PLANES in MADYMO are a subtype of the entity SURFACE that contains also the subtypes CYLINDER and ELLIPSOID. The IDs must be unique within the SURFACE type. In HyperMesh these entities are mapped onto two different entities (mbplanes and ellipsoids) with their own unique numbering. Therefore, during export the mbplanes are renumbered to get unique MADYMO IDs for the PLANEs.

Marc Interface

Overview of the Marc interface.

Follow these guidelines when creating a model for use with the HyperMesh Marc interface.

- HyperMesh treats Marc as a card image code.
- The Marc interface is based on Marc version 12; 2008r1 release.
- The current interface supports linear and nonlinear static analysis.

For 2D models, use the template `stress2d.tpl`. For 3D models, use the template `stress3d.tpl`. 2D and 3D models cannot be combined. 2D models should use the xy-plane.

Warnings and error messages are written to a file called `marc.msg`. Unrecognized cards are written to an `*.hmx` file. These files are created in the same directory from where HyperMesh is launched.

Any data which is currently not supported in HyperMesh is stored as unsupported data and is added to the model if the model is exported from HyperMesh.

The Marc solver interface supports PATRAN exported in Marc format.

Nastran Interface

Overview of the Nastran interface.

- Two sub-profiles are available for the Nastran interface:
  - MSC Nastran
  - NX Nastran
- During input, the Nastran interface assumes that the continuation line always follows the line before, therefore no continuation cards are needed in the input file. During export, HyperMesh writes "+" or "*" as a continuation card to ensure that the continuation line follows the reference card.
- Warnings and error messages are written to an open text file for your review. You can save this file. All the lines in the input file that are unrecognized by the translator are written to unsupported cards, which can be edited.
- The Nastran input translator supports single, double, free, and fixed formats.
- Two templates are available for export and viewing (editing) Nastran cards. The Nastran/general template outputs all cards in a single precision, fixed format. The nastran/generallf template outputs System, Grid, Element, Load, Subcase, Property, and Material cards in a fixed, double precision (Long) format.
- By selecting the Nastran user profile, you can work with pre-defined panels that are more specific to Nastran usage.
- Symbolic substitution is supported as parameters in the MSC Nastran user profile for properties and materials. All the attributes in materials and properties can be parameterized.
- Before exporting or editing Nastran cards, a Nastran template must be loaded. Setting the user profile to Nastran automatically loads the Nastran template.
- HyperMesh reads and writes certain HyperMesh commands in the Nastran bulk data file as comments. These comment cards enable HyperMesh to preserve pre-defined preferences across sessions. It is strongly suggested that you do not hand edit these cards.

Thermal Surface Elements for Nastran

HyperMesh supports the modeling of boundary condition surface elements used in heat transfer solution sequences in Nastran (such as SOL400).

HyperMesh currently has full support of CHBDYE elements, and partial support of CHBDYG elements.

Thermal Surface Element	Import	Export	Edit	Create
CHBDYE	Y	Y	Y	Y
CHBDYG	Y	Y	Y	N

These cards are represented as slave elements as part of a CONDUCTION, CONVECTION, or RADIATION group in HyperMesh, and are created through the Interfaces panel.

Convection

- Group type = CONVECTION
- Group Card Image = PCONV
- Slave element type = CHBDYE/CHBDYG
- Additional associated cards = CONV: Represented as a checkbox inside slave element card preview.

Radiation

- Group type = RADIATION
- Group Card Image = RADM
- Slave element type = CHBDYE/CHBDYG
- Additional associated cards = VIEW: Represented as a loadcollector in HyperMesh

CHBDYE Side Conventions for 3D Elements

To automatically calculate the appropriate value of the SIDE parameter for newly created CHBDYE slaves on solid elements according to the specified face nodes and break angle, go to the Interfaces panel, Add subpanel, and set the slave option to **face**.

CHBDYE Side Conventions for 2D Elements

For CHBDYE slaves on 2D shell elements, HyperMesh will automatically assign a SIDE parameter value of 1 (top) or 6 (bottom), depending on CHBDYE normal direction. Use the Normals panel inside the Tool menu page to change CHBDYE normal direction.

Regardless of base shell element normal direction, visualization of CHBDYE normal in HyperMesh will always identify the direction the CHBDYE slave is facing.

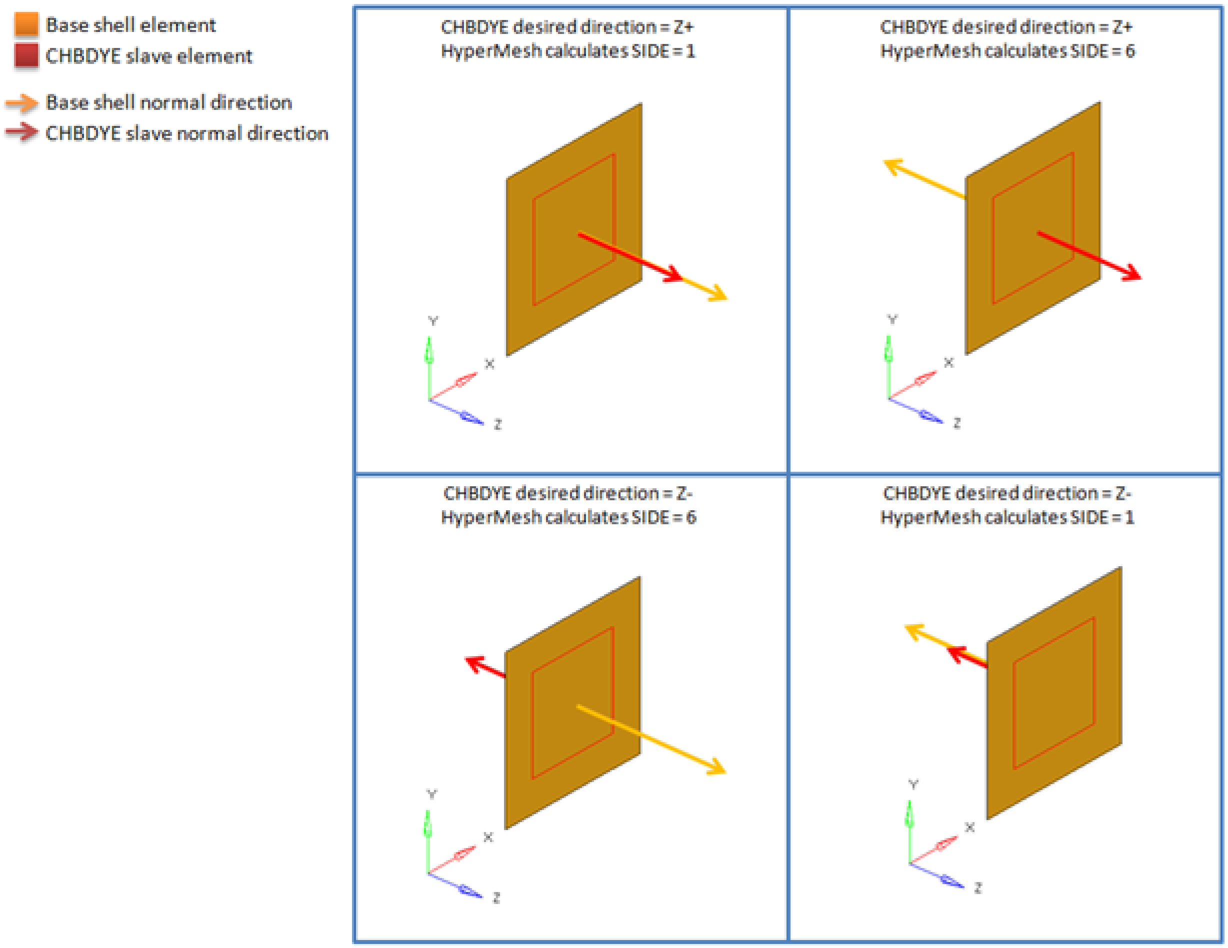


Figure 73: Resulting SIDE Value for CHBDYE Slave on 2D Shell Element

CHBDYG Side Conventions for 2D Elements

Visualization of CHBDYG normal in HyperMesh identifies the direction the CHBDYG slave is facing. CHBYG node order is adjusted accordingly. Use the Normals panel inside the Tool menu page to change CHBDYG normal direction.

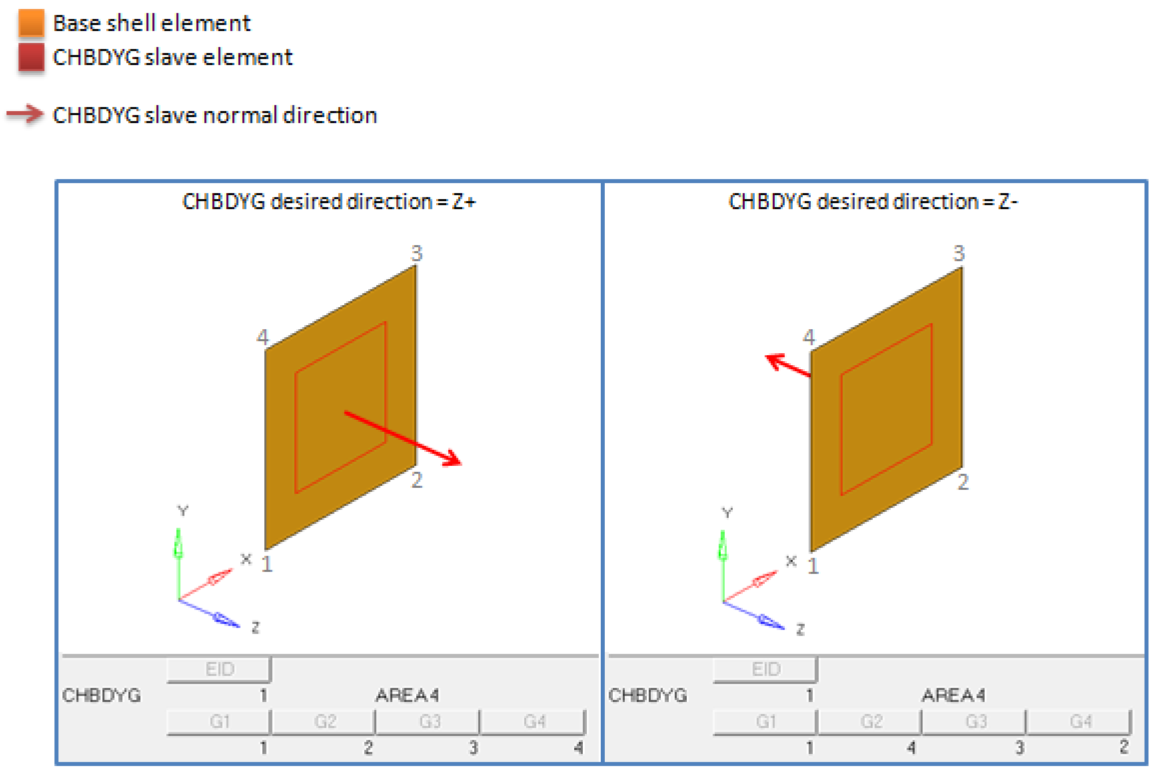


Figure 74: Resulting Node Order for CHBDYG Slave on 2D Shell Element

PAM-CRASH 2G Interface

Overview of the PAM-CRASH 2G interface.

HyperMesh provides a complete pre-processing environment for preparing PAM-CRASH 2G data decks for analysis. HyperMesh can read existing PAM-CRASH 2G decks, create a model, display and edit PAM-CRASH 2G cards as they will look in the deck, and write a deck for analysis. Although HyperMesh also offers limited post-processing capabilities in results translation you are encouraged to exclusively use HyperView. To create PAM-CRASH 2G decks in HyperMesh, you must load the Pamcrash2G user profile with the appropriate template to access the full pre-processing capability.

Import and Export

- PAM-CRASH 2G version 2012 - 2017.
- The Pamcrash2G input translator also supports reading older Pamcrash2G files 2002 - 2011 and map into the loaded template.
- Pamcrash and Pamcrash2G 2002 - 2011 templates are available in the installation and can be loaded. However they are no longer maintained.
- During import all the GES associated with the solver keywords are mapped to entity sets in HyperMesh and associated with the solver entity. They are exported back as GES during export.
- In addition to the custom export options available in the Export Browser, several templates are available to export specific solver entities only.

Table 5:

Template Name	Description
control_modeldoc	Exports only the model documentation.
ctrlvol	Exports only the airbag and chamber definitions.
groups_nsmas	Exports only the non-structural mass cards.
groups_rigwa	Exports only the rigid wall cards.
groups_slint	Exports only the sliding interface cards.
crosssection	Exports only the cross section cards.
load	Exports only the load and load collector cards.
materials	Exports only the material cards.
property	Exports only the property cards.
sensor	Exports only the sensor cards.
sets	Exports only the PAM-CRASH 2G groups.

Template Name	Description
curves	Exports only the function cards.
parts	Exports only the part cards.

Mass Calculations

- The mass of each element is calculated by density * volume. Density is retrieved from the material associated with the element's component. Several assumptions are made in mass calculation based on PAM-CRASH 2G solver.
- For rigid bodies (RBODY) of type 3 with a user-imposed center of gravity and mass and inertia properties, the structural mass of the PART components are replaced by the rigid body mass. All other keywords besides the PART keyword are neglected in the RBODY type 3 mass calculation.
- The following table summarizes how each supported element except shells and solid in the HyperMesh PAM-CRASH 2G interface performs these calculations:

PAM-CRASH 2G Keyword	Area	Volume	Mass	Notes
MASS /	0	0	$M = \sqrt{M_x^2 + M_y^2 + M_z^2}$	
SLIPR /	0	0	0	
RETRA /	0	0	0	
SENPT /	0	0	0	
PLINK /	0	0	0	
RBODY /	0	0	$RBODY_MASS	Rigid body type 3.
NODCO /	0	0	0	
SPRING /	0	0		Material type 220.
SPRGBM /	0	0		Material type 223.
BAR /	0	$V = A_{bar} * Length$ $V_{205} = 0.0$	$M = p * V$ $M_{205} = Length * \frac{Mass}{Length}$	Material types 200, 201, 202, 203, 204, and 205.
JOINT /	0	0	0	
KJOIN /	0	0	0	
NSMAS	0	0	MASS	Non-Structural Mass Definition

PAM-CRASH 2G Keyword	Area	Volume	Mass	Notes
				- Standard Input Format
NSMAS2	0	0	MASS / MLEN / MARE / MVOL	Non-Structural Mass Definition - Alternate Mass Distribution Method

Permas Solver Interface

Overview of the Permas interface.

The HyperMesh Permas interface enables you to:

- Read in a Permas `.dat` file
- Read and write `.gz` files
- Specify linear, nonlinear static and frequency analyses
- Preview and edit Permas cards
- Output an HyperView `.h3d` file for post-processing

HyperMesh supports Permas `.dat` files for direct import. Permas `.uci` command files are not supported.

The Permas interface supports names up to 40 characters in length.

MPC ID Pools

The HyperMesh Permas Interface allows multipoint constraints (MPC) to use the same the ID as other elements, such as beams, shells or solids within HyperMesh. In other words, it can be said that selected elements and MPCs can use different "ID pools."

Some restrictions apply. To explain, a list of MPC types and which HyperMesh entity they are mapped to is provided:

Table 6:

HyperMesh entity	MPC type
rigid	ASSIGN
	RIGID
	SAME
	JOIN
rbe3	WLSCON
equation	GENERAL
group	ISURFACE
	WLSSURFACE
	WLDSURFACE

While rigid and rbe3 elements use the same ID pool in HyperMesh, equations and groups have separate ones. Thus, it is possible to create duplicate IDs for MPCs as well. To resolve this, during export a solver option is available in the Export tab to renumber MPCs if duplicate IDs are present. Renumber MPCs if

IDs are duplicated. This option will check for the highest rigid/rbe3 ID and renumber equation entities accordingly. Groups are not included in the renumbering functionality as contact IDs would be affected by renumbering.

Due to the mapping scheme explained above, it is possible to import MPCs with duplicate IDs, for example, MPC JOIN and GENERAL equation and groups although Permas does not allow it.

Before working with your model, ensure that the Permas user profile is loaded. The user profile provides a customized HyperMesh environment for working with Permas.

OptiStruct Interface

Overview of the OptiStruct interface.

- The OptiStruct user profile is designed to support OptiStruct, version 2019, although several entries are unsupported.
- Elements of different types (solids, shells, bars, and so on) should be organized into separate components. This is most important for shell and solid elements where the component card image contains the referenced property definition.
- On importing an input file:
    - The interface requires that continuation cards follow the referencing card. An error message is issued if the continuation card does not follow the referencing card.
    - Warnings and error messages are displayed in an Import Process Messages pop-up window and are also written to a file in the current run directory called `optistruct.msg`.
    - If a line in the input file that is not a continuation line and starts with a keyword that is not recognized or supported, then the entire card gets written to the appropriate unsupported card section (CTRL_UNSUPPORTED_CARD, SUBCASE_UNSUPPORTED or BULK_UNSUPPORTED_CARD). Also, if the name of a `PARAM` or `DOPTPRM` entry is unrecognized, it is stored as an UNSUPPORTED_PARAM or UNSUPPORTED_DOPTPRM, respectively.
    - If the continuation line for a supported card is formatted incorrectly or if a certain feature for a recognized keyword is not supported, then the corresponding line gets written to the `.hmx` file.
- The bulk data input translator supports free, fixed and large field formats.
- Two templates are available for export and viewing (editing) OptiStruct cards. The OptiStruct template outputs all cards in single precision, fixed format. The optistructlf template outputs System, Grid, Element, Load, Subcase, Property, Material, and Optimization cards in fixed, double precision (Long) format.
- HyperMesh reads and writes certain HyperMesh commands to the OptiStruct input files as comments. These comment cards enable the preservation of pre-defined preferences across sessions. It is strongly suggested that you do not hand edit these cards.
- All settings necessary to interface with OptiStruct can be loaded by their respective user profiles.

Radioss Interface

Overview of the Radioss interface.

HyperMesh provides a complete pre-processing environment for preparing Radioss data decks for analysis.

HyperMesh can read existing Radioss decks, create a model, display and edit Radioss cards as they will look in the deck, and write a deck for analysis.

To create Radioss decks in HyperMesh, you must load the Radioss user profile with the appropriate template to access the full pre-processing capability.

Import and Export

- HyperMesh supports Radioss solver versions for import and export till Radioss 2018.
- Solver specific import options are available during import in the Solver Options tab.
- HyperMesh supports Radioss Dummy models.
- Most ID's in the solver deck are preserved in HyperMesh. If a keyword is not supported in a dedicated HyperMesh entity to ensure its unique ID-Pool, then HyperMesh renumbers those keywords when ID conflicts are detected. The new ID's are posted during the import process.
- The Radioss interface supports a smart, reliable FE input reader that warns you when your input deck contains unsupported fields and unsupported data lines.
- HyperMesh supports undefined entities, meaning, entity IDs which are referenced in keywords (for example a Material ID in a PART) but not defined in the deck. In this case, HyperMesh creates a default card (for example a material of type elastic is then created) in order to preserve the ID. This keyword has the **Defined** checkbox toggled off and is automatically not exported.

Duplicate ID's

- Several Radioss keywords are mapped to one HyperMesh entity in some instances. By default the Radioss interface does not allow duplicate IDs within the same HyperMesh entity, with the exception of elements. Radioss does allow duplicate IDs across cards mapped to one HyperMesh entity. In HyperMesh, ID flexibility similar to Radioss can be enabled by switching on the Duplicate ID option in the Preferences menu.
- Duplicate IDs are supported for the following HyperMesh entities in the Radioss user profile: elements, properties, entity sets, and sensors.

Rigid Body Management

Any RBODY created with less than 10000 slave nodes is shown with the spider connecting the master node to the slave nodes. If the RBODY has more than 10000 slave nodes, then it is shown with a single link connecting the master node to one of the slave node. All other options are similar for both.

Mass Calculation

The mass of each element is calculated by density * volume. Density is retrieved from the material associated with the element's component. Currently mass calculation for RBODY, RBE3, RBE2 is not supported.

Samcef Interface

Overview of the Samcef interface.

The Samcef interface enables you to:

- Specify linear analysis (module ASEF)
- Specify a linear, nonlinear static analysis
- Export a `.dat` file.
- Read and write a `.dat` file
- Preview and edit Samcef cards

In addition:

- Samcef `.sdb` files are not supported
- Warnings and error messages are written to a file named `samcef.msg`. Unrecognized lines are written to a `*.hmx` file. These files are created in the same directory from where HyperMesh is launched.

The following commands are supported:

- Nodes (.NOE)
- Elements (.MAI, .MCE MEAN, .MCC MEAN, .MCE BUSH)
- Property (.PHP, .BPR, .MCC BUSH)
- Material (.MAT)
- Loads (.CLM)
- System (.FRA)
- Output request (.SAI)
- Hypothesis (.HYP)
- Entity Set (.SEL NOE, .SEL MAI)
- Contactsurfs (.SEL FACE)
- Composite definition (.PLI, .LAM, .ETA)
- Contact definition (.MCT, .STI)

Contact .MCT

Link between HyperMesh keywords and the Samcef contact .MCT parameters.

For a complete definition of the card .MCT, refer to the Samcef help.

Selection of "contact resolution method" *[Option_number]*

"Uncoupled" (default)
: Write OPT 3

"Coupled"
: Write OPT 2.

"Sliding magnitude" *[Contact_characteristics]*. You can control this option by two choices:

« Moderate » (default)
Nothing is written on MCT Card.

« Large »
NLIM -1 must be written on the MCT definition

"Initial gap" *Contact_characteristics*. You can control this option by two choices:

"As mesh is" (default)
Nothing is written on the MCT definition.

"No initial gap"
UN3 1 must be written on the card definition.

"Contact offset" *Contact_characteristics*. You can control this option by two choices:

"As mesh is" (default)
Nothing is written on the MCT definition.

"Contact offset"
You must enter a value XX; DMIN XX must be written on the card definition.

"Kind of contact" *Contact_characteristics*. You can control this option by two choices:

"Standard" (default)
Nothing is written on the MCT definition.

"Normal direction glue"
OPCO 2 must be written on the card definition.

Selection of "Curvature smoothing contact faces" *[Smoothing_characteristics]*

"Non activated" (default)
Nothing is written by default on MCT card.

"Activated"
Write KSMO 1 on MCT card.

Selection of "Curvature smoothing contact faces" *[Smoothing_characteristics]*

"Non activated" (default)
Nothing is written by default on MCT card.

"Activated"
Write KSMO 1 on MCT card.
Blank text line *Additionnal_user_text*.

Enter free text on the current panel, on the Samcef MCT card. The text is saved with the contact definition in HyperMesh.

Examples:

```
.MCT I 10 GROUP 17 GTAR 23 OPT 3 ! Contact default
.MCT I 20 GROUP 65 GTAR 12 OPT 2 NLIM -1 UN3 1 DMIN -0.0245 KSMO 1 ! Advanced contact
```

.MCT I 30 GROUP 45 GTAR 75 OPT 2 UN3 1 OCPO 2 KSMO 1 ! Glue contact

Contact .STI

Link between HyperMesh keywords and the Samcef contact .STI parameters.

For a complete definition of the card .STI, refer to the Samcef help.

Selection of “Node project” [Option_number]

- “Non activated‟ by default ; write PROJ 0
- “Activated”; nothing to write on STI card

Example:

.STI I 1 GROUP 17 23 PROJ 0 ! Glue default

.STI I 2 GROUP 35 64 ! Advanced glue

Entities and Solver Interfaces 3

A solver interface is made up of a template and a FE-input reader.

This chapter covers the following:

Altair HyperMesh Entities

Entities can have none or multiple card images associated to them. Card images are defined within a solver interface template and allow for creation, editing, and deletion of a solver card within a HyperMesh model. Entities contain two types of data; data names and attributes.

Data names

Part of the entity data structure itself and are available to all instantiations of the entity regardless if the entity has an associated card image or not.

Attributes

Additional data, defined in a solver interface template, which are necessary to store solver specific data for a card image associated with an entity.

Entities can be subdivided into Collectors, Collected Entities, Named Entities, Optimization Entities, and Morphing Entities.

Solver Interfaces

A solver interface is made up of a template and a FE-input reader. A template defines the mapping between solver cards and entities, the attributes necessary to store data for solver cards, and the format which the solver cards are exported from a HyperMesh database. FE-input readers perform the function of reading solver decks and importing solver cards into the appropriate entities with the appropriate card images, data names, and attributes set as defined by the template. Furthermore, FE-input readers require template attribute definitions to perform their tasks.

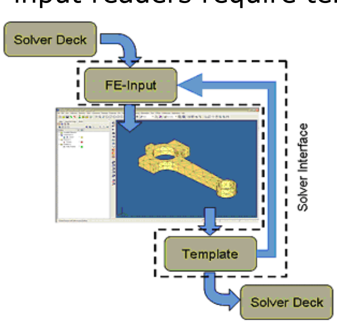


Figure 75: Schematic of the Solver Interfacing Architecture

Templates

Below is an example that demonstrates the interaction between entities and templates with data names and attributes for an OptiStruct `MAT1` card.

(1)	(2)	(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
MAT1	*MID*	*E*	*G*	*NU*	*RHO*	*A*	*TREF*	*GE*	
	ST	*SC*	*SS*						

The HyperMesh material named entity has data names of name and ID. Therefore the template would also have to define attributes for *E*, *G*, *NU*, *RHO*, *A*, *TREF*, *GE*, *ST*, *SC*, and *SS* in order to completely define and store the OptiStruct `MAT1` solver card within a HyperMesh material named entity with a `MAT1` card image. Templates define attributes using the `*defineattribute()` command. The example template code uses the `*defineattribute()` command to define these attributes.

In order to associate the OptiStruct `MAT1` solver card to the HyperMesh material named entity with a `MAT1` card image the template would contain a `*materials(MAT1)` definition block to define this association. Within the `*materials(MAT1)` definition block a `MAT1` card image would be defined using a `*beginmenu()` definition block . This `*beginmenu()` definition block is read every time a materials named entity with a `MAT1` card image is card edited using the card editor within HyperMesh. In addition, an export format for the OptiStruct `MAT1` solver card would be defined using a `*format()` definition block within the `*materials(MAT1)` definition block. This `*format()` definition block is read every time an export of the HyperMesh database is requested which contains a materials named entity with a `MAT1` card image. Example template code which performs these definitions for the OptiStruct `MAT1` solver card is given.

In the example below you can find this example template code in `[Install Directory]\hm\examples\templates\ExampleTemplate.tpl`.

```
*codename(ExampleTemplate,100)
//MAT1 Attributes
*defineattribute(MAT1,1,integer,none)
*defineattribute(E,2,real,none)
*defineattribute(G,3,real,none)
*defineattribute(NU,4,real,none)
*defineattribute(RHO,5,real,none)
*defineattribute(A,6,real,none)
*defineattribute(TREF,7,real,none)
*defineattribute(GE,8,real,none)
*defineattribute(ST,9,real,none)
*defineattribute(SC,10,real,none)
*defineattribute(SS,11,real,none)
//Materials Named Entity - MAT1 Card Image and Export Format
*materials(MAT1)
 //MAT1 Card Image
 *beginmenu()
```

```
          *menustring("MAT1    ")
          *menufield("ID",integer,id,8)
          *menufield("E",real,$E,8)
          *menufield("G",real,$G,8)
          *menufield("NU",real,$NU,8)
          *menufield("RHO",real,$RHO,8)
          *menufield("A",real,$A,8)
          *menufield("TREF",real,$TREF,8)
          *menufield("GE",real,$GE,8)
          *menulineend()
          *menustring("        ")
          *menufield("ST",real,$ST,8)
          *menufield("SC",real,$SC,8)
          *menufield("SS",real,$SS,8)
          *menulineend()
 *endmenu()
 //MAT1 Export Format
 *format()
          *string("MAT1    ")
          *field(integer,id,8)
          *field(real,$E,8)
          *field(real,$G,8)
          *field(real,$NU,8)
          *field(real,$RHO,8)
          *field(real,$A,8)
          *field(real,$TREF,8)
          *field(real,$GE,8)
          *end()
          *string("        ")
          *field(real,$ST,8)
          *field(real,$SC,8)
          *field(real,$SS,8)
 *end()
*output()
```

FE-Input Readers

FE-input readers perform the function of reading solver decks and importing solver cards into the appropriate entities with the appropriate card images, data names, and attributes set as defined by the template. Furthermore, FE-input readers require template attribute definitions to perform their tasks.

In the FE-Input code example below, you can find this example template code in `[Install Directory]\hm\examples\feinput\ExampleFEInput.cxx`.

```
#include <iostream>
#include <fstream>
#include <cstring>
#include "hmlib.h"
#include "hminlib.h"
using namespace std;
//Material Data Structure
int nummaterials;
struct materials {
 char name[12];
 int id;
 double E;
 double G;
 double NU;
 double RHO;
 double A;
 double TREF;
```

```
 double GE;
 double ST;
 double SC;
 double SS;
} material[100];
//Function Prototypes
int get_data(char *fileptr);
entityfunctionptr HM_getfunction(int function, HM_entitytype entities);
int HM_getMaterials();
int main(int argc, char *argv[])
{
 /* The main function calls get_data to process the data in the solver deck,
  initializes HyperMesh, sets the solver to 100 (the same number defined in
  the template), reads the model and passes material data structures to HyperMesh,
  and finally closes the connection between HM and the FE-input reader. */
 get_data(argv[1]);
 HMIN_init("ExampleFEInput", "10.0", argc, argv);
 HMIN_setsolver(100);
 HMIN_readmodel(HM_getfunction);
 HMIN_close();
 return(0);
}
int get_data(char *fileptr)
{
 /* This function opens a solver deck defined as the first argument on the
 input line and reads the solver deck for MAT1 cards.  If a MAT1 card is found
 then the MAT1 solver card is read and a material data structure is populated. */
 ifstream infile;
 char token[9];
 char line[128];
 //Open Solver Deck
 infile.open(fileptr, ios::in);
 if (infile.fail())
        return(1);
 //Read Solver Deck for MAT1 Solver Cards and Populate Material Data Structure
 nummaterials = 0;
 while (!infile.eof())
 {
        infile.get(token, 9);
        if (strcmp(token, "MAT1    ") == 0)
        {
               //Name
               strcpy_s(material[nummaterials].name, "material");
               //id
               infile.get(token, 9);
               material[nummaterials].id = atoi(token);
               //E
               infile.get(token, 9);
               material[nummaterials].E = atof(token);
               //G
               infile.get(token, 9);
               material[nummaterials].G = atof(token);
               //NU
               infile.get(token, 9);
               material[nummaterials].NU = atof(token);
               //RHO
               infile.get(token, 9);
               material[nummaterials].RHO = atof(token);
               //A
               infile.get(token, 9);
               material[nummaterials].A = atof(token);
               //TREF
               infile.get(token, 9);
```

```
                  material[nummaterials].TREF = atof(token);
                  //GE
                  infile.get(token, 9);
                  material[nummaterials].GE = atof(token);
                  infile.get();
                  //Blank Field
                  infile.get(token, 9);
                  //ST
                  infile.get(token, 9);
                  material[nummaterials].ST = atof(token);
                  //SC
                  infile.get(token, 9);
                  material[nummaterials].SC = atof(token);
                  //SS
                  infile.get(token, 9);
                  material[nummaterials].SS = atof(token);
                  infile.get();
                  nummaterials++;
          }
          else
                  infile.getline(line, sizeof(line));
 }
 return(0);
}
entityfunctionptr HM_getfunction(int function, HM_entitytype entities)
{
 /* This user-defined function is passed into hminlib and is
 used by hminlib to find all of the user-defined functions
 which perform reading and information passing.  Note
 that if a user-defined function is not required, this function
 must return NULL. */
 switch (function)
 {
          case HMIN_OPENFUNCTION:
                  break;
          case HMIN_ENTITYOPENFUNCTION:
                  break;
          case HMIN_ENTITYGETFUNCTION:
                  switch (entities)
                  {
                          case HM_ENTITYTYPE_NULL:
                                  break;
                          case HM_ENTITYTYPE_CARDS:
                                  break;
                          case HM_ENTITYTYPE_SYSTCOLS:
                                  break;
                          case HM_ENTITYTYPE_SYSTS:
                                  break;
                          case HM_ENTITYTYPE_NODES:
                                  break;
                          case HM_ENTITYTYPE_VECTORCOLS:
                                  break;
                          case HM_ENTITYTYPE_VECTORS:
                                  break;
                          case HM_ENTITYTYPE_MATS:
                                  return(HM_getMaterials);
                          case HM_ENTITYTYPE_PROPS:
                                  break;
                          case HM_ENTITYTYPE_COMPS:
                                  break;
                          case HM_ENTITYTYPE_GROUPS:
                                  break;
                          case HM_ENTITYTYPE_ELEMS:
```

```
                break;
            case HM_ENTITYTYPE_LOADCOLS:
                break;
            case HM_ENTITYTYPE_EQUATIONS:
                break;
            case HM_ENTITYTYPE_LOADS:
                break;
            case HM_ENTITYTYPE_GEOMETRY:
                break;
            case HM_ENTITYTYPE_LINES:
                break;
            case HM_ENTITYTYPE_SURFS:
                break;
            case HM_ENTITYTYPE_POINTS:
                break;
            case HM_ENTITYTYPE_ASSEMS:
                break;
            case HM_ENTITYTYPE_CURVES:
                break;
            case HM_ENTITYTYPE_PLOTS:
                break;
            case HM_ENTITYTYPE_BLOCKS:
                break;
            case HM_ENTITYTYPE_TITLES:
                break;
            case HM_ENTITYTYPE_SETS:
                break;
            case HM_ENTITYTYPE_OUTPUTBLOCKS:
                break;
            case HM_ENTITYTYPE_LOADSTEPS:
                break;
            case HM_ENTITYTYPE_SENSORS:
                break;
            case HM_ENTITYTYPE_DESIGNVARS:
                break;
            case HM_ENTITYTYPE_BEAMSECTCOLS:
                break;
            case HM_ENTITYTYPE_BEAMSECTS:
                break;
            case HM_ENTITYTYPE_OPTITABLEENTRS:
                break;
            case HM_ENTITYTYPE_OPTIFUNCTIONS:
                break;
            case HM_ENTITYTYPE_OPTIRESPONSES:
                break;
            case HM_ENTITYTYPE_DVPRELS:
                break;
            case HM_ENTITYTYPE_OPTICONSTRAINTS:
                break;
            case HM_ENTITYTYPE_DESVARLINKS:
                break;
            case HM_ENTITYTYPE_OBJECTIVES:
                break;
            case HM_ENTITYTYPE_CONTROLVOLS:
                break;
            case HM_ENTITYTYPE_MULTIBODIES:
                break;
            case HM_ENTITYTYPE_ELLIPSOIDS:
                break;
            case HM_ENTITYTYPE_OPTICONTROLS:
                break;
            case HM_ENTITYTYPE_OPTIDSCREENS:
                break;
```

```
                        case HM_ENTITYTYPE_TAG:
                               break;
                        case HM_ENTITYTYPE_MBJOINT:
                               break;
                        case HM_ENTITYTYPE_MBPLANE:
                               break;
                        case HM_ENTITYTYPE_DOBJREFS:
                               break;
                        case HM_ENTITYTYPE_CONTACTSURFS:
                               break;
                        case HM_ENTITYTYPE_CONNECTORS:
                               break;
                        case HM_ENTITYTYPE_SYMMETRYS:
                               break;
                        case HM_ENTITYTYPE_HANDLES:
                               break;
                        case HM_ENTITYTYPE_DOMAINS:
                               break;
                        case HM_ENTITYTYPE_SHAPES:
                               break;
                        case HM_ENTITYTYPE_SOLIDS:
                               break;
                        case HM_ENTITYTYPE_MORPHCONSTRAINTS:
                               break;
                        case HM_ENTITYTYPE_HYPERCUBES:
                               break;
                        case HM_ENTITYTYPE_DDVALS:
                               break;
                        case HM_ENTITYTYPE_BAGS:
                               break;
                        case HM_ENTITYTYPE_MAX:
                               break;
                }
                break;
        case HMIN_ENTITYCLOSEFUNCTION:
                break;
        case HMIN_NAMEFUNCTION:
                break;
        case HMIN_MOVEFUNCTION:
                break;
        case HMIN_COLORFUNCTION:
                break;
        case HMIN_ASSOCIATEFUNCTION:
                break;
        case HMIN_CEDATAFUNCTION:
                break;
        case HMIN_METADATAFUNCTION:
                break;
        case HMIN_CLOSEFUNCTION:
                break;
 }

 return(NULL);

}

int HM_getMaterials()
{
 /* This function writes each material data structure to HyperMesh.        */

 int i;

 //Write each material data structure to HyperMesh
```

```
 for (i=0; i<nummaterials; i++)
 {
         HMIN_material_write(material[i].id, material[i].name);
         HMIN_writeattribute_int(HM_ENTITYTYPE_MATS, material[i].id, 1, 0, 1, 1);
         HMIN_writeattribute_double(HM_ENTITYTYPE_MATS, material[i].id, 2, 0, 1,
material[i].E);
         HMIN_writeattribute_double(HM_ENTITYTYPE_MATS, material[i].id, 3, 0, 1,
material[i].G);
         HMIN_writeattribute_double(HM_ENTITYTYPE_MATS, material[i].id, 4, 0, 1,
material[i].NU);
         HMIN_writeattribute_double(HM_ENTITYTYPE_MATS, material[i].id, 5, 0, 1,
material[i].RHO);
         HMIN_writeattribute_double(HM_ENTITYTYPE_MATS, material[i].id, 6, 0, 1,
material[i].A);
         HMIN_writeattribute_double(HM_ENTITYTYPE_MATS, material[i].id, 7, 0, 1,
material[i].TREF);
         HMIN_writeattribute_double(HM_ENTITYTYPE_MATS, material[i].id, 8, 0, 1,
material[i].GE);
         HMIN_writeattribute_double(HM_ENTITYTYPE_MATS, material[i].id, 9, 0, 1,
material[i].ST);
         HMIN_writeattribute_double(HM_ENTITYTYPE_MATS, material[i].id, 10, 0, 1,
material[i].SC);
         HMIN_writeattribute_double(HM_ENTITYTYPE_MATS, material[i].id, 11, 0, 1,
material[i].SS);
 }

 return(0);
}
```

Collectors and Collected Entities

Collectors are named organizational containers for collected entities. Collected entities are nameless entities which must reside within one, and only one, collector. Collected entities are mutually exclusive to a collector.

Collector Entities

An example of a collector is the component collector which collects collected entities such as points, lines, surface, solids, elements, and connectors for model organization purposes.

The current collector is bolded in the Model Browser. Newly created entities are automatically organized within the respective current collector.

Collectors control the display state, on or off, of all their collected entities as a group. The display state of collectors can be controlled in the Model Browser.

With the exception of assemblies and bags, operations performed on a collector affect the collected entities within the collector. For example, if you delete a vector collector, the vectors within the vector collector are also deleted.

Collected Entities

Examples of collected entities include points, lines, surfaces, solids, elements, and connectors, which are collected by a component collector.

Newly created entities are automatically organized within the respective current collector.

Include Files and SolverSubmodels

Include Files

Include files enable solver decks to be split into separate files for organizational purposes, and provide a mechanism to include these files in a master solver beck, typically using an include statement.

Solver include files can generally have any solver card defined within them, therefore the include file entity has the capability to store and organize all entities. Every entity is stored and organized into an include file. There is a special include file called the Master Model which corresponds to the master solver deck and is automatically created for every HyperMesh model. New entities are automatically stored and organized into the current include file. If there are no include files defined then entities are automatically stored and organized into the Master Model.

The contents of the Master Model can be observed by changing the entity view in the Model Browser to Include View.

Every include file has a name and a location with a full or relative path.

Include file structures can be imported and exported with solver decks. By default, the solver include file structure is preserved during import. Include files are generated to match and organize all data within the solver include files into the appropriate include file. Solver include file references are also preserved in the Master Model. During export, all entities are written into their corresponding solver include files along with their references in the master solver deck. Change this functionality by selecting a different setting from the Include export option in the **File Options** dialog that opens when you export a solver deck.

Abaqus

- Complete information is required for each keyword. A keyword and its data lines must be part of the same file.
- Complete information is required for cards that contain multiple sub-keywords. For example, all sub-keywords and their data lines in a *Material card must be part of the same include file.
- Complete information is required for *Step cards. All history keywords and their data lines must be a part of the same file.
- The Abaqus solver interface is comprised of four types of include files, which define the sequence of the *Include keywords in the model.
    - Model (start) is written at the beginning of the deck, after the *Node block.
    - Model (middle) is written in the middle of the deck, after the *Material block.
    - Model is written at the end of the model definition.
    - History keywords are written after the model definition.
- Include file names are sorted according to their names in the browser. The sequence in the exported model is primarily determined by include file type. Within each type the sequence is determined by the order in which they are created.
- Include file path syntax.
    - File names can include a full or relative path name. Relative path names must be in relation to the directory from which the job was started. If a path is not specified, it is assumed that the file is located in the same directory from which the job was submitted.
    - From HyperMesh, it is, however, not always possible to predict the directory from which the job will finally be submitted. Therefore a relative path must be defined. This relative path should be defined with respect to the folder where the corresponding *Include keyword appears. If you run the job from a different folder in subsequent runs, you must also update the path name.

LS-DYNA

- The LS-DYNA keywords *INCLUDE, *INCLUDE_STAMPED_PART, *INCLUDE_STAMPED_PART_SET, *INCLUDE_TRANSFORM, *INCLUDE_COMPENSATION_OPTION are mapped to include files.
- You can switch to different types of includes (with the exception of *INCLUDE_TRANSFORM and *INCLUDE_STAMPED_PART, *INCLUDE_STAMPED_PART_SET) using the context menu Include File option in the Model Browser.
- INCLUDE_TRANSFORM is managed using the Transformation Manager.
- During import, if the same include file is referenced more than once using *INCLUDE_TRANSFORM, then it is imported, but appended with .#, where # = 1…n and is shown in the Model Browser. These will not be exported unless you clear the **Instance** checkbox.
- During import, *INCLUDE_STAMPED_PART and *INCLUDE_STAMPED_PART_SET are imported, by default, as read only to preserve the associativity.
- During import, an include file can be read when *INCLUDE comes after *INCLUDE_TRANSFORM.

- In the solver options of the Import browser, an option to choose the type of the Include file is available at the time of importing a file as include to the file in session. This allows you to set the include file type as one of the following: INCLUDE, INCLUDE_STAMPED_PART, INCLUDE_STAMPED_PART_SET, INCLUDE_COMPENSATION_options. This is the only option to attach any file of type INCLUDE_STAMPED_PART, INCLUDE_STAMPED_PART_SET.

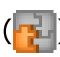
Radioss

- Radioss Keyword #include is mapped to the Include files.
- A solver deck can be directly imported as an Include file, when the import option **Import as include** is enabled in the Import browser.
- The order of the include files in the Model browser is dependent from their appearance order in the solver deck.
- During import, if the same Include file is referred more than once, then it is imported but appended with .# where # = 1…n and shown in the Model browser. These will not be exported unless you clear the **Instance** checkbox.

SolverSubmodels

SolverSubmodels allow you to manage ID offsets as well as define transformations on solver entities or Include files.

SolverSubmodels are generally used to position a sub-assembly of a model by defining transformations on it, and, manage the entities IDs in the sub-assembly by defining IDs offsets.

The SolverSubmodel is differentiated from a solver Include files by its entity icon ().

In the Radioss solver profile, the //SUBMODEL keyword is mapped to the SolverSubmodels.

The definition of a Position entity and Transformations on a SolverSubmodel allows you to position the content of a SolverSubmodel in the space.

Create SolverSubmodels

Manage ID offsets as well as define transformations on solver entities or Include files using the SolverSubmodel entity.

1. Use the Model or Solver browser to create a SolverSubmodel.
    - In the Model browser, right-click and select **Create** > **Solversubmodel** from the context menu.
    - In the Solver browser, right-click and select **Create** > **Submodel** from the context menu.
2. In the **Create SUBMODEL** dialog, define the SolverSubmodel.
    a) In the **Name** field, enter a name.
    b) Under **Offsets**, define all entity ID offsets.
    c) Click **Create**.
3. Modify ID offsets in the Entity Editor, or by dragging-and-dropping entities or Include Files into the SolverSubmodel in the browser.

ID offsets will be automatically assigned on affected entities during a move operation (drag-and-drop) of those entities (or Include files) into the Submodel.

Changing the ID offsets in the SolverSubmodel Entity Editor will not renumber the entity IDs already defined in the SolverSubmodel. You can manage ID offsets of entities defined in a SolverSubmodel in the **ID Manager** using the **Edit ID Offsets** option accessed from the contextual menu.

Tip:

Moving entities or Include files into SolverSubmodel when ID offsets are defined, may result in ID conflicts during ID offset operation. In that case, you will receive a message to inform you about such IDs have conflicts, and you can choose to perform an automatic renumbering of conflicting entities or stop the move operation.

Assemblies

Assemblies collect and organize sub-assemblies and components into hierarchical data structures which are intended to reflect the data structure of the product being modeled.

Operations performed on an assembly do not affect the sub-assemblies or components collected within the assembly. For example, if you delete an assembly, the sub-assemblies or components in the assembly are not deleted.

PAM-CRASH Cards

Card	Description
MBSYS/	Describes an assembly of multibodies

Radioss Cards

Card	Description
/SUBSET	Describes subsets.

Card	Description
	Note: Block Format Keyword

Components and Geometric/FE Entities

Components

Components collect and organize points, lines, surfaces, solids, elements and connectors. Components are intended to be organizational containers for the geometry and FE idealization of a physical part which makes up a product.

Components can be organized into one or more assemblies, therefore, components are not mutually exclusive to an assembly. It is not recommended to organize components into multiple assemblies if it can be avoided. The components which are not organized into an assembly can be observed by changing the entity view in the Model Browser to Assembly View.

Components can be assigned properties and materials. Component property and material assignments are solver interface dependent. In general, when a component is assigned a property or material, that property or material assignment is applied to all elements organized within that component. The method of assigning properties and materials at the component level is therefore referred to as indirect property and material assignment. Direct property and material assignment is performed directly on the elements themselves. Direct property and material assignments always take precedence over indirect property and material assignments.

ANSYS Cards

Card	Description
*HM_COMP	

EXODUS Cards

Card	Description
Beam	
ConMass	
ContactTie	

Card	Description
ElementBlocks	
Shell	
Solid	
Spring	
Truss	

LS-DYNA Cards

Card	Description
*PART	Defines parts for part adaptivity.
*PART_COMPOSITE	Provides a simplified method of defining a composite material model for shell elements that eliminates the need for user defined integration rules and part IDs for each composite layer.
*PART_COMPOSITE_CON	Allows part based contact parameters to be used with the automatic contact types a3, 4, a5, a10, 13, 15 and 26.
*PART_COMPOSITE_TSHELL	
*PART_COMPOSITE_LONG	
*PART_CONTACT	Allows part-based contact parameters to be used with the automatic contact types a3, a5, a10, a13, 15 and 26
*PART_CONTACT_PRINT	
*PART_INERTIA	Allows the inertial properties and initial conditions to be defined rather than calculated from the finite element mesh.
*PART_INERTIA_CONTACT	
*PART_INERTIA_CONTACT_PRI	
*PART_INERTIA_PRINT	
*PART_MOVE	Translate a part by an incremental displacement in either a local or a global coordinate system.
*PART_PRINT	Allows user control over whether output data is written into the ASCII files `MATSUM` and `RBDOUT`.

Card	Description
*PART_REPOSITION	Applies to deformable materials and is used to reposition deformable materials attached to rigid dummy components whose motion is controlled by either `CAL3D` or `MADYMO`.
*PART_REPOSITION_CON	
*PART_REPOSITION_CON	
*PART_REPOSITION_PRI	

PAM-CRASH Cards

Card	Description
PART /	Part_3D
PART /	Part_2D
PART /	Part_1D
PART /	Part_LINK

Permas Cards

Card	Description
$ELPROP	Assignment of geometrical data and material to elements.

Radioss Cards

Card	Description
/PART	Defines a part, which combines material and property information. Optionally, the interface gap can be specified.

Card	Description
	Note: Block Format Keyword

Geometric Entities

Every point, line, surface or solid must be organized into one component, and therefore are mutually exclusive to a component.

Points

A point is a zero-dimensional geometric entity.

A free point is a zero-dimensional geometry entity in space that is not associated with a surface. It is displayed as a small "x". These types of points are typically used for weld locations and connectors.

A fixed point is a zero-dimensional geometry entity that is associated with a surface. It is displayed as a small "o". The automesher places a node at each fixed point on the surface being meshed. A fixed point that is placed at the junction of three or more non-suppressed edges is called a vertex or vertex point. Such vertices cannot be suppressed (removed).

Lines

A line represents a curve in space and is not attached to any surface or solid. A line is a one-dimensional geometric entity.

A line can be composed of one or more line types. Each line type in a line is referred to as a segment. The end point of each line segment is connected to the first point of the next segment. A joint is the common point between two line segments. Line segments are maintained as a single line entity, so operations performed on the line affect each segment of the line. In general, HyperMesh automatically uses the appropriate number and type of line segments to represent the geometry.

Lines are different from surface edges and are sometimes handled differently for certain operations.

Surfaces

A surface represents the geometry associated with a physical part. A surface is a two-dimensional geometric entity that may be used in automatic mesh generation.

A surface is comprised of one or more faces. Each face contains a mathematical surface and edges to trim the surface, if required. When a surface has several faces, all of the faces are maintained as a single surface entity. Operations performed on the surface affect all the faces that comprise the surface. In general, HyperMesh automatically uses the appropriate number of and type of surface faces to represent the geometry.

The perimeter of a surface is defined by edges. There are four types of surface edges:

Free edges
A free edge is an edge that is owned by only one surface.

Free edges are colored red by default.
On a clean model consisting of surfaces, free edges appear only along the outer perimeter of the part and around any interior holes. Free edges that appear between two adjacent surfaces indicate the existence of a gap between the two surfaces. The automesher will leave a gap in the mesh wherever there is a gap between two surfaces.

Shared edges

A shared edge is an edge that is owned, or shared, by two adjacent surfaces.
Shared edges are colored green by default.
When the edge between two surfaces is a shared edge, there is no gap or overlap between the two surfaces - they are geometrically continuous. The automesher always places seed nodes along the length a shared edge and will produce a continuous mesh without any gaps along that edge. The automesher will not construct any individual elements that cross over a shared edge.

Suppressed edges

A suppressed edge is shared by two surfaces but it is ignored by the automesher.
Suppressed edges are colored blue by default.
Like a shared edge, a suppressed edge indicates geometric continuity between two surfaces but, unlike a shared edge, the automesher will mesh across a suppressed edge as if it were not even there. The automesher does not place seed nodes along the length of a suppressed edge and, consequently, individual elements will span across it. By suppressing undesirable edges you are effectively combining surfaces into larger logical meshable regions.

Non-manifold edges

A non-manifold edge is owned by three or more surfaces.
Non-manifold edges are colored yellow by default.
They typically occur at "T" intersections between surfaces or when 2 or more duplicate surfaces exist. The automesher always places seed nodes along their length and will produce a continuous mesh without any gaps along that edge. The automesher will not construct any individual elements that cross over a T-joint edge. These edges cannot be suppressed.

Surface edges are different from lines and are sometimes handled differently for certain operations.

The connectivity of surface edges constitutes the geometric topology.

Solids

Solids are closed volume of surfaces that can take any shape. Solids are three-dimensional entities that can be used in automatic tetra and solid meshing.

The surfaces defining a solid can belong to multiple component collectors. The display of a solid and its bounding surfaces are controlled only by the component collector to which the solid belongs.

Faces

A face is a single Non-uniform Rational B-Spline (NURBS) and is the smallest area entity. It has a separate underlying mathematical definition, specified when it was created.

All faces are represented mathematically with the following formulations:

- plane
- cylinder/cone
- sphere

- torus
- NURBS

A surface can be made up of a single face type or of multiple face types. Multiple types are used for more complex surfaces that contain sharp corners or highly complex shapes.

Finite Element Entities

Supported finite element entities.

Elements
Nodes

Load Collectors, Loads and Equations

Load Collectors

Load collectors collect and organize loads, constraints, and equations.

Abaqus Cards

Loads or constraints that are to be used as history data (under *STEP) should be collected into load collectors with the HISTORY card image. These load collectors also need to be added to the corresponding load steps (*STEP). In contrast, loads or constraints for model data should be collected into load collectors with INITIAL_CONDITION card image. They will automatically be written out in the model portion of the Abaqus input deck.

> **Note:** All loads and boundary conditions on sets can be expanded to individual nodes and elements by selecting the **Expand load on sets** option in the **File Options** dialog, which is invoked upon importing a solver deck. If a **HMLOAD_SETS_EXPAND comment is found in the input file, all loads and boundary conditions on sets are expanded to individual nodes and elements.

Card	Description
*CFILM	Defines film coefficients and associated sink temperatures at one or more nodes or vertices. **Note:** Only in HISTORY card image.
*CONNECTOR HARDENING	

Card	Description
*CONNECTOR LOAD	Specifies loads for available components of relative motion in connector elements. **Note:** Only in HISTORY card image.
*CONNECTOR MOTION	Specifies the motion of available components of relative motion in connector elements. **Note:** In both HISTORY and INTIAL_CONDITION card image.
*DSLOAD	Specifies distributed surface loads. **Note:** Only in HISTORY card image.
*INERTIA RELIEF	Applies inertia-based load balancing. **Note:** Only in HISTORY card image of Standard template.
*INITIAL_CONDITION_FL	Specifies initial pressures for hydrostatic fluid filled cavities.
*INITIAL_CONDITION_TE	Specifies initial temperatures for heat transfer analysis.
*INITIAL_CONDITION_TY	Specifies initial stresses.
*INITIAL_CONDITION_VE	Specifies initial velocities for dynamic analysis.
*SFILM	Define film coefficients and associated sink temperatures over a surface for heat transfer analysis. **Note:** Only in HISTORY card image.

EXODUS Cards

Card	Description
Acceleration	

Card	Description
Boundary	
Force	
Flux	
Moment	
Thermal	
Velocity	

LS-DYNA Cards

Load collector information is specified with a required $HMNAME comment card and an optional $HMCOLOR comment card. If an input translator encounters one of these comments while reading a load card, a new load collector is created. For the comments to be valid, they must follow a load keyword or the last line of the previous Structured block. The loads that follow a $HMNAME LOADCOLS comment are read into that collector. If there is a new keyword or structured block, the previous load collector information is ignored.

For non-HyperMesh generated input decks, loads are divided into collectors based on classification. The following load collectors are created:

- Mechanical loads for forces and moments
- Constraints/Displacements
- Velocities
- Accelerations
- Pressures

If translational or rotational constraints are defined in the input model, they are placed in a separate load collector named Nodal Constraints.

Load collectors are not used by LS-DYNA, but are useful for visualization. Additional load collectors can be defined to describe other entities.

Card	Description
*BOUNDARY_CONVECTIC	Define convection boundary conditions for a thermal or coupled thermal/ structural analysis. Two cards are defined for each option.
*BOUNDARY_NON_REFLE	Define a non-reflecting boundary.
*BOUNDARY_NON_REFLE	Define a non-reflecting boundary.
*BOUNDARY_RADIATION_	Defines surface segment sets that transfer energy by radiation to the environment.

Card	Description
*BOUNDARY_SPC_SET	Define nodal single point constraints.
*BOUNDARY_SPC_SET_II	
*BOUNDARY_TEMPERATU	Define temperature boundary conditions for a thermal or coupled thermal/ structural analysis.
*CONSTRAINED_RIGID_E	Stops the motion based on a time dependent constraint. The stopper overrides prescribed motion boundary conditions (except relative displacement) operating in the same direction for both the master and slaved rigid bodies.
*DEFINE_CURVE_FEEDBA	Define information that is used as the solution evolves to scale the ordinate values of the specified load curve ID.
*DEFINE_CURVE_FEEDBA	
*DEFORMABLE_TO_RIGII	Define materials to be switched to rigid at the start of the calculation. **Note:** Select an arraycount for the PSID and MRB pairs.
*DEFORMABLE_TO_RIGII	Define materials to be switched to rigid or to deformable at some stage in the calculation. **Note:** Change the option to automatic and card edit. In the D2R fields enter the number of PIDs that need to be converted to Rigid. Create an entity set of comps of the slave PIDs and select the set.
*DEFORMABLE_TO_RIGII	Inertial properties can be defined for the new rigid bodies that are created when the deformable parts are switched. These can only be defined in the initial input if they are needed in a later restart.
*INITIAL_AXIAL_FORCE_	Initialize axial force in the beam for modeling bolt
*INITIAL_DETONATION	Define points to initiate the location of high explosive detonations in part IDs which use the material (type 8) *MAT_HIGH_EXPLOSIVE_BURN.
*INITIAL_STRESS_SECTI	Initialize stress in solid sections
*INITIAL_TEMPERATURE_	Define initial nodal point temperatures using nodal set IDs or node numbers.
*INITIAL_VEHICLE_KINEI	Define initial kinematical information for a vehicle.
*INITIAL_VELOCITY	Define initial nodal point translational velocities using nodal set IDs. This may also be used for sets in which some nodes have other velocities.

Card	Description
	Note: InitialVel This card changes the INITV definition on Control Card 11. Only the first card defined is valid for Structured.
*INITIAL_VELOCITY_GEN	Define initial velocities for rotating and translating bodies.
*INITIAL_VELOCITY_GEN	Define a time to initialize velocities after time zero.
*INITIAL_VELOCITY_RIGI	Define the initial translational and rotational velocities at the center of gravity for a rigid body or a nodal rigid body.
*INTERFACE_SPRINGBAC	Define a material subset for an implicit springback calculation in LS-DYNA and any nodal constraint to eliminate rigid body degrees-of-freedom.
*INTERFACE_SPRINGBAC	
*INTERFACE_SPRINGBAC	Define a material subset for an implicit springback calculation in LS-DYNA and any nodal constraints to eliminate rigid body degrees-of-freedom.
*INTERFACE_SPRINGBAC	
*INTERFACE_SPRINGBAC	
*INTERFACE_SPRINGBAC	
*INTERFACE_SPRINGBAC	
*INTERFACE_SPRINGBAC	
*INTERFACE_SPRINGBAC	
*INTERFACE_SPRINGBAC	
*LOAD_BEAM_SET	Defines load on beam element set
*LOAD_BLAST	Define an airblast function for the application of pressure loads due to explosives in conventional weapons.
*LOAD_BODY_GENERALI	Define body force loads due to a prescribed base acceleration or prescribed angular velocity over a subset of the complete problem.
*LOAD_BODY_PARTS	Define body force loads due to a prescribed base acceleration or angular velocity using global axes directions. **Note:** Select component set.

Card	Description
*LOAD_BODY_X	Define body force loads due to a prescribed base acceleration using global axes directions. **Note:** Activate the proper option and enter the data. Only the first card defined is valid for Structured.
*LOAD_BODY_Y	**Note:** Activate the proper option and enter the data. Only the first card defined is valid for Structured.
*LOAD_BODY_Z	**Note:** Activate the proper option and enter the data. Only the first card defined is valid for Structured.
*LOAD_BODY_RX	Define body force loads due to a prescribed angular velocity using global axes directions. **Note:** Activate the proper option and enter the data. Only the first card defined is valid for Structured.
*LOAD_BODY_RY	**Note:** Activate the proper option and enter the data. Only the first card defined is valid for Structured.
*LOAD_BODY_RZ	**Note:** Activate the proper option and enter the data. Only the first card defined is valid for Structured.
*LOAD_BRODE	Define Brode function for application of pressure loads due to explosion.
*LOAD_MASK	Apply a distributed pressure load over a three-dimensional shell part.
*LOAD_NODE_SET	Apply a concentrated nodal force to a node or a set of nodes.
*LOAD_RIGID_BODY	
*LOAD_SEGMENT_SET	Apply the distributed pressure load over each segment in a segment set.
*LOAD_SHELL_SET	Apply the distributed pressure load over one shell element or shell element set.
*LOAD_SUPERELASTIC_F	

Card	Description
*LOAD_THERMAL_CONST	Define nodal sets giving the temperature that remains constant for the duration of the calculation.
*LOAD_THERMAL_LOAD_	
*LOAD_THERMAL_VARIAI	Define nodal sets giving the temperature that is variable in the duration of the calculation.

Nastran Cards

Nastran supports specific and generic load collectors. Specific load collectors have a card image which can be edited to group other load collectors together for simultaneous application in a single subcase, or to provide special information for a specific analysis type (such as modal analysis). Use specific load collectors for specialized loading cards, such as SPCADD, MPCADD, EIGRL, EIGB, EIGC, EIGP, EIGR, FREQ, FREQ1, LOAD, GRAV, RFORCE, and TEMPD. Generic load collectors do not have a card image. Use generic load collectors to collect loads and constraints for display purposes and to assign an ID to the loads.

General boundary conditions, such as loads and constraints, should not be collected into specific load collectors. Organizing loads and constraints into a specific load collector may result in an error termination.

When a Nastran deck is imported into HyperMesh, loads that have the same SID are collected into the same load collector. If a load collector already exists in the database with the same SID, one of the following can occur:

- If overwrite is off (default), the new load collector's ID is offset and all loads in that collector will have a new SID upon export.
- If overwrite is on, the new load collector replaces the existing load collector. The original load collector and the loads it contains are deleted.

Card	Description
ACSRCE	Defines the power versus frequency curve for a simple acoustic source.
AEFACT	Defines real numbers for aeroelastic analysis.
AEPARM	Defines a general aerodynamic trim variable degree-of-freedom (aerodynamic extra point).
AESTAT	Specifies rigid body motions to be used as trim variables in static aeroelasticity.
AEFORCE	Defines a vector of absolute or "per unit dynamic pressure" forces associated with a particular control vector.
BCPARA	Defines contact parameters.

Card	Description
BCRPARA	
BMFACE	
DAMPING	Defines a parameter and hybrid damping specification.
DELAY	Defines the time delay term in the equations of the dynamic loading function. **Note:** Supported as constraints.
DLOAD	Defines a dynamic loading condition for frequency response or transient response problems as a linear combination of load sets defined via RLOAD1 or RLOAD2 entries for frequency response or TLOAD1 or TLOAD2 entries for transient response
DTI SPECSEL	Defines table data blocks
EIGB	Defines data needed to perform buckling analysis
EIGC	Defines data needed to perform complex eigenvalue analysis
EIGP	Defines poles that are used in complex eigenvalue extraction by the Determinant method
EIGR	Defines data needed to perform real eigenvalue analysis
EIGRL	Defines data needed to perform real eigenvalue (vibration or buckling) analysis with the Lanczos method
FLFACT	Used to specify density ratios, Mach numbers, reduced frequencies, and velocities for flutter analysis.
FLUTTER	Defines data needed to perform flutter analysis.
FREQ	Defines a set of frequencies to be used in the solution of frequency response problems. **Note:** Defined using FREQi.
FREQ1	Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, frequency increment, and the number of increments desired. **Note:** Defined using FREQi.

Card	Description
FREQ2	Alternative form of frequency list. Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, final frequency, and the number of logarithmic increments desired. **Note:** Defined using FREQi.
FREQ3	Frequency List, Alternate Form 3. Defines a set of frequencies for the modal method of frequency response analysis by specifying the number of frequencies between modal frequencies. **Note:** Defined using FREQi.
FREQ4	Frequency List, Alternate Form 4. Defines a set of frequencies for the modal method of frequency response analysis by specifying the amount of "spread" around each modal frequency and the number of equally spaced frequencies within the spread. **Note:** Defined using FREQi.
FREQ5	Frequency List, Alternate Form 5. Defines a set of frequencies for the modal method of frequency response analysis by specification of a frequency range and fractions of the natural frequencies within that range. **Note:** Defined using FREQi.
GRAV	Defines acceleration vectors for gravity or other acceleration loading
GUST	Selects the gust field in an aeroelastic response problem.
HYBDAMP	Defines a hybrid modal damping for direct dynamic solutions.
LOAD	Defines a static load as a linear combination of load sets defined via FORCE, MOMENT, FORCE1, MOMENT1, FORCE2, MOMENT2, PLOAD, PLOAD1, PLOAD2, PLOAD4, PLOADX1, SLOAD, RFORCE, and GRAV entries.
LSEQ	Defines a sequence of static load sets.
MARCOUT	Selects output to be saved on the Marc t16 end/or t19 file(s) used in SOL 600 only.
MFLUID	Defines the properties of an incompressible fluid volume for the purpose of generating a virtual mass matrix

Card	Description
MKAERO1	Provides a table of Mach numbers (m) and reduced frequencies (k) for aerodynamic matrix calculation.
MPCADD	Defines a multipoint constraint set as a union of multipoint constraint sets defined via MPC entries.
NLAUTO	Defines parameters for automatic or fixed load/time stepping used in SOL 600 only.
NLDAMP	Defines damping constants for nonlinear analysis when Marc is executed from SOL 600 only.
NLPARM	Defines a set of parameters for nonlinear static analysis iteration strategy
NLRGAP	Defines a nonlinear radial (circular) gap for transient response or nonlinear harmonic response.
NLSTEP	Describes the control parameters for Mechanical, Thermal and Coupled Analysis in SOL 400 and for Linear Contact Analysis in SOL 101.
NLSTRAT	Defines strategy parameters for nonlinear structural analysis used in SOL 600 only.
NOLIN1	Defines a forcing function for transient responses or nonlinear harmonic responses.
NSMADD	Defines non structural mass as the sum of the sets listed.
NTHICK	Defines nodal thickness values for beams, plates and/or shells.
RADCAV	Identifies the characteristics of each radiant enclosure.
RANDPS	Defines load set power spectral density factors for use in random analysis having the frequency dependent form: $S_{jk}(F) = (X + iY)G(F)$
RFORCE	Defines a static loading condition due to an angular velocity and/or acceleration
RLOAD1	Defines a frequency-dependent dynamic load of the form: $\{P(f)\} = \{A\}[C(f) + iD(f)]e^{i\{0 - 2\pi f \tau\}}$ for use in frequency response problems
RLOAD2	Defines a frequency-dependent dynamic excitation of the form: $\{P(f)\} = \{A\} \cdot B(f)e^{i\{\phi(f) + 0 - 2\pi f \tau\}}$

Card	Description
	for use in frequency response problems
RSPEC	Defines a directional combination method, modal combination method, excitation direction(s), response spectra and scale factors for response spectrum analysis.
SPC1	Defines a single-point constraint, alternate form.
SPCADD	Defines a single-point constraint set as a union of single-point constraint sets defined on SPC or SPC1 entries
SPCR	Defines an enforced relative displacement value for a load step in SOL 400 and SOL 600.
TABDMP1	Defines modal damping as a tabular function of natural frequency
TABLED1	Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads
TABLED2	Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table
TABLED3	Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table
TABLED4	Defines the coefficients of a power series for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table
TABLEM1	Defines a tabular function for use in generating temperature-dependent material properties.
TABLEM2	Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.
TABLEM3	Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.
TABLEM4	Defines coefficients of a power series for use in generating temperature-dependent material properties. Also contains parametric data for use with the table.
TABLES1	Defines a tabular function for stress-dependent material properties such as the stress-strain curve (MATS1 entry), creep parameters (CREEP entry) and hyperelastic material parameters (MATHP entry).

Card	Description
TABLEST	Specifies the material property tables for nonlinear elastic temperature-dependent materials
TABRND1	Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced by the RANDPS entry.
TEMPD	Defines a temperature value for all grid points of the structural model that have not been given a temperature on a TEMP entry.
TIC	Defines values for the initial conditions of variables used in structural transient analysis. Both displacement and velocity values may be specified at independent degrees-of-freedom. This entry may not be used for heat transfer analysis.
TLOAD1	Defines a time-dependent dynamic load or enforced motion of the form: $$\{P(t)\} = \{A \cdot F(t-\tau)\}$$ for use in transient response analysis
TLOAD2	Defines a time-dependent dynamic excitation or enforced motion of the form: $$\{P(t)\} = \begin{cases} 0 & , t < (T1+\tau) \text{ or } t > (T2+\tau) \\ \{A\}\tilde{t}^{B} e^{C\tilde{t}} \cos(2\pi F\tilde{t} + P) , & (T1+\tau) \le t \le (T2+\tau) \end{cases}$$ for use in a transient response problem, where: $$\tilde{t} = t - T1 - \tau$$
TRIM	Specifies constraints for aeroelastic trim variables. The SPLINE1 and SPLINE4 entries need to be here for the finite plate spline.
TSTEP	Defines time step intervals at which a solution will be generated and output in transient analysis
TSTEPNL	Defines parametric controls and data for nonlinear transient structural or heat transfer analysis. TSTEPNL is intended for SOLs 129, 159, and 99.
UXVEC	Specification of a vector of aerodynamic control point (extra point) values.
VIEW	Defines radiation cavity and shadowing for radiation view factor calculations.

Card	Description
VIEW3D	Defines parameters to control and/or request the Gaussian Integration method of view factor calculation for a specified cavity.

OptiStruct Cards

Specific load collectors are used for specialized loading cards, such as EIGRL, SPCADD, GRAV, RLOAD and DTABLEi. Specific load collectors have a card image which can be edited to group other load collectors together for simultaneous application in a single subcase, or to provide special information for a specific analysis type (such as modal analysis).

General boundary conditions, such as loads and constraints, should not be collected into specific load collectors. Organizing loads and constraints into a specific load collector may result in an error termination.

Card	Description
ACSRCE	Defines acoustic source as a function of power vs. frequency. **Note:** Bulk Data Entry
CDSMETH	Can be used in the component dynamic synthesis method for generating component dynamic matrices at each loading frequency. **Note:** Bulk Data Entry
CMSMETH	Defines the CMS (Component Mode Synthesis) method, frequency upper limit, number of modes, and starting `SPOINT` ID to be used in a CMS solution. The eigenvalue solver is also specified. In addition, preload as well as loads for reduction and residual vector generation can be defined. Also, an ASCII file containing `CELAS4` and `CDAMP3` element data and/or their corresponding design variable definitions can be generated for `DMIG` to allow the use of the component modes in optimization runs. **Note:** Bulk Data Entry
CNTSTB	Defines parameters for stabilization control of surface-to-surface contact and large displacement node-to-surface contact. A `CNTSTB` Bulk Data Entry should be referenced by a `CNTSTB` Subcase Information entry to be applied in a particular subcase.

Card	Description
	Note: Bulk Data Entry A `CNTSTB` Bulk Data Entry should be referenced by a `CNTSTB` Subcase Information entry to be applied in a particular subcase.
DLOAD	Defines a dynamic loading condition for frequency response problems as a linear combination of load sets defined via `RLOAD1` and `RLOAD2` entries, or for transient problems as a linear combination of load sets defined via `TLOAD1` and `TLOAD2` entries, or acoustic source `ACSRCE` entries. **Note:** Bulk Data Entry
DTI, SPECSEL	Correlates spectra lines specified on `TABLED1` entries with damping values. **Note:** Bulk Data Entry
EIGC	Defines data required to perform complex eigenvalue analysis. **Note:** Bulk Data Entry
EIGRA	Defines the data required to perform real eigenvalue analysis with the Automated Multi-Level Sub-structuring technique. **Note:** Bulk Data Entry
EIGRL	Defines data required to perform real eigenvalue analysis (vibration or buckling) with the Lanczos Method. **Note:** Bulk Data Entry
FATDEF	Defines elements, and associated fatigue properties, for consideration in a fatigue analysis. **Note:** Bulk Data Entry
FATEVNT	Defines loading events for fatigue analysis. **Note:** Bulk Data Entry

Card	Description
FATLOAD	Defines fatigue loading parameters. **Note:** Bulk Data Entry
FATPARM	Used to define parameters required for a Fatigue Analysis. **Note:** Bulk Data Entry
FATSEQ	Defines a loading sequence for a Fatigue Analysis. **Note:** Bulk Data Entry
FLLWER	Define parameters for the calculation of loads dependent on deformation. Two type of loads, pressure load (only `PLOAD4` Bulk Data Entries) and concentrated force (only `FORCE1`/`FORCE2` Bulk Data Entries) can use this entry to control the options for Follower Loads. **Note:** Bulk Data Entry
FREQ	Defines a set of frequencies to be used in the solution of frequency response problems. **Note:** Bulk Data Entry
FREQ1	Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, frequency increment, and the number of increments desired. **Note:** Bulk Data Entry Defined using FREQi.
FREQ2	Defines a set of frequencies to be used in the solution of frequency response problems by specification of a starting frequency, final frequency, and the number of logarithmic increments desired. **Note:** Bulk Data Entry Defined using FREQi.

Card	Description
FREQ3	Defines a set of frequencies for the modal method of frequency response analysis by specifying the number of frequencies between modal frequencies. **Note:** Bulk Data Entry Defined using FREQi.
FREQ4	Defines a set of frequencies for the modal method of frequency response analysis by specifying the amount of "spread" around each modal frequency and the number of equally spaced frequencies within the spread. **Note:** Bulk Data Entry Defined using FREQi.
FREQ5	Defines a set of frequencies for the modal method of frequency response analysis by specification of a frequency range and fractions of the natural frequencies within that range. **Note:** Bulk Data Entry Defined using FREQi.
FSI	Defines the settings for Fluid-Structure Interaction Analysis with AcuSolve. **Note:** Bulk Data Entry
GRAV	Defines the gravity vectors for use in determining gravity loading for the static structural model. It can also be used to define the EXCITEID field (Amplitude "A") of dynamic loads in `RLOAD1`, `RLOAD2`, `TLOAD1`, `TLOAD2` and `NLOAD1` Bulk Data Entries for dynamic solution sequences. **Note:** Bulk Data Entry
HYBDAMP	This Bulk Data Entry defines the application of modal damping to the residual structure in a Direct Transient or Frequency Response analysis. **Note:** Bulk Data Entry
INVELB	Defines initial velocity in a multibody situation.

Card	Description
	Note: Bulk Data Entry
LOAD	The `LOAD` is equivalent to the `LOADADD`. **Note:** Bulk Data Entry
MBACT	Defines the entity/set that needs to be activated in the multibody system for the subsequent simulation. **Note:** Bulk Data Entry
MBDEACT	Defines the entity/set that needs to be deactivated in the multibody system for the subsequent simulation. **Note:** Bulk Data Entry
MBLIN	Defines the parameters for a multibody system linear analysis. **Note:** Bulk Data Entry
MBREQ	Defines a multibody as a combination of request sets defined via `MBREQE` and `MBREQM`. **Note:** Bulk Data Entry
MBREQE	Defines a multibody solver output request to output the results of a set of expressions. **Note:** Bulk Data Entry
MBREQM	Defines a multibody solver output request to output displacement, velocity, acceleration, or force with respect to markers. **Note:** Bulk Data Entry
MBSEQ	Defines the simulation sequence for the multibody solver.

Card	Description
	Note: Bulk Data Entry
MBSIM	Defines the parameters for a multibody simulation. **Note:** Bulk Data Entry
MBSIMP	Defines the simulation parameters for subsequent multibody simulation. **Note:** Bulk Data Entry
MBVAR	Defines a multibody solver variable which can be referred to by multiple expressions. **Note:** Bulk Data Entry
MFLUID	Defines the parameters and damp shell elements for a fluid volume. **Note:** Bulk Data Entry
MLOAD	Defines a multibody as a linear combination of load sets defined via `GRAV`, `MBFRC`, `MBFRCC`, `MBFRCE`, `MBMNT`, `MBMNTC`, `MBMNTE`, `MBSFRC`, `MBSFRCC`, `MBSFRCE`, `MBSMNT`, `MBSMNTC`, and `MBSMNTE`. **Note:** Bulk Data Entry
MOTION	Defines a multibody as a combination of motion sets defined via `MOTNJ`, `MOTNJC`, `MOTNJE`, `MOTNG`, `MOTNGC`, and `MOTNGE`. **Note:** Bulk Data Entry
MPCADD	Defines a multipoint constraint set as a union of multipoint constraint sets defined via `MPC` entries. **Note:** Bulk Data Entry
NLADAPT	The `NLADAPT` Bulk Data Entry defines parameters for time-stepping and convergence criteria in Nonlinear Analysis.

Card	Description
	Note: Bulk Data Entry Refers to Subcase information entry `NLADAPT`.
NLOAD	Defines a loading condition for nonlinear problems as a linear combination of load sets defined via `NLOAD1`. **Note:** Bulk Data Entry
NLOAD1	Defines a time-dependent load or enforced motion for use in geometric nonlinear analysis. **Note:** Bulk Data Entry
NLOUT	Defines incremental result output parameters for small displacement nonlinear analysis and large displacement analysis. **Note:** Bulk Data Entry Refers to Subcase information entry `NLOUT`.
NLPARM	Defines parameters for Nonlinear Static Analysis, Nonlinear Direct Transient Analysis, and Heat Transfer Analysis solution control. **Note:** Bulk Data Entry
NLPARMX	Defines additional parameters for geometric nonlinear implicit static analysis. **Note:** Bulk Data Entry
NSMADD	Defines non-structural mass as the sum of the sets listed. **Note:** Bulk Data Entry
PEAKOUT	Defines criteria used for the automatic identification of loading frequencies at which result peaks occur. Other result output may then be requested at these "peak" loading frequencies. This feature is only supported for frequency response solution sequences.

Card	Description
	Note: Bulk Data Entry
PFAT	Defines element properties for fatigue analysis. **Note:** Bulk Data Entry
PFPATH	Defines a one-step transfer path analysis. **Note:** Bulk Data Entry
PTADD	Defines a pretension load as a linear combination of load sets defined via `PTFORCE`, `PTFORC1`, `PTADJST` and `PTADJS1` entries. **Note:** Bulk Data Entry
RANDPS	Defines load set power spectral density factors for use in random analysis having the frequency dependent form S_{jk} (F) = (X + iY) G(F). **Note:** Bulk Data Entry
RFORCE	Defines a static loading condition due to a centrifugal force field. It can also be used to define the *EXCITEID* field (Amplitude "A") of dynamic loads in `RLOAD1`, `RLOAD2`, `TLOAD1` and `TLOAD2` Bulk Data Entries. `RFORCE` is used as a linear dead-load in Large Displacement Nonlinear Analysis. **Note:** Bulk Data Entry
RGYRO	Includes data required to perform Rotor Dynamics analysis in Modal Frequency Response Analysis and/or Modal Complex Eigenvalue Analysis. The `RGYRO` Bulk Data Entry is referenced by a corresponding `RGYRO` Subcase Information Entry in a specific subcase. **Note:** Bulk Data Entry
RLOAD1	Defines a frequency-dependent dynamic load of the form: $\mathbf{f}(\Omega) = A(C(\Omega) + iD(\Omega))e^{i(\theta - 2\pi\Omega\tau)}$ for use in frequency response problems. `RLOAD1` (Form 1) can be used when the frequency-dependent dynamic load input is available in real/imaginary number format.

Card	Description
	Note: Bulk Data Entry
RLOAD2	Defines a frequency-dependent dynamic load of the form $\mathbf{f}(\Omega) = A * B(\Omega)e^{i(\phi(\Omega)+\theta-2\pi\Omega\tau)}$ for use in frequency response problems. RLOAD2 (Form 2) can be used when the frequency-dependent dynamic load input is available in magnitude/phase number format. **Note:** Bulk Data Entry
RSPEC	Specifies directional combination method, modal combination method, excitation direction(s), response spectra and scale factors. **Note:** Bulk Data Entry
RSPEED	Specifies a set of reference rotor speed values for asynchronous analysis in Rotor Dynamics. **Note:** Bulk Data Entry
SOLVTYP	Defines the solver type to be used for static, dynamic analysis and geometric nonlinear implicit analysis. **Note:** Bulk Data Entry
SPCADD	Defines a single-point constraint set as a union of single-point constraint sets defined via SPC or SPC1 entries. **Note:** Bulk Data Entry
TABDMP1	Defines modal damping as a tabular function of natural frequency. **Note:** Bulk Data Entry
TABFAT	Defines y values of each point on the loading time history. **Note:** Bulk Data Entry

Card	Description
TABLED1	Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. **Note:** Bulk Data Entry
TABLED2	Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table. **Note:** Bulk Data Entry
TABLED3	Defines a tabular function for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table. **Note:** Bulk Data Entry
TABLED4	Defines the coefficients of a power series for use in generating frequency-dependent and time-dependent dynamic loads. Also contains parametric data for use with the table. **Note:** Bulk Data Entry
TABLEM1	Defines a tabular function for use in generating temperature-dependent material properties. **Note:** Bulk Data Entry
TABLEM2	Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table. **Note:** Bulk Data Entry
TABLEM3	Defines a tabular function for use in generating temperature-dependent material properties. Also contains parametric data for use with the table. **Note:** Bulk Data Entry

Card	Description
TABLEM4	Defines coefficients of a power series for use in generating temperature-dependent material properties. Also contains parametric data for use with the table. **Note:** Bulk Data Entry
TABLES1	Defines a tabular function for use as stress-strain curve in elasto-plastic material properties `MATS1`, `MATX33`, `MATX65`, `MATHF`, as well as material curve in nonlinear material properties `MATX36`, `MATX42`, and `MATX70`. The `TABLES1` entry can also be used to define the corresponding material curves on the `MATHE` Bulk Data Entry. **Note:** Bulk Data Entry
TABLEST	Specifies the material property tables for elasto-plastic, temperature-dependent materials. **Note:** Bulk Data Entry
TABRND1	Defines power spectral density as a tabular function of frequency for use in random analysis. Referenced on the `RANDPS` entry. **Note:** Bulk Data Entry
TEMPD	Defines a temperature value for all grid points of the structural model that have not been given a temperature on a `TEMP` entry. **Note:** Bulk Data Entry
TICA	Defines values for the initial velocity of a set of grids along and about an axis for explicit analysis. **Note:** Bulk Data Entry
TLOAD1	Defines a time-dependent dynamic load or enforced motion. **Note:** Bulk Data Entry
TLOAD2	Defines a time-dependent dynamic excitation or enforced motion.

Card	Description
	Note: Bulk Data Entry
TSTEP	Defines time step parameters for control and intervals at which a solution will be generated and output in transient analysis. **Note:** Bulk Data Entry
TSTEPNL	Defines parameters for geometric nonlinear implicit dynamic analysis strategy. **Note:** Bulk Data Entry
TSTEPNX	Defines additional parameters for geometric nonlinear implicit dynamic analysis. **Note:** Bulk Data Entry
UNBALNC	Defines the unbalanced rotating load during a rotor dynamic analysis in Frequency Response solution sequences. The unbalanced load is specified in a cylindrical system where the rotor rotation axis is the Z-axis. **Note:** Bulk Data Entry
XHISADD	Defines a time history output set as a union of time history outputs defined via `XHIST` entries. **Note:** Bulk Data Entry
XSTEP	Defines explicit analysis control. **Note:** Bulk Data Entry

PAM-CRASH Cards

Card	Description
ACC3D /	Imposed accelerations

Card	Description
	Note: Keyword input
ACFLD /	Acceleration field **Note:** Keyword input
BDFOR /	Body force constraints **Note:** Keyword input
BOUNC /	Specify boundary conditions on the base body **Note:** Keyword input
CONLO /	Concentrated nodal load **Note:** Keyword input
DAMP /	Nodal damping group cards **Note:** Keyword input
DIS3D /	Imposed displacement **Note:** Keyword input
DIS3DM /	Imposed minimum displacement **Note:** Keyword input
DIS3DX /	Imposed maximum displacement **Note:** Keyword input
INVEL /	Specify the initial velocity of the base body

Card	Description
	Note: Keyword input
PREFA /	Pressure on shells **Note:** Keyword input
PREBM /	Pressure on beams **Note:** Keyword input
RAC3D /	Imposed rotational acceleration **Note:** Keyword input
RAN3D /	Imposed angular rotations **Note:** Keyword input
RDA3D /	Radial 3D boundary conditions **Note:** Keyword input
RDD3D /	Radial 3D boundary conditions **Note:** Keyword input
RDV3D /	Radial 3D boundary conditions **Note:** Keyword input
RVE3D /	Imposed rotational velocities **Note:** Keyword input
VEL3D /	Imposed velocities

Card	Description
	Note: Keyword input

Permas Cards

Card	Description
$ADDMODES	Definition of static mode shapes to be added to the set of eigenmodes used for transformation to modal space. **Note:** Available as a load collector when the source = loads. To change the LPAT = field, set the AddmodeLoads toggle to LOADSELECT, which ensures that each data line will define different ADDMODES. In the NoOfLoads_AddMode = field, enter the number of load patterns to assign the ADDMODES to.
$CONTVAL	Assignment of properties to contacts referenced by contact identifier or name. **Note:** Supported as a load collector (card image LOADS). To create a card, use an existing load collector or create a new one with card image LOADS and select the CONTVAL check box. A maximum number of 5 keywords are allowed per load collector (load pattern).
$PRETENSION LOAD	Assignment of load properties to pretension threads/areas referenced by identifier or name. **Note:** Supported as a load collector (card image LOADS). To create a card, use an existing load collector or create a new one with card image LOADS and select the PRETENSION checkbox.

Card	Description
$SUPPRESS	Definition of suppressed degrees of freedom. The degrees of freedom given on the header line are suppressed for all nodes listed within the data block.

Radioss Cards

Assign loads to a collector by creating individual loads, all of the same type and degree of freedom, and storing them in the appropriate load collector, or by identifying the nodes on which loads/BCs act by selecting them through a set. The selection of the set is possible by editing the card image of the load collector.

Card	Description
/ACTIV	Describes the deactivation/activation of element groups. **Note:** Block Format Keyword
/ALE/BCS	Describes the ALE boundary conditions. **Note:** Block Format Keyword
/BCS	Defines boundary conditions on node groups for translational and rotational motion. **Note:** Block Format Keyword
/CLOAD	Defines a concentrated force or moment applied to each node of a prescribed nodal group. **Note:** Block Format Keyword
/CONVEC	Describes the free or forced convective flux. **Note:** Block Format Keyword
/DFS/DETLINE	Enable explosive material ignition from a detonation line [A,B]. **Note:** Block Format Keyword
/DFS/DETPOINT	Locates the detonation point and set lighting time for an explosive material law.

Card	Description
	Note: Block Format Keyword
/DFS/LASER	Enable to model laser impact taking into account laser-matter interaction. **Note:** Block Format Keyword
/DFS/WAV_SHA	Enables to shape detonation wave to take into account obstacles. **Note:** Block Format Keyword
/EBCS	Describes the elementary boundary condition sets. **Note:** Block Format Keyword
/GRAV	Defines gravity load on node group. **Note:** Block Format Keyword
/IMPACC	Defines imposed accelerations on a group of nodes. **Note:** Block Format Keyword
/IMPDISP	Defines imposed displacements on a group of nodes. **Note:** Block Format Keyword
/IMPTEMP	Defines imposed temperatures on a group of nodes. **Note:** Block Format Keyword
/IMPVEL	Defines imposed velocities on a group of nodes. **Note:** Block Format Keyword
/INITEMP	Describes the initial nodal temperature.

Card	Description
	Note: Block Format Keyword
/INIVEL	Defines initial velocity on a group of nodes. **Note:** Block Format Keyword
/INIVEL/AXIS	Initialize both translational and rotational velocities on a group of nodes in a given coordinate system. **Note:** Block Format Keyword
/LOAD/CENTRI	Computes a load according to the rotational velocity around an axis. **Note:** Block Format Keyword
/LOAD/PFLUID	Simulates hydrodynamic fluid pressure on a structure. The fluid pressure is calculated according to the specified fluid velocity, orientation of the structural surface against the fluid vector and the height of the fluid column above the surface of the structure. **Note:** Block Format Keyword
/PLOAD	Defines pressure load on a surface. **Note:** Block Format Keyword
/SPHBCS	Describes the SPH symmetry conditions. **Note:** Block Format Keyword
/SPH/INOUT	Describes the SPH inlet/outlet conditions.

Card	Description
	Note: Block Format Keyword

Loads

Every load and constraint must be organized into one load collector, and therefore are mutually exclusive to a load collector.

Load Configurations

Load entities have an associated load configuration. A load configuration determines how to draw, store, and work with a load.

Accelerations

Configuration 9 - Acceleration loads allow for an acceleration (length/time^2) to be defined on the model.

Accelerations are displayed as a vector with the letter A at the tail end in the modeling window.

Constraints

Configuration 3 - Constraints allow for constrained degrees of freedom to be defined on the model.

Constraints are displayed with a triangle that connects to the node, with the dof numbers that apply to the node beside the triangle in the modeling window.

Fluxes

Configuration 6 - Flux loads are defined as an amount that flows through a unit area per unit time (amount/length^2/time). Fluxes are typically used in modeling transport phenomena such as heat transfer, mass transfer, fluid dynamics, and electromagnetism.

Fluxes are displayed as a thick arrow labeled with the word "flux" in the modeling window.

Forces

Configuration 1 - Force loads allow for a concentrated force (mass*length/time^2) to be applied to the model.

Forces are displayed as a vector with the letter F at the tail end in the modeling window.

Moments

Configuration 2 - Moment loads allow for a concentrated moment (length*force) to be applied to the model.

Moments are displayed with a double-headed vector with the letter M at the tail end in the modeling window.

Pressures

Configuration 4 - Pressure loads allow for a pressure (force*length^2) to be applied to the model.

For most solvers, the pressure load is considered as force/area, therefore the magnitude of the pressure is multiplied by the calculated area of the elements to which it is applied and resolved as concentrated force loads at the associated nodes.

Pressures are displayed as a vector with the letter P at the tail end in the modeling window.

Temperatures

Configuration 5 - Temperature loads allow for a concentrated temperature to be applied to the model.

Temperatures are displayed as a vertical line with the letter T at the top in the modeling window.

Velocities

Configuration 8 - Velocity loads allow for a velocity (length/time) to be applied to the model.

Velocities are displayed as a vector with the letter V at the tail end in the modeling window.

Abaqus Cards

Loads or constraints that are to be used as history data (under *STEP) should be collected into load collectors with the HISTORY card image. These load collectors also need to be added to the corresponding load steps (*STEP). In contrast, loads or constraints for model data should be collected into load collectors with INITIAL_CONDITION card image. They will automatically be written out in the model portion of the Abaqus input deck.

> **Note:** All loads and boundary conditions on sets can be expanded to individual nodes and elements by selecting the **Expand load on sets** option in the **File Options** dialog, which is invoked upon importing a solver deck. If a **HMLOAD_SETS_EXPAND comment is found in the input file, all loads and boundary conditions on sets are expanded to individual nodes and elements.

Card	Supported Load Types	Description
*BOUNDARY (electric potential, dof 9)	Flux	Specifies flux boundary conditions for piezoelectric analysis.
*BOUNDARY (structural)	Constraint	Creates structural boundary conditions.
*BOUNDARY (temperature, dof 11)	Temperatures	Specifies temperature boundary conditions.
*CECHARGE	Flux	Specifies concentrated electric charges for piezoelectric analysis.
*CECURRENT	Flux	Specifies concentrated current in electric conduction.

Card	Supported Load Types	Description
*CFLUX	Flux	Specify concentrated fluxes in heat transfer or mass diffusion analyses.
*CLOAD	Force	Creates concentrated forces.
*CLOAD	Moment	Creates concentrated moments.
*COUPLING	Constraint	Define a surface-based coupling constraint
*DECHARGE	Pressure	Distributes electric charges for piezoelectric analysis.
*DFLUX	Pressure	Specify distributed fluxes in heat transfer or mass diffusion analyses.
*DISTRIBUTING	Constraint	Define a distributing coupling constraint
*DISTRIBUTING COUPLING	Elements	Specify nodes and weighting for distributing coupling elements
*DLOAD	Pressure	Specifies distributed loads
*FILM	Pressure	Define film coefficients and associated sink temperatures.
*KINEMATIC	Multi-point Constraints	Define a kinematic coupling constraint
*KINEMATIC COUPLING	Multi-point Constraints	Constrain all or specific degrees of freedom of a set of nodes to the rigid body motion of a reference node
*MPC	Multi-point Constraints	Define multi-point constraints
*RADIATE	Pressure	Specify radiation conditions in heat transfer analyses
*TEMPERATURE	Temperature	Specifies predefined temperature field.

ANSYS Cards

Card	Supported Load Types	Description
BF	Flux	Defines a nodal body force load.
BF_FLUE	Flux	
BF_HGEN	Flux	

Card	Supported Load Types	Description
BF_TEMP	Temperatures	
BFE_FLUE	Flux	Defines an element body force load.
BFE_HGEN	Flux	
BFE_TEMP	Flux	
CE_STRUCT	Equation	
CE_THERM	Equation	
CE_MAG	Equation	
CE_ELEC	Equation	
ConvBulkTe	Pressure	
ConvFilmCo	Pressure	
D_A	Constraint	Vector magnetic potential.
D_CONSTRNT	Constraint	Defines DOF constraints at nodes.
D_MAG	Constraint	Scalar magnetic potential.
D_PRES	Constraint	
D_TEMP	Temperature	
D_VOLT	Constraint	
F_FLOW	Flux	Specifies force loads at nodes.
F_HEAT	Flux	
FLOTRAN	Pressure	Specifies "FLOTRAN data settings" as the subsequent status topic. **Note:** FLOTRAN surface load label “FSI [fluid-structure interaction flag]” is available under pressure load. You must use DOF1 to add value for this label.
FORCE	Force	Selects the element nodal force type for output.
FORCE2	Moment	

Card	Supported Load Types	Description
FSI	Pressure	
HFLUX	Pressure	
IC_A	Constraint	
IC_CONSTRN	Constraint	Specifies initial conditions at nodes.
IC_MAG	Constraint	
IC_PRES	Constraint	
IC_TEMP	Temperature	
IC_VOLT	Constraint	
PRESSURE	Pressure	
RDSF_EMI	Pressure	
RDSF_ENCL	Pressure	
SFE	Pressure Convection Heatflux	Defines elemental surface load. **Note:** Structural, thermal and Fluid labels are covered.
SFE	Structural Thermal Fluid	Surface load Structural label: PRES Thermal label: CONV, HLFUX Fluid label: FSI

LS-DYNA Cards

Several load types cause three cards to be output for x, y, and z components. During input, these are grouped into one load. Loads cannot be applied to sets, components, or boxes. Load curves are input and output. Use the Card Editor to select load curves. Unless mentioned in the Notes column, load cards cannot be edited.

Card	Supported Load Types	Description
*BOUNDARY_PRESCRIBE	Constraints Type 2; Card 26, VAD = 2	Define an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes.

Card	Supported Load Types	Description
	DEATH BIRTH	DOF 4, -4, 8, -8, 9, -9, 10, -10, 11, -11 are not supported.
*BOUNDARY_PRESCRIBE	Velocity Type 1; Card 26; VAD = 0	Define an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes. DOF 4, -4, 8, -8, 9, -9, 10, -10, 11, -11 are not supported
*BOUNDARY_PRESCRIBE	Acceleration Type 1 Card 26 VAD = 1	Define an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes. DOF 4, -4, 8, -8, 9, -9, 10, -10, 11, -11 are not supported
*BOUNDARY_PRESCRIBE	Velocity	
*BOUNDARY_PRESCRIBE	Constraints Type 2; Card 26, VAD = 2 dynaName DEATH BIRTH	
*BOUNDARY_PRESCRIBE	PID DOF VAD LCID SF VID DEATH BIRTH	Define an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes. RIGID_LOCAL and _SET options are supported.
*BOUNDARY_PRESCRIBE	Dyna_Name PID DOF VAD LCID SF	Define an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes.

Card	Supported Load Types	Description
	VID DEATH BIRTH	
*BOUNDARY_PRESCRIBE	PID DOF VAD LCID SF VID DEATH BIRTH Title	
*BOUNDARY_PRESCRIBE	Dyna_Name PID DOF VAD LCID SF VID DEATH BIRTH	
*BOUNDARY_PRESCRIBE	NSIDDOF VAD LCID SF VID DEATH BIRTH TITLE	Define an imposed nodal motion (velocity, acceleration, or displacement) on a node or a set of nodes.
*BOUNDARY_PRESCRIBE	Dyna_Name NSID DOF VAD	

Card	Supported Load Types	Description
	LCID SF VID DEATH BIRTH	
*BOUNDARY_SPC_NODE	Constraints Type 1; Card 13 SPC CID	Define nodal single point constraint
*BOUNDARY_SPC_NODE_	Constraints dynaName CID	
*BOUNDARY_TEMPERATU	Temperature LCID LOC	Define temperature boundary conditions for a thermal or coupled thermal/structural analysis.
*CONSTRAINED_GLOBAL	Constraints	Define a global boundary constraint plane.
*INITIAL_TEMPERATURE_	Temperatures LOC	Define initial nodal point temperatures using nodal set IDs or node number.s
*INITIAL_VELOCITY	Type 2 Card 30 INITV = 3	Define initial nodal point translational velocities using nodal set IDs. For structured output, global velocity is set to 0.0. For structured input, non-zero values for INITV = 1 or INITV = 5 create velocities. INITV values of 2, 4, 6, and 7 are ignored.
*INITIAL_VELOCITY_NOD	Rotation	Define initial nodal point velocities for a node.
*LOAD_BEAM_ELEMENT	Pressure	Defines load on beam elements
*LOAD_MASK	N/A	Apply a distributed pressure load over a three-dimensional shell part
*LOAD_NODE_POINT	Force Type 1 Card 23	Apply a concentrated nodal force to a node or a set of nodes. LS-DYNA Load Configs 1, 2, 3 and 4

Card	Supported Load Types	Description
	Point Loads FollowerForce	A load curve can be selected for these loads.
*LOAD_NODE_POINT	Moment Type 1 Card 23 Point Loads	Apply a concentrated nodal force to a node or a set of nodes. LS-DYNA Load Configs 5, 6 7 and 8.
*LOAD_SEGMENT	Pressure LCID AT	Apply the distributed pressure load over one triangular or quadrilateral segment defined by four, six or eight nodes.
*LOAD_SEGMENT_ID	Pressure LCID AT	Apply the distributed pressure load over one triangular or quadrilateral segment defined by four, six or eight nodes.
*LOAD_SHELL_ELEMENT	Pressure AT LCIDoption	Apply the distributed pressure load over one shell element or shell element set.
*LOAD_SHELL_ELEMENT_	Pressure LCID AT	Apply the distributed pressure load over one shell element or shell element set.
*LOAD_SHELL_PRESSURI	Pressure Type 2 Card 24 Pressure BC	Apply the distributed pressure load over one shell element or shell element set.
*LOAD_THERMAL_CONST	Temperature N/A	Define nodal sets giving the temperature that remains constant for the duration of the calculation.
*LOAD_THERMAL_VARIAI	Temperature TS	Define nodal temperature that is variable during the calculation.

Card	Supported Load Types	Description
	LCID	

Nastran Cards

Note: Other loads such as SPCADD, MPCADD, FREQ, FREQ1, EIGR, EIGRL, EIGC, EIGP, EIGB, GRAV, and RFORCE are supported as load collectors.

Card	Supported Load Types	Description
ASET	Constraints	Defines degrees-of-freedom in the analysis set (a-set)
ASET1	Constraints	Defines degrees-of-freedom in the analysis set (a-set)
BNDFIX1	Constraints	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.
BOLTFOR	Flux	
BSET1	Constraints	Defines analysis set (a-set) degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode synthesis calculations.
CSET1	Constraints	Defines analysis set (a-set) degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component modes calculations.
DAREA	Constraints	Defines scale (area) factors for static and dynamic loads. In dynamic analysis, DAREA is used in conjunction with RLOADi and TLOADi entries.
DEFORM	Flux	Defines enforced axial deformation for one-dimensional elements for use in statics problems.
FORCE	Force	Requests the form and type of element force output or particle velocity output in coupled fluid-structural analysis.
MOMENT	Moment	Defines a static concentrated moment at a grid point by specifying a scale factor and a vector that determines the direction.

Card	Supported Load Types	Description
OMIT1	Constraints	Defines degrees-of-freedom to be excluded (o-set) from the analysis set (a-set).
PLOAD	Pressure	Defines a uniform static pressure load on a triangular or quadrilateral surface comprised of surface elements and/or the faces of solid elements.
PLOAD1	Pressure	Defines concentrated, uniformly distributed, or linearly distributed applied loads to the CBAR or CBEAM elements at user-chosen points along the axis.
PLOAD2	Pressure	Defines a uniform static pressure load applied to CQUAD4, CSHEAR, or CTRIA3 two-dimensional elements. The THRU field is supported for feinput only. On export, additional pressure cards for the range specified are written.
PLOAD4	Pressure	Defines a pressure load on a face of a CHEXA, CPENTA, CTETRA, CTRIA3, CTRIA6, CTRIAR, CQUAD4, CQUAD8, or CQUADR element. The THRU field is supported for feinput only. On export, additional pressure cards for the range specified are written. Unequal nodal pressures are now supported. The average pressure value is used as the magnitude of the pressure for visualization only. The individual field values, P1-P4, can be viewed or edited using the card editor. Updating the magnitude of pressure from the Pressures panel will have no effect on PLOAD4 cards defined using unequal nodal pressures.
QBDY1	Flux	Defines a uniform heat flux into CHBDYj elements.
QSET1	Constraints	Defines generalized degrees-of-freedom (q-set) to be used for generalized dynamic reduction or component mode synthesis.
QVOL	Flux	Volume Heat Addition - Defines a rate of volumetric heat addition in a conduction element.

Card	Supported Load Types	Description
SPC	Constraints	Defines a set of single-point constraints and enforced motion (enforced displacements in static analysis and enforced displacements, velocities or acceleration in dynamic analysis). Constraints on nodes are supported through SPC cards. PS field in GRID card is not supported. Upon import, any PS entry on the GRID card will be converted into an SPC card.
SPC1	Constraints	Defines a set of single-point constraints. Supported for feinput only. On export, equivalent SPC cards are written. Alternate format with THRU in the fifth field is supported.
SPCD	Constraints	Defines an enforced displacement value for static analysis and an enforced motion value (displacement, velocity or acceleration) in dynamic analysis.
SUPORT	Constraints	Defines determinate reaction degrees-of-freedom in a free body.
SUPORT1	Constraints	Defines determinate reaction degrees-of-freedom (r-set) in a free body-analysis. SUPORT1 must be requested by the SUPORT1 Case Control command.
TIC(D)	Constraints	Transient Initial Condition - Defines values for the initial conditions of variables used in structural transient analysis.
TIC(V)	Constraints	Transient Initial Condition - Defines values for the initial conditions of variables used in structural transient analysis.
TEMP	Temperatures	Defines temperature at grid points for determination of thermal loading, temperature-dependent material properties, or stress recovery.
TEMPBC	Temperatures	Defines the temperature boundary conditions for heat transfer analysis.
USET	Constraints	Defines a degree-of-freedom set.

Card	Supported Load Types	Description
USET1	Constraints	Defines a degrees-of-freedom set.

OptiStruct Cards

General boundary conditions, such as loads and constraints, should not be collected into specific load collectors. Organizing loads and constraints into a specific load collector may result in an error termination.

Card	Supported Load Types	Description
ASET	Constraints	Defines the boundary degrees-of-freedom of a superelement assembly for matrix reduction.
ASET1	Constraints	Defines the boundary degrees-of-freedom of a superelement assembly for matrix reduction.
DAREA	Constraints	Defines scale (area) factors for dynamic loads. `DAREA` is used in conjunction with `RLOAD1`, `RLOAD2`, `TLOAD1`, and `TLOAD2` entries.
DELAY	Constraints	Defines the time delay term τ in the equations of the dynamic loading function. `DELAY` is used in conjunction with `RLOAD1`, `RLOAD2`, `TLOAD1`, and `TLOAD2` entries.
DEFORM	Flux	Defines enforced axial deformation for one-dimensional elements for use in statics problems.
DPHASE	Constraints	Defines the phase lead term θ in the equation of the dynamic loading function. `DPHASE` is used in conjunction with `RLOAD1` and `RLOAD2` entries.
FORCE	Force	Defines a static force at a grid point or a *SET* of grid points by specifying a vector.
FORCE1	Force	Used to define a static force by specification of a value and two grid points that determine the direction. It can also be used to define the EXCITEID field (Amplitude "A") of dynamic loads in `RLOAD1`, `RLOAD2`, `TLOAD1` and `TLOAD2` Bulk Data Entries. Additionally, the `FORCE1` entry can be defined as Follower Loads in Large Displacement Nonlinear Analysis.

Card	Supported Load Types	Description
MBFRC	Force	Defines a constant force at a grid point by specifying a vector.
MBFRCC	Force	Defines a curve force at a grid point by specifying a vector.
MBMNT	Moment	Defines a constant moment at a grid point by specifying a vector.
MBMNTC	Moment	Defines a curve moment at a grid point by specifying a vector.
MOMENT	Moment	Defines a static moment at a grid point or a *SET* of grid points by specifying a vector.
MOMENT1	Moment	Defines a static moment by specification of a value and two grid points, which determine the direction. It can also be used to define the *EXCITEID* field (Amplitude *A*) of dynamic loads in `RLOAD1`, `RLOAD2`, `TLOAD1` and `TLOAD2` Bulk Data Entries.
MOTNG	Constraint	Defines a constant grid point motion.
MOTNGC	Constraint	Defines a grid point motion vs. time by specifying a curve.
PLOAD	Pressure	Defines a static pressure load on a triangular or quadrilateral element. It can also be used to define the *EXCITEID* field (Amplitude "*A*") of dynamic loads in `RLOAD1`, `RLOAD2`, `TLOAD1` and `TLOAD2` Bulk Data Entries.
PLOAD1	Pressure	Defines concentrated, uniformly distributed, or linearly distributed applied loads to the `CBAR` or `CBEAM` elements or a *SET* of such elements at user-chosen points along the axis. It can also be used to define the *EXCITEID* field (Amplitude "*A*") of dynamic loads in `RLOAD1`, `RLOAD2`, `TLOAD1` and `TLOAD2` Bulk Data Entries.
PLOAD2	Pressure	Defines a uniform static pressure load applied to two-dimensional elements, or a *SET* of such elements.
PLOAD4	Pressure	Defines a load on a face of a `HEXA`, `PENTA`, `TETRA`, `PYRA`, `TRIA3`, `TRIA6`, `QUAD4`, or `QUAD8` element.

Card	Supported Load Types	Description
		It can also be used to define the *EXCITEID* field (Amplitude "*A*") of dynamic loads in `RLOAD1`, `RLOAD2`, `TLOAD1` and `TLOAD2` Bulk Data Entries. Additionally, the `PLOAD4` entry can be defined as Follower Loads in Large Displacement Nonlinear Analysis.
QBDY1	Flux	Defines a uniform heat flux for `CHBDYE` elements.
QVOL	Flux	Defines a rate of volumetric heat addition in a conduction element.
SPC	Constraint	Defines sets of single-point constraints, enforced displacements for static analysis, and thermal boundary conditions for heat transfer analysis.
SPCD	Constraint	Defines an enforced displacement value for static analysis, an enforced displacement, velocity or acceleration for dynamic analysis and a thermal boundary condition for heat transfer (or transient heat transfer) analysis. It can also be used to define the *EXCITEID* field (Amplitude "A") of dynamic loads in `RLOAD1`, `RLOAD2`, `TLOAD1` and `TLOAD2` Bulk Data Entries.
SUPORT	Constraint	Defines determinate reaction degrees-of-freedom in a free body.
SUPORT1	Constraint	Defines determinate reaction degrees-of-freedom in a free body.
TEMP	Temperature	Defines temperature at grid points or a `SET` of grid points for determination of Thermal Loading and Stress recovery.
TIC(D) or (V)	Constraint	Defines values for the initial conditions of variables used in structural transient analysis and explicit analysis. Both displacement and velocity values may be specified at independent degrees-of-freedom.
USET	Constraint	Defines a set of degrees-of-freedom.

Card	Supported Load Types	Description
USET1	Constraint	Defines a set of degrees-of-freedom.

PAM-CRASH Cards

Card	Supported Load Types	Description
ACC3D /	Acceleration	Imposed accelerations
BOUNC /	Constraints	Define boundary condition
CONLO /	Force(1)	Concentrated nodal load
DIS3D /	Constraints	Imposed displacement
DIS3DM /	Constraints	Imposed minimum displacement
DIS3DX /	Constraints	Imposed maximum displacement
INVEL /	Velocity	Define initial velocity
PREFA /	Pressure(1)	Pressure on shells
RAC3D /	Acceleration	Imposed rotational acceleration
RAN3D /	Constraints	Imposed angular rotations
RDA3D /	Acceleration	Radial 3D boundary conditions
RDD3D /	Constraints	
RDV3D /	Velocity	Radial 3D boundary conditions
RVE3D /	Velocity	Imposed rotational velocities
RWALL /		Rigid wall definition
SECFO_PLANE /		

Card	Supported Load Types	Description
VEL3D /	Velocity	Imposed velocities

Permas Cards

Card	Supported Load Types	Description
$ADDMODES	Constraints	Definition of static mode shapes to be added to the set of eigenmodes used for transformation to modal space. If static mode shapes will be added directly to nodes or nodesets (SOURCE=INPUT), the $ADDMODES can be created through the Constraints panel. Click sysid to specify the system regarding to which the modes shall be applied. Use the DOFTYPE button to select an option: DISP, TEMP, PRES, POTE and MATH.
$ADDMODES	Pressure	Definition of static mode shapes to be added to the set of eigenmodes used for transformation to modal space. If mode shapes will be applied based on the natural deformation of elements (SOURCE=INPUT) the $ADDMODES keyword needs to be created here.
$CONLOAD	Force	Definition of concentrated loads at nodal point degrees of freedom.
$CONLOAD	Moment	Definition of concentrated loads at nodal point degrees of freedom.
$DISLOAD PRESS	Pressure	Definition of pressure loads for elements, where loads are given for elements or element sets. Applicable to shells and solids, but also axisymmetric solid elements. Therefore please apply a pressure on an HM shell element. Face identifiers are written in this case. On import IDS ELNODES or NODES will be resolved into ELGEO (face identifiers).

Card	Supported Load Types	Description
$DISLOAD TEMP	Pressure	Nodal temperatures defined on elements or element sets.
$DISLOAD TEMPFILM	Pressure	Surrounding temperatures for convective heat transfer applied on elements or element sets.
$DISLOADN TEMP	Temperature	Nodal temperatures definition applied on nodes or node sets
$DISLOADN TEMPFILM	Temperature	Surrounding temperatures for convective heat transfer applied on nodes or node sets.
$INIVAL	Constraints	Definition of initial values for nodal point degrees of freedom. For $INIVAL source parameter INPUT is currently supported to specify the initial values based on nodal points.
$INERTIA	Pressure	Definition of inertia forces acting on entire component or element sets. Available are force distributions due to linear acceleration, constant or accelerated rotation and coriolis acceleration. Only ACCELERATION and GRAVITY are supported. This card is created in the Pressure panel. Assign to a set of elements, and the set statement displays in the card image. To create the card without a set, create a pressure on a 'dummy' element; the card will be created without a set and can be applied to the whole model.
$INERTIAX	Pressure	Definition of inertia forces acting on entire axisymmetric component or element sets. Available are force distributions due to linear acceleration and constant rotation. Only ACCELERATION and GRAVITY are supported. This card is created in the Pressure panel. Assign to a set of elements, and the set statement displays in the card image. To create the card without a set, create a pressure on a 'dummy' element; the card will be created without a set and can be applied to the whole model.
$MPC GENERAL/ $MPCVAL	Equation	Multipoint constraint definition.

Card	Supported Load Types	Description
		Both cards are created simultaneously in the Equation panel. The equation needs to be placed into a load collector with card image SUPRESS. By attaching the load collector to a load step with 'CONSTRAINTS' attribute set, the $MPCVAL card gets written in the desired $CONSTRAINTS variant.
$PRESCRIBE/ PREVAL	Constraints	Prescribed degrees of freedom/Nodal point values (implemented as HyperMesh constraints)
$SUPPRESS	Constraints	Suppressed degrees of freedom

Samcef Cards

Card	Supported Load Types	Description
.CLM FIX	Constraint	Defines a set of single-point constraints
.CLM DEP	Constraint	Defines sets of enforced displacements
.CLM CHA COMP 123	Force	Defines a static force at a grid point by specifying a vector and a value.
.CLM FOL COMP 123	Force	Defines a follower force at a grid point by specifying a vector and a value
.CLM CHA COMP 456	Moment	Defines a static moment at a grid point by specifying a vector and a value.
.CLM FOL COMP 456	Moment	Defines a follower moment by specifying a vector and a value

Card	Supported Load Types	Description
.CLM PRESSURE	Pressure	Defines a static pressure load on any elements type

Equations

Equation entities contain mathematical equations that define more complex loads. They are used to define linear constraints in local and global coordinate systems.

Every equation must be organized into one load collector, and therefore are mutually exclusive to a load collector.

Equations are displayed as lines between the dependent node and the independent node(s) with EQ displayed at the dependent node of the equation in the modeling window.

Equations are used in Nastran as MPC (Multipoint Constraint Equation) or in Abaqus as *equation.

Equations are organized into load collectors.

Abaqus Cards

Loads or constraints that are to be used as history data (under *STEP) should be collected into load collectors with the HISTORY card image. These load collectors also need to be added to the corresponding load steps (*STEP). In contrast, loads or constraints for model data should be collected into load collectors with INITIAL_CONDITION card image. They will automatically be written out in the model portion of the Abaqus input deck.

> **Note:** All loads and boundary conditions on sets can be expanded to individual nodes and elements by selecting the **Expand load on sets** option in the **File Options** dialog, which is invoked upon importing a solver deck. If a **HMLOAD_SETS_EXPAND comment is found in the input file, all loads and boundary conditions on sets are expanded to individual nodes and elements.

Card	Description
*EQUATION	Define linear multi-point constraints.

Card	Description
	Note: Explicit node IDs are supported. Node sets are not supported. Equations are considered as loads and therefore, they are collected in load collectors. Upon export, they write to the bulk data portion of the Abaqus deck.

ANSYS Cards

Card	Description
CE	Defines a constraint equation relating to degrees of freedom.

LS-DYNA Cards

Several load types cause three cards to be output for x, y, and z components. During input, these are grouped into one load. Loads cannot be applied to sets, components, or boxes. Load curves are input and output. Use the Card Editor to select load curves. Unless mentioned in the Notes column, load cards cannot be edited.

Card	Description
*CONSTRAINED_LINEAR	Define linear constraint equations between displacements and rotations, which can be defined in a local coordinate system.
*CONSTRAINED_LINEAR_	Define linear constraint equations between displacements and rotations, which can be defined in global coordinate systems.

Nastran Cards

Note: Other loads such as SPCADD, MPCADD, FREQ, FREQ1, EIGR, EIGRL, EIGC, EIGP, EIGB, GRAV, and RFORCE are supported as load collectors.

Card	Description
MPC	Defines a multipoint constraint equation of the form.

Card	Description
	Note: Individual weight factors can be created on the nodes of an MPC equation using the update functionality in the Equations panel.

OptiStruct Cards

Card	Description
MPC	Defines a multipoint constraint equation of the form. **Note:** Bulk Data Entry

Permas Cards

Card	Description
$MPC GENERAL	General linear constraint equation.

Part Assemblies and Parts

HyperMesh supports CAE and PDM parts and part assemblies. CAE parts and part assemblies can be imported into HyperMesh, or can be manually created in the Part Browser. PDM parts and part assemblies can be imported into HyperMesh via a PDM generated `PLMXML` file. You cannot modify or delete any part assembly or part level entities.

Part Assemblies

Part assemblies collect and organize sub part assemblies and/or parts.

A part assembly can temporarily contain a component if it does not contain part assemblies or parts.

You can create, review, and edit part assemblies in the Part Browser.

Parts

A part represents and organizes subsystems of a physical part into a hierarchal data structure, which reflects the data structure of the product being modeled.

Parts can be organized into a part assembly in the Part Browser. A part can only contain components.

You can create, review, and edit parts in the Part Browser.

System Collectors and Systems

System Collectors

System collectors collect and organize systems.

LS-DYNA Cards

System collector cards can be previewed, but not edited.

Card	Description
*DEFINE_TRANSFORMATI	Define a transformation for the `INCLUDE_TRANSFORM` keyword option. **Note:** Transformations can be created using the Transformation Manager.
*INCLUDE_PATH	Defines path for the include file location. **Note:** Card Edit is not available. Keyword details can be accessed from the Include file options in the Include browser.
*INCLUDE_STAMPED_PAF	Attaches a file that has stamping results for a part in the model as an include file. **Note:** Card Edit is not available. Keyword details can be accessed from the Include file options in the Include browser.
*INCLUDE_STAMPED_PAF	Attaches a file that has stamping results for a collection of parts (set) in the model as an include file.

Card	Description
	Note: Card Edit is not available. Keyword details can be accessed from the Include file options in the Include browser.
*INCLUDE_TRANSFORM	Include file that supports offset on its content IDs and transforms on its contents. **Note:** HyperMesh support offset on the entity ID's. During import, offsets are applied on to the ID's of the corresponding include_transform file contents. During export, offset is subtracted from the ID's. The current release only supports Input and output therefore the offsets cannot be changed.
*NODE_TRANSFORM	Transformation defined on node set. **Note:** Transformations can be created using the Transformation Manager.

PAM-CRASH Cards

Card	Description
TRANSFORMATION /	

Radioss Cards

Card	Description
/TRANSFORM/ROT	Defines a rotation for a node group around a defined axis, center of rotation and rotation angle. **Note:** Block Format Keyword
/TRANSFORM/TRA	Defines a translation for a node group with a defined vector. **Note:** Block Format Keyword
/TRANSFORM/SCA	Defines a scale for a node group with defined scale center and scale factor.

Card	Description
	Note: Block Format Keyword
/TRANSFORM/SYM	Defines a symmetry for a node group normal to the plane defined by a vector. **Note:** Block Format Keyword

Systems

System entities, commonly called coordinate systems, can be defined as rectangular, cylindrical, or spherical coordinate systems.

Every system must be organized into one system collector, and therefore are mutually exclusive to a system collector.

Several systems may be nested. There are two types of system assignments to entities; as a reference system, or as a displacement system. A system may be a reference system, a displacement system, or both.

Reference System

Defines the geometric positions of entities.
Entities that can be assigned a reference system include systems, nodes, and loads. By default, each of these entities is defined in the global system with an ID of zero. Entity data is always displayed and reviewed transformed into the global system. When a reference system is deleted, the position of the entity assigned that reference system is maintained relative to the global system in the transformation process. For example, if you define the nodes of a cylindrical structure in a cylindrical reference coordinate system, and then delete the cylindrical reference coordinate system in which the nodes are defined, the model retains its cylindrical shape and also its location in space but is now referenced to the global system.

Displacement System

Defines the nodal degree of freedom coordinate system assigned to a node.
The only entity that may be assigned a displacement system is a node. When you delete a displacement system, the nodal degrees of freedom are not transformed to the global system, so all degree of freedom definitions after the deletion of the displacement system are now simply in the global system.

Abaqus Cards

A system can be exported as a *SYSTEM or *TRANSFORM card depending on the nodal assignment of the system. To export an *ORIENTATION card it is required to enable the option in the card image of the system card.

Card	Description
*ORIENTATION	Define a local axis system for material or element property definition, for kinematic coupling constraints, for free directions for inertia relief loads, or for connectors. **Note:** The *ORIENTATION card needs a name in Abaqus. Since systems do not have a name, a name needs to be entered in the system card image. The restriction of one system per system collector has been removed with version 10.0 – SA1-130. DEFINITION = NODES option with only two nodes is converted to DEFINITION=COORDINATES upon import from an input file. *ORIENTATION with SYSTEM = Z RECTANGULAR is converted to RECTANGULAR upon import from an input file.
*SYSTEM	Specify a local coordinate system in which to define nodes. **Note:** Whenever a system is assigned to nodes with the set reference option from the Systems panel activated, a *SYSTEM card is exported before the node block of its assignment.
*TRANSFORM	Specify a local coordinate system at nodes. **Note:** If assigned to individual nodes, on export each *TRANSFORM card creates references to an automatically generated *NSET card. This *NSET card is followed by the list of the nodes that are assigned to the coordinate system with the set displacement option. Systems can be assigned to node sets with the same assignment procedure.

ANSYS Cards

Card	Description
LOCAL	Defines a local coordinate system by location and orientation. **Note:** Even if KCN>10, 10 is added to the current value.
LOCAL	Defines a local coordinate system by location and orientation.

Card	Description
	Note: Type = PRM not supported

EXODUS Cards

Card	Description
Coordinate Frame	Defined for reference to materials and boundary conditions.

LS-DYNA Cards

Systems cards can be previewed, but not edited.

Card	Description
*DEFINE_COORDINATE_N	Define a local coordinate system with three nodes.
*DEFINE_COORDINATE_S	Define a local coordinate system with three points.
*DEFINE_COORDINATE_V	Define a local coordinate system with two vectors.

Nastran Cards

Card	Description
CORD1R	Defines a rectangular coordinate system using three grid points.
CORD2R	Defines a rectangular coordinate system using the coordinates of three points.
CORD1C	Defines a cylindrical coordinate system using three grid points.
CORD2C	Defines a cylindrical coordinate system using the coordinates of three points.
CORD1S	Defines a spherical coordinate system by reference to three grid points.

Card	Description
CORD2S	Defines a spherical coordinate system using the coordinates of three points.

OptiStruct Cards

Card	Description
CORD1C	Defines a cylindrical coordinate system using three grid points. The first point is the origin, the second lies on the Z-axis, and the third lies in the X-Z plane. **Note:** Bulk Data Entry Created from the create by node reference subpanel when cylindrical is the chosen type.
CORD1R	Defines a rectangular coordinate system using three grid points. The first point is the origin, the second lies on the Z-axis, and the third lies in the X-Z plane. **Note:** Bulk Data Entry Created from the create by node reference subpanel when rectangular is the chosen type.
CORD1S	Defines a spherical coordinate system using three grid points. The first point is the origin, the second lies on the Z-axis, and the third lies in the X-Z plane. **Note:** Bulk Data Entry Created from the create by node reference subpanel when spherical is the chosen type.
CORD2C	Defines a cylindrical coordinate system using three grid points specified with respect to a reference coordinate system. The coordinates of the three non-collinear grid points are used to uniquely define the coordinate system. The first point is the origin, the second lies on the Z-axis, and the third lies in the X-Z plane.

Card	Description
	Note: Bulk Data Entry Created from the create by axis direction subpanel when cylindrical is the chosen type. Various combinations of axes and planes are allowed to be indicated in the create by axis direction subpanel, but will write out the appropriate coordinates to define the z- axis and the x-z plane.
CORD2R	Defines a rectangular coordinate system by using three grid points. The coordinates of the three non-collinear grid points are used to uniquely define the coordinate system. The first point is the origin, the second lies on the Z-axis, and the third lies in the X-Z plane. **Note:** Bulk Data Entry Created from the create by axis direction subpanel when rectangular is the chosen type. Various combinations of axes and planes are allowed to be indicated in the create by axis direction subpanel, but will write out the appropriate coordinates to define the z- axis and the x-z plane.
CORD2S	Defines a spherical coordinate system three grid points. The coordinates of the three non-collinear grid points are used to uniquely define the coordinate system. The first point is the origin, the second lies on the Z-axis, and the third lies in the X-Z plane. **Note:** Bulk Data Entry Created from the create by axis direction subpanel when spherical is the chosen type. Various combinations of axes and planes are allowed to be indicated in the create by axis direction subpanel, but will write out the appropriate coordinates to define the z- axis and the x-z plane.
CORD3R	Defines a rectangular coordinate system using three grid points. The first point is the origin, the second lies on the X-axis, and the third lies in the X-Y plane.

Card	Description
	Note: Bulk Data Entry Created from the create by node reference subpanel when rectangular is the chosen type.
CORD4R	Defines a rectangular coordinate system using three grid points specified with respect to the basic coordinate system. The coordinates of the three non-collinear grid points are used to uniquely define the coordinate system. The first point is the origin, the second lies on the X-axis, and the third lies in the X-Y plane. **Note:** Bulk Data Entry Created from the create by axis direction subpanel when rectangular is the chosen type. Various combinations of axes and planes are allowed to be indicated in the create by axis direction subpanel, but will write out the appropriate coordinates to define the z- axis and the x-z plane. Editing the card image for a CORD2R will allow you to define a CORD4R.

PAM-CRASH Cards

Card	Description
FRAME /	Local frame definition - system collectors **Note:** If there is a $HMMOVE directive found for a system and that system collector exists in the model, the system is placed in that collector. Otherwise, a separate system collector is made for the frame.
FRAME /	Local frame definition - systems **Note:** If a base node is not given, system is created at the global origin 0, 0, 0. In case of frame definition with nodes, system is created at the first node.

Card	Description
TRSFM /	Select elements and nodes subject to transformation.

Permas Cards

Card	Description
$RSYS	Reference system **Note:** Assign the system to nodes to write the RSYS parameter in to the nodal coordinates card $COORD.
$ROTB	Analysis or displacement system assigned to nodes. **Note:** Assign a system as displacement system to nodes to receive this card.

Radioss Cards

Card	Description
/FRAME/FIX	Describes the frames. **Note:** Block Format Keyword
/FRAME/MOV	Describes moving frames. Relative motion with respect to a reference frame. **Note:** Block Format Keyword
/FRAME/MOV2	Describes moving frames. Relative motion with respect to a reference frame. Moving frame definition differs from `/FRAME/MOV`. **Note:** Block Format Keyword
/FRAME/NOD	Describes the node defined moving frame. **Note:** Block Format Keyword

Card	Description
/SKEW/FIX	Describes the fixed skew frames. **Note:** Block Format Keyword
/SKEW/MOV	Describes a moving local coordinate system. **Note:** Block Format Keyword
/SKEW/MOV2	Describes a moving local coordinate system. **Note:** Block Format Keyword

Samcef Cards

Card	Description
.FRA	Coordinate system definition **Note:** I frame_nr TYPE chosen_frame_type ORIGIN frame_origin V1 axis_definition V2 axis_definition V3 axis_definition

Vector Collectors and Vectors

Vector Collectors

Vector collectors collect and organize vectors.

Nastran Cards

Vectors can be used to define orientation directions for some 1D elements and forces, or to define the SNORM card. For orientation vectors, it is not necessary to load any card image data onto the vector collector. For SNORM vectors, you must load the SNORM card image onto the vector collector. Once this is done, all vectors organized into that vector collector will write out as SNORM vectors to the Nastran bulk data file.

Card	Description
SNORM	Defines a surface normal vector at a grid point for CQUAD4, CQUADR, CTRIA3, and CTRIAR shell elements. **Note:** There is no card image associated with the collector. In order to view the actual SNORM cards, each vector must be individually card edited. Loading the SNORM card image onto the collector assigns the SNORM type onto all of the vectors contained in that collector.

PAM-CRASH Cards

Card	Description
ADAPT /	
PICK /	
SUBDF /	Substructure definition

Vectors

Vector entities allow for the definition of a vector in 3D space. Vectors can be created using three methods; base & magnitude, two nodes, or cross-product.

LS-DYNA Cards

Card	Description
*DEFINE_SD_ORIENTATI(	Define orientation vectors for discrete springs and dampers.

Card	Description
*DEFINE_VECTOR	Define a vector by defining the coordinates of two points.

Beamsection Collectors and Beamsection

Beamsection Collectors

Beamsection collectors collect and organize beamsections and are used in HyperBeam to organize 1D beam section data.

Nastran Cards

Card	Description
BELTS /	Output activation and format/file selection for kinematic animation output.

Beamsections

Beamsection entities store 1D beam cross-section data.

Beamsections can be created from geometry, elements, or from solver standard sections, that is, I-Sections, H-Sections, and so on.

All sections can be created and modified in HyperBeam. Generic sections and Standard sections can also be created in the Model Browser and modified in the Property Editor.

The default section type and attribute values assigned to beam section vary based on solver interface.

On import, each 1D beam property card within a solver deck is automatically imported as a beamsection entity and a property entity with associated beamsection. The beamsection entity holds the 1D beam section data (A, I, and so on..., and/or Dimensions) and is associated to the property entity which has a 1D property card image. The beamsection association to a property is what transfers the 1D section data to the 1D property solver card for export. Editing of all 1D beam section data is accomplished through HyperBeam.

Generic Sections

Generic sections define sections without defining actual cross-section geometry. Areas, inertias, centroids, and other coefficients are supported directly through spreadsheet data entry of values.

Shell Sections

Shell sections define thin cross-sections with geometric lines or 1D elements. Once the cross-section is created, it can be further edited in HyperBeam.

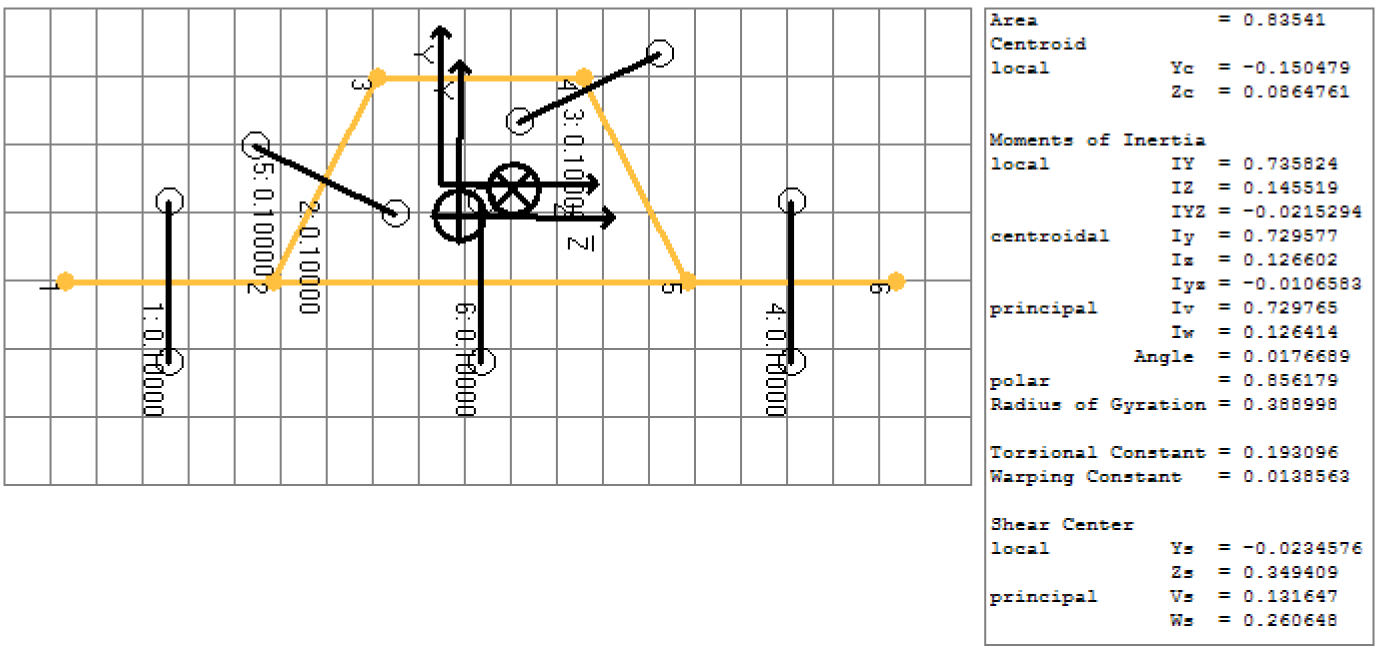


Figure 76:

Solid Sections

Solid sections to define solid cross-sections with surfaces, lines that form a closed loop, or 2D elements. Once the cross-section is created it can be further edited in HyperBeam.

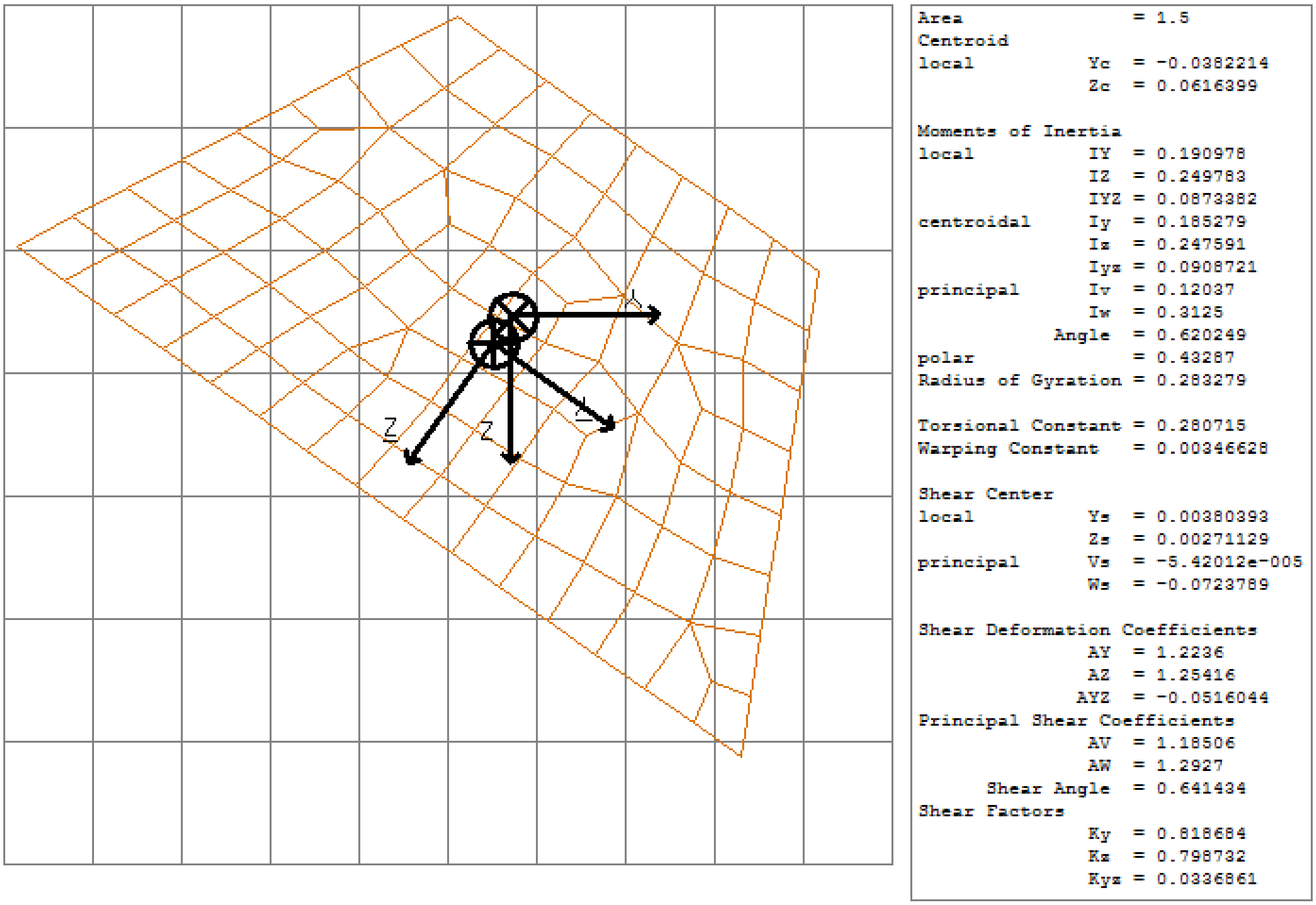


Figure 77:

Standard Sections

Standard sections to define solver supported cross-sections. Each supported solver interface has a library of supported solver cross-sections. For standards sections, only the dimensions of the section are necessary as input.

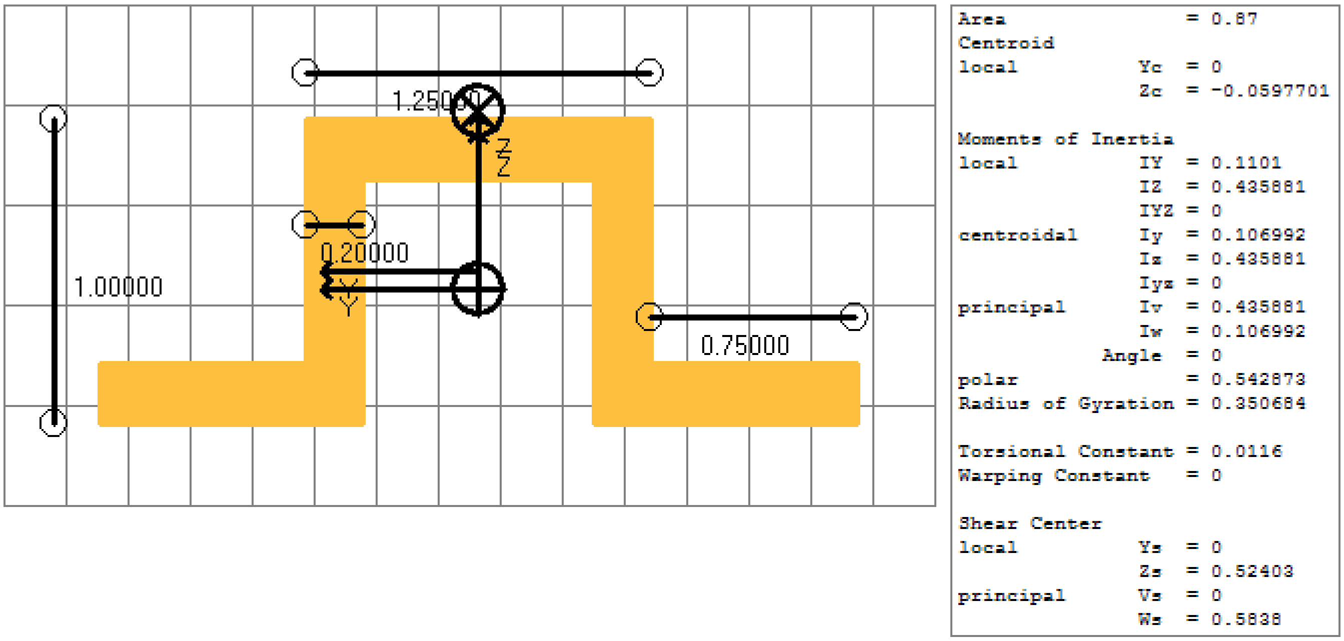


Figure 78:

Bags

Bags collect and organize entities.

Operations performed on a bag do not affect the entities collected within the bag. For example, if you delete a bag, the entities collected within the bag are not deleted.

The entities that are permissible for a bag entity to collect are determined by the configuration of the bag.

Currently, only the optimization configuration of bag entity can be created, edited, and deleted. Optimization problem configurations of the bag entity can be created, edited, and deleted in the Model Browser Optimization View.

All other configurations of bag entities can be created, edited, and deleted using the Tcl Modify Commands `*bagcreate` and `*bagentityupdate`.

Generic

Configuration 1 - Collect and organize any entities, including other bag entities.

Optimization Problem

Configuration 2 - Collect and organize Optimization entities.

- Design Variables
- Design Variable Links
- Objective
- Design Variable Property Relationships

- Objective References
- Optimization Constraints
- Optimization Constraint Screenings
- Optimization Controls
- Optimization Equations
- Optimization Responses
- Optimization Table Entries
- Discrete Design Variables

FBD Forces (All Loads)

Configuration 3 - Collect and organize nodes, element sets, systems, and load collectors.

FBD Forces (Applied Loads Only)

Configuration 4 - Collect and organize nodes, element sets, systems, and load collectors.

FBD Forces (Reaction Loads Only)

Configuration 5 - Collect and organize nodes, element sets, systems, and load collectors.

FBD Displacements

Configuration 6 - Collect and organize node sets, element sets, systems, and load collectors.

Resultant Force & Moment

Configuration 7 - Collect and organize systems, load collectors, and FBD cross-section bag entities.

FBD Cross-section

Configuration 8 - Collect and organize nodes, node sets, element sets, and systems.

ADM Part

Configuration 9 - Collect and organize any entities, including other bag entities.

ADM Material

Configuration 10 - Collect and organize any entities, including other bag entities.

Multibodies

Multibodies collect and organize ellipsoids, multibody planes, and multibody joints and are typically used in multi-body analysis.

Ellipsoids, multibody planes, and multibody joints can be organized into a multibody using the Organize panel. Every ellipsoid, multibody plane, and multibody joint must be organized into one, and only one, multibody and therefore are mutually exclusive to a multibody. Newly created ellipsoids, multibody planes, and multibody joints are automatically organized into the current multibody.

Operations performed on a multibody collector affect ellipsoids, multibody planes, and multibody joints within the multibody collector. For example, if you delete a multibody collector, the ellipsoids, multibody planes, and multibody jonts within the multibody collector are also deleted.

MADYMO

Card	Description
BODY.DEFORMABLE	Deformable body. body local system and center of gravity are not used. Enter the number of MODEs in MODE_LIST and select each applicable MODE element. Select the DAMPING check box to apply MODAL_DAMP.
BODY.FLEXIBLE_BEAM	Flexible beam. body local system and center of gravity are not used.
BODY.RIGID	This element contains the information necessary to define a unique rigid body: mass, inertia matrix and location of center of gravity. body local system = ORIENT_INERTIA. If you 'use' the referenced system, no system of a JOINT or BODY should be selected. center of gravity = CENTRE_OF_GRAVITY
JOINT	
SURFACE.CYLINDER	Hyper-elliptical cylinder.

Ellipsoids

No solver support is currently supported for ellipsoids.

Multibody Planes

MADYMO

Card	Description
SURFACE.PLANE	Rectangular plane. multibody = BODY N1 = POINT_1

Card	Description
	N2 = POINT_2 N3 = POINT_3 To create a SURFACE under the SYSTEM.REF_SPACE, a reference to a null body must be selected because a reference to a multibody is required when creating a multibody plane. A null body can be created like any other BODY (card image is not relevant and should not be used), Nullbody should be put under the SYSTEM.REF_SPACE assembly.

Multibody Joints

MADYMO

Card	Description
JOINT.BRAC	Card image = BRAC
JOINT.CYLI	Card image = CYLI D1 and R1 can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT.
JOINT.FREE	Card image = FREE D1 through D3, R1 through R3 and ORIENT can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT. When importing a model containing non-zero values for Q1 through Q7, these values are translated into the values for displacement and rotation; since no values for Q1 through Q7 can be set, no values will be exported.
JOINT.FREE_BRYANT	Card image = FREE D1 through D3, R1 through R3 and ORIENT can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT. When importing a model containing non-zero values for Q1 through Q7, these values are translated into the values for displacement and rotation; since no values for Q1 through Q7 can be set, no values will be exported.
JOINT.FREE_EULER	Card image = FREE D1 through D3, R1 through R3 and ORIENT can not be changed, because they are defined by the position and orientation of the systems connected

Card	Description
	by the JOINT. When importing a model containing non-zero values for Q1 through Q7, these values are translated into the values for displacement and rotation; since no values for Q1 through Q7 can be set, no values will be exported.
JOINT.FREE_ROT_DISP	Card image = FREE_BRYANT D1 through D3, R1 through R3 and ORIENT can not be changed, because they are defined by the position and orietation of the systems connected by the JOINT. Any JOINT.FREE_ROT_DISP is translated into a JOINT.FREE during import of the model. When importing a model containing non-zero values for Q1 through Q7, these values are translated into the values for displacement and rotation; since no values for Q1 through Q7 can be set, no values will be exported.
JOINT.PLAN	Card image = PLAN R1, D2 and D3 can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT.
JOINT.REVO	Card image = REVO R1 can not be changed, because it is defined by the position and orientation of the systems connected by the JOINT.
JOINT.REVO_TRAN	Card image = REVO_TRAN D1 and R2 can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT.
JOINT.SPHE	Card image = SPHE R1 through R3 and ORIENT can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT. When importing a model containing non-zero values for Q1 through Q4, these values are translated into the values for rotation; since no values for Q1 through Q4 can be set, no values will be exported.
JOINT.SPHE_BRYANT	Card image = SPHE_BRYANT R1 through R3 can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT.
JOINT.SPHE_EULER	Card image = SPHE_EULER R1 through R3 can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT.

Card	Description
JOINT.TRAN	Card image = TRAN D1 can not be changed, because it is defined by the position and orientation of the systems connected by the JOINT.
JOINT.TRAN_REVO	Card image = TRAN_REVO D1 and R2 can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT.
JOINT.TRAN_UNIV	Card image = TRAN_UNIV D1, R2 and R3 can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT.
JOINT.UNIV	Card image = UNIV R1 and R2 can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT.
JOINT.UNIV_TRAN	Card image = UNIV_TRAN D1, R2 and R3 can not be changed, because they are defined by the position and orientation of the systems connected by the JOINT.
JOINT.USER	Card image = USER

Named Entities

Entities which are given a name but are not collected or organized into containers.

Examples of named entities include materials and properties.

Some named entities also have a display state, on or off, which control the display of that entity in the modeling window.

All named entities have active and export states. The active state of a named entity controls the display state of the named entity and the listing of the named entity in the Model Browser.

The export state of a named entity controls the export of that entity to a solver deck. The active and export states of named entities can be controlled in the Entity State Browser.

Accelerometers

Accelerometer entities measure acceleration in the simulation.

In the LS-DYNA interface, Accelerometer entities are defined by three nodes. The first node defines the base point, and the other two nodes define a local coordinate system in which accelerations are measured. There are additional fields that enable you to adding a mass or subtract the acceleration due to gravity from the computed acceleration output.

In the Radioss interface, Accelerometer entities are defined by a node and a local coordinate system of type SKEW. This coordinate system is not mandatory. If it is not defined, accelerations are measured in the global coordinate system, otherwise, in the defined local coordinate system

LS-DYNA Cards

Card	Description
*ELEMENT_SEATBELT_AC	Defines a seat belt accelerometer.

Radioss Cards

Card	Description
/ACCEL	Defines an accelerometer.

Ale Fsi Projection

Ale Fsi Projection entities provide a coupling method for simulating the interaction between a Lagrangian material set (structure) and ALE material set (fluid).

The nearest ALE nodes are projected onto the Lagrangian structure surface at each time step. This method does not conserve energy, as mass and momentum are transferred via constrained based approach.

LS-DYNA Cards

Card	Description
*ALE_FSI_PROJECTION	Provides a coupling between Lagrangian material set (structure) and ALE material set (fluid).

Ale Reference System Curve

Ale Reference System Curve entities defines a motion and/or a deformation prescribed for a geometric entity, where a geometric entity may be any part, part set, node set, or segment set.

The motion or deformation is completely defined by the 12 parameters & these are defined in terms of 12 load curves. This command is required only when PRTYPE = 3 in the *ALE_REFERENCE_SYSTEM_GROUP command

LS-DYNA Cards

Card	Description
*ALE_REFERENCE_SYSTE	Motion and/or a deformation prescribed for a geometric entity, where a

(or mesh). This command defines the type of reference system or transformation that a geometric entity undergoes. In other words, it prescribes how certain mesh can translate, rotate, expand, contract, or be fixed in space, and so on.

LS-DYNA Cards

Card	Description
*ALE_REFERENCE_SYSTE	Defines the type of reference system or transformation that a geometric entity undergoes.

Ale Reference System Node

Ale Reference System Node entities defines a group of nodes that control the motion of an ALE mesh.

It is used only when PRTYPE = 5 or 7 in a corresponding *ALE_REFERENCE_SYSTEM_GROUP card.

LS-DYNA Cards

Card	Description
*ALE_REFERENCE_SYSTE	Defines a group of nodes that control the motion of an ALE mesh.

Ale Reference System Switch

Ale Reference System Switch entities allows for the time-dependent switches between different types of reference systems, that is, switching to multiple PRTYPEs at different times during the simulation.

This command is required only when PRTYPE = 6 in ARSG card.

LS-DYNA Cards

Card	Description
*ALE_REFERENCE_SYSTE	Allows for the time-dependent switches between different types of reference systems.

Ale Smoothing

Ale Smoothing entities constraint keeps an ALE slave node at its initial parametric location along a line between two other ALE nodes.

If these nodes are not ALE nodes, the slave node has to follow their motion. This constraint is active during each mesh smoothing operation. This keyword can be used with ALE solids, AL beams.

LS-DYNA Cards

experimentally measured but the ALE inflator needs one additional state variable - the inlet gas velocity which is impractical to obtain.

LS-DYNA Cards

Card	Description
*ALE_TANK_TEST	Provides curve through an engineering approximation for gas inflator’s experimentally measured two curves with additional state variable needed for ALE inflator.

Blocks

Block entities are enclosed volumes represented by a "box" or block.

Nodes, elements, points, lines, surfaces, solids, loads, equations, systems, vectors, and connectors can be reviewed and saved within a block.

LS-DYNA Cards

Card	Description
*DEFINE_BOX	Defines a box to select entities in the model.
*DEFINE_BOX_COARSEN	Defines a box to select elements in the model that are protected from mesh coarsening.
*DEFINE_BOX_ADAPTIVE	Defines a box to select elements in the model that will be adaptive meshed during solver run.
*DEFINE_BOX_SPH	Defines a box to select SPH elements for use to define motion on the SPH particles.

Card	Description
*DEFINE_BOX_DRAWBEA	Define a box around the drawbead to define elements for contact with drawbead.

Bodies

Body entities define a kinematic assembly made of FE parts or nodes.

Bodies are created and organized in the Mechanism Browser.

Boxes

Box entities are enclosed volumes, which are represented by a box.

Nodes, elements, lines, and surfaces can be reviewed and saved within a box.

Radioss Cards

Card	Description
/BOX/CYLIN	Describes a cylinder box for entities selection.
/BOX/SPHER	Describes a spherical box for entities selection.
/BOX/RECTA	Describes a rectangle box for entities selection. **Note:** Block Format Keyword

Constrained Extra Nodes

Constrained extra node entities define and store the keywords *CONSTRAINED_EXTRA_NODES_NODE and *CONSTRAINED_EXTRA _NODES_SET.

LS-DYNA Cards

Card	Description
*CONSTRAINED_EXTRA_I	Define extra nodes for rigid body.

Card	Description
*CONSTRAINED_EXTRA_I	Define extra nodes for rigid body.

Constraints

Constraint entities define kinematical constraints on a body at a specified node or point location.

Constraints are created and organized in the Mechanism Browser.

Contact Surfaces

Contact surface entities define and store contact definitions typically used in contact analysis.

Contact surfaces are defined using elements (1D/2D/3D) and their respective facecodes. A contact surface is displayed as an arrow on the selected element faces in the modeling window. The direction of the arrow is along the element normal that defines the contact surface.

EXODUS Cards

Card	Description
SideSets	Provides a second means of applying loads and boundary conditions to the model.

LS-DYNA Cards

Card	Description
*SET_SEGMENT	Definition segements on element faces.

Nastran Cards

Card	Description
ELIST	Defines a list of CQUAD4 and CTRIA3 structural elements for virtual fluid mass.

OptiStruct Cards

Master Slave contact is represented using contact surfaces.

Card	Description
ELIST	Specifies damp shell elements for a fluid volume. `ELIST` entries are referenced by the `MFLUID` entry.
SURF	Defines a face of a 2D or 3D element as part of a surface.

Permas Cards

Card	Description
$SURFACE	Surface definition. **Note:** The name of the contact surface is getting exported to the *`SFSET`* parameter. On import, the contact surface will be named after the *`SFSET`* parameter. Known limitations: • The reader does not combine two individual $SURFACE cards into a surface set if the carry the same name. • Several surfaces can be combined by an $SFSET card instead.

Radioss Cards

Card	Description
/LINE/SEG	Definition of a line. **Note:** Block Format Keyword
/SURF/SEG	Defines a face of a 2D or 3D element as part of a surface.

Card	Description
	Note: Block Format Keyword

Samcef Cards

Card	Description
.SEL FACE	Defines a set of faces of 3D elements

Control Volumes

Control volume entities define and store control volumes typically used in safety analysis.

Control volumes do not have a display state.

LS-DYNA Cards

Use the Control Volume panel to control volume objects within a model. The *AIRBAG_OPTION card is output from this panel. The POP option is supported for WANG_NEFSKE options.

Card	Description
*AIRBAG_ADIABATIC_GA	Define an airbag or control volume
*AIRBAG_ADVANCED_AL	*DEFINE_ALEBAG_BAG, *DEFINE_ALEBAG_HOLE, and *DEFINE_ALEBAG_INFLATOR to define ALE type AIRBAG in modular way.
*AIRBAG_ALE	Provides a simplified approach to defining the deployment of the airbag using the ALE capabilities with an option to switch from the initial ALE method to control volume method at a chosen time.
*AIRBAG_HYBRID_CHEMI	Define an airbag or control volume
*AIRBAG_HYBRID_ID	Define an airbag or control volume
*AIRBAG_HYBRID_JETTIN	
*AIRBAG_HYBRID_JETTIN	
*AIRBAG_INTERACTION_	Define two connected airbags which vent into each other
*AIRBAG_LINEAR_FLUID_	

Card	Description
*AIRBAG_LOAD_CURVE_	
*AIRBAG_PARTICLE	To define an airbag using the particle method.
*AIRBAG_REFERENCE_GE	If the reference configuration of the airbag is taken as the folded configuration, the geometrical accuracy of the deployed bag will be affected by both the stretching and the compression of elements during the folding process.
*AIRBAG_REFERENCE_GE	
*AIRBAG_REFERENCE_GE	
*AIRBAG_SHELL_REFERE	
*AIRBAG_SIMPLE_AIRBA	
*AIRBAG_SIMPLE_PRESS	
*AIRBAG_WANG_NEFSKE	
*AIRBAG_WANG_NEFSKE	
*AIRBAG_WANG_NEFSKE	
*AIRBAG_WANG_NEFSKE	
*AIRBAG_WANG_NEFSKE	
*AIRBAG_WANG_NEFSKE	
*AIRBAG_WANG_NEFSKE	
*AIRBAG_WANG_NEFSKE	
*AIRBAG_WANG_NEFSKE	
*AIRBAG_WANG_NEFSKE	
*DEFINE_ALEBAG_BAG	Defines the surface that constitutes the airbag's outer surface for ALE airbag.
*DEFINE_ALEBAG_HOLE	Defines the surface that constitutes the vents for ALE airbag.
*DEFINE_ALEBAG_INFLAT	Defines the inflator for ALE airbag.
*DEFINE_CPM_BAG_INTE	Allows interaction between two particle bags.
*DEFINE_CPM_CHAMBER	Defines airbag chambers for air particle initialization or chamber interaction.

Card	Description
*DEFINE_CPM_GAS_PROI	Defines extended gas thermodynamic properties.
*DEFINE_CPM_VENT	Defines extended vent hole options.
*INITIAL_FOAM_REFEREI	The reference configuration allows stresses to be initialized in the following hyperelastic material models: 2, 7, 21, 23, 27, 31, 38, 57, 73, 83, 132 and 181.

PAM-CRASH Cards

Card	Description
BAGIN	Airbag definition
CHAMBER	Multiple chamber definitions
END_BAGIN	Terminates the whole general airbag definition
END_CHAMBER	Ends each chamber description
EXT_SKIN	Chamber outer skin elements **Note:** Sub-keyword of `CHAMBER`
FPM	FPM definition card **Note:** Sub-keyword of `BAGIN`
FPM_HOLE	Hole for SPHCEL **Note:** Sub-keyword of `CHAMBER`
GAS	Chamber GAS definition **Note:** Sub-keyword of BAGIN Sub-keyword of `CHAMBER`
GEN_INI_COND	Chamber gas initial condition **Note:** Sub-keyword of `BAGIN`

Card	Description
INFLATOR	Airbag inflator definition **Note:** Sub-keyword of `CHAMBER`
INI_COND	Initial condition of gas **Note:** Sub-keyword of `CHAMBER`
JET	Jet definition **Note:** Sub-keyword of `CHAMBER`
LEAKAGE	Vents and holes definition in airbag **Note:** Sub-keyword of `CHAMBER`
LOCAL_H	FPM - Local smoothing length card **Note:** Sub-keyword of `BAGIN`
WALL_FABRIC	Define chamber wall fabric **Note:** Sub-keyword of `CHAMBER`
WALL_OPENING	Chamber wall opening **Note:** Sub-keyword of `CHAMBER`

Radioss Cards

Card	Description
/MONVOL	Describes the monitored volume types. **Note:** Block Format Keyword

Card	Description
/MONVOL/AIRBAG1	Describes one-chambered airbag with hybrid input of injected gas. This keyword is similar to `/MONVOL/AIRBAG` (Obsolete), but has more flexible input. **Note:** Block Format Keyword
/MONVOL/AREA	Describes the monitored volume type AREA. **Note:** Block Format Keyword
/MONVOL/COMMU1	Describes multi-chambered airbag with hybrid input of injected gas. This keyword is similar to obsolete keyword `/MONVOL/COMMU`, but has more flexible input. **Note:** Block Format Keyword
/MONVOL/FVMBAG1	Describes Finite Volume Method Airbag, which has more flexible input than the similar obsolete keyword `/MONVOL/FVMBAG`. **Note:** Block Format Keyword
/MONVOL/GAS	Describes the perfect gas monitored volume type. **Note:** Block Format Keyword
/MONVOL/PRES	Describes the pressure load curve monitored volume type. **Note:** Block Format Keyword
/REFSTA	Describes the reference state for elements belonging to a part, using given nodal coordinates.

Card	Description
	Note: Block Format Keyword You need to create a new include file, set the include file type as REFSTA and create this card in the include file.

Cross Sections

Cross section entities store cross section definitions used in a crash analysis.

A cross section is defined by a set of nodes and elements. These sets can be explicitly defined in the entity or computed by intersecting a given geometry (plane, circle) with a set of elements or components.

Supported Solver Cards

Solver cards supported for cross sections.

LS-DYNA Cards

Card	Description
*DATABASE_CROSS_SEC	XsectionPlane
*DATABASE_CROSS_SEC	Define a cross section for resultant forces written to an `ASCII` file SECFORC.

PAM-CRASH 2G

No graphics are display for SECFO cards.

Card	Description
SECFO / NTYP = PLANE	Transmission force at cutting plane Graphics available.
SECFO / NTYP = CONTACT	Cumulated contact forces No graphics available.
SECFO / NTYP = LINK	Cumulated link forces No graphics available.

Card	Description
SECFO / NTYP = SECTION	Transmission force at section No graphics available.
SECFO / NTYP = SUPPORT	Support reaction force No graphics available.
SECFO / NTYP = VOLFRAC	Fraction of volume affected by a defined criterion No graphics available.
SECFO / NTYP = CONT_MS	Contact forces between Master and Slave nodal groups No graphics available.
SECFO / NTYP = DETECT	Accumulated nodal mass in selected volume No graphics available.

Radioss Cards

Card	Description
/SECT	A section is a set of nodes and a set of elements. **Note:** Block Format Keyword
/SECT/CIRCLE	A section is a set of nodes and a set of elements. Sets are built automatically, by intersecting the concerned groups of elements with a disc. **Note:** Block Format Keyword
/SECT/PARAL	A section is a set of nodes and a set of elements. Sets are built automatically, by intersecting the concerned groups of elements with a parallelogram.

Card	Description
	Note: Block Format Keyword

Create Cross Sections

Create cross sections using the Cross Section Assistant.

Restriction: Available in the Radioss, LS-DYNA, PAM-CRASH 2G user profiles.

1. In the Model Browser, right-click and select **Create** > **Cross Section** from the context menu. The **Cross Section Assistant** dialog opens.
2. In the **Cross Section Name** field, enter a name for the cross section.

   Note: If **Create Multiple** is enabled, the cross section name defines the prefix of each cross section created.

3. In the **Cross Section Type** field, select the type of cross section to create.
4. In the **Entity Selection** field, use the entity selector to select the mesh that will be cut by the cross section.
5. To create multiple cross sections simultaneously, select the **Create Multiple** checkbox.

   Note: When this checkbox is enabled, a cross section path definition will replace the Base Node and Normal Definition parameters.

6. If you are creating multiple cross sections, in the **Cross Section Path** field, use the entity selector to select a node path for the creation of each cross section.
7. If you are creating a single cross section:
   a) In the **Base Node** field, define the location of the cross section.

      Manually enter the coordinates of the cross section, or click select the base node in the graphics area.

   b) In the **Normal Direction** field, define the normal of the cutting plane to be used for the generation of the cross section.

      Manually enter the coordinates of the vector, or click to pick two or three nodes.

8. Click **Create**.

HyperMesh automatically fits the cross section to the selected mesh. If you are using the Radioss user profile, the cross section time history Output Blocks is automatically created.

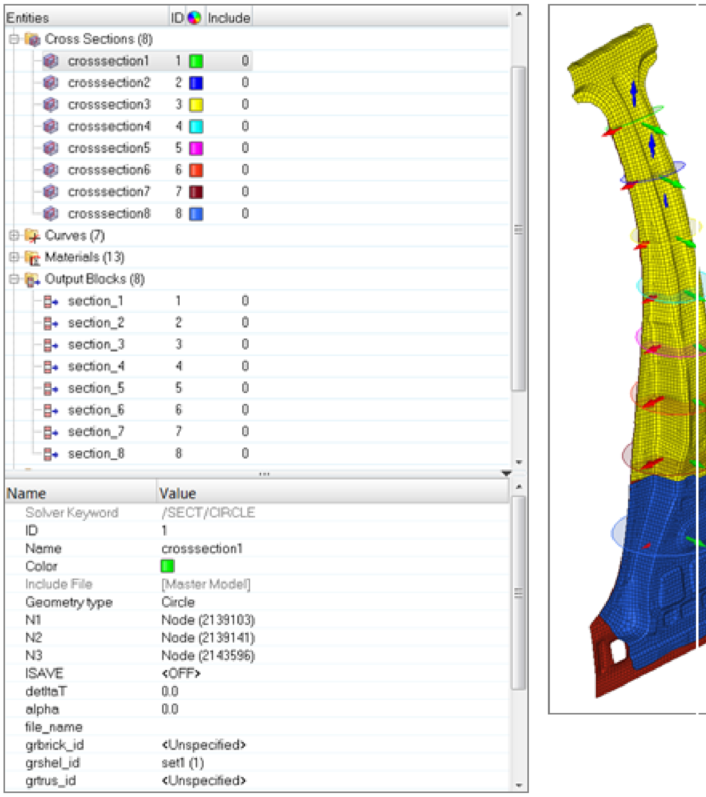


Figure 79: Radioss Cross Section

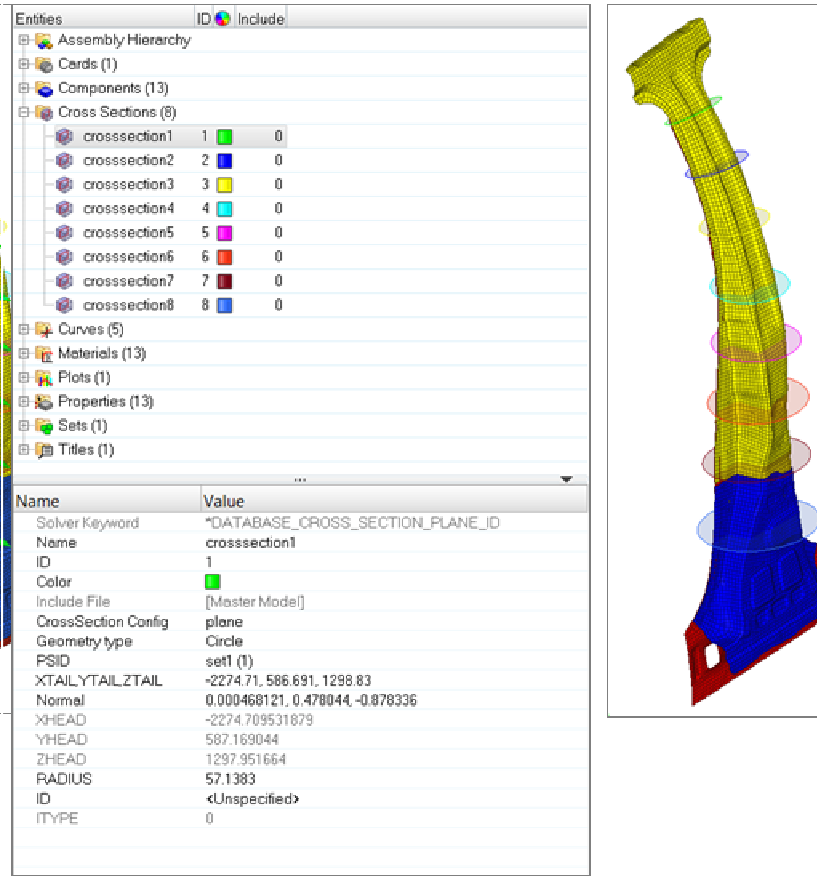


Figure 80: LS-DYNA Cross Section

Curves

Curve entities define and store xy data and are associated with a plot entity.

Curves do not have a display state. However, the display of a curve in a xy plot window is controlled by the display state and active state of its associated plot.

Abaqus Cards

Curves are exported as *AMPLITUDE card in the Abaqus input file.

Card	Description
*AMPLITUDE	Define an amplitude curve

Card	Description
	Note: For Standard and Explicit profiles: The TABULAR definition reads either pairwise entries (new) or four pairs of values per dataline (old). The EQUALLY SPACED option has been updated to the new format as well. The old format is required by users using pre 6.9 Abaqus solver. The reader can read both formats. The card will be exported the same way as it was imported. The card image allows you to switch between both formats.

EXODUS Cards

Card	Description
Function	Time, frequency and/or spatially dependent functions for transient and frequency response analysis.

LS-DYNA Cards

The output of curves creates a *DEFINE_CURVE or Structured Card 22 using Option 0.

*DEFINE_CURVE can be changed to *DEFINE_TABLE in the card previewer. When DEFINE_CURVE is changed to DEFINE_TABLE, the number of curves the table should contain depends on the XY curves that are referenced by a load, material, component, property, and so on are output.

Upon input, the *DEFINE_CURVE and *DEFINE_TABLE/Card 22 cards are read and placed in a plot called LS-DYNA Load Curves. Upon input, references to curves are preserved and are output along with the card, such as material, component, property and load.

Card	Description
*DEFINE_CURVE	Define a curve.
*DEFINE_CURVE_SMOOT	Define a smoothly varying curve using few parameters.
*DEFINE_CURVE_TRIM	Define a curve for trimming.
*DEFINE_CURVE_TRIM_3	Define a curve for trimming. Trimming is processed based on the element normal rather than the vector.
*DEFINE_FUNCTION	Define a function that can be referenced by a limited number of keyword options.

Card	Description
*DEFINE_TABLE	Define a table.
*DEFINE_TABLE_2D	Define a table.
*DEFINE_TABLE_3D	Define a table.

OptiStruct Cards

Multi-body dynamics curves are represented as curves.

Card	Description
MBCRV	Specifies the data used to define a curve. **Note:** Bulk Data Entry

PAM-CRASH Cards

Card	Description
FUNCT/	Definition of curves.

Permas Cards

Card	Description
$FUNCTION	Function definition.

Radioss Cards

Card	Description
/FUNCT	Defines a function (For example: stress (Y-axis) as a function of strain (X-axis)). **Note:** Block Format Keyword
/MOVE_FUNC	Describes the function scale and shift.

Card	Description
	Note: Block Format Keyword

Dummy

Dummy entities are defined by bodies representing the different kinematic assemblies of the dummy.

Dummy entities are the root of the hierarchy in the Dummy Browser. Dummy entities can only be imported into HyperMesh.

Element Clusters

Element clusters are used to describe an assembly of elements for post-processing and failure control.

In Radioss, element clusters are defined with a set of elements (brick or spring) in the related keyword /CLUSTER, inside of which a failure model can also be specified.

Element clusters have a display state, either ON or OFF, which controls the display of an element cluster in the graphics area. To hide or show an element cluster, activate the "Hide Attached" option in the Model Browser configuration window. You can control the display state of an element cluster using the icon next to the element cluster entity in the Model Browser.

Element clusters also have an active and export state. The active state of an element cluster controls the display state and the listing in the Model Browser and any of its views. If an element cluster entity is active, its display state can be turned ON or OFF and is listed in the Model Browser and any of its views. If an element cluster entity is inactive, its display state is turned OFF permanently and is not listed in the Model Browser or any of its views.

The export state of an element cluster entity controls whether that element cluster is exported when the custom export option is used. The all export option is not affected by the export state of an element cluster. The active and export states of an element cluster can be controlled using the Entity State Browser.

Table 7:

Card	Description	Parameters	Notes
/CLUSTER	Define an element cluster		Related keywords: /TH/CLUSTER and /ANIM/VECT/CLUSTER

Failures

Failures describe material failure criteria to be coupled to material models.

In Radioss, Failure models are defined with the keywords /FAIL.

Failure entities have a display state, on or off, which controls the display of these entities in the graphics area. To hide or show a Failure entity, the "Hide Attached" option in the Model Browser configure window, must be activated. The display state of a Failure entity can be controlled using the icon next to the entity in the Model Browser.

Failure entities also have an active and export state. The active state of a Failure entity controls its display state and its listing in the Model Browser along with its views. If a Failure entity is active, then its display state is available to be turned on or off and it is listed in the Model Browser along with its views. If a Failure entity is inactive, then its display state is turned off permanently and it is not listed in the Model Browser or any of its views.

The export state of a Failure entity controls whether or not this entity is exported when the custom export option is used. The all export option is not affected by the export state. The active and export states of Failure entities can be controlled using the Entity State Browser.

Radioss Cards

Card	Solver Description	Application	Notes
/FAIL/BIQUAD	Strain failure model	Ductile metal	Direct input on effective plastic strain to failure
/FAIL/CHANG	Chang-Chang model	Composite	Failure criteria for composites
/FAIL/CONNECT	Failure	Connection spot weld	Normal and Tangential failure model
/FAIL/EMC	Extended Mohr Coulomb failure model	Metal	Failure dependent on effective plastic strain

Card	Solver Description	Application	Notes
/FAIL/ENERGY	Energy isotrop	Metal, plastic	Specific energy
/FAIL/FLD	Forming limit diagram	Metal Forming	Fld
/FAIL/HASHIN	Composite model	Composite	Hashin model
/FAIL/JOHNSON	Ductile failure model	Ductile metal	Johnson-Cook
/FAIL/LAD_DAMA	Composite delamination	Composite	Ladeveze delamination model
/FAIL/NXT	NT failure model	Metal Forming	Similar to FLD, but based on stresses
/FAIL/PUCK	Composite model	Composite	Puck model
/FAIL/SNCONNECT	Failure	Connection, spot weld	Failure criteria for plastic strain
/FAIL/SPALLING	Ductile + Spalling	Ductile metal	Spalling + Johnson-Cook
/FAIL/TAB1	Strain failure model	Ductile metal	Based on damage accumulation using user-defined functions
/FAIL/TBUTCHER	Tuler-Butcher model	Ductile metal	Failure due to fatigue
/FAIL/TENSSTRAIN	Traction	Metal, plastic	Strain failure
/FAIL/USERi	User failure model	n/a	n/a
/FAIL/WIERZBICKI	Ductile material	Ductile metal	Bao-Xue-Wierzbicki model
/FAIL/WILKINS	Ductile failure model	Ductile metal	Wilkins model
/FAIL/XFEM	XFEM (Extended Finite Element Method		

Features

Feature entities track and manage certain geometric features.

Dimension features facilitate the change of dimensions in a model, and store information such as the surface vertices, parametrization, and control options required to edit a dimension.

For a dimension manipulator to be shown in the modeling window, both of its associated vertices must be displayed.

Fields

Field entities store spatially varying values, which can later be realized (mapped) to the element and node data of a new target mesh.

Field mapping can be used to:

- Transfer temperature result data in an analysis results file from a thermal analysis to a structural model (a different mesh to that of the thermal model) as nodal temperature loads. (Continuous field)
- Transfer displacement values in an analysis results file from a coarse structural analysis to a detailed structural model (sub modeling based on enforce displacement from a global model) as nodal enforced displacement. (Continuous field)
- Transfer temperature or pressure values in a `.csv` file (x,y,z, temperature) to a structural model as nodal temperature loads or element pressure loads. (Discrete field)
- Transfer pressure data in an analysis deck from an existing model to a new structural model (different mesh) as element pressures. (Continuous field)
- Transfer two-dimensional parametric data (u,v, data value) in a `.csv` file to a 3D surface model. Data vales can be temperature or pressure. Mapping is done in a parametric system. (Continuous field)
- Transfer property ID, material orientation, and/or load data in an analysis input deck from an existing model to a new structural model (different mesh). (Continuous field)
- Transfer 2D/axis-symmetric model temperate or pressure data to a new 3D model as temperate or pressure loads. (Continuous field)
- Bar/Beam 1D to 2D/3D element mapping.
- 3D to 2D/3D element mapping.
- Mapping types (for discrete/`.csv` data only.)
- Generic field mapping.
- Reviewing and transforming source data.
- Nodal force balancing using OptiStruct.

You can create field entities in the Model Browser, and edit their corresponding attributes in the Entity Editor. All fields are stored in the Field folder within the Model Browser.

Creating Fields

In this topic you will learn how to create fields.

Before creating a field, import the target model or analysis file that contains the new mesh into your current HyperMesh session.

1. In the Model Browser, right-click and select **Create** > **Field** from the context menu.
2. In the Entity Editor, select one of the following field types:

- **Continuous** fields evaluate data based on the shape function of source mesh data and its nodal or elemental values. If the location of the x,y,z points or u,v points are well structured in a rectangular array, a source mesh can be constructed.

  Source data contains the mesh and the nodal or elemental values. The target mesh is different from the source mesh.

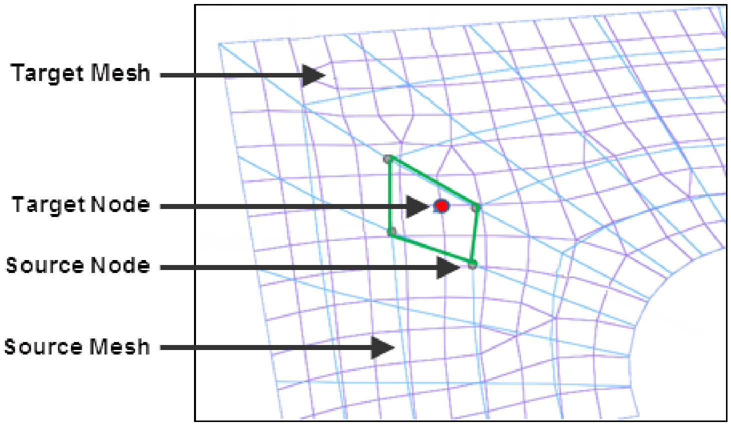


Figure 81:

- **Discrete** fields provide data on discrete x,y,z locations (real) or u,v locations (parametric), without the source mesh. Depending on the fitness of the data points, the closest point approach or inverse distance linear interpolation method will be used to map.

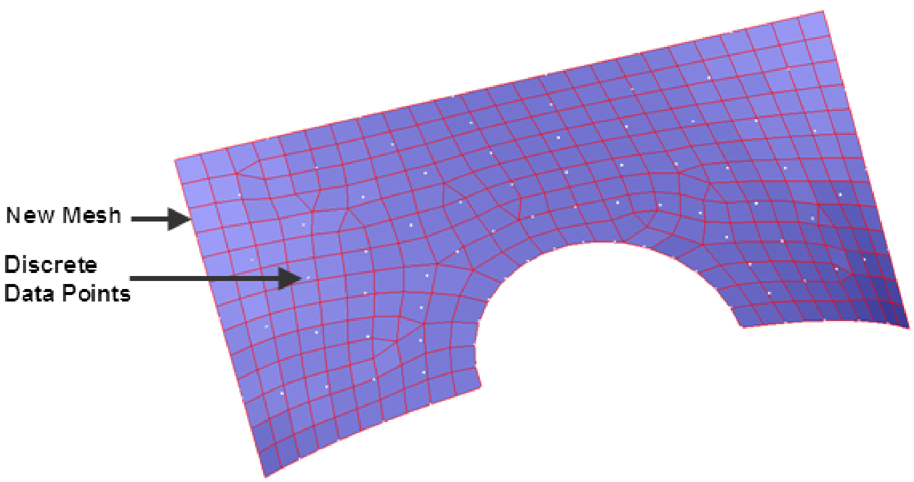


Figure 82:

3. Select one of the following System types:

- **Real** coordinate systems (x,y,z values)

centerx	centery	centerz	PLOAD4_P1
45.4873	-25.8606	4.327155	12350000
45.93106	-25.0764	4.900927	12114000
66.21854	-61.8255	2.234271	12987000
66.46476	-61.9586	2.249586	12047000

Figure 83:

- **Parametric** systems (u,v tables)

0	0	0.2	0.05	0.8	1
0	672.6	642	611.5	580.9	550.3
0.25	644.6	615.3	586	55[illegible]	[illegible]
0.75	476.6	454.9	433.2	41[illegible]	[illegible]
1	476.6	454.9	433.2	41[illegible]	[illegible]

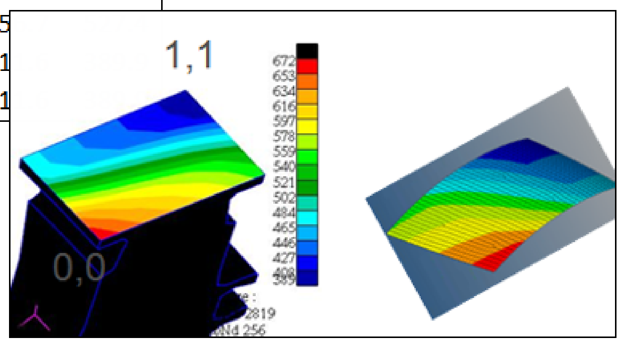


Figure 84:

4. Select one of the following Sources to create the field:
   - **Results:** loads an existing results file.
   - **Current Model:** imports an analysis data file (`.bdf`, `.fem`, `.inp`, and so on). If you select Current Model, you will see the following fields:
     - **Data Source Type:**
     - **Source Entity List:**
   - **CSV file:** loads a `.csv` file containing x,y,z, value1, value2 discrete data.
   - **Matrix:** loads a matrix file which will be used to generate data using matrix query tools.
   - **HV Current Contour:**

Realizing (Mapping) Fields

In this topic you will learn how to realize, or map, fields.

Field realization creates pressures and temperature loads, and maps property IDs. To map the spatially varying values stored in a field entity to the element and node data of the new target mesh, you must realize the field entity.

The following types of mapping are supported: node temperature loads, node displacement, element pressure, element property IDs and indirect thickness, and element material orientation.

1. In the Model Browser, right-click on the field that you would like to realize and select Realize from the menu.
2. In the **Field Realization** dialog, define realization settings:
    a) Using the Entity selector, select the elements or nodes to be realized.
    b) Select a field type.
    c) Select the type of interpolation to perform: shape function (based on mapping), proximity (based on close point evaluation), or linear interpolation (based on inverse distance).
    d) Click **Apply**.

    Once the field has been realized, the new mapped values can be visualized using contour, or they can be exported to solver decks. When a solver deck is exported, mapped loads will be available on the new mesh.

Mapping Examples

The following tasks show examples of various mapping strategies.

Continuous Temperature/Displacement Mapping Example

This task is an example of how to map fields.

Continuous temperature or displacement mapping from a results file to a new model.

1. Import the model containing the new mesh into HyperMesh.

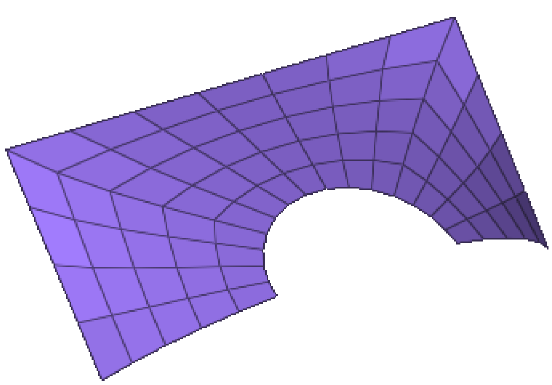


Figure 85:

2. In the Model Browser, right-click and select **Create** > **Field** from the menu.
3. In the Entity Editor, edit the field's corresponding attributes:
    a) Set Type to continuous.
    b) Set System Type to real.
    c) Set Source to results.

d) In the File field, locate the results file with temperatures or displacements. Results files can be of type `.opt`, `.odb`, `.xdb`, or any HyperView results file.

4. In the Model Browser Field folder, right-click on the **field entity** and select **Realize** from the menu.
5. In the **Field Realization** dialog, define the realization settings:

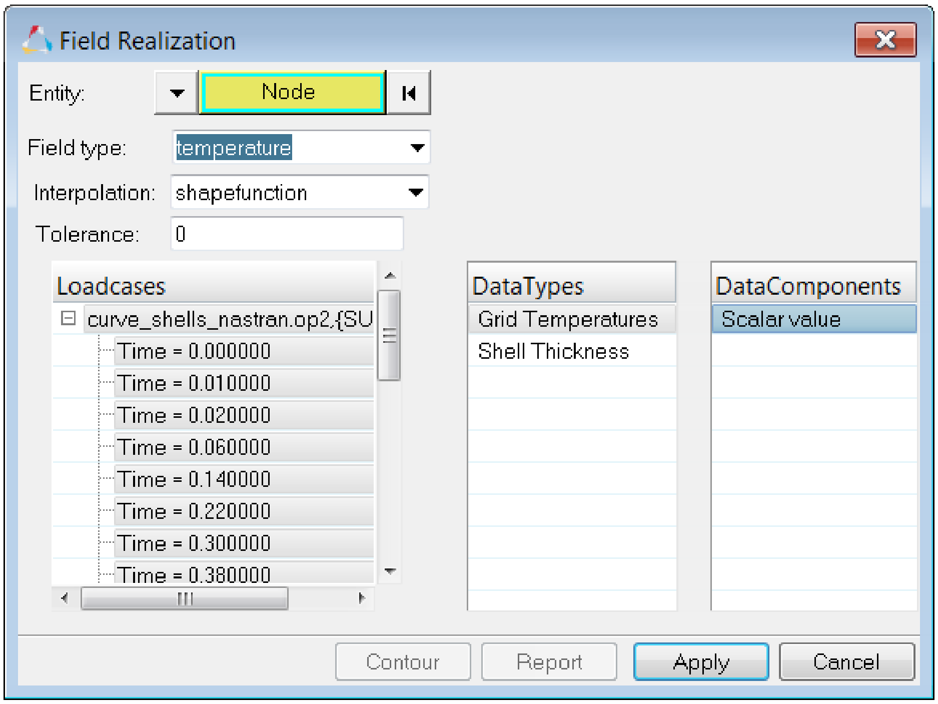


Figure 86:

a) With the Entity selector set to Node, select the target nodes (nodal temperature loads) that you would like to map.

b) Set Field Type to temperature.

c) Set Interpolation to shape function.

d) Under Loadcases, select the loadcases you would like to transfer. Each load case selection will create a new load.

e) Under DataTypes, select the temperature.

f) Under DataComponents, select scalar values or displacement x, y, and z values.

g) Click **Apply**.
Temperatures from the result file are now mapped to the new mesh.

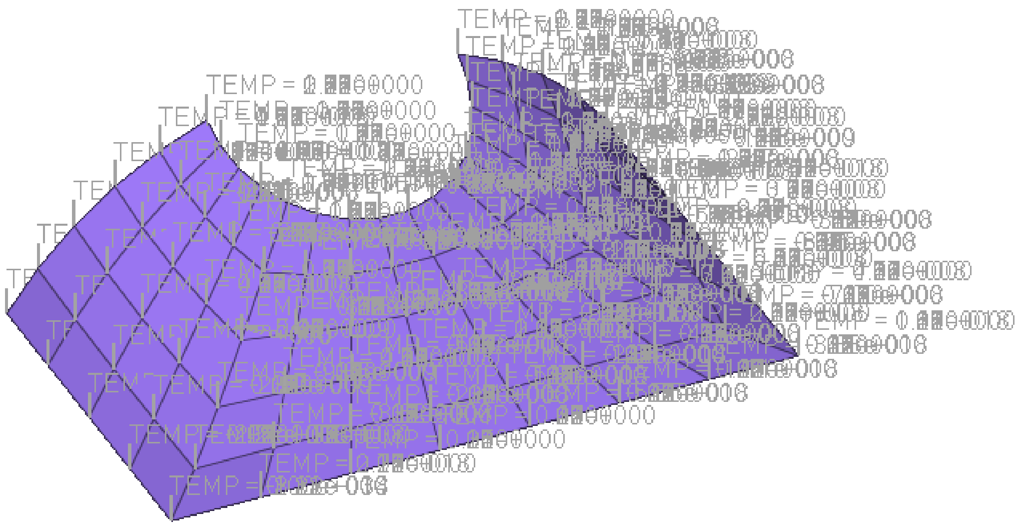


Figure 87:

6. Click **Contour** to view the contour for each load case (select the correct load case).

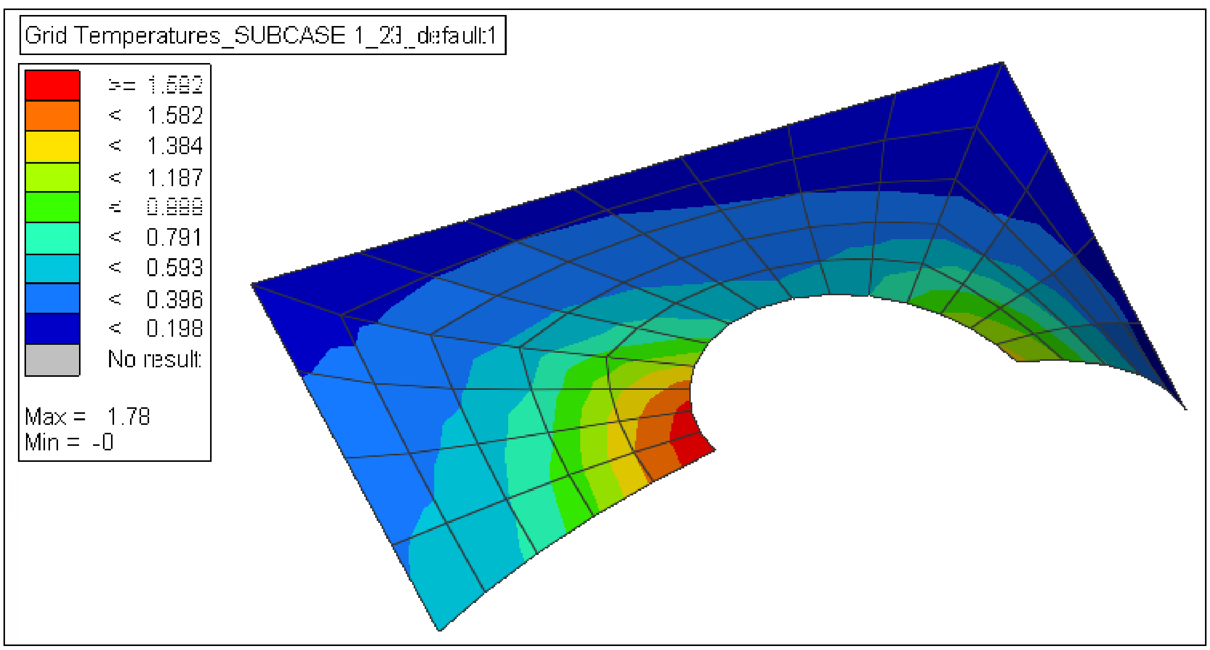


Figure 88:

7. Click **Report** to view the values in table format.

Discrete Temperature Mapping Example

This task is an example of discrete mapping of temperatures or pressures.

In this task you will learn how to use discrete mapping of temperatures or pressures from a `.csv` file.

1. Import the model containing the new mesh in HyperMesh.

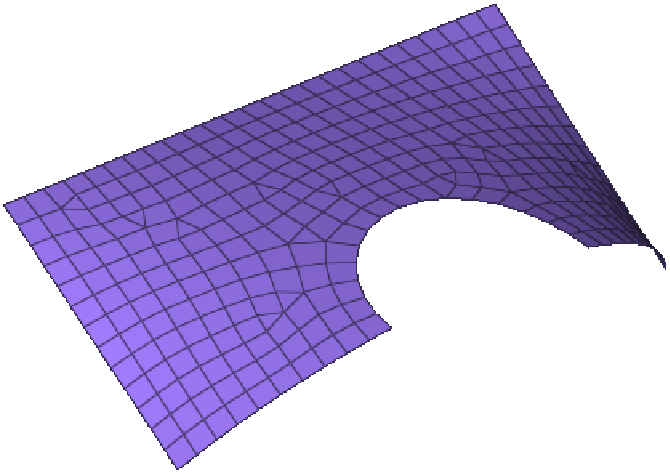


Figure 89:

2. In the Model Browser, right-click and select **Create** > **Field** from the menu.
3. In the Entity Editor, edit the field's corresponding attributes:
   a) Set Type to discrete.
   b) Set System Type to real.
   c) Set Source to `.csv` file.
   d) In the File field, locate the `.csv` results file that contains the temperatures or pressures.
4. In the Model Browser, right-click on the **field entity** and select **Realize** from the menu.
5. In the **Field Realization** dialog, define the realization settings:

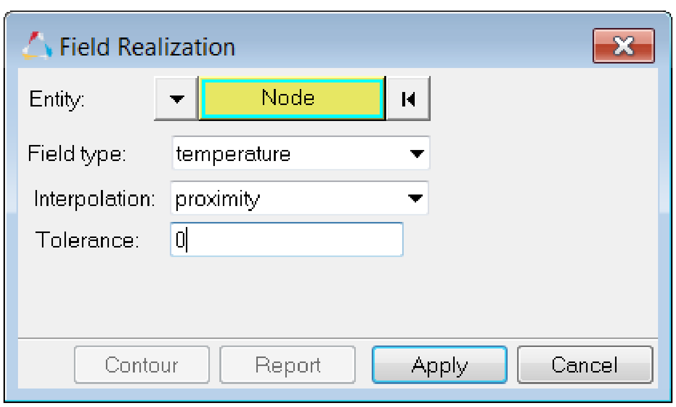


Figure 90:

   a) With the Entity selector set to Node, select the nodes you would like to map.
   b) Set Field Type to temperature.
   c) Set Interpolation to proximity (finds the closest nodes) or linear (finds nodes using the inverse distance average).
   d) In the Tolerance field, type the perpendicular projection tolerance. Nodes outside of the 14.0 limit will not be mapped.
   e) Click **Apply**.

Temperatures from the `.csv` file are now mapped to the new mesh.

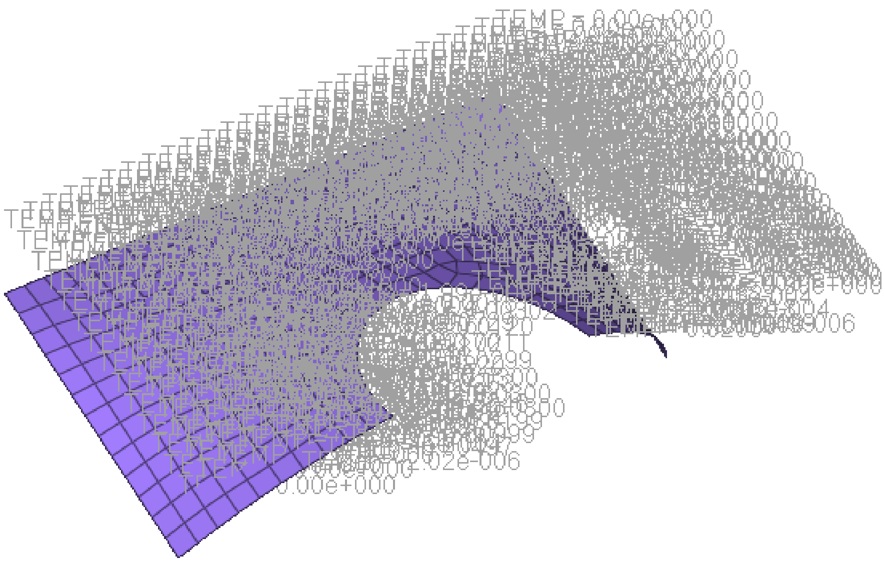


Figure 91:

f) Click **Contour** to see the mapping.

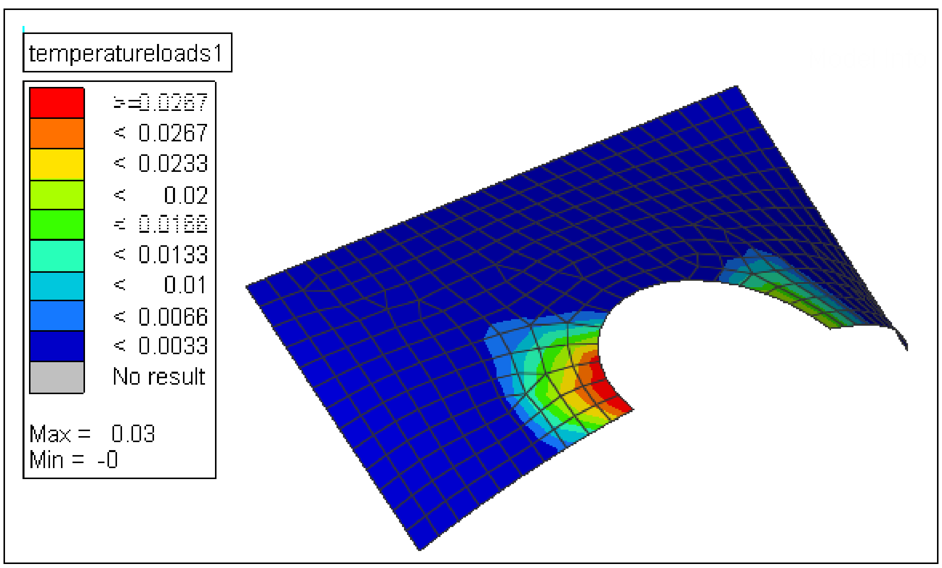


Figure 92:

Discrete Mapping Using a .csv File and System

This task is an example of mapping using data from a `.csv` file.

Data in a `.csv` file can be based on a local coordinate system. The format of the `.csv` file is x, y, z, value1, value2, and so forth. The x, y, z data can be in a global or local system , including cylindrical.

The following mapping methods can also be selected:

- Proximity
- Linear Interpolation
- Inverse Distance
- Triangulation

1. In the Model Browser, right-click and select **Create** > **Field** from the menu.

2. In the Entity Editor, edit the field's corresponding attributes:
   a) Set Type to discrete.
   b) Set System Type to real.
   c) Set Source to `.csv` file.
   d) In the File field, locate the `.csv` file.
   e) In the Systems field, select the system. The system should match the x, y, z data in the `.csv` file.

3. In the Model Browser, right-click on the **field entity** and select **Realize** from the menu.

4. In the **Field Realization** dialog, define the realization settings:

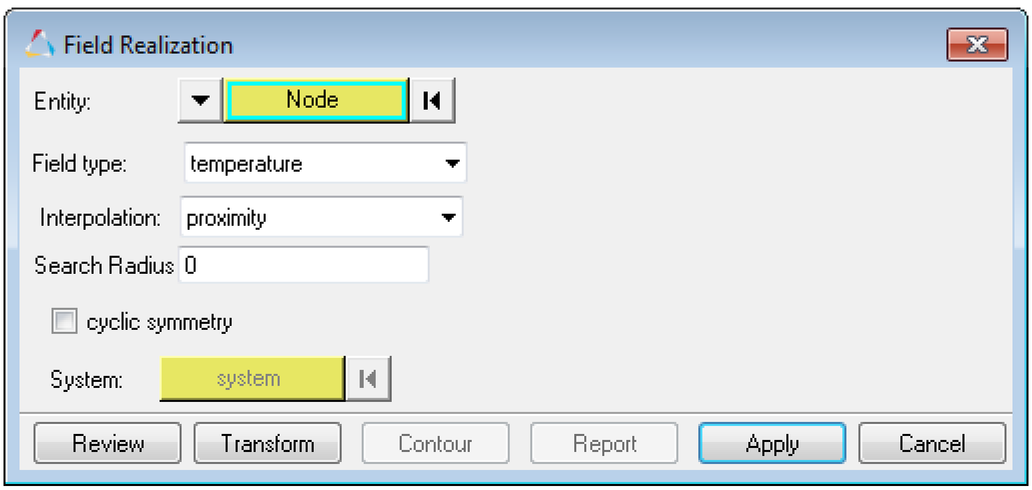


Figure 93:

   a) With the Entity selector, select nodes or elements that you would like to map.
   b) In the Field Type field, select a realization method.
   c) Set Interpolation to proximity (finds the closest nodes) or linear (finds nodes using the inverse distance average).
   d) In the Search Radius field, type a search radius.
   e) Click **Apply**.
   f) Click Contour to see the mapping.

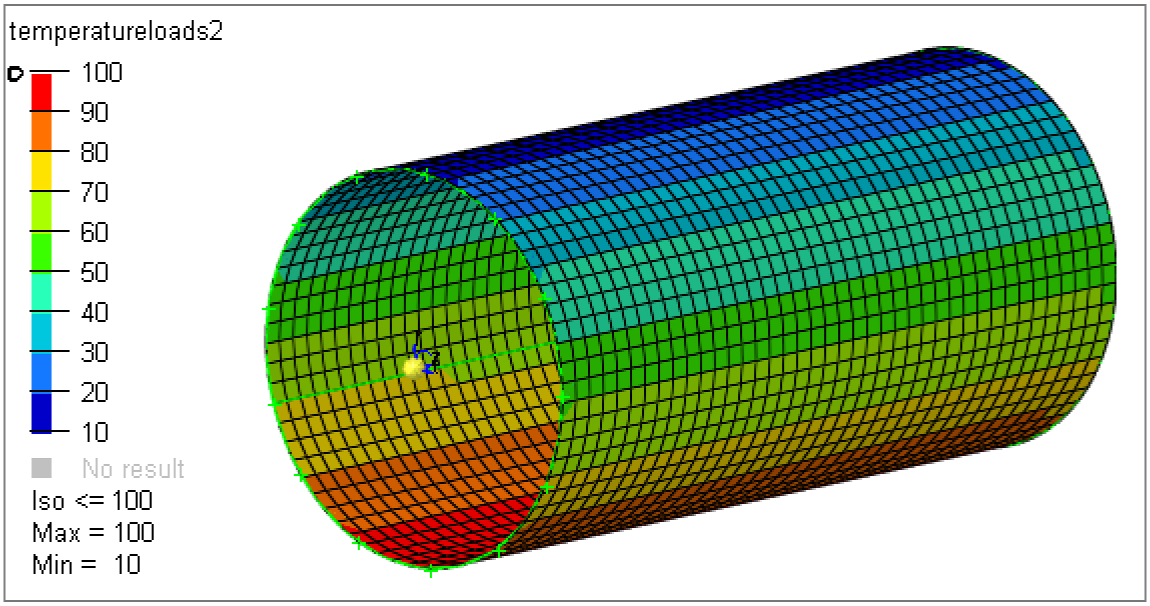


Figure 94:

Transfer Load Mapping Examples

This method is useful when transferring data from existing solver decks (`.bdf`, `.fem`, `.inp`, and so forth) to new meshed models. Old mesh shape functions are used with this method.

The x,y,z values in `.csv` files can be used without old mesh data. Discrete mapping (close point approach) will be used instead, which is less accurate than continuous shape function mapping. The matrix utility can be used to export x,y,z value data from old solver decks to a `.csv` file.

Transfer Load Data from an Existing Model to a New Model

This task is an example of transferring load data to a new model.

In this task you will learn how to transfer data from an existing model to a new model.

1. Import the model or analysis file containing transfer loads into HyperMesh.

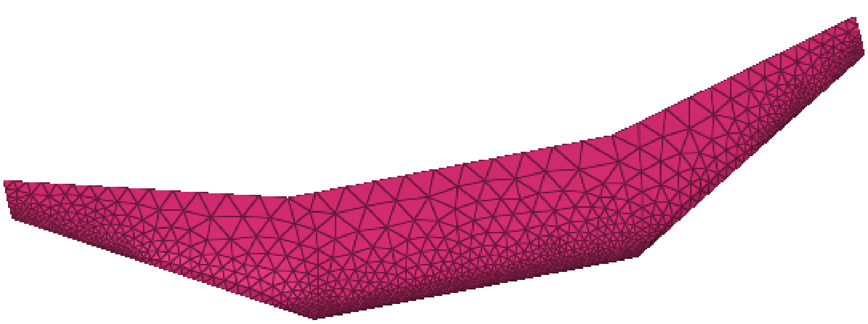


Figure 95:

2. In the Model Browser, right-click and select **Create** > **Field** from the menu.
3. In the Entity Editor, edit the field's corresponding attributes:

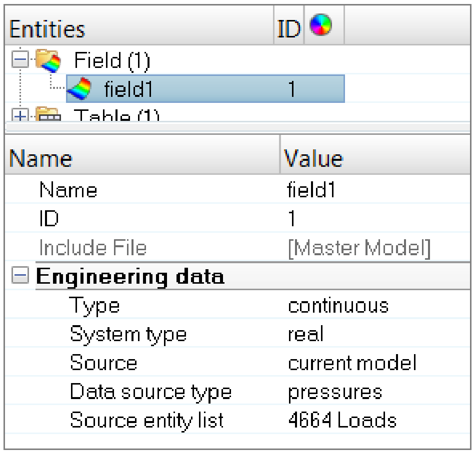


Figure 96:

a) Set **Type** to continuous.
b) Set **System Type** to real.
c) Set **Source** to current model.
d) Set **Data Source** to pressures.
e) In the **Source Entity List** field, use the elements selector to select pressure loads on the old mesh.

After selecting source entities, a Table entity is automatically created and populated with values associated to the nodes/element.

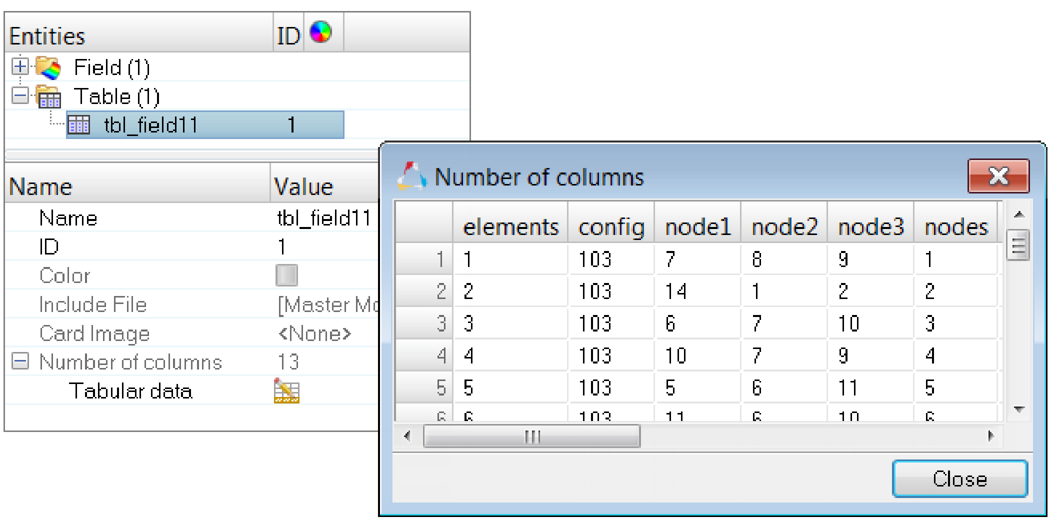


Figure 97:

4. Delete the old mesh and loads. Do not delete the field you just created.
5. Import the new mesh into HyperMesh.

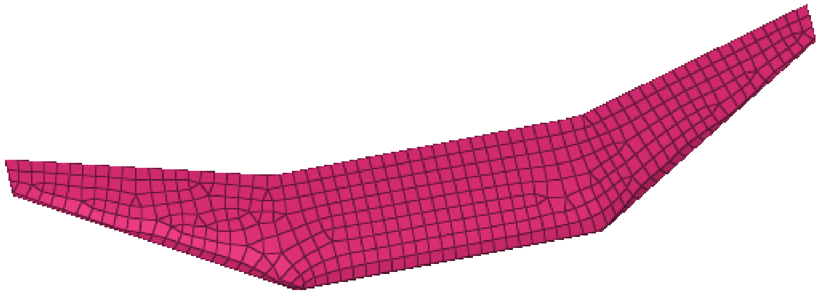


Figure 98:

6. In the Model Browser, right-click on the **field entity** and select **Realize** from the menu.

7. In the **Field Realization** dialog, define the realization settings:

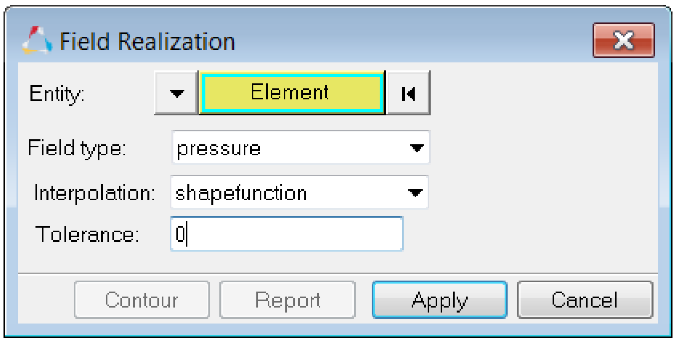


Figure 99:

a) Using the **Entity: Node** selector, select the nodal temperature load target nodes to be mapped.

b) Set **Field type** to pressure.

c) Click **Apply**.
Transfer loads are mapped to the new mesh, thereby transferring the thickness, material angle, or ply information to the new elements.

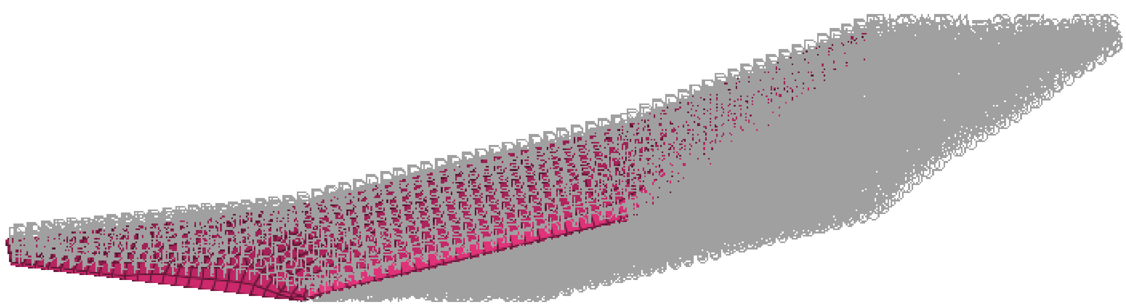


Figure 100:

d) Click **Contour** to view the contour of the pressure loads transferred to the new mesh.

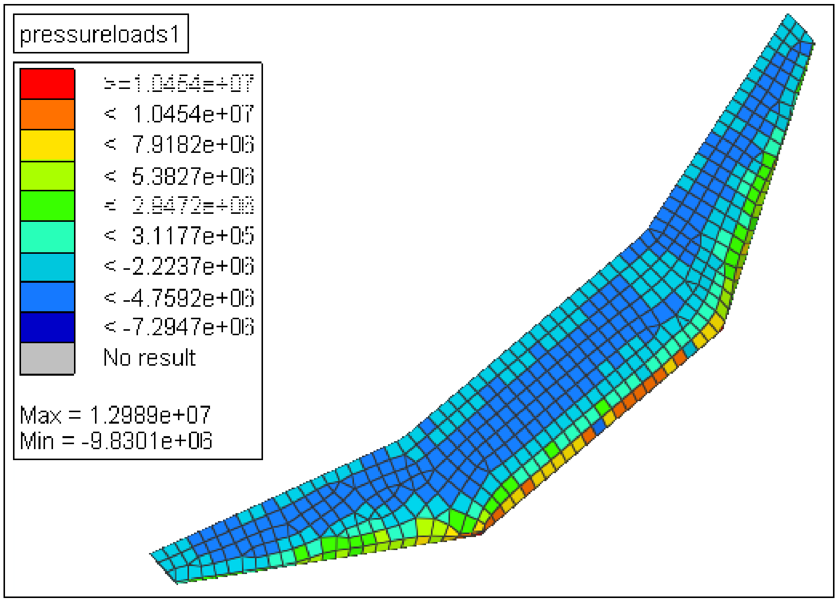


Figure 101:

Transfer Property IDs from an Existing Model to a New Model

This task is an example of transferring property IDs to a new model.

In this task you will learn how to transfer property IDs from an existing model to a new model.

1. Import the model or analysis file containing property IDs into HyperMesh.

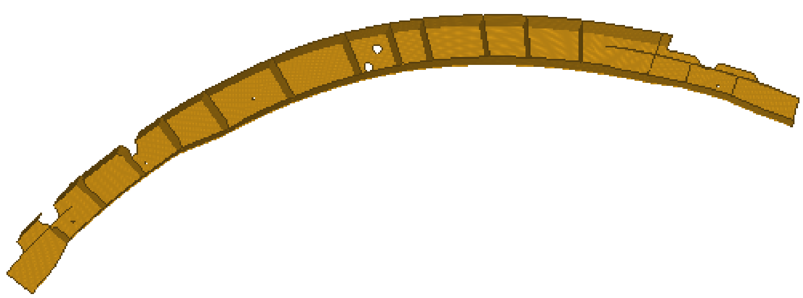


Figure 102:

2. In the Model Browser, right-click and select **Create** > **Field** from the menu.
3. In the Entity Editor, edit the field's corresponding attributes:

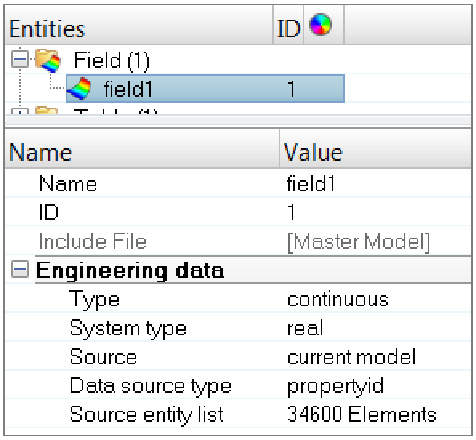


Figure 103:

a) Set **Type** to continuous.

b) Set **System Type** to real.

c) Set **Source** to current model.

d) Set **Data Source Type** to propertyid.

e) In the **Source Entity List** field, use the elements selector to select property IDs from the current model.
After selecting source entities, a Table entity is automatically created and populated with values associated to the nodes/element.

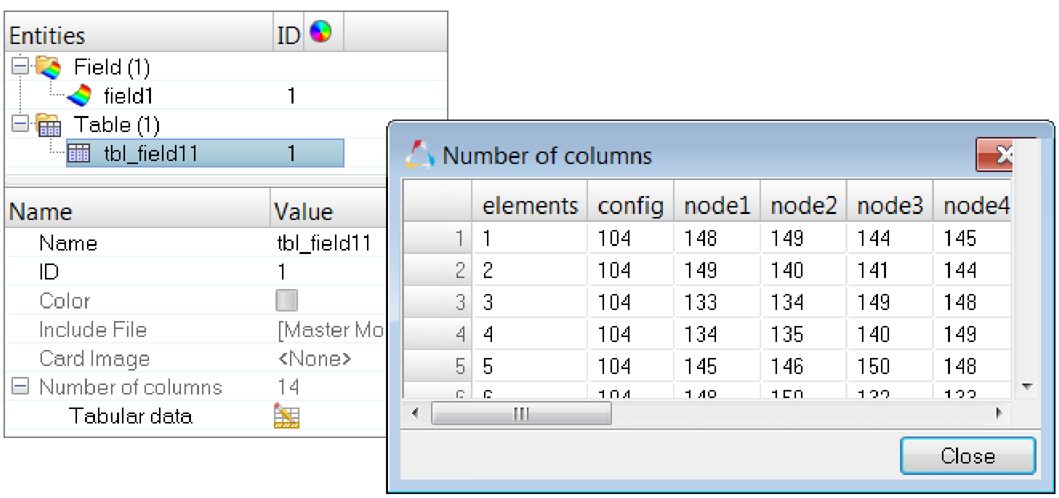


Figure 104:

4. Delete the old mesh. Do not delete the field or properties created.

5. Import the new mesh into HyperMesh, or re-mesh the model.

6. In the Model Browser, right-click on the **field entity** and select **Realize** from the menu.

7. In the **Field Realization** dialog, define the realization settings:

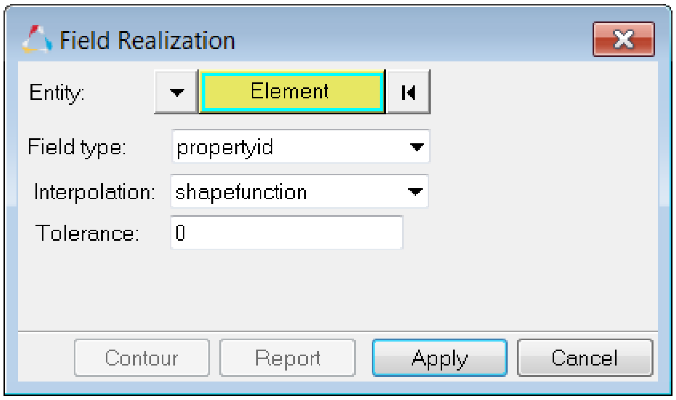


Figure 105:

a)

Transfer 2D Parametric Data in a .csv File to a 3D Surface Model

Use this method to map data from an old geometry model to a new geometry model during CAD parametric updates. This method uses geometry to node associativity.

In this example you will learn that though the u, v values are used as discrete data from a `.csv` file, a 2D mesh using the u, v locations is created internally. The shape function mapping is used in the 2D u, v space.

1. In the Model Browser, right-click and select **Create** > **Field** from the menu.
2. In the Entity Editor, edit the field's corresponding values:

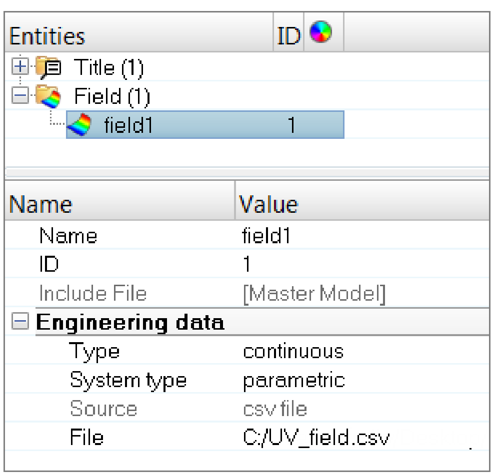


Figure 106:

   a) Set **Type** to continuous.
   b) Set **System Type** to parametric.
   c) Set **Source** to `.csv` file.
   d) In the **File** field, locate the `.csv` file containing the rectangular array of u, v data.
3. Import the new model in HyperMesh.

4. Mesh the surface of the model. The surface of the model should be associated with the target mesh. Every node should be mapped to a u, v.
5. In the Model Browser, right-click on the **field entity** and select **Realize** from the menu.
6. In the **Field Realization** dialog, define the realization settings:

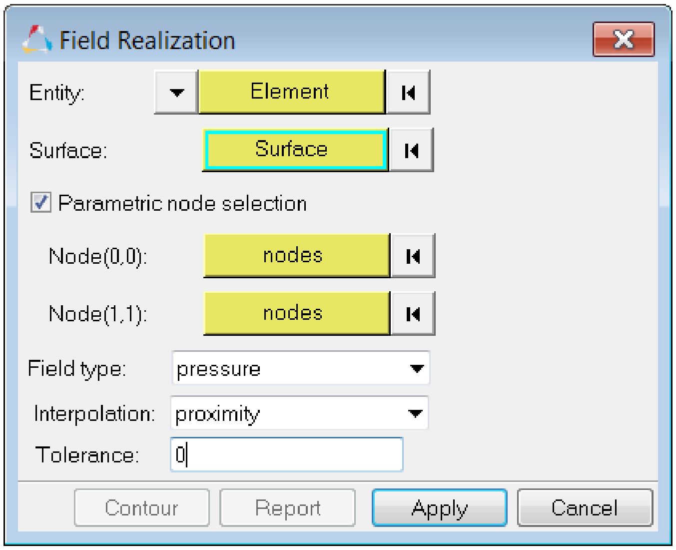


Figure 107:

   a) Using the **Entity: Element** selector, select the elements to map.
   b) Using the **Surface:** selector, select the surface to map.
   c) Using the Node(0,0) and Node(1,1) selectors, select the two nodes that coincide with (0,0) and (1,1) of the u, v surface. You can rotate the position by selecting two nodes which define the corner locations of the u, v.
   d) Click **Apply**.
   The 2D parametric data is mapped to the model.

Transfer Axis Symmetric Model Temperature/Pressure Data to a 3D Model

Use this method to map data from an axis symmetric model to a 3D model.

An axis symmetric model may contain nodal temperatures or pressures in the R, Z plane. You will need an axis symmetric mesh and the data in a `.csv` file. This data can be mapped to a full 3D model.

Limitation: 3D model nodes when rotated to the R, Z plane should fall inside the axis symmetric model.

1. In the Model Browser, right-click and select **Create** > **Field** from the menu.
2. In the Entity Editor, edit the field's corresponding attributes:

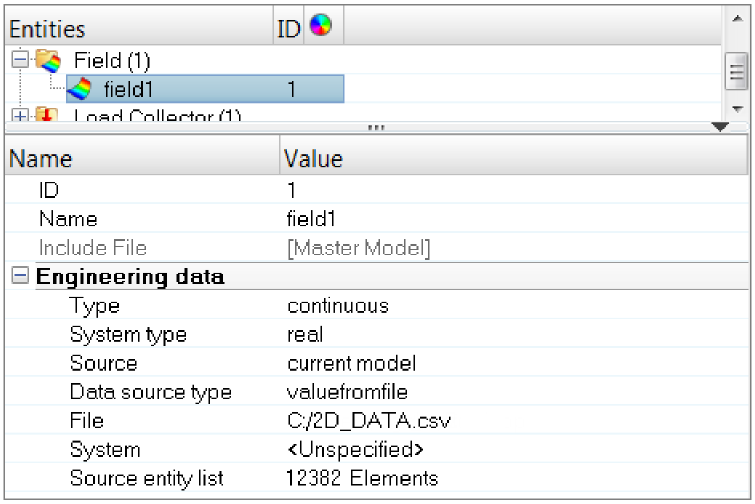


Figure 108:

a) Set **Type** to continuous.
b) Set **System Type** to real.
c) Set **Source** to current model.
d) Set **Data Source Type** to value from file.
e) In the **File** field, open the `.csv` file.
f) In the **System** field, select the system in which the `.csv` file x, y, z data is created (the default is global).
g) In the **Source Entity List** field, select element from the current model.

3. After creating the model, delete the current model (not the field).
4. Import the new model into HyperMesh.
5. Import the new 3D model (target model).
6. In the Model Browser, right-click on the **field entity** you just created and select **Realize** from the menu.
7. In the **Field Realization** dialog, define the realization settings:

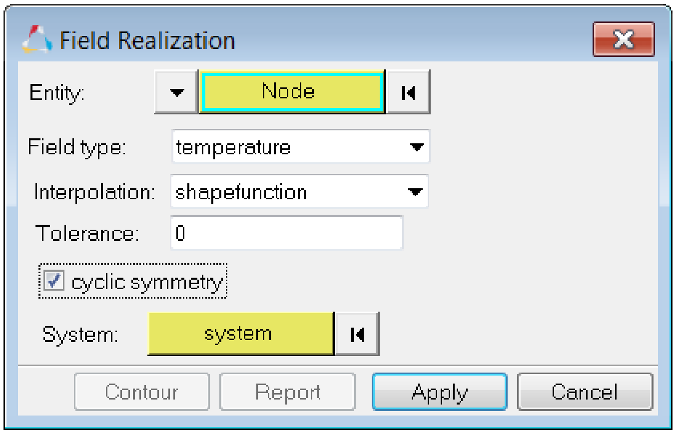


Figure 109:

a) Using the **Entity elements/nodes** selector, select the elements or nodes to map.
b) Set **Field Type** to temperature.
c) Set **Interpolation** to shape function or proximity.
d) Check **Cyclic Symmetry** and then select the cylindrical **System** where the axis symmetry mesh resides.
e) Click **Apply**.

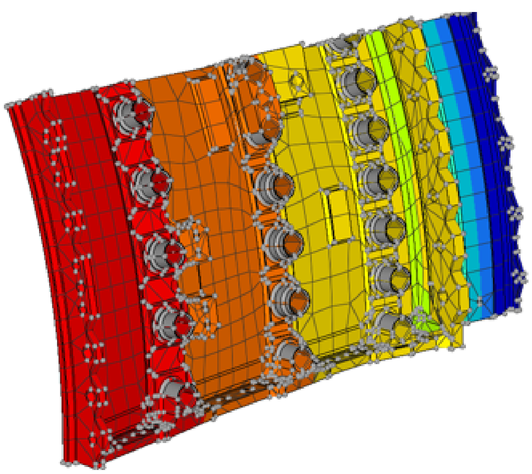


Figure 110:

Bar/Beam 1D to 2D/3D Element Mapping

Use 1D source element data (results and current model with `.csv`) to map to 2D/3D target elements.

In this task you will learn how to map bar/beams to 2D/3D elements.

1. Create or open a model containing 2D/3D target elements.
2. Create and define a field entity:
   a) In the Model Browser, right-click and select **Create** > **field** from the menu.

b) In the Entity Editor, set Source Type to results file, or current model with `.csv` results file-based results of temperature or displacements from 1D source elements.

3. In the Model Browser, right-click on the field and select **Realize** from the menu.

4. In the **Field Realization** dialog, define the realizations settings:

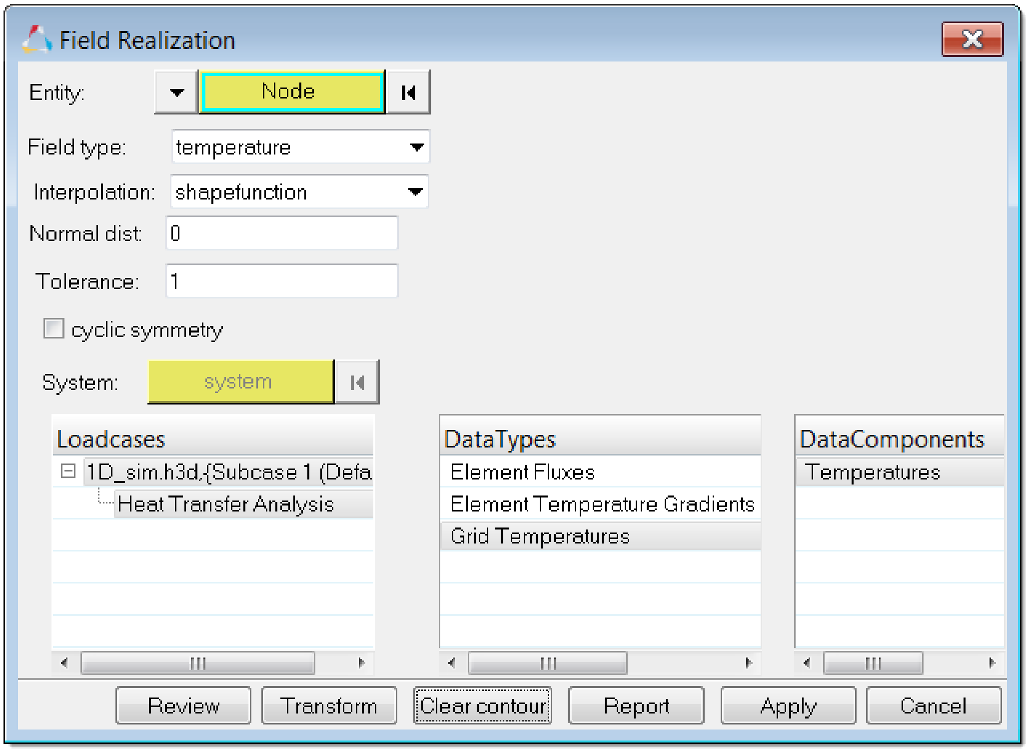


Figure 111:

a) Set **Field Type** to temperature or displacement.

b) Set **Interpolation** to shapefunction.

c) Select the appropriate **Loadcases**, **DataTypes**, and **DataComponents**.

d) Click **Apply**.

5. In the **Field Realization** dialog, click the **Contour** button to contour the results.

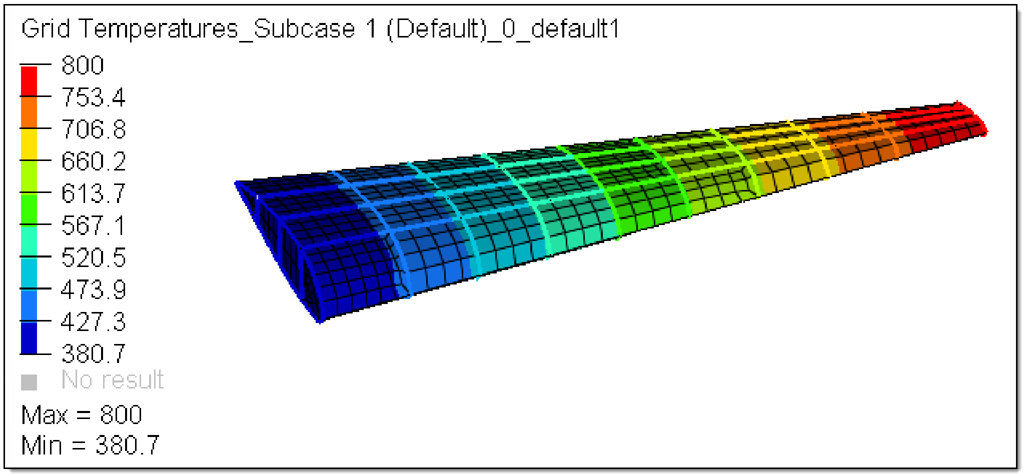


Figure 112:

3D to 2D/3D Element Mapping

3D linear or quadratic source element data can be mapped to 2D/3D target elements. 3D source elements can have loads or results which can be mapped.

In this task you will map 3D elements to other 2D/3D elements.

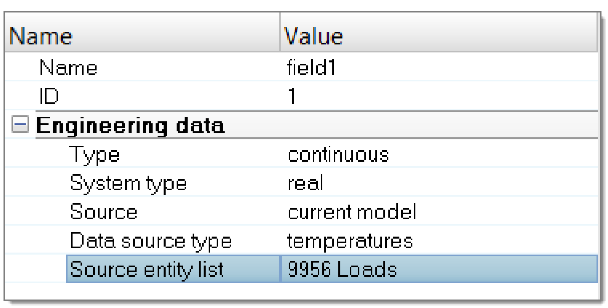
1. Import or open a 3D/2D target model.

   You can import a 3D model from solver files (current model), or results files containing temperatures, pressures, or other loads. When importing, use ID offset in order to preserve the new model's IDs.

2. Create and define a field entity.

   a) In the Model Browser, right-click and select **Create** > **Field** from the menu.

   b) In the Entity Editor, define the current model using loads or results.



Name	Value
Name	field1
ID	1
Engineering data	
Type	continuous
System type	real
Source	current model
Data source type	temperatures
Source entity list	9956 Loads

Figure 113:

3. Delete the source model.

4. Realize the field:

   a) In the Model Browser, right-click and select **Realize** from the menu.

   b) In the **Field Realization** dialog, define the realization settings.

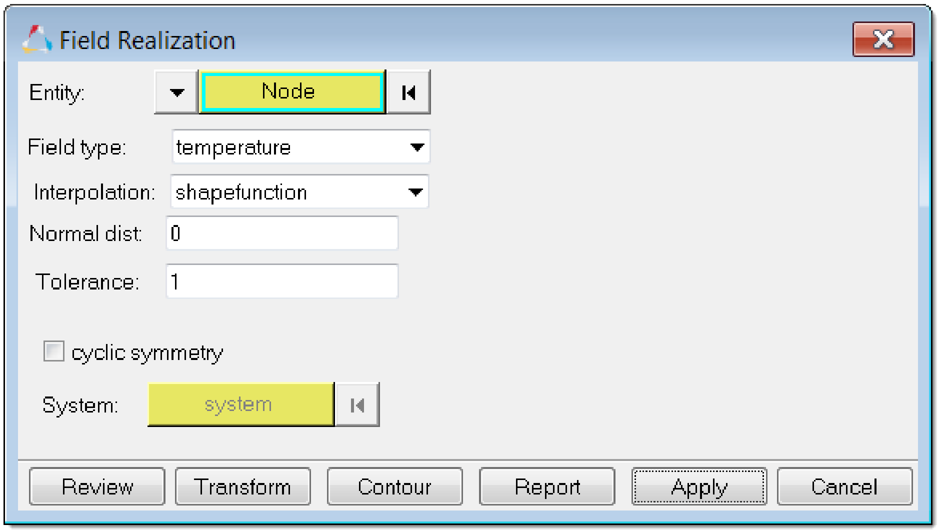


Figure 114:

c) Click **Apply**.

5. Click **Contour** to contour the target elements of a mapped temperature load from another 3D source model.

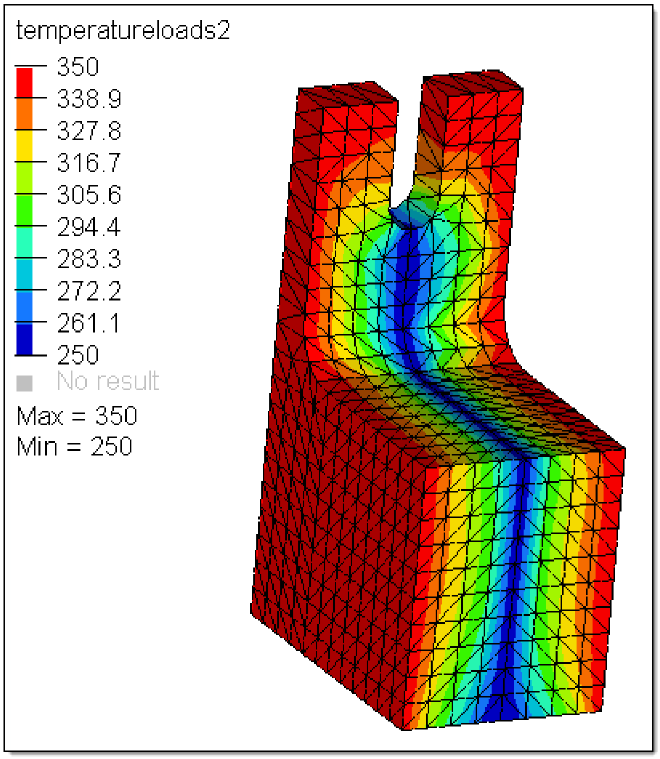


Figure 115:

Mapping Types

These tasks will introduce you to the various mapping types.

Triangulate XYZ points based on `.csv` file source data before mapping, and use linear shape function mapping (2D/3D target elements).

1. Create a field entity using a discrete `.csv` file.

Name	Value
Name	field1
ID	1
Engineering data	
Type	discrete
System type	real
Source	csv file
File	C:/Discreate_triangluation/2D_DATA_csv.csv

Figure 116:

2. Realize the field using the triangulation method. A source mesh is automatically created using the xyz points. Mapping is calculated using the linear shape function.

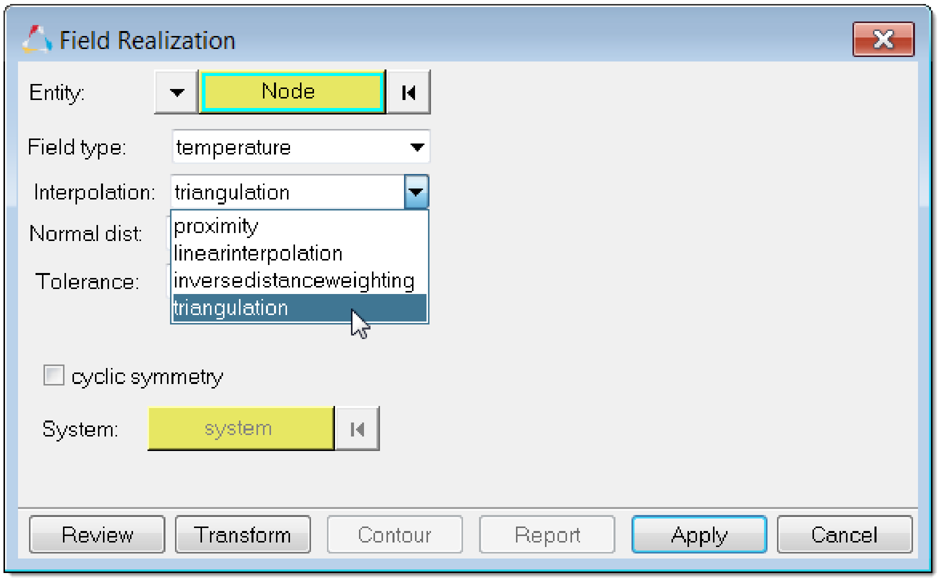


Figure 117:

3. Contour the mapped temperature.

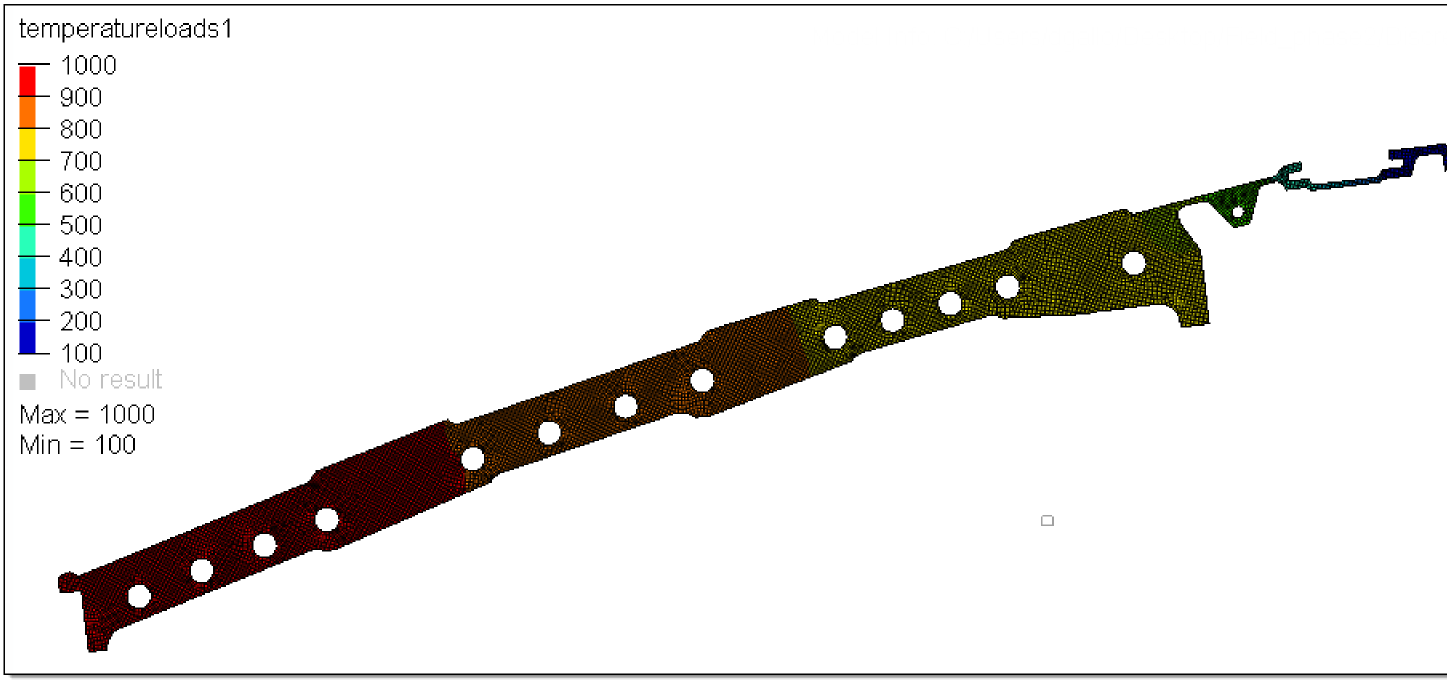


Figure 118:

Inverse Distance Weighted Mapping

This task makes use of the inverse distance weighting method.

1. Create a field entity using a discrete `.csv` file.

2. Realize the field using the inverse distance weighting method. A source mesh is automatically created using the xyz points in the `.csv` file, and mapping is calculated using the linear shape function.

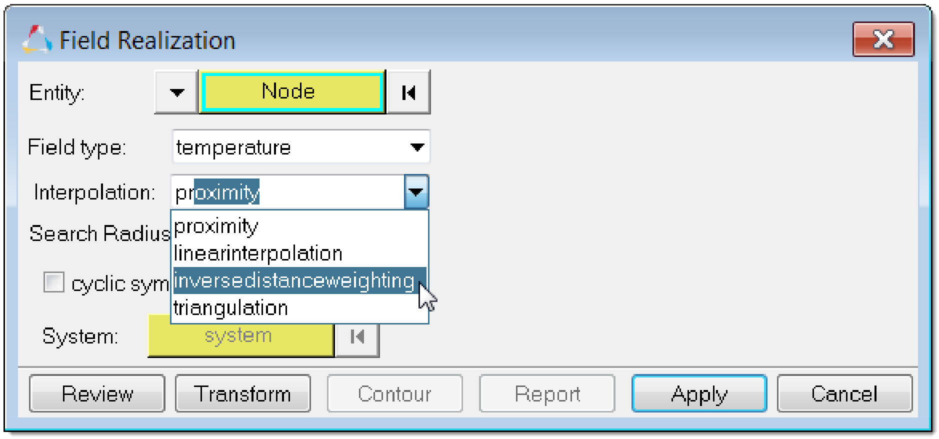


Figure 119:

Shape Function and Proximity Mapping

In this task you will map target nodes, which fall outside the source elements, using the close point approach. Nodes that fall inside the source elements will use the shape function.

If the target model is slightly bigger than the source model, within the tolerance specified in the **Field Realization** dialog, the close point method will be used to find the values of the nodes outside the model. Use **shapefunctionandproximity** instead of **shapefunction**.

Provide the correct **Search Radius** to detect the nodes that fall outside of the source model. If the Search Radius = 0 , the distance will be automatically found and searched.

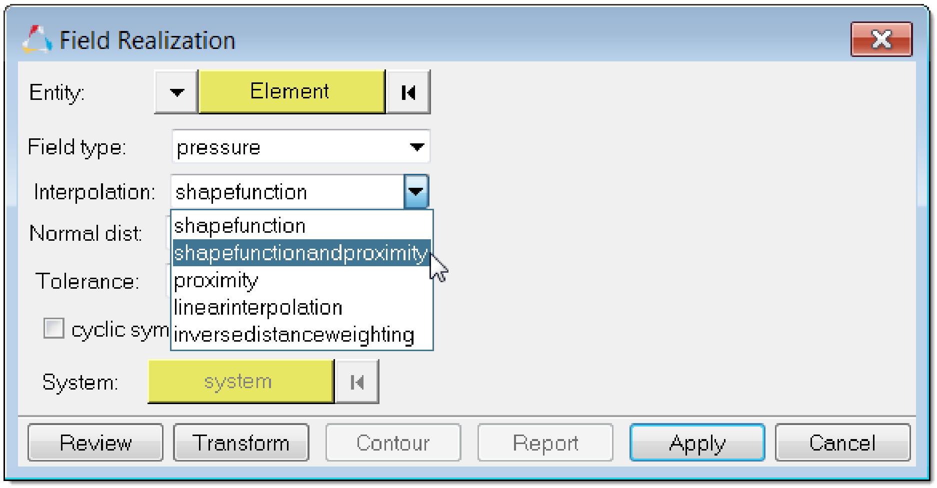


Figure 120:

Generic Field Mapping

Generic field mapping (table mapping) uses matrix columns (source data) to map and export target elements to a new Excel table.

Supported field mapping entities include: Temperature, Displacement, Pressure, and Property ID. These entities are standard HyperMesh database entries. Entries that are not standard HyperMesh supported field entities can be mapped using the Matrix Browser and a table entity.

1. Open the model containing the source data.
2. Create and define a field entity:
   a) In the Model Browser, right-click and select **Create** > **Field** from the menu.
   b) In the Entity Editor, set **Type** to continuous, **System Type** to real, and **Source** to matrix.
   c) Right-click on the **Source** field and select **Invoke** from the menu.
3. In the **Field** dialog, select the source data to map.

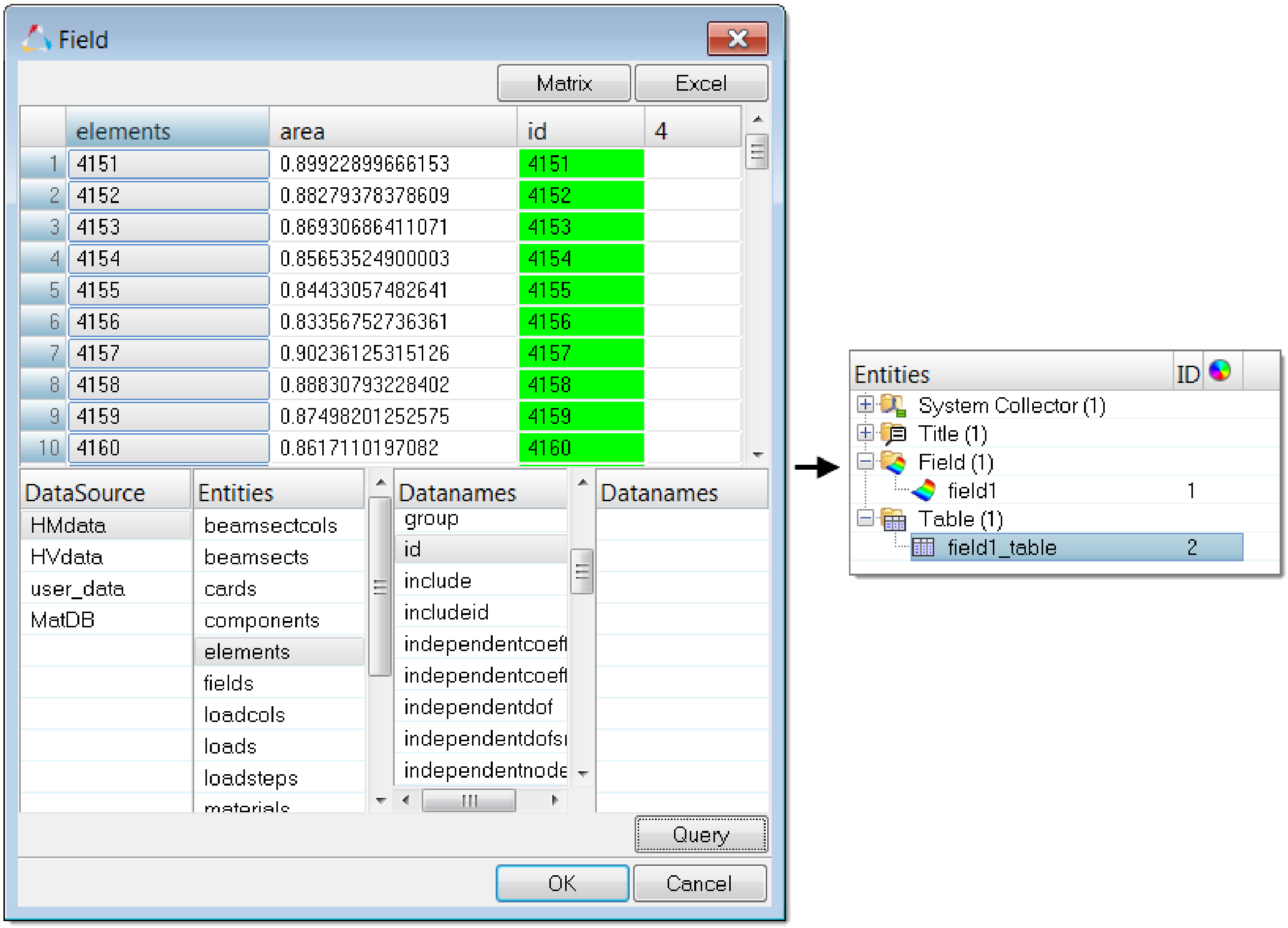


Figure 121:

a) From the **DataSource** list, select HMdata.
b) From the **Entities** list, select elements or nodes.
c) Click the **Query** button.
d) In the panel area, use the elems/nodes selector to select elements/nodes and then click **Proceed**.
e) In the matrix, click the **Elements** column.
f) From the **Datanames** list, select the type of data to import and then click the **Query** button.
g) Click **OK**.
A table entity containing all of the mapping entities based on the source model is automatically created.

4. Delete the source model.

5. Realize the field:

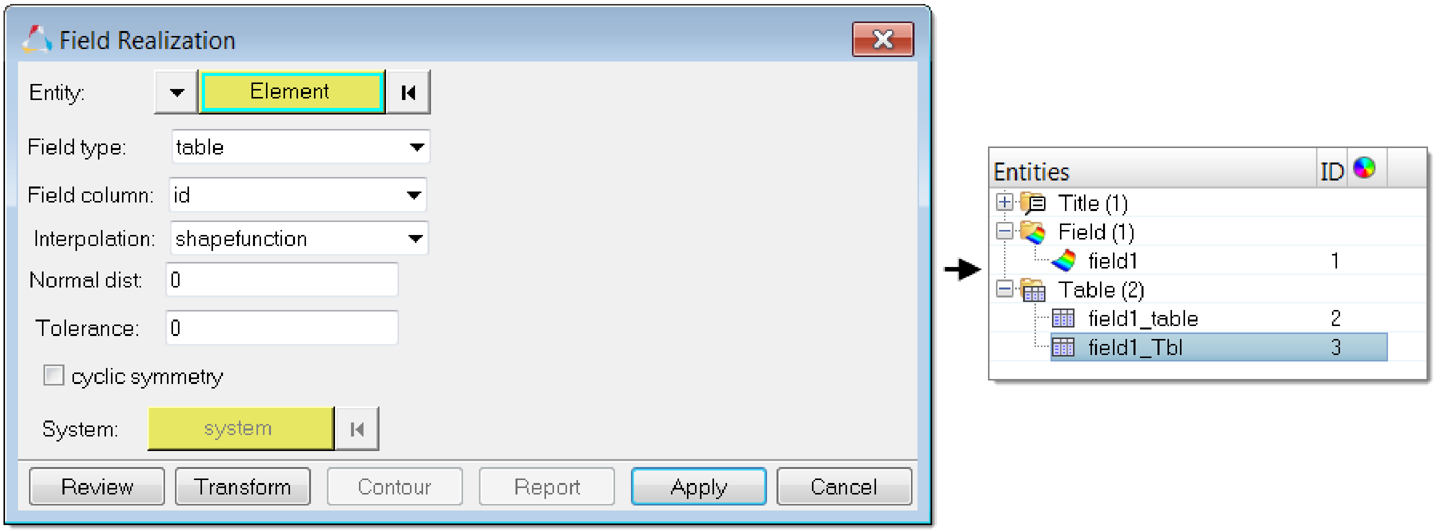


Figure 122:

a) In the Model Browser, right-click on the **field** and select **Realize** from the menu.

b) Set **Field Type** to table.

c) Select the table created by the Matrix Browser.

d) Set **Field Column** to the column name used in the Matrix Browser.

e) Click **Apply**.

A new table entity is created based on the target element/nodes after mapping (field realization).

6. Export target elements to Excel:

a) Open the **Matrix browser** from the menu bar by clicking **Tools** > **Matrix Browser**.

b) Set **Worksheet** to the new table created during mapping.

c) Click **Excel**.

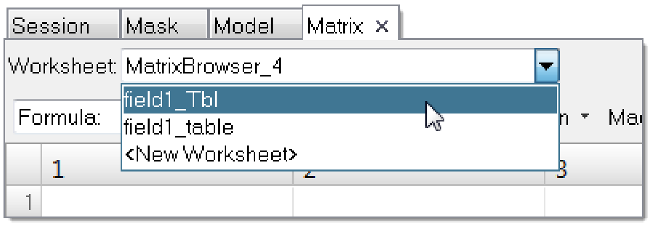


Figure 123:

Reviewing and Transforming Source Data

When source data is not in the correct location and overlaps with the target model, the tools provided in the **Field Realization** dialog can be used to transform the source model to the target model's location with linear transformation, rotation, or scale methods.

The transformed source model will be stored with the field entity and will remain in the database for future reviews.

1. Open or import the target model.
2. Create and define a field entity:
   a) In the Model Browser, right-click and select **Create** > **Field** from the menu.
   b) In the Entity Editor, define the source model data (results, csv, and current model).

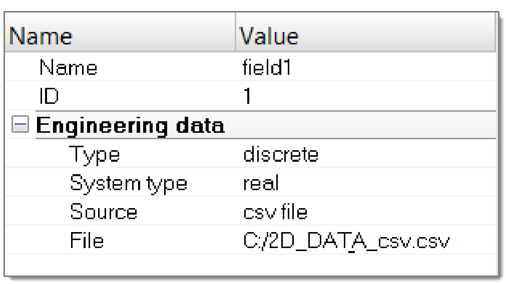


Figure 124:

3. Realize the field:
   a) In the Model Browser, right-click and select **Realize** from the menu.
   b) In the **Field Realization** dialog, define the realization settings.

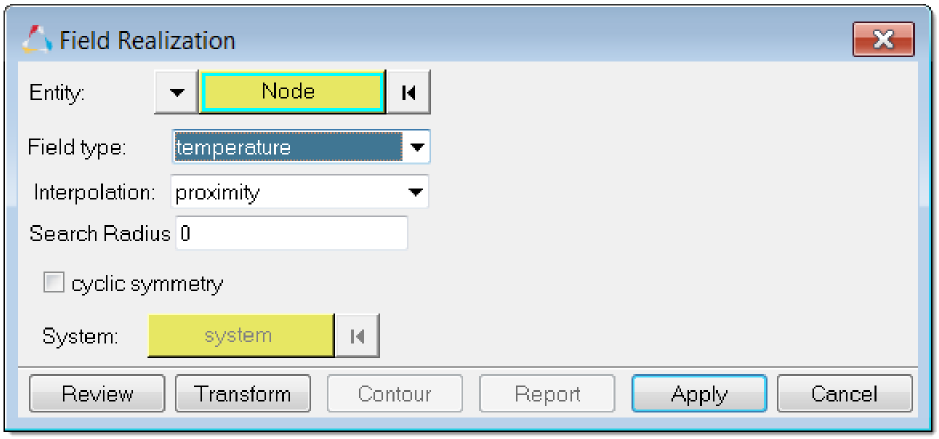


Figure 125:

c) Click **Apply**.

4. In the **Field Realization** dialog, click **Review** to display the target model (green) and the source model (pink).

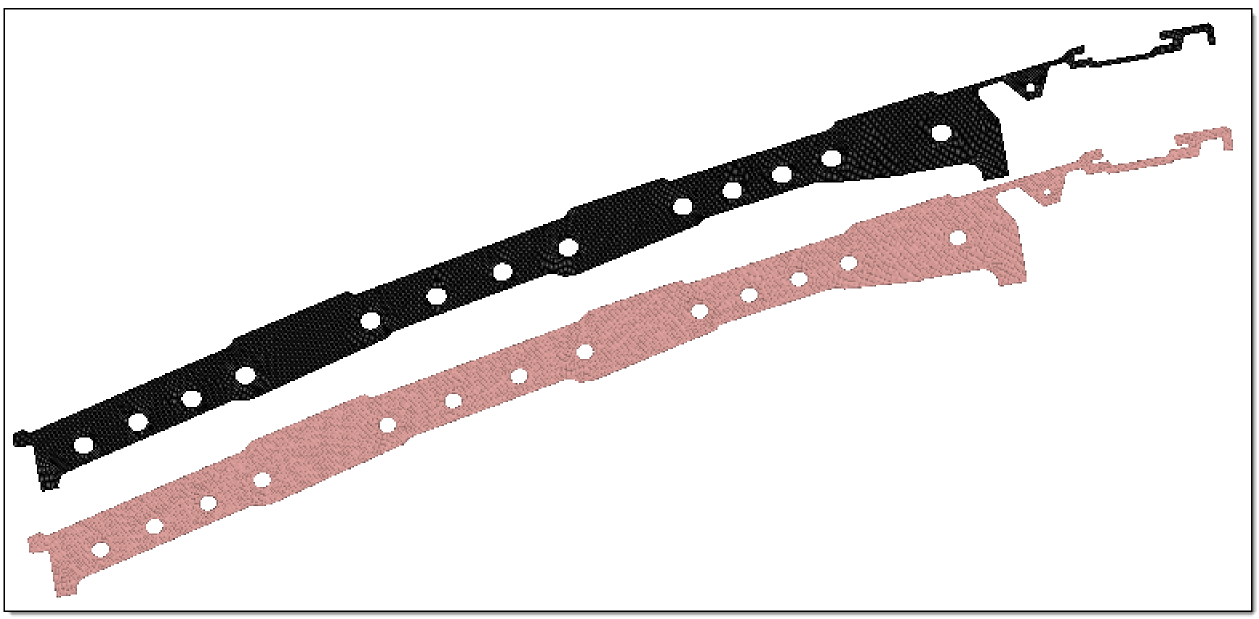


Figure 126:

5. Transform the source data:

a) In the **Field Realization** dialog, click the **Transform** button.

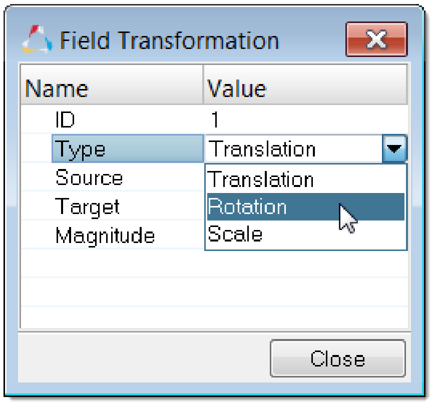


Figure 127:

b) In the **Field Transformation** dialog, define settings in the **Type** field to perform either a Translation, Rotation, or Scale transformation of the source node.
c) Click the **Close** button to perform the transformation operation.
d) Click **Reset Review** to display the updated location of the new source model.
e) Repeat this operation until the source model and the target model are in the same location.

> **Note:** Manually define the vector direction by entering the **Source** node x,y,z location, **Target** node x,y,z location, and **Magnitude** of the vector.

Nodal Force Balancing using OptiStruct

Perform nodal force balance mapping using OptiStruct to ensure mapped target nodal forces and source nodal forces are in equilibrium.

Forces from a source model are mapped to new target model nodal forces. The total load calculation on the source model and the new mapped model must be the same in order to ensure forces are balanced.

Equilibrium mapping is performed using the linear OptiStruct solver, therefore you must have the OptiStruct installed with license on the same installation directory as HyperWorks Desktop.

1. Open or import the target model. When importing, use ID Offset to preserve the source model ID.

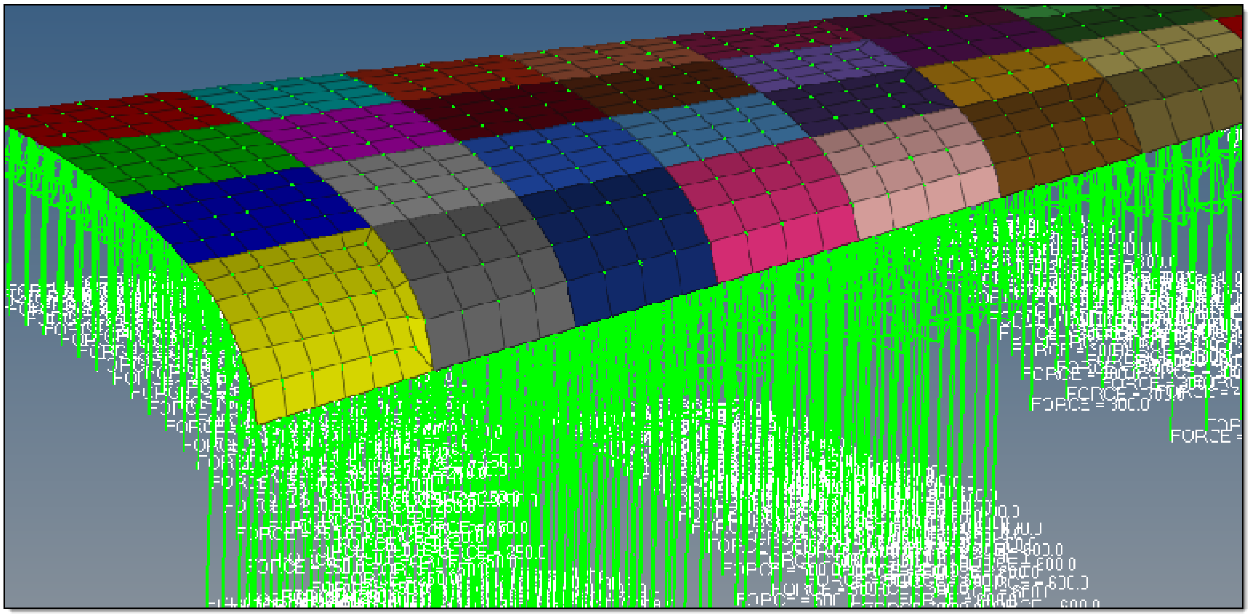


Figure 128:

2. Create and define a field entity:

a) In the Model Browser, right-click and select **Create** > **Field** from the menu.

b) In the Entity Editor, define the nodal field using the current model and forces data source, then select the loads to map.

Name	Value
Name	field1
ID	1
Engineering data	
Type	continuous
System type	real
Source	current model
Data source type	forces
Source entity list	0 Loads

Figure 129:

3. Realize the field:

a) In the Model Browser, right-click on the **field** and select **Realize** from the menu.

b) In the **Field Realization** dialog, define the realization settings.

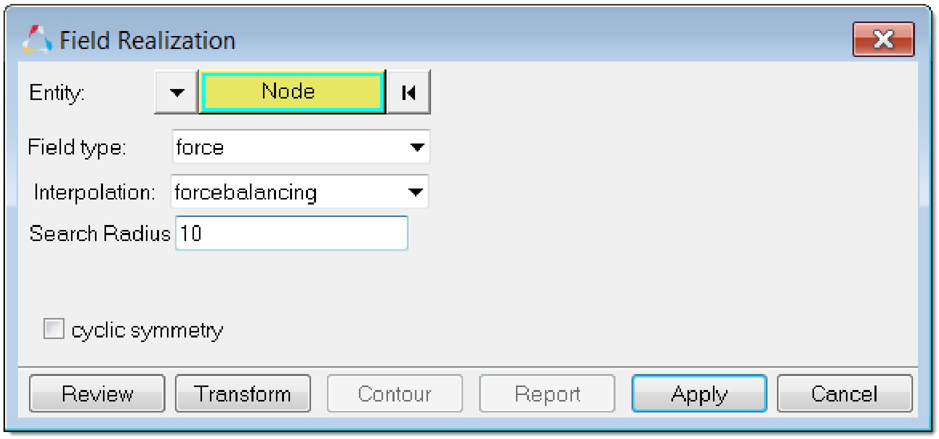


Figure 130:

c) Set **Interpolation** to forcebalancing.
d) Click **Apply**.
New forces and moments are created and applied to the nodes of the target mesh.

Figure 131:

Groups

Group entities define and store interfaces and rigid walls typically used in contact analysis.

Interfaces

Configurations 1 - 4: Define contact interactions between various parts of the model.

Rigid Walls

Configuration 5: Define a contact or sliding rigid wall in an analysis code.
In all solver interfaces except for LS-DYNA, rigid walls are created as group entities with configuration 5. In the LS-DYNA solver interface, a rigid wall is created as a rigid wall entity.

ALE Setup

Configuration 6: Define input data pertaining to the Arbitrary-Lagrangian-Eulerian LS-DYNA capability.

Abaqus Cards

Card	Description
*BLOCKAGE	Control contacting surfaces for blockage.

Card	Description
	Note: Must be used in conjunction with the *SURFACE INTERACTION card.
*CHANGE FRICTION	Change friction properties. **Note:** The Standard template only. It must be added to a load step (*STEP).
*CLEARANCE	Specify a particular initial clearance value and a contact direction for the slave nodes on a surface. **Note:** Must be added to a load step (*STEP) in explicit template.
*COHESIVE BEHAVIOR	Used to define surface-based cohesive behavior in a mechanical contact analysis. **Note:** It must be used in conjunction with the *SURFACE INTERACTION option.
*CONTACT(General Contact)	Begin the definition of general contact.
*CONTACT CLEARANCE	Define contact clearance properties.
*CONTACT CLEARANCE ASSIGNMENT	Assign contact clearances between surfaces in the general contact domain.
*CONTACT CONTROLS	Specify additional controls for contact. **Note:** Must be added to a load step (*STEP).
*CONTACT CONTROLS ASSIGNMENT	Assign contact controls for the general contact algorithm. **Note:** This card is a sub-option in the *CONTACT card image.
*CONTACT DAMPING	Define viscous damping between contacting surfaces. **Note:** This card is a sub-option in the *SURFACE INTERACTION card image.

Card	Description
*CONTACT EXCLUSIONS	Specify self-contact surfaces or surface pairings to exclude from the general contact domain. **Note:** This card is a sub-option in the *CONTACT card image.
*CONTACT FORMULATION	Specify a nondefault contact formulation for the general contact algorithm. **Note:** This card is a sub-option in the * CONTACT card image.
*CONTACT INCLUSIONS	Specify self-contact surfaces or surface pairings to include in the general contact domain. **Note:** This card is a sub-option in the *CONTACT card image.
*CONTACT INTERFERENCE	Prescribe time-dependent allowable interferences of contact pairs and contact elements. **Note:** The Standard template only. Must be added to a load step (*STEP).
*CONTACT PAIR	Define surfaces that contact each other.
*CONTACT PROPERTY ASSIGNMENT	Assign contact properties for the general contact algorithm. **Note:** This card is a sub-option in the *CONTACT card image.
*CONTROLS	Reset solution controls. **Note:** The Standard template only.
*CONTROLS	Reset solution controls. **Note:** The Standard template only.
*COUPLING	Define a surface-based coupling constraint where the *SURFACE card points to elements.

Card	Description
	Note: The *COUPLING is also supported as rigid elements (COUP_KIN) and RBE3 (COUP_DIS) when *SURFACE points to nodes.
*DEBOND	Used to specify that crack propagation may occur between two surfaces that are initially partially bonded. **Note:** The Standard template only. The *FRACTURE CRITERION option must appear immediately following this option.
*DIAGNOSTICS	Control diagnostic messages. **Note:** Explicit template only. Must be added to a load step (*STEP).
*DISTRIBUTING	Define a distributing coupling constraint. **Note:** This card is a sub-option in the *COUPLING card image. It is also supported as COUP_DIS type rbe3 elements.
*FASTENER (SPOT WELD)	Define mesh-independent fasteners. **Note:**
*FILTER	Define a filter for output filtering. **Note:** Explicit template only
*FIXED MASS SCALING	Specify mass scaling at the beginning of the step. **Note:** Explicit template only. Must be added to a load step (*STEP).
*FRACTURE CRITERION	Used to specify the criterion for crack propagation along initially partially bonded surfaces.

Card	Description
	Note: It must appear immediately following the *DEBOND option in Abaqus/Standard and after the *COHESIVE BEHAVIOR option in Abaqus/Explicit.
*FRICTION	Specify a friction model. **Note:** This card is a sub-option in the *SURFACE INTERACTION card image.
*FRICTION	Specify a friction model. **Note:** This card is a sub-option in the *CHANGE FRICTION card image.
*INITIAL CONDITIONS	Prescribe initial conditions for an analysis.
*INTEGRATED OUTPUT SECTION	Define an integrated output section over a surface with a local coordinate system and a reference point.
*KINEMATIC	Define a kinematic coupling constraint. **Note:** This card is a sub-option in the *COUPLING card image. It is also supported as COUP_KIN type rigid elements.
*MODEL CHANGE	Remove or reactivate elements and contact pairs. **Note:** The Standard template only. Must be added to a load step (*STEP).
*PRE-TENSION SECTION	Associate a pre-tension node with a pre-tension section.
*SHELL TO SOLID COUPLING	Define a surface-based coupling between a shell edge and a solid face. **Note:**
*SURFACE	Define a surface or region in a model. **Note:**

Card	Description
*SURFACE BEHAVIOR	Define alternative pressure-overclosure relationships for contact. **Note:** This card is a sub-option in the *SURFACE INTERACTION card image.
*SURFACE INTERACTION	Define surface interaction properties. **Note:** Explicit template only. This card is defined from the Property panel in case of Standard 3D and Standard 2D templates.
*SURFACE PROPERTY ASSIGNMENT	Assign surface properties to a surface for the general contact algorithm. **Note:** This card is a sub-option in the *CONTACT card image.
*TIE	Define surface-based tie and cyclic symmetry constraints or coupled acoustic-structural interactions.
*VARIABLE MASS SCALING	Specify mass scaling during the step. **Note:** Explicit template only. Must be added to a load step (*STEP).

EXODUS Cards

Card	Description
TiedJoint	

LS-DYNA Cards

AnLS-DYNA entity that utilizes a *SET_ [NODE, SHELL, PART, and so on] keyword card belongs to a group, with the exception of Rigid Bodies/RBE2's.

The difference among configurations is the type of entities contained within a group.

Config 1
: Contains master and slave elements.

Config 2
: Contains master elements and slave nodes.

Config 3

Contains slave elements.

Config 4

Contains slave nodes.

Sliding interfaces specifications.

-
- The Keyword _TITLE option is supported. The _THERMAL(IREAD==3) option is not supported.
- Use the additional cards option in Keyword decks to select number of lines of data. If this is on, two additional cards are available.
- In Structured, additional cards are controlled by using the *IREAD* variable. Valid values are 0, 1, and 2.
- Boxes, part sets, and sets are supported.
- The $HMNAME fields are used for names. When using the _TITLE option, the 70-character field is considered a comment.
- If the line following the keyword (No TITLE option), or the first line of the Structured card contains $HM_NAME, the name supplied is read and used as the group's name. If the string $HM_ID also exists, this is used as the group’s ID. NAME is 16 characters, starting in Column 9. ID field is 8 characters, starting in Column 35.
- The solver interface type defines the general type of the LS-DYNA Sliding Interface. Use the Property Editor to make changes to the LS-DYNA type.

Card	**Description**
*BOUNDARY_AMBIENT_EOS	Defines the IDs of 2 load curves: 1) internal energy per unit reference specific volume and 2) relative volume.
*BOUNDARY_FLUX_SET	Define flux boundary conditions for a thermal or coupled thermal/structural analysis.
*BOUNDARY_SPH_FLOW	Define a flow of particle
*CONSTRAINED_BUTT_WELD	Define a line of coincident nodes that represent a structural butt weld between two parts defined by shell elements.
*CONSTRAINED_LAGRANGE_IN_SOLID	Provides the coupling mechanism for modeling Fluid-Structure Interaction.
*CONSTRAINED_RIGID_BODIES	Merge two rigid bodies.
*CONSTRAINED_SPOTWELD	Defines massless spot welds between non-contiguous nodal pairs.

Card	Description
*CONSTRAINED_TIE-BREAK	Define a tied shell edge to shell edge interface that can release locally as a function of plastic strain of the shells surrounding the interface nodes.
*CONSTRAINED_TIED_NODES_FAILURE	Define a tied node set with failure based on plastic strain.
*CONTACT_AIRBAG_SINGLE_SURFACE(ID)	Define a contact interface.
*CONTACT_AIRBAG_SINGLE_SURFACE_MPP(ID)	
*CONTACT_AUTO_MOVE	Move the master surface in a contact definition to close an initial gap between the slave and master surfaces.
*CONTACT_AUTOMATIC_GENERAL(ID)	Define a contact interface.
*CONTACT_AUTOMATIC_GENERAL_INTERIOR(ID)	Define a contact interface.
*CONTACT_AUTOMATIC_GENERAL_INTERIOR_MPP(	
*CONTACT_AUTOMATIC_GENERAL_MPP(ID)	
*CONTACT_AUTOMATIC_NODES_TO_SURFACE(ID)	Define a contact interface.
*CONTACT_AUTOMATIC_NODES_TO_SURFACE_MPI	
*CONTACT_AUTOMATIC_NODES_TO_SURFACE_SM(	Define a contact interface.
*CONTACT_AUTOMATIC_NODES_TO_SURFACE_SM(	
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_S	
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_S	
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_S	
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_S	
*CONTACT_AUTOMATIC_SINGLE_SURFACE(ID)	
*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTA	Define a contact interface.
*CONTACT_AUTOMATIC_SINGLE_SURFACE_MPP(ID	
*CONTACT_AUTOMATIC_SINGLE_SURFACE_SMOOT	
*CONTACT_AUTOMATIC_SINGLE_SURFACE_SMOOT	
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE(ID	

Card	Description
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_M	
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_M	
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_M	
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_O	
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_S	
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_S	
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_T	
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_T	
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_T	
*CONTACT_CONSTRAINT_NODES_TO_SURFACE(ID	
*CONTACT_CONSTRAINT_NODES_TO_SURFACE_MI	
*CONTACT_CONSTRAINT_SURFACE_TO_SURFACE(I	
*CONTACT_CONSTRAINT_SURFACE_TO_SURFACE_	
*CONTACT_DRAWBEAD(ID)	
*CONTACT_DRAWBEAD_MPP(ID)	
*CONTACT_ENTITY(ID)	Define a contact entity
*CONTACT_ENTITY_MPP(ID)	
*CONTACT_ERODING_NODES_TO_SURFACE(ID)	
*CONTACT_ERODING_NODES_TO_SURFACE_MPP(I	
*CONTACT_ERODING_SINGLE_SURFACE(ID)	
*CONTACT_ERODING_SINGLE_SURFACE_MPP(ID)	
*CONTACT_ERODING_SURFACE_TO_SURFACE(ID)	
*CONTACT_ERODING_SURFACE_TO_SURFACE_MPF	
*CONTACT_FORCE_TRANSDUCER_CONSTRAINT(ID	
*CONTACT_FORCE_TRANSDUCER_CONSTRAINT_M	
*CONTACT_FORCE_TRANSDUCER_PENALTY(ID)	

Card	Description
*CONTACT_FORCE_TRANSDUCER_PENALTY_MPP(ID	
*CONTACT_FORMING_NODES_TO_SURFACE(ID)	
*CONTACT_FORMING_NODES_TO_SURFACE_MPP(I	
*CONTACT_FORMING_NODES_TO_SURFACE_SMOO	
*CONTACT_FORMING_NODES_TO_SURFACE_SMOO	
*CONTACT_FORMING_ONEWAY_SURFACE_TO_SUR	
*CONTACT_FORMING_ONEWAY_SURFACE_TO_SUR	
*CONTACT_FORMING_ONEWAY_SURFACE_TO_SUR	
*CONTACT_FORMING_ONEWAY_SURFACE_TO_SUR	
*CONTACT_FORMING_ONEWAY_SURFACE_TO_SUR	
*CONTACT_FORMING_ONEWAY_SURFACE_TO_SUR	
*CONTACT_FORMING_ONEWAY_SURFACE_TO_SUR	
*CONTACT_FORMING_ONEWAY_SURFACE_TO_SUR	
*CONTACT_FORMING_SURFACE_TO_SURFACE(ID)	
*CONTACT_FORMING_SURFACE_TO_SURFACE_CON	
*CONTACT_FORMING_SURFACE_TO_SURFACE_CON	
*CONTACT_FORMING_SURFACE_TO_SURFACE_CON	
*CONTACT_FORMING_SURFACE_TO_SURFACE_CON	
*CONTACT_FORMING_SURFACE_TO_SURFACE_MOI	
*CONTACT_FORMING_SURFACE_TO_SURFACE_MPP	
*CONTACT_FORMING_SURFACE_TO_SURFACE_SMO	
*CONTACT_FORMING_SURFACE_TO_SURFACE_SMO	
*CONTACT_INTERIOR(ID)	Define interior contact for foam hexahedral and tetrahedral elements.
*CONTACT_INTERIOR_MPP(ID)	
*CONTACT_NODES_TO_SURFACE(ID)	

Card	Description
*CONTACT_NODES_TO_SURFACE_INTERFERENCE(I	
*CONTACT_NODES_TO_SURFACE_INTERFERENCE_	
*CONTACT_NODES_TO_SURFACE_MPP(ID)	
*CONTACT_NODES_TO_SURFACE_SMOOTH(ID)	
*CONTACT_NODES_TO_SURFACE_SMOOTH_MPP(ID	
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE(ID)	
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INT	
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INT	
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INT	
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INT	
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_MPI	
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_SM	
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_SM	
*CONTACT_RIGID_BODY_ONE_WAY_TO_RIGID_BC	Define rigid surface contact.
*CONTACT_RIGID_BODY_ONE_WAY_TO_RIGID_BC	
*CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BC	
*CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BC	
*CONTACT_RIGID_NODES_TO_RIGID_BODY(ID)	
*CONTACT_RIGID_NODES_TO_RIGID_BODY_MPP(I	
*CONTACT_RIGID_SURFACE(ID)	
*CONTACT_RIGID_SURFACE_MPP(ID)	
*CONTACT_SINGLE_EDGE(ID)	
*CONTACT_SINGLE_EDGE_MPP(ID)	
*CONTACT_SINGLE_SURFACE(ID)	
*CONTACT_SINGLE_SURFACE_MPP(ID)	
*CONTACT_SLIDING_ONLY(ID)	

Card	Description
*CONTACT_SLIDING_ONLY_MPP(ID)	
*CONTACT_SLIDING_ONLY_PENALTY(ID)	
*CONTACT_SLIDING_ONLY_PENALTY_MPP(ID)	
*CONTACT_SPOTWELD(ID)	
*CONTACT_SPOTWELD_MPP(ID)	
*CONTACT_SPOTWELD_WITH_TORSION(ID)	
*CONTACT_SPOTWELD_WITH_TORSION_MPP(ID)	
*CONTACT_SURFACE_TO_SURFACE(ID)	
*CONTACT_SURFACE_TO_SURFACE_CONTRACTION	
*CONTACT_SURFACE_TO_SURFACE_INTERFERENCI	
*CONTACT_SURFACE_TO_SURFACE_INTERFERENCI	
*CONTACT_SURFACE_TO_SURFACE_INTERFERENCI	
*CONTACT_SURFACE_TO_SURFACE_INTERFERENCI	
*CONTACT_SURFACE_TO_SURFACE_MPP(ID)	
*CONTACT_SURFACE_TO_SURFACE_SMOOTH(ID)	
*CONTACT_SURFACE_TO_SURFACE_SMOOTH_MPP(	
*CONTACT_SURFACE_TO_SURFACE_THERMAL_FRI(	
*CONTACT_THERMAL_SURFACE_TO_SURFACE_(ID)	
*CONTACT_THERMAL_SURFACE_TO_SURFACE_MPP	
*CONTACT_TIEBREAK_NODES_TO_SURFACE(ID)	
*CONTACT_TIEBREAK_NODES_TO_SURFACE_MPP(	
*CONTACT_TIEBREAK_SURFACE_TO_SURFACE(ID)	
*CONTACT_TIEBREAK_SURFACE_TO_SURFACE_MPI	
*CONTACT_TIED_NODES_TO_SURFACE(ID)	
*CONTACT_TIED_NODES_TO_SURFACE_CONSTRAI	
*CONTACT_TIED_NODES_TO_SURFACE_CONSTRAI	

Card	Description
*CONTACT_TIED_NODES_TO_SURFACE_MPP(ID)	
*CONTACT_TIED_NODES_TO_SURFACE_OFFSET(ID	
*CONTACT_TIED_NODES_TO_SURFACE_OFFSET_M	
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE(ID)	
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAI	
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAI	
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_CON	
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_CON	
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_MPP(	
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_OFFS	
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_OFFS	
*CONTACT_TIED_SURFACE_TO_SURFACE(ID)	
*CONTACT_TIED_SURFACE_TO_SURFACE_CONSTR	
*CONTACT_TIED_SURFACE_TO_SURFACE_CONSTR	
*CONTACT_TIED_SURFACE_TO_SURFACE_MPP(ID)	
*CONTACT_TIED_SURFACE_TO_SURFACE_OFFSET(	
*CONTACT_TIED_SURFACE_TO_SURFACE_OFFSET_	
*CONTACT_TIED_SURFACE_TO_SURFACE_TITLE(ID	
*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFAC	Define a two-dimensional contact or slide line.
*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFAC	
*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFAC	
*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFAC	
*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFAC	
*CONTACT_2D_AUTOMATIC_TIED_TITLE	Defines a two-dimensional contact or slide line. May be used with rigid body material.

Card	Description
*DATABASE_FSI	Used to output information about certain coupled Lagrangian surfaces.
*DATABASE_NODAL_FORCE_GROUP	Define a nodal force group for output into ASCII file `NODFOR` and the binary file `XTFILE`.
*ELEMENT_TRIM	Define a part subset to be trimmed by *DEFINE_CURVE_TRIM
*INITIAL_GAS_MIXTURE	Used to specify a) which ALE multi-material groups may be present inside an ALE mesh set at time zero and b) the corresponding reference gas temperature and density which define the initial thermodynamic state of the gases.
*INITIAL_VOID (PART and SET)	Define initial voided part set IDs or part numbers.
*INITIAL_VOLUME_FRACTION	Define initial volume fractions of different materials in multi-material ALE elements.
*INITIAL_VOLUME_FRACTION_GEOMETRY	Volume filling command for defining the volume fractions of various ALE multi-material group that can occupy certain regions in some specified ALE mesh set.
*SET_MULTI-MATERIAL_GROUP_LIST(TITLE)	Defines an ALE multi-material set ID which contains a collection of one or more ALE multi-material group IDs.

Nastran Cards

Contact, thermal analysis definitions and non-structural mass are represented using group entities.

Card	Description
AESURF	Specifies an aerodynamic control surface as a member of the set of aerodynamic extra points.
BCBODY	Defines a flexible or rigid contact body in 2D or 3D. **Note:** Defined using the Interfaces panel.
BCBODY1	Defines a flexible or rigid contact body in 2D and 3D.
BCONECT	Defines the touching and touched contact bodies.

Card	Description
BCTABL1	Defines a Contact Table **Note:** Not supported in the BCTABLE Manager.
BCTABLE	Defines a contact table. **Note:** Use BCTABLE Manager tool to create BCTABLE, located in the utilities tab inside NASTRAN1
BCTSET	**Note:** Defined using the Interfaces panel
BSURF	Defines a contact body or surface defined by Element IDs. **Note:** Defined using the Interfaces panel
BSURFS	3D Contact Region Definition by Solid Elements. **Note:** NX Nastran only. Defined using the Interfaces panel
CONDUCTION	Defines CHBDYE slave elements used for thermal conduction analysis. **Note:** Defined using the Interfaces panel
CONVECTION	Defines CHBDYE slave elements used for thermal convection analysis, and also allows for CONV continuation cards to be defined. **Note:** Has PCONV card image. Defined using the Interfaces panel.
NSM1	Defines non-structural mass per unit length/area on properties or elements. **Note:** Defined using the NSM panel.
NSML1	Defines lumped non-structural mass on properties or elements. **Note:** Defined using the NSM panel.

Card	Description
PCONV	Specifies the free convection boundary condition properties of a boundary condition surface element used for heat transfer analysis. **Note:** Defined using the Interfaces panel, using CONVECTION group type.
RADIATION	Defines CHBDYE slave elements used for thermal radiation analysis. **Note:** Has RADM card image. Defined using the Interfaces panel.
RADM	Defines the radiation properties of a boundary element for heat transfer analysis. **Note:** Defined using the Interfaces panel, using RADIATION group type.
SPBLND1	Defines a strip based blending of two splines.
SPBLND2	Defines a curve based blending of two splines.
SPRELAX	Defines relaxation of a spline based on an adjacent spline.
SPLINE1	Defines a surface spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.
SPLINE2	Defines a beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries defined by regular arrays of aerodynamic points.
SPLINE4	Defines a curved surface spline for interpolating motion or forces for aeroelastic problems on general aerodynamic geometries.
SPLINE5	Defines a 1D beam spline for interpolating motion and/or forces for aeroelastic problems on aerodynamic geometries.
SPLINE6	Defines a 6DOF or 3DOF finite surface spline for interpolating motion and/or forces between two meshes.
SPLINE7	Defines a 6DOF finite beam spline for interpolating motion and/or forces between two meshes.

Card	Description
SPLINRB	Defines a rigid body spline for interpolating motion or forces for aeroelastic problems on general aerodynamic geometries.

OptiStruct Cards

Contact, thermal analysis definitions, multi-body dynamics bodies, non-structural mass, rigid walls and section outputs are represented using group entities.

Card	Description
CONTACT	Defines a contact interface for Small Displacement Nonlinear Analysis (NLSTAT), Nonlinear Transient Dynamics (NLGEOM), and Contact-based Thermal Analysis (HEAT). **Note:** Bulk Data Entry
CONDUCTION	Defines CHBDYE slave elements used for thermal conduction analysis. **Note:** Bulk Data Entry
CONVECTION	Defines CHBDYE slave elements used for thermal conduction analysis, and also allows for CONV continuation cards to be defined. **Note:** Bulk Data Entry
GROUND	Defines a ground body out of a list of finite element properties, elements, and grid points. **Note:** Bulk Data Entry
MBCNTDS	Defines a Multibody Contact between a set of nodes and a deformable surface. **Note:** Bulk Data Entry
MBCNTR	Defines a Multibody contact between rigid bodies. **Note:** Bulk Data Entry
NODESET	

Card	Description
NSM1	Defines non-structural mass per unit area or per unit length for a list of elements or properties. **Note:** Bulk Data Entry
NSML1	Defines lumped non-structural mass for a list of elements or properties. **Note:** Bulk Data Entry
PCONV	Defines a free convection boundary condition properties. **Note:** Bulk Data Entry
PFBODY	Defines a flexible body out of a list of finite element properties, elements, and grid points. **Note:** Bulk Data Entry
PRBODY	Defines a rigid body out of a list of finite element properties, elements and grid points. **Note:** Bulk Data Entry
SECT	Defines a section for force output in geometric nonlinear analysis. **Note:** Bulk Data Entry
TIE	Defines a tied contact in Linear Static Analysis (STATICS) and Small Displacement Nonlinear Analysis (NLSTAT). Penalty-based and MPC-based TIE contacts are available and can be selected using `CONTPRM`, `TIE`. **Note:** Bulk Data Entry
XDAMP	Defines the values for Raleigh damping for geometric nonlinear dynamic analysis.

Card	Description
	Note: Bulk Data Entry

PAM-CRASH 2G Cards

Card	Description
CNTAC / NTYPE = 1	Special sliding without separation
CNTAC / NTYPE = 10	Internal solid anti-collapse contact
CNTAC / NTYPE = 21	Body-to-multiplane contact
CNTAC / NTYPE = 33	Symmetric node-to-segment contact with edge treatment
CNTAC / NTYPE = 34	Non-symmetric node-to-segment contact with edge treatment
CNTAC / NTYPE = 36	Self-impacting node-to-segment contact with edge treatment
CNTAC / NTYPE = 37	Enhanced self-impacting contact for airbags
CNTAC / NTYPE = 43	Edge-to-edge master-slave contact
CNTAC / NTYPE = 44	Node-to-segment-oriented contact with smooth contact surface
CNTAC / NTYPE = 46	Edge-to-edge self-impacting contact
CNTAC / NTYPE = 54	Non-symmetric-oriented contact
CNTAC / NTYPE = 61	Node-to-analytical surface contact
CNTAC / NTYPE = 154	Implicit small sliding contact
MASS_GES /	Added mass definition (only when IDNOD = 0)
NSMAS /	Non-structural mass definition
NSMAS2/	Alternate mass distribution method

Card	Description
TIED /	Node-surface tied interface

Permas Cards

Card	Description
$CONTACT	Contact definitions
$MPC ISURFACE	Coupling of two surfaces
$MPC WLSSURFACE/ WLDSURFACE	Weld connection between nodes and surfaces
$PRETENSION PLANE	Pretension definition without detailing the threaded connection.
$PRETENSION THREAD	Modeling a threaded pretension section

Radioss Cards

Card	Description
/INICONT	Container definition for ALE material. **Note:** Block Format Keyword
/INIVOL	Defines material in the ALE containers. **Note:** Block Format Keyword
/INTER	Describes the interfaces. **Note:** Block Format Keyword
/INTER/LAGMUL/TYPE7	Describes the Lagrange Multiplier interface TYPE7. Multi usage impact interface between a master surface and a list of slaves nodes. **Note:** Block Format Keyword
/INTER/TYPE1	Defines a fluid-structure interaction. Lagrangian elements (structure) can interact with ALE elements, which model a viscous fluid.

Card	Description
	Note: Block Format Keyword
/INTER/TYPE2	Defines a TYPE2 tied interface that connects a set of slave nodes to a master surface. It can be used to connect coarse and fine meshes, model spotwelds, rivets, and so on. **Note:** Block Format Keyword
/INTER/TYPE3	Simulates impacts between two surfaces (with oriented segments). This interface works properly if the two surfaces are simply convex. **Note:** Block Format Keyword
/INTER/TYPE5	Simulates impacts between a master surface and a list of slave nodes. **Note:** Block Format Keyword
/INTER/TYPE6	Simulates contact between two rigid bodies with tabulated input of the contact force. It works similar to interface TYPE3. Contact force between the bodies can be input as a function of maximal penetration. The interface also allows you to input a force function for unloading. **Note:** Block Format Keyword
/INTER/TYPE7	Multi-usage impact interface, modeling contact between a master surface and a group of slave nodes. It is also possible to consider heat transfer and heat friction. **Note:** Block Format Keyword
/INTER/TYPE8	Simulates drawbeads. It is mainly used in the process industry to model metal forming. Drawbeads are used to prevent the metal strip from sliding during the stamping process. Normals of the master segments must be oriented toward the slave nodes (unsorted group). **Note:** Block Format Keyword
/INTER/TYPE9	Defines the ALE Lagrange with void opening and free space. Non-impacted ALE nodes are on a free surface. The grid velocity is equal to the material

Card	Description
	velocity in normal direction. The normal of the master surface elements must be oriented toward the slave nodes. **Note:** Block Format Keyword
/INTER/TYPE10	Tied contact with void. **Note:** Block Format Keyword
/INTER/TYPE11	Simulates impact between edge to Edge or lines. A line can be a beam or truss element or a shell edge or spring elements. **Note:** Block Format Keyword
/INTER/TYPE12	Defines fluid to fluid contact and enables the transmission of flow between two ALE surfaces (master and slave side). The slave node velocities are interpolated from master surface values. Then convective fluxes are calculated between the two surfaces. **Note:** Block Format Keyword
/INTER/TYPE14	Simulates impacts between a hyper-ellipsoidal rigid master surface and a list of slave nodes. The hyper-ellipsoidal surface is treated as an analytical surface (hyper-ellipsoidal surfaces are only discretized for post-processing). For this interface, generally, use a mesh whose size is finer than the lowest semi- axis of master surface. The master surface must be a MADYMO hyper-ellipsoidal surface or a Radioss hyper-ellipsoidal surface. **Note:** Block Format Keyword
/INTER/TYPE15	Ellipsoidal Surfaces to Elements Contact. It is a penalty contact interface without damping. This interface replaces interface TYPE14, especially if the mesh is coarser than the ellipsoid size. The slave surface must be a set of 3-node or 4-node segments (i.e. any kind of surface; except ELLIPS and MDELLIPS surfaces). The master surface must be a MADYMO hyper-ellipsoidal surface or a Radioss hyper-ellipsoidal surface, and Interface does not allow penetrations up to half the ellipsoid. **Note:** Block Format Keyword

Card	Description
/INTER/TYPE18	Defnes the Euler/Lagrange or ALE/Lagrange contact. Euler-Lagrange or ALE-Lagrange contact is the contact between a Lagrangian master surface and a list of Eulerian or ALE slaves nodes. Material velocity for all slave nodes is imposed by master surface with a penalty formulation. ALE slave node grid velocity is not modified by this interface. **Note:** Block Format Keyword
/INTER/TYPE19	Combination of interface TYPE7 and TYPE11, with common input based on the same slave/master surfaces. Slave node group for interface TYPE7, as well as slave and master line segments used by equivalent TYPE11 interface are virtually generated from these input surfaces. **Note:** Block Format Keyword
/INTER/TYPE20	A general single surface or surface to surface contact interface. Edge to edge contact is also possible. Penalty stiffness is constant and therefore the time step is not affected (for standard penalty stiffness). This contact interface can replace interface TYPE3, TYPE5, TYPE7, TYPE11 or TYPE19. The interface is basically defined in terms of one or two surfaces. If only one surface is used, this surface is self-impacting. If two surfaces are defined, nodes of surface two impact surface one. A symmetric treatment can be activated. Edges of surface one and two can be taken into account for the contact. Nodes can be added to surface. **Note:** Block Format Keyword
/INTER/TYPE21	Specific interface between a non-deformable master surface and a slave surface designed for stamping. All nodes of the master surface must belong to the rigid body. **Note:** Block Format Keyword
/INTER/TYPE23	Defines a contact interface for airbag fabrics, modeling contact between a master surface and a slave surface which are supposed to belong to an airbag. This is a soft penalty contact which can deal with penetrations and intersections often coming in the folded airbag mesh. This interface can be used for self-impacting.

Card	Description
	Note: Block Format Keyword

Samcef Cards

Card	Description	Examples
.BOLT	Defines a bolt contact.	.BOLT I 1 NDIST 1155 MACRO 2090 $ GROUP "PRETENS_1" METHOD 1 ! "BOLT_1" .CLM NOEUD I 1155 CHA COMP 1 VAL 150000.0 .BOLT I 2 NDIST 1156 MACRO 3090 FIX 2 $ GROUP "PRETENS_2" METHOD 2 ! "BOLT_2" .CLM NOEUD I 1156 DEP VAL 10.5 COMP 1
.MCT	Defines a flexible contact.	.MCT I 10 GROUP 17 GTAR 23 OPT 3 ! Contact default .MCT I 20 GROUP 65 GTAR 12 OPT 2 NLIM -1 UN3 1 DMIN -0.0245 KSMO 1 ! Advanced contact .MCT I 30 GROUP 45 GTAR 75 OPT 2 UN3 1 OCPO 2 KSMO 1 ! Glue contact
.STI	Defines a glue contact.	.STI I 1 GROUP 17 23 PROJ 0! Glue default .STI I 2 GROUP 35 64 ! Advanced glue
.ZYG	Defines a periodic condition or cyclic condition - explicit mode contact.	.ZYG GROUP "slaves_nodes" "master" $ TRAN 0.0 1.0 0.0 ! "contact_Periodic" .ZYG GROUP "slaves_nodes" "master" $ WAVE 1 $ AXE X $ ORIGIN 0.88366595 1.02703438 -40.0 $ ANGL 45.0 $ PROJ 0 ! "contact_cyclic"

Card	Description	Examples
.ZYG_AUTO	Defines a cyclic condition - automatic mode contact.	.ZYG AUTO GROUP "auto_elements" $ WAVE 0 $ AXE Y $

Card	Description	Examples
		ORIGIN -8.41253531 5.4064082 -16.47058824 ! "contact_Auto_Cyclic"

Hourglass

Hourglass entities define hourglass and bulk viscosity properties which are referenced via HGID in the `*PART` command.

LS-DYNA Cards

Card	Description
*HOURGLASS	Define hourglass and bulk viscosity properties which are referenced via HGID in the `*PART` command

Interface Component

Interface Component entities create an interface in interface file for use in subsequent linking calculations.

Each definition consists of a set of cards that define the interface. Interfaces may consist of a set of segments for later use with *INTERFACE_LINKING_SEGMENT, an ordered line of nodes for use with *INTERFACE_LINKING_EDGE, or an unordered set of nodes for use with *INTERFACE_LINKING_NODE.

LS-DYNA Cards

Card	Description
*INTERFACE_COMPONEN (NODE & SEGMENT)	Creates an interface in interface file for use in subsequent linking calculations.

Interface Linking

Interface Linking Discrete keyword link node(s) to an interface in an existing interface file.

This link applies to all element types. With this command, nodes in a node set must be given in the same order as they appear in the interface file.

Interface Linking Edge keyword link a series of nodes to an interface in an existing interface force file.

Interface Linking Segment keyword link segments to an interface in an existing interface file.

LS-DYNA Cards

Card	Description
*INTERFACE_LINKING (DISCRETE, EDGE and SEGMENT)	Link entities to an interface in existing interface file.

Joints

Joint entities define the kinematic relationship between two bodies (for Ball, Cylinder, Revolute, Slider joints) or three bodies (for DoubleSlider joints).

Joints are created and organized in the Mechanism Browser.

Laminates

Laminate entities define laminates, which make up a laminated structure by defining the stacking sequence of ply entities that make up the laminated structure.

Ply Laminates

Define laminates which make up flat or slightly curved laminated structures.
Ply laminates stack ply entities. The stack direction for the plies of a ply laminate is in the direction of the element's normal.

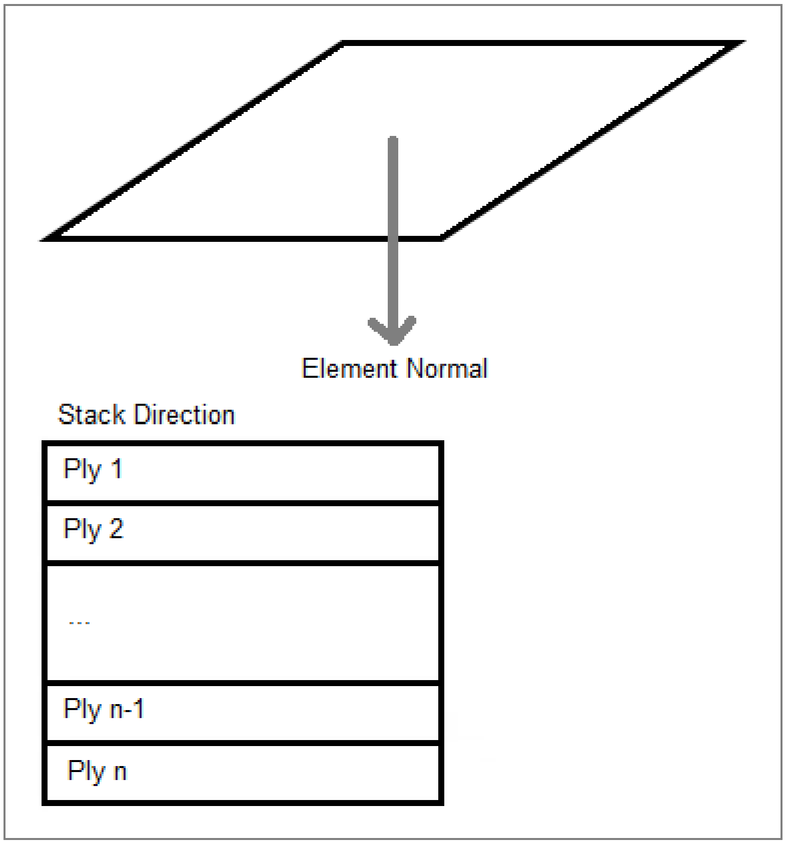


Figure 132:

Sub-Laminates

Similar to ply laminates in that they also stack ply entities. However, sub-laminates define only a portion of a laminate rather than a complete laminate structure.

The stack direction for the plies of a sublaminate is defined by an interface definition within an associated interface laminate. However, the ply order defined within a sublaminate must remain in the defined order. An interface definition of an interface laminate defines which ply of the sublaminate is on “top” and which is on the “bottom” relative to the elements normal.

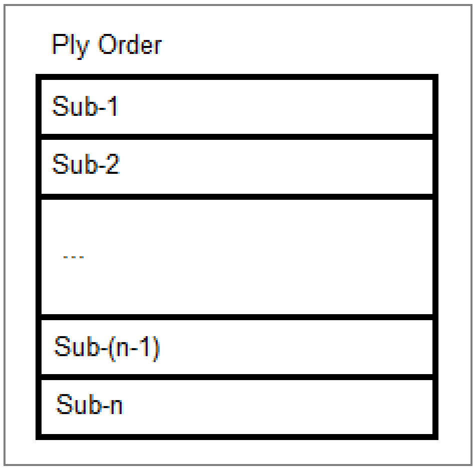


Figure 133:

Interface Laminates

Define laminates which make up complex laminated structures that wrap around corners. Interface laminates stack sublaminates. The stack direction for the sublaminates of an interface laminate is in the direction of the element's normal. The exact stacking sequence of the plies of the sublaminates is defined by the interface definitions within an interface laminate. An interface definition defines which surface plies of two sublaminates touch, or interface, with each other. Each sublaminate stacked within an interface laminate must have at least one interface definition.

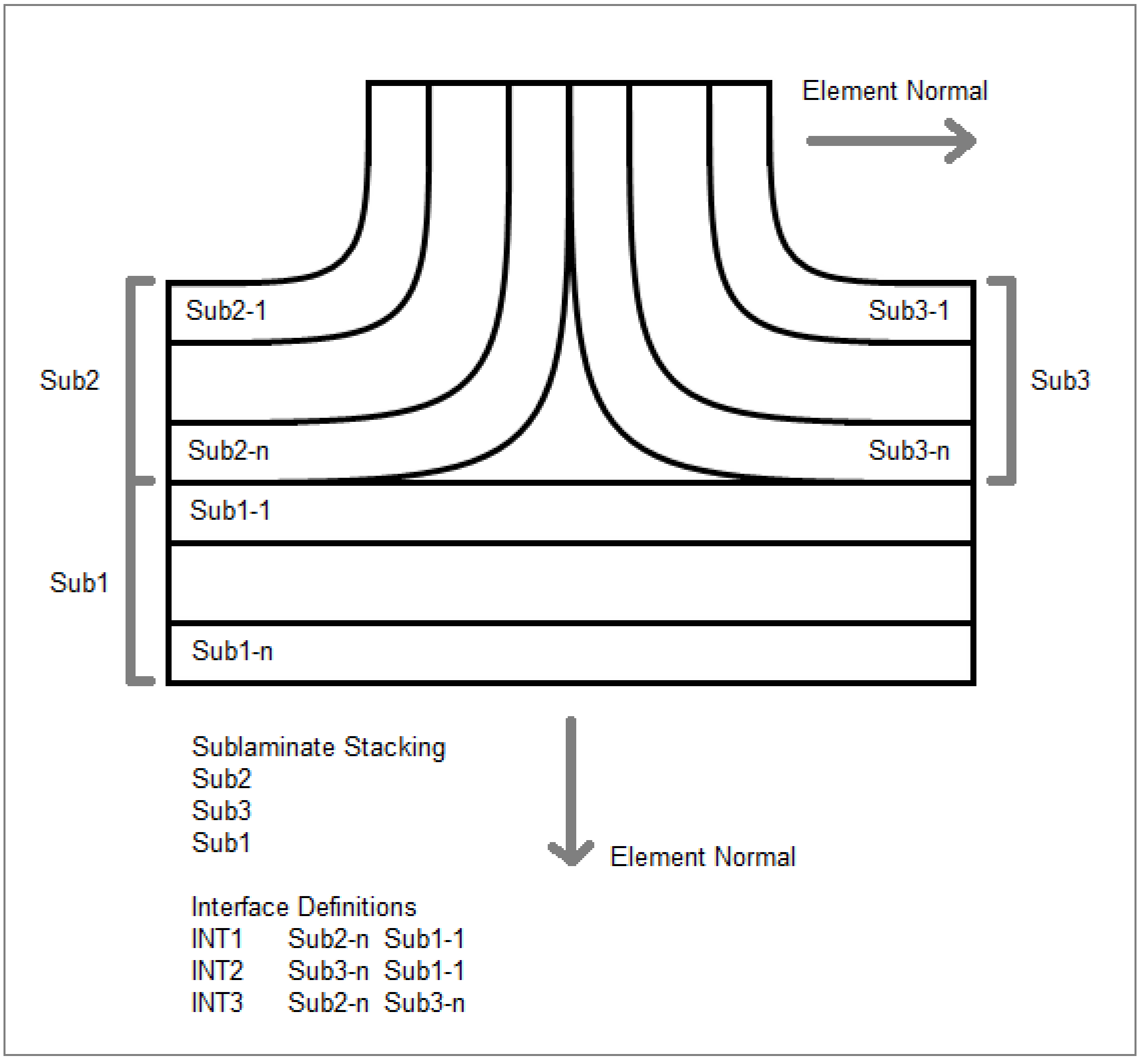


Figure 134:

Supported Solver Cards

Solver cards supported for laminates.

Abaqus Cards

If a laminate is realized, as many composite properties as needed are created to represent the ply and laminate based definition. A ‘template’ composites property (SHELL SECTION or SHELL GENERAL SECTION) has to be assigned all concerned elements first, as the algorithm derives the new properties from the same.

The laminate name ends up in the *LAYUP* parameter of the Abaqus property.

Nastran Cards

If a laminate is realized, as many composite properties (PCOMPG) as needed are created to represent the ply and laminate based definition. A 'template' composites property PCOMPP has to previously be assigned all concerned elements. The PCOMPP property is not exported.

OptiStruct Cards

Laminate realization can be used to convert a ply based model into a zone based model. In this case the export state of all ply related entities (PCOMPP property, PLY and STACK) will be set so that they are not exported.

To generate PCOMPG properties representing the ply and laminate definition, a PCOMPP card has to be defined and assigned to the elements involved in the composites definition before the realization is started.

Card	Description
STACK	Defines the stacking information and stacking sequence for ply-based composite definition. **Note:** Bulk Data Entry

Radioss Cards

The stacking definition of the property cards /PROP/STACK (TYPE17), /PROP/TYPE51, and /STACK are represented as a laminate entity.

Samcef Cards

Laminates created will be selected in the definition of the composite properties (.ETASHELL or .ETASOLID).

Create and Realize Laminates

Overview of how to create and realize laminates.

Create Laminates

Radioss

1. In the Model Browser, right-click and select **Create** > **Laminate** from the context menu.
2. In the **Create Laminate** dialog, define attributes accordingly.

Option	Description
Type	Defines the type of laminate definition.
Name	Name of laminate.
Save as	Duplicate an existing laminate.
Card image	Card image of the property that will be linked with the laminate (/PROP/TYPE17 > P17_STACK or /PROP/TYPE51 > P51_STACK), or will create /STACK and /PROP/PCOMPP if card image LAM_STACK is used.
Color	Laminate color.
Laminate option	Only the Total option is available.
Define laminate	Select plies, and define the stacking sequence. Phi and Zi will define the attributes of the same name in /PROP/TYE17 or /PROP/TYPE51. The stacking sequence can be modified at any time using the Entity Editor. You can modify the stacking sequence of a set of defined plies at any time by right-clicking on the laminate in the Model Browser and selecting **Edit** from the context menu.

3. Click **Create**.

When you create a laminate entity via the Model Browser, a corresponding property entity with the /PROP/TYPE17 (STACK) or /PROP/TYPE51 card image is created, and assigned to the laminate entity. If you select the card image LAM_STACK, HyperMesh will create the corresponding /STACK keyword and the /PROP/PCOMPP property, and link the laminate with the created property.

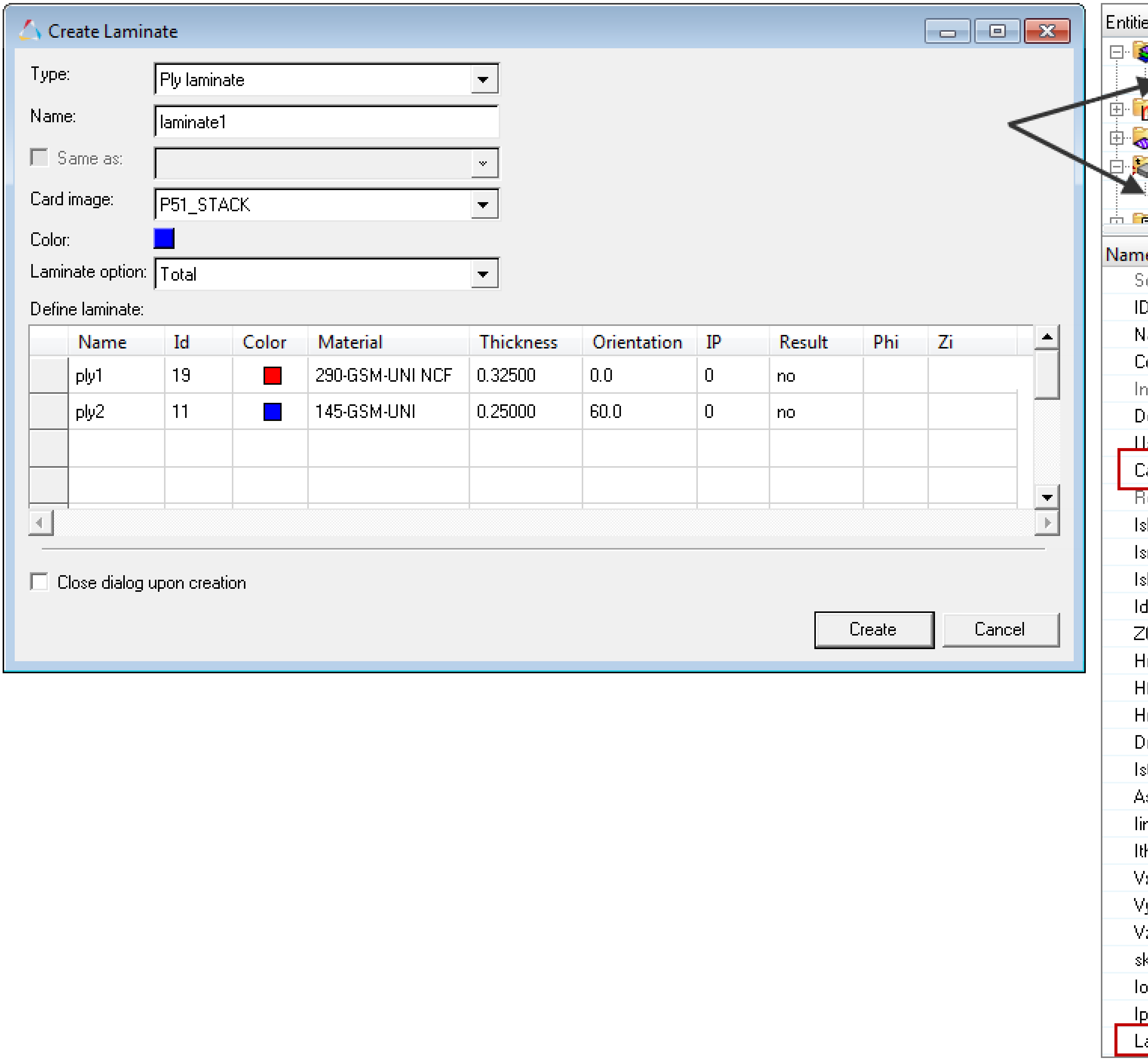


Figure 135:

If you create a property entity with the /PROP/STACK (TYPE17) or /PROP/TYPE51 card image, you can manually assign the corresponding laminate entity in the Entity Editor, Laminate field, using the Laminate selector.

Name	Value
Solver Keyword	/PROP/TYPE51/
ID	2
Name	UNI-0-60-COMBINED_laminate
Color	
Include	[Master Model]
Defined	
User Comments	Do Not Export
Card Image	P51_STACK
Regular_OR_encrypted_flag	Regular
Ishell	
Ismstr	
Ish3n	
Idrill	
Z0	
Hm	
Hf	
Hr	
Dm	
Istrain	
Ashear	
Iint	
Ithick	
Vx	
Vy	
Vz	
skew_ID	
Iorth	
Ipos	
Laminate	(1) laminate1

Figure 136:

Realize Laminates

Before you can realize laminates, a ‘template’ property must be assigned to all elements referred to through the ply and laminate entities.

In the Model Browser, right-click on a laminate and select **Realize** from the context menu.

A ply based model is converted into a zone based model. The realization algorithm creates as many properties as needed (as a copy from the template property) to represent the ply/laminate definition. Each region of the model with a unique set of layers will get its own property.

Load Steps

Load step entities define and store load cases for a given analysis.

Load steps are defined by selecting associated load collectors and output blocks.

Abaqus Cards

A load step corresponds to a *STEP definition in Abaqus model history. Load collectors, output blocks and groups within a load step are exported under the corresponding *STEP block in the Abaqus input deck.

Card	Description
*ADAPTIVE MESH	Defines an adaptive mesh domain and specifies the frequency and intensity of adaptive meshing for that domain. **Note:** Defined in the load step card image.
*BUCKLE	Obtain eigenvalue buckling estimates. **Note:** Defined in the load step card image.
*BULK VISCOSITY (Explicit)	Modify bulk viscosity parameters. **Note:** History data.
*COUPLED TEMP-DISPLACEMENT	Analyze problems where the simultaneous solution of the temperature and stress/displacement fields are necessary. **Note:** Defined in the load step card image.
*DYNAMIC (Explicit)	Dynamic stress/displacement analysis. **Note:** Defined in the load step card image.
*DYNAMIC (Standard)	Dynamic stress/displacement analysis.

Card	Description
	Note: Defined in the load step card image.
*FILE FORMAT	Specify format for results file output and invoke zero-increment results file output. **Note:** Defined in the load step card image.
*FREQUENCY	Extract natural frequencies and modal vectors. **Note:** Defined in the load step card image.
*GEOSTATIC	Verify that the geostatic stress field is in equilibrium with the applied loads and boundary conditions on the model and to iterate, if needed, to obtain equilibrium.
*HEAT TRANSFER	Transient or steady-state uncoupled heat transfer analysis. **Note:** Defined in the load step card image.
*LOAD CASE	Begin a load case definition for multiple load case analysis. **Note:** Defined in the load step card image.
*MODAL DYNAMIC	Dynamic time history analysis using modal superposition **Note:** Defined in the load step card image.
*MONITOR	Define a degree of freedom to monitor. **Note:** Defined in the load step card image.
*PRINT	Request or suppress output to the message file in an Abaqus/Standard analysis or to the status file in an Abaqus/Explicit analysis. **Note:** Defined in the load step card image.
*RADIATION_ VIEWFACTOR	Control the calculation of viewfactors during a cavity radiation analysis.

Card	Description
	Note: Defined in the load step card image; visible in *Heat Transfer analysis procedure.
*RESPONSE SPECTRUM	Calculates estimates of peak values of nodal and element responses. **Note:** Defined in the load step card image.
*RESTART WRITE	Save and reuse data and analysis results. **Note:** Defined in the load step card image.
*STATIC	Static stress/displacement analysis. **Note:** Defined in the load step card image.
*STEADY STATE DYNAMICS	Steady-state dynamic response based on harmonic excitation. **Note:** Defined in the load step card image.
*STEP	Begin a step. **Note:** Parameters are defined in the load step card image.
*VISCO	Transient, static, stress/displacement analysis with time-dependent material response (creep, swelling, and viscoelasticity) **Note:** Defined in the load step card image.

ANSYS Cards

Card	Description
ACEL	Specifies the linear acceleration of the structure.
ANTYPE	Specifies the analysis type and restart status.
CECMOD	

Card	Description
CMACEL	Specifies the translational acceleration of an element component.
CMDOMEGA	Specifies the rotational acceleration of an element component about a user-defined rotational axis.
CMOMEGA	Specifies the rotational velocity of an element component about a user-defined rotational axis.
EQSLV	Specifies the type of equation solver.
LSSOLVE	Reads and solves multiple load steps.
NLGEOM	Includes large-deflection effects in a static or full transient analysis.
NSUBST	Specifies the number of substeps to be taken this load step.
OMEGA	Specifies the rotational velocity of the structure.
OUTRES	Controls the solution data written to the database.
TIME	Sets the time for a load step.

Nastran Cards

Load step entities directly correspond to Nastran subcase definitions.

Load steps can reference constraint (SPC), static load (LOAD), multi-point constraint (MPC), fictitious support (SUPORT1), non-linear parameters (NLPARM), eigenvalue extraction data (METHOD), frequency range (FREQ), damping (SDAMPING), dynamic load (DLOAD), thermal loading (TEMP), and so on subcase definitions. Other input data is automatically generated (the SUBCASE header) or may be added to the subcase definition.

Card	Description
NLSTEP	Describes the control parameters for mechanical, thermal and coupled analysis in SOL 400 and for linear contact analysis in SOL 101.
SUBCASE	Delimits and identifies a subcase.
SUBCOM	Defines a combination subcase. **Note:** Automatically created when a new load step of type “Combination Subcase Delimiter” is created. The SUBCOM ID matches the ID of the HyperMesh load step entity.
SUBSEQ	Subcase sequence coefficients.

Card	Description
	Note: Only available for Combination Subcase Delimiter load steps. Enable SUBSEQ to input combination coefficients.
TSTEPNL	Defines parametric controls and data for nonlinear transient structural or heat transfer analysis.

OptiStruct Cards

Loadsteps directly correspond to OptiStruct subcases. Load steps can reference static loads (LOAD), constraints (SPC), and dynamic loads (DLOAD).

Supported card parameters are dependent upon the selected analysis type (solution sequence).

Card	Description
ACCELERATION	Requests acceleration vector output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select ACCELERATION.
ANALYSIS	Used in the I/O Options section to request that only a finite element analysis be performed (optimization input is ignored). It may also be used in the I/O Options or Subcase Information sections to identify the solution sequence for all subcases or for individual subcases, respectively. **Note:** Subcase Information Entry Enable the subcase option ANALYSIS and then select an analysis type.
CMSMETH	Requests that only a component mode synthesis solution be performed and to select a component mode synthesis method definition to be used. **Note:** Subcase Information Entry
CSTRAIN	Requests ply strain output for elements referencing `PCOMP`, `PCOMPP` or `PCOMPG` properties for all subcases or individual subcases, respectively.

Card	Description
	Note: I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select CSTRAIN.
CSTRESS	Requests ply stress output for elements referencing `PCOMP`, `PCOMPP` or `PCOMPG` properties for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select CSTRESS.
DESOBJ	Selects a single response definition as the objective function of an optimization, or to select system response definitions when the objective function is the least squares sum of these definitions. The `DESOBJ` command also indicates if this response is to be minimized or maximized. **Note:** Subcase Information Entry Part of the optimization problem setup, created when you define an objective.
DESSUB	Used within a subcase definition, to select a constraint set that is subcase dependent. **Note:** Subcase Information Entry Part of the optimization problem setup, created when you define a dconstraint.
DISPLACEMENT	Requests displacement vector output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select DISPLACEMENT.
DLOAD	Selects a dynamic load to be applied in a transient and frequency response analysis subcase. The `DLOAD` command can also be used to select a user-defined loading curve to be applied in a nonlinear static analysis subcase. **Note:** Subcase Information Entry Enable the subcase definition DLOAD and select load collectors with dynamic loading information (DLOAD, RLOAD1, RLOAD2, TLOAD1, TLOAD2).

Card	Description
EIGVRETRIEVE	Retrieves eigenvalue and eigenvector results of a Normal Modes Analysis from an external data file (`.eigv`). **Note:** Subcase Information Entry
EIGVSAVE	Outputs eigenvalue and eigenvector results of a Normal Modes Analysis to an external data file (`.eigv`). **Note:** Subcase Information Entry
ELFORCE	Requests structural element force output and elemental fluid particle velocity output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select ELFORCE.
ERP	Requests equivalent radiated power output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select ERP.
ESE	Requests strain energy and strain energy density output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select ESE.
EXCLUDE	Selects a set of elements to be excluded from a Linear Buckling Analysis. **Note:** Subcase Information Entry Enable the subcase option OUTPUT to select EXCLUDE, then select a SET definition.
FATDEF	Selects a `FATDEF` Bulk Data Entry that will define the elements, and their associated fatigue properties, to be considered for Fatigue Analysis. **Note:** Subcase Information Entry

Card	Description
FATPARM	Selects a `FATPARM` Bulk Data Entry that will define the parameters to be used for a Fatigue Analysis. **Note:** Subcase Information Entry
FATSEQ	Indicates that a subcase is a fatigue analysis subcase and to select a `FATSEQ` Bulk Data Entry that will define the loading sequence for the Fatigue Analysis. **Note:** Subcase Information Entry
FREQUENCY	Selects the set of forcing frequencies to be solved in a frequency response problem. **Note:** Subcase Information Entry Enables you to select load collectors with frequency information (FREQ, FREQ1, FREQ2, FREQ3, FREQ4, FREQ5).
GLOBSUB	Selects a subcase that references the global structure for Local-Global Analysis and a set of grid points in the local structure that defines the cut surface for displacement. **Note:** Subcase Information Entry
GPFORCE	Requests grid point force balance output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select GPFORCE.
GPSTRESS	Requests grid point stresses output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select GPSTRESS.
IC	Selects initial conditions for Transient and Explicit Analysis. **Note:** Subcase Information Entry

Card	Description
INVEL	Selects a multibody initial velocity set to be applied in a multibody problem. **Note:** Subcase Information Entry
LABEL	Provides a subcase with a label. **Note:** Subcase Information Entry
LOAD	Selects a static load set to be applied in linear static solutions. **Note:** Subcase Information Entry Enables you to select load collectors with static load information (FORCE, MOMENT, PLOAD, PLOAD2, PLOAD4, LOAD), or inertial loading information (GRAV, RFORCE).
MBSIM	Selects a Multibody simulation definition to be applied in a Multibody problem. **Note:** Subcase Information Entry
METHOD	Selects a method for real eigenvalue extraction. **Note:** Subcase Information Entry Select load collectors with an EIGRL card image for METHOD(STRUCT) or METHOD(FLUID).
MLOAD	Selects a Multibody load set to be applied in a Multibody problem. **Note:** Subcase Information Entry Enables you to select load collectors with multi-body dynamic load information (GRAV, MBFRC, MBFRCC, MBMNT, MBMNTC, MLOAD).
MODEWEIGHT	Defines a multiplier for computed eigenvalues that are to be used in the calculation of the "weighted reciprocal eigenvalue" and "combined compliance index" optimization responses.

Card	Description
	Note: Subcase Information Entry Part of the optimization problem setup, created when you define a response.
MOTION	Selects a Multibody motion set to be applied in a multibody problem. **Note:** Subcase Information Entry Enables you to select load collectors with multi-body dynamic motion information (MOTION, MOTNG, MOTNGC).
MPC	Selects a multi-point constraint set. **Note:** Subcase Information Entry
MPCFORCE	Requests multi-point force of constraint vector is output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry
NLPARM	Activates nonlinear solution methods for this subcase and to select the parameters used for Nonlinear Quasi-static Analysis, Geometric Nonlinear Implicit Analysis, and Nonlinear Direct Transient Analysis. **Note:** Subcase Information Entry
NSM	Selects a non-structural mass set for mass generation. The selector command must appear before the first `SUBCASE` statement. **Note:** Subcase Information Entry
OFREQUENCY	Requests a set of frequencies for output requests for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select OFREQUENCY.
OLOAD	Requests the form of applied load vector output and temperature load output for all subcases or individual subcases, respectively.

Card	Description
	Note: I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select OLOAD.
OMODES	Requests a set of modes for output requests for all subcases or for individual subcases, respectively. This command is applicable for normal modes and linear buckling solution sequences only. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select OMODES.
RESVEC	Controls the calculation of residual vectors. **Note:** Subcase Information Entry
RSPEC	References combination rules, excitation degrees-of-freedom, and input spectra for use in response spectrum analysis. **Note:** Subcase Information Entry
RWALL	Selects rigid walls for geometric nonlinear analysis. **Note:** Subcase Information Entry
SDAMPING	Applies modal damping as a function of natural frequency in modal solutions. **Note:** Subcase Information Entry
SOLVTYP	Defines the solver type to be used for static, dynamic analysis and geometric nonlinear implicit analysis. **Note:** Bulk Data Entry
SPC	Selects a single-point constraint set. **Note:** Subcase Information Entry

Card	Description
SPCFORCES	Requests single-point force of constraint vector output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry
STATSUB	Selects a static solution subcase. **Note:** Subcase Information Entry
STRAIN	Requests strain output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select STRAIN.
STRESS	Requests stress output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select STRESS.
SUBCASE	Indicates the start of a new subcase definition. **Note:** Subcase Information Entry
SUBCOM	Delimits and identifies a combination subcase. **Note:** Subcase Information Entry
SUBMODEL	Selects a submodel as a set of elements. Subcase entries specific to the selected element set can be used to solve the submodel without affecting the rest of the structure. **Note:** Subcase Information Entry
SUBSEQ	Gives the coefficients for forming a linear combination of the previous static subcases. **Note:** Subcase Information Entry
SUBTITLE	Defines the subtitle for all subcases or for individual subcases, respectively.

Card	Description
SUPORT1	Selects a fictitious support set to be applied to the model. **Note:** Subcase Information Entry
TEMP	Defines temperature at grid points or a `SET` of grid points for determination of Thermal Loading and Stress recovery.
TSTEP	Selects integration for transient analysis. **Note:** Subcase Information Entry
TTERM	Used in a geometric nonlinear subcase to define the termination time. **Note:** I/O Options and Subcase Information Entry
VELOCITY	Requests velocity vector output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Enable the subcase option OUTPUT to select VELOCITY.
WEIGHT	Defines a weighting factor (multiplier) for the compliances of individual linear static solution subcases, which are used in the calculation of the "weighted compliance" and "combined compliance index" optimization responses. **Note:** Subcase Information Entry
XHIST	Selects time history output for geometric nonlinear analysis. **Note:** Subcase Information Entry

Permas Cards

Card	Description
$CONSTRAINTS	Constraint variant bracket header line.

Card	Description
	Note: Set AnalysisProcedure to CONSTRAINTS.
$FREQLOAD	Definition of frequency dependent dynamic loads for use in frequency response analysis. **Note:** Only available when AnalysisProcedure is set to LOADING.
$LOADING	Loading variant bracket header line. **Note:** Set AnalysisProcedure to LOADING.
$NLLOAD	Define a nonlinear static load history. **Note:** Only available when AnalysisProcedure is set to LOADING.
$SITUATION	Situation definition header line. **Note:** Set AnalysisProcedure to SITUATION.
$TRANSLOAD	Definition of time dependent dynamic loads for use in transient response analysis. **Note:** TRANSLOAD cards and FREQLOAD cards are mutually exclusive.

Materials

Material entities define and store material definitions for a model.

Materials do not have a display state in the modeling window. You can color the model according to the colors assigned to each material, which is based on element material relationships, by changing the color mode to material.

Element material relationships are dependent on the solver interface. In general, when a component is assigned a material, that material assignment is applied to all elements collected by that component. The method of assigning materials at the component level is therefore referred to as indirect material assignment. Direct material assignment is performed directly on the elements themselves, typically via a property assignment. Direct material assignments always take precedence over indirect property and material assignments.

Abaqus Cards

The material keywords *MATERIAL, *GASKET MATERIAL, and *CONNECTOR BEHAVIOR are supported in the ABAQUS_MATERIAL, GASKET_MATERIAL and CONNECTOR_BEHAVIOR card images, respectively.

Abaqus has a large selection of material types, many of which are not supportered. In the Abaqus solver interface, material cards can be imported as generic materials. Generic materials are assigned the GENERIC_MATERIAL card image, and all material sub-options, parameters, and data lines are imported as simple text.

The validity or syntax of data is not checked when material cards are imported as generic materials. You must manually check the validity of the data. This method is helpful when material models are already defined, and are imported for the purpose of adding them to the corresponding sectional properties. No editing, updating, or review of the material data is intended.

The Generic Material setting can be enabled in the **File Options** dialog, that opens when you import a solver deck. You can also add an **HM_GENERIC_MATERIAL comment before a material card to have it imported as a generic material.

User comments blocks are supported for all materials. These comments are preserved during import and export of the Abaqus solver deck.

Card	Description
*BIAXIAL TEST DATA	Provides biaxial test data (compression and/or tension).
*BRITTLE CRACKING	Define cracking and postcracking properties for the brittle cracking material model.
*BRITTLE FAILURE	Used with the brittle cracking material model to specify brittle failure of the material.
*BRITTLE SHEAR	Define the postcracking shear behavior of a material used in a brittle cracking model.
*CLAY HARADENING	Define piecewise linear hardening/softening of the Cam-clay plasticity yield surface.
*CLAY PLASTICITY	Specify the plastic part of the material behavior for elastic-plastic materials that use the extended Cam-clay plasticity model.
*COMBINED TEST DATA	Simultaneously defines the normalized shear and bulk compliance or relaxation moduli as functions of time.

Card	Description
	Note: Must be used in conjunction with the *VISCOELASTIC option. Cannot be used if the *SHEAR TEST DATA and *VOLUMETRIC TEST DATA options are used.
*CONDUCTIVITY	Defines thermal conductivity. **Note:** Sub-option in the ABAQUS_MATERIAL card image.
*CONNECTOR BEHAVIOR	Begins the specification of a connector behavior.
*CONNECTOR CONSTITUTIVE REFERENCE	Defines reference lengths and angles to be used in specifying connector constitutive behavior. **Note:** Sub-option in the CONNECTOR_BEHAVIOR card image.
*CONNECTOR CONTACT FORCE	Defines the damping behavior for connector elements. **Note:** Sub-option in the CONNECTOR_BEHAVIOR card image.
*CONNECTOR DAMPING	Defines connector damping behavior. **Note:** Sub-option in the CONNECTOR_BEHAVIOR card image.
*CONNECTOR DERIVED COMPONENTS	Define user-customized components from numbered components.
*CONNECTOR ELASTICITY	Defines connector elastic behavior. **Note:** Sub-option in the CONNECTOR_BEHAVIOR card image.
*CONNECTOR FAILURE	Defines a failure criterion for connector elements. **Note:** Sub-option in the CONNECTOR_BEHAVIOR card image. Only available in the Explicit template.
*CONNECTOR FRICTION	Defines friction forces and moments in connector elements.

Card	Description
(Abaqus 6.4 version)	**Note:** Sub-option in the CONNECTOR_BEHAVIOR card image.
*CONNECTOR FRICTION (Abaqus 6.5 or later version)	Defines friction forces and moments in connector elements. **Note:** Sub-option in the CONNECTOR_BEHAVIOR card image. A *FRICTION card is needed, which can be created as a property using the FRICTION card image.
*CONNECTOR HARDENING	Defines the initial yield surface size and, optionally, the post-yield hardening behavior in connector available components of relative motion. **Note:** Sub-option in the CONNECTOR_BEHAVIOR card image.
*CONNECTOR LOCK	Defines a locking criterion for connector elements. **Note:** Sub-option in the CONNECTOR_BEHAVIOR card image.
*CONNECTOR PLASTICITY	Defines plasticity behavior in connector elements. It must be used in conjunction with the *CONNECTOR HARDENING option.
*CONNECTOR POTENTIAL	Define a restricted set of mathematical functions to represent yield or limiting surfaces in the space spanned by connector available components.
*CONNECTOR STOP	Defines connector stops for connector elements. **Note:** Sub-option in the CONNECTOR_BEHAVIOR card image.
*CREEP	Defines a creep law. **Note:** Sub-option in both the ABAQUS_MATERIAL and *GASKET MATERIAL card images. Only available in the Standard templates.
*CRUSHABLE FOAM	Defines the crushable foam plasticity model. **Note:** Sub-option in the ABAQUS_MATERIAL card image.
*CRUSHABLE FOAM HARDENING	Defines hardening for the crushable foam plasticity model.

Card	Description
	Note: Sub-option in the ABAQUS_MATERIAL card image.
*DAMPING	Defines material damping. **Note:** Sub-option in the ABAQUS_MATERIAL card image.
*DETONATION POINT	Defines detonation points for a JWL explosive equation of state. **Note:** Sub-option in the ABAQUS_MATERIAL card image. Available when *EOS, Type=JWL is selected. Only available for Abaqus/Explicit.
*DENSITY	Defines material mass density. **Note:** Sub-option in the ABAQUS_MATERIAL card image.
*DIELECTRIC	Defines dielectric material properties. **Note:** Sub-option in the ABAQUS_MATERIAL card image.
*ELASTIC	Defines elastic material properties. **Note:** Sub-option in the ABAQUS_MATERIAL card image.
*EOS	Defines a hydrodynamic material model in the form of an equation of state. **Note:** Sub-option in the ABAQUS_MATERIAL card image. It is only available for Abaqus/Explicit.
*EOS COMPACTION	Defines plastic compaction behavior for a hydrodynamic material. **Note:** Sub-option in the ABAQUS_MATERIAL card image. Available when *EOS, Type=USUP / TABULAR is selected. It is only available for Abaqus/Explicit.
*EXPANSION	Defines thermal expansion.

Card	Description
	Note: Sub-option in both the ABAQUS_MATERIAL and *GASKET MATERIAL card images.
*FABRIC	Define the in-plane behavior of a fabric material under plane stress conditions.
*FLUID BEHAVIOR	Defines fluid behavior for a fluid cavity.
*GASKET BEHAVIOR	Begins the specification of a gasket behavior. **Note:** Only available in the Standard templates.
*GASKET CONTACT AREA	Defines a gasket contact area or contact width for average pressure output. **Note:** Sub-option in the *GASKET_MATERIAL card image.
*GASKET ELASTICITY	Defines elastic properties for the membrane and transverse shear behaviors of a gasket. **Note:** Sub-option in the *GASKET_MATERIAL card image.
*GASKET THICKNESS BEHAVIOR	Defines a gasket thickness-direction behavior. **Note:** Sub-option in the *GASKET_MATERIAL card image.
*GAS SPECIFIC HEAT	Defines the specific heat of reacted gas products for an ignition and growth equation of state. **Note:** Sub-option in the ABAQUS_MATERIAL card image. Available when *EOS, Type=IGNITION AND GROWTH is selected. Only available for Abaqus/Explicit.
*HYPERELASTIC	Defines elastic properties for approximately incompressible elastomers.

Card	Description
	Note: Sub-option in the ABAQUS_MATERIAL card image. Supported sub-options: *BIAXIAL TEST DATA *PLANAR TEST DATA *UNIAXIAL TEST DATA *VOLUMETRIC TEST DATA
*HYPERFOAM	Defines elastic properties for a hyperelastic foam. **Note:** Sub-option in the ABAQUS_MATERIAL card image. Supported sub-options: *BIAXIAL TEST DATA *PLANAR TEST DATA *SIMPLE SHEAR TEST DATA *UNIAXIAL TEST DATA *VOLUMETRIC TEST DATA
*LOADING DATA	Define the loading response data for the uniaxial behavior of connector elements.
*LOW DENSITY FOAM	Define material coefficients for low-density foam materials.
*MATERIAL	Begins the definition of a material.
*MULLINS EFFECT	Defines Mullins effect material parameters for elastomers. **Note:** Sub-option in the ABAQUS_MATERIAL card image. Supported sub-options: *BIAXIAL TEST DATA *PLANAR TEST DATA *UNIAXIAL TEST DATA
*PIEZOELECTRIC	Defines piezoelectric material properties. **Note:** Sub-option in the ABAQUS_MATERIAL card image. Only available in the Standard templates.
*PLANAR TEST DATA	Provides planar test (or pure shear) data (compression and/or tension).

Card	Description
	Note: This option can be used only in conjunction with the *HYPERELASTIC option, the *HYPERFOAM option, and the *MULLINS EFFECT option. This type of test does not define the hyperelastic material constants fully; at the least, uniaxial or biaxial test data should also be given.
*PLASTIC	Defines a metal plasticity model.
*RATE DEPENDENT	Defines a rate-dependent viscoplastic model. **Note:** Sub-option in the ABAQUS_MATERIAL card image.
*REACTION RATE	Defines the reaction rate for an ignition and growth equation of state. **Note:** Sub-option in the ABAQUS_MATERIAL card image. Available when *EOS, Type=IGNITION AND GROWTH is selected. Only available for Abaqus/Explicit.
*SHEAR FAILURE	Defines a shear failure model and criterion. **Note:** Sub-option in the ABAQUS_MATERIAL card image. Only available for Abaqus/Explicit.
*SHEAR TEST DATA	Provides shear test data. **Note:** Can be used only in conjunction with the *VISCOELASTIC option.
*SIMPLE SHEAR TEST DATA	Provides simple shear test data. **Note:** Can be used only in conjunction with the *HYPERFOAM option.
*SPECIFIC HEAT	Defines specific heat. **Note:** Sub-option in the ABAQUS_MATERIAL card image.

Card	Description
*UNIAXIAL	Indicate the start of shear or uniaxial test data along a particular direction to define the behavior of a fabric material.
*UNIXIAL TEST DATA	Provides uniaxial test data (compression and/or tension). **Note:** Can be used only in conjunction with the *HYPERELASTIC option, the *HYPERFOAM option, and the *MULLINS EFFECT option.
*UNLOADING DATA	Define unloading response for the uniaxial behavior of connector elements.
*USER MATERIAL	Defines material constants for use in subroutine UMAT, UMATHT, or VUMAT. **Note:** Sub-option in the ABAQUS_MATERIAL card image.
*USER OUTPUT VARIABLES	Defines the number of user variables. **Note:** Sub-option in both the ABAQUS_MATERIAL and *GASKET MATERIAL card images.
*VISCOELASTIC	Defines dissipative behavior for use with elasticity. **Note:** Sub-option in the ABAQUS_MATERIAL card image. Supported sub-options: *COMBINED TEST DATA *SHEAR TEST DATA *VOLUMETRIC TEST DATA For the sub-options, the parameters `SHRINF` and `VOLINF` are supported.
*VOLUMETRIC TEST DATA	Provides volumetric test data.

ANSYS Cards

If an unsupported field in a card is found, a message is displayed on the status bar. Messages are also printed to the file `ansys.msg`. General slash commands, `SOLUTION` commands, `POST1` commands, and `POST26` commands are referred to as control cards. Unrecognized cards are written to a `*.hmx` file.

Card	Description
MAT	Sets the element material attribute pointer.
MP	Defines a linear material property as a constant or a function of temperature.
MPDATA	Defines property data to be associated with the temperature table.
MPDATA	Defines property data to be associated with the temperature table.
MPTEMP	Defines a temperature table for material properties. **Note:** Supports temperature tables for each material attribute.
MPTEMP	**Note:** Supports temperature tables for each material attribute.
TB	Activates a data table for nonlinear material properties or special element input.
TBDATA	Defines data for the data table.

EXODUS Cards

Card	Description
Acoustic	
Anisotropic	
Isotropic	
Isotropic_Viscoelastic	
Orthotropic	
Stochastic	

Feko

Supported media definitions and assignments are (anisotropic) dielectric media (with optional magnetic properties) and metallic media. Feko's default Free space, Perfect electric conductor, and Perfect magnetic conductor materials are also supported.

Properties are defined in HyperMesh to map the required material assignments to mesh elements.

For Feko wire segments (Bar2 and Bar3) a Property with the Card Image Segment must be defined and applied to the Components that contain segments. The Segment radius, Core medium, and Surrounding medium must be set in such a Property.

For Feko triangles (Tria3 and Tria6) a Property with the Card Image = Triangle must be defined and applied to the Components that contain triangles. The Front and Back medium, and the Face medium must be defined. For Metallic faces, the Thickness must be defined. For the boundary surface between two dielectric regions (or between Free space and a dielectric) the Face Medium should be left as <Unspecified>.

For Feko tetrahedra (Tetra4) a Property with the Card Image = Tetrahedron must be defined and applied to the Components that contain tetrahedral mesh elements. The Volume medium must be set as either a dielectric medium or Perfect electric conductor.

LS-DYNA Cards

LS-DYNA allows you to program your own materials that can be used in a simulation. Unsupported LS-DYNA materials and user defined LS-DYNA materials are assigned the MAT_UNSUPPORTED card image.

HyperMesh imports unsupported material with the MAT_UNSUPPORTED card image, and preserves their corresponding IDs and associated components.

In the MAT_UNSUPPORTED card image, all material sub-options, parameters, and data lines are supported as simple text. The validity or syntax of any data is not checked in this mode. You must manually check the validity of the data. No editing, updating, or review of the material data is intended. Also, time step calculation and mass calculation are not available for the component that refers to this material.

Card	**Description**
DI	Defines the dielectric or metallic medium properties.
SK	Assigns a material property to a surface.

Card	**Description**
*MAT_ACOUSTIC (*MAT_090)	Appropriate for tracking low pressure stress waves in an acoustic media such as air or water and can be used only with the acoustic pressure element formulation. **Note:** Material Type 90
*MAT_ALE_INCOMPRESSI (*MAT_160)	Solves imcompressible flows with the ALE solver. It should be used with the element formulation 6 and 12 in *SECTION_SOLID.

Card	Description
	Note: Material Type 160
*MAT_ANISOTROPIC_ELA (*MAT_002_ANIS)	Valid for modeling the elastic-orthotropic behavior of solids, shells and thick shells. **Note:** Material Type 2
*MAT_ANISOTROPIC_ELA (*MAT_157)	Valid for modeling the elastic-orthotropic behavior of solids, shells and thick shells and solid elements. **Note:** Material Type 157
*MAT_ANISOTROPIC_PLA (*MAT_103_P)	Simplified version of the Material Type 103. Applies only to shell elements. **Note:** Material Type 103P
*MAT_ANISOTROPIC_VIS (*MAT_103)	Applies to shell and brick elements. **Note:** Material Type 103
*MAT_ARRUDA_BOYCE_R (*MAT_127)	Provides a hyperelastic rubber model combined optionally with linear viscoelasticity. **Note:** Material Type 127
*MAT_ARUP_ADHESIVE (*MAT_169)	Used for adhesive bonding in aluminum structures. **Note:** Material Type 169
*MAT_BAMMAN (*MAT_051)	Models temperature and rate dependent plasticity with a fairly complex model that has many input parameters. **Note:** Material Type 51
*MAT_BAMMAN_DAMAGE (*MAT_052)	Extension of model 51 which includes the modeling of damage. **Note:** Material Type 52

Card	Description
*MAT_BARLAT_ANISOTRO (*MAT_033)	Used for modeling anisotropic material behavior in forming processes. **Note:** Material Type 33
*MAT_BARLAT_YLD2000 (*MAT_133)	Developed to overcome some shortcomings of the six parameters Barlat model implemented at Material Type 33. Available for shell elements only. **Note:** Material Type 133
*MAT_BARLAT_YLD96 (*MAT_033_b)	Used for modeling anisotropic material behavior in forming processes in particular for aluminum alloys. Available for shell elements only. **Note:** Material Type 33b
*MAT_BILKHU/ DUBOIS_FOAM (*MAT_075)	Used for the simulation of isotropic crushable forms. **Note:** Material Type 75
*MAT_BLATZ-KO_FOAM (*MAT_038)	Used for the definition of rubber-like foams of polyurethane. **Note:** Material Type 38
*MAT_BLATZ-KO_RUBBER (*MAT_007)	Used for the modeling of nearly incompressible continuum rubber. **Note:** Material Type 7
*MAT_BOLT_BEAM (*MAT_208)	Used with beam elements using ELFORM=6 (Discrete Beam). **Note:** Material Type 208
*MAT_BRITTLE_DAMAGE (*MAT_096)	
*MAT_CABLE_DISCRETE_ (*MAT_071)	Permits elastic cables to be realistically modeled; thus, no force will develop in compression. **Note:** Material Type 71

Card	Description
*MAT_CELLULAR_RUBBEF (*MAT_087)	Provides a cellular rubber model with confined air pressure combined with linear viscoelasticity. **Note:** Material Type 87
*MAT_CLOSED_CELL_FO/ (*MAT_053)	Used for the modeling of low density, closed cell polyurethane foam. **Note:** Material Type 53
*MAT_CODAM2 (*MAT_219)	A sub-laminate-based continuum damage mechanics model for fiber reinforced composite laminates made up of transversely isotropic layers. Used for brick, shell, and thick shell elements. **Note:** Material Type 219
*MAT_COHESIVE_ELASTI (*MAT_184)	Simple cohesive elastic model for use with solid element types 19 and 20 and is not available for other solid element formulations. **Note:** Material Type 184
*MAT_COHESIVE_GENER. (*MAT_186)	Cohesive material model that includes three general irreversible mixed-mode interaction cohesive formulations with arbitrary normalized traction-separation law given by a load curve. **Note:** Material Type 186
*MAT_COHESIVE_MIXED_ (*MAT_138)	Cohesive material model that includes a bilinear traction-separation law with quadratic mixed mode delamination criterion and a damage formulation. **Note:** Material Type 138
*MAT_COHESIVE_MIXED_ (*MAT_240)	Cohesive material formulation limited to linear softening with mixed mode delamination criterion and a damage formulation. **Note:** Material Type 240
*MAT_COHESIVE_TH (*MAT_185)	Cohesive material for use with solid element types 19 and 20. Not available for any other solid element formulation.

Card	Description
	Note: Material Type 185
*MAT_COMPOSITE_DAMA (*MAT_022)	An orthotropic material with optional brittle failure for composites can be defined. **Note:** Material Type 22
*MAT_COMPOSITE_FAILU	**Note:** Material Type 59
*MAT_COMPOSITE_FAILU (*MAT_059_SHELL)	**Note:** Material Type 59
*MAT_COMPOSITE_FAILU (*MAT_059_SOLID)	**Note:** Material Type 59
*MAT_COMPOSITE_LAYUI (*MAT_116)	Used for modeling the elastic responses of composite layups that have an arbitrary number of layers through the shell thickness. **Note:** Material Type 116
*MAT_COMPOSITE_MATR (*MAT_117)	Used for modeling the elastic responses of composites where a pre-integration is used to compute the extensional, bending, and coupling stiffness coefficients for use with the Belytschko Tsay resultant shell formulation. **Note:** Material Type 117
*MAT_COMPOSITE_MSC (*MAT_161)	Used to model the progressive failure analysis for composite materials consisting of unidirectional and woven fabric layers. **Note:** Material Type 161
*MAT_COMPOSITE_MSC_ (*MAT_162)	Used to model the progressive failure analysis for composite materials consisting of unidirectional and woven fabric layers. **Note:** Material Type 162

Card	Description
*MAT_CONCRETE_DAMAC (*MAT_072)	Used to analyze buried steel reinforced concrete structures subjected to impulsive loadings. **Note:** Material Type 72
*MAT_CONCRETE_DAMAC (*MAT_072R3)	Used to analyze buried steel reinforced concrete structures subjected to impulsive loadings. **Note:** Material Type 72R3
*MAT_CONCRETE_EC2 (*MAT_172)	Represents plain concrete only, reinforcing steel only, or a smeared combination of concrete and reinforcement. **Note:** Material Type 172
*MAT_CORUS_VEGTER (*MAT_136)	Plane stress orthotropic material model for metal forming. **Note:** Material Type 136
*MAT_CRUSHABLE_FOAM (*MAT_063)	Used to model crushable foam with optional damping and tension cutoff. **Note:** Material Type 63
*MAT_CSCM (*MAT_159)	Concrete material **Note:** Material Type 159
*MAT_CSCM_CONCRETE (*MAT_159_CONCRETE)	Concrete material **Note:** Material Type 159
*MAT_DAMPER_NONLINE (*MAT_S05)	Used for discrete springs and dampers. **Note:** Material Type SD-5
*MAT_DAMPER_VISCOUS (*MAT_S02)	Used for discrete springs and dampers. **Note:** Material Type SD-2

Card	Description
*MAT_DESHPANDE_FLECI (*MAT_154)	Used for modeling aluminum foam used as a filler material in aluminum extrusions to enhance the energy absorbing capability of the extrusion. For solid elements. **Note:** Material Type 154
*MAT_ELASTIC (*MAT_001)	Isotropic elastic material that is available for beam, shell and solid elements. **Note:** Material Type 1
*MAT_ELASTIC_FLUID (*MAT_001_FLUID)	Isotropic elastic material available for beam, shell and solid elements. **Note:** Material Type 1
*MAT_ELASTIC_PLASTIC_ (*MAT_010)	Used for the modeling of an elastic-plastic hydrodynamic material. **Note:** Material Type 10
*MAT_ELASTIC_PLASTIC_ (*MAT_004)	Temperature dependent material coefficients can be defined. **Note:** Material Type 4
*MAT_ELASTIC_SPRING_ (*MAT_074)	Permits elastic springs with damping to be combined and represented with a discrete beam element type 6. **Note:** Material Type 74
*MAT_ELASTIC_VISCOPL/ (*MAT_106)	Elastic viscoplastic material with thermal effects. **Note:** Material Type 106
*MAT_ELASTIC_WITH_VI (*MAT_060)	Used to simulate forming of glass products at high temperatures. **Note:** Material Type 60
*MAT_ELASTIC_6DOF_SP (*MAT_093)	Defined for simulating the effects of nonlinear elastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees of freedom.

Card	Description
	Note: Material Type 93
*MAT_EMMI (*MAT_151)	The Evolving Microstructural Model of Inelasticity (EMMI) is a temperature and rate-dependent state variable model developed to represent the large deformation of metals under diverse loading conditions. This model is available for 3D solid elements, 2D solid elements and thick shell forms 3 and 5. **Note:** Material Type 151
*MAT_ENHANCED_COMP((*MAT_054)	Enhanced versions of the composite model Material Type 22. **Note:** Material Types 54-55
*MAT_FABRIC (*MAT_034)	Developed for airbag materials. **Note:** Material Type 34
*MAT_FINITE_ELASTIC_S (*MAT_112)	An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. **Note:** Material Type 112
*MAT_FLD_TRANSVERSEI (*MAT_039)	Used for simulating sheet forming processes with anisotropic material. **Note:** Material Type 39
*MAT_FLD_3_PARAMETEF (*MAT_190)	Used for modeling sheets with anisotropic materials under plane stress conditions. **Note:** Material Type 190
*MAT_FORCE_LIMITED (*MAT_029)	With this material model, for the Belytschko-Schwer beam only, plastic hinge forming at the ends of a beam can be modeled using curve definitions. **Note:** Material Type 29
*MAT_FRAZER_NASH_RU	This model defines rubber from uniaxial test data.

Card	Description
(*MAT_031)	**Note:** Material Type 31
*MAT_FU_CHANG_FOAM (*MAT_083)	Rate effects can be modeled in low and medium density foams. **Note:** Material Type 83
*MAT_FU_CHANG_FOAM_ (*MAT_083_DAMAGE_DE	Rate effects can be modeled in low and medium density foams. **Note:** Material Type 83
*MAT_GAS_MIXTURE (*MAT_148)	Used for the simulation of thermally equilibrated ideal gas mixtures. **Note:** Material Type 148
*MAT_GENERAL_JOINT_D (*MAT_097)	Used to define a general joint constraining any combination of degrees of freedom between two nodes. **Note:** Material Type 97
*MAT_GENERAL_NONLINI (*MAT_121)	Very general spring and damper model. **Note:** Material Type 121
*MAT_GENERAL_NONLINI (*MAT_119)	Very general spring and damper model. **Note:** Material Type 119
*MAT_GENERAL_SPRING_	Permits elastic and elastoplastic springs with damping to be represented with a discrete beam element type 6 using six springs each acting about one of the six local degrees of freedom. **Note:** Material Type 196
*MAT_GENERAL_VISCOEI (*MAT_076)	Provides a general viscoelastic Maxwell model having up to 6 terms in the prony series expansion and is useful for modeling dense continuum rubbers and solid explosives.

Card	Description
	Note: Material Type 76
*MAT_GEOLOGIC_CAP_M (*MAT_025)	This is an inviscid two invariant geologic cap model. **Note:** Material Type 25
*MAT_GEPLASTIC_SRATE (*MAT_101)	Characterizes General Electric's commercially available engineering thermoplastics subjected to high strain rate events. **Note:** Material Type 101
*MAT_GURSON (*MAT_120)	Gurson dilatational-plastic model. Available for shell and solid elements. **Note:** Material Type 120
*MAT_GURSON_JC (*MAT_120_JC)	Enhancement of Material Type 120. Gurson model with additional Johnson-Cook failure criterion. **Note:** Material Type 120B
*MAT_GURSON_RCDC (*MAT_120_RCDC)	This is an enhancement of material Type 120. This is the Gurson model with the Wilkins Rc-Dc fracture model added. This model is available for shell and solid elements. **Note:** Material Type 120C
*MAT_HIGH_EXPLOSIVE_ (*MAT_008)	Used fo the modeling of the detonation of a high explosive. **Note:** Material Type 8
*MAT_HILL_FOAM (*MAT_177)	Highly compressible foam. **Note:** Material Type 177
*MAT_HILL_3R (*MAT_122)	Planar anisotropic material model with 3 R values. **Note:** Material Type 122

Card	Description
*MAT_HILL_90 (*MAT_243)	Used for modeling sheets with anisotropic materials under plane stress conditions. **Note:** Material Type 243
*MAT_HONEYCOMB (*MAT_026)	The major use of this material model is for honeycomb and foam materials with real anisotropic behavior. **Note:** Material Type 26
*MAT_HYDRAULIC_GAS_I (*MAT_070)	Special purpose element represents a combined hydraulic and gas-filled damper which has a variable orifice coefficient. **Note:** Material Type 70
*MAT_HYPERELASTIC_RU (*MAT_077_H)	Provides a general hyperelastic rubber model combined optionally with linear viscoelasticity. **Note:** Material Type 77
*MAT_INELASTIC_SPRIN((*MAT_094)	Elastoplastic springs with damping are represented with a discrete beam element type 6. **Note:** Material Type 94
*MAT_INELASTIC_6DOF_ (*MAT_095)	Defined for simulating the effects of nonlinear inelastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees of freedom. **Note:** Material Type 95
*MAT_ISOTROPIC_ELAST (*MAT_013)	Non-iterative plasticity with simple plastic strain failure model. **Note:** Material Type 13
*MAT_ISOTROPIC_ELAST (*MAT_012)	Very low cost isotropic plasticity model for three-dimensional solids. **Note:** Material Type 12

Card	Description
*MAT_JOHNSON_COOK (*MAT_015)	The Johnson/Cook strain and temperature sensitive plasticity is sometimes used for problems where the strain rates vary over a large range and adiabatic temperature increases due to plastic heating causes material softening. **Note:** Material Type 15
*MAT_JOHNSON_HOLMQU (*MAT_110)	Used for modeling ceramics, glass, and other brittle materials. **Note:** Material Type 110
*MAT_JOHNSON_HOLMQU (*MAT_111)	Used for modeling concrete subjected to large strains, high strain rates, and high pressures. **Note:** Material Type 111
*MAT_JOHNSON_HOLMQU (*MAT_241)	Used for modeling ceramics, glass, and other brittle materials. **Note:** Material Type 241
*MAT_KELVIN-MAXWELL_VISCOELASTIC (*MAT_061)	Used for modeling viscoelastic bodies, such as foams. **Note:** Material Type 61
*MAT_KINEMATIC_HARDE (*MAT_242)	Used to model metal sheets under cyclic plasticity loading and with anisotropy in plane stress condition. **Note:** Material Type 242
*MAT_KINEMATIC_HARDE (*MAT_226)	Used to model metal sheets under cyclic plasticity loading and with anisotropy in place stress condition. **Note:** Material Type 226
*MAT_KINEMATIC_HARDE (*MAT_125)	**Note:** Material Type 125

Card	Description
*MAT_LAMINATED_COMP (*MAT_058)	Depending on the type of failure surface, may be used to model composite materials with unidirectional layers, complete layers, complete laminates, and woven fabrics. **Note:** Material Type 58
*MAT_LAMINATED_GLASS (*MAT_032)	With this material model, a layered glass including polymeric layers can be modeled. **Note:** Material Type 32
*MAT_LAYERED_LINEAR_ (*MAT_114)	Layered elastoplastic material with an arbitrary stress versus strain curve and an arbitrary strain rate dependency can be defined. **Note:** Material Type 114
*MAT_LINEAR_ELASTIC_I (*MAT_066)	Used for simulating the effects of a linear elastic beam by using six springs each acting about one of the six local degrees of freedom. **Note:** Material Type 66
*MAT_LOW_DENSITY_FO (*MAT_057)	Used for modeling high density foams. **Note:** Material Type 57
*MAT_LOW_DENSITY_SYI (*MAT_179)	Used for modeling rate independent low density foams, which have the property that the hysteresis in the loading-unloading curve is considerably reduced after the first loading cycle. **Note:** Material Type 179
*MAT_LOW_DENSITY_SYI (*MAT_179_WITH_FAILUI	Used for modeling rate independent low density foams, which have the property that the hysteresis in the loading-unloading curve is considerably reduced after the first loading cycle. **Note:** Material Type 179
*MAT_LOW_DENSITY_SYI (*MAT_180)	Used for modeling rate independent low density foams, which have the property that the hysteresis in the loading-unloading curve is considerably reduced after the first loading cycle.

Card	Description
	Note: Material Type 180
*MAT_LOW_DENSITY_SY (*MAT_180_WITH_FAILU	Used for modeling rate independent low density foams, which have the property that the hysteresis in the loading-unloading curve is considerably reduced after the first loading cycle. **Note:** Material Type 180
*MAT_LOW_DENSITY_VIS (*MAT_073)	Used for modeling Low Density Urethane Foam with high compressibility and with rate sensitivity which can be characterized by a relaxation curve. **Note:** Material Type 73
*MAT_MICROMECHANICS (*MAT_235)	Used for modeling the elastic response of loose fabric used in inflatable structures, parachutes, body armor, blade containments, and airbags. **Note:** Material Type 235
*MAT_MODIFIED_CRUSH (*MAT_163)	Dedicated to modeling crushable foam with optional damping, tension cutoff, and strain rate effects. **Note:** Material Type 163
*MAT_MODIFIED_HONEY (*MAT_126)	Used for aluminum honeycomb crushable foam materials with anisotropic behavior. **Note:** Material Type 126
*MAT_MODIFIED_PIECEW (*MAT_123)	An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain-rate dependency can be defined. **Note:** Material Type 123
*MAT_MODIFIED_PIECEW (*MAT_123_RATE)	An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. **Note:** Material Type 123

Card	Description
*MAT_MODIFIED_PIECEW (*MAT_123_RTCL)	An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. **Note:** Material Type 123
*MAT_MODIFIED_ZERILL (*MAT_065)	Rate and temperature sensitive plasticity model which is sometimes preferred in ordinance design calculations. **Note:** Material Type 65
*MAT_MOMENT_CURVATU (*MAT_166)	Beam material for performing non-liner elastic or multi-linear plastic analysis. **Note:** Material Type 166
*MAT_MOONEY_RIVLIN_I (*MAT_027)	A two-parametric material model for rubber can be defined. **Note:** Material Type 27
*MAT_MTS (*MAT_088)	Available for applications involving large strains, high pressures and strain rates. **Note:** Material Type 88
*MAT_NONLINEAR_ELAST (*MAT_067)	Used for simulating the effects of nonlinear elastic and nonlinear viscous beams by using six springs each acting about one of the six local degrees of freedom. **Note:** Material Type 67
*MAT_NONLINEAR_ORTH (*MAT_040)	Used fo the definition of an orthotropic nonlinear elastic material based on a finite strain formulation with the initial geometry as the reference. **Note:** Material Type 40
*MAT_NONLINEAR_PLAST (*MAT_068)	Used for simulating the effects of nonlinear elastoplastic, linear viscous behavior of beams by using six springs each acting about one of the six local degrees of freedom.

Card	Description
	Note: Material Type 68
*MAT_NULL (*MAT_009)	Allows equations of state to be considered without computing deviatoric stresses. **Note:** Material Type 9
*MAT_OGDEN_RUBBER (*MAT_077_O)	Provides the Ogden (1984) rubber model combined optionally with linear viscoelasticity. **Note:** Material Type 77
*MAT_ORIENTED_CRACK (*MAT_017)	This material may be used to model brittle materials which fail due to large tensile stresses. **Note:** Material Type 17
*MAT_ORTHOTROPIC_EL/ (*MAT_002)	Valid for modeling the elastic-orthotropic behavior of solids, shells and thick shells. **Note:** Material Type 2
*MAT_ORTHOTROPIC_SI (*MAT_221)	An orthotropic material with optional simplified damage and optional failure for composites can be defined. Only valid for 3D solid elements with reduced or full integration. **Note:** Material Type 221
*MAT_ORTHOTROPIC_TH (*MAT_021)	A linearly elastic, orthotropic material with orthotropic thermal expansion. **Note:** Material Type 21
*MAT_ORTHOTROPIC_VIS (*MAT_086)	Allows the definition of an orthotropic material with a viscoelastic part. Applies to shell elements. **Note:** Material Type 86

Card	Description
*MAT_PIECEWISE_LINEA (*MAT_024)	An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. **Note:** Material Type 24
*MAT_PLASTICITY_COMP (*MAT_124)	An isotropic elastic-plastic material where unique yield stress versus plastic strain curves can be defined for compression and tension. **Note:** Material Type 124
*MAT_PLASTICITY_COMP (*MAT_155)	An isotropic elastic-plastic material where unique yield stress versus plastic strain curves can be defined for compression and tension. **Note:** Material Type 155
*MAT_PLASTIC_KINEMAT (*MAT_003)	Suited to model isotropic and kinematic hardening plasticity with the option of including rate effects. **Note:** Material Type 3
*MAT_PLASTICITY_POLYN (*MAT_089)	An elasto-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. **Note:** Material Type 89
*MAT_PLASTICITY_WITH_ (*MAT_082, *MAT_081)	An elasto-visco-plastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. **Note:** Material Types 81-82
*MAT_PLASTICITY_WITH_ (*MAT_081_ORTHO)	Invokes an orthotropic damage model. **Note:** Material Types 81-82
*MAT_PLASTICITY_WITH_ (*MAT_082_ORTHO_RCD	Invokes the damage model developed by Wilkins. **Note:** Material Types 81-82

Card	Description
*MAT_PML_ELASTIC (*MAT_230)	A perfectly-matched layer (PML) material. An absorbing layer material used to simulate wave propagation in an unbounded isotropic elastic medium. Only available for solid 8-node bricks (element type 2). **Note:** Material Type 230
*MAT_PML_ELASTIC_FLU (*MAT_230_FLUID)	A perfectly-matched layer (PML) material with a pressure fluid constitutive law. Used in a wave-absorbing layer adjacent to a fluid material (*MAT_ELASTIC_FLUID) in order to simulate wave propagation in an unbound fluid medium. **Note:** Material Type 230
*MAT_POLYMER (MAT_168)	Used for brick elements. **Note:** Material Type 168
*MAT_POWDER (*MAT_271)	Used to analyze the compaction and sintering of cemented carbides. Only available for solid elements. **Note:** Material Type 271
*MAT_POWER_LAW_PLAS (*MAT_018)	This is an isotropic plasticity model with rate effects which uses a power law hardening rule. **Note:** Material Type 18
*MAT_PSEUDO_TENSOR (*MAT_016)	This model has been used to analyze buried steel reinforced concrete structures subjected to impulsive loadings. **Note:** Material Type 16
*MAT_RATE_SENSITIVE_I (*MAT_141)	Used to model the simulation of an isotropic ductile polymer with strain rate effects. Known as the modified Ramaswamy-Stouffer model. **Note:** Material Type 141

Card	Description
*MAT_RATE_SENSITIVE_I (*MAT_064)	Used to model strain rate sensitive elasto-plastic material with a power law hardening. **Note:** Material Type 64
*MAT_RESULTANT_ANISO (*MAT_170)	This model is available for the Belytschko-Tsay and the C0 triangular shell elements and is based on a resultant stress formulation. **Note:** Material Type 170
*MAT_RESULTANT_PLAST (*MAT_028)	A resultant formulation for beam and shell elements including elasto-plastic behavior can be defined. **Note:** Material Type 28
*MAT_RIGID (*MAT_020)	Parts made from this material are considered to belong to a rigid body (for each part ID). **Note:** Material Type 20
*MAT_RIGID_DISCRETE (*MAT_220)	Rigid material for shells or solids. **Note:** Material Type 220
*MAT_SAMP-1 (*MAT_187)	Uses an isotropic C-1 smooth yield surface for the description of non-reinforced plastics. **Note:** Material Type 187
*MAT_SCHWER_MURRAY (*MAT_145)	The Schwer & Murray Cap Model, known as the Continuous Surface Cap Model, is a three invariant extension of the Geological Cap Model (Material Type 25) that also includes viscoplasticity for rate effects and damage mechanics to model strain softening. **Note:** Material Type 145
*MAT_SEATBELT (*MAT_B01)	Define a seat belt material. **Note:** Material Type B01

Card	Description
*MAT_SEISMIC_BEAM (*MAT_191)	Enables lumped plasticity to be developed at the 'node 2' end of Belytschko-Schwer beams (resultant formulation). **Note:** Material Type 191
*MAT_SHAPE_MEMORY (*MAT_030)	This material model describes the superelastic response present in shape-memory alloys that is the peculiar material ability to undergo large deformations with full recovery in loading-unloading cycles. **Note:** Material Type 30
*MAT_SID_DAMPER_DIS((*MAT_069)	The side impact dummy uses a damper that is not adequately treated by the nonlinear force versus relative velocity curves since the force characteristics are dependent on the displacement of the piston. **Note:** Material Type 69
*MAT_SIMPLIFIED_JOHN! (*MAT_098)	Used for problems where the strain rates vary over a large range. **Note:** Material Type 98
*MAT_SIMPLIFIED_JOHN! (*MAT_099)	Implemented with multiple through thickness integration points. Extension of Model 98 to include orthotropic damage as a means of treating failure in aluminum panels. **Note:** Material Type 99
*MAT_SIMPLIFIED_RUBBI (*MAT_181)	Provides a rubber and foam model defined by a single uniaxial load curve or by a family of uniaxial curves at discrete strain rates. **Note:** Material Type 181
*MAT_SIMPLIFIED_RUBBI (*MAT_181_WITH_FAILUI	Provides a rubber and foam model defined by a single uniaxial load curve or by a family of uniaxial curves at discrete strain rates. **Note:** Material Type 181
*MAT_SIMPLIFIED_RUBBI (*MAT_183)	Provides an incompressible rubber model defined by a single uniaxial load curve for loading (or a table if rate effects are considered) and a single uniaxial load curve for unloading.

Card	Description
	Note: Material Type 183
*MAT_SOIL_AND_FOAM (*MAT_005)	Simple model that works in some ways like a fluid. **Note:** Material Type 5
*MAT_SOIL_AND_FOAM_ (*MAT_014)	The input for this model is the same as for *MAT_SOIL_AND_FOAM; however, when the pressure reaches the failure pressure, the element loses its ability to carry tension. **Note:** Material Type 14
*MAT_SOIL_CONCRETE (*MAT_078)	Permits concrete and soil to be efficiently modeled. **Note:** Material Type 78
*MAT_SPECIAL_ORTHOTF (*MAT_130)	Applies to Belytschko-Tsay and the C0 triangular shell elements. **Note:** Material Type 130
*MAT_SPOTWELD (*MAT_100)	Applies to beam elements Type 9 and to solid elements Type 1 with Type 6 hourglass controls. **Note:** Material Type 100
*MAT_SPOTWELD_DAIML (*MAT_100_DAIMLER_CH	Applies to solid elements Type 1 with Type 6 hourglass controls. **Note:** Material Type 100
*MAT_SPOTWELD_DAMA(FAILURE (*MAT_100_DAMAGE-FAILURE)	Applies to beam element type 9 and to solid element type 1 with type 6 hourglass controls. **Note:** Material Type 100
*MAT_SPRING_ELASTIC (*MAT_S01)	Used for discrete springs and dampers. Provides a translational or rotational elastic spring located between two nodes.

Card	Description
	Note: Material Type SD-1
*MAT_SPRING_ELASTOPL (*MAT_S03)	Used for discrete springs and dampers. Provides an elastoplastic translational or rotational spring with isotropic hardening located between two nodes. **Note:** Material Type SD-3
*MAT_SPRING_GENERAL_ (*MAT_S06)	Used for discrete springs and dampers. Provides a general nonlinear translational or rotational spring with arbitrary loading and unloading definitions. **Note:** Material Type SD-6
*MAT_SPRING_INELASTI (*MAT_S08)	Used for discrete springs and dampers. Provides an inelastic tension or compression only, translational or rotational spring. **Note:** Material Type SD-8
*MAT_SPRING_MAXWELL (*MAT_S07)	Used for discrete springs and dampers. Provides a three Parameter Maxwell Viscoelastic translational or rotational spring. **Note:** Material Type SD-7
*MAT_SPRING_NONLINEA (*MAT_S04)	Used for discrete springs and dampers. Provides a nonlinear elastic translational and rotational spring with arbitrary force versus displacement and moment versus rotation, respectively. **Note:** Material Type SD-4
*MAT_STEEL_EC3	Tables and formulae from Eurocode 3 are used to derive the mechanical properties and their variation with temperature, although these can be overriden by user-defined curves. **Note:** Material Type 202
*MAT_STEINBERG (*MAT_011)	This material is available for modeling materials deforming at very high strain rates (>105) and can be used with solid elements.

Card	Description
	Note: Material Type 11
*MAT_STEINBERG_LUND (*MAT_011_LUND)	This material is a modification of the Steinberg model to include the rate model of Steinberg and Lund (1989). **Note:** Material Type 11
*MAT_STRAIN_RATE_DEF (*MAT_019)	A strain rate dependent material can be defined. **Note:** Material Type 19
*MAT_TABULATED_JOHNS (*MAT_224)	Defines an elasto-viscoplastic material with arbitrary stress versus strain curve(s), and arbitrary strain rate dependency. **Note:** Material Type 224
*MAT_TEMPERATURE_DEI (*MAT_023)	Defines an orthotropic elastic material with arbitrary temperature dependency. **Note:** Material Type 23
*MAT_THERMAL_ISOTROI (*MAT_T01)	Defines isotropic thermal properties. **Note:** Thermal Material Property Type 1
*MAT_THERMAL_ISOTROI (*MAT_T06)	Defines isotropic thermal properties that are temperature dependent specified by load curves. **Note:** Thermal Material Property Type 6
*MAT_THERMAL_ORTHOT (*MAT_T02)	Defines orthotropic thermal properties. **Note:** Thermal Material Property Type 2
*MAT_THERMO_ELASTO_ (*MAT_188)	Defines creep separately from plasticity. **Note:** Material Type 188

Card	Description
*MAT_TRANSVERSELY_AN (*MAT_142)	Used for an extruded foam material that is transversely istropic, crushable, and of low density with no significant Poisson effect. **Note:** Material Type 142
*MAT_TRANSVERSELY_AN (*MAT_037)	Simulates sheet forming processes with anisotropic material. **Note:** Material Type 37
*MAT_TRANSVERSELY_AN (*MAT_037_NLP_FAILURE	Simulates sheet forming processes with anisotropic material. **Note:** Material Type 37
*MAT_TRANSVERSELY_AN (*MAT_037_ECHANGE)	Simulates sheet forming processes with anisotropic material. **Note:** Material Type 37
*MAT_TRIP (*MAT_113)	Isotropic elasto-plastic material model that applies to shell elements only. **Note:** Material Type 113
*MAT_UHS_STEEL (*MAT_244)	Material for hot stamping and press hardening. **Note:** Material Type 244
*MAT_UNSUPPORTED	
*MAT_USER_DEFINED_M.	User can supply their own subroutines. **Note:** Material Types 41-50
*MAT_VACUUM (*MAT_140)	Dummy material representing a vacuum in a multi-material Euler/ALE model. **Note:** Material Type 140
*MAT_VISCOELASTIC (*MAT_006)	Used for the modeling of viscoelastic behavior for beams (Hughes-Liu), shells, and solids.

Card	Description
	Note: Material Type 6
*MAT_VISCOELASTIC_HI (*MAT_178)	Highly compressible foam. **Note:** Material Type 178
*MAT_VISCOELASTIC_LO (*MAT_234)	Used for modeling the elastic and viscoelastic response of loose fabric used in body armor, blade containments, and airbags. **Note:** Material Type 234
*MAT_VISCOPLASTIC_MI (*MAT_225)	An elasto_viscoplastic material with an arbitrary stress versus strain curve and arbitrary strain rate dependency can be defined. Kinematic, isotropic, or a combination or kinematic and isotropic hardening can be specified. Also, failure based on plastic strain can be defined. **Note:** Material Type 225
*MAT_VISCOUS_FOAM (*MAT_062)	Used to represent the Confor Foam on the ribs of EuroSID side impact dummy. **Note:** Material Type 62
*MAT_WINFRITH_CONCR (*MAT_084)	Only Type 84 includes rate effects. Model is a smeared crack, smeared rebar model implemented in the 8-node single integration point continuum element. **Note:** Material Type 84 and Type 85
*MAT_WOOD (*MAT_143)	Wood material. **Note:** Material Type 143
*MAT_WOOD_FIR (*MAT_143_FIR)	Wood material. **Note:** Material Type 143
*MAT_WOOD_OPTION	Transversely isotropic material and is available for solid elements.

Card	Description
	Note: Material Type 143
*MAT_WOOD_PINE (*MAT_143_PINE)	Wood material. **Note:** Material Type 143
*MAT_WTM_STM	Anisotropic-viscoplastic material model. **Note:** Material Type 135
*MAT_WTM_STM_PLC (*MAT_135_PLC)	Anisotropic material. **Note:** Material Type 135PLC
*MAT_1DOF_GENERALIZE (*MAT_146)	Linear or spring damper that allows different degrees of freedom at two nodes to be coupled. **Note:** Material Type 146
*MAT_3-PARAMETER_BARLAT (*MAT_036)	Used for modeling sheets with anisotropic materials under plane stress conditions. **Note:** Material Type 36
*MAT_3-PARAMETER_BARLAT_NLF (*MAT_036_NLP)	Used for modeling sheets with anisotropic materials under plane stress conditions. **Note:** Material Type 36

Nastran Cards

The PCOMP card contains all information regarding composite materials, including the orientation of the longitudinal direction of each ply. The material longitudinal axis of the element is obtained either by rotating the x axis of the element THETA degree (from THETA field in the element card) counterclockwise, or by projecting the x axis of a system (from MCID field in the element card) onto the surface of the element. Each ply orientation, shown a as ply direction, is obtained by rotating the material longitudinal axis THETAi degree (from the THETAi field in the PCOMP card) counterclockwise.

Card	Description
MAT1	Defines the material properties for linear isotropic materials.
MAT2	Defines the material properties for linear anisotropic materials for two-dimensional elements.
MAT4	Defines the constant or temperature-dependent thermal material properties for conductivity, heat capacity, density, dynamic viscosity, heat generation, reference enthalpy, and latent heat associated with a single-phase change.
MAT5	Defines the thermal material properties for anisotropic materials.
MAT8	Defines the material property for an orthotropic material for isoparametric shell elements.
MAT9	Defines the material properties for linear, temperature-independent, anisotropic materials for solid isoparametric elements.
MAT10	Defines material properties for fluid elements in coupled fluid-structural analysis.
MATEP	Specifies elasto-plastic material properties to be used for large deformation analysis. Used in SOL 600 only.
MATHE	Specifies hyperelastic (rubber-like) material properties for nonlinear (large strain and large rotation) analysis in SOL 600 and MD Nastran SOL 400 only.
MATHP	Specifies material properties for use in fully nonlinear (i.e., large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).
MATG	Specifies gasket material properties to be used in SOL 600 and MD Nastran SOL 400.
MATHP	Specifies material properties for use in fully nonlinear (that is, large strain and large rotation) hyperelastic analysis of rubber-like materials (elastomers).
MATEP	Specifies elasto-plastic material properties.
MATS1	Specifies stress-dependent material properties for use in applications involving nonlinear materials.
MATT1	Specifies temperature-dependent material properties on MAT1 entry fields via TABLEMi entries.
MATT2	Specifies temperature-dependent material properties on MAT2 entry fields via TABLEMj entries.

Card	Description
MATT4	Specifies table references for temperature-dependent MAT4 material properties.
MATT8	Specifies temperature-dependent material properties on MAT8 entry fields via TABLEMi entries.
MATT9	Specifies temperature-dependent material properties on MAT9 entry fields via TABLEMk entries.

OptiStruct Cards

Card	Description
MAT1	Defines the material properties for linear, temperature-independent, and isotropic materials. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
MAT2	Defines the material properties for linear, temperature-independent, and anisotropic materials for two-dimensional elements. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
MAT3	Defines the material properties for linear, temperature-independent, and orthotropic materials used by the CTAXI and CTRIAX6 axisymmetric elements. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
MAT4	Defines constant thermal material properties for conductivity, density, and heat generation. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
MAT5	Defines the thermal material properties for anisotropic materials.

Card	Description
	Note: Bulk Data Entry Exported in large field format by `optistructlf` template.
MAT8	Defines the material properties for linear temperature-independent orthotropic material for two-dimensional elements. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
MAT9	Defines the material properties for linear, temperature-independent, and anisotropic materials for solid elements. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
MAT9ORT	Defines the material properties for linear, temperature-independent, and orthotropic materials for solid elements in terms of engineering constants. **Note:** Bulk Data Entry
MAT10	Defines material properties for fluid elements in coupled fluid-structural analysis. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
MATFAT	Defines material properties for fatigue analysis. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
MATF1	Specifies frequency-dependent material properties on `MAT1` entry fields via *TABLEDi* entries. **Note:** Bulk Data Entry
MATF2	Specifies frequency-dependent material properties on `MAT2` entry fields via *TABLEDi* entries.

Card	Description
	Note: Bulk Data Entry
MATF8	Specifies frequency-dependent material properties on `MAT8` entry fields via *TABLEDi* entries. **Note:** Bulk Data Entry
MATF9	Specifies frequency-dependent material properties on `MAT9` entry fields via *TABLEDi* entries. **Note:** Bulk Data Entry
MATF10	Specifies frequency-dependent material properties on `MAT10` entry fields via *TABLEDi* entries. **Note:** Bulk Data Entry
MATHE	Defines material properties for nonlinear hyperelastic materials. The Polynomial form is available and various material types can be defined by specifying the corresponding coefficients. **Note:** Bulk Data Entry
MATPE1	Defines the material properties for poro-elastic materials. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
MATT1	Specifies temperature-dependent material properties on `MAT1` entry fields via *TABLEMi* entries. **Note:** Bulk Data Entry
MATT2	Specifies temperature-dependent material properties on `MAT2` entry fields via *TABLEMj* entries. **Note:** Bulk Data Entry

Card	Description
MATT4	Defines temperature-dependent material properties for the corresponding `MAT4` Bulk Data Entry fields via *TABLEMi* entries. **Note:** Bulk Data Entry
MATT5	Defines temperature-dependent material properties on `MAT5` entry fields via *TABLEMi* entries. **Note:** Bulk Data Entry
MATT8	Specifies temperature-dependent material properties on `MAT8` entry fields via *TABLEMi* entries. **Note:** Bulk Data Entry
MATT9	Specifies temperature-dependent material properties on `MAT9` entry fields via *TABLEMk* entries. **Note:** Bulk Data Entry
MGASK	Defining the material properties for gasket-like materials. **Note:** Bulk Data Entry

PAM-CRASH Cards

Card	Description
MAT_SECURE	
PLY	
TYPE 1	Elastic Plastic **Note:** Post-Yield behavior - defined by Yield Stress list box.
TYPE 2	Crushable Foam
TYPE 5	Linear Viscoelastic

Card	Description
TYPE 7	Isotropic - Elastic-Plastic-Hydrodynamic
TYPE 11	Blatz-Ko Rubber
TYPE 16	Elastic-Plastic with Damage
TYPE 17	Hyperleastic Mooney-Rivlin **Note:** To enter LTC on card 4, a curve must exist.
TYPE 18	Hyperelastic Hart-Smith
TYPE 20	Inelastic Crushable Foam
TYPE 21	Elastic Foam **Note:** Curve definition may be defined with points or with curve entities.
TYPE 22	Non-linear Viscoelastic
TYPE 25	Solid for Foam Side Impact Barriers
TYPE 26	Elastic Plastic with Gurson Damage Model
TYPE 30	Unidirectional Composite Bi-Phase **Note:** To enter IPLY on card 3, a material collector with a defined PLY_DATA card image must exist.
TYPE 31	Unidirectional Composite Non-linear **Note:** To enter IPLY on card 3, a material collector with a defined PLY_DATA card image must exist.
TYPE 36	Elastic/Stiffening - Plastic with Failure
TYPE 37	Viscoelastic Ogden Rubber
TYPE 41	Improved Side Impact Barrier TYPEerial **Note:** Full material input and simplified material input are available.

Card	Description
TYPE 42	
TYPE 45	General Non-linear Strain Rate Dependent
TYPE 52	Elastic-Plastic Solid with Fracture Criteria
TYPE 61	Elastic-Plastic 8 Node Thick Shell
TYPE 62	Elastic-Plastic for 8-Node Thick Shell Elements with Total Lagrangian Formulation
TYPE 71	Elastic-Plastic with EWK Damage and Failure
TYPE 99	Null TYPEerial for Solid
TYPE 100	Null TYPEerial
TYPE 101	Elastic
TYPE 102	Elastic Plastic
TYPE 103	Elastic Plastic
TYPE 105	Elastic Plastic with Isotropic Damage
TYPE 106	Elastic Plastic with Anisotropic Damage
TYPE 107	
TYPE 108	Anisotropic Elastic Plastic
TYPE 109	**Note:** TYPE 109 data will not be exported (not valid for PAM 2006 and above).
TYPE 110	Super Elastic
TYPE 115	Elastic Plastic with Gurson Damage
TYPE 116	Elastic Plastic with Isotropic Damage
TYPE 117	**Note:** Type 117 data will not be exported (not valid for Pam 2006 and above).
TYPE 118	Anisotropic Elastic Plastic
TYPE 121	CRASH/FORMING, Non-linear Visco Elastic

Card	Description
TYPE 126	Glass
TYPE 128	Anisotropic Elastic Plastic
TYPE 130	Multilayered Shell Elements **Note:** To specify a ply database, a material collector with the PLY_DATA card image must exist in the database. Ply auxiliary variables default to blank and can be overridden.
TYPE 131	Multilayered Shell Elements **Note:** To specify a ply database, a material collector with the PLY_DATA card image must exist in the database. Ply auxiliary variables default to blank and can be overridden.
TYPE 132	Multilayered Shell Elements **Note:** To specify a ply database, a material collector with the PLY_DATA card image must exist in the database. Ply auxiliary variables default to blank and can be overridden.
TYPE 143	Elastic-Plastic with Elastic Stiffening
TYPE 150	Layered TYPEerials for Membrane Elements
TYPE 151	Fabric Membrane Elements with Non Linear Fibre
TYPE 161	Elastic for 4-Node Thick Shell
TYPE 162	Elastic Plastic with 4-Node Thick Shell
TYPE 171	Elastic Plastic with EWK Damage
TYPE 200	Null TYPERerial
TYPE 201	Elastic **Note:** Card fields vary depending upon the element type selected (beam or bar).
TYPE 202	Elastic-Plastic

Card	Description
	Note: Card fields vary depending upon the element type selected. Post-Yield behavior - defined by Yield Stress list box.
TYPE 203	Nonlinear for BAR **Note:** Number of editable fields depends on NLOAD, NUNLD, and NRELD.
TYPE 204	Nonlinear for BAR/DASHPOT **Note:** Force-deflection curve specification requires existence of curves in the database.
TYPE 205	Nonlinear Tension Only Barn **Note:** NLOAD can be set to 0 or to a curve (right-click field label to reset NLOAD curve selection). Other fields for material type 205 depend on the value of NLOAD.
TYPE 212	Elastic-Plastic for Beam Elements **Note:** Post-Yield behavior - defined by Yield Stress list box.
TYPE 213	Elastic-Plastic for Beam Elements **Note:** Post-Yield behavior - defined by Yield Stress list box. Specification of the cross section description through the list box affects the layout of cards 8 through NIPS 8.
TYPE 214	Global Beam Column **Note:** Curves must exist in the model before specifying curve fields.
TYPE 220	Nonlinear 6-DOF Spring/Dashpot **Note:** Curves must exist in the model before specifying curve fields.

Card	Description
TYPE 221	Spherical Joint Elements **Note:** Curves must exist in the model before specifying curve fields.
TYPE 222	Flexion Torsion Joint Elements **Note:** Curves must exist in the model before specifying curve fields.
TYPE 223	Nonlinear 6-DOF Spring-Beam Elements **Note:** Curves must exist in the model before specifying curve fields.
TYPE 224	6-DOF Penalty Spring Beam Elements **Note:** 6-DOF penalty spring beam elements
TYPE 230	KineTYPEic Joint Elements **Note:** Curves must exist in the model before specifying curve fields.
TYPE 301	SLINK, ELINK elements or TIED interface.
TYPE 302	PLINK elements.
TYPE 303	LLINK elements
TYPE 304	TIED Interfaces
PLYDATA	Only TSAI-WU failure criterion supported

Permas Cards

Card	Description
$COMPRESS	Fluid compressibility
$CONDUCTIVITY	Heat conductivity

Card	Description
$DAMPING	Structural damping
$DENSITY	Material density
$DIELECTRIC	Definition of dielectricity.
$ELASTIC	Linear elastic material data.
$ELCONDUCT	Definition of electric conductivity.
$ENTER MATERIAL	Material input bracket header line.
$FLDENS	Definition of fluid material density.
$FLUID	Opens the bracket for definition of a fluid material.
$GASKET	Definition of material for gaskets.
$GSKLOAD	Definition of the loading behavior for gasket material.
$GSKUNLOAD	Definition of the unloading behavior for gasket material.
$HARDENING	Hardening
$HEATCAP	Heat capacity
$MATERIAL	Definition of homogenous material.
$PERMEABILITY	Definition of magnetic permeability.
$PLASTIC	Plasticity data
$SURFABS	Definition of absorption at the boundary surface of a fluid.
$THERMEXP	Thermal expansion coefficients
$VOLDRAG	Definition of volumetric drag of a fluid.
$YIELD	Yield limit

Radioss Cards

Radioss allows you to program your own materials that can be used in a simulation. Unsupported Radioss materials and user defined Radioss materials are assigned the MAT_UNSUPPORTED card image.

HyperMesh imports unsupported materials with the MAT_UNSUPPORTED card image, and preserves their corresponding IDs and associated components.

In the MAT_UNSUPPORTED card image, all material sub-options, parameters, and data lines are supported as simple text. HyperMesh does not check the validity or syntax of any data in this mode.

You must manually check the validity of the data. No editing, updating, or review of the material data is intended. Also time step calculation and mass calculation are not available for the component that refers to this material.

Card	Description
/ALE/MAT	Describes the ALE material. **Note:** Block Format Keyword
/MAT/LAW12 (3D_COMP)	Describes a solid material using the Tsai-Wu formulation that is usually used to model composites. This material is assumed to be 3D orthotropic-elastic before the Tsai-Wu criterion is reached. The material becomes nonlinear afterwards. The Tsai-Wu criterion can be set dependent on the plastic work and strain rate in each of the orthotropic directions and in shear to model material hardening. Stress based orthotropic criterion for brittle damage and failure is available. This material is a generalization and improvement of `/MAT/LAW14 (COMPSO)`. **Note:** Block Format Keyword
/MAT/B-K-EPS	Describes the boundary conditions material in flow analysis (ALE or EULER). It is based on boundary material `/MAT/LAW11 (BOUND)` activating boundary turbulence modeling and adding 2 input lines for k - ε parameters. It is compatible for 2D and 3D analysis. It is not compatible with Multi-material ALE laws, `LAW37 (BIMAT)` and `/MAT/LAW51 (MULTIMAT)`. **Note:** Block Format Keyword
/MAT/LAW57 (BARLAT3)	Describes plasticity hardening defined by a user function and can be used only with shell elements. This is an elasto-plastic orthotropic law for modeling anisotropic materials in forming processes especially aluminum alloys. This material law must be used with property set type `/PROP/TYPE9 (SH_ORTH)` or `/PROP/TYPE10 (SH_COMP)`.

Card	Description
	Note: Block Format Keyword
/MAT/LAW20 (BIMAT)	ALE multi-material law for 2D analysis. **Note:** Block Format Keyword
/MAT/LAW37 (BIPHAS)	Describes the hydrodynamic bi-material liquid gas material. It is not recommended to use this multi-material laws (LAW37) with Radioss single precision engine. **Note:** Block Format Keyword
/MAT/LAW34 (BOLTZMAN)	Describes the Boltzmann (visco-elastic) material. This law is applicable only for solid elements and can be used to model polymers and elastomers. **Note:** Block Format Keyword
/MAT/LAW11 (BOUND)	Describes the boundary conditions material in flow analysis (ALE or EULER). It is compatible for 2D and 3D analysis. It is not compatible with Multi-Material ALE laws, `LAW37 (BIMAT)` and `/MAT/LAW51 (MULTIMAT)`. In case of turbulence, activate boundary turbulence modeling using `/MAT/B-K-EPS` and input κ - $\dot{\varepsilon}$ boundary conditions. **Note:** Block Format Keyword
/MAT/LAW15 (CHANG)	Models composite shell elements, similar to LAW25. The plastic behavior is based on the Tsai-Wu criteria (`/MAT/LAW25 (COMPSH)` for Tsai-Wu description) and failure is based on the Chang-Chang failure criterion is used. **Note:** Block Format Keyword
/MAT/LAW25 (COMPSH)	Two variations of the same material LAW25 are implemented:Tsai-wu formulation and CRASURV formulation.

Card	Description
	Note: Block Format Keyword
/MAT/LAW14 (COMPSO)	Describes an orthotropic solid material using the Tsai-Wu formulation that is mainly designed to model uni-directional composites. This material is assumed to be 3D orthotropic-elastic before the Tsai-Wu criterion is reached. The material becomes nonlinear afterwards. The nonlinearity in direction 3 is the same as that in direction 2 to represent the behavior of a composite matrix material. The Tsai-Wu criterion can be set dependent on the plastic work and strain rate in each of the orthotropic directions and in shear to model material hardening. Stress based orthotropic criterion for brittle damage and failure is available. `/MAT/LAW12 (3D_COMP)` is an improved version of this material and should be used instead of LAW14. **Note:** Block Format Keyword
/MAT/LAW24 (CONC)	Models brittle elastic-plastic behavior of reinforced concrete. The law assumes that the two failure mechanisms are tensile cracking and compressive crushing of the concrete material. This keyword is compatible only with solid elements. **Note:** Block Format Keyword
/MAT/LAW59 (CONNECT)	Describes the Connection material, which can be used to model spotweld, welding line, glue, or adhesive layers in laminate composite material. Elastic and elastoplastic behavior in normal and shear directions can be defined. The curves that represent plastic behavior can be specified for different strain rates. This material is applicable only to solid hexahedron elements (`/BRICK`) and the material time-step does not depend on element height. **Note:** Block Format Keyword

Card	Description
/MAT/LAW68 (COSSER)	Describes the honeycomb material. **Note:** Block Format Keyword
/MAT/LAW44 (COWPER)	The Cowper-Symonds law models an elasto-plastic material. The basic principle is the same as the standard Johnson-Cook model; the only difference between the two laws lies in the expression for strain rate effect on flow stress. **Note:** Block Format Keyword
/MAT/LAW22 (DAMA)	This law is identical to Johnson-Cook material (`/MAT/LAW2`), except that the material undergoes damage if plastic strains reach a user-defined value (ε_{dam}). This law can be applied to both shell and solid elements. **Note:** Block Format Keyword
/MAT/LAW21 (DPRAG)	Specific interface between a non-deformable master surface and a slave surface designed for stamping. All nodes of the master surface must belong to the rigid body. **Note:** Block Format Keyword
/MAT/LAW10 (DPRAG1)	This law, based on Drücker-Prager yield criteria, is used to model materials with internal friction such as rock-concrete. The plastic behavior of these materials is dependent on the pressure in the material. This law is similar to LAW21 (`/MAT/LAW21 (DRAGP)`); the only difference being that in this law, the pressure is defined as a cubic function of volumetric strain, and hence requires the input of certain coefficients. This law is compatible only with solid elements. **Note:** Block Format Keyword

Card	Description
/MAT/LAW1 (ELAST)	Defines an isotropic, linear elastic material using Hooke's law. This law represents a linear relationship between stress and strain. It is available for truss, beam (type 3 only), shell and solid elements. **Note:** Block Format Keyword
/MAT/LAW65 (ELASTOMER)	Describes non-linear elastoplastic material with strain rate dependent loading and unloading behavior. **Note:** Block Format Keyword
/MAT/LAW58 (FABR_A)	Describes a hyperelastic anisotropic fabric material. It uses an anisotropic coordinate system with anisotropy angle, following element deformation. The material formulation provides coupling between warp and weft directions in order to reproduce physical interaction between fibers. The shear degree of freedom is fully decoupled from the translational degrees of freedom. Optionally, nonlinear stress-strain curves for loading and unloading can be specified for warp, weft directions and in shear. **Note:** Block Format Keyword
/MAT/LAW19 (FABRI)	Defines an elastic orthotropic material and is available only for shell elements. It is used to model airbag fabrics. **Note:** Block Format Keyword
/MAT/LAW33 (FOAM_PLAS)	Models a visco-elastic plastic foam material. This law is applicable only for solid elements and is typically used to model low density, closed cell polyurethane foams such as impact limiters. **Note:** Block Format Keyword

Card	Description
/MAT/LAW70 (FOAM_TAB)	Describes the visco-elastic foam tabulated material. This material law can be used only with solid elements. **Note:** Block Format Keyword
/MAT/LAW35 (FOAM_VISC)	Describes a visco-elastic foam material using Generalized Maxwell-Kelvin-Voigt model where viscosity is based on Navier equations. This law is applicable only for shell and solid elements and can be used for open cell foams, polymers, elastomers, seat cushions and dummy paddings. **Note:** Block Format Keyword
/MAT/GAS	Describes the gas molecular weight and specific heat coefficients. **Note:** Block Format Keyword
/MAT/LAW16 (GRAY)	This material law is based on Gray EOS and Johnson-Cook yield criteria. **Note:** Block Format Keyword
/MAT/LAW52 (GURSON)	This law is based on the Gurson constitutive law, which is used to model visco-elastic-plastic strain rate dependent porous metals. **Note:** Block Format Keyword
/MAT/LAW63 (HANSEL)	This law describes the trip steel plastic material. This material law can be used only with shell elements. **Note:** Block Format Keyword
/MAT/LAW32 (HILL)	Describes the Hill orthotropic plastic material. It is applicable only to shell elements. This law differs from `LAW43 (HILL_TAB)` only in the input of yield stress.

Card	Description
	Note: Block Format Keyword
/MAT/LAW72 (HILL_MMC)	Describes the anisotropic hill material with a modified Mohr fracture criteria. This law is available for shell and solid. **Note:** Block Format Keyword
/MAT/LAW43 (HILL_TAB)	Describes the Hill orthotropic material and is applicable only to shell elements. This law differs from `LAW32 (HILL)` only in the input of yield stress (here it is defined by a user function). **Note:** Block Format Keyword
/MAT/LAW73	Describes the Thermal Hill orthotropic material and is applicable only to shell elements. This law differs from `/MAT/LAW43 (HILL_TAB)` by the fact that yield stress not only depends on strain rate and plastic strain, but also on temperature (it is defined by a user table). **Note:** Block Format Keyword
/MAT/LAW28 (HONEYCOMB)	Describes a three dimensional nonlinear elasto-plastic material, usually used to model honeycomb or foam material. Nonlinear elasto-plastic behavior can be specified for each othrotropic direction and shear as function of strain or volumetric strain. All degrees of freedom are uncoupled and the material is fully compressible. Tension and shear strain based failure criteria can be specified. **Note:** Block Format Keyword
/MAT/LAW4 (HYD_JCOOK)	Represents an isotropic elasto-plastic material using the Johnson-Cook material model. This model expresses material stress as a function of strain, strain rate and temperature. This material may account for the nonlinear dependence between pressure and volumetric strain when

Card	Description
	corresponding equation of state is specified. A built-in failure criterion based on the maximum plastic strain is available. This material law is compatible with solid elements only. **Note:** Block Format Keyword
/MAT/LAW3 (HYDPLA)	Represents an isotropic elasto-plastic material using the Johnson-Cook material model. This model expresses material stress as a function of strain and may account for the nonlinear dependence between pressure and volumetric strain when corresponding equation of state is specified. A built-in failure criterion based on the maximum plastic strain is available. This material law is compatible with solid elements only. **Note:** Block Format Keyword
/MAT/LAW6 (HYDRO)	Describes the hydrodynamic viscous fluid material using a polynomial EOS. **Note:** Block Format Keyword
/MAT/LAW79 (JOHN_HOLM)	Describes the behavior of brittle materials, such as ceramics and glass. The implementation is the second Johnson Holmquist model: JH-2. **Note:** Block Format Keyword
/MAT/LAW5 (JWL)	Describes the Jones-Wilkins-Lee EOS for detonation products of high explosives. **Note:** Block Format Keyword
/MAT/LAW6 (K-EPS)	Describes the k - $\dot{\varepsilon}$ turbulence viscous material for fluid. **Note:** Block Format Keyword

Card	Description
/MAT/LAW40 (KELVINMAX)	Describes the generalized Maxwell-Kelvin material. This law can only be used with solid elements. **Note:** Block Format Keyword
/MAT/LAW66	Models an isotropic tension-compression elasto-plastic material law using user-defined functions for the work-hardening portion of the stress-strain (plastic strain vs. stress). This law can be defined for compression and tension. **Note:** Block Format Keyword
/MAT/LAW69	This law (extension of `/MAT/LAW42 (OGDEN)`) defines a hyperelastic and incompressible material specified using the Ogden, Mooney-Rivlin material models. It is generally used to model incompressible rubbers, polymers, foams, and elastomers. Material parameters are computed from engineering stress-strain curve from uniaxial tension and compression tests. It is used with shell and solid elements. **Note:** Block Format Keyword
/MAT/LAW74	Describes the Thermal Hill orthotropic 3D material and is applicable only to solid elements. The yield stress may depend on strain rate, or on both strain rate and temperature. **Note:** Block Format Keyword
/MAT/LAW77	This open cell foam material law is a generalization of `LAW70`. It accounts for a non-viscous compressible ideal gas flow inside of the foam and its interaction with the foam structure. ALE simulation of the gas flow and Lagrangian simulation of the foam deformation is performed on the same elements system. Interaction between the gas flow and the structure is through Darcy law and direct application of the gas pressure to the structure.

Card	Description
	Note: Block Format Keyword
/MAT/LAW78	This law is the Yoshida-Uemori model for describing the large-strain cyclic plasticity of metals. The law is based on the framework of two surfaces theory: the yielding surface and the bounding surface. During the plastic deformation, a yield surface will move within the bounding surface and will never change its size, and the bounding surface can change both in size and location. The plastic-strain dependency of the Young's modulus and the work-hardening stagnation effect are also taken into account. Concerning SPH, it is compatible with solid only, this can be verified with the /SPH/WavesCompression test. The solid version is only isotropic. The shell version is anisotropic based on Hill criterion. **Note:** Block Format Keyword
/MAT/LAW80	Models the ultra-high strength steel behavior at high temperatures and the phase transformation phenomena from austenite to ferrite, pearlite, bainite and martensite during cooling. **Note:** Block Format Keyword
/MAT/LAW81	This law is based on Drücker-Prager yield criteria with cap. It has a strain-hardening cap model based on the principles of Foster. Plasticity has an isotropic hardening. Failure surface is limited to the standard linear Drücker-Prager relation, with symmetry around the pressure axis. This law is LAG, ALE and EULER compatible. **Note:** Block Format Keyword
/MAT/LAW82	Defines the Ogden material. This law is compatible with solid and shell elements. In general it is used to model polymers and elastomers.

Card	Description
	Note: Block Format Keyword
/MAT/LAW83	Describes the Connection material, which can be used to model spotweld, welding line, glue, or adhesive layers in laminate composite material. Elastic and elastoplastic behavior can be defined. The plastic behavior of the material can be coupled in normal and shear directions for corresponding strain-rates. This material is applicable only to solid hexahedron elements (`/BRICK`) and the material time-step does not depend on element height. **Note:** Block Format Keyword
/MAT/LAW41 (LEE-TARVER)	Describes detonation products using an ignition and growth model of a reactive material. The Lee-Tarver model is based on the assumption that ignition starts at local hot spots in the passage of shock front and grows outward from these sites. The reaction rate is controlled by the pressure and the surface area as in a deflagration process. **Note:** Block Format Keyword
/MAT/LAW46 (LES_FLUID)	Describes the viscous fluid material with sub-grid scale viscosity. **Note:** Block Format Keyword
/MAT/LAW51 (MULTIMAT)	Up to four material laws can be defined: elasto-plastic solid, liquid, gas and detonation products. The material boundaries inside an element are not explicitly defined, but an anti-diffusive technique is used to avoid expansion of transition zone (`/UPWIND` in Radioss Starter Input). **Note:** Block Format Keyword
/MAT/LAW42 (OGDEN)	Defines a hyperelastic, viscous, and incompressible material specified using the Ogden, Mooney-Rivlin

Card	Description
	material models. This law is generally used to model incompressible rubbers, polymers, foams, and elastomers. This material can be used with shell and solid elements. **Note:** Block Format Keyword
/MAT/LAW27 (PLAS_BRIT)	Combines an isotropic elasto-plastic Johnson-Cook material model with an orthotropic brittle failure model. Material damage is accounted for prior to failure. Failure and damage occur only in tension. This law is applicable only for shells. **Note:** Block Format Keyword
/MAT/LAW23 (PLAS_DAMA)	Models an isotropic elastic plastic material and combines Johnson-Cook material model with a generalized damage model. The law is applicable only for solid elements. **Note:** Block Format Keyword
/MAT/LAW2 (PLAS_JOHNS)	Represents an isotropic elasto-plastic material using the Johnson-Cook material model. This model expresses material stress as a function of strain, strain rate and temperature. A built-in failure criterion based on the maximum plastic strain is available. **Note:** Block Format Keyword
/MAT/LAW60 (PLAS_T3)	Models an isotropic elasto-plastic material using user-defined functions for the work-hardening portion of the stress-strain curve (that is, plastic strain vs. stress) for different strain rates. It is similar to LAW36, except yield stress is a nonlinear interpolation from the functions. **Note:** Block Format Keyword
/MAT/LAW36 (PLAS_TAB)	Models an isotropic elasto-plastic material using user-defined functions for the work-hardening

Card	Description
	portion of the stress-strain curve (for example, plastic strain vs. stress) for different strain rates. **Note:** Block Format Keyword
/MAT/LAW75 (POROUS)	Describes the P-α porous material model. This material describes ductile Porous material with Herrmann model. It only works with 8-node brick element and is not compatible with ALE. **Note:** Block Format Keyword
/MAT/LAW54 (PREDIT)	Describes the predit material. This material law is only used with `/PROP/TYPE36 (PREDIT)`. **Note:** Block Format Keyword
/MAT/LAW13 (RIGID)	Models part(s) as rigid bodies. **Note:** Block Format Keyword
/MAT/LAW76 (SAMP)	Describes a semi-analytical elasto-plastic material using user-defined functions for the work-hardening portion for tension, compression and shear (stress as function of strain). **Note:** Block Format Keyword
/MAT/LAW26 (SESAM)	This ALE material law describes a SESAME tabular EOS, used with a Johnson-Cook yield criterion. SESAME EOS covers a wide range of phases including solids, fluids and high temperature/high density plasmas, and the well-known transitions between these various phases. It requires SESAME tables, which were developed at Los Alamos National Laboratory in USA. **Note:** Block Format Keyword
/MAT/LAW49 (STEINB)	Defines an elastic plastic material with thermal softening.

Card	Description
	Note: Block Format Keyword
/MAT/LAW18 (THERM)	Describes thermal material. **Note:** Block Format Keyword
/MAT/LAW53 (TSAI_TAB)	Describes the law that is a uni-directional orthotropic elasto-plastic law and is only used with solid elements. **Note:** Block Format Keyword
/MAT/LAW64 (UGINE_ALZ)	Describes the Ugine & Alz trip steel material. This material law can be used only with shell elements. **Note:** Block Format Keyword
/MAT/USERij	Describes the user material. **Note:** Block Format Keyword
/MAT/LAW50 (VISC_HONEY)	Describes the honeycomb material with strain rate dependency (based on material LAW28 + strain rate dependency). **Note:** Block Format Keyword
/MAT/LAW62 (VISC_HYP)	Describes the hyper visco-elastic material. This law is compatible with solid and shell elements. In general it is used to model polymers and elastomers. **Note:** Block Format Keyword
/MAT/LAW38 (VISC_TAB)	Describes the visco-elastic foam tabulated material and can only be used with solid elements. **Note:** Block Format Keyword

Card	Description
/MAT/LAW0 (VOID)	Defines elements to act as a void, or an empty space. **Note:** Block Format Keyword
/MAT/LAW48 (ZHAO)	Describes the Zhao material law used to model an elasto-plastic strain rate dependent materials. The law is applicable only for solids and shells. The global plasticity option for shells (N=0 in shell property keyword) is not available in the actual version. **Note:** Block Format Keyword
/VISC/PRONY	This is an isotropic visco-elastic Maxwell model that can be used to add visco-elasticity to certain shell and solid element material models. The visco-elasticity is input using a Prony series. **Note:** Block Format Keyword

Samcef Cards

Card	Description
.MAT, ANISOTROPIC	Define the properties of one or several materials.
.MAT, ISOTROPIC	Define the properties of one or several materials.
.MAT, ORTHOTROPIC	Define the properties of one or several materials.

Mechanisms

Mechanism entities are the root of the hierarchy in the Mechanism Browser.

Output Blocks

Output block entities define and store solver output requests.

Output blocks do not have a display state.

Abaqus Cards

Output blocks are added to load steps (*STEP). Output requests organized into the output blocks are written out within the corresponding step definition in the Abaqus input deck. Output blocks must be added to the corresponding load steps for them to be written out to the Abaqus deck. All types of output requests in an output block are defined from the card image.

Card	Description
*CONTACT FILE	Define results file requests for contact variables.
*CONTACT OUTPUT	Specify contact variables to be written to the output database.
*CONTACT PRINT	Define print requests for contact variables.
*EL FILE	Define results file requests for element variables.
*EL PRINT	Define data file requests for element variables.
*ELEMENT OUTPUT	Define output database requests for element variables.
*ENERGY FILE	Write energy output to the results file.
*ENERGY OUTPUT	Define output database requests for whole model or element set energy data.
*ENERGY PRINT	Print a summary of the total energies
*INCREMENTATION OUTPUT	Define output database requests for time incrementation data. **Note:** Explicit template only. History data. Only for *OUTPUT, HISTORY.
*INTEGRATED OUTPUT	Specify variables integrated over a surface to be written to the output database.

Card	Description
	Note: Only for *OUTPUT, HISTORY
*MODAL OUTPUT	Write generalized coordinate (modal amplitude) data to the output database during a mode-based dynamic or complex eigenvalue extraction procedure. **Note:** Standard 2D and 3D templates only. History data. Only for *OUTPUT, HISTORY.
*NODE FILE	Define results file requests for nodal data
*NODE OUTPUT	Define output database requests for nodal data
*NODE PRINT	Define print requests for nodal variables
*OUTPUT	Define output requests to the output database

LS-DYNA Cards

Data on LS-DYNA output blocks cannot be edited.

Card	Description
*DATABASE_HISTORY_BE	Describe time history for beam elements.
*DATABASE_HISTORY_BE	Describe time history for beam element sets.
*DATABASE_HISTORY_DI	
*DATABASE_HISTORY_DI	
*DATABASE_HISTORY_NO	Describe time history for nodes.
*DATABASE_HISTORY_ NODE_LOCAL(ID)	Describe time history for nodes.
*DATABASE_HISTORY_ NODE_SET	Describe time history for node sets.
*DATABASE_HISTORY_NO	Describe time history for node sets.
*DATABASE_HISTORY_NO	Describe time history for node sets.
*DATABASE_HISTORY_SE	Define time history on seat belt element.

Card	Description
*DATABASE_HISTORY_SH	Describe time history for shell elements.
*DATABASE_HISTORY_SH	Describe time history for shell elements.
*DATABASE_HISTORY_SH	Describe time history for shell element sets.
*DATABASE_HISTORY_SO	Describe time history for solid elements.
*DATABASE_HISTORY_SO	Describe time history for solid elements.
*DATABASE_HISTORY_SO	Describe time history for solid element sets.
*DATABASE_HISTORY_SP	Describe time history for SPH elements.
*DATABASE_HISTORY_TS	Describe time history for TSHELL elements.
*DATABASE_HISTORY_TS	Describe time history for TSHELL elements.
*DATABASE_HISTORY_TS	Describe time history for TSHELL element sets.

OptiStruct Cards

Card	Description
XHIST	Defines the time history output request for geometric nonlinear analysis. **Note:** Bulk Data Entry

PAM-CRASH Cards

Keyword selection for output blocks is supported as GES selection.

The appropriate keyword is output when selecting elements for time-history output.

Card	Description
PLANE	
SECFO /	Section definition for force output. **Note:** Slave nodes and master elements define the cross section. To define non-shell elements, create an entity set first. The master definition must be by sets.

Card	Description
SECTION	
SENPT /	Sensor point output definition.
SENPTG /	Sensor point output definition.
SUPPORT	
THELE /	Element time history. **Note:** Uses element output block.
THLOC /	Local frame definition for node output. **Note:**
THNOD /	Nodal time history. **Note:** Uses output block including nodes.
VOLFRAC	

Permas Cards

Card	Description
$FREQUENCY	Definition of frequency list for frequency response analysis. **Note:** The FREQUENCY option is available in the results bracket.
$NLRESULTS	Output request for nonlinear static analysis.
$TIMESTEPS	Specify specific time steps when results are written.

Radioss Cards

Card	Description
/TH/ACCEL/	Describes the time history for accelerometers.

Card	Description
	Note: Block Format Keyword Options to assign ATH...ITH
/TH/BEAM/	Describes the time history for beams. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/BRIC/	Describes the time history for bricks. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/CYL_JO/	Describes the time history for cylindrical joints. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/FRAME/	Describes the time history for frames. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/GAUGE	Describes the time history for gauges. **Note:** Block Format Keyword
/TH/INTER/	Describes the time history for interfaces. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/MONVOL/	Describes the time history for monitored volume. **Note:** Block Format Keyword Options to assign ATH...ITH

Card	Description
/TH/NODE/	Describes the time history for nodes. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/QUAD/	Describes the time history for a quad. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/PART/	Describes the time history for parts. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/RBODY/	Describes the time history for rigid bodies. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/RWALL/	Describes the time history for rigid wall. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/SECTIO/	Describes the time history for sections. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/SH3N/	Describes the time history output for 3-noded shell elements. Output parameters are components of stress and strain tensor expressed in different coordinate systems, plastic strains, strain rate, internal energy, element thickness, element deletion flag, and so on. **Note:** Block Format Keyword Options to assign ATH...ITH

Card	Description
/TH/SHEL/	Describes the time history output for 4-node shell elements. Output parameters are components of stress and strain tensor expressed in different coordinate systems, plastic strains, strain rate, internal energy, element thickness, element deletion flag, and so on. **Note:** Options to assign ATH...ITH
/TH/SPHCEL/	Describes the SPH cell output to time history. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/SPRING/	Describes the time history for springs. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/SUBS/	Describes the time history for subsets. **Note:** Block Format Keyword Options to assign ATH...ITH
/TH/TRUSS/	Describes the time history for trusses. **Note:** Block Format Keyword Options to assign ATH...ITH

Parameters

Parameter entities parameterize FE entity and geometric entity attributes.

Abaqus Cards

To properly read and import parameters, independent parameters must be defined before dependent parameters. If parameters are not properly defined, Abaqus will abort the evaluation, and an error message will display.

In the image below, parameter2 must be defined ahead of parameter1.

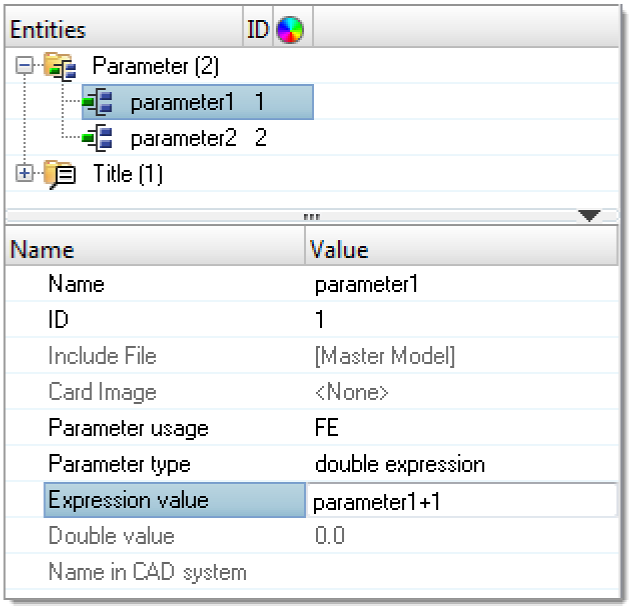


Figure 137:

Parameters are evaluated and their values are determined by the solver. The final value, after the evaluation of the parameter, will be written in the input deck. The order in which the parameters are written determines the value assigned to the parameter. The value of Parameter1, regardless of where Parameter1 is written in the input deck, will be 2.

```
*PARAMETER
parameter1 = 1.0
**HMNAME PROPERTY          1 property1
*SHELL SECTION, ELSET=property1, MATERIAL=Steel
<parameter1>,           5
*PARAMETER
Parameter1 = 2.0
```

During export, parameters are exported in the order in which they were created. Parameter information is written at the top of the input deck. If a parameter is in an Include file, then it will be written at the top of the Include file.

The keywords, *Parameter Dependence and *Parameter Table are not yet supported.

Card	Description
*PARAMETER	Defines numerical values through algebraic expressions of parameter names referenced throughout the input file.

Card	Description
	Note: Supported: • Parameters can be created, imported, and exported. • Edit parameters in the Property Editor. • Parameter expressions are evaluated on the fly. • Parameter expressions can reference another parameter. • If you edit the value of a parameter, the other parameters that references it will be updated accordingly. • Edit parameterized variables in the Property Editor. • You can create and assign parameters from the fields of an entity in Property Editor. • Load magnitude, element node ID, Node coordinate, and so on can all be parameterized. Limitations: • Support is limited to all properties and materials. • Parameterization of entity IDs is not supported. • Unsupported parameterized entities and fields are replaced with corresponding parameter values during import, to avoid the loss of data. • In macros and browsers, if any parameterized variable is changed, you will receive a warning message in the message bar and the value will not be updated. • The creation of duplicate parameters is not supported. During import, duplicate parameters and their corresponding expression will be replaced with the resolved value in the input deck.

LS-DYNA Cards

Card	Description
*PARAMETER *PARAMETER_LOCAL	Defines numerical values of parameter names referenced throughout the input file.

Card	Description
	Note: Supported: • Parameters can be created, imported, and exported. • Edit parameter expressions in the Property Editor. • Parameter expressions are evaluated on the fly. • A parameter expression can reference another parameter. • If you edit the value of a parameter, the other parameters that references it will be updated accordingly. • Edit parameterized values in the Property Editor. • Same parameters can be used for negation by prefixing negative signs "-". • You can create and assign parameters from the fields of an entity in Property Editor. • Parameters can be referenced in angular brackets, along with the usual reference using "&". • When more than one *PARAMETER with the same name exist in a deck, the first one will be retained. • Parameters with same names are maintained when *PARAMTER_LOCAL is used and they are specified in different include files. • Free format is supported. Limitations: • Support is limited to all FE entities except elements and nodes. • Parameterization of entity ID's is only supported for Components, Properties, Materials and Curve. • Parameterization of entity ID's is not supported. • Unsupported parameterized entities and fields are replaced with corresponding parameter values during import, to avoid the loss of data. • In macros and browsers, if any parameterized variable is changed a warning message will appear in the message bar and the value will not be updated.
*PARAMETER_EXPRE *PARAMETER_EXPRE	Defines numerical values through algebraic expressions of parameter names referenced throughout the input file.

Card	Description
	Note: Supported: • Parameters can be created, imported, and exported. • You can edit parameters and parameter expressions in the Property Editor. • Parameter expressions are evaluated on the fly. • Parameter expression can reference another parameter. • If you edit the value of a parameter, the other parameters that references it will be updated accordingly. • Same parameters can be used for negation by prefixing negative signs "-". • Edit parameterized variables in the Property Editor. • Parameters can be referenced in angular brackets, along with the usual reference using "&". • When more than one *PARAMETER_EXPRESSION with the same name exists in a deck, the first one will be retained. • Parameters with the same names are maintained when *PARAMTER_EXPRESSION_LOCAL is used and they are specified in different include files. • Free format is supported. Limitations: • Support is limited to all FE entities except elements and nodes. • Parameterization of entity ID's is only supported for Components, Properties, Materials and Curves. • Parameterization of entity ID's is not supported. • Unsupported parameterized entities and fields are replaced with corresponding parameter values during import, to avoid the loss of data. • In Macros and Browser, if any parameterized variable is changed a warning message will appear in the message bar and the value will not be updated.

Nastran Cards

Card	Description
%DEFREPSYM	Defines default parameter values.

Card	Description
	Note: Can be created, imported, and exported. Edit in the Property Editor.
%DEFREPWIDTH	Defines default width information. **Note:** Automatically created during export when a parameter is created in HyperMesh.
%SETREPSYM	Defines default values. **Note:** Cannot be created in HyperMesh, only imported and exported.
%UNDEFREPSYM	Clears or unsets default parameter values. **Note:** Cannot be created in HyperMesh, only imported and exported.
%UNSETREPSYM	Clears or unsets parameter values. **Note:** Cannot be created in HyperMesh, only imported and exported.

Materials that can be parameterized in the NastranMSC user profile.

- MAT1
- MAT10
- MAT2
- MAT4
- MAT5
- MAT8
- MAT9
- MATEP
- MATG
- MATHE
- MATHP

Properties that can be parameterized in the NastranMSC user profile.

- PAABSF

- PACBAR
- PACINF
- PAERO2
- PBEND
- PBUSH
- PBUSH1D
- PCOMP
- PCOMPG
- PCOMPP
- PCONVM
- PWELD
- PDAMP
- PELAS
- PELAST
- PFAST
- PGAP
- PHBDY
- PMASS
- PROD
- PSEAM
- PSHEAR
- PSHELL
- PSHELL1
- PTUBE
- PVISC

OptiStruct Cards

- The parameter entity is supported in the OptiStruct interface, however support is limited to parameters that are created in HyperMesh and associated with parameters in HyperStudy.
- Scope is limited to material and property entity types. Also, within those entity types, parameterization is restricted to template attributes of type double.
- Parameterized attributes are unparameterized with their original value during export.
- Parameters cannot be exported or imported.

Radioss Cards

Card	Description
/GLOBAL/INT_EXPR/	Defines numerical values through algebraic expressions of parameter names referenced throughout the input file.

<table>
<tr><th>Card</th><th>Description</th></tr>
<tr><td></td><td>Note: Supported:
Parameters can be created, imported, and exported.
Edit parameter expressions in the =Property Editor.
Parameter expressions are evaluated on the fly.
Parameter expressions can reference another parameter.
If you edit the value of a parameter, the other parameters that references it will be updated accordingly.
Same parameters can be used for negation by prefixing negative signs “-“.
Edit parameterized variables in the Property Editor.
Limitations:
Support is limited to all FE entities except elements and nodes.
Parameterization of entity IDs is not supported.
Unsupported parameterized entities and fields are replaced with corresponding parameter values during import, to avoid the loss of data.
In macros and browsers, if any parameterized variable is changed, you will receive a warning message in the message bar and the value will not be updated.</td></tr>
<tr><td>/GLOBAL/INTEGER/</td><td>Defines numerical values of parameter names referenced throughout the input file.</td></tr>
</table>

Card	Description
	Note: Supported: • Parameters can be imported, exported, and created. • Edit parameter expressions in the Property Editor. • Create and assign parameters on the fly in Property Editor. • Same parameters can be used for negation by prefixing negative signs "-". • Edit parameterized variables in the Property Editor. Limitations: • Support is limited to all FE entities except elements and nodes. • Parameterization of entity IDs is not supported. • Unsupported parameterized entities and fields are replaced with corresponding parameter values during import, to avoid the loss of data. • In macros and browser, if any parameterized variable is changed, you will receive a warning message in the message bar and the value will not be updated.
/GLOBAL/REAL/	Defines numerical values of parameter names referenced throughout the input file.

Card	Description
	Note: Supported: • Parameters can be imported, exported, and created. • Edit parameter expressions in the Property Editor. • Create and assign parameters on the fly in Property Editor. • Same parameters can be used for negation by prefixing negative signs "-". • Edit parameterized variables in the Property Editor. Limitations: • Support is limited to all FE entities except elements and nodes. • Parameterization of entity IDs is not supported. • Unsupported parameterized entities and fields are replaced with corresponding parameter values during import, to avoid the loss of data. • In macros and browser, if any parameterized variable is changed, you will receive a warning message in the message bar and the value will not be updated.
/GLOBAL/REAL_EXPR/	Defines numerical values through algebraic expressions of parameter names referenced throughout the input file.

Card	Description
	Note: Supported: • Parameters can be imported, exported, and created. • Edit parameter expressions in the Property Editor. • Parameter expressions are evaluated on the fly. • Parameter expressions can reference another parameter. • If you edit the value of a parameter, the other parameters that references it will be updated accordingly. • Same parameters can be used for negation by prefixing negative signs "-". • Edit parameterized variables in the Property Editor. Limitations: • Support is limited to all FE entities except elements and nodes. • Parameterization of entity IDs is not supported. • Unsupported parameterized entities and fields are replaced with corresponding parameter values during import, to avoid the loss of data. • In macros and browser, if any parameterized variable is changed, you will receive a warning message in the message bar and the value will not be updated.

Samcef Cards

- The parameter entity is supported in the Samcef interface, however these parameters cannot be directly exported in HyperStudy.
- Scope is limited to material and property entity types.
- Parameters are exported as the macro command /ABRE in the deck.
- Parameters can be exported or imported.

Plies

Ply entities define a FEA ply which is the FEA correlation to a physical ply.

Physical plies manufacture laminates which make up composite structures. A physical ply has attributes of material, shape (area), thickness, and fiber orientation; where its shape is any complex flat pattern that can be cut from a roll of material. Similarly, a FEA ply is composed of the same data attributes as a physical ply (material, shape/area, thickness, and fiber orientation). The shape of a FEA ply can either be defined by closed lines or approximated from the elements which most closely represent its actual complex shape. In the case where plies are defined on lines, perform a realization to convert this information into a definition by elements. Ply data defined on lines is imported from Catia Composites Parts Design (CPD) data.

Plies defined by elements

The shape (area) of an FEA ply is defined by selecting elements which most closely represent the complex shape of a physical ply. In Figure 138, an elliptical physical ply shape is defined by the brown line. The corresponding FEA ply shape is defined by the gray shaded elements of the associated FEA mesh. Typically, if an element's centroid exists within the bounds of the physical ply shape, that element is considered part of the FEA ply shape.

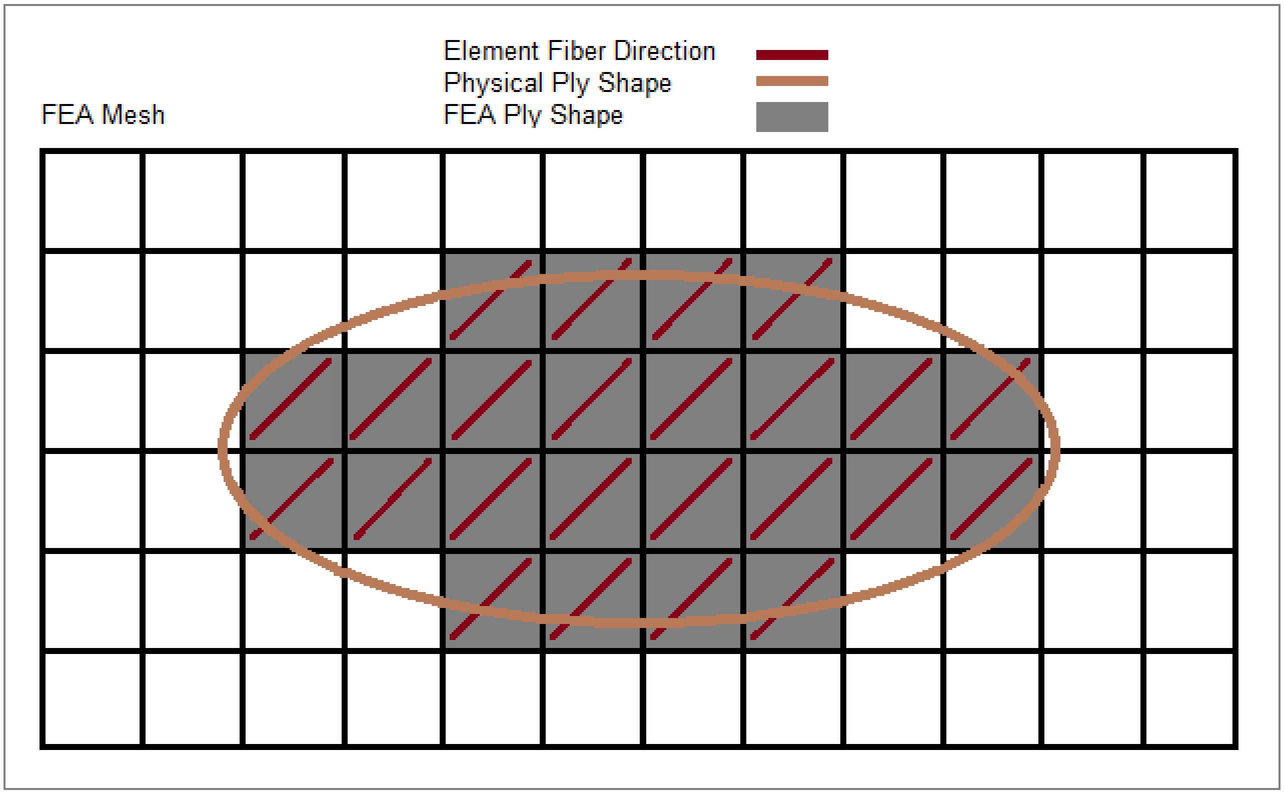


Figure 138:

Plies defined by lines

Plies can be defined by selecting lines which build a closed area. If CPD data is imported from Catia files, plies are defined on lines.

Once a mesh is available, plies defined by lines can be converted into plies defined by elements by performing a realization. Realization/conversion methods include:

Project normal to target mesh

If the element centroid projected along its normal lies within the geometrical ply definition, it is associated with this ply.

Normal by ply contour

Projection along a normal on a surface derived from the lines of a ply contour.

Specified direction

Manually define a projection direction.

The ply thickness is typically defined as the final cured thickness of a single ply of material. In addition, the ply can be made of any material: isotropic, orthotropic, anisotropic, or any other material law.

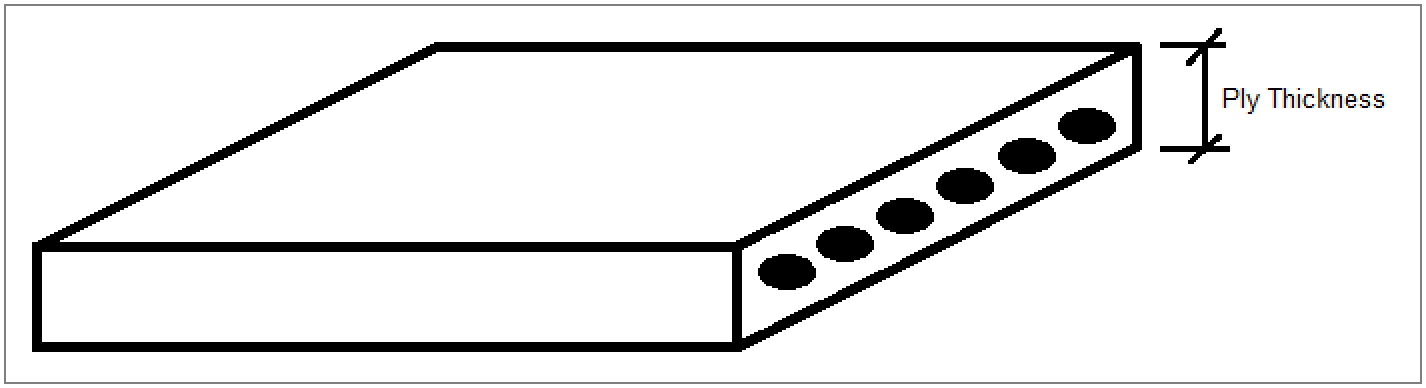


Figure 139:

The fiber orientation of a ply defines the direction fibers lay within that ply. The ply fiber orientation is typically an integer value between -90 and 90. The fiber orientation of a ply is always defined relative to each elements material direction using right hand rule around the elements normal, or thru-thickness direction, to define positive angles. Even though a ply's fiber orientation is a constant integer, element material directions can vary from element to element, and this allows varying fiber directions within a ply to be modeled. Element material directions are defined differently from solver interface to solver interface.

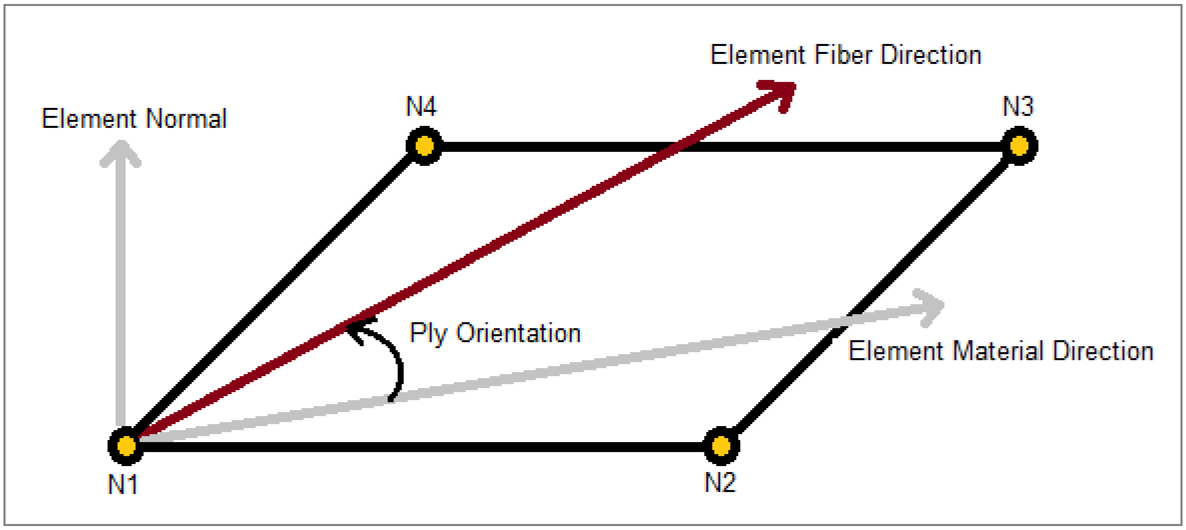


Figure 140:

Once all of the plies which make up a composite structure are defined, just as in the actual hand-layup manufacturing process, plies are stacked in a specific given order within the laminate entity to define a laminate of the structure. It is possible to stack plies, whether they are defined based on geometry or elements.

Turn the display of plies on/off or change the way plies appear in the modeling window with the Ply layers display setting, which can be accessed from the Mesh Display settings on the View Controls toolbar.

Supported Solver Cards

Solver cards supported for plies.

Abaqus Cards

The Abaqus solver has no direct match for a ply, therefore the ply entity does not have a card image in the Abaqus solver interface. Ply entity names will end up in the ply name of the composite properties SHELL SECTION and SHELL GENERAL SECTION, if used in a laminate entity and realized into a zone based model.

Nastran Cards

The Nastran solver has no direct match for a ply, therefore the ply entity does not have a card image in the Nastran solver interface. Ply entity IDs will end up as global ply ID in the PPOMG property card, if used in a laminate entity and realized into a zone based model.

OptiStruct Cards

The OptiStruct PLY card is represented as a ply entity. Plies can be created from the selection of individual elements or from predefined element sets which define the ply shape.

Card	Description
PLY	Defines the properties of a ply used in ply-based composite definition.

Card	Description
	Note: Can only be created and edited in the Ply Editor from the Model Browser.

Radioss Cards

The property card /PROP/PLY (TYPE19) and keyword /PLY are represented as a ply entity.

Samcef Cards

The Samcef .PLI card is represented as a ply entity.

Create and Realize Plies

Overview of how to create and realize plies.

Radioss

Create Plies

1. In the Model Browser, right-click and select **Create** > **Ply** from the context menu.
2. In the **Create Ply** dialog, define attributes accordingly.

Option	Description
Name	Ply name. This will define the prop_title attribute of /PROP/TYPE19.
Same as	Duplicate an existing ply.
Card Image	Keyword used for the ply creation. /PROP/PLY if P19_PLY card image is used, or /PLY if PLY card image is used.
Color	Color of ply.
Dummy ply	Create simple ply without a card image.
Material Type	Type of material assigned to ply.
Material	Material assigned to ply. This will define the mat_ID attribute in /PROP/TYPE19.

Option	Description
Thickness	Ply thickness. This will define the t attribute in /PROP/TYPE19
System	Reference system used for the angle calculation of the ply. The system is not linked to an attribute of the /PROP/TYPE19, but can be defined by importing external composite data like FiberSim information. This information will be taken into account during the realization of the ply.
Orientation	Fiber angle on the ply. This will define the delta_phi attribute in /PROP/TYPE19.
Integration points	Number of integration point through the ply thickness. This will define the Npt_ply attribute in /PROP/TYPE19.
Drape table	Drape table to link to the ply. The drape table will not linked to an attribute of the /PROP/TYPE19, but can be defined by importing external composite data like FiberSim information. If specified, a realization of the ply is needed to obtain the initial fiber angle defined with the Radioss cards /INISHE/ORTH_LOC and /INSH3N/ORTH_LOC.
Shape	Shape of the ply. The attributes grsh4n_ID and grsh3n_ID of /PROP/TYPE19 are automatically generated. If the shape is defined by a line or surface, which may be the case when importing CAD/CPD data, a realization of the ply is needed to generate the FE element sets defining the FE shape.
Base Surfaces	Base surfaces to be used during ply realization. This is useful when the projection of the shape is not found on a unique surface.

3. Click **Create**.

You can edit a ply by right-clicking on it in the Model Browser and selecting **Edit** from the context menu.

Realize Plies

Realize plies when ply definitions need to be imported from external composite data, such as FiberSim or CAD/CPD.

Before you realize plies, make sure the Radioss composite property card /PROP/TYPE17 or /PROP/TYPE51 is assigned to the composite components on which the realization should be performed.

During the realization process, the shape of the ply will be projected on the selected FE mesh and, if drape information is available, the corresponding Radioss cards /INISHE/ORTH_LOC, /INSH3N/ORTH_LOC, or /DRAPE will be generated.

1. In the Model Browser, right-click on the Plies folder and select **Realize** from the context menu.
2. In the **Ply Realization** dialog, define realization settings accordingly.

   > **Note:** For each case (FiberSim or CAD/CPD), a dedicated projection method is proposed in the **Projection options** field.

3. Click **Realize**.

Plots

Plot entities associate and organize curve entities within a xy plot window.

Positions

Position entities allow you to apply a Transformation sequence on set of nodes or SolverSubmodels.

Abaqus Cards

Card	Description
*NMAP	Maps nodes from one coordinate system to another, and rotates, translates, or scales the nodal coordinates.

Card	Description
	Note: A *NMAP keyword is generated for each Transformation listed in the given Position entity.

Radioss

In the Radioss solver profile, there is no solver keywords mapped in Position entity. The Position entity is used to manage the Transformations applied on a set of nodes or a Submodel. Position and Transformations are managed in the Position/Transformation Tool.

Pretensioner

Pretensioner entities define pretensioners.

LS-DYNA Cards

Card	Description
*ELEMENT_SEATBELT_PR	Defines pretensioners.

Properties

Property entities define and store 1D, 2D, and 3D property definitions for a model.

Properties do not have a display state in the modeling window. You can color the model according to the colors assigned to each property, which is based on element property relationships, by changing the color mode to propety.

Element property relationships are dependent on the solver interface. In general, when a component is assigned a property, that property assignment is applied to all elements collected by that component. The method of assigning properties at the component level is therefore referred to as indirect property assignment. Direct property assignment is performed directly on the elements themselves, typically via a property assignment. Direct property assignments always take precedence over indirect property and material assignments.

Abaqus Cards

Card	Description
*ADAPTIVE CONTROLS	Controls various aspects of the adaptive meshing and advection algorithms applied to an adaptive mesh domain.
*BEAM ADDED INERTIA	Define additional beam inertia.
*BEAM GENERAL SECTION	Specify a beam section when numerical integration over the section is not required. **Note:** Only one *BEAM GENERAL SECTION card is output per component, therefore the beam elements in each component must have the same cross-sectional properties.
*BEAM SECTION	Specify a beam section when numerical integration over the section is required. **Note:** Only one *BEAM SECTION card is output per component, therefore the beam elements in each component must have the same cross-sectional properties.
*COHESIVE SECTION	Specify element properties for cohesive elements.
*CONNECTOR SECTION	Specify connector attributes for connector elements. **Note:** The following types are supported: Standard template: , Explicit template: All listed above, as well as ROTATION-ACCELEROMETER
*CONTACT DAMPING	Define viscous damping between contacting surfaces. **Note:** This card is a sub-option in the *SURFACE INTERACTION card image.
*DASHPOT	Define dashpot behavior.

Card	Description
	Note: Only one *DASHPOT card is output per component, therefore the spring elements in each component must have the same properties. When the *DASHPOT card is written for DASHPOT1 elements, both dof1 and dof2 are written, but Abaqus only reads dof1. For DASHPOTA elements, choose the DASHPOTA option in the *DASHPOT card image.
*ELEMENT PROPERTIES	
*EULERIAN SECTION	Define properties of Eulerian continuum elements, including the list of materials that may occupy the elements.
*FASTENER (SPOT WELD) PROPERTY	Prescribe mesh-independent fastener properties.
*FLUID BULK MODULUS	Define compressibility for a hydraulic fluid. **Note:** This option is used to define compressibility for the hydraulic fluid model. It can only be used in conjunction with the *FLUID BEHAVIOR option or the *FLUID PROPERTY option.
*FLUID DENSITY	Specify hydrostatic fluid density. **Note:** This option is used to define the reference fluid density for fluid cavities. It is only applicable for hydraulic and pneumatic fluids, and should not be used for user-defined fluids. The *FLUID DENSITY option can only be used in conjunction with the *FLUID BEHAVIOR option or the *FLUID PROPERTY option.
*FLUID EXPANSION	Specify the thermal expansion coefficient for a hydraulic fluid. **Note:** This option is used to define thermal expansion coefficients for the hydraulic fluid model. It can only be used in conjunction with the *FLUID BEHAVIOR option or the *FLUID PROPERTY option.
*FLUID PROPERTY	Define properties for hydrostatic fluid elements.
*FRICTION	Specify a friction model.

Card	Description
	Note: This card is a sub-option in the *SURFACE INTERACTION card image. It is also supported as a separate card image to allow for it to be used as a sub-option of the *CONNECTOR FRICTION card (in *CONNECTOR BEHAVIOR).
*GAP	Specify clearance and local geometry for GAP-type elements. **Note:** Only one *GAP card is output per component, therefore the gap elements in each component must have the same properties. Not available in the Explicit template.
*GASKET SECTION	Specify element properties for gasket elements. **Note:** Not available in the Explicit template.
*ITS	Define the properties for ITS-type elements.
*JOINT	Define properties for JOINTC elements. **Note:** Only one *JOINT card is output per component, therefore the spring elements in each component must have the same properties. The *SPRING and *DASHPOT cards in the *JOINT property behave the same way as the individual cards mentioned above. See the How do I section below for more information. Not available in the Explicit template.
*M1	Define the first bending moment behavior of beams.
*M2	Define the second bending moment behavior of beams.
*MASS	Specify a point mass. **Note:** Only one *MASS card is output per component, therefore the mass elements in each component must have the same properties.
*MEMBRANE SECTION	Specify section properties for membrane elements.

Card	Description
*NONSTRUCTURAL MASS	Specify mass contribution to the model from nonstructural features. **Note:** Only available in the Explicit template.
*PHYSICAL CONSTANTS	Specify physical constants.
*REBAR LAYER	Reinforcement definition **Note:** The keyword is available in the card image of the SHELL SECTION (homogeneous and composite), MEMBRANE SECTION, and SURFACE SECTION.
*RIGID BODY	Define a set of elements as a rigid body and define rigid element properties. **Note:** For Analytical Rigid Surfaces, the *ANALYTICAL SURFACE* parameter should point to the corresponding ANALYTICAL_RIGID_SURFACE group from the card image of the *RIGID BODY card.
*ROTARY INERTIA (no longer listed on panel)	Define rigid body rotary inertia. **Note:** Only one *ROTARY INERTIA card is output per component, therefore the ROTARY1 elements in each component must have the same properties.
*SECTION CONTROLS	Specify section controls.
*SHELL GENERAL SECTION	Define a general, arbitrary, elastic shell section.
*SHELL SECTION	Specify a shell cross-section.
*SOLID SECTION	Specify element properties for solid, infinite, acoustic, and truss elements.
*SPRING	Define spring behavior.

Card	Description
	Note: Only one *SPRING card is output per component, therefore the spring elements in each component must have the same properties. When the *SPRING card is written for SPRING1 elements, both dof1 and dof2 are written, but Abaqus only reads dof1. For SPRINGA elements, choose the SPRINGA option in the *SPRING card image.
*SURFACE BEHAVIOR	Define alternative pressure-overclosure relationships for contact. **Note:** This card is a sub-option in the *SURFACE INTERACTION card image.
*SURFACE INTERACTION	Define surface interaction properties. **Note:** For Abaqus Explicit template, this card is defined as a group.
*SURFACE PROPERTY / *EMISSIVITY	Define surface properties for cavity radiation analysis. It must immediately precede the *EMISSIVITY option.
*SURFACE SECTION	Specify section properties for surface elements.
*SURFACE SMOOTHING	Create a surface smoothing definition for contact interactions. It must be used in conjunction with the *CONTACT PAIR option.
*TRANSVERSE SHEAR STIFFNESS	Define transverse shear stiffness for beams and shells.

Card	Description
	Note: This option must be used in conjunction with the *BEAM GENERAL SECTION, *BEAM SECTION, *COHESIVE SECTION, *SHELL GENERAL SECTION, or the *SHELL SECTION options. The transverse shear stiffness defined with this option affects only the transverse shear flexible elements whose section properties are defined by the immediately preceding section option.

ANSYS Cards

Card	Description
SECTYPE	Associates section type information with a section ID number.

LS-DYNA Cards

Card	Description
*CONSTRAINED_JOINT_S TORSION	Define optional rotational and translational joint stiffness for joints.
*CONSTRAINED_JOINT_S	Define optional rotational and translational joint stiffness for joints.
*CONSTRAINED_JOINT_S	Define optional rotational and translational joint stiffness for joints.
*DAMPING_PART_MASS	Define mass weighted damping by part ID
*DAMPING_PART_MASS_	
*DAMPING_PART_STIFFN	Assign Rayleigh stiffness damping coefficient by part ID
*DAMPING_PART_STIFFN	
*DAMPING_RELATIVE	Apply damping relative to the motion of a rigid body.
*DEFINE_CONNECTION_I	Define failure related parameters for solid element spot weld failure by *MAT_SPOTWELD_DAIMLERCHRYSLER.
*EOS_GRUNEISEN (EOS 4)	Equation of state Form 4.

Card	Description
*EOS_IDEAL_GAS (EOS 12)	Equation of state for 12 for modeling ideal gas.
*EOS_IGNITION_AND_GI (EOS 7)	Equation of state Form 7.
*EOS_JWL (EOS 2)	Equation of state Form 2.
*EOS_LINEAR_POLYNOMI (EOS 1)	Equation of state Form 1. Define coefficients for linear polynomial EOS and initialize the initial thermodynamic state of the material.
*EOS_LINEAR_POLYNOMI (EOS 6)	Equation of state Form 6.
*EOS_PROPELLANT_DEFL (EOS 10)	Equation of state Form 10. Added to model airbag propellants.
*EOS_RATIO_OF_POLYNC (EOS 5)	Equation of state Form 5.
*EOS_SACK_TUESDAY (EOS 3)	Equation of state Form 3.
*EOS_TABULATED (EOS 9)	Equation of state Form 9.
*EOS_TABULATED_COMP. (EOS 8)	Equation of state Form 8.
*EOS_TENSOR_PORE_CC (EOS 11)	Equation of state Form 11.
*INTEGRATION_BEAM	Define user defined through the thickness integration rules for the beam element.
*INTEGRATION_SHELL	Define user defined through the thickness integration rules for the shell element.
*MAT_ADD_EROSION	Many of the consitutive models in LS-DYNA do not allow failure and erosion. This option provides a way of including failure in these models although the option can also be applied to constitutive models of other failure/erosion criterion.
*SECTION_BEAM(TITLE)	Define cross sectional properties for beam, truss, discrete beam and cable elements.
*SECTION_BEAM_AISC	Defines cross-sectional properties for beams and trusses.

Card	Description
*SECTION_DISCRETE(TIT	Define spring and damper elements for translation and rotation.
*SECTION_POINT_SOUR(	Provides the inlet boundary condition for single gas flow (inflation potential) via a set of point source(s).
*SECTION_POINT_SOUR(	Provides: (a) an element formulation for a solid ALE part of the type similar to ELFORM=11 of *SECTION_SOLID and (b) the inlet gas injection boundary condition for multiple-gas mixture in-flow via a set of point sources.
*SECTION_SEATBELT(TIT	Define section properties for the seat belt elements.
*SECTION_SHELL(TITLE)	Define section properties for shell elements.
*SECTION_SHELL_ALE(TI	Define section properties for shell elements.
*SECTION_SHELL_EFG(T	Define section properties for shell elements.
*SECTION_SOLID(TITLE)	Define section properties for solid continuum and fluid elements.
*SECTION_SOLID_ALE(TI	Define section properties for solid continuum and fluid elements.
*SECTION_SOLID_EFG(T	Define section properties for solid continuum and fluid elements.
*SECTION_SPH	Define section properties for SPH particles.
*SECTION_SPH_TENSOR	
*SECTION_SPH_USER	
*SECTION_TSHELL(TITLE	Define section properties for SPH particles.

Nastran Cards

Only one card image can be loaded into each property collector.

1D elements can be grouped into components with 2D and 3D elements for display purposes. The component groupings are maintained on export and import.

Properties for PBAR and PBEAM cards can be manually input in the card image or automatically created.

The HM_ELAS card defines properties for an HM_Spring element.

The Nastran and OptiStruct solver interfaces allow the property between groups to have the same ID. For example, PBAR3, PSHELL 3 and PSOLID 3. Duplicate IDs within the same group is not allowed.

Nastran and OptiStruct properties are grouped as follows:

0D_Rigids

PMASS

1D

PBAR, PBARL, PBEAM, PBEAML, PBEAND, PROD, PTUBE, PWELD

SPRING_GAP

PBUSH, PBUSH1D, PDAMP, PELAS, PGAP, PVISC

2D

PSHELL, PSHEAR, PCOMP, PCOMG

3D

PSOLID

Card	Description
BCBDPRP	Defines contact body parameters.
BCONPRG	Defines geometric contact parameters of touching bodies.
BCONPRP	Defines physical contact parameters of touching bodies.
PAABSF	Defines the properties of a frequency-dependent acoustic absorber element.
PACABS	Defines the properties of the acoustic absorber element.
PACBAR	Defines the properties of the acoustic barrier element. **Note:** PACBAR is referenced by a CHACBR entry only. Either FRESON or KRESON must be specified, but not both.
PACINF	Defines the properties of acoustic conjugate infinite elements.
PAERO1	Defines associated bodies for the panels in the Doublet-Lattice method.
PAERO2	Defines the cross-sectional properties of aerodynamic bodies.
PAXSYMH	Defines the properties of a linear axisymmetric harmonic element.
PBAR	Defines the properties of a simple beam element (CBAR entry).
PBARL	Defines the properties of a simple beam element (CBAR entry) by cross-sectional dimensions.
PBEAM	Defines the properties of a beam element (CBEAM entry). This element may be used to model tapered beams. **Note:** Blank fields are not supported for intermediate stations. Appropriate default values are inserted during feinput.
PBEAML	Defines the properties of a beam element by cross-sectional dimensions.

Card	Description
	Note: Blank fields are not supported for intermediate stations. Appropriate default values are inserted during feinput.
PBEND	Defines the properties of a curved beam, curved pipe, or elbow element (CBEND entry).
PBUSH	Defines the nominal property values for a generalized spring-and-damper structural element.
PBUSH1D	Defines linear and nonlinear properties of a one-dimensional spring and damper element (CBUSH1D entry).
PBUSHT	Defines the frequency dependent properties or the stress dependent properties for a generalized spring and damper structural element.
PCOMP	Defines the properties of an n-ply composite material laminate.
PCOMPG	Defines global (external) ply IDs and properties for a composite material laminate.
PCOMPLS	Defines global (external) ply IDs and properties for a composite material laminate in SOL 600 and SOLs 400 and 700.
PCONVM	Specifies forced convection boundary condition properties for a boundary condition surface element.
PDAMP	Specifies the damping value of a scalar damper element using defined CDAMP1 or CDAMP3 entries.
PELAS	Specifies the stiffness, damping coefficient, and stress coefficient of a scalar elastic (spring) element (CELAS1 or CELAS3 entry).
PELAST	Defines the frequency dependent properties for a PELAS Bulk Data entry.
PFAST	Defines the CFAST fastener property values.
PGAP	Defines the properties of the gap element (CGAP entry).
PHBDY	Property entry referenced by a CHBDYP element to give auxiliary geometric information for it.
PLPLANE	Defines the properties of a fully nonlinear (i.e., large strain and large rotation) hyperelastic plane strain or axisymmetric element.

Card	Description
PLSOLID	Defines a fully nonlinear (i.e., large strain and large rotation) hyperelastic solid element.
PMASS	Specifies the mass value of a scalar mass element (CMASS1 or CMASS3 entries).
PROD	Defines the properties of a rod element (CROD entry).
PSEAM	Defines the PSEAM property values.
PSHEAR	Defines the properties of a shear panel (CSHEAR entry).
PSHELL	Defines the membrane, bending, transverse shear, and coupling properties of thin shell elements.
PSHELL1	Defines the properties of SOL 700 shell elements, which are more complicated than the shell elements defined using the PSHELL entry.
PSHLN1	
PSOLID	Defines the properties of solid elements (CHEXA, CPENTA, and CTETRA entries).
PTUBE	Defines the properties of a thin-walled cylindrical tube element (CTUBE entry).
PVISC	Defines properties of a one-dimensional viscous damping element (CVISC entry).
PWELD	Defines the properties of connector (CWELD) elements.

OptiStruct Cards

Only one property definition is allowed on each property collector. For definitions like PMASS, which allow more than one definition on the same card, this is separated on import into four different cards.

1D elements can be organized into components with 2D and 3D elements, and these component groupings are maintained on export and import. However, this usage is not recommended.

Card	Description
HM_ELAS	Defines properties for a HM_Spring element. **Note:** Bulk Data Entry
PAABSF	Defines the properties of the fluid acoustic absorber element.

Card	Description
	Note: Bulk Data Entry
PAXI	Defines the properties of axisymmetric elements. Referenced by `CTAXI` entry. **Note:** Bulk Data Entry Referenced by `CTAXI`.
PBAR	Defines the properties of a simple beam (bar), which is used to create bar elements via the `CBAR` entry. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
PBARL	Defines the properties of a simple beam (bar) by cross-sectional dimensions, which is used to create bar elements via the `CBAR` entry. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
PBEAM	Defines the properties of beam elements defined via the `CBEAM` entry. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
PBEAML	Defines the properties of a beam element by cross-sectional dimensions that are used to create beam elements via the `CBEAM` entry. **Note:** Bulk Data Entry
PBUSH	Defines the nominal property values for a generalized spring-damper-mass structural element. **Note:** Bulk Data Entry
PBUSH1D	Defines the linear and nonlinear properties for a one-dimensional spring-and-damper structural element.

Card	Description
	Note: Bulk Data Entry
PCOMP	Defines the structure and properties of an n-ply composite laminate material. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
PCOMPG	Defines the structure and properties of a composite laminate material, allowing for global ply identification. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
PCOMPP	Defines the properties of a composite laminate material used in ply-based composite definition. **Note:** Bulk Data Entry Exported in large field format by `optistructlf` template.
PCONT	Defines properties of a `CONTACT` interface. **Note:** Bulk Data Entry
PCONTX	Defines properties of a `CONTACT` interface for geometric nonlinear analysis. **Note:** Bulk Data Entry
PDAMP	Specifies the damping of a scalar damper element using defined `CDAMP1` or `CDAMP3` entry. **Note:** Bulk Data Entry
PELAS	Used to define the stiffness and stress coefficient of a scalar elastic element (spring) by means of the `CELAS1` or `CELAS3` entry. **Note:** Bulk Data Entry

Card	Description
PFAST	Define properties of connector (CFAST) elements. **Note:** Bulk Data Entry
PGAP	Defines properties of the gap (CGAP or CGAPG) elements. **Note:** Bulk Data Entry
PGASK	Defining the properties for solid gasket elements. **Note:** Bulk Data Entry
PLSOLID	Defines the properties of nonlinear hyperelastic solid elements, referenced by CHEXA, CPENTA, and CTETRA Bulk Data Entries. The MATHE hyperelastic material can be referenced to define corresponding material properties. **Note:** Bulk Data Entry
PMASS	Defines the mass value of a scalar mass element (CMASS1 or CMASS3 entry). **Note:** Bulk Data Entry
PROD	Defines the properties of a rod, which is referenced by the CROD entry. **Note:** Bulk Data Entry Exported in large field format by optistructlf template.
PSHEAR	Defines the properties of a shear panel. **Note:** Bulk Data Entry
PSHELL	Defines the membrane, bending, transverse shear, and membrane-bending coupling of shell elements. **Note:** Bulk Data Entry Exported in large field format by optistructlf template.
PSOLID	Defines the properties of solid elements, referenced by CHEXA, CPENTA, CPYRA and CTETRA Bulk Data Entries.

Card	Description
	Note: Bulk Data Entry
PTUBE	Defines the properties of a thin-walled cylindrical tube element, referenced by a `CTUBE` entry. **Note:** Bulk Data Entry
PVISC	Defines properties of a one-dimensional viscous damping element (`CVISC` entry). **Note:** Bulk Data Entry
PWELD	Defines properties of connector (`CWELD`) elements. **Note:** Bulk Data Entry

PAM-CRASH Cards

Card	Description
FRICT /	Friction modeling definition.
GASPEC /	Specification of air bag gas.
RUPMO /	Rupture model definition.

Permas Cards

Card	Description
$GEODAT BEAM	Beam
$GEODAT CONA	Surface convection
$GEODAT CONS	Shell surface convection
$GEODAT DAMPER	Viscous damper
$GEODAT FLANGE	Flange

Card	Description
$GEODAT GASKET	Gasket
$GEODAT MASS	Mass
$GEODAT SCALAR	Scalar
$GEODAT SHELL	Shell
$GEODAT SOLID	Solid
$GEODAT SPRING	Spring
$GEODAT SPRINGX	Spring

Radioss Cards

Radioss allows you to program your own properties, mostly for springs, that can be used in a simulation. Unsupported Radioss properties and user defined Radioss properties are assigned the PROP_UNSUPPORTED card image.

HyperMesh imports unsupported properites with the PROP_UNSUPPORTED card image, and preserves their corresponding IDs and associated components.

In the PROP_UNSUPPORTED card image, all property sub-options, parameters, and data lines are supported as simple text. HyperMesh does not check the validity or syntax of any data in this mode. You must manually check the validity of the data. No editing, updating, or review of the property data is intended. Also time step calculation and mass calculation are not available for the component that refers to this property.

Card	Description
/ADMESH/SET	Defines the criteria for adaptive meshing in parts. This keyword is not available for SPMD computation. **Note:** Block Format Keyword
/DAMP/	Defines the Rayleigh mass and stiffness damping coefficients applied to a set of nodes. The damping can be applied to any nodal DOF either in local or global coordinate system. **Note:** Block Format Keyword
/EOS/GRUNEISEN	Describes the Gruneisen equation of state.

Card	Description
	Note: Block Format Keyword
/EOS/POLYNOMIAL	Describes the Linear polynomial equation of state P(ρ ,E). **Note:** Block Format Keyword
/EOS/PUFF	Describes the linear polynomial equation of state P(ρ,E) . **Note:** Block Format Keyword
/EOS/SESAME	Describes the SESAME table equation of state. **Note:** Block Format Keyword
/EOS/TILLOTSON	Describes the Tillotson equation of state. **Note:** Block Format Keyword
/FAIL	Describes the failure models. **Note:** Block Format Keyword
/FAIL/CHANG	Describes the Chang failure model. **Note:** Block Format Keyword
/FAIL/CONNECT	Describes the failure model for CONNECTION material with elongation criteria and/or energy criteria. **Note:** Block Format Keyword
/FAIL/ENERGY	Describes a specific energy failure model. **Note:** Block Format Keyword
/FAIL/FLD	Describes the forming limit.

Card	Description
	Note: Block Format Keyword
/FAIL/HASHIN	Describes the Hashin failure model. This failure model is available for Shell and Solid. **Note:** Block Format Keyword
/FAIL/JOHNSON	Describes the failure criteria by Johnson-Cook failure model. **Note:** Block Format Keyword
/FAIL/LAD_DAMA	Describes the Ladeveze failure model for delamination (interlaminar fracture). **Note:** Block Format Keyword
/FAIL/PUCK	Describes the Puck failure model. **Note:** Block Format Keyword
/FAIL/SPALLING	Describes the Spalling and Johnson-Cook failure model. **Note:** Block Format Keyword
/FAIL/TAB1	Describes the strain failure model based on damage accumulation using user-defined functions. **Note:** Block Format Keyword
/FAIL/TBUTCHER	Describes a Tuler-Butcher failure model. **Note:** Block Format Keyword
/FAIL/TENSSTRAIN	Describes a strain failure model. **Note:** Block Format Keyword

Card	Description
/FAIL/USERi (USER 1, 2, 3)	Describes a user failure model. **Note:** Block Format Keyword
/FAIL/WIERZBICKI	Describes the BAO-XUE-Wierzbicki failure model. **Note:** Block Format Keyword
/FAIL/WILKINS	Describes the Wilkins failure model. **Note:** Block Format Keyword
/FAIL/XFEM	Describes a XFEM (eXtended Finite Element Method) failure model. **Note:** Block Format Keyword
/LEAK/MAT	Specifies effective leakage area of porous airbag fabric materials LAW19 and LAW58 as function of time, pressure, area and other parameters. **Note:** Block Format Keyword
/PROP	Describes the property sets. **Note:** Block Format Keyword
/PROP/TYPE3 (BEAM)	Describes the beam property for torsion, bending, membrane or axial deformation. **Note:** Block Format Keyword
/PROP/TYPE43 (CONNECT)	Designed for spotweld, welding line or glue type connections. Only used with `/MAT/CONNECT` material law. **Note:** Block Format Keyword
/PROP/TYPE14 (FLUID)	Describes the general fluid property set. **Note:** Block Format Keyword

Card	Description
/PROP/INJECT1	Describes mass injected for each constituent gas. **Note:** Block Format Keyword
/PROP/INJECT2	Describes molar fraction injected for each constituent gas and total mass injected. **Note:** Block Format Keyword
/PROP/TYPE18 (INT_BEAM)	Describes the integrated beam property set. This beam model is based on Timoshenko theory and takes into account transverse shear strain without warping in torsion. It can be used for deep beam cases (short beams). Beam section and position of integration points can be either used as predefined or prescribed directly. **Note:** Block Format Keyword
/PROP/TYPE33 (KJOINT)	Describes the joint type spring. **Note:** Block Format Keyword
/PROP/TYPE45 (KJOINT2)	Describes the joint type spring between two rigid bodies. **Note:** Block Format Keyword
/PROP/TYPE19 (PLY)	Defines the ply property set used in ply-based composite definition. It is used in combination with `/PROP/STACK (/PROP/TYPE17)` or `/PROP/TYPE51` to create ply-based sandwich composite properties. **Note:** Block Format Keyword
/PROP/TYPE15 (POROUS)	Describes the porous solid element property set (extended Darcy's law). **Note:** Block Format Keyword
/PROP/TYPE5 (RIVET)	Describes the rivet property set. **Note:** Block Format Keyword

Card	Description
/PROP/TYPE10 (SH_COMP)	Defines the composite shell property set. It is possible to define composite with several layers and each layer with individual orthotropic direction. **Note:** Block Format Keyword
/PROP/TYPE16 (SH_FABR)	Defines the anisotropic layered shell property set. This property is currently only compatible with Elastic Anisotropic Fabric (`/MAT/LAW58 (FABR_A)`) and only one layer is allowed. **Note:** Block Format Keyword
/PROP/TYPE9 (SH_ORTH)	Defines the orthotropic shell property. **Note:** Block Format Keyword
/PROP/TYPE19 (PLY)	Defines the ply property set used in ply-based composite definition. It is used in combination with `/PROP/STACK (/PROP/TYPE17)` or `/PROP/TYPE51` to create ply-based sandwich composite properties. **Note:** Block Format Keyword
/PROP/TYPE11 (SH_SANDW)	Defines the sandwich shell property set. It is possible to define sandwich composite with several layers and each lay with individual material, thickness, layer position and orthotropic direction. This property is only compatible with Material Laws 15, 25, 27, 36, 60, 72 and user laws. **Note:** Block Format Keyword
/PROP/TYPE1 (SHELL)	Describes the shell property set which used for 3-node or 4-node shell element. Belytschko, QBAT or QEPH shell formulation are available. **Note:** Block Format Keyword
/PROP/TYPE6 (SOL_ORTH)	Describes the orthotropic solid property set. This property set is used to define the fiber plane for `/MAT/LAW14(COMPS0)`, the steel reinforcement direction for `/MAT/LAW24 (CONC)` or the cell direction for `/MAT/LAW28 (HONEYCOMB)`. This property is only available for 8-node linear solid elements (`/BRICK`), tetrahedron elements (`/TETRA4` and `/TETRA10`), and 2D solid elements (`/QUAD`). Quadratic bricks (`/BRIC20` and `/SHEL16`) and pentahedron elements (`/PENTA6`) are not compatible with this property.

Card	Description
	Note: Block Format Keyword
/PROP/TYPE14 (SOLID)	Defines the general solid property set. **Note:** Block Format Keyword
/PROP/SPH	Defines the axisymmetric spring property set. **Note:** Block Format Keyword
/PROP/TYPE25 (SPR_AXI)	Defines the axisymmetric spring property set. **Note:** Block Format Keyword
/PROP/TYPE13 (SPR_BEAM)	A beam type spring property that works as a beam element with six independent modes of deformation. This spring accounts for non-linear stiffness, damping and different unloading. Deformation, force and energy-based failure criteria are available. **Note:** Block Format Keyword
/PROP/TYPE44 (SPR_CRUS)	A spring element property that represents a simple macro model of a crushable frame in compression, tension, torsion and bending. Originally, this element was developed in cooperation with PSA PEUGEOT CITROËN. **Note:** Block Format Keyword
/PROP/TYPE8 (SPR_GENE)	A spring property that works with six independent modes of deformation. This spring accounts for non-linear stiffness, damping and different unloading. Deformation, force and energy based failure criteria are available. The general spring property is often used to model a joint connection between two parts. **Note:** Block Format Keyword
/PROP/TYPE32 (SPR_PRE)	Describes the pretension spring property set. **Note:** Block Format Keyword

Card	Description
/PROP/TYPE12 (SPR_PUL)	A pulley spring property set (with one translational DOF) used to model a pulley. **Note:** Block Format Keyword
/PROP/TYPE26 (SPR_TAB)	Defines the tabulated spring property. **Note:** Block Format Keyword
/PROP/TYPE4 (SPRING)	Defines spring property with one translational DOF. This spring accounts for non-linear stiffness, damping and different unloading. Deformation based failure criteria is available. **Note:** Block Format Keyword
/PROP/TYPE17 (STACK)	Defines the sandwich shell property set using the stack and ply approach. **Note:** Block Format Keyword
/PROP/TYPE2 (TRUSS)	Defines the truss property set. **Note:** Block Format Keyword
/PROP/TYPE22 (TSH_COMP)	Defines the composite thick shell property set. **Note:** Block Format Keyword
/PROP/TYPE21 (TSH_ORTH)	Defines the orthotropic thick shell property set. **Note:** Block Format Keyword
/PROP/TYPE20 (TSHELL)	Defines the general thick shell property set. **Note:** Block Format Keyword
/PROP/USER	User-defined property. **Note:** Block Format Keyword

Card	Description
/PROP/TYPE0 (VOID)	Defines the void property set. **Note:** Block Format Keyword
/THERM_STRESS/MAT	Adds thermal expansion property for Radioss material (shell and solid). **Note:** Block Format Keyword

Samcef Cards

Card	Description
.BPR	Define beam profiles.
.ETASHELL	Used to assign the laminate to the elements. The Projection method is supported.
.ETASOLID	Used to assign the laminate to the elements. The Projection method is supported.
.MCCBUSH	Defines the property on the BUSH element.
.PHP SHELL	Assign physical properties to an existing mesh.
SOLIDMAT	Assign physical properties to an existing mesh. **Note:** This is a dummy property that creates a link between the elements and the material, as it is not possible to directly assign a material to the elements.

Regions

Region entities store information used to facilitate and automate modeling practices and processes. It enables a selection which can be common across design changes or other models, provided region data is the same.

Region entities support two configurations of input: By ID and By Metadata. Both inputs are supported for geometry only, and can be used for selection purposes.

For example, when using mesh controls, it is possible to mesh a surface which has been tagged with metadata (or ID). If the design changes and a new version is authored, then you can quickly reapply the same mesh controls as long as the metadata (or ID) is still applied to the new CAD version. Region entities enables an automated re-meshing process that is consistently repeatable.

Retractors

Retractor entities define retractors.

LS-DYNA Cards

Card	Description
*ELEMENT_SEATBELT_RE	Defines retractors.

Rigid Walls

Rigid wall entities provide a method for treating a contact between a rigid surface and nodal points of a deformable body. In the LS-DYNA and Radioss user profiles, rigid walls can be created in the Model and Solver browsers.

LS-DYNA Cards

Card	Description
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	

Card	Description
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_GEOMETRIC	
*RIGIDWALL_PLANAR	Define planar rigid walls with either finite or infinite size.
*RIGIDWALL_PLANAR_FI	
*RIGIDWALL_PLANAR_FI	
*RIGIDWALL_PLANAR_FI	
*RIGIDWALL_PLANAR_FI	
*RIGIDWALL_PLANAR_FI	
*RIGIDWALL_PLANAR_FI	
*RIGIDWALL_PLANAR_MO	
*RIGIDWALL_PLANAR_MO	
*RIGIDWALL_PLANAR_MO	
*RIGIDWALL_PLANAR_MO	
*RIGIDWALL_PLANAR_MO	

Card	Description
*RIGIDWALL_PLANAR_OF	
*RIGIDWALL_PLANAR_OF	
*RIGIDWALL_PLANAR_OF	
*RIGIDWALL_PLANAR_OF	

OptiStruct

Card	Description
RWALADD	Defines a rigid wall set as a union of rigid walls defined via `RWALL` entries. **Note:** Bulk Data Entry
RWALL	Defines a rigid wall for geometric nonlinear analysis. **Note:** Bulk Data Entry

Radioss Cards

Card	Description
/RWALL/PLANE	Infinite Plane rigid wall
/RWALL/CYL	Infinite Cylinder rigid wall
/RWALL/SPHER	Spherical rigid wall
/RWALL/PARAL	Parallelogram rigid wall

Card	Description
/RWALL/THERM	ALE infinite plane rigid wall with thermal conductivity

Sensors

Sensor entities define and store sensors typically used in safety analysis.

Sensors do not have a display state in the modeling window.

LS-DYNA Cards

Card	Description
*DATABASE_CPM_SENSO	Activates an ASCII file "cpm_sensor". Its input defines sensors' locations based on the positions of some Lagrangian segments. The output gives the history of the velocity, temperature, density, and pressure averaged on the number of particles contained in the sensors.
*PART_SENSOR	Links part/component to a sensor to activate and deactivate during the analysis. **Note:** This is supported as an attribute to an element to maintain its associativity with element inside HyperMesh.
*SENSOR_CONTROL	Applies sensor result on an entity during run.
*SENSOR_DEFINE_ELEMI	Strain gauge type sensor.
*SENSOR_DEFINE_FORCI	Force transducer type sensor.
*SENSOR_DEFINE_NODE	Accelerometer type sensor.
*SENSOR_DEFINE_CALC-MATH	Perform mathematical calculations on sensor values.
*SENSOR_SWITCH	Compares sensor value with input value and gives a logical signal.

Card	Description
*SENSOR_SWITCH_CALC LOGIC	Performs mathematical calculations on the logical signal from sensor logic.

Nastran

Card	Description
MONDSP1	Defines a virtual point displacement response at a user-defined reference location (coordinates and coordinates system) as a weighted average of the motions at a set of grid points.
MONPNT1	Defines an integrated load monitor point at a point (x,y,z) in a user defined coordinate system.
MONPNT2	Element Monitor Output Results Item.
MONPNT3	Sums select Grid Point Forces to a user chosen monitor point.

OptiStruct Cards

Card	Description
MARKER	Defines a marker by associating a grid and a coordinate system. **Note:** Bulk Data Entry

PAM-CRASH Cards

Card	Description
SENSO /	Definition of a sensor. **Note:** Types 2 and 4 are not supported.
SENSOR/	Definition of a sensor.

Card	Description
	Note: Types 2 and 4 are not supported.

Radioss Cards

Card	Description
/GAUGE	Defines gauges. **Note:** Block Format Keyword
/SENSOR	Defines a sensor for geometric nonlinear analysis. Sensors may be used to activate loads (see `NLOAD1`). **Note:** Block Format Keyword
/SENSOR/ACCE	Defines an accelerometer.. **Note:** Block Format Keyword Available in the /SENSOR card image, when the Sensor Type is set to Accelerometer - Type 1.
/SENSOR/AND	ON if sensors SID1 and SID2 are ON **Note:** Block Format Keyword Available in the /SENSOR card image, when the Sensor Type is set to ON as long as S1 AND S2 - Type 4.
/SENSOR/DIST	Defines pressure gauge. **Note:** Block Format Keyword Available in the /SENSOR card image, when the Sensor Type is set to Nodal Distance - Type 2.
/SENSOR/GAUGE	
/SENSOR/INTER	Defines interface activation and deactivation.

Card	Description
	Note: Block Format Keyword Available in the /SENSOR card image, when the Sensor Type is set to Interface Act and Deact - Type 6.
/SENSOR/NOT	ON if sensor SID1 is OFF. **Note:** Block Format Keyword Available in the /SENSOR card image, when the Sensor Type is set to On as long as S1 is OFF - Type 8.
/SENSOR/OR	ON if sensors SID1 or SID2 is ON. **Note:** Block Format Keyword Available in the /SENSOR card image, when the Sensor Type is set to On as long as S1 OR S2 - Type 5.
/SENSOR/RBODY	Activation of sensor by Rigid Body force criteria. **Note:** Block Format Keyword Available in the /SENSOR card image, when the Sensor Type is set to RBody - Type11.
/SENSOR/RWAL	Activation/Deactivation of sensor by contact with rigid wall. **Note:** Block Format Keyword Available in the /SENSOR card image, when the Sensor Type is set to Rigid wall Act and Deact - Type 7.
/SENSOR/SECT	Activation of sensor by Section force criteria. **Note:** Block Format Keyword Available in the /SENSOR card image, when the Sensor Type is set to Sect - Type12.
/SENSOR/SENS	Activation with sens_ID1; Deactivation with sens_ID2.

Card	Description
	Note: Block Format Keyword Available in the /SENSOR card image, when the Sensor Type is set to Active S1, Deact S2 - Type 3.
/SENSOR/TIME	Start time. **Note:** Block Format Keyword Available in the /SENSOR card image, when the Sensor Type is set to Start Time - Type 0.

Sets

Set entities define and store lists of entity IDs for entities, sets of sets, or lists of set IDs.

Sets can only be generated for certain entity types.

Sets do not have a display state in the modeling window.

Abaqus Cards

User Comment blocks are supported for all sets, and will be preserved during the import and export of the Abaqus input deck.

Sets using the *GENERATE* parameter can be expanded upon import by enabling the Expand sets defined by range option from the **File Options** dialog, which opens when you import a solver deck. This option is useful when node/element IDs are renumbered during import.

Card	Description
*DISTRIBUTION	Define spatial distributions.
*ELSET	Assign elements to an element set.
*EMBEDDED ELEMENT	Specify an element or a group of elements that lie embedded in a group of “host” elements in a model.
*NODAL THICKNESS	Define shell or membrane thickness at nodes.

Card	Description
*NSET	Assign nodes to a node set.

ANSYS Cards

Card	Description
CMGRP	Groups components and assemblies into an assembly. **Note:** This is supported as sets of sets.

EXODUS Cards

Card	Description
NodeSets	Provides a means to reference a group of nodes with a single ID. Also used to specify load or boundary conditions, or to identify nodes for a special output request.

LS-DYNA Cards

Default LS-DYNA attribute values for a set can be edited, whereas individual values cannot be edited.

Card	Description
*ALE_MULTI-MATERIAL_GROUP	Defines the appropriate ALE material groupings for interface reconstruction when many ALE Multi-Material Groups are present in a model.
*DEFINE_HEX_SPOTWELI	Assembly of elements that describes a spotweld.
*DEFINE_HEX_SPOTWELI	
*SET_BEAM(TITLE)	Define a set of beam elements.
*SET_BEAM_ADD	Define a beam set by combining beam sets.
*SET_BEAM_GENERATE(T	Generates a block of beam element IDs between a starting ID and an ending ID.
*SET_DISCRETE(TITLE)	Define a set of discrete elements.
*SET_DISCRETE_ADD	Define a discrete set by combining discrete sets.

Card	Description
*SET_DISCRETE_GENERA	Generates a block of discrete element IDs between a starting ID and an ending ID.
*SET_NODE_ADD	Define a node set by combining node sets.
*SET_NODE_ADD_ADVAN	Define a node set by combining node sets or by combining NODE, SHELL, SOLID, BEAM, SEGMENT, DISCRETE and THICK SHELL sets.
*SET_NODE_COLUMN	Define a nodal set with some identical or unique attributes.
*SET_NODE_LIST(TITLE)	Define a nodal set with some identical or unique attributes.
*SET_NODE_LIST_GENEF	Generate a block of node IDs between a starting nodal ID number and an ending nodal ID number.
*SET_PART_ADD	Define a part set by combining part sets.
*SET_PART_COLUMN(TIT	Define a set of parts with optional attributes.
*SET_PART_LIST_GENER	Generate a block of part IDs between a starting part ID number and an ending part ID number.
*SET_SEGMENT_GENERA	Definition of contact surface from parts, elements, box.
*SET_SHELL_ADD	Define a shell set by combining shell sets.
*SET_SHELL_COLUMN	Define a set of shell elements with optional identical or unique attributes.
*SET_SHELL_LIST(TITLE)	Define a set of shell elements with optional identical or unique attributes.
*SET_SHELL_LIST_GENEI	Define a set of shell elements with optional identical or unique attributes.
*SET_SOLID(TITLE)	Define a set of solid elements.
*SET_SOLID_ADD	Define a solid set by combining solid sets.
*SET_SOLID_GENERAL	
*SET_SOLID_GENERATE(	Generate a block of solid element IDs between a starting ID and an ending ID.
*SET_TSHELL(TITLE)	Define a set of thick shell elements.
*SET_TSHELL_GENERAL	

Card	Description
*SET_TSHELL_GENERATE	Generate a block of thick shell element IDs between a starting ID and an ending ID.

Nastran Cards

When reading input decks that were not created in HyperMesh, an attempt is made to create two sets for each set found: one containing elements and one containing nodes. You can delete the unnecessary set. Sets that are created are maintained as node or element sets by using the $HMSET comment cards.

Card	Description
AECOMP	Defines a component for use in monitor point definition or external splines.
AECOMPL	Defines a component for use in aeroelastic monitor point definition or external splines as a union of other components.
AELIST	Defines a list of aerodynamic elements or grid ID’s.
BCPROP	Defines a 3D contact region by element properties.
BNDFREE1	
BOLT	Defines a rigid bolt by a set of MPC constraints.
BOLTLD	
CSUPER	Defines the grid or scalar point connections for identical or mirror image superelements or superelements from an external source.
CSUPEXT	Assigns exterior points to a superelement.
ERPPNL	Defines one or more panels by referencing sets of elements or properties.
PANEL	Defines one or more panels by referencing sets of grid points, elements or properties.
ROTORG	Specifies grids that compose the rotor line model.
RSPINR	Specifies the relative spin rates between rotors for complex eigenvalue, frequency response, and static analysis. **Note:** Supported in conjunction with ROTORG.
RSPINT	Specifies rotor spin rates for Transient analysis.

Card	Description
	Note: Supported in conjunction with ROTORG.
SEBNDRY	Defines a list of grid points in a partitioned superelement for the automatic boundary search between a specified superelement or between all other superelements in the model.
SEBSET1	Defines boundary degrees-of-freedom to be fixed (b-set) during generalized dynamic reduction or component mode calculations.
SECSET1	Defines boundary degrees-of-freedom to be free (c-set) during generalized dynamic reduction or component mode synthesis calculations.
SEQSET1	Defines the generalized degrees-of-freedom of the superelement to be used in generalized dynamic reduction or component mode synthesis.
SESET	Defines interior grid points for a superelement.
SET	Defines a set of element or grid point numbers to be plotted. **Note:** Node and element sets supported with the THRU option.
SET1	Defines a list of structural grid points or element identification numbers.
SET3	Defines a list of grids, elements or points.
SEUSET1	Defines a degree-of-freedom set for a superelement.

OptiStruct Cards

OptiStruct sets are represented in HyperMesh as entity sets. The sets can be composed of grids, elements, design variables, MBD entities, mode numbers, frequencies or times for reference by other input definitions. In addition to the definition of entity sets through the explicit selection of the constituents, it is possible to define a set of nodes or a set of elements through a combination of formulaic expressions.

Sets that are created in HyperMesh are maintained on I/O by using the $HMSET comment cards.

Card	Description
BORE	Defines the surface, coordinate system, and parameters required to request Bore Deformation output for Static Analysis Subcases. **Note:** Bulk Data Entry

Card	Description
ERPPNL	Defines one or more sets of elements as panels for equivalent radiated power output for a frequency response analysis of a coupled fluid-structural model. **Note:** Bulk Data Entry
MBDCRV	Defines an ordered list of grids as a Multibody Deformable Curve. **Note:** Bulk Data Entry
MBDSRF	Defines a multibody deformable surface. **Note:** Bulk Data Entry
MBPCRV	Defines a Multi-body Parametric Curve using node sets. **Note:** Bulk Data Entry
PANEL	Defines up to four sets of grid points or elements as panels for panel participation output for a frequency response analysis of a coupled fluid-structural model. **Note:** Bulk Data Entry
PANELG	Defines a set of grid points and/or elements as generic panel. **Note:** Bulk Data Entry
ROTORG	Specifies grids that determine the Rotor Line model. **Note:** Bulk Data Entry
RSPINR	Defines the relative spin rates between rotors during a rotor dynamic analysis in Modal Complex Eigenvalue or Frequency Response solution sequences. **Note:** Bulk Data Entry

Card	Description
SET	Defines a set of grids, elements, design variables, MBD entities, mode numbers, frequencies or times for reference by other input definitions. **Note:** Bulk Data Entry Sets of integer and real values are supported as entity sets.
SET3	Defines a set of grids or elements. **Note:** Bulk Data Entry

PAM-CRASH Cards

During import, entity sets are automatically generated. PAM-CRASH cards with general entity selection generate entity sets.

Card	Description
GES /	
GROUP /	Keyword selection. **Note:** GROUP / card and general entity selection (GES) are mapped as set of sets. A set is created if only one keyword is used. A set of sets will be created: • If the definition uses more than one keyword. • Unresolved groups are used in the definition. • More than one `GRP` keyword is present in the definition. • A GROUP definition is always implemented as a set of set.

Permas Cards

Card	Description
ESET	Definition of new element sets. An element set may be defined by a list of element numbers or other element set names or using some generation rules.

Card	Description
ESETBIN	Definition of element set bins. An element set bin is defined by a list of element set names.
NSET	Definition of new node sets. A node set may be defined by a list of node numbers or other node set names or using some generation rules.
NSETBIN	Definition of node set bins. A node set bin is defined by a list of node set names.
SFSET	Definition of new surface sets. A surface set may be defined by a list of surface numbers or other surface set names or using some generation rules.

Radioss Cards

Sets of different types but with the same ID are supported.

Card	Description
/GRBEAM	Defines the beam groups. **Note:** Block Format Keyword
/GRBEAM/BEAM	Defines the beam number groups. **Note:** Block Format Keyword
/GRBEAM/BOX	Defines the beam groups which are inside the box. **Note:** Block Format Keyword All elements having all nodes inside the box or on its surface are selected.
/GRBEAM/BOX2	Defines the beam groups which are inside the box. **Note:** Block Format Keyword All elements with at least one node inside the box or on its external surface are selected.
/GRBEAM/GRBEAM	Defines the beam groups which are selected from other beam groups. **Note:** Block Format Keyword

Card	Description
/GRBEAM/MAT	Defines the beam material groups which belong to given materials. **Note:** Block Format Keyword
/GRBEAM/PART	Defines the beam part groups which belong to given parts. **Note:** Block Format Keyword
/GRBEAM/PROP	Defines the beam property groups which belong to given properties. **Note:** Block Format Keyword
/GRBEAM/SUBSET	Defines the beam subset groups which belong to givens subsets. **Note:** Block Format Keyword
/GRBRIC	Defines the brick groups. **Note:** Block Format Keyword
/GRBRIC/BOX	Defines the brick box groups which are inside the box. **Note:** Block Format Keyword All elements having all nodes inside the box or on its surface are selected.
/GRBRIC/BOX2	Defines the brick box groups which are inside the box. **Note:** Block Format Keyword All elements with at least one node inside the box or on its external surface are selected.
/GRBRIC/BRIC	Defines the brick number groups. **Note:** Block Format Keyword
/GRBRIC/GRBRIC	Defines the brick groups which are selected from other brick groups.

Card	Description
	Note: Block Format Keyword
/GRBRIC/MAT	Defines the brick material groups which belong to given materials. **Note:** Block Format Keyword
/GRBRIC/PART	Defines the brick part groups which belong to given parts. **Note:** Block Format Keyword
/GRBRIC/PROP	Defines the brick property groups which belong to given properties. **Note:** Block Format Keyword
/GRBRIC/SUBSET	Defines the brick subset groups which belong to given subsets. **Note:** Block Format Keyword
/GRNOD	Defines the node groups. **Note:** Block Format Keyword
/GRNOD/BOX	Defines the node box groups which are inside the box. **Note:** Block Format Keyword
/GRNOD/GENE	Defines the node groups. **Note:** Block Format Keyword
/GRNOD/GRBEAM	Defines the node groups which are selected from beam groups. **Note:** Block Format Keyword
/GRNOD/GRBRIC	Defines the node groups which are selected from brick groups.

Card	Description
	Note: Block Format Keyword
/GRNOD/GRQUAD	Defines the node groups which are selected from quad groups. **Note:** Block Format Keyword
/GRNOD/GRSH3N	Defines the 3 node shell groups which are selected from 3-node shell groups. **Note:** Block Format Keyword
/GRNOD/GRSHEL	Defines the 4 node shell groups which are selected from 4-node shell groups. **Note:** Block Format Keyword
/GRNOD/GRSPRI	Defines the node spring groups which are selected from spring groups. **Note:** Block Format Keyword
/GRNOD/GRTRUS	Defines the node truss groups which are selected from truss groups. **Note:** Block Format Keyword
/GRNOD/MAT	Defines the node material groups which belong to given materials. **Note:** Block Format Keyword
/GRNOD/NODE	Defines the node groups. **Note:** Block Format Keyword
/GRNOD/NODENS	Defines the unsortable node number groups. **Note:** Block Format Keyword
/GRNOD/PART	Defines the node part groups which belong to given parts.

Card	Description
	Note: Block Format Keyword
/GRNOD/PROP	Defines the node part groups which belong to given parts. **Note:** Block Format Keyword
/GRNOD/SUBSET	Defines the node subset groups which belong to given subset. **Note:** Block Format Keyword
/GRNOD/SURF	Defines the node surface groups which belong to given surfaces. **Note:** Block Format Keyword
/GRPART	Defines the part groups. **Note:** Block Format Keyword
/GRQUAD	Defines the quad groups. **Note:** Block Format Keyword
/GRQUAD/BOX	Defines the quad groups are inside the box. **Note:** Block Format Keyword All elements having all nodes inside the box or on its surface are selected.
/GRQUAD/BOX2	Defines the quad groups are inside the box. **Note:** Block Format Keyword All elements with at least one node inside the box or on its external surface are selected.
/GRQUAD/GRQUAD	Defines the quad groups which are selected from other quad groups. **Note:** Block Format Keyword

Card	Description
/GRQUAD/MAT	Defines the quad material groups which belong to given materials. **Note:** Block Format Keyword
/GRQUAD/PART	Defines the quad part groups which belong to given parts. **Note:** Block Format Keyword
/GRQUAD/PROP	Defines the quad property groups which belong to given properties. **Note:** Block Format Keyword
/GRQUAD/QUAD	Defines the quad number groups. **Note:** Block Format Keyword
/GRQUAD/SUBSET	Defines the quad subset groups which belong to given subsets **Note:** Block Format Keyword
/GRSH3N	Defines the 3-node shell groups. **Note:** Block Format Keyword
/GRSH3N/BOX	Defines the 3-node shell groups which are inside the box. **Note:** Block Format Keyword All elements having all nodes inside the box or on its surface are selected.
/GRSH3N/BOX2	Defines the 3-node shell groups which are inside the box. **Note:** Block Format Keyword All elements with at least one node inside the box or on its external surface are selected.
/GRSH3N/GRSH3N	Defines the 3-node shell groups which are selected from other 3-node shell groups.

Card	Description
	Note: Block Format Keyword
/GRSH3N/MAT	Defines the 3-node shell material groups which belong to given materials. **Note:** Block Format Keyword
/GRSH3N/PART	Defines the 3-node shell part groups which belong to given parts. **Note:** Block Format Keyword
/GRSH3N/PROP	Defines the 3 node shell groups which belong to given properties. **Note:** Block Format Keyword
/GRSH3N/SH3N	Defines the 3-node shell groups. **Note:** Block Format Keyword
/GRSH3N/SUBSET	Defines the 3-node shell subset groups which belong to given subsets. **Note:** Block Format Keyword
/GRSHEL	Defines the shell groups. **Note:** Block Format Keyword
/GRSHEL/BOX	Defines the shell box groups which are inside the box. **Note:** Block Format Keyword All elements having all nodes inside the box or on its surface are selected.
/GRSHEL/BOX2	Defines the shell box groups which are inside the box. **Note:** Block Format Keyword All elements with at least one node inside the box or on its external surface are selected.

Card	Description
/GRSHEL/GRSHEL	Defines the shell groups which are selected from other shell groups. **Note:** Block Format Keyword
/GRSHEL/MAT	Defines the shell material groups which belong to given materials. **Note:** Block Format Keyword
/GRSHEL/PART	Defines the shell part groups which belong to given parts. **Note:** Block Format Keyword
/GRSHEL/PROP	Defines the shell property groups which belong to given properties. **Note:** Block Format Keyword
/GRSHEL/SHEL	Defines the shell number groups. **Note:** Block Format Keyword
/GRSHEL/SUBSET	Defines the shell subset groups which belong to given subsets. **Note:** Block Format Keyword
/GRSPRI	Defines the spring groups. **Note:** Block Format Keyword
/GRSPRI/BOX	Defines the spring groups which are inside the box. **Note:** Block Format Keyword All elements having all nodes inside the box or on its surface are selected.
/GRSPRI/BOX2	Defines the spring groups which are inside the box.

Card	Description
	Note: Block Format Keyword All elements with at least one node inside the box or on its external surface are selected.
/GRSPRI/GRSPRI	Defines the spring groups which are selected from other spring groups. **Note:** Block Format Keyword
/GRSPRI/PART	Defines the spring part groups which belong to given parts. **Note:** Block Format Keyword
/GRSPRI/PROP	Defines the spring property groups which belong to given properties. **Note:** Block Format Keyword
/GRSPRI/SPRI	Defines the spring groups. **Note:** Block Format Keyword
/GRSPRI/SUBSET	Defines the spring subset groups which belong to given subsets. **Note:** Block Format Keyword
/GRTRUS	Defines the truss groups. **Note:** Block Format Keyword
/GRTRUS/BOX	Defines the truss box groups which are inside the box. **Note:** Block Format Keyword All elements having all nodes inside the box or on its surface are selected.
/GRTRUS/BOX2	Defines the truss box groups which are inside the box.

Card	Description
	Note: Block Format Keyword All elements with at least one node inside the box or on its external surface are selected.
/GRTRUS/GRTRUS	Defines the truss groups which are selected from other truss groups. **Note:** Block Format Keyword
/GRTRUS/MAT	Defines the truss material groups which belong to given materials. **Note:** Block Format Keyword
/GRTRUS/PART	Defines the truss part groups which belong to given parts. **Note:** Block Format Keyword
/GRTRUS/PROP	Defines the truss property groups which belong to given properties. **Note:** Block Format Keyword
/GRTRUS/SUBSET	Defines the truss subset groups which belong to given subsets. **Note:** Block Format Keyword
/GRTRUS/TRUS	Defines the truss groups. **Note:** Block Format Keyword
/LINE	Defines the line. **Note:** Block Format Keyword
/LINE/BOX	Defines the lines inside the box. **Note:** Block Format Keyword All elements having all nodes inside the box or on its surface are selected.

Card	Description
/LINE/BOX2	Defines the lines inside the box. **Note:** Block Format Keyword All elements with at least one node inside the box or on its external surface are selected.
/LINE/EDGE	Defines the selected edges from given surfaces as lines. **Note:** Block Format Keyword
/LINE/GRBEAM	Defines the lines selected from beam groups. **Note:** Block Format Keyword
/LINE/GRTRUS	Defines the lines selected from truss groups. **Note:** Block Format Keyword
/LINE/MAT	Defines the lines which belong to given materials. **Note:** Block Format Keyword
/LINE/PROP	Defines the lines which belong to given properties. **Note:** Block Format Keyword
/LINE/SURF	Defines the lines which belong to given surfaces. **Note:** Block Format Keyword
/SURF	Defines the surface definition. **Note:** Block Format Keyword
/SURF/BOX	Defines the surfaces inside the box.

Card	Description
	Note: Block Format Keyword All segments supported by solids, shells and 3-node shells with all nodes inside the box or on its external surface are selected (segments lying on solid elements are not considered).
/SURF/BOX/ALL	Defines the surfaces inside the box (include inner segments). **Note:** Block Format Keyword All segments supported by solids, shells and 3-node shells with all nodes inside the box or on its external surface are selected (segments lying on solid elements are not considered).
/SURF/BOX/EXT	Defines the surfaces which are inside the box (exclude inner segments). **Note:** Block Format Keyword All segments supported by solids, shells and 3-node shells with all nodes inside the box or on its external surface are selected (segments lying on solid elements are not considered).
/SURF/BOX2	Defines the surfaces inside the box. **Note:** Block Format Keyword All segments with at least one node inside the box or on its surface are selected.
/SURF/GRSHELL	Defines the surface selected from shell groups. **Note:** Block Format Keyword
/SURF/MAT	Defines the material surfaces which belong to given materials. **Note:** Block Format Keyword
/SURF/MAT/ALL	Defines the material surfaces which belong to given materials. **Note:** Block Format Keyword
/SURF/MAT/EXT	Defines the external material surfaces which belong to given materials.

Card	Description
	Note: Block Format Keyword
/SURF/PART	Defines the parts surfaces which belong to given parts. **Note:** Block Format Keyword
/SURF/PART/ALL	Defines the parts surfaces which belong to given parts. **Note:** Block Format Keyword
/SURF/PART/EXT	Defines the external parts surfaces which belong to given parts. **Note:** Block Format Keyword
/SURF/PROP	Defines the property surfaces which belong to given properties. **Note:** Block Format Keyword
/SURF/PROP/ALL	Defines the property surfaces which belong to given properties. **Note:** Block Format Keyword
/SURF/PROP/EXT	Defines the external property surfaces which belong to given properties. **Note:** Block Format Keyword
/SURF/SUBSET	Defines the subset surfaces which belong to given subsets. **Note:** Block Format Keyword
/SURF/SUBSET/ALL	Defines the subset surfaces which belong to given subsets. **Note:** Block Format Keyword
/SURF/SUBSET/EXT	Defines the external subset surfaces which belong to given subsets.

Card	Description
	Note: Block Format Keyword
/SURF/SURF	Defines the surface selected from other surfaces. **Note:** Block Format Keyword

Samcef Cards

Card	Description
.SEL NOE	Defines a set of grids.
.SEL MAI	Defines a set of elements.

Slip Rings

Slip ring entities define sliprings.

LS-DYNA Cards

Card	Description
*ELEMENT_SEATBELT_SL	Defines sliprings.

Solver Masses

Solver masses are used to model lumped mass finite element mesh.

Solver mass entities have a display state, on or off, which controls the display of these entities in the graphics area. The display state of a Solver Mass entity can be controlled using the icon next to the entity in the Model Browser.

Solver Mass entities also have an active and export state. The active state of a Solver Mass entity controls its display state, its listing in the Model Browser, and any of its views. If a Solver Mass entity is active, then its display state is available to be turned on or off and it is listed in the Model Browser along with its views. If a Solver Mass entity is inactive, then its display state is turned off permanently and it is not listed in the Model Browser or any of its views.

The export state of a Solver Mass entity controls whether or not this entity is exported when the custom export option is used. The all export option is not affected by the export state. The active and export states of Solver Mass entities can be controlled using the Entity State Browser.

Solver Mass entities have also a Review mode which is accessible from the context menu of the entity.

Radioss Cards

Card	Description
/ADMAS/0	Mass is added to each node of node group. Mass Type=0 in Solver Mass Entity Editor.
/ADMAS/1	Mass/N is added to each node of node group. N being the total number of nodes in the node group. Mass Type=1 in Solver Mass Entity Editor.
/ADMAS/2	Mass/Area - additional surface mass applied on the shell area. Mass Type=2 in Solver Mass Entity Editor.
/ADMAS/3	Additional mass distributed on the part-group. Mass Type=3 in Solver Mass Entity Editor.
/ADMAS/4	Final mass distributed on the part-group. Mass Type=4 in Solver Mass Entity Editor.
/ADMAS/5	Mass is added to each single node. Mass Type=5 in Solver Mass Entity Editor.
/ADMAS/6	Additional mass distributed on each single part. Mass Type=6 in Solver Mass Entity Editor.
/ADMAS/7	Final mass distributed on each single part. Mass Type=7 in Solver Mass Entity Editor.

Note: A special Review is available for Solver Mass entity with Mass Type=5. It displays the mass distribution on all the nodes referenced in the Solver Mass, with a dynamic legend:

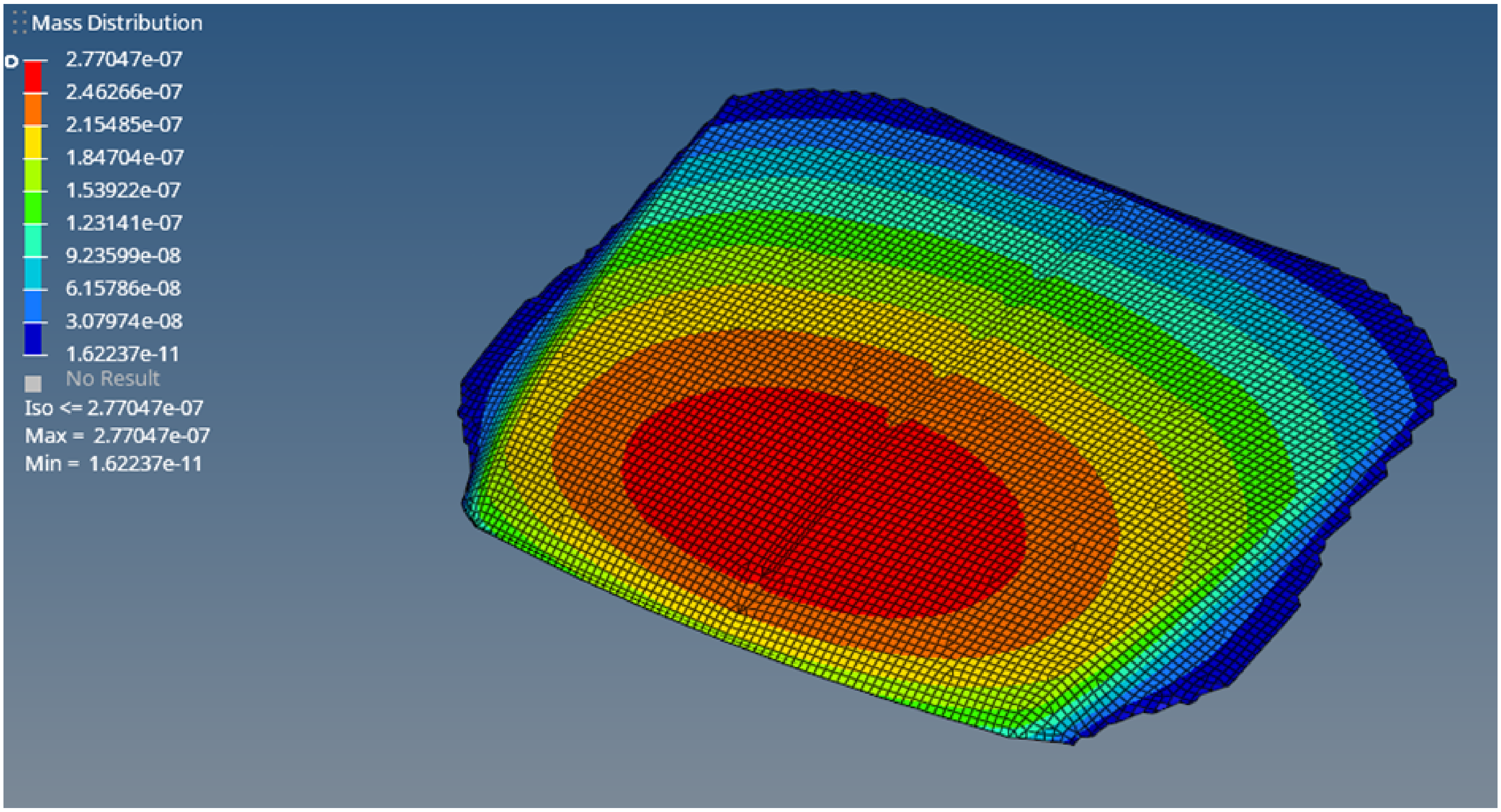


Figure 141:

Tables

Table entities collect data points.

Abaqus Cards

Card	Description
*DISTRIBUTION ANGLE	Creates a *Distribution table for angles. **Note:** Can be referenced in *Orientation. A default angle must be provided, which will be used for elements without angle values.
*DISTRIBUTION DRAPE	Creates a *Distribution table for thickness and angle.

Card	Description
	Note: Contains both the angle and thickness, but during export separate *Distribution keywords are written. Can be referenced by *Shell Section with composite ply information. A default angle and thickness must be provided, which will be used for elements without thickness and angle values.
*DISTRIBUTION OFFSET	Creates a *Distribution table for offset. The offset values define the offset and thickness of elements. **Note:** Can be referenced by *Shell Section using the thickness option. A default offset must be provided, which will be used for elements without offset values.
*DISTRIBUTION ORIENTATION	Creates a *Distribution table for orientation. The orientation values define the orientation of elements. **Note:** Can be referenced by *Orientaion. A default orientation value must be provided, which will be used for elements without orientation values.
*DISTRIBUTION THICKNESS	Creates a *Distribution table for thickness. The thickness values define the thickness of elements. **Note:** Can be referenced by *Shell Section using the thickness option. A default thickness value must be provided, which will be used for elements without thickness values.
*SPECTRUM	Creates a *Spectrum table for *RESPONSE SPECTRUM ANALYSIS. This value can be defined as a function of spectrum, frequency, and damping.

Card	Description
	Note: Can be referenced by *RESPONSE SPECTRUM.

Tags

Tag entities tag a piece of information, called the body, onto a node, element, line, surface, point, or solid within the model.

Limitations

Tags are automatically deleted when:

- A tag is on a surface edge and the edge is split, toggled (in either direction), or surface normals are adjusted.
- A tag is on a free line and the free line is split.
- A tag is on a surface vertex and all but one (owning parent of the vertex) of the associated surfaces are deleted, or the vertex is toggled (in either direction).

Terminations

Termination entities provides an alternative way to stop the calculation before the termination time is reached.

Termination entities have a display state, on or off, which controls the display of the entity in the graphics area. The graphic representation depends on the referenced entity defined in the Termination card. For example, *TERMINATION_BODY refers to a component. This component will be shown/hidden when the Termination will be shown/hidden. The display state of Termination entity can be controlled using the icon next to the entity in the Model Browser.

Termination entities also have an active and export state. The active state of a Termination entity controls the display state of the Termination entity and the listing of the Termination entity in the Model Browser and any of its views. The export state of a Termination entity controls whether or not that entity is exported when the custom export option is used. The all export option is not affected by the export state of a Termination entity. The active and export states of Terminations entities can be controlled using the Entity State Browser.

LS-DYNA Cards

Card	Description
*TERMINATION_BODY	Termination based on rigid body displacements.

Card	Description
*TERMINATION_CONTACT	Termination when the magnitude of the contact interface resultant force is zero.
*TERMINATION_CURVE	Termination when the load curve value returns to zero.
*TERMINATION_DELETED	Termination when the number of deleted shells for a specified part exceeds the defined value.
*TERMINATION_DELETED	Termination when the number of deleted solids for a specified part exceeds the defined value.
*TERMINATION_NODE	Termination when the position of the specified node reaches either the maximum or minimum values defined.
*TERMINATION_SENSOR	Termination when the switch condition defined in *SENSOR_SWITCH is met.

Titles

Title entities attach a title box with text to the modeling window, or to a node, element, load, or system.

A title entity will move with the model if it is attached to a node, element, load, or system. The title entity is static if it is attached to the modeling window.

Transformations

Transformation entities define solver transformations, and are used to define a transformation sequence in a Position entity, to be applied on a set of nodes or on a SolverSubmodel.

Abaqus

Transformation entities hold the transformation information that is provided in the datalines of the *NMAP keyword.

Card	Description
*NMAP	Maps nodes from one coordinate system to another, and rotates, translates, or scales the nodal coordinates.

Card	Description
	Note: More than one transformation can be applied on a given node set using additional `*NMAP` keywords. A given transformation can be applied multiple times using separate `*NMAP` keywords for each required instance. The coordinates for **Point A** and **Point B** cannot be the same. If the **Magnitude** is greater than 0, the nodes will be translated in the direction of the vector AB; if the Magnitude is less than 0, the nodes will be translated in the opposite direction of the vector AB. If the Magnitude is equal to 0, an error message will be issued when the transformation is applied. A positive **angle** results in a counter clockwise rotation; a negative angle results in a clockwise rotation. If the Angle is equal to 0, an error message will be issued when the transformation is applied. If the **scale** factor along the X, Y, and Z axes are uniform, then the **Uniform Scale** checkbox can be selected and one **Scale size** can be specified. If the scale factor is different along each axis, then it can be entered separately by clearing the Uniform scale checkbox.

Radioss

Transformations entities hold the Radioss keywords /TRANSFORM.

Supported Card	Solver Description
//SUBMODEL	Defines a part of the model with separate definition of numeration, unit system and Radioss version.
/TRANSFORM/TRA	Defines a translation for a set of nodes or a submodel. A transformation cannot be applied on a node set and a submodel using the same keyword.
/TRANFORM/ROT	Defines a rotation for a set of nodes or a submodel. A transformation cannot be applied on a node set and a submodel using the same keyword.
/TRANFORM/SCA	Defines scaling factors in X,Y,Z directions for a set of nodes.

Supported Card	Solver Description
/TRANSFORM/SYM	Defines a symmetry on a set of nodes, normal to the plane defined by a vector.

Morphing Entities

Entities used for morphing the shape of FE model.

Domains

Domain entities divide the model into different domains during morphing.

Each domain contains either elements (for 1D, 2D, 3D, or general domains), a series of nodes (for edge domains), or a group of nodes (for global domains).

The shape of a domain changes when the handles associated with a domain move, which in turn changes the position of the nodes inside those domains.

Domains do not have an active or export state.

Domain Types

1D Domain
Group of 1D elements, such as bars and rigid elements.

2D Domain
Group of shell elements.

3D Domain
Group of solid elements.

General Domain
Group of any element type.

Edge Domain
Series of nodes that are commonly found along the edges of 2D and 3D domains.

Global Domain
Group of nodes.

Handles

Handle entities control the shape of domains during morphing.

A handle can be associated with any domain, but must be associated with one and only one domain. When a handle associated with a 1D, 2D, 3D, general, or edge domain is moved, it moves the nodes within that domain along with the nodes in any other domain that the handle is touching. A handle associated with the global domain affects only the nodes in that domain. You can make large-scale changes to your model by moving the handles on the global domain and make small-scale changes to your model by moving the handles on local (1D, 2D, 3D, general, and edge) domains. You can also make both large-scale and small-scale changes to your model and have their effects combined in a logical manner.

Handles can also be dependent on other handles, either in the global domain or in local domains. This allows handles to control, but not restrict, the movements of other handles. A handle dependent on another handle inherits the perturbations applied to its parent handle which are added to any perturbations applied to the dependent handle directly. This allows you to set up a hierarchical control system for the handles in the model, such as making three handles which control the positions of three different holes dependent on a handle in the middle of all of them. When the middle handle is moved, the other handles move the same way, which in turn move all of the holes. Moving any one of the dependent handles moves only the hole it controls. You may also make a handle dependent on multiple handles. The dependent handle will be perturbed an amount equal to the average perturbation applied to the parent handles based on distances between them.

Global handles are red and are associated to global domains. Local handles are orange and are associated to local domains. Both global and local handles can have dependent handles which are of varying colors and sizes.

Handles do not have an active or export state.

Morph Constraints

Morph constraint entities restrict the movements of nodes or force compliance with dimensional requirements during morphing.

Morph constarints do not have an active or export state.

Morph Constraint Types

Fixed
: Selected nodes do not move when the mesh morphs around them.

Cluster
: Nodes move with the rest of the mesh, but remain fixed relative to each other. The cluster will move, but will also not stretch or become deformed.

Smooth
: When handles move on a selected edge domain, the edge morphs according to spline-based motion rather than linear motion, creating smooth edge curves instead of straight edge lines. You may also choose whether to apply the spline-based motion to the nodes directly, or to dependent handles between the morphed handles.

Along Vector
: Nodes will move only in the direction of a selected vector.

Along Line
: Nodes will move, but will either move along the length of a selected line or will be bounded by the selected line.

On Plane
: Nodes will move, but will either move across the surface of a selected plane, or be bounded by the selected plane.

On Surface

Nodes will move, but will either move across a selected surface, or be bounded by the selected surface.

On Elements

Nodes move, but will either move along the surface of a selected mesh, or be bounded by the selected mesh. This is similar to the along surface option but without requiring surface geometry.

On Equation

Nodes will move, but will either move across the surface of a function, or be bounded by the surface of the function. The surface of a function is defined at the threshold where f(x,y,z) is equal to zero.

Along dofs

Nodes will move, but can have one, two, or all three of their x, y, or z (or r, theta, or phi) coordinates fixed for any given coordinate system, local or global.

Match Elems

Two groups of elements will be forced to match each other regardless of the morphing applied to the nodes on either group. The matching can be forced relative to the normal direction of the elements, relative to a vector, or by selecting nodes to orient one group of elements relative to another.

> **Note:** Three or more groups of elements can be joined together by using this constraint more than once.

Tangency

Continuous tangency can be enforced across a pair of edge or 2D domains. Also, edge domains may have their ends constrained to defined angles, be joined in a master-slave tangency, or have one edge "attached" to another.

Length

Used in conjunction with one or more morphing shapes, to enforce a specific, maximum, or minimum distance as measured along the nodes in a specified node list.

Angle

Used in conjunction with one or more morphing shapes, to control the angle of three selected nodes. These nodes will move, even relative to each other, but the angle they form remains either fixed, above a certain value, or below a certain value depending on your choice.

Radius

Used in conjunction with one or more morphing shapes, to control the radius of a given edge domain.

The radius is measured using an edge domain and one of the following methods for finding the center of curvature: an axis, a line, a node, or inferred from the plane of the edge domain.

For example, if an edge domain lies along a cylinder which has been cut by a plane or planes that are not perpendicular to the axis of the cylinder, the axis of the cylinder can be selected as the center for the radius of the domain. The radius of the domain would then be measured with respect to the assigned axis.

Arc Angle

Used in conjunction with one or more morphing shapes, to control the arc angle of a given edge domain. As for the radius option, the arc angle is measured using an edge domain and one of the following methods for finding the center of curvature: an axis, a line, a node, or inferred from the plane of the edge domain.

Area

Used in conjunction with one or more morphing shapes, to control the area of a number of elements. The selected elements will deform during morphs, but their total surface area remains either constant, above a certain value, or below a certain value, depending in your choice.

Volume

Used in conjunction with one or more morphing shapes, to control the volume of a number of elements. The selected elements will deform during morphs, but the volume of the selected elements remains constant, below a maximum value, or above a minimum value.

Mass

Used in conjunction with density data supplied by a property card, to control the mass of a number of elements. The selected elements will deform during morphs, but the total mass of the selected elements remains equal to, above, or below a specified number, even as the elements change shape during morphing.

Morph Volumes

Morph volume entities are highly deformable six-sided prisms which surround a portion of the FE mesh, and can be used to manipulate a mesh by manipulating the shape of the morph volume.

Morph volumes are very malleable; the length and curvature of each edge can be modified independently of the others, and adjacent morph volumes can be linked through various tangency conditions. This malleability allows you to enclose a given mesh with morph volumes, alter the morph volumes to fit your model, and then change the shape of your model by modifying the morph volumes.

Morph volumes support tangency between adjoining edges and allow for multiple control points along their edges. Handles placed at the corners and along the edges of the morph volumes allow for the morphing of the morph volumes which in turn morphs the mesh inside the morph volumes.

Morph volumes can morphed independently of the enclosed mesh. For example, to change the shape of your morph volumes without affecting the mesh you can set the morph volumes to be inactive. This allows you to use all of the morphing capabilities to modify the shape and position of your morph volumes to better fit your mesh. Then you can switch the morph volumes back to being active and use them to morph the mesh.

Morph volumes will only morph the mesh for nodes that have been registered. In some cases, nodes within morph volumes are automatically registered when the morph volumes are created, while in others only the selected nodes or nodes on selected elements are registered. If the morph volumes do not appear to be morphing nodes inside them, you may need to register those nodes.

Morph volumes do not have an active or export state.

Shapes

Shape entities are collections of handle and/or node perturbations from the initial configuration of the FE mesh before morphing.

When you morph your model, the morph is stored internally as a collection of perturbations which you can then undo, redo, and/or save as a shape.

Shapes do not have an active or export state.

Symmetries

Symmetry entities define planes of symmetry within a model so that morphs can be applied in a symmetric fashion.

Symmetries do not have an active or export state.

There are two basic symmetry groups: reflective and non-reflective. Symmetries can be combined, but you must be careful not to create confusing symmetrical arrangements. Symmetries can also be applied to unconnected domains. In this case, the symmetric handle linking works the same as that for connected domains, but the influences between handles and nodes for non-reflective symmetries do not extend across to all domains.

Reflelctive Symmetires

Reflective symmetries link handles in a symmetric fashion so that the movements of one handle will be reflected and applied to the symmetric handles. You can also use reflective symmetries to reflect morphs performed on domains when using the alter dimensions.

Reflective symmetries are one plane, two plane, three plane, and cyclical.

One Plane
: A mirror is placed at the origin perpendicular to the selected axis (default = x-axis).

Two Plane
: Two mirrors are placed at the origin perpendicular to the selected axis and the subsequent axis (that is x and y, y and z, z and x) (default = x and y-axis).

Three Plane
: Three mirrors are placed at the origin perpendicular to all three axes.

Cyclical
: Two mirrors are placed along the selected axis (default = z-axis) and running through the origin with a given angle in between that is a factor of 360. The result is a wedge that is reflected a certain number of times about the selected axis.

Reflective symmetries can be defined as either unilateral or multilateral and either approximate or enforced.

Unilateral Symmetries

One side governs the other, but not vice versa.

For example, handles created and morphs applied to handles on the positive side of the symmetry are reflected onto the other side or sides of the symmetry, but handles created or morphs applied to handles on the other side or sides of the symmetry are not reflected.

Multilateral Symmetries

All sides govern all other sides.

For example, a handle created or a morph applied to a handle on any side is reflected to all the other sides.

Approximate Symmetries

Contain handles that are not symmetric to other handles. This option is best for asymmetrical, but similar, domains or for a cyclical symmetry applied to a mesh that sweeps through an arc but not a full circle.

For example, handles created on any side of the symmetry are not reflected to the other sides.

Enforced Symmetries

Cannot contain handles that are not symmetric on all other sides.

For example, handles created or deleted on any side of the symmetry are created or deleted on the other sides so that the symmetry is maintained. When a reflective symmetry is created with the enforced option, additional handles may also be created to meet the enforcement requirements.

> **Note:** Handles created due to the enforced symmety may not be located on any mesh, however, they will always be assigned to the nearest domain and will affect nodes in that domain.

Non-Reflective Symmetries

Non-reflective symmetries change the way that handles influence nodes as well as link the symmetric handles so that the movement of one affects the others.

The handles for a domain with non-reflective symmetry will act as if they are the shape of the symmetry type. For instance, a domain with linear symmetry causes handle movements to act on the domain as if the handle was a line in the direction of the x-axis. A domain with circular symmetry causes handle movements to act on the domain as if the handle was a circle centered around the z-axis. The edges of a domain affect how influences between handles and nodes are calculated. Non-reflective symmetries work best for domains that are shaped like the symmetry type and have a regular mesh. For example, a circular symmetry works best for a round domain with a concentric mesh.

Non-reflective symmetries are linear, circular, planar, radial 2D, cylindrical, radial + linear, radial 3D, and spherical.

Linear

Handle acts as a line drawn through the handle location parallel to the selected axis (default = x-axis).

Circular

Handle acts as a circle drawn through the handle position about the selected axis (default = z-axis).

Planar

Handle acts as a plane drawn through the handle location perpendicular to the selected axis (default = x-axis).

Radial 2D

Handle acts as a ray drawn through the handle position originating from and extending perpendicular to the selected axis (default = z-axis).

Cylindrical

Handle acts as a cylinder drawn through the handle position about the selected axis (default = z-axis).

Radial + Linear

Handle acts as a plane drawn through the handle position extending from the selected axis (default = z-axis).

Radial 3D

Handle acts as a ray drawn through the handle position originating from origin.

Spherical

Handle acts as a sphere drawn through the handle position centered on the origin.

Optimization Entities

Entities used to set up optimization problems.

Design Variables

Design variable entities define and store design variables for optimization problems.

Design variables do not have a display state in the modeling window.

Nastran Cards

Card	Description
DESVAR	Defines a design variable for design optimization. **Note:** Design variable definition.
DLINK	Relates one design variable to one or more other design variables. **Note:** Design variable link.

OptiStruct Cards

Card	Description
DCOMP	Defines manufacturing constraints for composite sizing optimization. **Note:** Bulk Data Entry
DESVAR	Defines a design variable. **Note:** Bulk Data Entry
DSHAPE	Defines parameters for the generation of free-shape design variables. **Note:** Bulk Data Entry
DSHUFFLE	Defines parameters for the generation of composite shuffling design variables.

Card	Description
	Note: Bulk Data Entry
DSIZE	Defines parameters for the generation of free-size design variables. **Note:** Bulk Data Entry
DTPG	Defines parameters for the generation of topography design variables. **Note:** Bulk Data Entry
DTPL	Defines parameters for the generation of topology design variables. **Note:** Bulk Data Entry
DVGRID	Defines the relationship between a design variable and a grid point location. **Note:** Bulk Data Entry Exported in large field format by both the `optistruct` and `optistructlf` templates.

Radioss Cards

Card	Description
DESVAR	Defines an optimization design variable (max. or min.).
DSHAPE	Design variable for Free-Shape optimization.
DSIZE	Design variable for Free-Size optimization.
DTPG	Design variable for Topography optimization.
DTPL	Design variable for Topology optimization.
DVPREL1	Defined linearly, relates a design variable to an analysis model property using the equation.

Card	Description
DVGRID	Defines the relationship between design variable and grid point location.

Design Variable Links

Design variable link entities define links between design variables for optimization problems.

Design variable links do not have a display state in the modeling window.

Nastran Cards

Card	Description
DLINK	Relates one design variable to one or more other design variables.
DLINK2	

OptiStruct Cards

Card	Description
DLINK	Defines a link between one design variable and one or more other design variables.
DLINK2	Defines a link of one design variable to one or more other design variables defined by a `DEQATN` card. The equation inputs come from the referenced `DESVAR` values and the constants defined on the `DTABLE` card.

Design Variable Property Relationships

Design variable property relationship entities define relationships between design variables and properties for optimization problems.

Design variable property relationships do not have a display state in the modeling window.

Nastran Cards

Card	Description
DVCREL1	Defines the relation between a connectivity property and design variables.

Card	Description
	Note: Generic Property
DVCREL2	Defines the relation between a connectivity property and design variables with a user-supplied equation.
DVMREL1	Defines the relation between a material property and design variables.
DVMREL2	Defines the relation between a material property and design variables with a user-supplied equation.
DVPREL1	Defines the relation between an analysis model property and design variables. **Note:** Generic Property
DVPREL2	Defines the relation between an analysis model property and design variables with a user-supplied equation. **Note:** Function Property

OptiStruct Cards

Card	Description
DVCREL1	Linearly relates a design variable to an analysis model element property using the equation: $MP_i = C_0 + \sum_i COEF_i \cdot DVID_i$ `CBAR`, `CELAS2`, `CELAS4`, `CMASS2`, `CMASS4`, `CDAMP2`, `CDAMP4`, `CONM1`, `CONM2`, `CONROD`, `CQUAD4`, `CTRIA3`, `CQUAD8`, `CTRIA6` elements can be selected. **Note:** Bulk Data Entry
DVCREL2	Relates design variables to an analysis model element property using a relationship defined by a `DEQATN` card. The equation inputs come from the referenced DESVAR values and constants defined on a `DTABLE` card. `CBAR`, `CELAS2`, `CELAS4`, `CMASS2`, `CMASS4`, `CDAMP2`, `CDAMP4`, `CONM1`, `CONM2`, `CONROD`, `CQUAD4`, `CTRIA3`, `CQUAD8`, `CTRIA6` elements can be selected.

Card	Description
	Note: Bulk Data Entry
DVLREL1	Linearly relates a design variable to an analysis model element.
DVLREL2	Relates a design variable to an analysis model loading using a relationship defined by a DEQATN card. The equation inputs come from the referenced DESVAR values and the constants defined on the DTABLE card.
DVMREL1	Linearly relates a design variable to an analysis model material property using the equation: $MP_i = C_0 + \sum_i COEF_i \cdot DVID_i$ MAT1, MAT2, MAT8 and MAT9 materials can be selected. **Note:** Bulk Data Entry
DVMREL2	Relates design variables to an analysis model material property using a relationship defined by a `DEQATN` card. The equation inputs come from the referenced `DESVAR` values and constants defined on a `DTABLE` card. MAT1, MAT2, MAT8 and MAT9 materials can be selected. **Note:** Bulk Data Entry Requires a dequation definition.
DVPREL1	Linearly relates a design variable to an analysis model property using the equation: $P_i = C0 + \sum_i COEF_i * DDVI_i$ `PBAR`, `PBARL`, `PBEAM`, `PBEAML`, `PBUSH`, `PCOMP`, `PCOMPG*`, `PCOMPP`, `PDAMP`, `PELAS`, `PMASS`, `PROD`, `PSHEAR`, `PSHELL`, `PVISC` properties and `PLY` entities can be selected. **Note:** Bulk Data Entry For `PCOMPG`, either global plies or property specific plies may be selected.
DVPREL2	Relates a design variable to an analysis model property using a relationship defined by a `DEQATN` card. The equation inputs come from the referenced `DESVAR` values and the constants defined on the `DTABLE` card. `PBAR`, `PBARL`, `PBEAM`, `PBEAML`, `PBUSH`, `PCOMP`, `PCOMPG*`, `PCOMPP`, `PDAMP`, `PELAS`, `PMASS`, `PROD`, `PSHEAR`, `PSHELL`, `PVISC` properties and `PLY` entities can be selected.

Card	Description
	Note: Bulk Data Entry Requires a dequation definition. For PCOMPG, either global plies or property specific plies may be selected.

Discrete Design Variables

Discrete design variable entities define and store discrete design variables for optimization problems.

Discrete design variables do not have a display state in the modeling window.

Nastran Cards

Card	Description
DDVAL	Define real, discrete design variable values for discrete variable optimization. Supported as a ddval entity.

OptiStruct Cards

Card	Description
DDVAL	Define real, discrete design variable values for discrete variable optimization or to define relative rotor spin rates in rotor dynamics.

Card	Description
	Note: Bulk Data Entry

Optimization Responses

Optimization response entities define and store model responses for optimization problems.

Optimization responses do not have a display state in the modeling window.

Nastran Cards

Card	Description
DRESP1	Defines a set of structural responses that is used in the design either as constraints or as an objective. **Note:** Generic response
DRESP2	Defines equation responses that are used in the design, either as constraints or as an objective. **Note:** Function response

OptiStruct Cards

Card	Description
DRESP1	A response or a set of responses that are the result of a design analysis iteration. These responses can be used as a design objective or as design constraints. **Note:** Bulk Data Entry
DRESP2	When a desired response is not directly available from OptiStruct, it may be calculated using `DRESP2`. This response can be a functional combination of any set of responses resulting from the design analysis iteration. Responses defined in this manner can be used as design objectives or constraints. The

Card	Description
	DRESP2 card identifies the equation to use for the response relationship and the input values to evaluate the response function. **Note:** Bulk Data Entry
DRESP3	When a desired response is not available from OptiStruct, either directly or via equations, it may be calculated through external user-supplied functions implemented in shared/dynamic libraries or external files. The DRESP3 card identifies the external function to be called and defines the parameters to be transferred to that function. **Note:** Bulk Data Entry
DSYSID	Defines responses and their target values for a system identification problem. **Note:** Bulk Data Entry
MODEWEIGHT	Defines a multiplier for computed eigenvalues that are to be used in the calculation of the "weighted reciprocal eigenvalue" and "combined compliance index" optimization responses. **Note:** Subcase Information Entry
PSD STRAIN	
PSD STRESS	
RMS STRAIN	
RMS STRESS	
WEIGHT	Defines a multiplier for computed eigenvalues that are to be used in the calculation of the "weighted reciprocal eigenvalue" and "combined compliance index" optimization responses.

Card	Description
	Note: Subcase Information Entry

Radioss Cards

Card	Description
DRESSP1	Defines a response (mass, displacement, stress, velocity, and so on) or a set of responses.
DRESP2	Design Response via Equations for Design Optimization.

Optimization Constraints

Optimization constraint entities define and store constraints on model responses for optimization problems.

Optimization constraints do not have a display state in the modeling window.

Nastran Cards

Card	Description
DCONADD	Defines the design constraints for a subcase as a union of `DCONSTR` entries. **Note:** Collects constraints
DCONSTR	Defines design constraints. **Note:** Constraint to define lower and upper bounds.
DESGLB	Selects the design constraints to be applied at the global level in a design optimization task. **Note:** Global constraint; belongs in the subcase section.
DESSUB	Selects the design constraints to be used in a design optimization task for the current subcase.

Card	Description
	Note: Constraint dependent on the load step; Belongs in the subcase section.

OptiStruct Cards

Card	Description
DCONADD	Creates a combination of several `DCONSTR` sets that can be referenced by a subcase. **Note:** Bulk Data Entry
DCONSTR	Defines design constraint upper and lower bounds where response is defined by `DRESP1`, `DRESP2`, and `DRESP3` cards. **Note:** Bulk Data Entry
DESGLB	Used before the first subcase statement, to select a constraint set that is not subcase dependent. **Note:** I/O Options and Subcase Information Entry
DESSUB	Used within a subcase definition, to select a constraint set that is subcase dependent. **Note:** Subcase Information Entry

Radioss Cards

Card	Description
DCOMP	Defines manufacturing constraints for composite sizing optimization.

Card	Description
DCONSTR	Defines constrains a design response by specifying its lower and upper bounds.

Optimization Equations

Optimization equation entities define and store equations for optimization problems.

Optimization equations do not have a display state in the modeling window.

Nastran Cards

Card	Description
DEQATN	Defines one or more equations for use in design sensitivity or p-element analysis. **Note:** Equations referenced on `DRESP2`, `DVPREL2`

OptiStruct Cards

Card	Description
DEQATN	Specifies one or more equations for use in optimization.

Card	Description
	Note: Bulk Data Entry

Radioss Cards

Card	Description
DEQATN	Specifies one or more equations for use in optimization.

Optimization Table Entries

Optimization table entry entities define and store table data for optimization problems.

Optimization table entires do not have a display state in the modeling window.

Nastran Cards

Card	Description
DTABLE	Defines a table of real constants that are used in equations. **Note:** Table entries referenced on `DRESP2`, `DVPREL2`

OptiStruct Cards

Card	Description
DTABLE	List of constants to be used in functions defined by `DEQATN`.

Card	Description
	Note: Bulk Data Entry

Objectives

Objective entities define and store an objective for an optimization problem.

Objectives do not have a display state in the modeling window.

Nastran Cards

Card	Description
DESOBJ	Selects the `DRESP1` or `DRESP2` entry to be used as the design objective. **Note:** Objective function, can be in or out of the load step; Belongs in the subcase section.
MAXMIN	Objective functions for maxmin problems.
MINMAX	Objective functions for minmax problems.

OptiStruct Cards

Card	Description
DESOBJ	Selects a single response definition as the objective function of an optimization, or to select system response definitions when the objective function is the least squares sum of these definitions. The `DESOBJ` command also indicates if this response is to be minimized or maximized. **Note:** Subcase Information Entry
MINMAX or MAXMIN	Selects normalized response or system identification definitions as the objective function for a "Minmax" or "Maxmin" optimization.

Card	Description
	Note: Subcase Information Entry

RADIOSS Cards

Card	Description
DESOBJ	Specifies a response as the objective function of an optimization.

Objective References

Objective reference entities define and store objective references for an optimization problem.

Objective references do not have a display state in the modeling window.

OptiStruct Cards

Card	Description
DOBJREF	Defines a response and its reference values for a minmax (maxmin) optimization problem.

Card	Description
	Note: Bulk Data Entry

Optimization Constraint Screenings

Optimization constraint screening entities define and store constraint screening data for optimization problems.

Optimization constraint screenings do not have a display state in the modeling window.

Nastran Cards

Card	Description
DSCREEN	Defines screening data for constraint deletion.

OptiStruct Cards

Card	Description
DSCREEN	Defines design response screening data. **Note:** Bulk Data Entry
SHAPEOPT	Constraint screenings.

Card	Description
	Note: Bulk Data Entry

Optimization Controls

Optimization control entities define and store controls for optimization problem run.

Optimization controls do not have a display state in the modeling window.

Nastran Cards

Card	Description
DOPTPRM	Overrides default values of parameters used in design optimization.

OptiStruct Cards

Card	Description
DOPTPRM	Defines design optimization parameters by overriding the defaults. **Note:** Bulk Data Entry If an unsupported argument is encountered on importing a DOPTPRM card, the data is stored as UNSUPPORTED_DOPTPRM on the DOPTPRM card. It is also possible to create an unsupported DOPTPRM card using the UNSUPPORTED_DOPTPRM option on the opticontrol card image.
DOPTPRM, NESLEXPD	Specifies the number of time steps retained for optimization from each `EXPDYN` subcase. **Note:** Bulk Data Entry
DOPTPRM, NESLIMPD	Specifies the number of time steps retained for optimization from each `IMPDYN` subcase.

Card	Description
	Note: Bulk Data Entry
DOPTPRM, NESLNLGM	Specifies the number of time steps retained for optimization from each `NLGEOM` subcase. **Note:** Bulk Data Entry

RADIOSS Cards

Card	Description
BULK	Used to insert bulk format data.
BULKFMT	Used to define different OptiStruct formats (FREE, LARGEFREE, FIXED, or LARGEFIXED) during the Radioss-to-OptiStruct conversion.
BULKMAT	Used to insert bulk format material card.
BULKPROP	Used to insert bulk format property card.
ESLPART	This keyword defines parts in the model, which will be included in ESLM optimization.

Control Cards

Control card entities create solver control cards such as results file I/O options, CPU and memory limits, and others.

A solver interface template must be loaded which defines control cards in order to create and edit control cards.

Abaqus

Control cards define model information, which needs to be specified only once in the input file.

Card	Description
*CONSTRAINT CONTROLS	Reset overconstraint checking controls.
*DEPVAR	Specify solution-dependent state variables.
*HEADING	Print a heading on the output.
*PREPRINT	Select printout for the analysis input file processor.
*RESTART	Save and reuse data and analysis results.

ANSYS

The input translator recognizes ANSYS cards. If an unsupported field is found in a card, a message is displayed on the status bar. The messages are also printed to the file `ansys.msg`. General slash commands, `SOLUTION` commands, `POST1` commands, and `POST26` commands are referred to as control cards. Unrecognized cards are written to a `*.hmx` file.

Card	Description
ACEL	Specifies the linear acceleration of the structure.
ALPHAD	Defines the mass matrix multiplier for damping.
ANTYPE	Specifies the analysis type and restart status.
ARCLEN	Activates the arc length method.
ARCTRM	Controls termination of the arc-length solution
/ASSIGN	Reassigns a file name to an ANSYS file identifier.
AUTOTS	Specifies whether to use automatic time stepping or load stepping.
/BATCH	Sets the program mode to "batch"

Card	Description
BETAD	Defines the stiffness matrix multiplier for damping.
BFUNIF	Assigns a uniform body force load to all nodes.
BUCOPT	Specifies buckling analysis options
CGLOC	Specifies the origin location of the acceleration coordinate system.
CGOMGA	Specifies the rotational velocity of the global origin.
CMDOMEGA	Specifies the rotational acceleration of an element component about a user-defined rotational axis.
CMOMEGA	Specifies the rotational velocity of an element component about a user-defined rotational axis.
CNVTOL	Sets convergence values for nonlinear analyses.
/COM	Places a comment in the output.
/COPY	Copies a file.
CRPLIM	Specifies the creep criterion for automatic time stepping.
DCGOMG	Specifies the rotational acceleration of the global origin.
/DELETE	Deletes a file.
DELTIM	Specifies the time step sizes to be used for this load step.
DMPRAT	Sets a constant damping ratio.
DOF	Adds degrees of freedom to the current DOF set.
DOMEGA	Specifies the rotational acceleration of the structure.
EMUNIT	Specifies the system of units for magnetic field problems.
EQSLV	Specifies the type of equation solver.

Card	Description
ERESX	Specifies extrapoloation of integration point results.
EORIENT	Reorient SOLID elements' axes
ETABLE	Fills a table of element values for further processing.
EXPASS	Specifies an expansion pass of an analysis.
HARFRQ	Defines the frequency range in the harmonic response analysis.
HREXP	Specifies the phase angle for the harmonic analysis expansion pass.
HROPT	Specifies harmonic analysis options.
HROUT	Specifies the harmonic analysis output options.
IRLF	Specifies that inertia relief calculations are to be performed.
KBC	Specifies stepped or ramped loading within a load step.
KUSE	Specifies whether or not to reuse the triangularized matrix.
LNSRCH	Activates a line search to be used with Newton-Raphson.
LSSOLVE	Reads and solves multiple load steps.
LVSCALE	Scales the load vector for mode superposition analyses.
MDAMP	Defines the damping ratios as a function of mode.
MODE	Specifies the harmonic loading term for this load step.
MODOPT	Specifies modal analysis options.
MXPAND	Specifies the number of modes to expand and write for a modal or buckling analysis.
NCNV	Sets the key to terminate an analysis.

Card	Description
NEQIT	Specifies the maximum number of equilibrium iterations for nonlinear analyses.
NLGEOM	Includes large-deflection effects in a static or full transient analysis.
NROPT	Specifies the Newton-Raphson options in a static or full transient analysis.
NSUBST	Specifies the number of substeps to be taken this load step.
NUMOFF	
OMEGA	Specifies the rotational velocity of the structure.
OUTRES	Controls the solution data written to the database.
/POST1	Enters the database results post-processor.
PRED	Activates a predictor in a nonlinear analysis.
PRESOL	Prints the solution results for elements.
PSTRES	Specifies whether prestress effects are calculated or included.
RSYS	Activates a coordinate system for printout or display of results.
SLOAD	Loads a pretension section.
/SOLU	Enters the solution processor.
SOLU	Specifies solution summary data per substep to be stored.
SOLVE	Starts a solution.
SSTIF	Activates stress stiffness effects in a nonlinear analysis.
/STITLE	Defines subtitles.
SUBOPT	Specifies options for subspace iteration eigenvalue extraction.
/SYS	Passes a command string to the operating system.

Card	Description
TIME	Sets the time for a load step.
TIMINT	Turns on transient effects.
TINTP	Defines transient integration parameters.
/TITLE	Defines a main title.
TOFFST	Specifies the temperature offset from absolute zero to zero.
TOTAL	Specifies automatic MDOF generation.
TREF	Defines the reference temperature for the thermal strain calculations.
TRNOPT	Specifies transient analysis options.
TUNIF	Assigns a uniform temperature to all nodes.
/UNITS	Annotates the database with the system of units used.
UNSU_END	
UNSU_PREP7	

EXODUS

Card	Description
OUTPUTS	Determines which data will be written to the output file.
PARAMETERS	Input parameters that are independent of the solution method or solver.

Card	Description
SOLUTION	Solution method and options to be selected in the solution.

LS-DYNA

Card	Description
*CONTROL_ACCURACY	Define control parameters that can improve the accuracy of the calculation.
*CONTROL_ADAPSTEP	Define control parameters for contact interface force update during each adaptive cycle.
*CONTROL_ADAPTIVE	Activate adaptive meshing.
*CONTROL_ADAPTIVE_CURVE	To refine the element mesh along a curve.
*CONTROL_ALE	Set global control parameters for the Arbitrary Lagrange-Eulerian and Eulerian calculations.
*CONTROL_BULK_VISCOSITY	Reset the default values of the bulk viscosity coefficients globally.
*CONTROL_CHECK	Check for various problems in the mesh.
*CONTROL_COARSEN	Adaptively de-refine (coarsen) a shell mesh by selectively merging four adjacent elements into one.
*CONTROL_CONTACT	Change defaults for computation with contact surfaces.
*CONTROL_COUPLING	Change defaults for MADYMO3D/CAL3D coupling.
*CONTROL_CPM	Global control parameters for CPM (Corpuscular Particle Method).
*CONTROL_CPU	Control CPU time.
*CONTROL_DYNAMIC_RELAXATION	Initialize stresses and deformation in a model to simulate a preload.
*CONTROL_EFG	Define controls for the mesh-free computation.
*CONTROL_ENERGY	Provide controls for energy dissipation options

Card	Description
*CONTROL_EXPLOSIVE_SHADOW	Compute detonation times in explosive elements for which there is no direct line of sight.
*CONTROL_HOURGLASS	Set the default values of the hourglass control to override the default values.
*CONTROL_IMPLICIT_AUTO	Define parameters for automatic time step control during implicit analysis.
*CONTROL_IMPLICIT_BUCKLE	Activate implicit buckling analysis when termination time is reached.
*CONTROL_IMPLICIT_DYNAMICS	Activate implicit dynamic analysis and define time integration constants.
*CONTROL_IMPLICIT_EIGENVALUE	Activate implicit eigenvalue analysis and define associated input parameters.
*CONTROL_IMPLICIT_FORMING	Activate implicit static analysis, primarily for metal forming processes.
*CONTROL_IMPLICIT_GENERAL	Activate implicit analysis and define associated control parameters.
*CONTROL_IMPLICIT_INERTIA_RELIEF	Allows analysis of linear static problems that have rigid body modes.
*CONTROL_IMPLICIT_MODES	Request calculation of constraint and/or attachment modes for later use in modal analysis using *PART_MODES.
*CONTROL_IMPLICIT_SOLUTION	Optional card that applies to implicit calculations.Used to specify whether a linear or nonlinear solution is desired.
*CONTROL_IMPLICIT_SOLVER	Optional card that applies to implicit calculations. The linear equation solver performs the CPU-intensive stiffness matrix inversion.
*CONTROL_IMPLICIT_STABILIZATION	Optional card that applies to implicit calculations. Artificial stabilization is required for multi-step unloading in implicit springback analysis.
*CONTROL_IMPLICIT_TERMINATION	Specify termination criteria for implicit transient simulations.

Card	Description
*CONTROL_MPP_DECOMPOSITION_AUTOMATIC	Instructs the program to apply a simple heuristic to try to determine the proper decomposition for the simulation.
*CONTROL_MPP_DECOMPOSITION_CHECK_SPEED	Modifies the decomposition depending on the relative speed of the processors involved.
*CONTROL_MPP_DECOMPOSITION_CONTACT_DIST	Ensures that the indicated contact interfaces are distributed across all processors, which can lead to better load balance for large contact interfaces.
*CONTROL_MPP_DECOMPOSITION_CONTACT_ISOL	Ensures that the indicated contact interferences are isolated on a single processor, which can lead to decreased communication.
*CONTROL_MPP_DECOMPOSITION_FILE	Allow for pre-decomposition and a subsequent run or runs without having to do the decomposition.
*CONTROL_MPP_DECOMPOSITION_METHOD	Specify the decomposition method to use.
*CONTROL_MPP_DECOMPOSITION_NUMPROC	Specify the number of processors for decomposition.
*CONTROL_MPP_DECOMPOSITION_SHOW	Allows display of the final decomposition.
*CONTROL_MPP_DECOMPOSITION_TRANSFORMATI	Specifies transformations to apply to modify the decomposition.
*CONTROL_MPP_IO_LSTC_REDUCE	Controls the decomposition of the model for solving in multiple CPU's.
*CONTROL_MPP_IO_NOD3DUMP	Suppress the output of all dump files.
*CONTROL_MPP_IO_NODUMP	Suppresses the output of all dump files and full deck restart files.
*CONTROL_MPP_IO_NOFULL	Suppresses the output of the full deck restart files.
*CONTROL_MPP_IO_SWAPBYTES	Swap bytes on some of the output files.
*CONTROL_OUTPUT	Set miscellaneous output parameters.
*CONTROL_PARALLEL	Control parallel processing usage for shared memory computers by defining the number of processors invoking the optional consistency of the global vector assembly.

Card	Description
*CONTROL_REMESHING	Provide control over the remeshing of solids which are meshed with the solid tetrahedron element type 13.
*CONTROL_RIGID	Special control options related to rigid bodies and the rigid-flexible bodies.
*CONTROL_SHELL	Provide controls for computing shell response.
*CONTROL_SOLID	Provide controls for solid element response.
*CONTROL_SOLUTION	Specify the analysis solution procedure if thermal only or coupled thermal analysis is performed.
*CONTROL_SPH	Provide controls for computing SPH particles.
*CONTROL_SPOTWELD_BEAM	Provides factors for scaling the failure force resultants of beam spot welds as a function of their parametric location on the contact segment and the size of the segment.
*CONTROL_STRUCTURED	Write out a LS-DYNA structured input deck for version 970.
*CONTROL_STRUCTURED_TERM	Write out a LS-DYNA structured input deck for version 970. Termination will occur after the structured input file is written.
*CONTROL_SUBCYCLE	Control time step subcycling.
*CONTROL_TERMINATION	Stop the job.
*CONTROL_THERMAL_NONLINEAR	Set parameters for a nonlinear thermal or coupled structural/thermal analysis.
*CONTROL_THERMAL_SOLVER	Set options for the thermal solution in a thermal only or coupled structural-thermal analysis.
*CONTROL_THERMAL_TIMESTEP	Set time step controls for the thermal solution in a thermal only or a coupled structural/thermal analysis.
*CONTROL_TIMESTEP	Set structural time step size control using different options.
*DAMPING_FREQUENCY_RANGE	Provides approximately constant damping over a range of frequencies.

Card	Description
*DAMPING_GLOBAL	Define mass weighted nodal damping that applies globally to the nodes of deformable bodies and to the mass center of rigid bodies.
*DATABASE_ABSTAT	Specify time interval and file type for Airbag statistics time history file output.
*DATABASE_AVSFLT	Specify time interval for AVS database output.
*DATABASE_BINARY_D3PART	Dt for partial output states.
*DATABASE_BINARY_D3PLOT	Dt for complete output states.
*DATABASE_BINARY_D3DRLF	Dynamic relaxation database.
*DATABASE_BINARY_D3THDT	Dt for time history data of element subsets.
*DATABASE_BINARY_D3DUMP	Binary output restart files. Define output frequency in cycles.
*DATABASE_BINARY_FSIFOR	ALE interface force database
*DATABASE_BINARY_INTFOR	Dt for output of contact interface data.
*DATABASE_BINARY_RUNRSF	Binary output restart file. Define output frequency in cycles.
*DATABASE_BINARY_XTFILE	Flag to specify output of extra time history data to XTFILE at same time as `D3THDT` file.
*DATABASE_BNDOUT	Specify time interval and file type for Boundary condition forces and energy time history file output.
*DATABASE_DEFGEO	Specify time interval deformed geometry file output.
*DATABASE_DEFORC	Specify time interval and file type for discrete element forces time history file output.
*DATABASE_ELOUT	Specify time interval and file type for element data time history file output.
*DATABASE_EXTENT_AVS	Specify output database to be written.
*DATABASE_EXTENT_BINARY	Specify output database to be written.
*DATABASE_EXTENT_D3PART	Controls, to some extent, the content of specific output databases.

Card	Description
*DATABASE_EXTENT_MOVIE	Specify output database to be written.
*DATABASE_EXTENT_MPGS	
*DATABASE_EXTENT_SSSTAT	
*DATABASE_FORMAT	
*DATABASE_GCEOUT	Specify time interval and file type for Geometric contact entities force time history file output
*DATABASE_GLSTAT	Specify time interval and file type for global model data time history file output.
*DATABASE_JNTFORC	Specify time interval and file type for joint force time history file output
*DATABASE_MASSOUT	Export nodal masses into ASCII file MASSOUT.
*DATABASE_MATSUM	Specify time interval and file type for material energies time history file output
*DATABASE_MOVIE	Specify time interval for movie output
*DATABASE_MPGS	Specify time interval for MPGS
*DATABASE_NCFORC	Specify time interval and file type for nodal interface forces time history file output
*DATABASE_NODFOR	Specify time interval and file type for nodal force groups time history file output
*DATABASE_NODOUT	Specify time interval and file type for nodal point data time history file output
*DATABASE_OPTION	Control cards for all ASCII output
*DATABASE_RBDOUT	Specify time interval and file type for rigid body data time history file output.
*DATABASE_RCFORC	Specify time interval and file type for resultant interface forces time history file output.
*DATABASE_RWFORC	Specify time interval and file type for rigid wall forces time history file output.
*DATABASE_SBTOUT	Specify time interval and file type for Seat belt time history file output.

Card	Description
*DATABASE_SECFORC	Specify time interval and file type for cross section forces time history file output
*DATABASE_SLEOUT	Specify time interval and file type for sliding interface energy time history file output
*DATABASE_SPCFORC	Specify time interval and file type for SPC reaction forces time history file output
*DATABASE_SPHOUT	Specify time interval and file type for SPH element data time history file output
*DATABASE_SPRING_FORWARD	Create spring forward nodal force file.
*DATABASE_SSSTAT	Specify time interval and file type for Subsystem data time history file output
*DATABASE_SUPERPLASTIC_FORMING	Specify the output intervals to the superelastic forming output files.
*DATABASE_SWFORC	Specify time interval and file type for nodal constraint reaction forces time history file output
*DATABASE_TPRINT	Specify time interval and file type for thermal time history file output. Thermal output from a coupled structural/thermal or thermal only analysis.
*DATABASE_TRACER	Tracer particles will save a history of either a material point or a spatial point into an ASCII file, TRHIST.
*DATABASE_TRHIST	Tracer particle history information
*INTERFACE_COMPENSATION_NEW	Calculate the deviation of the part from its intended design of the stamped part and automatically compensate the tool to minimize the deviation, modify the trimming curve after the die modification, and automatically detect the undercut problem
*STRESS_INITIALIZATION	Allows all or selected parts to be initialized on restart, using data from the d3dump or runrsf database.
*STRESS_INITIALIZATION_DISCRETE	Initializes all discrete parts from the old parts.

Card	Description
*STRESS_INITIALIZATION_SEATBELT	Initializes all seatbelt parts from the old parts.

Nastran

Card	Description
ACMODL	Defines modeling parameters for the interface between the fluid and the structure.
AERO	Gives basic aerodynamic parameters for unsteady aerodynamics.
AEROS	Defines basic parameters for static aeroelasticity.
AUTOSPC	Constrains stiffness singularities via m-sets or s-sets.
B2GG	Selects direct input damping matrix or matrices.
B2PP	Selects direct input damping matrix or matrices.
BEGIN BULK	Designates the end of the Case Control Section and/or the beginning of a Bulk Data Section. **Note:** When the model is created from HyperMesh, this card is on by default.
BULK-UNSUPPORTED_CARD	**Note:** On import, all unsupported bulk data will be written into this card. You can edit the card.
CASE-UNSUPPORTED_CARD	**Note:** If CEND and SUBCASE exist, on import, all unsupported cards between CEND and the first SUBCASE will be written into this card. You can edit the card. If SUBCASE does not exist and BEGIN BULK exists, all unsupported cards between CEND and BEGIN BULK will be written into this card.

Card	Description
CEND	**Note:** When the model is created from HyperMesh, this card is on by default.
DIAG	Requests diagnostic output or special options.
ECHO	Bulk Data Echo Request
ENDDATA	Designates the end of the Bulk Data Section. **Note:** When the model is created from HyperMesh, this card is on by default.
EXEC_UNSUPPORTED_CARDS	**Note:** If CEND exists in the imported deck, on import, all unsupported cards before CEND will be written into this card. You can edit the card. If CEND does not exist, on import, all unsupported cards before BEGIN BULK will be written into this card.
ERP	Equivalent Radiated Power Panel Participation Factor Output Request
FLSPOUT	Control for fluid-structure mode participation output.
FLUX	Request gradient and flux output for all heat transfer analysis subcases or individual heat transfer analysis subcases respectively.
GLOBAL	
GLOBAL CASE CONTROL	
GLOBAL OUTPUT REQUEST	
GRDSET	Defines default options for fields 3, 7, 8, and 9 of all `GRID` entries
IC	

Card	Description
ID	Specifies a comment.
INCLUDE_BULK	Inserts an external file into the input file
INCLUDE_CTRL	Inserts an external file into the input file
INCLUDE_EXEC	Inserts an external file into the input file
K2GG	
K2PP	Selects direct input stiffness matrices, which are not included in normal modes
M2GG	Selects direct input mass matrix or matrices.
MAXLINES	Sets the maximum number of output lines. **Note:** The maximum number of lines is 999999999 (Nastran default).
MONITOR_POINTS	Defines an integrated load monitor point.
NLMOPTS	Specifies nonlinear material optional schemes in SOL 400. Extends to material options, property options including property mapping, boundary condition options.
OMIT_BEGIN_BULK	**Note:** By selecting this control card, you are forcing HyperMesh to not write BEGIN BULK CARD.
OMIT_CEND	**Note:** By selecting this control card, you are forcing HyperMesh to not write the CEND card.
OMIT_END_BULK	**Note:** By selecting this card, you are forcing HyperMesh to not write the END_BULK card.
P2G	Selects direct input load matrices.
PARAM	Specifies values for parameters.

Card	Description
	Note: On import, all unsupported PARAM cards are written into the UNSUPPORTED PARAMS block within PARAM CARD in the Control Cards panel. You can edit this data block.
PRESSURE	Pressure output request
RADSET	Specifies which radiation cavities are to be included for radiation enclosure analysis.
RESTART	Requests that data stored in a previous run be used in the current run.
RESVEC	Controls residual vectors
RIGID	Rigid Element Method
SET CARD	Defines a set of element or grid point numbers to be plotted. **Note:** Only real number sets can be created using control cards.
SOL	Specifies the solution sequence or main subDMAP to be executed.
SUBTITLE	Defines a subtitle that will appear on the second heading line of each page of printer output.
SWLDPRM	Overrides default values of parameters for CFAST, CWELD, and CSEAM connectivity search.
TIME	Sets the maximum CPU and I/O time.
TITLE	Defines a character string that will appear on the first heading line of each page of NX Nastran printer output.

Card	Description
TSTEPNL	Defines parameters for geometric nonlinear implicit dynamic analysis strategy.

OptiStruct

I/O Option Entries are supported as control cards when appearing before the first SUBCASE statement. In many cases, information supplied on these entries is overridden by repeated definitions within subcases.

Card	Description
ACCELERATION	Requests acceleration vector output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
ACMODL	Defines model parameters for the fluid-structure interface. **Note:** Bulk Data Entry
ANALYSIS	Used in the I/O Options section to request that only a finite element analysis be performed (optimization input is ignored). It may also be used in the I/O Options or Subcase Information sections to identify the solution sequence for all subcases or for individual subcases, respectively. **Note:** Subcase Information Entry Supported as an output option on the subcase definition when it appears within a subcase.
ASSIGN	Used in the I/O Options section to identify external files and their contents.

Card	Description
	Note: I/O Options and Subcase Information Entry Supported as an output option on the subcase definition when it appears within a subcase.
AUTOSPC	Used to automatically constrain stiffness singularities and near singularities with single point constraints. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_CASE_CONTROL`.
B2GG	Selects a direct input viscous damping matrix. **Note:** Subcase Information Entry Global matrix selectors.
BULK_UNSUPPORTED_CARDS	**Note:** If a line (not a continuation line) occurs after the BEGIN BULK statement in an input file and starts with a keyword that is not recognized or supported, then the entire card gets written to BULK_UNSUPPORTED_CARDS. It is also possible to manually define an unsupported OptiStruct card using the BULK_UNSUPPORTED_CARDS. BULK_UNSUPPORTED_CARDS are exported near the bottom of the exported OptiStruct input file, just before the `ENDDATA` statement.
CFAILURE	Requests failure index output for elements referencing `PCOMP`, `PCOMPP` or `PCOMPG` properties for all subcases or individual subcases, respectively.

Card	Description
	Note: I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. CFAILURE is supported for static analysis only.
CHECK	Requests that only a model check be performed. **Note:** I/O Options and Subcase Information Entry
CMDE	Used above the first `SUBCASE` or within a `SUBCASE` definition to request component modal synthesis damping energy output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
CMKE	Used above the first `SUBCASE` or within a `SUBCASE` definition to request component modal synthesis kinetic energy output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
CMSE	Used above the first `SUBCASE` or within a `SUBCASE` definition to request component modal synthesis strain energy output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.

Card	Description
CONTF	Requests contact results output for all Nonlinear Analysis subcases or individual nonlinear analysis subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
CONTPRM	Defines the default properties of all contacts and sets parameters that affect all contacts. The default values set here can be overridden by values explicitly specified on `PCONT`, `PCONTX`, and `CONTACT` cards. **Note:** Bulk Data Entry
CSTRAIN	Requests ply strain output for elements referencing `PCOMP`, `PCOMPP` or `PCOMPG` properties for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
CSTRESS	Requests ply stress output for elements referencing `PCOMP`, `PCOMPP` or `PCOMPG` properties for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.

Card	Description
CTRL_UNSUPPORTED_CARDS	**Note:** If a line (not a continuation line) occurs before the BEGIN BULK statement and before the first SUBCASE statement and starts with a keyword that is not recognized or supported, then the entire card gets written to CTRL_UNSUPPORTED_CARDS. It is also possible to manually define data cards appearing above the first SUBCASE statement using the CTRL_UNSUPPORTED_CARDS. CTRL_UNSUPPORTED_CARDS are exported near the top of the exported OptiStruct input file, just before the first SUBCASE statement.
DAMAGE	Requests fatigue damage results output for all fatigue subcases or individual fatigue subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
DEBUG	Supports internal issues for debugging purposes in HyperMesh. **Note:** I/O Options and Subcase Information Entry Some special or custom features can be accessed through the use of 'debug, <string>, <real>' statements.
DENSITY	Requests density output for a topology optimization. **Note:** I/O Options and Subcase Information Entry

Card	Description
DENSRES	Controls the frequency of output of design results (density, shape, or thickness). **Note:** I/O Options Entry
DESHIS	Controls the creation of the optimization history file `.hgdata`. **Note:** I/O Options and Subcase Information Entry
DESVARG	Defines an override for design variable settings. **Note:** Bulk Data Entry
DGLOBAL	Defines input parameters required for the Global Search Option (GSO). **Note:** Bulk Data Entry
DISPLACEMENT	Requests displacement vector output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
DMIGNAME	Defines the name given to the reduced matrices written to an external data file. **Note:** I/O Options and Subcase Information Entry
DSA	Requests Design Sensitivity Analysis results in a Frequency Response Analysis. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.

Card	Description
DTI_UNITS	Defines units for multibody, component mode synthesis (flexible-body preparation), and geometric nonlinear solution sequences. **Note:** Bulk Data Entry
ECHO	Outputs the interpreted forms of Subcase Information and Bulk Data Entries to the `.out` file. **Note:** I/O Options and Subcase Information Entry
EDE	Requests element energy loss per cycle and element energy loss per cycle density output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
EIGVNAME	Defines the prefix to be used for external eigenvalue data files (`.eigv`). **Note:** I/O Options and Subcase Information Entry
EKE	Requests kinetic energy and kinetic energy density output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
ELFORCE	Requests structural element force output and elemental fluid particle velocity output for all subcases or individual subcases, respectively.

Card	Description
	Note: I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
ENERGY	Requests energy output for all geometric nonlinear analysis subcases or individual geometric nonlinear analysis subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
ERP	Requests equivalent radiated power output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
ESE	Requests strain energy and strain energy density output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.

Card	Description
FLUX	Requests temperature gradient and flux output for all steady-state heat transfer analysis subcases, transient heat transfer analysis subcases or individual heat transfer analysis subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
FORMAT	Indicates the format in which results are to be output. **Note:** I/O Options Entry Formats are: H3D (default), HM, FLX, OPTI, OUTPUT2, PUNCH, PATRAN, APATRAN, NONE.
FOS	Requests fatigue factor of safety, hydrostatic pressure, and shear stress output for all fatigue subcases or individual fatigue subcases. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
GAPPRM	Defines parameters that control connectivity and configuration checks for gap elements (`CGAP` and `CGAPG`). Most of these parameters also affect contact elements that are automatically created on CONTACT interfaces. **Note:** Bulk Data Entry
GLOBAL_CASE_CONTROL	Handles the data selectors FREQ, METHOD, MPC, SDAMPING and SPC appearing above the first `SUBCASE` statement.

Card	Description
	It also handles the data selector `DESVAR`, used to select a set of design variables for use in an optimization run. It also handles the output control `OMODES`, used to define a set of modes for output requests. **Note:** This control card OMODES is also supported as an output option on the subcase definition when it appears within a subcase.
GPFORCE	Requests grid point force balance output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
GPSTRAIN	Requests grid point strain output for all or individual subcases. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`
GPSTRESS	Requests grid point stresses output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
GRDSET	Defines default options for fields 3, 7, and 8 of all `GRID` entries.

Card	Description
	Note: Bulk Data Entry
HISOUT	Controls the amount of data printed to the `.hgdata` file. **Note:** I/O Options and Subcase Information Entry
INCLUDE_BULK	**Note:** This control card is retained to support old database files.
INCLUDE_CTRL	**Note:** This control card is retained to support old database files.
INFILE	Identifies the file containing the Bulk Data Entries. **Note:** I/O Options Entry Its extension must be `.fem`.
K2GG	Selects a direct input stiffness matrix. **Note:** Subcase Information Entry Global matrix selectors
K2PP	Selects a direct input stiffness matrix, which is not included in normal modes. **Note:** Subcase Information Entry Global matrix selectors
K42GG	Selects a direct input stiffness matrix. **Note:** Subcase Information Entry Global matrix selectors

Card	Description
LIFE	Requests output of fatigue life results for all fatigue subcases or individual fatigue subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
LOADLIB	Defines the external libraries and external files to be loaded into OptiStruct. External functions can be implemented within dynamic libraries (`.dll`) under Windows, shared libraries (`.so`) under Linux, by using HyperMath (`.hml`) on Windows and Linux, and external files by using Microsoft Excel (`.xls` or `.xlsx`) on Windows. **Note:** I/O Options and Subcase Information Entry
M2GG	Selects a direct input mass matrix. **Note:** Subcase Information Entry Global matrix selectors
MBFORCE	Requests force output for a set of joints and/or force elements from multibody dynamics subcases. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
MODALDE	Used above the first SUBCASE or within a SUBCASE definition to request modal damping energy (the energy loss per cycle) output for all subcases or individual subcases, respectively. Note that this modal damping energy only includes the energy contribution from viscous dampers.

Card	Description
	Structural damping and modal damping are not included. **Note:** I/O Options and Subcase Information Entry Found under GLOBAL_OUPUT_REQUEST.
MODALKE	Used above the first SUBCASE or within a SUBCASE definition to request modal kinetic energy output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under GLOBAL_OUPUT_REQUEST.
MODALSE	Used above the first SUBCASE or within a SUBCASE definition to request modal strain energy output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under GLOBAL_OUPUT_REQUEST.
MODEL	Requests output of only a subset of the model and related results for H3D and OUTPUT results files as well as for CMS superelements. **Note:** I/O Options and Subcase Information Entry This option is intended for multi-body dynamics and transient solution sequences with which users often require results for only a subsection of a model, but it is applied to all solution sequences.

Card	Description
Model Documentation	**Note:** $HMBEGINDOC and $HMENDDOC indicate a section of comment cards which are supported on import and export. The comments are stored on control card Model Documentation. This information is exported at the top of the exported OptiStruct input file.
MPCFORCE	Requests multi-point force of constraint vector is output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
MSGLMT	Limits the number of ERROR, WARNING and INFORMATION messages output, or to elevate a WARNING or INFORMATION message to an ERROR. **Note:** I/O Options and Subcase Information Entry
NLLOAD	Used to request nonlinear load output for Transient Response Analysis for each time step for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
OFREQUENCY	Requests a set of frequencies for output requests for all subcases or individual subcases, respectively.

Card	Description
	Note: I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
OLOAD	Requests the form of applied load vector output and temperature load output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
OSDIAG	**Note:** Some special diagnostic information can be processed through the use of 'osdiag, <integer>, <integer>, <real>, <real>' statements.
OTIME	Requests a set of times for output requests for Transient Analysis for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
OUTFILE	Defines the prefix for the results files output. **Note:** I/O Options and Subcase Information Entry
OUTPUT	Controls the format of results output and the creation of certain results files.

Card	Description
	Note: I/O Options Entry
P2G	Used before the first subcase to select a direct input load matrix. **Note:** Subcase Information Entry Global matrix selectors
PARAM	Defines values for parameters used during analysis and optimization. **Note:** Bulk Data Entry If an unsupported argument is encountered on importing a PARAM card, the data is stored as UNSUPPORTED_PARAMS on the PARAM card. It is also possible to create an unsupported PARAM card using the UNSUPPORTED_PARAMS option.
PEAKOUT	Defines criteria used for the automatic identification of loading frequencies at which result peaks occur. Other result output may then be requested at these "peak" loading frequencies. This feature is only supported for frequency response solution sequences. **Note:** Bulk Data Entry
PFGRID	Requests output of acoustic grid participation factors for all Modal Frequency Response subcases. The output will be in the `.h3d` file. **Note:** I/O Options and Subcase Information Entry
PFMODE	Requests output of modal participation factors for all modal frequency response subcases.

Card	Description
	Note: I/O Options and Subcase Information Entry
PFPANEL	Requests output of acoustic panel participation factors for all Modal Frequency Response subcases. **Note:** I/O Options and Subcase Information Entry
POWERFLOW	Requests output of the power flow field. **Note:** I/O Options Entry Found under GLOBAL_OUPUT_REQUEST.
PRESSURE	Requests the output of pressure results. Analogous to the `DISPLACEMENT` command. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
PRETBOLT	Requests output of pretension force/adjustment values in the pretension bolts for all pretensioning and pretensioned subcases. **Note:** I/O Options and Subcase Information Entry Found under GLOBAL_OUPUT_REQUEST.
PROPERTY	Requests the output of the property definitions used in the final iteration of an optimization.

Card	Description
	Note: I/O Options and Subcase Information Entry
REQUEST	Selects a multibody request definition to be used in a multibody problem. **Note:** I/O Options and Subcase Information Entry
RESPRINT	Rorces all unretained responses of a certain type to be printed to the output file, provided they are referenced either as an objective or a constraint. This also applies to manufacturing constraints for composites. **Note:** I/O Options and Subcase Information Entry
RESTART	Indicates that the current optimization is to be restarted from the final iteration of a previous optimization. **Note:** I/O Options and Subcase Information Entry
RESULTS	Determines the frequency of output of analysis results for all subcases or for individual subcases, respectively. **Note:** I/O Options Entry
SACCELERATION	Requests the form and type of modal participation accelerations output for all subcases or individual subcases, respectively.

Card	Description
	Note: I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
SCREEN	Controls the output of model, analysis, and optimization information to the UNIX or DOS shell. **Note:** I/O Options and Subcase Information Entry
SDISPLACEMENT	Requests the form and type of modal participation displacements output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
SENSITIVITY	Requests the output of the responses and sensitivities for size and shape design variables to a Microsoft Excel spreadsheet. **Note:** I/O Options and Subcase Information Entry
SENSOUT	Controls the frequency of output of responses and sensitivities for size and shape design variables to a Microsoft Excel spreadsheet. **Note:** I/O Options and Subcase Information Entry
SHAPE	Requests altered shape output for a shape optimization.

Card	Description
	Note: I/O Options and Subcase Information Entry
SHRES	Controls the frequency of output of the state files (`.#.sh` file and the `.#.grid` file). **Note:** I/O Options Entry
SINTENS	Used in the I/O Options section to request Sound Intensity output for all frequency response subcases. The `SINTENS` command can be used in the I/O Options or Subcase Information sections to request Sound Intensity output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
SPCFORCE	Requests single-point force of constraint vector output for all subcases or individual subcases, respectively. **Note:** SPCFORCE I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
SPL	Requests Sound Pressure output for all subcases or individual subcases, respectively. `SPL` can only be requested for frequency response subcases. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.

Card	Description
SPOWER	Requests Sound Power output for all subcases or individual subcases, respectively. SPOWER can only be requested for frequency response subcases. **Note:** I/O Options and Subcase Information Entry Found under GLOBAL_OUPUT_REQUEST.
STRAIN	Requests strain output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under GLOBAL_OUPUT_REQUEST. Supported as an output option on the subcase definition when it appears within a subcase.
STRESS	Requests stress output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under GLOBAL_OUPUT_REQUEST. Supported as an output option on the subcase definition when it appears within a subcase.
SUBTITLE	Defines the subtitle for all subcases or for individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Individual subcases may have their own SUBTITLE definitions which are supported on the subcase definition. These will override the default subtitle.
SVELOCITY	Requests the form and type of modal participation velocities output for all subcases or individual subcases, respectively.

Card	Description
	Note: I/O Options and Subcase Information Entry Found under GLOBAL_OUPUT_REQUEST. Supported as an output option on the subcase definition when it appears within a subcase.
SWLDPRM	Defines values of parameters used during the CWELD and CSEAM connectivity search. **Note:** Bulk Data Entry
SYSSETTING	Alters system settings. Any setting defined here may be over-ridden by command line arguments (Run Options). Most of these options can also be specified in one of the config files (Configuration File). **Note:** I/O Options Entry
THERMAL	Requests temperature output for all heat transfer analysis subcases or individual heat transfer analysis subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under GLOBAL_OUPUT_REQUEST. Supported as an output option on the subcase definition when it appears within a subcase.
THICKNESS	Requests thickness output for elements referencing a PSHELL, PCOMP or PCOMPP properties in size/free-size optimization and analysis runs. **Note:** I/O Options and Subcase Information Entry
THIN	Requests thinning and thickness output for all geometric nonlinear analysis subcases or

Card	Description
	individual geometric nonlinear analysis subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`.
TITLE	Defines the title for the OptiStruct job. **Note:** I/O Options and Subcase Information Entry
TMPDIR	Chooses the directory in which the scratch files are to be written. **Note:** I/O Options and Subcase Information Entry OptiStruct allows multiple TMPDIR entries, but only one instance is currently supported.
UNITS	Defines a system of units for the model. **Note:** I/O Options and Subcase Information Entry
VELOCITY	Requests velocity vector output for all subcases or individual subcases, respectively. **Note:** I/O Options and Subcase Information Entry Found under `GLOBAL_OUPUT_REQUEST`. Supported as an output option on the subcase definition when it appears within a subcase.
XSHLPRM	Defines default shell element parameters for geometric nonlinear analysis. **Note:** Bulk Data Entry

Card	Description
XSOLPRM	Defines default `SOLID` properties for geometric nonlinear analysis. **Note:** Bulk Data Entry

PAM-CRASH 2G

Card	Description
AIRBAGCHECK	Airbag check option (Prime Control keywords) AIRBAGCHECK is executed by default to check the airbag's closed volume.
ANALYSIS	Analysis definition (Prime Control keywords) If AUTOSLEEP is not present inside the input file or AUTOSLEEP is not set to YES, the solver stops if the license tokens are not available at the beginning of the computation.
AUTOSLEEP	Autosleep option (Prime Control keywords)
CCTRL /	Contact Control Cards (Standard Control keywords)
CKCTRL/	Check Control Cards (Standard Control keywords)
COUPLING	Coupling (Optional Control keywords)
DATACHECK	Data Check option (Prime Control keywords)
DCOMP	DMP Domain Decomposition Method option (Prime Control keywords)
DEBUG	Debug option (Prime Control keywords)
DRAPP	Draping Database Path Name Definition (Standard Control keywords)
DRAPF /	Import Composite Layer Draping File Definition (Standard Control keywords)
ECTRL /	Element Control Cards (Standard Control keywords)
EIGEN /	Eigenmode Extraction and Damping Matrix Generation Analyses (Standard Control keywords)
EOCTRL/	ERF Control Cards (Standard Control keywords)
ENDDATA	End of Data option (General Model setup)

Card	Description
	Note: Cards after the ENDDATA card are ignored. This card is not created during FE input process, but it will be exported automatically.
FILE	File Root Name option (Prime Control keywords)
FPCTRL/	FPM Control Definition (Standard Control keywords)
HEAT /	Heat Analysis Control (Standard Control keywords)
ICTRL /	Implicit General Control Cards Definition (Standard Control keywords)
INPUTVERSION	Input Version Definition (Prime Control keywords)
MAXMEMORY	Maximum Memory Check option (Prime Control keywords)
MERGEGAP	Common Node Merge option (Prime Control keywords)
METRIC/	Metric Input Definition (Standard Control keywords)
METRICCHECK	Metric Check option (Prime Control keywords)
MSTRM /	Mass Trimming Definition (Standard Control keywords)
OCTRL /	Output Parameters Control Cards (Standard Control keywords)
ORTHF /	Import Element Orientation Definition (Standard Control keywords)
ORTHP	Element Orientation File Path Name Definition (Standard Control keywords)
PFLOW /	Porous Flow Analysis Definition (Standard Control keywords)
PIPE	Process Independent Parallel Execution (PIPE) option (Optional Control keywords)
PRCTRL/	Print Control Definition (Standard Control keywords)
PROFILE_DMP	DMP Profile Guided Domain Decomposition option (Prime Control keywords)
LCTRL /	Load Control Definition (Standard Control keywords)
MCTRL /	Metric Control Definition (Standard Control keywords)
RESTARTFILES	Cyclic Restart File option (Prime Control keywords)
RUNEND/	End of Run Definition (Standard Control keywords)
SHELLCHECK	Shell Element Data Check option (Prime Control keywords)

Card	Description
SIGNAL	Job Signal option (Prime Control keywords)
SOLIDCHECK	Solid Element Data Check option (Prime Control keywords)
SOLVER	Solver Selection option (Prime Control keywords)
SPCTRL/	Smoothed Particle Hydrodynamics (SPH) Control Cards (Standard Control keywords)
STOPRUN	Stop Run option (Prime Control keywords)
SUBRUN	Substructure Run option (Prime Control keywords)
TCTRL /	Time Step Definition (Standard Control keywords)
TITLE /	Title Card or Header on Output Listing Card (Standard Control keywords)
UNIT	Unit System Definition (Prime Control keywords)
SECURE_NODE_VISU	Secured Nodes Visualization option (PAMCOMFORT) (Optional Control keywords)
$HMUNSUPPORTED CARDS	Unsupported cards **Note:** All unsupported information found in the input deck is imported in this card. You can edit this card.
$HMBEGINDOC $HMENDDOC	Imported model documentation **Note:** The input deck can contain model documentation text that is imported and exported via the comment lines. For decks not written by HyperMesh, all comment lines before the first non-comment line are treated as model documentation. For decks written by HyperMesh, the documentation lines must be placed in a block that begins with the line $HMBEGINDOC and ends with the line $HMENDDOC. This card is always overwritten by the current model documentation.
$HMBEGINDOC $HMENDDOC	Model documentation

Card	Description
	Note: This card is not created during FE input process. You can create this card, but once this card is defined, only this will be exported. You can access macros from the PAM-CRASH solver interface to append or overwrite the information from Imported Model Documentation to this card.

Permas

Card	Description
$COMPONENT	Component input bracket header line.
$ECHO	Controls the echo print of Permas data input lines.
$MODDAMP	Definition of viscous or structural modal damping. **Note:** This option is available in the SYSTEM card as a separate checkbox.
$SYSTEM	Opens the bracket for input of system data.

Radioss

Card	Description
/@TFILE	The time at which the time history file begins. **Note:** Engine Keyword
/@TFILE/Keyword2	The time at which the time history file begins. **Note:** Engine Keyword
/ABF	Describes the output of `.abf` files. (`.abf` [binary] files are optimized for fast plotting of very large data sets and is intended for creating 2D and 3D plots using HyperGraph and HyperGraph 3D).

Card	Description
	Note: Engine Keyword
/ADMESH/GLOBAL	Defines the global parameters for adaptive meshing. This keyword is not available for SPMD computation. **Note:** Block Format Keyword
/ALE	Sets parameters for DONEA grid velocity formulation of ALE. **Note:** Engine Keyword
/ALE/2	Sets parameters for Radioss grid velocity formulation for ALE. **Note:** Engine Keyword
/ALE/3	When this keyword is used instead of `/ALE` or `/ALE/2`, no grid calculation is performed. Grid does not move, unless constrained otherwise. **Note:** Engine Keyword
/ALE/4	Sets the Radioss standard formulation. **Note:** Engine Keyword
/ALE/GRID/DISP	The displacement of a grid node depends on the displacements of the neighboring grid nodes. **Note:** Block Format Keyword
/ALE/GRID/DONEA	Velocity of a given grid node depends on velocity and displacements of neighboring grid nodes. **Note:** Block Format Keyword

Card	Description
/ALE/GRID/SPRING	Grid nodes are connected with nonlinear springs (those springs have no effect on the material, only on the movement of the grid). **Note:** Block Format Keyword
/ALE/GRID/STANDARD	Describes the standard formulation for ALE grid velocity computation. It is an improved `/ALE/GRID/SPRING` formulation based on edge springs and anti-shear springs. **Note:** Block Format Keyword
/ALE/GRID/ZERO	Describes the Euler formulation. **Note:** Block Format Keyword
/ALESUB	Subcycling on Lagrangian parts. **Note:** Engine Keyword
/AMS	Describes the part group on which the advanced mass scaling is applied. **Note:** Block Format Keyword
/ANALY	Defines the type of analysis and sets analysis flags. **Note:** Block Format Keyword
/ANIM	Generates animation files containing results according to the keywords specified. The keywords are specified in the topics within Animation and Post-processing Output Files. **Note:** Engine Keyword
/ANIM/BRICK/TENS	Generates animation files containing tensor data (mean value or value in specified integration

Card	Description
	point) for brick elements, in global coordinate system. **Note:** Engine Keyword
/ANIM/BRICK/TENS/STRAIN	Generates animation files containing strain tensor for a specified integration point of a solid, in global coordinate system. **Note:** Engine Keyword
/ANIM/BRICK/TENS/STRESS	Generates animation files containing stress tensor for a specified integration point of a solid, in global coordinate system. **Note:** Engine Keyword
/ANIM/CUT/1	Defines a section cut on a deformed geometry. **Note:** Engine Keyword
/ANIM/CUT/2	Defines a section cut on the initial geometry. **Note:** Engine Keyword
/ANIM/CUT/3	Defines a section cut on a deformed geometry using three nodes. **Note:** Engine Keyword
/ANIM/DT	Write animation files (A-files) at a time frequency equal to T_{freq}, the first file being written at time T. The animation file name will be `RunnameAnnn`, where `Runname` is the Run Name and `nnn` is the file number. **Note:** Engine Keyword
/ANIM/Eltyp/FORC	Generates animation files containing force and moment data for the specified element type.

Card	Description
	Note: Engine Keyword
/ANIM/Eltyp/Restype	Generates animation files containing element data for the specified result. **Note:** Engine Keyword
/ANIM/GPS1	Generates animation files containing simple average GPS data. The average value at node is calculated by the mean stress of all elements, which are connected to this node. **Note:** Engine Keyword
/ANIM/GPS2	Generates animation files containing volume based averaged GPS data, which is done by the relative element volumes (connected to same node use Vi/ V_total). **Note:** Engine Keyword
/ANIM/GZIP	Generates animation files in GZIP format. **Note:** Engine Keyword
/ANIM/KEEPD	In animation files, keep deleted elements in their original part; otherwise group all deleted elements in an extra part (named "deleted elements"). **Note:** Engine Keyword
/ANIM/LSENSOR	Write additional animation files (Animation Output File (A-File)) at a time frequency equal to T_{freq}, the first file being written at sensor activation time. The sensor activation time is given by the sensor property set $sens\_ID_N$. **Note:** Engine Keyword

Card	Description
/ANIM/MASS	Generates animation files containing nodal masses. **Note:** Engine Keyword
/ANIM/MAT	Generates animation files with one part for each material. **Note:** Engine Keyword
/ANIM/NODA	Generates animation files containing nodal scalar data. **Note:** Engine Keyword
/ANIM/SENSOR	Write additional animation files (Animation Output File (A-File)) with a time frequency equal to T_{freq}, the first file being written once sensor activated. (The activation of sensor will be defined in /SENSOR). **Note:** Engine Keyword
/ANIM/SHELL/EPSP	Generates animation files containing plastic strain as function of a shell element integration point. **Note:** Engine Keyword
/ANIM/SHELL/PHI/ALL	Animation output of the angle between the element skew and direction 1 orthotropy for all layers. **Note:** Engine Keyword
/ANIM/SHELL/PHI/N	Animation output of the angle between the element skew and direction 1 orthotropy for layer N. **Note:** Engine Keyword

Card	Description
/ANIM/SHELL/TENS	Generates animation files containing shell tensor data for a specified result. **Note:** Engine Keyword
/ANIM/SHELL/USRi	Generates animation files containing shell element data for variables of user law on each integration point. **Note:** Engine Keyword
/ANIM/VECT	Generates animation files containing vectorial data for the specified variable. **Note:** Engine Keyword
/ANIM/VECT/FREAC	Generates animation files containing vectorial data for reaction forces of imposed velocities, displacements, accelerations and boundary conditions. **Note:** Engine Keyword
/ANIM/VERS	Generates animation files in Radioss environment post-processing formats 41 and 44. **Note:** Engine Keyword
/ARCH	Describes the architecture flag. **Note:** Block Format Keyword
/ATFILE	Defines the frequency of writing additional time history file of T-file "`RunnameTnnx`". **Note:** Engine Keyword
/BCS/ALE	Given node numbers will be constrained in specified directions X, Y or Z for grid DOF: Ω = 0.

Card	Description
	Note: Engine Keyword
/BCS/LAG	Given node numbers will be lagrangian in specified directions X, Y or Z: V = W. **Note:** Engine Keyword
/BCS/ROT	Given node numbers will be constrained in specified directions X, Y or Z for rotational DOF: Ω = 0. **Note:** Engine Keyword
/BCS/TRA	Given node numbers will be constrained in specified directions X, Y or Z for material translational DOF: V = 0. **Note:** Engine Keyword
/BCSR/ALE	Given node numbers will be released in specified directions X, Y or Z for grid DOF. **Note:** Engine Keyword
/BCSR/LAG	Given node numbers will no longer be lagrangian in specified directions X, Y or Z. **Note:** Engine Keyword
/BCSR/ROT	Given node numbers will be released in specified directions X, Y or Z for material translational DOF. **Note:** Engine Keyword
/BCSR/TRA	Given node numbers will be released in specified directions X, Y or Z for material translational DOF. **Note:** Engine Keyword

Card	Description
/BEGIN	Sets the run name, the version of the input manual, the number of starter run and input and work unit systems. **Note:** Block Format Keyword
/CAA	Describes the computation Aero-Acoustic formulation for Eulerian or quasi-Eulerian simulations. Internal forces are computed with finite volume formulation. It improves results for under integrated elements. By default artificial viscosity is canceled, (qa,qb) default values are zero. **Note:** Block Format Keyword
/DAMP	Defines the Rayleigh mass and stiffness damping coefficients applied to a set of nodes. The damping can be applied to any nodal DOF either in local or global coordinate system. **Note:** Block Format Keyword
/DEF_SHELL	Sets default values for certain parameters in shell property (`/PROP/SHELL`). **Note:** Block Format Keyword
/DEF_SOLID	Sets default values for certain parameters in solid property (`/PROP/TYPE14 (SOLID)` and `/PROP/TYPE20 (TSHELL)`). **Note:** Block Format Keyword
/DEL	Delete element numbers N_1, N_2, . . ., N_N. **Note:** Engine Keyword
/DEL/Eltyp/1	Delete element numbers N^1_{first} to N^1_{last}, . . ., N^I_{first} to N^I_{last}.

Card	Description
	Note: Engine Keyword
/DEL/INTER	Delete interface numbers N_1, N_2, . . ., N_N. **Note:** Engine Keyword
/DELINT	If a slave node is only connected to 2D solid elements that are all deleted, the node is deactivated from the interface. **Note:** Engine Keyword
/DT	Time step defaults for all elements. **Note:** Engine Keyword
/DT/AMS	Elementary time step for Advanced Mass Scaling. **Note:** Engine Keyword
/DT/Eltyp/Iflag	Time step for select entities. **Note:** Engine Keyword
/DT/Eltyp/Keyword3/Iflag	Element time step control. **Note:** Engine Keyword
/DT/NODA/Keyword3	Nodal time step control. **Note:** Engine Keyword
/DT/SHELL	The time step for shell elements using `/DT1` is lower but more accurate. Otherwise, this option is identical to `/DT1/SHELL`. **Note:** Engine Keyword

Card	Description
/DT/SHNOD or /DT/SHNOD/CST	Time step for shells. **Note:** Engine Keyword
/DT/SPHCEL	Generates the SPH cells time step. **Note:** Engine Keyword
/DT/SPHCEL/Keyword3	Generates the SPH cells time step control type. **Note:** Engine Keyword
/DT/THERM	Time step for thermal modeling. **Note:** Engine Keyword
/DT1/SHELL	The time step for shell elements using `/DT1` is lower, but more accurate. Otherwise, this option is identical to `/DT/SHELL`. **Note:** Engine Keyword
/DTIX	Sets initial and maximum time step for this run. **Note:** Engine Keyword
/DTSDE	The time step for degenerated or PENTA6 solid elements using this option is higher (almost factor 2). **Note:** Engine Keyword
/DYREL	Dynamic relaxation. **Note:** Engine Keyword
/DYREL/1	Dynamic relaxation applied to node group identifier *grnd_ID*.

Card	Description
	Note: Engine Keyword
/END/ENGINE	This keyword has to be set at the end of the Engine input deck when using Single File Input. **Note:** Engine Keyword
/FVMBAG/MODIF	Activates FVM airbag merging parameter modifications in Engine file. **Note:** Engine Keyword
/FXINP	Generates input files for flexible bodies from eigenmodes and static modes computation. **Note:** Engine Keyword
/IMPL	Keywords used for implicit solution. Different implicit solver methods (linear, nonlinear, buckling or quasi-static), different options to control the implicit computation, different implicit time step method and implicit output messages are introduced. **Note:** Engine Keyword
/IMPL/AUTOSPC	A zero stiffness DOF will be constrained automatically. **Note:** Engine Keyword
/IMPL/BUCKL/1	Euler buckling modes will be computed. **Note:** Engine Keyword
/IMPL/BUCKL/2	Euler buckling modes will be computed based on actual pre-stress stat.

Card	Description
	Note: Engine Keyword
/IMPL/CHECK	Implicit model checking will be run. **Note:** Engine Keyword
/IMPL/DIVERG/n	Divergence criterion with successive increasing residual number for nonlinear analysis. **Note:** Engine Keyword
/IMPL/DT/1	Implicit time step control method 1. **Note:** Engine Keyword
/IMPL/DT/2	Implicit automatic time step control method 2. **Note:** Engine Keyword
/IMPL/DT/3	Implicit automatic time step control with Riks method. **Note:** Engine Keyword
/IMPL/DT/FIXPOINT	Users' input fix time points where implicit nonlinear computation will not miss with automatic time step control. **Note:** Engine Keyword
/IMPL/DT/STOP	The computation will be stopped, if *DT_min* is reached. If *DT_max* is reached, the computation will continue with constant time step set as *DT_max*. **Note:** Engine Keyword
/IMPL/DTINI	Initial time step for nonlinear implicit analysis.

Card	Description
	Note: Engine Keyword
/IMPL/DYNA/1	Describes the implicit dynamics with α-HHT method. **Note:** Engine Keyword
/IMPL/DYNA/2	Describes the implicit dynamics with a general Newmark method. **Note:** Engine Keyword
/IMPL/DYNA/DAMP	Rayleigh structure damping with dynamic implicit analysis. **Note:** Engine Keyword
/IMPL/DYNA/FSI	Defines the settings for Fluid Structure Interaction Analysis with AcuSolve. **Note:** Engine Keyword
/IMPL/GSTIF/OFF	Geometrical stiffness matrix will not be used for implicit nonlinear calculation. **Note:** Engine Keyword
/IMPL/INTER/KCOMP	Describes the stiffness matrix; due to contact interfaces will be assembled completely. **Note:** Engine Keyword
/IMPL/INTER/KNONL	Defines including some contact nonlinearities in PCG linear solver. **Note:** Engine Keyword

Card	Description
/IMPL/LBFGS/L	Change the maximum number of BFGS quasi-Newton method for implicit nonlinear. **Note:** Engine Keyword
/IMPL/LINEAR	Linear implicit solution will be computed. **Note:** Engine Keyword
/IMPL/LINEAR/INTER	Implicit linear analysis will take into account contact interfaces. **Note:** Engine Keyword
/IMPL/LRIGROT	Due to some linearization used in geometrical nonlinear analysis, small time step has to be chosen to avoid accumulated errors under large rotation. In this case, this option improves the accuracy using usual time step. **Note:** Engine Keyword
/IMPL/LSEARCH/n	Line search methods for nonlinear analysis. **Note:** Engine Keyword
/IMPL/LSEARCH/OFF	Deactivating line search for nonlinear analysis. **Note:** Engine Keyword
/IMPL/MONVOL/OFF	Describes the stiffness of gas in monitored volume type 3 (tire modeling). **Note:** Engine Keyword
/IMPL/NCYCLE/STOP	The computation will be stopped, if *Ncy_max* is reached. **Note:** Engine Keyword

Card	Description
/IMPL/NONLIN	Nonlinear implicit methods. **Note:** Engine Keyword
/IMPL/NONLIN/SMDISP	Only small displacement effects will be considered with this option. **Note:** Engine Keyword
/IMPL/PREPAT	Describes the implicit option for pre-conditioning. **Note:** Engine Keyword
/IMPL/PRINT/LINEAR	Printout frequency for linear solvers. **Note:** Engine Keyword
/IMPL/PRINT/NONLIN	Printout frequency for nonlinear implicit iterations. **Note:** Engine Keyword
/IMPL/PSTIF/OFF	Deactivating load (pressure) stiffness matrix. **Note:** Engine Keyword
/IMPL/QSTAT	Quasi-static implicit solution will be computed. **Note:** Engine Keyword
/IMPL/QSTAT/DTSCAL	Quasi-static implicit solution with a factor for inertia stiffness matrix. **Note:** Engine Keyword
/IMPL/RREF/INTERF/n	Different reference force residual computations used for stop criteria on nonlinear analysis with contact.

Card	Description
	Note: Engine Keyword
/IMPL/RREF/LIMIT	Define reference force residual limit values used in stop criteria for implicit nonlinear. **Note:** Engine Keyword
/IMPL/RREF/OFF	Deactivation reference residual option and using the previous one in stop criteria for implicit nonlinear analysis. **Note:** Engine Keyword
/IMPL/SBCS/MSGLV	Describes the message level for BCS (Boeing) solver. **Note:** Engine Keyword
/IMPL/SBCS/ORDER	Reordering method definition for BCS. **Note:** Engine Keyword
/IMPL/SBCS/OUTCORE	BCS solver will use minimum in-core memory. **Note:** Engine Keyword
/IMPL/SHPOFF	The full projection of QEPH shell element for warped elements will be deactivated to keep consistency between the stiffness matrix and the internal force computations. **Note:** Engine Keyword
/IMPL/SHPON	The full projection of QEPH shell element for warped elements will be activated for stiffness matrix to keep consistency with the internal force computations.

Card	Description
	Note: Engine Keyword
/IMPL/SINIT	Describes the initial stresses that will be imposed gradually. **Note:** Engine Keyword
/IMPL/SOLVER	Selects linear solver. **Note:** Engine Keyword
/IMPL/SPRBACK	Implicit spring-back will be run in this session. **Note:** Engine Keyword
/IMPL/SPRING	Describes the linear or nonlinear stiffness choices for nonlinear spring in implicit nonlinear analysis. **Note:** Engine Keyword
/IMPLICIT	Allows different default values which are suitable for implicit calculations. **Note:** Block Format Keyword
/INCMP	Quasi-incompressible formulation for fluid material (compatible with `/MAT/LAW6` and `/MAT/LAW11`). **Note:** Engine Keyword
/INISTA	Describes the initial state file. **Note:** Block Format Keyword
/INIV/AXIS/Keyword3	Initialize rotational velocity about an axis and translational velocity.

Card	Description
	Note: Engine Keyword
/INIV/AXIS/Keyword3/1	Initialize rotational velocity about an axis and translational velocity. **Note:** Engine Keyword
/INIV/ROT	Initialize rotational velocity in the specified direction X, Y or Z. **Note:** Engine Keyword
/INIV/ROT/Keyword3/1	Initialize rotational velocity in the specified direction X, Y or Z. **Note:** Engine Keyword
/INIV/TRA	Initialize translational velocity in the specified direction X, Y or Z. **Note:** Engine Keyword
/INIV/TRA/Keyword3/1	Initialize translational velocity in the specified direction X, Y or Z. **Note:** Engine Keyword
/INTER	Describes an interface. **Note:** Engine Keyword
/IOFLAG	Describes the input-output flags. **Note:** Block Format Keyword
/KEREL	Kinetic energy relaxation.

Card	Description
	Note: Engine Keyword
/KEREL/1	Kinetic energy relaxation applied to node group *grnd_ID*. **Note:** Engine Keyword
/KILL	The Engine is stopped if one of the following criteria is exceeded. **Note:** Engine Keyword
/MADYMO	Activates MADYMO- Radioss coupling. **Note:** Engine Keyword
/MEMORY	Describes the memory request. **Note:** Engine Keyword
/MON	Provides an estimation of the CPU time spent for each processor. **Note:** Engine Keyword
/OUTP	Write ASCII formatted output files. **Note:** Engine Keyword
/OUTP/LSENSOR	An `outp` file is written each time a sensor in the defined list is activated. **Note:** Engine Keyword
/PARITH	Turns the parallel arithmetic ON/OFF. **Note:** Engine Keyword

Card	Description
/PRINT	Sets printout frequency for output file. **Note:** Engine Keyword
/RAD2RAD/ON	To activate Multi-Domain coupling, Radioss Engine Input file must contain the following directive. **Note:** Engine Keyword
/RANDOM	Describes the nodal random noise to check stability of model by introducing random noise on nodal coordinates. **Note:** Block Format Keyword
/RBODY	Listed rigid bodies will be activated or deactivated. **Note:** Engine Keyword
/RERUN	Permits to continue a previous Radioss run. **Note:** Engine Keyword
/RFILE	Writes a Restart R-File. **Note:** Engine Keyword
/RFILE/n	Rewrites a Restart R-File. **Note:** Engine Keyword
/RUN	Identifies the run number. **Note:** Engine Keyword
/RUN/Run Name/Run Number/Restart Letter	Identifies the run number. **Note:** Engine Keyword

Card	Description
/SHSUB	Enables you to activate new sub-cycling option, for which $I_{subcycle}$=2 in the `/ANALY` Radioss Starter option should be specified. **Note:** Engine Keyword
/SHVER/V51	The new large rotational body motion formulation for QEPH, QBAT and DKT18 will be deactivated. **Note:** Engine Keyword
/SPHGLO	Describes the SPH global parameters. **Note:** Block Format Keyword
/SPMD	Sets SPMD parameters for Hybrid Massively Parallel Program (HMPP) computation. **Note:** Block Format Keyword
/STAMPING	Allows adapting error messages to stamping applications. **Note:** Block Format Keyword
/STATE/BRICK/AUX/FULL	Describes the internal variable state for solid. **Note:** Engine Keyword
/STATE/BRICK/ORTHO	Describes orthotropy direction for orthotropic solid or thick shells. **Note:** Engine Keyword
/STATE/BRICK/STAIN/FULL	Describes the strain state for solid. **Note:** Engine Keyword

Card	Description
/STATE/BRICK/STRAIN/GLOBFULL	Describes the strain state for solid in global reference system. **Note:** Engine Keyword
/STATE/BRICK/STRES/FULL	Describes the stress state for solid. **Note:** Engine Keyword
/STATE/BRICK/STRESS/GLOBFULL	Describes the stress state for solid in global reference system. **Note:** Engine Keyword
/STATE/DT	Writes the state file. **Note:** Engine Keyword
/STATE/NODE/TEMP	Describes the temperature for nodes. **Note:** Engine Keyword
/STATE/LSENSOR	A state file is written each time a sensor in the defined list is activated. **Note:** Engine Keyword
/STATE/SHELL/AUX/FULL	Describes the internal variable state for solid. **Note:** Engine Keyword
/STATE/SHELL/EPSP/FULL	Describes the Epsilon plastic state for shell. **Note:** Engine Keyword
/STATE/SHELL/ORTHL	Describes the orthotropy direction for shell. **Note:** Engine Keyword

Card	Description
/STATE/SHELL/STRAIN/FULL	Describes the strain state for shell. **Note:** Engine Keyword
/STATE/SHELL/STRESS/FULL	Describes the stress state for shell. **Note:** Engine Keyword
/STATE/STR_FILE	When writing the state file, `/INISHE`, `/INISH3` and `/INIBRI` will be written in a separate file which can be asked for being compressed (using GZIP) or not. The state file will then resume to the nodes coordinates and element connectivity. **Note:** Engine Keyword
/STOP	The Engine is stopped, due to reached energy error ratio criteria, total mass ratio criteria or nodal mass ratio criteria. **Note:** Engine Keyword
/TFILE	Defines the frequency of writing the time history file of T-file "`RunnameTnn`".
/TH/VERS	Generates the Time History files in Radioss environment post-processing format "Version Number". **Note:** Engine Keyword
/TITLE	Input a title. **Note:** Engine Keyword
/UNIT	Defines a local unit system for the keywords listed below. **Note:** Block Format Keyword

Card	Description
/UPWIND	Describes the upwind coefficient. **Note:** Block Format Keyword
/UPWM/SUPG	Streamline Upwind Petrov Galerkin formulation. **Note:** Engine Keyword
/UPWM/TG	Describes the Taylor Galerkin method for momentum advection. This method is not available with multi-material LAW51 (`/MAT/MULTIMAT`). **Note:** Engine Keyword
/VEL/ALE	With ALE links on grid velocities, given nodes are linked to the given master grid velocities. You can specify direction or a combination of directions. **Note:** Engine Keyword
/VEL/ROT	Given node numbers have the same rotational velocity in a specified direction X, Y or Z. **Note:** Engine Keyword
/VEL/TRA	Given node numbers have the same velocity in a specified direction X, Y or Z for material translational DOF. **Note:** Engine Keyword
/VERS	Identifies the input data version number.

Card	Description
	Note: Engine Keyword

Samcef

Card	Description
BOLT_OUTPUT	Defines the output code definitions for all .BOLT defined in the model.
UNIT	Defines the unit used to set up the model. **Note:** This control card is mandatory when beams are being defined in a model.
UNSUPPORTED_CARDS	

Undefined Entities

Undefined entities are entities that will not be written to the solver deck.

The import and export of undefined entities is fully supported for:

- Components
- Properties
- Materials
- Curves
- Sets
- Rigid Walls
- Cross Sections
- Accelerometers
- Boxes

Undefined entities imported from new include files will be organized in the master model.

If an entity is referenced in a solver deck, but is not defined, an undefined entity will be created and assigned the ID of the referenced entity on import. A default card image and a default name will also be assigned to the newly created undefined entity.

Referenced components are created as an undefined entity.

The defined state of an entity is indicated in the Model Browser, Defined column (Property and Material entity views), or in the Property Editor, Defined Entity field. An active checkbox indicates an entity is defined, whereas an inactive checkbox indicates an entity is undefined.

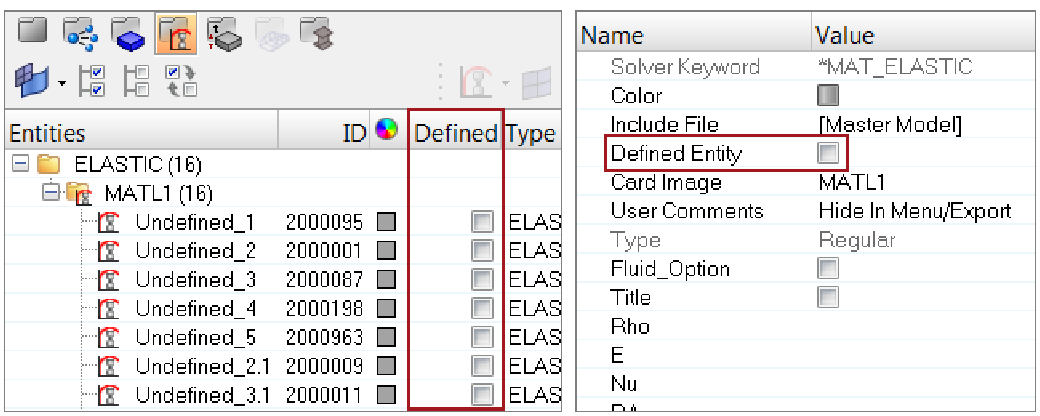


Figure 142:

Undefined entities exhibit the same behaviors as defined entities; they are visible in all relevant browsers and in the Property Editor. The defined state simply controls the export of the entity.

- If Defined is inactive, the entity not will be written to the solver deck.
- If Defined is active, the entity will be written to the solver deck.

The export status of undefined entities cannot be changed. If Defined is inactive, the export status will also be inactive in the Entity State Browser.

Resolve Undefined Entities

Undefined entities can be defined interactively in the session or via the import of a model that contains the referenced entities. This ensures the entity will be written to the solver deck.

- Define an entity interactively in the session by activating the **Defined** checkbox in the browser or Property Editor.

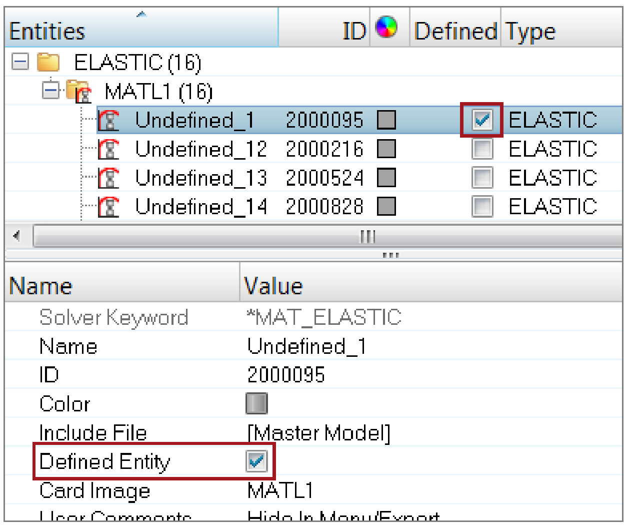


Figure 143:

- Define an entity via the import of a model containing the referenced entities.

  If additional entities residing in the solver deck do not exist, they will be imported as is.

  a) From the menu bar, click **File** > **Import** > **HyperMesh Model**.
  b) In the **Import File** dialog, open the model containing the referenced entities.
  c) In the **File Options** dialog, set the relevant option(s) to **Keep Incoming Attributes** and click **OK**.

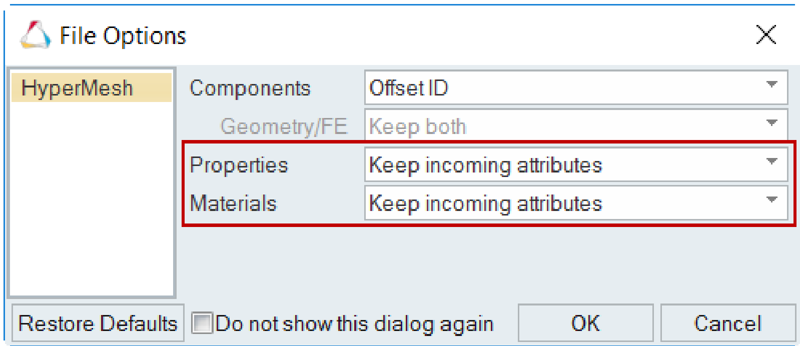


Figure 144:

Renumber Undefined Entities

The renumbering of undefined entities is restricted to ensure references to entities that reside in external Include files will not be invalidated.

1. In the Property Editor, ID field, enter a new number for the undefined entity.
2. In the dialog that appears, click **Yes** to execute the renumbering operation.

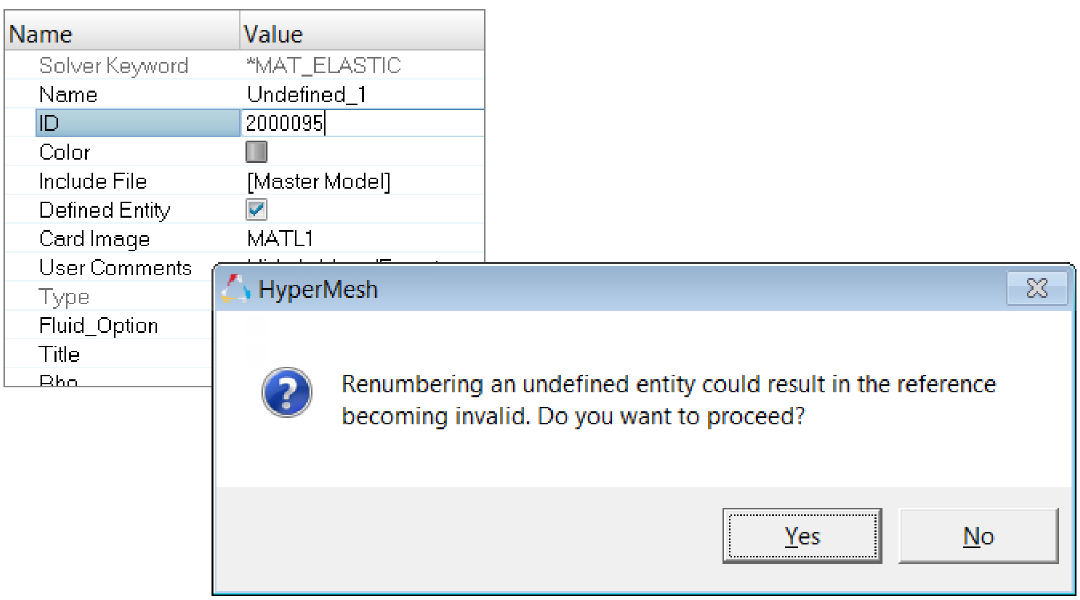


Figure 145:

Solver Encryption Entities

The encryption of solver entities is used to translate entity data into an undisclosed language.

Support of encrypted solver entities will allow you, for example, to manipulate encrypted dummies within the Dummy Browser.

You can view encrypted entities from the Model Browser, Solver Browser and References Browser. If an entity is encrypted, then its corresponding attributes will not be exposed in Property Editor, and curves cannot be reviewed or edited in the Curve Editor.

Undo and redo encryption actions made in HyperMesh using Undo/Redo.

LS-DYNA

Encrypted material and curve entities can be imported and exported.

PAM-CRASH 2G

Secure MATER, PART, FUNCT, and PLY can be imported and exported.

Radioss

Property and failure entities can be imported and exported.

Element Property and Material Assignement Rules

Element property and material assignment rules are based on the current user profile (solver interface).

There are two basic solver groups supported in HyperMesh; solver group1 and solver group2.

Solver Group 1

- OptiStruct
- Abaqus
- Nastran
- Samcef

Element property and material assignment rules for solver group 1:

- Components have no card images.
- Properties are assigned to elements or components using the following rules in order:
    1. If a property is assigned directly to an element, then that property is the elements property regardless of any other property assignments. Properties are assigned directly to elements on the Properties panel, Assign subpanel.
    2. If there is no property assigned directly to an element, then the property assigned to the component the element is organized into becomes the elements property. Properties are assigned to components on the Properties panel, Assign subpanel.
    3. If there is no property assigned to the component, then the element has no property assignment.
- Materials are always assigned to properties. Elements are assigned the material of their assigned property. If a property has no assigned material, then all elements assigned to that property have no material assignment. Materials are assigned to properties on the Properties panel, Assign subpanel.

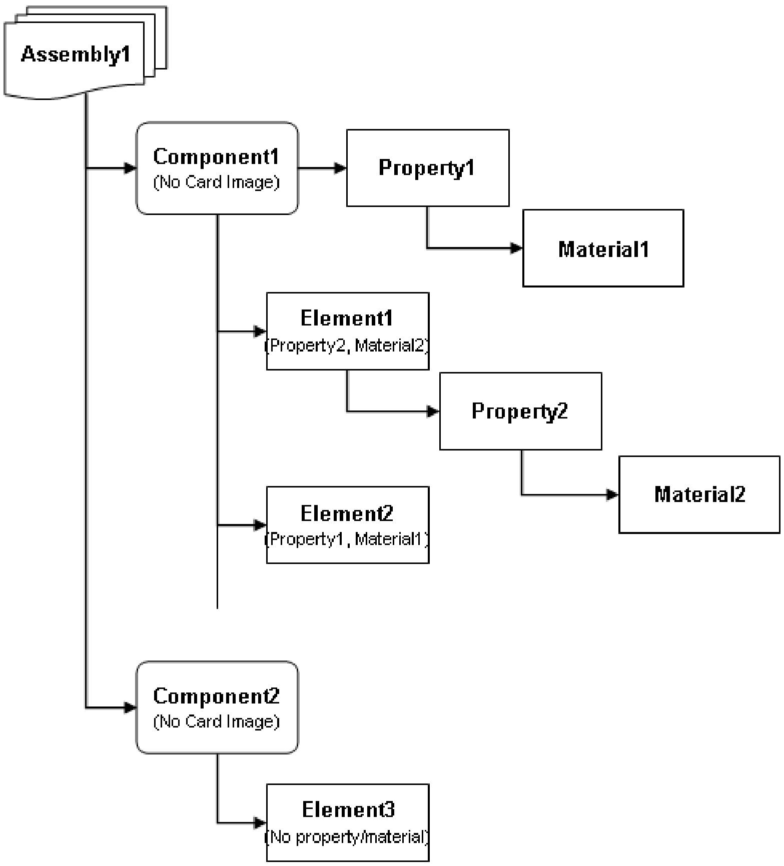


Figure 146: Solver Group1 Property/Material Assignment Schematic

Solver Group 2

- Radioss
- LS-DYNA
- PAM-CRASH
- ANSYS
- Permas

Element property and material assignment rules for solver group 2:

- Components have card images; typically "part" card images.
- Properties and materials are assigned to components only. There is no property or material assignment directly to elements. Properties and materials are assigned to components on the Components panel, Assign subpanel.
- Elements are assigned the property and material assigned to the component in which they are organized into.
- If a component is not assigned a property or material, then all elements within that component have no property or material assignment.

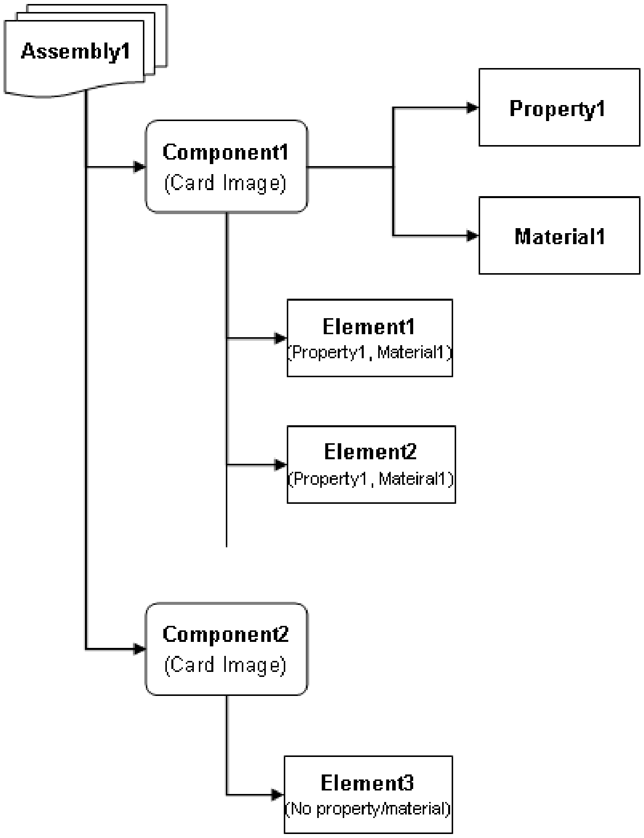


Figure 147: Solver Group 2 Property/Material Assignment Schematic

Supported Cards

Supported cards organized by solver.

Abaqus

- *AMPLITUDE
- *BEAM ADDED INERTIA
- *BEAM GENERAL SECTION
- *BEAM SECTION
- *BIAXIAL TEST DATA
- *BLOCKAGE
- *BOUNDARY
- *BRITTLE CRACKING
- *BRITTLE FAILURE
- *BRITTLE SHEAR
- *BUCKLE
- *BULK VISCOSITY
- *CECHARGE
- *CECURRENT
- *CFILM
- *CFLUX
- *CHANGE FRICTION
- *CLAY HARDENING
- *CLAY PLASTICITY
- *CLEARANCE
- *CLOAD
- *COHESIVE BEHAVIOR
- *COHESIVE SECTION
- *COMBINED TEST DATA
- *CONDUCTIVITY
- *CONNECTOR BEHAVIOR
- *CONNECTOR CONSTITUTIVE REFERENCE
- *CONNECTOR CONTACT FORCE
- *CONNECTOR DAMPING
- *CONNECTOR DERIVED COMPONENTS
- *CONNECTOR ELASTICITY
- *CONNECTOR FAILURE
- *CONNECTOR FRICTION
- *CONNECTOR HARDENING

*CONNECTOR LOAD
*CONNECTOR LOCK
*CONNECTOR MOTION
*CONNECTOR PLASTICITY
*CONNECTOR POTENTIAL
*CONNECTOR SECTION
*CONNECTOR STOP
*CONSTRAINT CONTROLS
*CONTACT (General Contact)
*CONTACT CLEARANCE
*CONTACT CLEARANCE ASSIGNMENT
*CONTACT CONTROLS
*CONTACT CONTROLS ASSIGNMENT
*CONTACT DAMPING (Explicit template)
*CONTACT DAMPING (Standard templates)
*CONTACT EXCLUSIONS
*CONTACT FILE
*CONTACT FORMULATION
*CONTACT INCLUSIONS
*CONTACT INTERFERENCE
*CONTACT OUTPUT
*CONTACT PAIR
*CONTACT PRINT
*CONTACT PROPERTY ASSIGNMENT
*CONTACT PROPERTY ASSIGNMENT, PROPERTY=OFFSET FRACTION
*CONTACT PROPERTY ASSIGNMENT, PROPERTY=THICKNESS
*CONTROLS
*COUPLED TEMP-DISPLACEMENT
*COUPLING
*CREEP
*CRUSHABLE FOAM
*CRUSHABLE FOAM HARDENING
*DAMPING
*DASHPOT
*DEBOND
*DECHARGE
*DENSITY
*DEPVAR
*DFLUX

*DIAGNOSTICS

*DIELECTRIC

*DISTRIBUTING

*DISTRIBUTING COUPLING

*DISTRIBUTION

*DISTRIBUTION_ANGLE

*DISTRIBUTION_DRAPE

*DISTRIBUTION_OFFSET

*DISTRIBUTION_ORIENTATION

*DISTRIBUTION_THICKNESS

*DLOAD

*DSLOAD

*DYNAMIC

*DYNAMIC (Explicit)

*EL FILE

*EL PRINT

*ELASTIC

*ELEMENT

*ELEMENT OUTPUT

*ELEMENT PROPERTIES

*ELGEN

*ELSET

*EMBEDDED ELEMENT

*ENERGY FILE

*ENERGY OUTPUT

*ENERGY PRINT

*EQUATION

*EULERIAN SECTION

*EXPANSION

*FABRIC

*FASTENER (SPOT WELD)

*FASTENER PROPERTY

*FILE FORMAT

*FILM

*FILTER

*FIXED MASS SCALING

*FLUID BEHAVIOR

*FLUID BULK

*FLUID DENSITY

*FLUID EXPANSION

*FLUID PROPERTY

*FRACTURE CRITERION

*FREQUENCY

*FRICTION (Explicit template)

*FRICTION (Standard templates)

*GAP

*GASKET BEHAVIOR

*GASKET CONTACT AREA

*GASKET ELASTICITY

*GASKET SECTION

*GASKET THICKNESS BEHAVIOR

*GEOSTATIC

*HEADING

*HEAT TRANSFER

*HYPERELASTIC

*HYPERFOAM

*INCREMENTATION OUTPUT

*INERTIA RELIEF

*INITIAL CONDITION (TYPE=STRESS)

*INITIAL CONDITION (TYPE=PRESSURE)

*INITIAL CONDITION (TYPE=TEMPERATURE)

*INITIAL CONDITION (TYPE=VELOCITY)

*INTEGRATED OUTPUT

*INTEGRATED OUTPUT SECTION

*ITS

*JOINT

*KINEMATIC

*KINEMATIC COUPLING

*LOAD CASE

*LOADING DATA

*LOW DENSITY FOAM

*M1

*M2

*MASS

*MATERIAL

*MEMBRANE SECTION

*MODAL_DAMPING

*MODAL DYNAMIC

*MODAL OUTPUT

*MODEL CHANGE

*MONITOR

*MPC

*MULLINS EFFECT

*NODAL THICKNESS

*NFILL

*NGEN

*NMAP

*NODE

*NODE FILE

*NODE OUTPUT

*NODE PRINT

*NONSTRUCTURAL MASS

*NSET

*ORIENTATION

*OUTPUT

*PARAMETER

*PHYSICAL CONSTANT

*PIEZOELECTRIC

*PLANAR TEST DATA

*PLASTIC

*PREPRINT

*PRE-TENSION SECTION

*PRINT

*RADIATE

*RADIATION VIEWFACTOR

*RATE DEPENDENT

*REBAR LAYER

*RELEASE

*RESPONSE SPECTRUM

*RESTART

*RESTART

*RIGID BODY

*ROTARY INERTIA

*SECTION CONTROLS

*SFILM

*SHEAR FAILURE

*SHEAR TEST DATA

Actran

AXISYMMETRY
BC_MESH
BOUNDARY_CONDITION
COUPLING_SURFACE
DIMENSION
DISCRETE
DISPLACEMENT
DISTRIBUTED_LOAD
DISTRIBUTED_PRESSURE
ELEMENT
FIELD_POINT_SURFACE
FREQUENCY_DOMAIN
IMPERVIOUS
INCIDENT_SURFACE
INFINITE ADMITTANCE
INFINITE_DOMAIN
INFINITE_ELEMENT
INFINITE_FLUID
INFINITE_MESH
INTERFACE
LIGHTHILL
MATERIAL (FLUID)
MATERIAL (SHELL)
MATERIAL (SOLID)
MATERIAL (POROUS_UP)
MATERIAL (POROUS_RIGID)
MATERIAL (VISCOTHERMAL)
MESH
MEAN_FLOW
MODAL_BASIS
MODAL_EXTRACTION
MODAL_SURFACE
NODE
OUTPUT_FRF
OUTPUT_MAP
POINT_LOAD
POROUS_UP
PRESSURE
RADIATING_SURFACE

RAYLEIGH_SURFACE

RIGID_POROUS

SAVE

SHELL

SOLID

SOLVER

SOURCE

STIFFENER

SUPER_CONNECTOR

SUPER_ELEMENT

TEMPERATURE_PRESSURE

TITLE

VELOCITY

VISCOTHERMAL

VISCOTHERMAL_FLUID

ANSYS

The input translator recognizes the ANSYS cards listed below. If an unsupported field is found in a card, a message is displayed on the status bar. The messages are also printed to the file `ansys.msg`. General slash commands, SOLUTION commands, POST1 commands, and POST26 commands are referred to as control cards. Unrecognized cards are written to a `*.hmx` file.

> **Note:** One component collector is created for every unique combination of Type, Real, Mat, and SECNUM. Surface loads on 1D elements is not supported. Property collectors are created for each real set defined in the ANSYS deck. Material collectors are also created for each material ID encountered. The component in HyperMesh is different from the component (CM) in ANSYS.

ACEL

ALPHAD

ANTYPE

ARCLEN

ARCTRM

/ASSIGN

AUTOTS

/BATCH

BEAM3

BEAM4

BEAM23

BEAM24

BEAM44

ConvFilmCo

/COPY

CP_ELEC

CP_STRUC

CRPLIM

D_CONSTRNT

D_TEMP

D_VOLT

DCGOMG

/DELETE

DELTIM

DIM

DMPRAT

DOF

DOMEGA

EMUNIT

EQSLV

ERESX

EORIENT

ETABLE

EXPASS

F_FLOW

F_HEAT

FLOTRAN

FLUID

FLUID29

FLUID30

FLUID80

FLUID116

FORCE

FORCE2

HARFRQ

HF118

HF119

HF120

HFLUX

HM_COMP

HREXP

HROPT

PSTRES
RBE3
RSYS
SECCONTROLS
SECDATA
SECOFFSET
SECTYPE
SFE
SHELL28
SHELL41
SHELL43
SHELL51
SHELL57
SHELL61
SHELL63
SHELL91
SHELL93
SHELL99
SHELL131
SHELL132
SHELL143
SHELL150
SHELL157
SHELL163
SHELL181
SHELL208
SHELL209
SHELL281
SLOAD
SOLID5
SOLID45
SOLID46
SOLID62
SOLID64
SOLID69
SOLID70
SOLID72
SOLID73
SOLID87

EXODUS

Truss
VELOCITY

Feko

/DI
/SK

LS-DYNA

*AIRBAG_ADIABATIC_GAS_MODEL
*AIRBAG_ALE
*AIRBAG_HYBRID
*AIRBAG_HYBRID_CHEMKIN
*AIRBAG_HYBRID_JETTING
*AIRBAG_HYBRID_JETTING_CM
*AIRBAG_INTERACTION
*AIRBAG_LINEAR_FLUID
*AIRBAG_LOAD_CURVE
*AIRBAG_PARTICLE
*AIRBAG_PARTICLE_DECOMPOSITION
*AIRBAG_PARTICLE_DECOMPOSITION_MOLEFRACTION
*AIRBAG_PARTICLE_MOLEFRACTION
*AIRBAG_PARTICLE_MPP
*AIRBAG_PARTICLE_MPP_DECOMPOSITION
*AIRBAG_PARTICLE_MPP_MOLEFRACTION
*AIRBAG_PARTICLE_MPP_DECOMPOSITION_MOLEFRACTION
*AIRBAG_REFERENCE_GEOMETRY
*AIRBAG_REFERENCE_GEOMETRY_BIRTH
*AIRBAG_REFERENCE_GEOMETRY_BIRTH_RDT
*AIRBAG_REFERENCE_GEOMETRY_RDT
*AIRBAG_SHELL_REFERENCE_GEOMETRY
*AIRBAG_SHELL_REFERENCE_GEOMETRY_RDT
*AIRBAG_SIMPLE_AIRBAG_MODEL
*AIRBAG_SIMPLE_PRESSURE_VOLUME
*AIRBAG_WANG_NEFSKE
*AIRBAG_WANG_NEFSKE_JETTING
*AIRBAG_WANG_NEFSKE_JETTING_CM
*AIRBAG_WANG_NEFSKE_JETTING_POP
*AIRBAG_WANG_NEFSKE_JETTING_POP_CM
*AIRBAG_WANG_NEFSKE_MULTIPLE_JETTING

*AIRBAG_WANG_NEFSKE_MULTIPLE_JETTING_CM
*AIRBAG_WANG_NEFSKE_MULTIPLE_JETTING_POP
*AIRBAG_WANG_NEFSKE_MULTIPLE_JETTING_POP_CM
*AIRBAG_WANG_NEFSKE_POP
*ALE_FSI_PROJECTION
*ALE_MULTI-MATERIAL_GROUP
*ALE_REFERENCE_SYSTEM_CURVE
*ALE_REFERENCE_SYSTEM_GROUP
*ALE_REFERENCE_SYSTEM_NODE
*ALE_REFERENCE_SYSTEM_SWITCH
*ALE_SMOOTHING
*ALE_TANK_TEST
*BOUNDARY_AMBIENT_EOS
*BOUNDARY_CONVECTION_SET
*BOUNDARY_CYCLIC
*BOUNDARY_FLUX_SET
*BOUNDARY_NON_REFLECTING
*BOUNDARY_NON_REFLECTING_2D
*BOUNDARY_PRESCRIBED_ACCELEROMETER_RIGID
*BOUNDARY_PRESCRIBED_MOTION_NODE(Accl)
*BOUNDARY_PRESCRIBED_MOTION_NODE(Disp)
*BOUNDARY_PRESCRIBED_MOTION_NODE(Vel)
*BOUNDARY_PRESCRIBED_MOTION_NODE
*BOUNDARY_PRESCRIBED_MOTION_RIGID(Accl)
*BOUNDARY_PRESCRIBED_MOTION_RIGID(Disp)
*BOUNDARY_PRESCRIBED_MOTION_RIGID(Vel)
*BOUNDARY_PRESCRIBED_MOTION_RIGID
*BOUNDARY_PRESCRIBED_MOTION_RIGID_LOCAL
*BOUNDARY_PRESCRIBED_MOTION_RIGID_LOCAL(Accl)
*BOUNDARY_PRESCRIBED_MOTION_RIGID_LOCAL(Disp)
*BOUNDARY_PRESCRIBED_MOTION_RIGID_LOCAL(Vel)
*BOUNDARY_PRESCRIBED_MOTION_RIGID_LOCAL
*BOUNDARY_PRESCRIBED_MOTION_SET
*BOUNDARY_PRESCRIBED_MOTION_SET(Accl)
*BOUNDARY_PRESCRIBED_MOTION_SET(Disp)
*BOUNDARY_PRESCRIBED_MOTION_SET(Vel)
*BOUNDARY_RADIATION_SET
*BOUNDARY_SPC_NODE
*BOUNDARY_SPC_NODE_BIRTH_DEATH

*BOUNDARY_SPC_SET
*BOUNDARY_SPC_SET_BIRTH_DEATH
*BOUNDARY_SPH_FLOW
*BOUNDARY_TEMPERATURE_NODE
*BOUNDARY_TEMPERATURE_SET
*CONSTRAINED_BUTT_WELD
*CONSTRAINED_EXTRA_NODES_NODE
*CONSTRAINED_EXTRA_NODES_SET
*CONSTRAINED_GENERALIZED_WELD_BUTT
*CONSTRAINED_GENERALIZED_WELD_COMBINED
*CONSTRAINED_GENERALIZED_WELD_CROSS_FILLET
*CONSTRAINED_GENERALIZED_WELD_FILLET
*CONSTRAINED_GENERALIZED_WELD_SPOT
*CONSTRAINED_GLOBAL
*CONSTRAINED_INTERPOLATION
*CONSTRAINED_INTERPOLATION_LOCAL
*CONSTRAINED_JOINT_CONSTANT_VELOCITY_FAILURE
*CONSTRAINED_JOINT_CONSTANT_VELOCITY
*CONSTRAINED_JOINT_CONSTANT_VELOCITY_LOCAL_FAILURE
*CONSTRAINED_JOINT_CONSTANT_VELOCITY_LOCAL
*CONSTRAINED_JOINT_CYLINDRICAL
*CONSTRAINED_JOINT_CYLINDRICAL_FAILURE
*CONSTRAINED_JOINT_CYLINDRICAL_LOCAL_FAILURE
*CONSTRAINED_JOINT_CYLINDRICAL_LOCAL
*CONSTRAINED_JOINT_GEARS_FAILURE
*CONSTRAINED_JOINT_GEARS
*CONSTRAINED_JOINT_GEARS_LOCAL_FAILURE
*CONSTRAINED_JOINT_GEARS_LOCAL
*CONSTRAINED_JOINT_LOCKING_FAILURE
*CONSTRAINED_JOINT_LOCKING
*CONSTRAINED_JOINT_LOCKING_LOCAL_FAILURE
*CONSTRAINED_JOINT_LOCKING_LOCAL
*CONSTRAINED_JOINT_PLANAR
*CONSTRAINED_JOINT_PLANAR_FAILURE
*CONSTRAINED_JOINT_PLANAR_LOCAL_FAILURE
*CONSTRAINED_JOINT_PLANAR_LOCAL
*CONSTRAINED_JOINT_PULLEY_FAILURE
*CONSTRAINED_JOINT_PULLEY
*CONSTRAINED_JOINT_PULLEY_LOCAL_FAILURE

*CONSTRAINED_JOINT_PULLEY_LOCAL
*CONSTRAINED_JOINT_RACK_AND_PINION_FAILURE
*CONSTRAINED_JOINT_RACK_AND_PINION
*CONSTRAINED_JOINT_RACK_AND_PINION_LOCAL_FAILURE
*CONSTRAINED_JOINT_RACK_AND_PINION_LOCAL
*CONSTRAINED_JOINT_REVOLUTE
*CONSTRAINED_JOINT_REVOLUTE_FAILURE
*CONSTRAINED_JOINT_REVOLUTE_LOCAL_FAILURE
*CONSTRAINED_JOINT_REVOLUTE_LOCAL
*CONSTRAINED_JOINT_ROTATIONAL_MOTOR_FAILURE
*CONSTRAINED_JOINT_ROTATIONAL_MOTOR
*CONSTRAINED_JOINT_ROTATIONAL_MOTOR_LOCAL_FAILURE
*CONSTRAINED_JOINT_ROTATIONAL_MOTOR_LOCAL
*CONSTRAINED_JOINT_SCREW_FAILURE
*CONSTRAINED_JOINT_SCREW
*CONSTRAINED_JOINT_SCREW_LOCAL_FAILURE
*CONSTRAINED_JOINT_SCREW_LOCAL
*CONSTRAINED_JOINT_SPHERICAL
*CONSTRAINED_JOINT_SPHERICAL_FAILURE
*CONSTRAINED_JOINT_SPHERICAL
*CONSTRAINED_JOINT_SPHERICAL_LOCAL_FAILURE
*CONSTRAINED_JOINT_SPHERICAL_LOCAL
*CONSTRAINED_JOINT_STIFFNESS_FLEXION-TORSION
*CONSTRAINED_JOINT_STIFFNESS_GENERALIZED
*CONSTRAINED_JOINT_STIFFNESS_TRANSLATIONAL
*CONSTRAINED_JOINT_TRANSLATIONAL
*CONSTRAINED_JOINT_TRANSLATIONAL_FAILURE
*CONSTRAINED_JOINT_TRANSLATIONAL
*CONSTRAINED_JOINT_TRANSLATIONAL_LOCAL_FAILURE
*CONSTRAINED_JOINT_TRANSLATIONAL_LOCAL
*CONSTRAINED_JOINT_TRANSLATIONAL_MOTOR_FAILURE
*CONSTRAINED_JOINT_TRANSLATIONAL_MOTOR
*CONSTRAINED_JOINT_TRANSLATIONAL_MOTOR_LOCAL_FAILURE
*CONSTRAINED_JOINT_TRANSLATIONAL_MOTOR_LOCAL
*CONSTRAINED_JOINT_UNIVERSAL
*CONSTRAINED_JOINT_UNIVERSAL_FAILURE
*CONSTRAINED_JOINT_UNIVERSAL_LOCAL_FAILURE
*CONSTRAINED_JOINT_UNIVERSAL_LOCAL
*CONSTRAINED_LAGRANGE_IN_SOLID

*CONSTRAINED_LAGRANGE_IN_SOLID_EDGE
*CONSTRAINED_NODAL_RIGID_BODY
*CONSTRAINED_NODAL_RIGID_BODY_INERTIA
*CONSTRAINED_NODAL_RIGID_BODY_SPC
*CONSTRAINED_NODAL_RIGID_BODY_SPC_INERTIA
*CONSTRAINED_NODE_SET
*CONSTRAINED_RIGID_BODIES
*CONSTRAINED_RIGID_BODY_STOPPERS
*CONSTRAINED_RIVET
*CONSTRAINED_SHELL_TO_SOLID
*CONSTRAINED_SPOTWELD
*CONSTRAINED_SPOTWELD_FILTERED_FORCE
*CONSTRAINED_TIE-BREAK
*CONSTRAINED_TIED_NODES_FAILURE
*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE
*CONTACT_2D_AUTOMATIC_SURFACE_TO_SURFACE_THERMAL
*CONTACT_2D_AUTOMATIC_TIED
*CONTACT_AIRBAG_SINGLE_SURFACE
*CONTACT_AIRBAG_SINGLE_SURFACE_MPP
*CONTACT_AUTO_MOVE
*CONTACT_AUTOMATIC_BEAMS_TO_SURFACE
*CONTACT_AUTOMATIC_BEAMS_TO_SURFACE_MPP
*CONTACT_AUTOMATIC_GENERAL_EDGEONLY
*CONTACT_AUTOMATIC_GENERAL_EDGEONLY_MPP
*CONTACT_AUTOMATIC_GENERAL
*CONTACT_AUTOMATIC_GENERAL_INTERIOR
*CONTACT_AUTOMATIC_GENERAL_INTERIOR_MPP
*CONTACT_AUTOMATIC_GENERAL_MPP
*CONTACT_AUTOMATIC_NODES_TO_SURFACE
*CONTACT_AUTOMATIC_NODES_TO_SURFACE_MPP
*CONTACT_AUTOMATIC_NODES_TO_SURFACE_SMOOTH
*CONTACT_AUTOMATIC_NODES_TO_SURFACE_SMOOTH_MPP
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_MPP
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_MPP
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION_MPP
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL

*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL_MPP
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_THERMAL_FRICTION
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_THERMAL_FRICTION_MPP
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_THERMAL
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_THERMAL_MPP
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK_MPP
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK_THERMAL_FRICTION
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK_THERMAL_FRICTION_MPP
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK_THERMAL
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK_THERMAL_MPP
*CONTACT_AUTOMATIC_SINGLE_SURFACE
*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR
*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR_MPP
*CONTACT_AUTOMATIC_SINGLE_SURFACE_MPP
*CONTACT_AUTOMATIC_SINGLE_SURFACE_SMOOTH
*CONTACT_AUTOMATIC_SINGLE_SURFACE_SMOOTH_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_COMPOSITE
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_COMPOSITE_THERMAL
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_COMPOSITE_THERMAL_FRICTION
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_THERMAL_FRICTION
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_THERMAL_FRICTION_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_THERMAL
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_THERMAL_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED_THERMAL_FRICTION
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED_THERMAL_FRICTION_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED_THERMAL
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED_THERMAL_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH_THERMAL

*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH_THERMAL_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_THERMAL_FRICTION
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_THERMAL_FRICTION_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_THERMAL
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_THERMAL_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_THERMAL_FRICTION
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_THERMAL_FRICTION_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_THERMAL
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_THERMAL_MPP
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIED_WELD
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIED_WELD_THERMAL
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIED_WELD_THERMAL_FRICTION
*CONTACT_CONSTRAINT_NODES_TO_SURFACE
*CONTACT_CONSTRAINT_NODES_TO_SURFACE_MPP
*CONTACT_CONSTRAINT_SURFACE_TO_SURFACE
*CONTACT_CONSTRAINT_SURFACE_TO_SURFACE_MPP
*CONTACT_CONSTRAINT_SURFACE_TO_SURFACE_THERMAL_FRICTION
*CONTACT_CONSTRAINT_SURFACE_TO_SURFACE_THERMAL_FRICTION_MPP
*CONTACT_CONSTRAINT_SURFACE_TO_SURFACE_THERMAL
*CONTACT_CONSTRAINT_SURFACE_TO_SURFACE_THERMAL_MPP
*CONTACT_DRAWBEAD
*CONTACT_DRAWBEAD_MPP
*CONTACT_ENTITY
*CONTACT_ERODING_NODES_TO_SURFACE
*CONTACT_ERODING_NODES_TO_SURFACE_MPP
*CONTACT_ERODING_SINGLE_SURFACE
*CONTACT_ERODING_SINGLE_SURFACE_MPP
*CONTACT_ERODING_SURFACE_TO_SURFACE
*CONTACT_ERODING_SURFACE_TO_SURFACE_MPP
*CONTACT_ERODING_SURFACE_TO_SURFACE_THERMAL_FRICTION
*CONTACT_ERODING_SURFACE_TO_SURFACE_THERMAL_FRICTION_MPP
*CONTACT_ERODING_SURFACE_TO_SURFACE_THERMAL
*CONTACT_ERODING_SURFACE_TO_SURFACE_THERMAL_MPP
*CONTACT_FORCE_TRANSDUCER_CONSTRAINT
*CONTACT_FORCE_TRANSDUCER_CONSTRAINT_MPP
*CONTACT_FORCE_TRANSDUCER_PENALTY
*CONTACT_FORCE_TRANSDUCER_PENALTY_MPP

*CONTACT_FORMING_NODES_TO_SURFACE
*CONTACT_FORMING_NODES_TO_SURFACE_MPP
*CONTACT_FORMING_NODES_TO_SURFACE_SMOOTH
*CONTACT_FORMING_NODES_TO_SURFACE_SMOOTH_MPP
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_MPP
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_MPP
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION_MPP
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL_MPP
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_THERMAL_FRICTION
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_THERMAL_FRICTION_MPP
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_THERMAL
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_THERMAL_MPP
*CONTACT_FORMING_SURFACE_TO_SURFACE
*CONTACT_FORMING_SURFACE_TO_SURFACE_MORTAR
*CONTACT_FORMING_SURFACE_TO_SURFACE_MORTAR_MPP
*CONTACT_FORMING_SURFACE_TO_SURFACE_MORTAR_THERMAL_FRICTION
*CONTACT_FORMING_SURFACE_TO_SURFACE_MORTAR_THERMAL_FRICTION_MPP
*CONTACT_FORMING_SURFACE_TO_SURFACE_MORTAR_THERMAL
*CONTACT_FORMING_SURFACE_TO_SURFACE_MORTAR_THERMAL_MPP
*CONTACT_FORMING_SURFACE_TO_SURFACE_MPP
*CONTACT_FORMING_SURFACE_TO_SURFACE_SMOOTH
*CONTACT_FORMING_SURFACE_TO_SURFACE_SMOOTH_MPP
*CONTACT_FORMING_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION
*CONTACT_FORMING_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION_MPP
*CONTACT_FORMING_SURFACE_TO_SURFACE_SMOOTH_THERMAL
*CONTACT_FORMING_SURFACE_TO_SURFACE_SMOOTH_THERMAL_MPP
*CONTACT_FORMING_SURFACE_TO_SURFACE_THERMAL_FRICTION
*CONTACT_FORMING_SURFACE_TO_SURFACE_THERMAL_FRICTION_MPP
*CONTACT_FORMING_SURFACE_TO_SURFACE_THERMAL
*CONTACT_FORMING_SURFACE_TO_SURFACE_THERMAL_MPP
*CONTACT_GUIDED_CABLE
*CONTACT_INTERIOR
*CONTACT_NODES_TO_SURFACE
*CONTACT_NODES_TO_SURFACE_INTERFERENCE
*CONTACT_NODES_TO_SURFACE_INTERFERENCE_MPP

*CONTACT_NODES_TO_SURFACE_MPP
*CONTACT_NODES_TO_SURFACE_SMOOTH
*CONTACT_NODES_TO_SURFACE_SMOOTH_MPP
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE_MPP
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE_THERMAL_FRICTION
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE_THERMAL_FRICTION_MPP
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE_THERMAL
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE_THERMAL_MPP
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_MPP
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_MPP
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION_MPP
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH_THERMAL_MPP
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_THERMAL_FRICTION
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_THERMAL_FRICTION_MPP
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_THERMAL
*CONTACT_ONE_WAY_SURFACE_TO_SURFACE_THERMAL_MPP
*CONTACT_RIGID_BODY_ONE_WAY_TO_RIGID_BODY
*CONTACT_RIGID_BODY_ONE_WAY_TO_RIGID_BODY_MPP
*CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BODY
*CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BODY_MPP
*CONTACT_RIGID_NODES_TO_RIGID_BODY
*CONTACT_RIGID_NODES_TO_RIGID_BODY_MPP
*CONTACT_RIGID_SURFACE
*CONTACT_SINGLE_EDGE
*CONTACT_SINGLE_EDGE_MPP
*CONTACT_SINGLE_SURFACE
*CONTACT_SINGLE_SURFACE_MPP
*CONTACT_SLIDING_ONLY
*CONTACT_SLIDING_ONLY_MPP
*CONTACT_SLIDING_ONLY_PENALTY
*CONTACT_SLIDING_ONLY_PENALTY_MPP
*CONTACT_SPOTWELD_BEAM_OFFSET
*CONTACT_SPOTWELD_BEAM_OFFSET_MPP
*CONTACT_SPOTWELD_CONSTRAINED_OFFSET

*CONTACT_SPOTWELD_CONSTRAINED_OFFSET_MPP
*CONTACT_SPOTWELD
*CONTACT_SPOTWELD_MPP
*CONTACT_SPOTWELD_WITH_TORSION
*CONTACT_SPOTWELD_WITH_TORSION_MPP
*CONTACT_SURFACE_TO_SURFACE_CONTRACTION_JOINT
*CONTACT_SURFACE_TO_SURFACE_CONTRACTION_JOINT_MPP
*CONTACT_SURFACE_TO_SURFACE_CONTRACTION_JOINT_THERMAL_FRICTION
*CONTACT_SURFACE_TO_SURFACE_CONTRACTION_JOINT_THERMAL_FRICTION_MPP
*CONTACT_SURFACE_TO_SURFACE_CONTRACTION_JOINT_THERMAL
*CONTACT_SURFACE_TO_SURFACE_CONTRACTION_JOINT_THERMAL_MPP
*CONTACT_SURFACE_TO_SURFACE
*CONTACT_SURFACE_TO_SURFACE_INTERFERENCE
*CONTACT_SURFACE_TO_SURFACE_INTERFERENCE_MPP
*CONTACT_SURFACE_TO_SURFACE_INTERFERENCE_THERMAL_FRICTION
*CONTACT_SURFACE_TO_SURFACE_INTERFERENCE_THERMAL_FRICTION_MPP
*CONTACT_SURFACE_TO_SURFACE_INTERFERENCE_THERMAL
*CONTACT_SURFACE_TO_SURFACE_INTERFERENCE_THERMAL_MPP
*CONTACT_SURFACE_TO_SURFACE_MPP
*CONTACT_SURFACE_TO_SURFACE_SMOOTH
*CONTACT_SURFACE_TO_SURFACE_SMOOTH_MPP
*CONTACT_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION
*CONTACT_SURFACE_TO_SURFACE_SMOOTH_THERMAL_FRICTION_MPP
*CONTACT_SURFACE_TO_SURFACE_SMOOTH_THERMAL
*CONTACT_SURFACE_TO_SURFACE_SMOOTH_THERMAL_MPP
*CONTACT_SURFACE_TO_SURFACE_THERMAL_FRICTION
*CONTACT_SURFACE_TO_SURFACE_THERMAL_FRICTION_MPP
*CONTACT_SURFACE_TO_SURFACE_THERMAL
*CONTACT_SURFACE_TO_SURFACE_THERMAL_MPP
*CONTACT_TIEBREAK_NODES_TO_SURFACE
*CONTACT_TIEBREAK_NODES_TO_SURFACE_MPP
*CONTACT_TIEBREAK_SURFACE_TO_SURFACE
*CONTACT_TIEBREAK_SURFACE_TO_SURFACE_MPP
*CONTACT_TIEBREAK_SURFACE_TO_SURFACE_THERMAL_FRICTION
*CONTACT_TIEBREAK_SURFACE_TO_SURFACE_THERMAL_FRICTION_MPP
*CONTACT_TIEBREAK_SURFACE_TO_SURFACE_THERMAL
*CONTACT_TIEBREAK_SURFACE_TO_SURFACE_THERMAL_MPP
*CONTACT_TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET
*CONTACT_TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET_MPP

*CONTACT_TIED_NODES_TO_SURFACE
*CONTACT_TIED_NODES_TO_SURFACE_MPP
*CONTACT_TIED_NODES_TO_SURFACE_OFFSET
*CONTACT_TIED_NODES_TO_SURFACE_OFFSET_MPP
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET_MPP
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET_MPP
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_MPP
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_OFFSET
*CONTACT_TIED_SHELL_EDGE_TO_SURFACE_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_CONSTRAINED_OFFSET
*CONTACT_TIED_SURFACE_TO_SURFACE_CONSTRAINED_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_CONSTRAINED_OFFSET
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_CONSTRAINED_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_OFFSET
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL_CONSTRAINED_OFFSET
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL_CONSTRAINED_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL_FRICTION_CONSTRAINED_OFFSET
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL_FRICTION_CONSTRAINED_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL_FRICTION
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL_FRICTION_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL_FRICTION_OFFSET
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL_FRICTION_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL_OFFSET
*CONTACT_TIED_SURFACE_TO_SURFACE_FAILURE_THERMAL_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE
*CONTACT_TIED_SURFACE_TO_SURFACE_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_OFFSET
*CONTACT_TIED_SURFACE_TO_SURFACE_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL_CONSTRAINED_OFFSET
*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL_CONSTRAINED_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL_FRICTION_CONSTRAINED_OFFSET

*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL_FRICTION_CONSTRAINED_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL_FRICTION
*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL_FRICTION_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL_FRICTION_OFFSET
*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL_FRICTION_OFFSET_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL
*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL_MPP
*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL_OFFSET
*CONTACT_TIED_SURFACE_TO_SURFACE_THERMAL_OFFSET_MPP
*CONTROL_ACCURACY
*CONTROL_ADAPSTEP
*CONTROL_ADAPTIVE
*CONTROL_ADAPTIVE_CURVE
*CONTROL_ALE
*CONTROL_BULK_VISCOSITY
*CONTROL_CHECK
*CONTROL_CHECK_SHELL
*CONTROL_COARSEN
*CONTROL_CONTACT
*CONTROL_COUPLING
*CONTROL_CPM
*CONTROL_CPU
*CONTROL_DEBUG
*CONTROL_DYNAMIC_RELAXATION
*CONTROL_EFG
*CONTROL_ENERGY
*CONTROL_EXPLOSIVE_SHADOW
*CONTROL_FORMING_AUTOCHECK
*CONTROL_FORMING_AUTOPOSITION_PARAMETER
*CONTROL_FORMING_AUTOPOSITION_PARAMETER_SET
*CONTROL_HOURGLASS
*CONTROL_IMPLICIT_AUTO
*CONTROL_IMPLICIT_BUCKLE
*CONTROL_IMPLICIT_DYNAMICS
*CONTROL_IMPLICIT_EIGENVALUE
*CONTROL_IMPLICIT_FORMING
*CONTROL_IMPLICIT_GENERAL
*CONTROL_IMPLICIT_INERTIA_RELIEF
*CONTROL_IMPLICIT_MODAL_DYNAMIC_DAMPING

*CONTROL_IMPLICIT_MODAL_DYNAMIC_DAMPING_FREQUENCY_RANGE
*CONTROL_IMPLICIT_MODAL_DYNAMIC_DAMPING_SPECIFIC
*CONTROL_IMPLICIT_MODAL_DYNAMIC_MODE_GENERATE
*CONTROL_IMPLICIT_MODAL_DYNAMIC_MODE_LIST
*CONTROL_IMPLICIT_MODES
*CONTROL_IMPLICIT_MODES_BINARY
*CONTROL_IMPLICIT_SOLUTION
*CONTROL_IMPLICIT_SOLVER
*CONTROL_IMPLICIT_STABILIZATION
*CONTROL_IMPLICIT_TERMINATION
*CONTROL_IMPLICIT_ROTATIONAL_DYNAMICS
*CONTROL_MPP_CONTACT_GROUPABLE
*CONTROL_MPP_DECOMPOSITION_ARRANGE_PARTS
*CONTROL_MPP_DECOMPOSITION_ARRANGE_PARTS_LOCAL
*CONTROL_MPP_DECOMPOSITION_AUTOMATIC
*CONTROL_MPP_DECOMPOSITION_BAGREF
*CONTROL_MPP_DECOMPOSITION_CHECK_SPEED
*CONTROL_MPP_DECOMPOSITION_CONTACT_DISTRIBUTE
*CONTROL_MPP_DECOMPOSITION_CONTACT_DISTRIBUTE_LOCAL
*CONTROL_MPP_DECOMPOSITION_CONTACT_ISOLATE
*CONTROL_MPP_DECOMPOSITION_DISABLE_UNREF_CURVES
*CONTROL_MPP_DECOMPOSITION_DISTRIBUTE_ALE_ELEMENTS
*CONTROL_MPP_DECOMPOSITION_DISTRIBUTE_SPH_ELEMENTS
*CONTROL_MPP_DECOMPOSITION_ELCOST
*CONTROL_MPP_DECOMPOSITION_FILE
*CONTROL_MPP_DECOMPOSITION_METHOD
*CONTROL_MPP_DECOMPOSITION_NUMPROC
*CONTROL_MPP_DECOMPOSITION_OUTDECOMP
*CONTROL_MPP_DECOMPOSITION_PARTS_DISTRIBUTE
*CONTROL_MPP_DECOMPOSITION_PARTS_DISTRIBUTE_LOCAL
*CONTROL_MPP_DECOMPOSITION_PARTSET_DISTRIBUTE
*CONTROL_MPP_DECOMPOSITION_PARTSET_DISTRIBUTE_LOCAL
*CONTROL_MPP_DECOMPOSITION_RCBLOG
*CONTROL_MPP_DECOMPOSITION_SCALE_CONTACT_COST
*CONTROL_MPP_DECOMPOSITION_SCALE_FACTOR_SPH
*CONTROL_MPP_DECOMPOSITION_SHOW
*CONTROL_MPP_DECOMPOSITION_TRANSFORMATION
*CONTROL_MPP_IO_LSTC_REDUCE
*CONTROL_MPP_IO_NOBEAMOUT

*CONTROL_MPP_IO_NOD3DUMP

*CONTROL_MPP_IO_NODUMP

*CONTROL_MPP_IO_NOFULL

*CONTROL_MPP_IO_SWAPBYTES

*CONTROL_OUTPUT

*CONTROL_PARALLEL

*CONTROL_REMESHING

*CONTROL_REMESHING_EFG

*CONTROL_RIGID

*CONTROL_SHELL

*CONTROL_SOLID

*CONTROL_SOLUTION

*CONTROL_SPH

*CONTROL_SPOTWELD_BEAM

*CONTROL_STAGED_CONSTRUCTION

*CONTROL_STRUCTURED

*CONTROL_STRUCTURED_TERM

*CONTROL_SUBCYCLE

*CONTROL_SUBCYCLE_MASS_SCALED_PART

*CONTROL_SUBCYCLE_MASS_SCALED_PART_SET

*CONTROL_TERMINATION

*CONTROL_THERMAL_NONLINEAR

*CONTROL_THERMAL_SOLVER

*CONTROL_THERMAL_TIMESTEP

*CONTROL_TIMESTEP

*DAMPING_FREQUENCY_RANGE

*DAMPING_FREQUENCY_RANGE_DEFORM

*DAMPING_GLOBAL

*DAMPING_PART_MASS

*DAMPING_PART_MASS_SET

*DAMPING_PART_STIFFNESS

*DAMPING_PART_STIFFNESS_SET

*DAMPING_RELATIVE

*DATABASE_ABSTAT

*DATABASE_ABSTAT_CPM

*DATABASE_ATDOUT

*DATABASE_AVSFLT

*DATABASE_BINARY_BLSTFOR

*DATABASE_BINARY_CPMFOR

*DATABASE_BINARY_D3CRACK

*DATABASE_BINARY_D3DRLF

*DATABASE_BINARY_D3DUMP

*DATABASE_BINARY_D3PART

*DATABASE_BINARY_D3PLOT

*DATABASE_BINARY_D3PROP

*DATABASE_BINARY_D3THDT

*DATABASE_BINARY_DEMFOR

*DATABASE_BINARY_FSIFOR

*DATABASE_BINARY_INTFOR

*DATABASE_BINARY_RUNRSF

*DATABASE_BINARY_XTFILE

*DATABASE_BNDOUT

*DATABASE_CPM_SENSOR

*DATABASE_CROSS_SECTION_PLANE

*DATABASE_CROSS_SECTION_SET

*DATABASE_CURVOUT

*DATABASE_DCFAIL

*DATABASE_DEFGEO

*DATABASE_DEFORC

*DATABASE_DISBOUT

*DATABASE_ELOUT

*DATABASE_EXTENT_AVS

*DATABASE_EXTENT_BINARY

*DATABASE_EXTENT_BINARY_COMP

*DATABASE_EXTENT_D3PART

*DATABASE_EXTENT_MOVIE

*DATABASE_EXTENT_MPGS

*DATABASE_EXTENT_SSSTAT

*DATABASE_FORMAT

*DATABASE_FSI

*DATABASE_FSI_SENSOR

*DATABASE_GCEOUT

*DATABASE_GLSTAT

*DATABASE_GLSTAT_MASS_PROPERTIES

*DATABASE_H3OUT

*DATABASE_HISTORY_BEAM

*DATABASE_HISTORY_BEAM_SET

*DATABASE_HISTORY_DISCRETE

*DATABASE_HISTORY_DISCRETE_SET
*DATABASE_HISTORY_NODE
*DATABASE_HISTORY_NODE_LOCAL
*DATABASE_HISTORY_NODE_SET
*DATABASE_HISTORY_NODE_SET_LOCAL
*DATABASE_HISTORY_SEATBELT
*DATABASE_HISTORY_SHELL
*DATABASE_HISTORY_SHELL_SET
*DATABASE_HISTORY_SOLID
*DATABASE_HISTORY_SOLID_SET
*DATABASE_HISTORY_SPH
*DATABASE_HISTORY_SPH_SET
*DATABASE_HISTORY_TSHELL
*DATABASE_HISTORY_TSHELL_SET
*DATABASE_JNTFORC
*DATABASE_MASSOUT
*DATABASE_MATSUM
*DATABASE_MOVIE
*DATABASE_MPGS
*DATABASE_NCFORC
*DATABASE_NCFORC_FILTER
*DATABASE_NODAL_FORCE_GROUP
*DATABASE_NODFOR
*DATABASE_NODOUT
*DATABASE_RBDOUT
*DATABASE_RCFORC
*DATABASE_RWFORC
*DATABASE_SBTOUT
*DATABASE_SECFORC
*DATABASE_SLEOUT
*DATABASE_SPCFORC
*DATABASE_SPHOUT
*DATABASE_SPRING_FORWARD
*DATABASE_SSSTAT
*DATABASE_SSSTAT_MASS_PROPERTIES
*DATABASE_SUPERPLASTIC_FORMING
*DATABASE_SWFORC
*DATABASE_TPRINT
*DATABASE_TRACER

*DATABASE_TRHIST
*DEFINE_ALEBAG_BAG
*DEFINE_ALEBAG_HOLE
*DEFINE_ALEBAG_INFLATOR
*DEFINE_BOX
*DEFINE_BOX_ADAPTIVE
*DEFINE_BOX_ADAPTIVE_LOCAL
*DEFINE_BOX_COARSEN
*DEFINE_BOX_COARSEN_LOCAL
*DEFINE_BOX_DRAWBEAD
*DEFINE_BOX_LOCAL
*DEFINE_BOX_SPH
*DEFINE_BOX_SPH_LOCAL
*DEFINE_CONNECTION_PROPERTIES
*DEFINE_CONNECTION_PROPERTIES_ADD
*DEFINE_COORDINATE_NODES
*DEFINE_COORDINATE_SYSTEM
*DEFINE_COORDINATE_VECTOR
*DEFINE_CPM_BAG_INTERACTION
*DEFINE_CPM_CHAMBER
*DEFINE_CPM_GAS_PROPERTIES
*DEFINE_CPM_VENT
*DEFINE_CURVE
*DEFINE_CURVE_FEEDBACK
*DEFINE_CURVE_FUNCTION
*DEFINE_CURVE_SMOOTH
*DEFINE_CURVE_TRIM
*DEFINE_CURVE_TRIM_3D
*DEFINE_FRICTION
*DEFINE_FUNCTION
*DEFINE_HEX_SPOTWELD_ASSEMBLY
*DEFINE_SD_ORIENTATION
*DEFINE_STOCHASTIC_VARIATION
*DEFINE_TABLE
*DEFINE_TABLE_2D
*DEFINE_TABLE_3D
*DEFINE_TRANSFORMATION
*DEFINE_VECTOR
*DEFINE_VECTOR_NODES

*DEFORMABLE_TO_RIGID
*DEFORMABLE_TO_RIGID_AUTOMATIC
*DEFORMABLE_TO_RIGID_INERTIA
*ELEMENT_BEAM
*ELEMENT_BEAM_ORIENTATION
*ELEMENT_BEAM_PID
*ELEMENT_BEAM_PID_ORIENTATION
*ELEMENT_BEAM_PID_SCALAR
*ELEMENT_BEAM_PID_SCALR
*ELEMENT_BEAM_SCALAR
*ELEMENT_BEAM_SCALAR_ORIENTATION
*ELEMENT_BEAM_SCALR
*ELEMENT_BEAM_SCALR_ORIENTATION
*ELEMENT_BEAM_SECTION
*ELEMENT_BEAM_SECTION_ORIENTATION
*ELEMENT_BEAM_SECTION_PID
*ELEMENT_BEAM_THICKNESS
*ELEMENT_BEAM_THICKNESS_ORIENTATION
*ELEMENT_BEAM_THICKNESS_PID
*ELEMENT_BEAM_THICKNESS_SCALAR
*ELEMENT_DISCRETE
*ELEMENT_DISCRETE_LCO
*ELEMENT_INERTIA
*ELEMENT_INERTIA_OFFSET
*ELEMENT_MASS
*ELEMENT_MASS_NODE_SET
*ELEMENT_MASS_PART
*ELEMENT_MASS_PART_SET
*ELEMENT_PLOTEL
*ELEMENT_SEATBELT
*ELEMENT_SEATBELT_ACCELEROMETER
*ELEMENT_SEATBELT_PRETENSIONER
*ELEMENT_SEATBELT_RETRACTOR
*ELEMENT_SEATBELT_SENSOR
*ELEMENT_SEATBELT_SLIPRING
*ELEMENT_SHELL
*ELEMENT_SHELL_BETA
*ELEMENT_SHELL_BETA_OFFSET
*ELEMENT_SHELL_BETA_COMPOSITE_LONG

*ELEMENT_SHELL_COMPOSITE
*ELEMENT_SHELL_COMPOSITE_LONG
*ELEMENT_SHELL_MCID
*ELEMENT_SHELL_MCID_COMPOSITE_LONG
*ELEMENT_SHELL_MICD_OFFSET
*ELEMENT_SHELL_OFFSET
*ELEMENT_SHELL_SHL4_TO_SHL8
*ELEMENT_SHELL_THICKNESS
*ELEMENT_SHELL_THICKNESS_BETA
*ELEMENT_SHELL_THICKNESS_BETA_OFFSET
*ELEMENT_SHELL_THICKNESS_MCID
*ELEMENT_SHELL_THICKNESS_MCID_OFFSET
*ELEMENT_SOLID
*ELEMENT_SOLID_ORTHO
*ELEMENT_SOLID_TET4TOTET10
*ELEMENT_SOLID_H20
*ELEMENT_SOLID_H8TOH20
*ELEMENT_SPH
*ELEMENT_TSHELL
*ELEMENT_TSHELL 6N
*ELEMENT_TSHELL 8N
*END
*EOS_GRUNEISEN
*EOS_IDEAL_GAS
*EOS_IGNITION_AND_GROWTH_OF_REACTION_IN_HE
*EOS_JWL
*EOS_LINEAR_POLYNOMIAL
*EOS_LINEAR_POLYNOMIAL_WITH_ENERGY_LEAK
*EOS_PROPELLANT_DEFLAGRATION
*EOS_RATIO_OF_POLYNOMIALS
*EOS_SACK_TUESDAY
*EOS_TABULATED
*EOS_TABULATED_COMPACTION
*EOS_TENSOR_PORE_COLLAPSE
*HOURGLASS
*INCLUDE
*INCLUDE_PATH
*INCLUDE_TRANSFORM
*INCLUDE_STAMPED_PART

*INCLUDE_STAMPED_PART_SET
*INCLUDE_COMPENSATION_BLANK_BEFORE_SPRINGBACK
*INCLUDE_COMPENSATION_BLANK_AFTER_SPRINGBACK
*INCLUDE_COMPENSATION_DESIRED_BLANK_SHAPE
*INCLUDE_COMPENSATION_COMPENSATED_SHAPE
*INCLUDE_COMPENSATION_CURRENT_TOOLS
*INITIAL_ALE_MAPPING
*INITIAL_AXIAL_FORCE_BEAM
*INITIAL_DETONATION
*INITIAL_FOAM_REFERENCE_GEOMETRY
*INITIAL_GAS_MIXTURE
*INITIAL_STRAIN_SHELL
*INITIAL_STRAIN_SOLID
*INITIAL_STRESS_BEAM
*INITIAL_STRESS_SECTION
*INITIAL_STRESS_SHELL
*INITIAL_STRESS_SOLID
*INITIAL_TEMPERATURE_NODE
*INITIAL_TEMPERATURE_SET
*INITIAL_VEHICLE_KINEMATICS
*INITIAL_VELOCITY
*INITIAL_VELOCITY_GENERATION
*INITIAL_VELOCITY_GENERATION_START_TIME
*INITIAL_VELOCITY_NODE
*INITIAL_VELOCITY_RIGID_BODY
*INITIAL_VOID_PART
*INITIAL_VOID_SET
*INITIAL_VOLUME_FRACTION
*INITIAL_VOLUME_FRACTION_GEOMETRY
*INTEGRATION_BEAM
*INTEGRATION_SHELL
*INTERFACE_COMPENSATION_NEW
*INTERFACE_COMPONENT_NODE
*INTERFACE_COMPONENT_SEGMENT
*INTERFACE_LINKING_DISCRETE_NODE_NODE
*INTERFACE_LINKING_DISCRETE_NODE_SET
*INTERFACE_LINKING_EDGE
*INTERFACE_LINKING_SEGMENT
*INTERFACE_SPRINGBACK_LSDYNA

*INTERFACE_SPRINGBACK_LSDYNA_NOTHICKNESS
*INTERFACE_SPRINGBACK_LSDYNA_THICKNESS
*INTERFACE_SPRINGBACK_NASTRAN
*INTERFACE_SPRINGBACK_NASTRAN_NOTHICKNESS
*INTERFACE_SPRINGBACK_NASTRAN_THICKNESS
*INTERFACE_SPRINGBACK_SEAMLESS
*INTERFACE_SPRINGBACK_SEAMLESS_NOTHICKNESS
*INTERFACE_SPRINGBACK_SEAMLESS_THICKNESS
*KEYWORD
*KEYWORD_ID
*LOAD_BEAM_ELEMENT
*LOAD_BEAM_SET
*LOAD_BLAST
*LOAD_BODY_GENERALIZED
*LOAD_BODY_GENERALIZED_SET_NODE
*LOAD_BODY_GENERALIZED_SET_PART
*LOAD_BODY_PARTS
*LOAD_BODY_RX
*LOAD_BODY_RY
*LOAD_BODY_RZ
*LOAD_BODY_VECTOR
*LOAD_BODY_X
*LOAD_BODY_Y
*LOAD_BODY_Z
*LOAD_BRODE
*LOAD_GRAVITY_PART
*LOAD_GRAVITY_PART_SET
*LOAD_MASK
*LOAD_NODE_POINT
*LOAD_NODE_SET
*LOAD_RIGID_BODY
*LOAD_SEGMENT
*LOAD_SEGMENT_SET
*LOAD_SHELL_ELEMENT
*LOAD_SHELL_SET
*LOAD_SUPERPLASTIC_FORMING
*LOAD_THERMAL_CONSTANT
*LOAD_THERMAL_CONSTANT_NODE
*LOAD_THERMAL_LOAD_CURVE

*LOAD_THERMAL_VARIABLE
*LOAD_THERMAL_VARIABLE_NODE
*NODE
*NODE_RIGID_SURFACE
*NODE_TRANSFORM
*PARAMETER
*PARAMETER_LOCAL
*PARAMETER_EXPRESSION
*PARAMETER_EXPRESSION_LOCAL
*PART
*PART_COMPOSITE
*PART_COMPOSITE_CONTACT
*PART_COMPOSITE_LONG
*PART_COMPOSITE_TSHELL
*PART_CONTACT
*PART_CONTACT_PRINT
*PART_INERTIA
*PART_INERTIA_PRINT
*PART_INERTIA_CONTACT
*PART_INERTIA_CONTACT_PRINT
*PART_MOVE
*PART_PRINT
*PART_REPOSITION
*PART_REPOSITION_PRINT
*PART_REPOSITION_CONTACT
*PART_REPOSITION_CONTACT_PRINT
*PART_SENSOR
*RIGIDWALL_GEOMETRIC_CYLINDER_DISPLAY
*RIGIDWALL_GEOMETRIC_CYLINDER
*RIGIDWALL_GEOMETRIC_CYLINDER_MOTION_DISPLAY
*RIGIDWALL_GEOMETRIC_CYLINDER_MOTION
*RIGIDWALL_GEOMETRIC_FLAT(FINITE)
*RIGIDWALL_GEOMETRIC_FLAT_DISPLAY(FINITE)
*RIGIDWALL_GEOMETRIC_FLAT_DISPLAY
*RIGIDWALL_GEOMETRIC_FLAT
*RIGIDWALL_GEOMETRIC_FLAT_MOTION(FINITE)
*RIGIDWALL_GEOMETRIC_FLAT_MOTION_DISPLAY(FINITE)
*RIGIDWALL_GEOMETRIC_FLAT_MOTION_DISPLAY
*RIGIDWALL_GEOMETRIC_FLAT_MOTION

*RIGIDWALL_GEOMETRIC_PRISM(FINITE)
*RIGIDWALL_GEOMETRIC_PRISM_DISPLAY(FINITE)
*RIGIDWALL_GEOMETRIC_PRISM_DISPLAY
*RIGIDWALL_GEOMETRIC_PRISM
*RIGIDWALL_GEOMETRIC_PRISM_MOTION(FINITE)
*RIGIDWALL_GEOMETRIC_PRISM_MOTION_DISPLAY(FINITE)
*RIGIDWALL_GEOMETRIC_PRISM_MOTION_DISPLAY
*RIGIDWALL_GEOMETRIC_PRISM_MOTION
*RIGIDWALL_GEOMETRIC_SPHERE_DISPLAY
*RIGIDWALL_GEOMETRIC_SPHERE
*RIGIDWALL_GEOMETRIC_SPHERE_MOTION_DISPLAY
*RIGIDWALL_GEOMETRIC_SPHERE_MOTION
*RIGIDWALL_PLANAR_FINITE_FORCES
*RIGIDWALL_PLANAR_FINITE
*RIGIDWALL_PLANAR_FINITE_MOVING_DISPLAY
*RIGIDWALL_PLANAR_FINITE_MOVING_FORCES_DISPLAY
*RIGIDWALL_PLANAR_FINITE_MOVING_FORCES
*RIGIDWALL_PLANAR_FINITE_MOVING
*RIGIDWALL_PLANAR_FORCES
*RIGIDWALL_PLANAR
*RIGIDWALL_PLANAR_MOVING_DISPLAY
*RIGIDWALL_PLANAR_MOVING_FORCES_DISPLAY
*RIGIDWALL_PLANAR_MOVING_FORCES
*RIGIDWALL_PLANAR_MOVING
*RIGIDWALL_PLANAR_ORTHO_FINITE_FORCES
*RIGIDWALL_PLANAR_ORTHO_FINITE
*RIGIDWALL_PLANAR_ORTHO_FORCES
*RIGIDWALL_PLANAR_ORTHO
*SECTION_ALE1D
*SECTION_ALE2D
*SECTION_BEAM
*SECTION_BEAM_AISC
*SECTION_DISCRETE
*SECTION_POINT_SOURCE
*SECTION_POINT_SOURCE_MIXTURE
*SECTION_SEATBELT
*SECTION_SHELL
*SECTION_SHELL_EFG
*SECTION_SHELL_THERMAL

*SECTION_SHELL_XFEM
*SECTION_SOLID
*SECTION_SOLID_EFG
*SECTION_SPH
*SECTION_SPH_USER
*SECTION_TSHELL
*SENSOR_CONTROL
*SENSOR_DEFINE_CALC-MATH
*SENSOR_DEFINE_ELEMENT
*SENSOR_DEFINE_ELEMENT_SET
*SENSOR_DEFINE_FORCE
*SENSOR_DEFINE_NODE
*SENSOR_DEFINE_NODE_SET
*SENSOR_SWITCH
*SENSOR_SWITCH_CALC-LOGIC
*SET_BEAM
*SET_BEAM_ADD
*SET_BEAM_COLLECT
*SET_BEAM_COLLECT(MASTER)
*SET_BEAM_GENERAL
*SET_BEAM_GENERAL_COLLECT
*SET_BEAM_GENERATE
*SET_BEAM_GENERATE_COLLECT
*SET_DISCRETE
*SET_DISCRETE_ADD
*SET_DISCRETE_GENERAL
*SET_DISCRETE_GENERATE
*SET_MULTI-MATERIAL_GROUP_LIST
*SET_NODE_ADD
*SET_NODE_ADD_ADVANCED
*SET_NODE_COLLECT(MASTER)
*SET_NODE_COLUMN
*SET_NODE_COLUMN_COLLECT
*SET_NODE_GENERAL
*SET_NODE_GENERAL_COLLECT
*SET_NODE_LIST
*SET_NODE_LIST_COLLECT
*SET_NODE_LIST_GENERATE
*SET_NODE_LIST_GENERATE_COLLECT

*SET_PART_ADD
*SET_PART_COLUMN
*SET_PART_COLUMN_COLLECT
*SET_PART_LIST
*SET_PART_LIST_COLLECT
*SET_PART_LIST_GENERATE
*SET_PART_LIST_GENERATE_COLLECT
*SET_PART_COLLECT(MASTER)
*SET_SEGMENT
*SET_SEGMENT_GENERAL
*SET_SHELL_LIST
*SET_SHELL_LIST_COLLECT
*SET_SHELL_ADD
*SET_SHELL_COLUMN
*SET_SHELL_GENERAL
*SET_SHELL_LIST_GENERATE
*SET_SOLID
*SET_SOLID_ADD
*SET_SOLID_GENERAL
*SET_SOLID_GENERATE
*SET_TSHELL
*SET_TSHELL_GENERAL
*SET_TSHELL_GENERATE
*SET_TSHELL_ADD
*STRESS_INITIALIZATION
*STRESS_INITIALIZATION_DISCRETE
*STRESS_INITIALIZATION_SEATBELT
*STRESS_INITIALIZATION_SEATBELT
*TERMINATION_BODY
*TERMINATION_CONTACT
*TERMINATION_CURVE
*TERMINATION_DELETED_SHELLS
*TERMINATION_DELETED_SHELLS_SET
*TERMINATION_DELETED_SOLIDS
*TERMINATION_DELETED_SOLIDS_SET
*TERMINATION_NODE
*TERMINATION_SENSOR
*TITLE
*MAT_ADD_AIRBAG_POROSITY_LEAKAGE

*MAT_ADD_COHESIVE
*MAT_ADD_EROSION
*MAT_ADD_FATIGUE
*MAT_ADD_GENERALIZED_DAMAGE
*MAT_ADD_PERMEABILITY
*MAT_ADD_PORE_AIR
*MAT_ADD_THERMAL_EXPANSION
001 - *MAT_ELASTIC
001 - *MAT_ELASTIC_FLUID
002_ANIS - *MAT_ANISOTROPIC_ELASTIC (*MAT_002_ANIS)
002 - *MAT_ORTHOTROPIC_ELASTIC (*MAT_002)
003 - *MAT_PLASTIC_KINEMATIC
004 - *MAT_ELASTIC_PLASTIC_THERMAL
005 - *MAT_SOIL_AND_FOAM
006 - *MAT_VISCOELASTIC
007 - *MAT_BLATZ-KO_RUBBER
008 - *MAT_HIGH_EXPLOSIVE_BURN
009 - *MAT_NULL
010 - *MAT_ELASTIC_PLASTIC_HYDRO
010 - *MAT_ELASTIC_PLASTIC_HYDRO_SPALL
011 - *MAT_STEINBERG
011 - *MAT_STEINBERG_LUND
012 - *MAT_ISOTROPIC_ELASTIC_PLASTIC
013 - *MAT_ISOTROPIC_ELASTIC_FAILURE
014 - *MAT_SOIL_AND_FOAM_FAILURE
015 - *MAT_JOHNSON_COOK
015 - *MAT_JOHNSON_COOK_STOCHASTIC
016 - *MAT_PSEUDO_TENSOR
017 - *MAT_ORIENTED_CRACK
018 - *MAT_POWER_LAW_PLASTICITY
019 - *MAT_STRAIN_RATE_DEPENDENT_PLASTICITY
020 - *MAT_RIGID
021 - *MAT_ORTHOTROPIC_THERMAL
021 - *MAT_ORTHOTROPIC_THERMAL_CURING
021 - *MAT_ORTHOTROPIC_THERMAL_FAILURE
022 - *MAT_COMPOSITE_DAMAGE
023 - *MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC
024 - *MAT_PIECEWISE_LINEAR_PLASTICITY
024 - *MAT_PIECEWISE_LINEAR_PLASTICITY_HAZ

024 - *MAT_PIECEWISE_LINEAR_PLASTICITY_LOG_INTERPOLATION

024 - *MAT_PIECEWISE_LINEAR_PLASTICITY_MIDFAIL

024 - *MAT_PIECEWISE_LINEAR_PLASTICITY_STOCHASTIC

025 - *MAT_GEOLOGIC_CAP_MODEL

026 - *MAT_HONEYCOMB

027 - *MAT_MOONEY-RIVLIN_RUBBER

028 - *MAT_RESULTANT_PLASTICITY

029 - *MAT_FORCE_LIMITED

030 - *MAT_SHAPE_MEMORY

031 - *MAT_FRAZER_NASH_RUBBER_MODEL

032 - *MAT_LAMINATED_GLASS

033 - *MAT_BARLAT_ANISOTROPIC_PLASTICITY

033_96 - *MAT_BARLAT_YLD96

034 - *MAT_FABRIC

034M - *MAT_FABRIC_MAP

036 - *MAT_3-PARAMETER_BARLAT

036 - *MAT_3-PARAMETER_BARLAT_NLP

037 - *MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC

037 - *MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC_ECHANGE

037 - *MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC_NLP_FAILURE

038 - *MAT_BLATZ-KO_FOAM

039 - *MAT_FLD_TRANSVERSELY_ANISOTROPIC

040 - *MAT_NONLINEAR_ORTHOTROPIC

041-050- *MAT_USER_DEFINED_MATERIAL_MODELS

051 - *MAT_BAMMAN

052 - *MAT_BAMMAN_DAMAGE

053 - *MAT_CLOSED_CELL_FOAM

054-055 - *MAT_ENHANCED_COMPOSITE_DAMAGE

057 - *MAT_LOW_DENSITY_FOAM

058 - *MAT_LAMINATED_COMPOSITE_FABRIC

059 - *MAT_COMPOSITE_FAILURE_SHELL_MODEL

059 - *MAT_COMPOSITE_FAILURE_SOLID_MODEL

059 - *MAT_COMPOSITE_FAILURE_SPH_MODEL

060 - *MAT_ELASTIC_WITH_VISCOCITY

060C- *MAT_ELASTIC_WITH_VISCOSITY_CURVE

061 - *MAT_KELVIN-MAXWELL_VISCOELASTIC

062 - *MAT_VISCOUS_FOAM

063 - *MAT_CRUSHABLE_FOAM

064 - *MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY

065 - *MAT_MODIFIED_ZERILLI_ARMSTRONG
066 - *MAT_LINEAR_ELASTIC_DISCRETE_BEAM
067 - *MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM
068 - *MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM
069 - *MAT_SID_DAMPER_DISCRETE_BEAM
070 - *MAT_HYDRAULIC_GAS_DAMPER_DISCRETE
071 - *MAT_CABLE_DISCRETE_BEAM
072 - *MAT_CONCRETE_DAMAGE
072R3 - *MAT_CONCRETE_DAMAGE_REL3
073 - *MAT_LOW_DENSITY_VISCOUS_FOAM
074 - *MAT_ELASTIC_SPRING_DISCRETE_BEAM
075 - *MAT_BILKU-DUBOIS_FOAM
076 - *MAT_GENERAL_VISCOELASTIC
077_H - *MAT_HYPERELASTIC_RUBBER
077_O - *MAT_OGDEN_RUBBER
078 - *MAT_SOIL_CONCRETE
079 - *MAT_HYSTERETIC_SOIL
081,082 - *MAT_PLASTICITY_WITH_DAMAGE
081,082 - *MAT_PLASTICITY_WITH_DAMAGE_ORTHO
081,082 - *MAT_PLASTICITY_WITH_DAMAGE_ORTHO_RCDC
081,082 - *MAT_PLASTICITY_WITH_DAMAGE_STOCHASTIC
083 - *MAT_FU_CHANG_FOAM
083 - *MAT_FU_CHANG_FOAM_DAMAGE_DECAY
083 - *MAT_FU_CHANG_FOAM_LOG_LOG_INTERPOLATION
084_085 - *MAT_WINFRITH_CONCRETE
086 - *MAT_ORTHOTROPIC_VISCOELASTIC
087 - *MAT_CELLULAR_RUBBER
088 - *MAT_MTS
089 - *MAT_PLASTICITY_POLYMER
090- *MAT_ACOUSTIC
093 - *MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM
094 - *MAT_INELASTIC_SPRING_DISCRETE_BEAM
095 - *MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAM
096 - *MAT_BRITTLE_DAMAGE
097 - *MAT_GENERAL_JOINT_DISCRETE_BEAM
098 - *MAT_SIMPLIFIED_JOHNSON_COOK
098 - *MAT_SIMPLIFIED_JOHNSON_COOK_STOCHASTIC
099 - *MAT_SIMPLIFIED_JOHNSON_COOK_ORTHOTROPIC_DAMAGE
100 - *MAT_SPOTWELD

100 - *MAT_SPOTWELD_DAMAGE-FAILURE
100_DA - *MAT_SPOTWELD_DAIMLERCHRYSLER
101 - *MAT_GEPLASTIC_SRATE_2000a
103 - *MAT_ANISOTROPIC_VISCOPLASTIC
103P - *MAT_ANISOTROPIC_PLASTIC
104 - *MAT_DAMAGE_1
105 - *MAT_DAMAGE_2
106 - *MAT_ELASTIC_VISCOPLASTIC_THERMAL
110 - *MAT_JOHNSON_HOLMQUIST_CERAMICS
111 - *MAT_JOHNSON_HOLMQUIST_CONCRETE
112 - *MAT_FINITE_ELASTIC_STRAIN_PLASTICITY
113 - *MAT_TRIP
114 - *MAT_LAYERED_LINEAR_PLASTICITY
115 - *MAT_UNIFIED_CREEP
115_O - *MAT_UNIFIED_CREEP_ORTHO
116 - *MAT_COMPOSITE_LAYUP
117 - *MAT_COMPOSITE_MATRIX
119 - *MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM
120 - *MAT_GURSON
120B - *MAT_GURSON_JC
120C - *MAT_GURSON_RCDC
121 - *MAT_GENERAL_NONLINEAR_1DOF_DISCRETE_BEAM
122 - *MAT_HILL_3R
122_3D - *MAT_HILL_3R_3D
123 - *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY
123 - *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY_RATE
123 - *MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY_RTCL
124 - *MAT_PLASTICITY_COMPRESSION_TENSION
125 - *MAT_KINEMATIC_HARDENING_TRANSVERSELY_ANISOTROPIC
125 - *MAT_KINEMATIC_HARDENING_TRANSVERSELY_ANISOTROPIC_NLP
126 - *MAT_MODIFIED_HONEYCOMB
127 - *MAT_ARRUDA_BOYCE_RUBBER
130 - *MAT_SPECIAL_ORTHOTROPIC
133 - *MAT_BARLAT_YLD2000
135 - *MAT_WTM_STM
135 - *MAT_WTM_STM_PLC
136 - *MAT_CORUS_VEGTER
138 - *MAT_COHESIVE_MIXED_MODE
140 - *MAT_VACUUM

141 - *MAT_RATE_SENSITIVE_POLYMER

142 - *MAT_TRANSVERSELY_ANISOTROPIC_CRUSHABLE_FOAM

143 - *MAT_WOOD

143 - *MAT_WOOD_FIR

143 - *MAT_WOOD_PINE

145 - *MAT_SCHWER_MURRAY_CAP_MODEL

146 - *MAT_1DOF_GENERALIZED_SPRING

148 - *MAT_GAS_MIXTURE

151 - *MAT_EMMI

154 - *MAT_DESHPANDE_FLECK_FOAM

155 - *MAT_PLASTICITY_COMPRESSION_TENSION_EOS

157 - *MAT_ANISOTROPIC_ELASTIC_PLASTIC

159 - *MAT_CSCM

159 - *MAT_CSCM_CONCRETE

160 - *MAT_ALE_INCOMPRESSIBLE

161 - *MAT_COMPOSITE_MSC

162 - *MAT_COMPOSITE_MSC_DMG

163 - *MAT_MODIFIED_CRUSHABLE_FOAM

165 - *MAT_PLASTIC_NONLINEAR_KINEMATIC

165B - *MAT_PLASTIC_NONLINEAR_KINEMATIC_B

166 - *MAT_MOMENT_CURVATURE_BEAM

168 - *MAT_POLYMER

169 - *MAT_ARUP_ADHESIVE

170 - *MAT_RESULTANT_ANISOTROPIC

172 - *MAT_CONCRETE_EC2

177 - *MAT_HILL_FOAM

178 - *MAT_VISCOELASTIC_HILL_FOAM

179 - *MAT_LOW_DENSITY_SYNTHETIC_FOAM

179 - *MAT_LOW_DENSITY_SYNTHETIC_FOAM_WITH_FAILURE

180 - *MAT_LOW_DENSITY_SYNTHETIC_FOAM_ORTHO

180 - *MAT_LOW_DENSITY_SYNTHETIC_FOAM_ORTHO_WITH_FAILURE

181 - *MAT_SIMPLIFIED_RUBBER_FOAM

181 - *MAT_SIMPLIFIED_RUBBER_FOAM_LOG_LOG_INTERPOLATION

181 - *MAT_SIMPLIFIED_RUBBER_FOAM_WITH_FAILURE

183 - *MAT_SIMPLIFIED_RUBBER_WITH_DAMAGE

183 - *MAT_SIMPLIFIED_RUBBER_WITH_DAMAGE_LOG_LOG_INTERPOLATION

184 - *MAT_COHESIVE_ELASTIC

185 - *MAT_COHESIVE_TH

186 - *MAT_COHESIVE_GENERAL

187 - *MAT_SAMP-1
188 - *MAT_THERMO_ELASTO_VISCOPLASTIC_CREEP
189 - *MAT_ANISOTROPIC_THERMOELASTIC
190 - *MAT_FLD_3-PARAMETER_BARLAT
191 - *MAT_SEISMIC_BEAM
196 - *MAT_GENERAL_SPRING_DISCRETE_BEAM
202 - *MAT_STEEL_EC3
208 - *MAT_BOLT_BEAM
214 - *MAT_DRY_FABRIC
219 - *MAT_CODAM2
220 - *MAT_RIGID_DISCRETE
221 - *MAT_ORTHOTROPIC_SIMPLIFIED_DAMAGE
224 - *MAT_TABULATED_JOHNSON_COOK
224_GYS - *MAT_TABULATED_JOHNSON_COOK_GYS
225 - *MAT_VISCOPLASTIC_MIXED_HARDENING
226 - *MAT_KINEMATIC_HARDENING_BARLAT89
230 - *MAT_PML_ELASTIC
230 - *MAT_PML_ELASTIC_FLUID
234 - *MAT_VISCOELASTIC_LOOSE_FABRIC
235 - *MAT_MICROMECHANICS_DRY_FABRIC
240 - *MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE
241 - *MAT_JOHNSON_HOLMQUIST_JH1
242 - *MAT_KINEMATIC_HARDENING_BARLAT2000
243 - *MAT_HILL_90
244 - *MAT_UHS_STEEL
248 - *MAT_PHS_BMW
249 - *MAT_REINFORCED_THERMOPLASTIC
251 - *MAT_TAILORED_PROPERTIES
252 - *MAT_TOUGHENED_ADHESIVE_POLYMER
254 - *MAT_GENERALIZED_PHASE_CHANGE
255 - *MAT_PIECEWISE_LINEAR_PLASTIC_THERMAL
260A - *MAT_STOUGHTON_NON_ASSOCIATED_FLOW
260A - *MAT_STOUGHTON_NON_ASSOCIATED_FLOW_XUE
260B - *MAT_MOHR_NON_ASSOCIATED_FLOW
260B - *MAT_MOHR_NON_ASSOCIATED_FLOW_XUE
261 - *MAT_LAMINATED_FRACTURE_DAIMLER_PINHO
262 - *MAT_LAMINATED_FRACTURE_DAIMLER_CAMANHO
270 - *MAT_CWM
271 - *MAT_POWDER

274 - *MAT_PAPER
275 - *MAT_SMOOTH_VISCOELASTIC_VISCOPLASTIC
277 - *MAT_ADHESIVE_CURING_VISCOELASTIC
279 - *MAT_COHESIVE_PAPER
280 - *MAT_GLASS
B01 - *MAT_SEATBELT
S1 - *MAT_SPRING_ELASTIC
S2 - *MAT_DAMPER_VISCOUS
S3 - *MAT_SPRING_ELASTOPLASTIC
S4 - *MAT_SPRING_NONLINEAR_ELASTIC
S5 - *MAT_DAMPER_NONLINEAR_VISCOUS
S6 - *MAT_SPRING_GENERAL_NONLINEAR
S7 - *MAT_SPRING_MAXWELL
S8 - *MAT_SPRING_INELASTIC
T1 - *MAT_THERMAL_ISOTROPIC
T10- *MAT_THERMAL_ISOTROPIC_TD_LC
T2 - *MAT_THERMAL_ORTHOTROPIC
U1 - *MAT_UNSUPPORTED

MADYMO

ACTUATOR
AIRBAG_CHAMBER
AMPLIFICATION.ABS_POLY
AMPLIFICATION.EXP
AMPLIFICATION.LOG
AMPLIFICATION.POLY
ANIMATION
BELT
BELT_FUSE
BELT_RETRACTOR
BELT_SEGMENT
BELT_TYING
BODY.DEFORMABLE
BODY.FLEXIBLE_BEAM
BODY.RIGID
CHAR_MOD
CHARACTERISTIC.CONTACT
CHARACTERISTIC.LOAD
CHARACTERISTIC.MATERIAL

HOLE
HOLE_AREA
HOLE_SUBSEGMENT
INFLATOR
INFLATOR.CHAR
INFLATOR.DEF
INFLATOR.REF
INJURY
JET
JOINT
LAYER
LOAD
MADYMO
MATERIAL.ANISO
MATERIAL.FABRIC
MATERIAL.FOAM
MATERIAL.HOLE
MATERIAL.HONEYCOMB
MATERIAL.HONEYCOMB_PLASTIC
MATERIAL.HYSISO
MATERIAL.INTERFACE
MATERIAL.ISOLIN
MATERIAL ISOLIN
MATERIAL ISOPLA
MATERIAL.KELVIN1D
MATERIAL.KELVIN1D_NL
MATERIAL.KELVIN3D
MATERIAL KELVIN3D_NL
MATERIAL LINVIS
MATERIAL MOONRIV
MATERIAL NULL
MATERIAL ORTHOLIN
MATERIAL ORTHOLIN_LAYERED
MATERIAL ORTHOPLA
MATERIAL RIGID
MATERIAL SANDWICH
MATERIAL STRAP
MATERIAL TONER
MATERIAL USER

MATERIAL.VISCO_NL

MODE

MODE_SHAPE

MOTION.NODE

MOTION_STRUCT_FE

OPERATOR

ORIENTATION.MATRIX

ORIENTATION.SCREW_AXIS

ORIENTATION.SUCCESSIVE_ROT

ORIENTATION.VECTOR

OUTPUT_AIRBAG_CHAMBER

OUTPUT_ANIMATION

OUTPUT_BELT

OUTPUT_BODY

OUTPUT_BODY_REL

OUTPUT_CONTACT

OUTPUT_CONTROL_SYSTEM

OUTPUT_CROSS_SECTION

OUTPUT_ELEMENT

OUTPUT_ELEMENT_INITIAL

OUTPUT_ENERGY_FE_MODEL

OUTPUT_JET

OUTPUT_JOINT_CONSTRAINT

OUTPUT_JOINT_DOF

OUTPUT_MARKER

OUTPUT_MOTION_STRUCT

OUTPUT_NODE

OUTPUT_NODE_INITIAL

OUTPUT_RESTRAINT

OUTPUT_SENSOR

OUTPUT_SYSTEM_COG

PART

PERMEABILITY

POINT_OBJECT

POINT_OBJECT_1_MB

POINT_OBJECT_2_MB

POINT_OBJECT_FE

POINT_OBJECT_1_FE

POINT_OBJECT_2_FE

SCALING
SELECT
SENSOR
SIGNAL
SPOTWELD.NODE_NODE
SPOTWELD.THREE_NODE
STATE
STRAP
SUPPORT
SURFACE.CYLINDER
SURFACE.PLANE
SWITCH
SYSTEM.MODEL
SYSTEM.REF_SPACE
TIME_HISTORY_CONTACT
TIME_HISTORY_ENERGY
TIME_HISTORY_FE
TIME_HISTORY_MB
TIME_HISTORY_SYSTEM
USER_INT

Marc

ASSUMED (ASSUMED STRAIN)
AUTO INCREMENT
Body 3D Deformable
Body 3D Rigid
CBUSH
Constant Dilatation
Contact Header
Contact Table
CONTACT_TYPE
Disp_chang
DIST_LOADS
DIST LOADS (CONTROL CARD)
E_1
E_2
E_3
E_5
E_6

E_80
E_81
E_82
E_83
E_84
E_89
E_95
E_96
E_98
E_114
E_115
E_116
E_117
E_118
E_119
E_120
E_124
E_125
E_126
E_127
E_128
E_129
E_130
E_134
E_138
E_139
E_140
E_157
E-195
ELSTO
FEATURE
Finite
Fixed_Acce
Fixed_Disp
Fixed_Pres
FOLLOW FOR
FOUNDATION
Init_Disp
INITIAL_vel

Nastran

AEFORCE

AELIST

AEPARM

AERO

AEROS

AESTAT

AESURF

ASCRCE

ASET

ASET1

AUTOSPC

B2GG

B2PP

BCBODY

BCBODY1

BCDPRP

BCONECT

BCONPRG

BCONPRP

BCPARA

BCPROP

BCTABLE

BCTABL1

BCTSET

BEGIN BULK

BMFACE

BNDFREE1

BNDFX1

BOLT

BOLTFOR

BSET1

BSURF

BSURFS

BULK-UNSUPPORTED_CARD

CAABSF

CACINF3

CACINF4

CAERO1

CAERO2

CASE-UNSUPPORTED CARD

CBAR

CBEAM

CBEND

CBUSH

CBUSH1D

CDAMP1

CDAMP2

CDAMP3

CDAMP4

CELAS1

CELAS2

CELAS3

CELAS4

CEND

CFAST

CGAP

CHACAB

CHBDYE

CHBDYP

CHEXA (20-noded)

CHEXA (8-noded)

CMASS1

CMASS2

CMASS3

CMASS4

CONM1

CONM2

CONROD

CONVECTION

CORD1C

CORD1R

CORD1S

CORD2C

CORD2R

CORD2S

CPENTA (6-noded)

CPENTA (15-noded)

CQUAD4

CQUAD8

CQUADR

CQUADX

CROD

CSET1

CSHEAR

CSUPER

CSUPEXT

CTETRA (4-noded)

CTETRA (10-noded)

CTRIA3

CTRIA6

CTRIAX

CTRIAX6

CTRIAR

CTUBE

CVISC

CWELD

DAMPING

DAREA

DCONADD

DCONSTR

DDVAL

DEFORM

DELAY

DEQATN

DESGLB

DESOBJ

DESSUB

DESVAR

DIAG

DLINK

DLINK2

DLOAD

DOPTPRM

DPHASE

DRESP1

DRESP2

DSCREEN

HM_SPRING
HYBDAMP
ID
INCLUDE BULK
INCLUDE_CTRL
INCLUDE_EXEC
K2GG
K2PP
LOAD
LOADSEQ
M2GG
MARCOUT
MAT1
MAT2
MAT4
MAT5
MAT8
MAT9
MAT10
MATEP
MATG
MATHE
MATHP
MATS1
MATT1
MATT4
MAXLINES
MAXMIN
MBOLT
MBOLTUS
MFLUID
MINMAX
MKAERO1
MOMENT
MONDSP1
MONPNT
MONPNT1
MONPNT2
MONPNT3

PCONVM
PDAMP
PELAS
PELAST
PFAST
PGAP
PHBDY
PLPLANE
PLOAD
PLOAD1
PLOAD2
PLOAD4
PLOTEL
PLSOLID
PMASS
PROD
PSEAM
PSHEAR
PSHELL
PSHLN1
PSOLID
PRESSURE
PTUBE
PVISC
PWELD
QBDY1
QSET1
QVOL
RADCAV
RADIATION
RADM
RADSET
RANDPS
RBAR
RBE2
RBE3
RESTART
RESVEC
RFORCE

OptiStruct

ACSRCE

ANALYSIS

ASET

ASET1

ASSIGN

AUTOSPC

B2GG

BMFACE

BORE

CAABSF

CAALOAD

CBAR

CBEAM

CBUSH

CBUSH1D

CDAMP1

CDAMP2

CDAMP3

CDAMP4

CDSMETH

CELAS1

CELAS2

CELAS3

CELAS4

CFAST

CGAP

CGAPG

CGASK6

CGASK8

CGASK12

CGASK16

CHACAB

CHBDYE

CHECK

CHEXA (8-noded)

CHEXA (20-noded)

CMASS1

CMASS2

CMASS3

CMASS4

CMBEAM

CMBUSH

CMBUSHC

CMBUSHE

CMBUSHT

CMSMETH

CMSPDP

CMSPDPC

CMSPDPE

CMSPDPT

CNTSTB

CONM1

CONM2

CONROD

CONTACT

CONTF

CONTPRM

CONV

CORD1C

CORD1R

CORD1S

CORD2C

CORD2R

CORD2S

CORD3R

CORD4R

CPENTA (6-noded)

CPENTA (15-noded)

CPYRA (5-noded)

CPYRA (13-noded)

CQUADR

CQUAD4

CQUAD8

CROD

CSHEAR

CSTRAIN

CSTRESS

CTAXI

CTETRA (4-noded)
CTETRA (10-noded)
CTRIAR
CTRIAX6
CTRIA3
CTRIA6
CTUBE
CVISC
CWELD
DAMAGE
DAREA
DCOMP
DCONADD
DCONSTR
DDVAL
DEBUG
DEFORM
DENSITY
DENSRES
DEQATN
DESGLB
DESHIS
DESOBJ
DESSUB
DESVAR
DESVARG
DGLOBAL
DISPLACEMENT
DLINK
DLINK2
DLOAD
DMIGNAME
DOBJREF
DOPTPRM
DPENTA6
DQUAD4
DRESP1
DRESP2
DRESP3

DSCREEN
DSHAPE
DSHUFFLE
DSIZE
DSYSID
DTABLE
DTETRA4
DTI_SPECSEL
DTI_UNITS
DTPG
DTPL
DTRIA3
DVCREL1
DVCREL2
DVGRID
DVMREL1
DVMREL2
DVPREL1
DVPREL2
ECHO
EIGC
EIGRA
EIGRL
EIGVNAME
EIGVRETRIEVE
EIGVSAVE
ELFORCE
ELIST
ENERGY
ERP
ERPPNL
ESE
EXCLUDE
FATDEF
FATEVNT
FATLOAD
FATPARM
FATSEQ
FLLWER

FLUX

FORCE

FORCE1

FORMAT

FREQ

FREQ1

FREQ2

FREQ3

FREQ4

FREQ5

FREQUENCY

FSI

GAPPRM

GLOBAL_CASE_CONTROL

GLOBSUB

GPFORCE

GPSTRAIN

GPSTRESS

GRAV

GRDSET

GRID

GROUND

HISOUT

HM_ELAS

HMSPRING

HYBDAMP

IC

INCLUDE_BULK

INCLUDE_CTRL

INFILE

INVEL

INVELB

JOINT

K2GG

K2PP

K42GG

LABEL

LIFE

LOAD

- MBCNTDS
- MBCNTR
- MBCRV
- MBDCRV
- MBDEACT
- MBDSRF
- MBFORCE
- MBFRC
- MBFRCC
- MBLIN
- MBMNT
- MBMNTC
- MBPCRV
- MBREQ
- MBREQE
- MBREQM
- MBSEQ
- MBSIM
- MBSIMP
- MBVAR
- METHOD
- MGASK
- MFLUID
- MINMAX
- MLOAD
- MODEL
- MODEWEIGHT
- MOMENT
- MOMENT1
- MOTION
- MOTNG
- MOTNGC
- MPC
- MPCADD
- MPCFORCE
- MPCFORCES
- MSGLMT
- NESLEXPD
- NESLIMPD

PEAKOUT (Load Collector)

PELAS

PFAST

PFAT

PFBODY

PFGRID

PFMODE

PFPANEL

PFPATH

PGAP

PGASK

PLOAD

PLOAD1

PLOAD2

PLOAD4

PLOTEL

PLOTEL3

PLOTEL4

PLY

PMASS

PRBODY

PRESSURE

PROD

PROPERTY

PSHEAR

PSHELL

PSOLID

PTADD

PTUBE

PVISC

PWELD

QBDY1

QVOL

RANDPS

RBAR

RBE2

RBE3

REQUEST

RESPRINT

RESTART

RESULTS

RESVEC

RFORCE

RGYRO

RLOAD1

RLOAD2

ROTORG

RROD

RSPEC

RSPEC (Case Control)

RSPEED

RSPINR

RWALL (Group)

RWALL (Case Control)

RWALLADD

SCREEN

SDAMPING

SECT

SENSITIVITY

SENSOUT

SET

SET3

SHAPE

SHAPEOPT

SHRES

SOLVTYP

SPC

SPCADD

SPCD

SPCFORCES

SPOINT

STACK

STATSUB

STRAIN

STRESS

SUBCASE

SUBCOM

SUBMODEL

PAM-CRASH 2G

FRAME/ IAXIS = 2 (rectangular)

FRAME/ IAXIS = 3 (rectangular)

FRAME/ IAXIS = 4

FRAME/ IAXIS = 5

FRICT/

FRICT / IFROPT = 1

FRICT / IFROPT = 2

FRICT / IFROPT = 3

FRICT / IFROPT = 4

FRICT / IFROPT = 5

FRICT / IFROPT = 6

FRICT / IFROPT = 10

FRICT / IFROPT = 11

FRICT / IFROPT = 12

FUNCT/

FUNCSW/

GASPEC/

GROUP/

HEAT/

ICTRL/

IMPFIL/

IMPORT/

INCLU/

INFLATOR

INPUTVERSION

INTEM/

INVEL/

JOINT/

KJOIN/

KJOIN / JTYP = SPHERICA

KJOIN / JTYP = TRANSLAT

KJOIN / JTYP = REVOLUTE

KJOIN / JTYP = CYLINDRI

KJOIN / JTYP = PLANAR

KJOIN / JTYP = UNIVERSA

KJOIN / JTYP = GENERAL

KJOIN / JTYP = FLEX-TOR

LAYER/

LCTRL/

MATER / MATYP = 99
MATER / MATYP = 100
MATER / MATYP = 101
MATER / MATYP = 103
MATER / MATYP = 105
MATER / MATYP = 106
MATER / MATYP = 107
MATER / MATYP = 108
MATER / MATYP = 109
MATER / MATYP = 110
MATER / MATYP = 115
MATER / MATYP = 116
MATER / MATYP = 117
MATER / MATYP = 118
MATER / MATYP = 121
MATER / MATYP = 126
MATER / MATYP = 127
MATER / MATYP = 128
MATER / MATYP = 131
MATER / MATYP = 143
MATER / MATYP = 150
MATER / MATYP = 151
MATER / MATYP = 161
MATER / MATYP = 162
MATER / MATYP = 171
MATER / MATYP = 180
MATER / MATYP = 181
MATER / MATYP = 182
MATER / MATYP = 183
MATER / MATYP = 184
MATER / MATYP = 185
MATER / MATYP = 200
MATER / MATYP = 201
MATER / MATYP = 202
MATER / MATYP = 204
MATER / MATYP = 205
MATER / MATYP = 212
MATER / MATYP = 213
MATER / MATYP = 214

PART / ATYPE = BSHEL

PART / ATYPE = TETRA

PART / ATYPE = SPHEL

PART / ATYPE = COS3D

PART / ATYPE = SHELL

PART / ATYPE = TSHEL

PART / ATYPE = MEMBR

PART / ATYPE = BAR

PART / ATYPE = BEAM

PART / ATYPE = ELINK

PART / ATYPE = GAP

PART / ATYPE = JOINT

PART / ATYPE = KJOIN

PART / ATYPE = LLINK

PART / ATYPE = MBKJN

PART / ATYPE = MBSPR

PART / ATYPE = MTOJNT

PART / ATYPE = PLINK

PART / ATYPE = SLINK

PART / ATYPE = SPRGBM

PART / ATYPE = SPRING

PART / ATYPE = TIED

PENTA6/

PFLOW/

PICEXP/

PICIMP/

PLINK/

PLY/

PLY / ITYP = 0

PLY / ITYP = 1, MATYP = 131

PLY / ITYP = 1, MATYP = 30

PLY / ITYP = 2, SIGMAy

PLY / ITYP = 2, CURVE

PLY / ITYP = 2, POWER

PLY / ITYP = 2, KRUPK

PLY / ITYP = 3, SIGMAy

PLY / ITYP = 3, CURVE

PLY / ITYP = 3, POWER

PLY / ITYP = 3, KRUPK

PLY / ITYP = 7

PLY / ITYP = 8, MATYP = 131

PLY / FAILINP = 1

PLY / FAILINP = 1, FAILTYP = 0

PLY / FAILINP = 1, FAILTYP = 1

PLY / FAILINP = 1, FAILTYP = 2

PLY / FAILINP = 1, FAILTYP = 3

PLY / FAILINP = 1, FAILTYP = 4

PLY / FAILINP = 1, FAILTYP = 5

PLY / FAILINP = 1, FAILTYP = 6

PLY / FAILINP = 1, FAILTYP = 7

PLY / FAILINP = 1, FAILTYP = 8

PLY / FAILINP = 1, FAILTYP = 9

PLY / FAILINP = 1, FAILTYP = 10

PRCTRL/

PREBM/

PREFA/

PROFILE_DMP

PYFUNC/

PYVAR/

RAC3D/

RAN3D/

RBODY/

RBODY / ITRB = 0

RBODY / ITRB = 1

RBODY / ITRB = 2

RBODY / ITRB = 3

RBODY / ITRB = 4

RDA3D/

RDD3D/

RDV3D/

RVE3D/

RESTARTFILES

RETRA/

RUNEND/

RUPMO/

RUPMO / IRUPT = 0

RUPMO / IRUPT = 1

RUPMO / IRUPT = 2

RUPMO / IRUPT = 3
RUPMO / IRUPT = 5
RUPMO / IRUPT = 6
RUPMO / IRUPT = 7
RVE3D/
SECFO/
SECFO / NTYP = CONTACT
SECFO / NTYP = LINK
SECFO / NTYP = PLANE
SECFO / NTYP = SECTION
SECFO / NTYP = SUPPORT
SECFO / NTYP = VOLFRAC
SECFO / NTYP = CONT_MS
SECFO / NTYP = DETECT
SECURE/
SECURE/ ENCRYTYP = MATER
SECURE/ ENCRYTYP = PLY
SECURE/ ENCRYTYP = FUNCT
SECURE/ ENCRYTYP = PART
SECURE_NODE_VISU
SENPT/
SENPTG/
SENSOR/
SENSOR/ ITYP = 1
SENSOR/ ITYP = 2
SENSOR/ ITYP = 3
SENSOR/ ITYP = 4
SENSOR/ ITYP = 5
SENSOR/ ITYP = 6
SENSOR/ ITYP = 7
SENSOR/ ITYP = 8
SENSOR/ ITYP = 9
SENSOR/ ITYP = 10
SENSOR/ ITYP = 11
SENSOR/ ITYP = 12
SENSOR/ ITYP = 13
SENSOR/ ITYP = 14
SHELL/
SHELLCHECK

Permas

$CONDUCTIVITY - materials
$CONDUCTIVITY - loads
$CONLOAD
$CONS3
$CONS4
$CONS6
$CONS8
$CONSTRAINTS
$CONTACT
$CONTVAL
$COOR
$DAMP1
$DAMP3
$DAMP6
$DAMPING
$DENSITY
$DIELECTRIC
$DISLOAD
$DISLOADN
$ECHO
$ELASTIC
$ELCONDUCT
$ELPROP
$ELSYS
$ENTER MATERIAL
$ESET
$ESETBIN
$FLA2
$FLA3
$FLDENS
$FLHEX8
$FLHEX20
$FLPENT6
$FLPENT15
$FLPYR5
$FLTET4
$FLTET10
$FLUID
$FREQLOAD

$MPC_SAME
$MPC WLDSURFACE
$MPC WLSCON
$MPC WLSSURFACE
$NLDAMP
$NLDAMPR
$NLSTIFF
$NLSTIFFR
$NSET
$NSETBIN
$PENTA15
$PENTA6
$PERMEABILITY
$PLASTIC
$PLOTA3
$PLOTA4
$PLOTA6
$PLOTA8
$PLOTL2
$PLOTL3
$PLY
$POINTS
$PRESCRIBE/PREVAL
$PRETENSION LOAD
$PRETENSION PLANE
$PRETENSION THREAD
$PYRA5
$QUAD4
$QUAM4
$REFSYS
$RIGID
$ROTB
$RSYS
$SAME
$SFSET
$SHEAR4
$SHELL3
$SHELL4
$SITUATION

Samcef

.ETASHELL
.ETASOLID
.FRA
.HYP
.LAM
.MAI
.MCC BUSH
.MCC MEAN
.MCE BUSH
.MCE MEAN
.MAT, ANISOTROPIC
.MAT, ISOTROPIC
.MAT, ORTHOTROPIC
.MCT
.NOE
.PHP SHELL
.PLI
.SEL FACE
.SEL NOE
.SEL MAI
.STI
.SUB
.UNIT
.ZYG
.ZYG_AUTO

Radioss

Radioss Starter

#include
#RADIOSS Starter
/ACCEL
/ACTIV
/ADMAS
/ADMESH/GLOBAL
/ADMESH/SET
/ADMESH/STATE/SH3N
/ADMESH/STATE/SHELL
/ALE/BCS
/ALE/CLOSE

/ALE/GRID/DISP
/ALE/GRID/DONEA
/ALE/GRID/SPRING
/ALE/GRID/STANDARD
/ALE/GRID/ZERO
/ALE/MAT
/ALE/MUSCL
/AMS
/ANALY
/ANIM/VERS
/ARCH
/BCS
/BEAM
/BEGIN
/BOX
/BRIC20
/BRICK
/CAA
/CLOAD
/CLUSTER
/CNODE
/CYL_JOINT
/DAMP
/DAMP/INTER
/DEF_SHELL
/DEF_SOLID
/DEFAULT/INTER/TYPE11
/DEFAULT/INTER/TYPE19
/DEFAULT/INTER/TYPE2
/DEFAULT/INTER/TYPE24
/DEFAULT/INTER/TYPE7
/DFS/DETCORD
/DFS/DETLINE
/DFS/DETPLAN
/DFS/DETPOIN
/DFS/DETPOIN/NODE
/DFS/DETPLAN/NODE
/DFS/DETLINE/NODE
/DFS/LASER

/DFS/WAV_SHA
/DRAPE
/END
/EOS/GRUNEISEN
/EOS/POLYNOMIAL
/EOS/PUFF
/EOS/SESAME
/EOS/TILLOTSON
/EOS/NOBEL-ABEL
/EOS/LSZK
/EOS/MURNAGHAN
/EOS/IDEAL-GAS
/EOS/OSBORNE
/EOS/STIFF-GAS
/EREF
/EULER/MAT
/FAIL/BIQUAD
/FAIL/CHANG
/FAIL/CONNECT
/FAIL/EMC
/FAIL/ENERGY
/FAIL/FABRIC
/FAIL/FLD
/FAIL/HASHIN
/FAIL/HC-DSSE
/FAIL/ORTHSTRAIN
/FAIL/JOHNSON
/FAIL/LAD_DAMA
/FAIL/NXT
/FAIL/PUCK
/FAIL/SNCONNECT
/FAIL/SPALLING
/FAIL/TAB1
/FAIL/TBUTCHER
/FAIL/TENSSTRAIN
/FAIL/USERi
/FAIL/WIERZBICKI
/FAIL/WILKINS
/FRAME/FIX

/FRAME/MOV
/FRAME/MOV2
/FRAME/NOD
/FUNCT
/GAUGE
/GAUGE/SPH
/GRAV
/GRBEAM
/GRBRIC
/GRNOD
/GRPART
/GRQUAD
/GRSH3N
/GRSHEL
/GRSPRI
/GRTRUS
/HEAT/MAT
/IMPACC
/IMPDIS/FGEO
/IMPDISP
/IMPLICIT
/IMPTEMP
/IMPVEL
/IMPVEL/FGEO
/IMPVEL/LAGMUL
/INIBRI
/INIBRI/FILL
/INICRACK
/INIQUA/DENS
/INIQUA/ENER
/INIQUA/EPSP
/INIQUA/STRESS
/INISH3/AUX
/INISH3/EPSP
/INISH3/EPSP_F
/INISH3/ORTH_LOC
/INISH3/ORTHO
/INISH3/STRA_F
/INISH3/STRS_F

/INISH3/STRS_F/GLOB
/INISH3/THICK
/INISHE/AUX
/INISHE/EPSP
/INISHE/EPSP_F
/INISHE/FAIL
/INISHE/ORTH_LOC
/INISHE/ORTHO
/INISHE/STRA_F
/INISHE/STRS_F
/INISHE/STRS_F/GLOB
/INISHE/THICK
/INISTA
/INITEMP
/INIVEL
/INIVEL/AXIS
/INIVOL
/INTER/LAGMUL/TYPE7
/INTER/SUB
/INTER/TYPE1
/INTER/TYPE10
/INTER/TYPE11
/INTER/TYPE12
/INTER/TYPE14
/INTER/TYPE15
/INTER/TYPE18
/INTER/TYPE19
/INTER/TYPE2
/INTER/TYPE20
/INTER/TYPE21
/INTER/TYPE22
/INTER/TYPE23
/INTER/TYPE24
/INTER/TYPE25
/INTER/TYPE3
/INTER/TYPE5
/INTER/TYPE6
/INTER/TYPE7
/INTER/TYPE8

/MAT/LAW82
/MAT/LAW83
/MAT/LAW84
/MAT/LAW87
/MAT/LAW88
/MAT/LAW90
/MAT/LAW92
/MAT/LAW93 (ORTH_HILL)
/MAT/LAW94 (YEOH)
/MAT/LAW95
/MAT/LAW97 (/MAT/JWLB)
/MAT/LAW100
/MONVOL/AIRBAG1
/MONVOL/AREA
/MONVOL/COMMU1
/MONVOL/FVMBAG1
/MONVOL/GAS
/MONVOL/PRES
/MOVE_FUNCT
/NODE
/PARAMETER
/PART
/PENTA6
/PLOAD
/PLY
/PROP/INJECT1
/PROP/INJECT2
/PROP/PCOMPP
/PROP/TYPE0 (VOID)
/PROP/TYPE1 (SHELL)
/PROP/TYPE2 (TRUSS)
/PROP/TYPE3 (BEAM)
/PROP/TYPE4 (SPRING)
/PROP/TYPE6 (SOL_ORTH)
/PROP/TYPE8 (SPR_GENE)
/PROP/TYPE9 (SH_ORTH)
/PROP/TYPE10 (SH_COMP)
/PROP/TYPE11 (SH_SANDW)
/PROP/TYPE12 (SPR_PUL)

/SKEW/FIX
/SKEW/MOV
/SKEW/MOV2
/SPH/INOUT
/SPH/RESERVE
/SPHBCS
/SPHCEL
/SPHGLO
/SPMD
/SPRING
/STACK
/STAMPING
//SUBMODEL
/SUBSET
/SURF/BOX
/SURF/DSURF
/SURF/GRBRIC/EXT
/SURF/GRBRIC/FREE
/SURF/GRSH3N
/SURF/GRSHEL
/SURF/MAT
/SURF/MAT/ALL
/SURF/MAT/EXT
/SURF/PART
/SURF/PART/ALL
/SURF/PART/EXT
/SURF/PLANE
/SURF/PROP
/SURF/PROP/ALL
/SURF/PROP/EXT
/SURF/SEG
/SURF/SUBMODEL
/SURF/SUBSET
/SURF/SUBSET/ALL
/SURF/SUBSET/EXT
/SURF/SURF
/TABLE
/TETRA4
/TETRA10

Radioss Engine

/ALE/GRID/DISP
/ALE/GRID/DONEA
/ALE/GRID/SPRING
/ALE/GRID/STANDARD
/ALE/GRID/ZERO
/ALE/SPRING
/ALE/STANDARD
/ALE/ZERO
/ALESUB
/ANIM/BRICK/DAMA
/ANIM/BRICK/TDEL
/ANIM/BRICK/TDET
/ANIM/BRICK/TENS
/ANIM/BRICK/TENS/STRAIN
/ANIM/BRICK/TENS/STRESS
/ANIM/BRICK/VDAM
/ANIM/CUT/1
/ANIM/CUT/2
/ANIM/CUT/3
/ANIM/DT
/ANIM/Eltyp/FORCE
/ANIM/Eltyp/Restype
/ANIM/ELEM/TDET
/ANIM/QUAD/TDET
/ANIM/ELEM/EINT
/ANIM/GPS/TENS
/ANIM/GPS1
/ANIM/GPS2
/ANIM/GZIP
/ANIM/LSENSOR
/ANIM/MASS
/ANIM/MAT
/ANIM/NODA
/ANIM/SHELL/DAMA
/ANIM/SHELL/EPSP
/ANIM/SHELL/NXTFACTOR
/ANIM/SHELL/PHI
/ANIM/SHELL/SIG1H
/ANIM/SHELL/SIG2H

/ANIM/SHELL/TDEL

/ANIM/SHELL/TENS

/ANIM/SHELL/IDPLY

/ANIM/SHELL/FLDZ

/AMIN/SHELL/FLDF

/ANIM/SENSOR

/ANIM/TITLE

/ANIM/VECT

/ANIM/VERS

/ATFILE

/BCS/ALE

/BCS/LAG

/BCS/ROT

/BCS/TRA

/BCSR/ALE

/BCSR/LAG

/BCSR/ROT

/BCSR/TRA

/DAMP

/DEL

/DEL/Eltyp/1

/DEL/INTER

/DELINT

/DT

/DT/CST_AMS

/DT/ALE

/DT/AMS

/DT/Eltyp/Iflag

/DT/Eltyp/Keyword3/Iflag

/DT/GLOB

/DT/NODA/Keyword3

/DT/SHELL

/DT/SHNOD

/DT/SPHCEL

/DT/SPHCEL/Keyword3

/DT/THERM

/DT1/SHELL

/DTIX

/DTSDE

/DYREL
/DYREL/1
/END/ENGINE
/FVMBAG/MODIF
/IMPL/AUTOSPC
/IMPL/BUCKL/1
/IMPL/BUCKL/2
/IMPL/CHECK
/IMPL/DIVERG/n
/IMPL/DT/1
/IMPL/DT/2
/IMPL/DT/3
/IMPL/DT/FIXPOINT
/IMPL/DT/STOP
/IMPL/DTINI
/IMPL/DYNA/1
/IMPL/DYNA/2
/IMPL/DYNA/DAMP
/IMPL/DYNA/FSI
/IMPL/GSTIF/OFF
/IMPL/INTER/KCOMP
/IMPL/INTER/KNONL
/IMPL/LBFGS/L
/IMPL/LINEAR
/IMPL/LINEAR/INTER
/IMPL/LSEARCH/n
/IMPL/LSEARCH/OFF
/IMPL/MONVOL/OFF
/IMPL/NCYCLE/STOP
/IMPL/NONLIN
/IMPL/NONLIN/SMDISP
/IMPL/NONLIN/SOLVINFO
/IMPL/PREPAT
/IMPL/PRINT/LINEAR
/IMPL/PRINT/NONLIN
/IMPL/PSTIF/OFF
/IMPL/QSTAT
/IMPL/QSTAT/DTSCAL
/IMPL/QSTAT/MRIGM

/IMPL/RREF/INTERF/n

/IMPL/RREF/LIMIT

/IMPL/RREF/OFF

/IMPL/SBCS/MSGLV

/IMPL/SBCS/ORDER

/IMPL/SBCS/OUTCORE

/IMPL/SHPOFF

/IMPL/SHPON

/IMPL/SINIT

/IMPL/SOLVER

/IMPL/SPRBACK

/IMPL/SPRING

/INCMP

/INIV/ROT

/INIV/ROT/Keyword3/1

/INIV/TRA

/INIV/TRA/Keyword3/1

/INTER

/KEREL

/KEREL/1

/KILL

/MADYMO

/MON

/OUTP

/OUTP/LSENSOR

/PARITH

/PRINT

/RAD2RAD/ON

/RERUN

/RBODY

/RFILE

/RFILE/n

/RUN

/RUN/Run Name/Run Number/Restart Letter

/SHSUB

/STATE/BRICK/AUX/FULL

/STATE/BRICK/FAIL

/STATE/BRICK/ORTHO

/STATE/BRICK/STRAIN/FULL

/STATE/BRICK/STRAIN/GLOBFULL
/STATE/BRICK/STRES/FULL
/STATE/BRICK/STRESS/GLOBFULL
/STATE/DT
/STATE/LSENSOR
/STATE/NO_DEL
/STATE/NODE/TEMP
/STATE/SHELL/AUX/FULL
/STATE/SHELL/EPSP/FULL
/STATE/SHELL/FAIL
/STATE/SHELL/ORTHL
/STATE/SHELL/STRAIN/FULL
/STATE/SHELL/STRESS/FULL
/STATE/STR_FILE
/STOP
/STOP/LSENSOR
/TFILE
/TH/VERS
/THERMAL
/TITLE
/UPWIND
/UPWM/SUPG
/UPWM/TG
/VEL/ALE
/VEL/ROT
/VEL/TRA
/VERS

CAD Interfaces

4

Support provided by the CAD readers and writers.

This chapter covers the following:

CAD Import

Learn about the CAD readers supported by HyperMesh and the options available for importing CAD geometry data.

CAD readers are dynamically loaded upon demand.

Supported CAD Readers

CAD readers supported by HyperMesh.

Format	Latest Version Supported	File Extensions Supported
ACIS	r21	`.sat`
AVEVA Marine	TXHSTL-R XML Export v1.3	`.xml`

CATIA	v4
	v5-6R2018 (R28)
	v6-2015X
		CATIA v4
		`.model`
		`.exp`
		CATIA v5
		`.CATProduct`
		`.CATPart`
		`.cgr`
		CATIA v6
		`.3dxml`
		`.3DRep`
		`.PLMBriefcase`

CATIA Composites Link		`.h5`
Creo	5.0	`.prt` `.asm`
DXF	AutoCAD 12	`.dxf`
FiberSim		`.h5`
IGES	v6.0 JAMA-IS	`.iges` `.igs`

Format	Latest Version Supported	File Extensions Supported
Inspire	2018.1	`.stmod`
Intergraph	schema 2013	`.xml`
Inventor	2018	`.ipt` `.iam`
JT	10.2	`.jt`
NX	10.0 11.0 12.0	`.prt`

Parasolid	v30.0.185	`.x_t`
		`.x_b`
		`.xmt_txt`
		`.xmt_bin`

PDGS	v26	`.fsf`
Rhino[1]	3	`.3dm`

SolidWorks	2018	`.sldasm`
		`.sldprt`
		`.SLDASM`
		`.SLDPRT`

STEP	AP203 AP214	`.step` `.stp`
VDAFS	v2	`.vda` `.vdafs`

1. Rhino is only supported on Windows.

ACIS Reader

Supported Entities

Entities supported by the ACIS reader.

- Coordinate systems
- Free points
- Free curves
- Surfaces
- Quilt bodies
- Solid bodies

Import Options

The ACIS reader uses the `ct_reader.ini` file.

@AttributesAsMetadata

Value	Description
`on`	Import all generic attributes (global and related to single entities) as metadata.
`off`	Do not import attributes related to single entities. Only import global attributes.
`%attribute1%attribute2%...`	Import specified generic attributes as metadata. The attribute names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '%' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '%'. HyperMesh will recognize it as it is the first character of the string.

@AvoidThirdPartyConversion

Value	Description
`on`	Avoid using the conversion functions provided by third-party (default).
`off`	Use the conversion functions provided by third-party.

@BodyIdAsMetadata

Value	Description
on	Assign body identifier as metadata. BODY_ID
off	Do not assign body identifier as metadata (default).

@CleanupTol

Value	Description
double	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
integer	Read color attributes of geometric entities as metadata with integer values. COLOR_RGB
on	Read color attributes of geometric entities as metadata. COLOR_RGB
off	Do not read color attributes (default).

@CreationType

Value	Description
Parts	Generate a full part-based hierarchy.
BOMOnly	Generate an empty part-based hierarchy.
TreeOfComponents	Generate an assemblies/components based hierarchy.

Value	Description
PackIntoSinglePart	Consolidate hierarchy into a single part.

@DensityAsMetadata

Value	Description
on	Read density value as metadata (default). DENSITY
off	Do not read density value.

@DoNotMergeEdges

Value	Description
on	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
off	Merge edges together during the import cleanup phase (default).

@ImportBlanked

Value	Description
on	Import entities that are hidden, blanked, or no show.
off	Do not import entities that are hidden, blanked, or no show (default).

@ImportCoordinateSystems

Value	Description
on	Import CAD coordinate systems as system collectors (default).
off	Do not import CAD coordinate systems.

@ImportForVisualizationOnly

Value	Description
`on`	Import the model for visualization purposes only. This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
`off`	Import the model in the normal fashion (default).

`@ImportFreeCurves`

Value	Description
`on`	Import free curves (wireframe entities) into the model (default).
`off`	Do not import free curves.

`@ImportFreePoints`

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

`@LayerAsMetadata`

Value	Description
`on`	Read layer value as metadata. `LAYER`
`off`	Do not read layer value (default).

`@LegacyHierarchyAsMetadata`

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part.

Value	Description
	`LEGACY_HIERARCHY`
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

@MetadataPrefix

Value	Description
`string`	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@OriginalIdAsMetadata

Value	Description
`on`	Import original CAD entity IDs as metadata. `ORIGINAL_ID`
`off`	Do not import original CAD entity IDs as metadata (default).

@ScaleFactor

Value	Description
`double`	Define the model scaling factor during import. Default is 1.0.

@ShrinkSurfaces

Value	Description
`on`	For trimmed surfaces, shrink the corresponding base surfaces to only enclose the extents of the external trimming contour, plus a narrow frame.
`off`	Do not modify the base surfaces (default).

@SkipCreationOfSolid

Value	Description
`on`	Read surfaces, but do not create solid entities.
`off`	Create solid entities (default).

@SplitComponents

Value	Description
`Body`	Generate body-based component (default if `CreationType=Parts`).
`Layer`	Generate layer-based component.
`Part`	Generate part-based component.
`General`	Keep component as in CAD (default if `CreationType=TreeOfComponents`).

@StitchingAcrossBodies

Value	Description
`on`	Stitch surfaces belonging to different components.
`off`	Do not stitch surfaces belonging to different components (default).

@UseParasolidNativeReader

Value	Description
`on`	Use the Parasolid reader to read embedded XT breps (default).
`off`	Also use the standard JT reader to read XT breps.

@TargetUnits

Value	Description
`CAD units`	Keep the units of the CAD files (default for GUI).
`MKS [m kg N s]`	Convert to the corresponding units system.
`MMKS [mm kg N s]`	Convert to the corresponding units system.

Value	Description
MPA [mm t N s]	Convert to the corresponding units system.
CGS [cm g dyn s]	Convert to the corresponding units system.
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.
IPS Std [in pounds lbf s]	Convert to the corresponding units system.
IPS Grav [in slinch lbf s]	Convert to the corresponding units system.
FPS Std [ft pounds lbf s]	Convert to the corresponding units system.
FPS Grav [ft slug lbf s]	Convert to the corresponding units system.
Scale factor	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

Supported Metadata

Metadata generated from the ACIS reader.

BODY_ID

Type	Entities	Description
string	points lines surfs	Identifier of the CAD body containing the entity. Generated when @BodyIDAsMetadata=on

COLOR_RGB

Type	Entities	Description
string	points lines surfs	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when @ColorsAsMetadata = on

Type	Entities	Description
	solids	

DENSITY

Type	Entities	Description
`double`	solids	The value of the density of a solid. Generated when `@DensityAsMetadata = on`

LAYER

Type	Entities	Description
`integer`	points lines surfs solids	Value of the layer of an entity. Generated when `@LayerAsMetadata=on`

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles

Type	Entities	Description
		6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

ORIGINAL_ID

Type	Entities	Description
`ORIGINAL_ID`	points lines surfs solids parts	The original CAD entity ID. Generated when `@OriginalIdAsMetadata = on`

AVEVA Marine Reader

Supported Entities

Entities supported by the AVEVA Marine reader.

- Plane panels
- Curved panels
- Knuckled panels
- Plane plates
- Curved plates
- Pillars
- Stiffeners (also with sub-flanges)
- Curved stiffeners
- Face plates (also with sub-flanges)
- Flanges

- Brackets

When `@CreationType = TreeOfComponents`, an assembly tree is created and organized as follows:

- 1 assembly corresponding to the whole ship (1 ship per part).
- 1 assembly per block.
    - 1 assembly per PlanePanel.
        - 1 component for the detailed contour of the current PlanePanel (with relevant option).
        - 1 component for the simple contour of the current PlanePanel (with relevant option).
        - 1 component per PlanePlateGroup. The material name, material side, thickness and offset are created as metadata. If material data are available, a PSHELL material is created.
            1. 1 surface per PlanePlate. The profile existing in the file is used for the external loop. Holes can be added as internal loops (with relevant option).
        - 1 component per PlanePillarGroup.
            1. 1 set of trace lines per PlanePillar.
            2. 1 surface per web (with relevant option).
        - 1 component per PlaneFlangeGroup.
            1. 1 set of trace lines per PlaneFlange.
        - 1 component per PlaneStiffenerGroup.
            1. 1 set of trace lines per PlaneStiffener.
            2. 1 surface per web and per flange (with relevant option).
        - 1 component per PlaneFaceplateGroup.
            1. 1 set of trace lines per PlaneFaceplate.
            2. 1 surface per web and per flange (with relevant option).
        - 1 assembly per sub-PlanePanel. The contents correspond to the ones for normal PlanePanels.
        - 1 assembly per PlaneBracketGroup.
            1. 1 component per bracket if no sub-elements exist (stiffeners, planeplates, flanges). 1 assembly if sub-elements exist.
                1. 1 surface per PlaneBracket. The profile existing in the file is used for the external loop. No holes are allowed by the AVEVA Marine format. If sub-elements are present in the current PlaneBracketGroup, a specific component is created for the surface in order to keep it separate from its sub-elements.
            2. 1 component per PlaneStiffenerGroup sub-element (no flanges inside the stiffener group are allowed).
            3. 1 component per PlaneFaceplateGroup sub-element (no flanges inside the faceplate group are allowed).
            4. 1 component per PlaneFlangeGroup sub-element.
        - 1 assembly per CurvedPanel.
            1. 1 component for the simple contour of the current CurvedPanel (with relevant option).
            2. 1 component per CurvedPlateGroup.

1 1 surface per CurvedPlate. The profile existing in the file is used for the external loop. Internal holes are implemented.

3 1 component per CurvedStiffenerGroup.

1 1 set of trace lines per CurvedStiffener.

- 1 assembly per KnuckledPanel.

When `@CreationType` is "`Parts`", a corresponding tree is created but no assembly entities are created, parts are created instead. The components are also created.

When `@CreationType` is "`BOM Only`", a corresponding tree is created with parts as in the previous case, but no components are actually created.

PlanePlates, CurvedPlates and PlaneBrackets are mapped as surfaces. Other objects are imported as curves, lying on the plates. Groups of objects may share similar properties (such as material, material side, thickness and offset). In this case, metadata are added to these objects, and if possible the material description is also created. Holes of curved surfaces are not taken into consideration for this release.

The reader instantiates objects in their nominal position, hence there may be gaps between panels, brackets, and so forth, due to idealizations that do not take thickness into account. Hence, no stitching between surfaces is performed on import.

Available material fields include Young's modulus, Poisson's ratio, expansion coefficient, and density. Yield stress and ultimate stress are not imported. Each material is associated with a unique grade name.

When a thickness is provided, the corresponding value is given to a HyperMesh properties.

Objects affected by material/thickness include PlanePlateGroup, PlanePillarGroup, PlaneFlangeGroup, PlaneStiffenerGroup, PlaneFaceplateGroup, PlaneBracketGroup, CurvedPlateGroup and CurvedStiffenerGroup.

Import Options

The AVEVA Marine reader uses the `aveva_reader.ini` file.

@BodyIdAsMetadata

Value	Description
`on`	Assign body identifier as metadata. `BODY_ID`
`off`	Do not assign body identifier as metadata (default).

@CleanupTol

Value	Description
`double`	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

`@ColorsAsMetadata`

Value	Description
`integer`	Read color attributes of geometric entities as metadata with integer values. `COLOR_RGB`
`on`	Read color attributes of geometric entities as metadata. `COLOR_RGB`
`off`	Do not read color attributes (default).

`@CreationType`

Value	Description
`Parts`	Generate a full part-based hierarchy.
`BOMOnly`	Generate an empty part-based hierarchy.
`TreeOfComponents`	Generate an assemblies/components based hierarchy.
`PackIntoSinglePart`	Consolidate hierarchy into a single part.

`@CurvedPlateColor`

Value	Description
`string`	A RGB description of the color to use for curved plates, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete `@TribonCurvedPlateColor` is corresponding to this option and still supported.

@CurvedSimpleContourColor

Value	Description
string	A RGB description of the color to use for curved simple contours, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete @TribonCurvedSimpleContourColor is corresponding to this option and still supported.

@CurvedStiffenerColor

Value	Description
string	A RGB description of the color to use for curved stiffeners, with values ranging from 0.0 to 1.0 (e.g. "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete @TribonCurvedStiffenerColor is corresponding to this option and still supported.

@DoNotMergeEdges

Value	Description
on	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
off	Merge edges together during the import cleanup phase (default).

@FlangeColor

Value	Description
string	A RGB description of the color to use for flanges, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete @TribonFlangeColor is corresponding to this option and still supported.

@ImportCutoutProfiles

Value	Description
on	Import cutout profiles.

Value	Description
off	Do not import cutout profiles (default).

@ImportFaceplatesAsSurfaces

Value	Description
on	Import faceplates as surfaces if the surface description is present in the file, otherwise import as curves (default).
off	Import faceplates as curves.

@ImportHoles

Value	Description
on	Create holes (default).
off	Do not create create holes.

@ImportNotchProfiles

Value	Description
on	Import notch profiles.
off	Do not import notch profiles (default).

@ImportPanelProfiles

Value	Description
on	Create curves along the panel profiles.
off	Do not create curves along the panel profiles (default).

@ImportPillarsAsSurfaces

Value	**Description**
`on`	Import pillars as surfaces if the surface description is present in the file, otherwise import as curves (default).
`off`	Import pillars as curves.

@ImportStiffenersAsSurfaces

Value	**Description**
`on`	Import stiffeners as surfaces if the surface description is present in the file, otherwise import as curves (default).
`off`	Import stiffeners as curves.

@ImportUnboundedCurvedPlates

Value	**Description**
`on`	Import curved plates lacking boundary descriptions (default).
`off`	Do not import curved plates lacking boundary descriptions.

@LegacyHierarchyAsMetadata

Value	**Description**
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

@MetadataPrefix

Value	**Description**
`string`	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@PlaneBracketColor

Value	Description
`string`	A RGB description of the color to use for plane brackets, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete `@TribonPlaneBracketColor` is corresponding to this option and still supported.

@PlaneDetailedContoursColor

Value	Description
`string`	A RGB description of the color to use for plane detailed contours, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete `@TribonPlaneDetailedContoursColor` is corresponding to this option and still supported.

@PlaneFaceplateColor

Value	Description
`string`	A RGB description of the color to use for plane faceplates, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete `@TribonPlaneFaceplateColor` is corresponding to this option and still supported.

@PlaneFlangeColor

Value	Description
`string`	A RGB description of the color to use for plane flanges, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete `@TribonPlaneFlangeColor` is corresponding to this option and still supported.

@PlanePillarColor

Value	Description
`string`	A RGB description of the color to use for plane pillars, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete `@TribonPlanePillarColor` is corresponding to this option and still supported.

@PlanePlateColor

Value	Description
`string`	A RGB description of the color to use for plane plates, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete `@TribonPlanePlateColor` is corresponding to this option and still supported.

@PlaneSimpleContourColor

Value	Description
`string`	A RGB description of the color to use for plane simple contours, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete `@TribonPlaneSimpleContourColor` is corresponding to this option and still supported.

@PlaneStiffenerColor

Value	Description
`string`	A RGB description of the color to use for plane stiffeners, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete @TribonPlaneStiffenerColor is corresponding to this option and still supported.

@PreferDetailed

Value	Description
`on`	When multiple representations of a AVEVA Marine object are available, import the most complex one (default).

Value	Description
`off`	When multiple representations of a AVEVA Marine object are available, import the least complex one.

@ScaleFactor

Value	Description
`double`	Define the model scaling factor during import. Default is 1.0.

@SplitComponents

Value	Description
`Part`	Generate part-based component (only allowed value if `CreationType=Parts`).
`General`	Keep component as in CAD (only allowed value if `CreationType=TreeOfComponents`).

@SubFaceplateColor

Value	Description
`string`	A RGB description of the color to use for sub-faceplates, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete `@TribonSubFaceplateColor` is corresponding to this option and still supported.

@SubStiffenerColor

Value	Description
`string`	A RGB description of the color to use for sub-stiffeners, with values ranging from 0.0 to 1.0 (for example "0.1,0.5,0.4"). If not specified, default color management will be utilized. The obsolete `@TribonSubStiffenerColor` is corresponding to this option and still supported.

@TargetUnits

Value	Description
`CAD units`	Keep the units of the CAD files (default for GUI).
`MKS [m kg N s]`	Convert to the corresponding units system.
`MMKS [mm kg N s]`	Convert to the corresponding units system.
`MPA [mm t N s]`	Convert to the corresponding units system.
`CGS [cm g dyn s]`	Convert to the corresponding units system.
`MMKNMS [mm kg kN ms]`	Convert to the corresponding units system.
`MMGNMS [mm g N ms]`	Convert to the corresponding units system.
`IPS Std [in pounds lbf s]`	Convert to the corresponding units system.
`IPS Grav [in slinch lbf s]`	Convert to the corresponding units system.
`FPS Std [ft pounds lbf s]`	Convert to the corresponding units system.
`FPS Grav [ft slug lbf s]`	Convert to the corresponding units system.
`Scale factor`	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

Supported Metadata

Metadata generated from the AVEVA Marine reader.

BENDING_RADIUS

Type	Entities	Description
`string`	assems	The value of the BendingRadius attribute for a plane flange group.

BODY_ID

Type	Entities	Description
`string`	points	Identifier of the CAD body containing the entity.

Type	Entities	Description
	lines surfs	Generated when `@BodyIDAsMetadata=on`

COLOR_RGB

Type	Entities	Description
`string`	comps	Three RGB values, ranging from 0 to 255, indicating the color for the object. Generated when the corresponding `@Tribon<object>Color` option is used.

COMP_ID

Type	Entities	Description
`string`	assems	The value of the CompId attribute for a plate or bracket.

DATA_TYPE

Type	Entities	Description
`string`	assems	The value of the DataType attribute for a panel.

EXTENT_MAX

Type	Entities	Description
`string`	assems	The value of the maximum extent of a block or panel, as retrieved from the Max attribute of the object.

EXTENT_MIN

Type	Entities	Description
`string`	assems	The value of the minimum extent of a block or panel, as retrieved from the Min attribute of the object.

FORCE_UNITS

Type	Entities	Description
`string`	assems	The value of the ForceUnits attribute for a ship units object.

FUNCTIONAL_PROPERTY

Type	Entities	Description
`string`	assems	The value of the FunctionalProperty attribute for a panel or a group of sub-objects of a panel.

HEIGHT

Type	Entities	Description
`string`	assems	The value of the Height attribute for a plane flange group.

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

LENGTH_UNITS

Type	Entities	Description
`string`	assems	The value of the LengthUnits attribute for a ship units object.

MATERIAL_DIRECTION

Type	Entities	Description
`string`	faces	The value of the MaterialDirection attribute for a flange, curved panel or knuckled panel.

MATERIAL_SIDE

Type	Entities	Description
`string`	assems	The value of the material side data for a plate group or bracket group.

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles 6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

OFFSET

Type	Entities	Description
string	assems	The value of the offset data for a plate group.

RENDERING_TYPE

Type	Entities	Description
string	assems	The value of the Type attribute for a ship rendering object.

RootSystemID

Type	Entities	Description
string	assems	Generated for root system assemblies. Its value is the XML field OID.

THICKNESS

Type	Entities	Description
string	assems	The value of the Thickness attribute for a plane flange group.

VERSION

Type	Entities	Description
string	assems	The value of the Version attribute for a ship.

WEIGHT_UNITS

Type	Entities	Description
`string`	assems	The value of the WeightUnits attribute for a ship units object.

CATIA Reader

Supported Entities

Entities supported the CATIA readers.

CATIA v4 Reader

Supported entities.

- Point (type 1)
- Line segment (type 2)
- Parametric curve (type 3)
- Conics (types 20-23)
- Composite curve (type 24)
- Parametric surface (type 5)
- Face (type 6)
- Volume (type 7)
- Coordinate system (type 8)
- Skin (type 13)
- Mock-up solids (type 17-1)
- Exact solids (type 17-2)
- Ditto (type 28)

CATIA v5 Reader

Supported entities.

- Free points
- Free curves
- Surfaces
- Skin bodies
- Solid bodies
- Facets/triangles
- Infinite planes
- Coordinate systems
- Publications
- Composite data
- Parts (`.CatPart`)

- Assemblies (`.CatProduct`)

Entities that are part of other entities are not created as independent entities.
Entities with non-Latin characters in their names are supported.

CATIA v6 Reader

Supported entities.

- Free points
- Free curves
- Surfaces
- Skin bodies
- Solid bodies
- Facets/triangles
- Infinite planes
- Coordinate systems
- Publications
- Composite data
- Parts (`.3dxml`, `.3DRep`)
- Assemblies (`.3dxml`)

Entities that are part of other entities are not created as independent entities.
Entities with non-Latin characters in their names are supported.

Import Options

The CATIA v5 and CATIA v4 readers use the `ct_reader.ini` file.

`@AttributesAsMetadata`

Value	Description
`on`	Import all generic attributes (global and related to single entities) as metadata.
`off`	Do not import attributes related to single entities. Only import global attributes.
`%attribute1%attribute2%`	Import specified generic attributes as metadata. The attribute names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '%' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '%'. HyperMesh will recognize it as it is the first character of the string.

@AvoidThirdPartyConversion

Value	Description
on	Avoid using the conversion functions provided by third-party (default).
off	Use the conversion functions provided by third-party.

@BodyIdAsMetadata

Value	Description
on	Assign body identifier as metadata. BODY_ID
off	Do not assign body identifier as metadata (default).

@CleanupTol

Value	Description
double	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
integer	Read color attributes of geometric entities as metadata with integer values. COLOR_RGB
on	Same as integer.
off	Do not read color attributes (default).

@CreationType

Value	Description
`Parts`	Generate a full part-based hierarchy.
`BOMOnly`	Generate an empty part-based hierarchy.
`TreeOfComponents`	Generate an assemblies/components based hierarchy.
`PackIntoSinglePart`	Consolidate hierarchy into a single part.

`@DensityAsMetadata`

Value	Description
`on`	Read density value as metadata (default). `DENSITY`
`off`	Do not read density value.

`@DisableSurfaceRelimination`

Value	Description
`on`	Disable the third party algorithm for performing base surface trimming during import. This option should be used in conjunction with `@ShrinkSurfaces`.
`off`	Do not disable the third party algorithm (default).

`@DoNotMergeEdges`

Value	Description
`on`	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
`off`	Merge edges together during the import cleanup phase (default).

`@FullNameAsMetadata`

Value	Description
on	Generate the full CAD name, as retrieved from the CAD part, as metadata. This consists of assembly name/part name/feature name/entity name. FULL_IDENTIFIER
off	Do not generate full name metadata (default).

@ImportBlanked

Value	Description
on	Enable the import of invisible (blanked/NO SHOW) components. This option, when used, takes priority over any other similar options/ settings.
off	Disable the import of invisible (blanked/NO SHOW) components (default). This option, when used, takes priority over any other similar options/ settings.

@ImportCoordinateSystems

Value	Description
on	Import CAD coordinate systems as system collectors (default).
off	Do not import CAD coordinate systems.

@ImportForVisualizationOnly

Value	Description
on	Import the model for visualization purposes only. This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.

Value	Description
off	Import the model in the normal fashion (default).

@ImportFreeCurves

Value	Description
on	Import free curves (wireframe entities) into the model (default).
off	Do not import free curves.

@ImportFreePoints

Value	Description
on	Import free points into the model (default).
off	Do not import free points.

@ImportPublicationData

Value	Description
on	Create regions from the geometry referenced in publications.
off	Do not create regions from the geometry referenced in publications (default).

@InfinitePlaneSizeFactor

Value	Description
double	A factor > 0 for sizing of infinite planes. The factor is a percentage of the modal size. The defaault is 0.5.

@LayerAsMetadata

Value	Description
on	Read layer value as metadata. LAYER
off	Do not read layer value (default).

@LegacyHierarchyAsMetadata

Value	Description
on	Generate metadata with the original CAD hierarchy within the part. LEGACY_HIERARCHY
off	Do not generate metadata with the original CAD hierarchy within the part (default).

@MaterialName

Value	Description
string	Name of the CAD attribute containing the PDM material name info of the current part. Default is "Material".

@MetadataPrefix

Value	Description
string	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@MeshFlag

Value	Description
string	Name of the CAD attribute containing the PDM mesh flag name info of the current part.

Value	Description
	Default is "MeshFlag".

@MID

Value	Description
string	Name of the CAD attribute containing the PDM material ID info of the current part. Default is "MID".

@OriginalIdAsMetadata

Value	Description
on	Import original CAD entity IDs as metadata. ORIGINAL_ID
off	Do not import original CAD entity IDs as metadata (default).

@PartNumber

Value	Description
string	Name of the CAD attribute containing the PDM part number info of the current part. Default is "PartNumber”.

@PID

Value	Description
string	Name of the CAD attribute containing PDM property ID info of the current part. Default is "PID”.

@PlyContourGapTolerance

Value	Description
`double`	Acceptable gap between curves of a ply contour (default 0.3). Meaningful if `@ReadCompositeData` is enabled.

`@ReadCompositeData`

Value	Description
`on`	Import CATIAV5 composite data.
`off`	Do not import CATIAV5 composite data (default).

`@ReadHiddenPlyBaseSurfaces`

Value	Description
`on`	Read hidden ply base surfaces. Meaningful if `@ReadCompositeData` is enabled.
`off`	Do not read hidden ply base surfaces (default).

`@RemoveTinySegInCompositeCurves`

Value	Description
`on`	Remove small segments present in composite curves and closes the gap by modifying the adjacent segments, while minimizing the distortion.
`off`	Do not remove small segments present in composite curves.

`@Revision`

Value	Description
`string`	Name of the CAD attribute containing PDM major revision info of the current part. Default is "Revision”.

`@ScaleFactor`

Value	Description
`double`	Define the model scaling factor during import. Default is 1.0.

`@ShrinkSurfaces`

Value	Description
`on`	For trimmed surfaces, shrink the corresponding base surfaces to only enclose the extents of the external trimming contour, plus a narrow frame.
`off`	Do not modify the base surfaces (default).

`@SkipCreationOfSolid`

Value	Description
`on`	Read surfaces, but do not create solid entities.
`off`	Create solid entities (default).

`@SplitComponents`

Value	Description
`Body`	Generate body-based component (default if `CreationType=Parts`).
`Layer`	Generate layer-based component.
`Part`	Generate part-based component.
`General`	Keep component as in CAD (default if `CreationType=TreeOfComponents`).

`@StitchingAcrossBodies`

Value	Description
`on`	Stitch surfaces belonging to different components.
`off`	Do not stitch surfaces belonging to different components (default).

@TagsAsMetadata

Value	Description
on	Create entity name as metadata (default). TAG
FromBody	Take entity from its parent body and create as metadata. TAG
off	Do not read tags.

@TargetUnits

Value	Description
CAD units	Keep the units of the CAD files (default for GUI).
MKS [m kg N s]	Convert to the corresponding units system.
MMKS [mm kg N s]	Convert to the corresponding units system.
MPA [mm t N s]	Convert to the corresponding units system.
CGS [cm g dyn s]	Convert to the corresponding units system.
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.
IPS Std [in pounds lbf s]	Convert to the corresponding units system.
IPS Grav [in slinch lbf s]	Convert to the corresponding units system.
FPS Std [ft pounds lbf s]	Convert to the corresponding units system.
FPS Grav [ft slug lbf s]	Convert to the corresponding units system.
Scale factor	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

@ThicknessName

Value	Description
string	Name of the CAD attribute containing the PDM thickness info of the current part. Default is "Thickness".

@TrimPlySurfaces

Value	Description
on	Trim ply surfaces when a ply points to one surface only. Meaningful if `@ReadCompositeData` is enabled.
off	Do not trim ply surfaces (default).

@UID

Value	Description
string	Name of the CAD attribute containing the PDM UID info of the current part. Default is "UID".

There is also the possibility to specify a default search path for files referenced by assemblies via the `CT_DefaultPartPath` environment variable.

By default, the referenced files will be read from the locations referenced in the assembly files.

If a referenced file cannot be found:

- If the `CT_DefaultPartPath` environment variable is not defined, an attempt will be made to locate the file in the directory where the assembly exists.
- If the `CT_DefaultPartPath` environment variable is defined, an attempt will be made to locate the file in the directories defined by the environment variable, following the order the directories are given (directories separated by semi-colons).

Supported Metadata

Metadata generated from the CATIA reader.

BODY_ID

Type	Entities	Description
`string`	points lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

COLOR_RGB

Type	Entities	Description
`string`	points lines surfs solids	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when `@ColorsAsMetadata = on`

COMMENT_BLOCK

Type	Entities	Description
`string`	assems	The command block text for a CATIA v4 file. This is only attached to the root assembly.

COORDINATE_SYSTEM_<name>

Type	Entities	Description
`string`	assems	The `<name>` field is the name of the local coordinate system and the value is the local coordinate system information.

DENSITY

Type	Entities	Description
`double`	solids	The value of the density of a solid. Generated when `@DensityAsMetadata = on`

FULL_IDENTIFIER

Type	Entities	Description
`string`	points lines surfs solids comps assems	A string indicating the name in the following format: `"part_name/name"` Generated when `@FullNameAsMetadata = on`

LAYER

Type	Entities	Description
`integer`	points lines surfs solids	Value of the layer of an entity. Generated when `@LayerAsMetadata=on`

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles

Type	Entities	Description
		6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

`ORIGINAL_ID`

Type	Entities	Description
`string`	points lines surfs solids parts	The original CAD entity ID. Generated when `@OriginalIdAsMetadata = on`

`TAG`

Type	Entities	Description
`string`	points lines surfs	The tag (name) of the entity as read from the CAD model, if one exists. Generated when `@TagsAsMetadata = on`

Type	Entities	Description
	solids	

CATIA Composites Link Reader

Use the CATIA Composites Link option to import drape data. This method is separate from the CATIA reader, where drape data is not imported.

You need to use the CATIA Composites Link (Simulyt) module and export a HDF5 file and then use that to import in HyperMesh using the CATIA Composites Link connection. This is similar to the FiberSim reader.

1. Import a CAD model (without composite data) using the CATIA geometry import method.
2. Use the CATIA Composites Link connection to import the HDF5 files created from the Simulyt interface in CATIA.
3. When realizing the plies, use the CATIA Composites Link drape map by proximity method in the **Ply Realization** dialog.
4. When exporting the composite data to CAD including draping, use the geometry export option **CATIA Composites**.

Creo Reader

Supported Entities

Entities supported by the Creo reader.

- Free points
- Free curves
- Surfaces
- Quilt bodies
- Solid bodies
- Coordinate systems
- Assemblies
- Assembly Level Features are currently not supported.
- Family Tables are currently not supported.

Entities with non-latin characters in their names are supported.

Import Options

The Creo reader uses the `ct_reader.ini` file.

`@AttributesAsMetadata`

Value	Description
on	Import all generic attributes (global and related to single entities) as metadata.
off	Do not import attributes related to single entities. Only import global attributes.
%attribute1%attribute2%.	Import specified generic attributes as metadata. The attribute names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '%' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '%'. HyperMesh will recognize it as it is the first character of the string.

@AvoidThirdPartyConversion

Value	Description
on	Avoid using the conversion functions provided by third-party (default).
off	Use the conversion functions provided by third-party.

@BodyIdAsMetadata

Value	Description
on	Assign body identifier as metadata. BODY_ID
off	Do not assign body identifier as metadata (default).

@CleanupTol

Value	Description
double	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
on	Read color attributes of geometric entities as metadata. COLOR_RGB
off	Do not read color attributes (default).

@CreationType

Value	Description
Parts	Generate a full part-based hierarchy.
BOMOnly	Generate an empty part-based hierarchy.
TreeOfComponents	Generate an assemblies/components based hierarchy.
PackIntoSinglePart	Consolidate hierarchy into a single part.

@DensityAsMetadata

Value	Description
on	Read density value as metadata (default). DENSITY
off	Do not read density value.

@DoNotMergeEdges

Value	Description
on	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
off	Merge edges together during the import cleanup phase (default).

@FullNameAsMetadata

Value	Description
`on`	Generate the full CAD name, as retrieved from the CAD part, as metadata. This consists of assembly name/part name/feature name/entity name. `FULL_IDENTIFIER`
`off`	Do not generate full name metadata (default).

`@ImportBlanked`

Value	Description
`on`	Import entities that are hidden, blanked, or no show.
`off`	Do not import entities that are hidden, blanked, or no show (default).

`@ImportCoordinateSystems`

Value	Description
`on`	Import CAD coordinate systems as system collectors (default).
`off`	Do not import CAD coordinate systems.

`@ImportForVisualizationOnly`

Value	Description
`on`	Import the model for visualization purposes only. This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
`off`	Import the model in the normal fashion (default).

`@ImportFreeCurves`

Value	Description
`on`	Import free curves (wireframe entities) into the model (default).

Value	Description
`off`	Do not import free curves.

@ImportFreePoints

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

@LayerAsMetadata

Value	Description
`on`	Read layer value as metadata. `LAYER`
`off`	Do not read layer value (default).

@LegacyHierarchyAsMetadata

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

@MaterialName

Value	Description
`string`	Name of the CAD attribute containing the PDM material name info of the current part. Default is "Material".

@MeshFlag

Value	Description
`string`	Name of the CAD attribute containing the PDM mesh flag name info of the current part. Default is "MeshFlag".

@MetadataPrefix

Value	Description
`string`	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@MID

Value	Description
`string`	Name of the CAD attribute containing the PDM material ID info of the current part. Default is "MID".

@OriginalIdAsMetadata

Value	Description
`on`	Import original CAD entity IDs as metadata. `ORIGINAL_ID`
`off`	Do not import original CAD entity IDs as metadata (default).

@PartNumber

Value	Description
`string`	Name of the CAD attribute containing the PDM part number info of the current part.

Value	Description
	Default is "PartNumber".

@PID

Value	Description
`string`	Name of the CAD attribute containing PDM property ID info of the current part. Default is "PID".

@Revision

Value	Description
`string`	Name of the CAD attribute containing PDM major revision info of the current part. Default is "Revision".

@ScaleFactor

Value	Description
`double`	Define the model scaling factor during import. Default is 1.0.

@ShrinkSurfaces

Value	Description
`on`	For trimmed surfaces, shrink the corresponding base surfaces to only enclose the extents of the external trimming contour, plus a narrow frame.
`off`	Do not modify the base surfaces (default).

@SkipCreationOfSolid

Value	Description
on	Read surfaces, but do not create solid entities.
off	Create solid entities (default).

@SplitComponents

Value	Description
Body	Generate body-based component (default if CreationType=Parts).
Layer	Generate layer-based component.
Part	Generate part-based component.
General	Keep component as in CAD (default if CreationType=TreeOfComponents).

@SplitComponentsByBody

Value	Description
on	Create one component per third-party body. There might be one or more third-party bodies per third-party component. This option cannot be used when @ReadCompositeData is enabled, or in conjunction with @SplitComponentsByPart.
off	Do not consider the division of third-party components into third-party bodies while creating components (default).

@StitchingAcrossBodies

Value	Description
on	Stitch surfaces belonging to different components.
off	Do not stitch surfaces belonging to different components (default).

@TagsAsMetadata

Value	Description
on	Create entity name as metadata (default). TAG
FromBody	Take entity from its parent body and create as metadata. TAG
off	Do not read tags.

@TargetUnits

Value	Description
CAD units	Keep the units of the CAD files (default for GUI).
MKS [m kg N s]	Convert to the corresponding units system.
MMKS [mm kg N s]	Convert to the corresponding units system.
MPA [mm t N s]	Convert to the corresponding units system.
CGS [cm g dyn s]	Convert to the corresponding units system.
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.
IPS Std [in pounds lbf s]	Convert to the corresponding units system.
IPS Grav [in slinch lbf s]	Convert to the corresponding units system.
FPS Std [ft pounds lbf s]	Convert to the corresponding units system.
FPS Grav [ft slug lbf s]	Convert to the corresponding units system.
Scale factor	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

@ThicknessName

Value	Description
`string`	Name of the CAD attribute containing the PDM thickness info of the current part. Default is "Thickness".

@UID

Value	Description
`string`	Name of the CAD attribute containing the PDM UID info of the current part. Default is "UID".

Supported Metadata

Metadata generated from the Creo reader.

BODY_ID

Type	Entities	Description
`string`	points lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

COLOR_RGB

Type	Entities	Description
`string`	points lines surfs solids	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when `@ColorsAsMetadata = on`

DENSITY

Type	Entities	Description
`double`	solids	The value of the density of a solid. Generated when `@DensityAsMetadata = on`

FULL_IDENTIFIER

Type	Entities	Description
`string`	points lines surfs solids comps assems	A string indicating the name in the following format: `"part_name/name"` Generated when `@FullNameAsMetadata = on`

LAYER

Type	Entities	Description
`integer`	points lines surfs solids	Value of the layer of an entity. Generated when `@LayerAsMetadata=on`

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles 6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

ORIGINAL_ID

Type	Entities	Description
`string`	points lines edges surfs solids parts	The original CAD entity ID. Generated when `@OriginalIdAsMetadata = on`

TAG

Type	Entities	Description
`string`	points lines edges	The tag (name) of the entity as read from the CAD model, if one exists. Generated when `@TagsAsMetadata = on`

Type	Entities	Description
	surfs solids	

DXF Reader

Supported Entities

Entities supported by the DXF reader.

- Free points (POINT)
- Free curves (LINE)
- Surfaces (3DFACE)
- Solid (SOLID)
- Facets/triangles (POLYLINE with Group Code 70=64 for polyface mesh)

Import Options

The DXF reader uses the `dxf_reader.ini` file.

@BodyIdAsMetadata

Value	Description
`on`	Assign body identifier as metadata. `BODY_ID`
`off`	Do not assign body identifier as metadata (default).

@CleanupTol

Value	Description
`double`	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
integer	Read color attributes of geometric entities as metadata with integer values. COLOR_RGB
on	Read color attributes of geometric entities as metadata. COLOR_RGB
off	Do not read color attributes (default).

@CreationType

Value	Description
Parts	Generate a full part-based hierarchy.
BOMOnly	Generate an empty part-based hierarchy.
TreeOfComponents	Generate an assemblies/components based hierarchy.
PackIntoSinglePart	Consolidate hierarchy into a single part.

@LegacyHierarchyAsMetadata

Value	Description
on	Generate metadata with the original CAD hierarchy within the part. LEGACY_HIERARCHY
off	Do not generate metadata with the original CAD hierarchy within the part (default).

@MetadataPrefix

Value	Description
string	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@ScaleFactor

Value	Description
double	Define the model scaling factor during import. Default is 1.0.

@SplitComponents

Value	Description
Part	Generate part-based component (only allowed value if CreationType=Parts).
General	Keep component as in CAD (only allowed value if CreationType=TreeOfComponents).

@TargetUnits

Value	Description
CAD units	Keep the units of the CAD files (default for GUI).
MKS [m kg N s]	Convert to the corresponding units system.
MMKS [mm kg N s]	Convert to the corresponding units system.
MPA [mm t N s]	Convert to the corresponding units system.
CGS [cm g dyn s]	Convert to the corresponding units system.
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.
IPS Std [in pounds lbf s]	Convert to the corresponding units system.
IPS Grav [in slinch lbf s]	Convert to the corresponding units system.
FPS Std [ft pounds lbf s]	Convert to the corresponding units system.
FPS Grav [ft slug lbf s]	Convert to the corresponding units system.

Value	Description
`Scale factor`	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

Supported Metadata

Metadata generated from the DXF reader.

BODY_ID

Type	Entities	Description
`string`	points lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

FiberSim Reader

The FiberSim reader is supported for Nastran and OptiStruct user profiles.

Supported Entities

Entities supported by the FiberSim HDF5 reader.

Plies

Name, thickness, and fiber orientation information is directly read and mapped as a Ply entity. Plies that point to woven and stack materials are split/separated into multiple plies with half the thickness and correct orientation angles. These split ply names always have _1 and _2 suffix in them for each identification.

Date Map/Table

Data map with element set (ply shapes), material orientation angles (orient1 , orinet2 , draping corrections) thickness corrections, reference direction and normal information for each ply is preserved/mapped in the table entity, therefore each ply has a table associated with it. Nodes and elements are not created in the database from FiberSim triangulation data to define ply shapes.

Instead, it preserves this information in a table so that when these plies are mapped (realized) on actual good mesh, HyperMesh uses this triangular information to define the ply boundary and extract the actual elements.

Laminates

One laminate per HDF5 component with all the ply sequence preserved as per layer_id value.

Materials

Material names and their mechanical properties are read and mapped to solver cards automatically depending on the user profile loaded while importing the model. Currently mechanical properties such as E1, E2, E3, G12, G13, G23, Alpha1, Alpha2 and Alpha_ref temperature are mapped to solver material attributes.

Rosette/Systems

All the system definitions available in the HDF5 file will be imported into one system collector. Currently HyperMesh does not preserve the ply and system relation.

IGES Reader

Supported Entities

Entities supported by the IGES reader.

- Circular arc (100)
- Composite curve (102)
- Conic arc (104)
- Copious data (106)
- Plane (108)
- Line (110)
- Parametric spline curve (112)
- Parametric spline surface (114)
- Point (116)
- Ruled surface (118, form 1 only)
- Surface of revolution (120)
- Tabulated cylinder (122)
- Direction (123)
- Transformation matrix (124)
- Flash (125)
- Rational B-spline curve (126)
- Rational B-spline surface (128)
- Offset surface (140)
- Boundary (141)
- Curve on a parametric surface (142)
- Bounded surface (143)

- Trimmed (parametric) surface (144)
- Manifold solid B-rep object (186)
- Plane surface (190)
- Right circular cylindrical surface (192)
- Right circular conical surface (194)
- Spherical surface (196)
- Toroidal surface (198)
- Angular dimension (202)
- Diameter dimension (206)
- General label (210)
- General note (212)
- Leader (214)
- Linear dimension (216)
- Radius dimension (222)
- General symbol (228)
- Sectioned area (230)
- Line font definition (304)
- Subfigure definition (308)
- Color definition (314)
- Form 7 group without back pointers (402)
- Drawing (404)
- Form 15 name (406)
- Singular subfigure instance (408)
- View (410)
- Vertex (502)
- Edge (504)
- Loop (508)
- Face (510)
- Shell (514)

Import Options

The IGES reader uses the `iges_reader.ini` file. The first section of this file contains instructions for reading each type of IGES entity. It is recommended that you do not change this section. The second section controls the options for the translator.

@CheckFacet

Value	Description
on	Based on the success of the normal faceting operation, more cleanup attempts may be required. One option is to mesh it in advance to check

Value	Description
	the faceting. This may slow down the import due to the possible use of meshing operations, but should result in cleaner surfaces (default).
`off`	Do not check the faceting and apply normal cleanup.

@CleanupTol

Value	Description
`double`	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
`on`	Read color attributes of geometric entities as metadata. `COLOR_RGB`
`off`	Do not read color attributes (default).

@CreationType

Value	Description
`Parts`	Generate a full part-based hierarchy.
`BOMOnly`	Generate an empty part-based hierarchy.
`TreeOfComponents`	Generate an assemblies/components based hierarchy.
`PackIntoSinglePart`	Consolidate hierarchy into a single part.

@DoNotMergeEdges

Value	Description
`on`	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.

Value	Description
off	Merge edges together during the import cleanup phase (default).

@GroupAsRegions

Value	Description
on	Create region entities corresponding to IGES groups.
off	Do not create region entities. (default).

@ImportBlanked

Value	Description
on	Import entities that are hidden, blanked, or no show.
off	Do not import entities that are hidden, blanked, or no show (default).

@ImportForVisualizationOnly

Value	Description
on	Import the model for visualization purposes only. This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
off	Import the model in the normal fashion (default).

@ImportFreeCurves

Value	Description
on	Import free curves (wireframe entities) into the model (default).
off	Do not import free curves.

@ImportFreePoints

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

`@ImportLayers`

Value	Description
`Layers to skip`	Enables the specification of layer numbers to import, in order to skip unwanted layers. To specify layer groupings, enter a hyphen between the beginning and ending values of the desired group. Groups are separated by commas. Example: `@ImportLayers = "1,2-5,100-200"`

`@ImportType`

Value	Description
`ASSEMBLY`	Generate an assembly tree corresponding to the one contained in the file (default). Meaningful if `@CreationType` = "`TreeOfComponents`".
`LAYERS_ONLY`	Create components based on the layer (=level) structure of the file. Meaningful if `@CreationType` = "`TreeOfComponents`".
`LAYERS_AND_GROUPS`	Create components corresponding to layers and groups contained in the file. Meaningful if `@CreationType` = "`TreeOfComponents`".

`@LayerAsMetadata`

Value	Description
`on`	Read layer value as metadata. `LAYER`

Value	Description
off	Do not read layer value (default).

@LegacyHierarchyAsMetadata

Value	Description
on	Generate metadata with the original CAD hierarchy within the part. LEGACY_HIERARCHY
off	Do not generate metadata with the original CAD hierarchy within the part (default).

@MetadataPrefix

Value	Description
string	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@PropertyAsGroup

Value	Description
NEVER_MANAGE	Do not generate special metadata (default). Meaningful if @CreationType = "TreeOfComponents".
MANAGE_CORRECT_ONLY	Read group ID and name as metadata. This is useful for special cases where the group ID and name have special meaning, such as to indicate a property ID and name. This applies to IGES entity type #402, form number #7. GROUP_ID GROUP_NAME Meaningful if @CreationType = "TreeOfComponents".

@ReadAsIndependent

Value	Description

`INDEPENDENT`
`PHYSICALLY_DEPENDENT`
`LOGICALLY_DEPENDENT`
	Enable the import of three types of independent entities.
	Use a semicolon to separate multiple values. When more than one value is used, both independent and logically dependent entities are treated as independent. Generally, this option should only be used for a particular vendor that marks some entities as dependent when they are imported. The reader imports the entities according to the value specified in the file. The default is `INDEPENDENT`.

@ScaleFactor

Value	Description
`double`	Define the model scaling factor during import. Default is 1.0.

@SkipConnectivityComputation

Value	Description
`on`	Import the model without computing connectivity. The surfaces will not be connected. This option speeds up import, but may not be suitable for any other purpose than visualization.
`off`	Import the model in the normal fashion (default).

@SkipCreationOfSolid

Value	Description
`on`	Read surfaces, but do not create solid entities.
`off`	Create solid entities (default).

@SkipEntities

Value	Description
`Entity types and subtypes to skip`	Specific entity types, or even subtypes (that is entity types with specific form numbers) that should be skipped during import. The list of types uses semicolons as separators. Example: `@SkipEntities = "ENTTYPE1;ENTTYPE2.FORM2"`

@SplitComponents

Value	Description
`Body`	Generate body-based component (default if `CreationType=Parts`).
`Layer`	Generate layer-based component.
`Part`	Generate part-based component.
`General`	Keep component as in CAD (default if `CreationType=TreeOfComponents`).

@TagsAsMetadata

Value	Description
`on`	Read tags of supported entities as metadata (default). `TAG`
`off`	Do not read tags.

@TargetUnits

Value	Description
`CAD units`	Keep the units of the CAD files (default for GUI).
`MKS [m kg N s]`	Convert to the corresponding units system.
`MMKS [mm kg N s]`	Convert to the corresponding units system.
`MPA [mm t N s]`	Convert to the corresponding units system.
`CGS [cm g dyn s]`	Convert to the corresponding units system.

Value	Description
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.
IPS Std [in pounds lbf s]	Convert to the corresponding units system.
IPS Grav [in slinch lbf s]	Convert to the corresponding units system.
FPS Std [ft pounds lbf s]	Convert to the corresponding units system.
FPS Grav [ft slug lbf s]	Convert to the corresponding units system.
Scale factor	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

@Transform402form16

Value	Description
on	Entities referenced by an entity type #402 from #16 are transformed from 2D local space into 3D absolute space. Early IGES files from SolidWorks require such an operation.
off	(default)

@TraverseGroups

Value	Description
on	The reader attempts to traverse group entities (default).
off	Ignore references to entities within a group.

@TrimRevolvedWithModelSpaceCurves

Value	Description
`on`	Attempt to compute the boundary definition by projecting 3D trimming curves (if such curves are available) onto the surface only for revolution surface entities (type #120).
`off`	Use parameter space trimming loops whenever possible. Given an IGES file containing correct data, this option is faster and more robust than reading object space loops (default).

`@TrimWithModelSpaceCurves`

Value	Description
`on`	Attempt to compute the boundary definition by projecting 3D trimming curves (if such curves are available) onto the surface. This is useful if the parameter space trimming loops in the file contain incorrect geometry data.
`off`	Use parameter space trimming loops whenever possible. Given an IGES file containing correct data, this option is faster and more robust than reading object space loops (default).

`@TrimWithPreferredRepresentation`

Value	Description
`on`	Attempt to create the boundary definition using 2D or 3D curves, based on the preferred representation provided by entity type #142. This option can be overridden by either `@TrimRevolvedWithModelSpaceCurves = on` or `@TrimWithModelSpaceCurves = on`.
`off`	(default)

`@UseAnsys128Format`

Value	Description
`on`	Attempt to read Ansys NURBS surface format.
`off`	(default)

@vendors

Value	Description
`List of vendor names`	Use vendor information to search the global section of the file to determine if it is from a particular vendor. Each vendor name is separated by semicolons and all spaces in the vendor name must be replaced by an underscore. Example: `@vendors = "vendor1;vendor2;vendor3"` After a vendor has been added to the list, options for that particular vendor can be specified. If a file is recognized as coming from a particular vendor, settings for that vendor take priority over "general" settings. Example: `@<vendor>.<option> = "<value>"`

Supported Metadata

Metadata generated from the IGES reader.

AUTHOR

Type	Entities	Description
`string`	comps	The author block as read from the 'G' section of the file. This is generated only when the block is found in the file.

AUTHORS_ORGANIZATION

Type	Entities	Description
`string`	comps	The author's organization block from the 'S' section of the file. This is generated only when the block is found in the file.

COLOR_RGB

Type	Entities	Description
`string`	lines surfs	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model.

Type	Entities	Description
		Generated when `@ColorsAsMetadata = on`

COMMENT_BLOCK

Type	Entities	Description
`string`	comps	The comment block from the 'S' section of the file. This is generated only when the block is found in the file.

DRAFTING_STANDARD

Type	Entities	Description
`integer`	comps	The drafting standard block from the 'G' section of the file. This is generated only when the block is found in the file.

FILE_NAME

Type	Entities	Description
`string`	comps	The file name block from the 'G' section of the file. This is generated only when the block is found in the file.

GROUP_ID

Type	Entities	Description
`integer`	comps	The IGES entity type #402, form number #7 group ID as read from the CAD model, if one exists. Generated when `@PropertyAsGroup = ManageCorrectOnly`

GROUP_NAME

Type	Entities	Description
`string`	comps	The IGES entity type #402, form number #7 group name as read from the CAD model, if one exists. Generated when `@PropertyAsGroup = ManageCorrectOnly`

IGES_VERSION

Type	Entities	Description
`integer`	comps	The IGES version block from the 'G' section of the file. This is generated only when the block is found in the file.

IMPORT_DATE

Type	Entities	Description
`string`	comps	The import date block from the 'G' section of the file. This is generated only when the block is found in the file.

LAYER

Type	Entities	Description
`integer`	points lines surfs solids	Value of the layer of an entity. Generated when `@LayerAsMetadata=on`

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines	Hierarchy of an entity within a part.

Type	Entities	Description
	surfs	Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles 6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

MODIFICATION_DATE

Type	Entities	Description
`string`	comps	The modification date block from the 'G' section of the file. This is generated only when the block is found in the file.

PREPROCESSOR_VERSION

Type	Entities	Description
`string`	comps	The preprocessor version block from the 'G' section of the file. This is generated only when the block is found in the file.

`PRODUCT_IDENTIFICATION`

Type	Entities	Description
`string`	comps	The product identification block from the 'G' section of the file. This is generated only when the block is found in the file.

`RECEIVING_PRODUCT_ID`

Type	Entities	Description
`string`	comps	The receiving product ID block from the 'G' section of the file. This is generated only when the block is found in the file.

`SYSTEM_ID`

Type	Entities	Description
`string`	comps	The system ID block from the 'G' section of the file. This is generated only when the block is found in the file.

`TAG`

Type	Entities	Description
`string`	points lines surfs	The tag (name) of the entity as read from the CAD model, if one exists. Generated when `@TagsAsMetadata = on`

`UNITS`

Type	Entities	Description
`string`	comps	The units block from the 'G' section of the file. This is generated only when the block is found in the file.

Inspire Reader

Supported Entities

Entities supported by the Inspire reader.

- Models
- Assemblies
- Parts
- Solids
- Sheets
- Curves
- Points
- Axis Systems
- Alternatives
  - Active alternatives are imported by default.
  - All alternatives are imported when hidden (blanked/no show) entities are imported.

The following properties associated to entities are supported:

- Visibility
- Color
- Transparency
- Active status
  - Active geometry is imported by default.
  - All geometry is imported when hidden (blanked/no show) entities are imported.

Import Options

The Inspire reader uses the `inspire_reader.ini` file.

`@BodyIdAsMetadata`

Value	Description
`on`	Assign body identifier as metadata. `BODY_ID`

Value	Description
`off`	Do not assign body identifier as metadata (default).

@CleanupTol

Value	Description
`double`	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
`integer`	Read color attributes of geometric entities as metadata with integer values. `COLOR_RGB`
`on`	Read color attributes of geometric entities as metadata. `COLOR_RGB`
`off`	Do not read color attributes (default).

@CreationType

Value	Description
`Parts`	Generate a full part-based hierarchy.
`BOMOnly`	Generate an empty part-based hierarchy.
`TreeOfComponents`	Generate an assemblies/components based hierarchy.
`PackIntoSinglePart`	Consolidate hierarchy into a single part.

@DensityAsMetadata

Value	Description
`on`	Read density value as metadata (default). `DENSITY`
`off`	Do not read density value.

@DoNotMergeEdges

Value	Description
`on`	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
`off`	Merge edges together during the import cleanup phase (default).

@ExtractManifoldFromGeneralBody

Value	Description
`on`	Extract the manifold body from a Parasolid general body that is non-manifold and/or of mixed topological dimensionality (default).
`off`	Extract Parasolid general bodies as defined.

@ImportBlanked

Value	Description
`on`	Import entities that are hidden, blanked, or no show.
`off`	Do not import entities that are hidden, blanked, or no show (default).

@ImportForVisualizationOnly

Value	Description
`on`	Import the model for visualization purposes only.

Value	Description
	This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
`off`	Import the model in the normal fashion (default).

@ImportFreeCurves

Value	Description
`on`	Import free curves (wireframe entities) into the model (default).
`off`	Do not import free curves.

@ImportFreePoints

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

@LegacyHierarchyAsMetadata

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

@MetadataPrefix

Value	Description
`string`	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

`@OriginalIdAsMetadata`

Value	Description
`on`	Import original Inspire and Parasolid entity IDs as metadata.
`off`	Do not import original Inspire and Parasolid entity IDs as metadata (default).

`@ScaleFactor`

Value	Description
`double`	Define the model scaling factor during import. Default is 1.0.

`@SkipCreationOfSolid`

Value	Description
`on`	Read surfaces, but do not create solid entities.
`off`	Create solid entities (default).

`@SplitComponents`

Value	Description
`Body`	Generate body-based component (default if `CreationType=Parts`).
`Part`	Generate part-based component.
`General`	Keep component as in CAD (default if `CreationType=TreeOfComponents`).

`@SplitPeriodicFaces`

Value	Description
`on`	Split periodic surfaces to improve the quality and robustness of the import (default).
`off`	Import periodic surfaces as a single surface with a seam edge.

@TargetUnits

Value	Description
`CAD units`	Keep the units of the CAD files (default for GUI).
`MKS [m kg N s]`	Convert to the corresponding units system.
`MMKS [mm kg N s]`	Convert to the corresponding units system.
`MPA [mm t N s]`	Convert to the corresponding units system.
`CGS [cm g dyn s]`	Convert to the corresponding units system.
`MMKNMS [mm kg kN ms]`	Convert to the corresponding units system.
`MMGNMS [mm g N ms]`	Convert to the corresponding units system.
`IPS Std [in pounds lbf s]`	Convert to the corresponding units system.
`IPS Grav [in slinch lbf s]`	Convert to the corresponding units system.
`FPS Std [ft pounds lbf s]`	Convert to the corresponding units system.
`FPS Grav [ft slug lbf s]`	Convert to the corresponding units system.
`Scale factor`	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

@UniqueIdAsMetadata

Value	Description
`on`	Split periodic surfaces to improve the quality and robustness of the import (default).
`off`	Import periodic surfaces as a single surface with a seam edge.

Supported Metadata

Metadata generated from the Inspire reader.

BODY_ID

Type	Entities	Description
`string`	points lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

COLOR_RGB

Type	Entities	Description
`string`	points lines surfs solids	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when `@ColorsAsMetadata = on`

DENSITY

Type	Entities	Description
`double`	solids	The value of the density of a solid. Generated when `@DensityAsMetadata = on`

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
`integer`	comps	Model units specified in the CAD file. 1 = inches

Type	Entities	Description
	parts	2 = millimeters 4 = feet 5 = miles 6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

ORIGINAL_ID

Type	Entities	Description
string	comps points lines surfs solids parts	The original Inspire ID for comps and the Parasolid entity IDs for the geometry. Generated when `@OriginalIdAsMetadata = on`

UNIQUE_ID

Type	Entities	Description
string	points lines surfs	A combination of the Inspire body ID and the Parasolid entity ID. Generated when `@OriginalIdAsMetadata = on`

Type	Entities	Description
	solids	

Intergraph Reader

Supported Entities

Entities supported by the Intergraph reader.

Systems

The following systems, listed in hierarchical order, are supported and created as assemblies or components, when `@CreationType = TreeOfComponents`.

- Generic systems. One assembly is created for each generic system.
    - Root systems. One assembly is created for each root system.
        - Plate systems. One component is created for each plate system.
        - Stiffener systems. By default one or more components (depending on the properties you created) are created for each stiffener child geometry; this behavior can be changed based on the ini option `@CreateMultipleComponentsForStiffeners`.
        - ER systems. Same as stiffener systems.
    - Bracket systems. One assembly is created for each bracket system.
        - Bracket system child. One component is created for each bracket child.
        - Stiffener systems
    - Beam systems. Same as stiffener systems.
    - Member systems. By default one component is created for each member part; this behavior can change based on the ini option `@CreateMultipleComponentsForLinearMembers`.

When `@CreationType` is "`Parts`", a corresponding tree is created but no assembly entities are created, parts are created instead. The components are also created.

When `@CreationType` is "`BOM Only`", a corresponding tree is created with parts as in the previous case, but no components are actually created.

Compartments

By default, no compartments are imported. To import compartments, the `@ImportCompartments` or `@ImportIGESCompartments` ini options should be set.

Topological Properties

By default no topological properties are created; this behavior can change based on the ini option `@ImportTopologicalProperties`.

Plate Geometries

The following plate geometries are supported and created:

- Planar

- Curved
- External (supported just in IGES format)

Stiffener/ER/Beam Systems Cross Section

The following stiffer/ER/beam system cross sections are supported and created:

- Flat bar
- Angle
- TBar
- IBar
- Channel
- Round
- Rectangle

Linear Members

The only member parts that are imported have a straight NURBS as a contour 3D, and a cross section of type round.

Materials and Properties

By default, materials and properties are created for the Nastran solver; a different solver can be selected with the ini option `@Solver`.
The following properties are created:

- For each plate system and bracket child system, a PSHELL property is created.
- For stiffener/beam/ER system children, the behavior depends on the analysis type (global or local analysis, see "GUI Options"):
    - Local analysis: By default, a component and a PROD property are created for each ER system child, a component and a PSHELL property are created for each web and flange of stiffener/beam system children.
    - Global analysis: By default, a component and a PROD property are created for each stiffener/beam/ER system child; if it is longitudinal, then a PBAR property is also created.

By default these properties are assigned to the corresponding components; this behavior can be changed by using the ini option `@AssignPropertiesToComponents`.
When `@CreationType` is "`Parts`" the thickness (assigned to the PSHELL property) is also assigned to the part.
Materials are created for plate systems, bracket, stiffener, beam, and ER children; for the Nastran solver, the material created is of type MAT1.
When `@CreationType` is "`Parts`" the material is also assigned to the part.

Import Options

The Intergraph reader uses the `intergraph_reader.ini` file.

`@AssignPropertiesToComponents`

Value	Description
`on`	Assign properties to the corresponding component (for components which have more than one property, properties are not assigned).

Value	Description
off	Do not assign properties to components (default).

@BodyIdAsMetadata

Value	Description
on	Assign body identifier as metadata. BODY_ID
off	Do not assign body identifier as metadata (default).

@CleanupTol

Value	Description
double	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@CreateMetadataForStiffenerNormals

Value	Description
on	Attach some metadata with information about normals to stiffener/beam/ER systems. START NORMAL, START POINT, END NORMAL, END POINT, MOUNTING RULE
off	Do not attach metadata with information about normals to stiffeners/beam/ER systems (default).

@CreateMultipleComponentsForLinearMembers

Value	Description
on	Create one component for each member part (default).

Value	Description
`off`	Group member parts belonging to the same member system based on thickness and diameter, that is, just one component is created for each set of data.

`@CreateMultipleComponentsForStiffeners`

Value	Description
`on`	Create one or more components for each stiffener/beam/ER geometry (one for each property; that is, if we have flange and web, then two different properties and components are created) (default).
`off`	Group stiffener/beam/ER geometries belonging to the same root stiffener/ beam/ER system based on their properties (that is, just one component is created for each set of property data).

`@CreationType`

Value	Description
`Parts`	Generate a full part-based hierarchy.
`BOMOnly`	Generate an empty part-based hierarchy.
`TreeOfComponents`	Generate an assemblies/components based hierarchy.
`PackIntoSinglePart`	Consolidate hierarchy into a single part.

`@DoNotMergeEdges`

Value	Description
`on`	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
`off`	Merge edges together during the import cleanup phase (default).

`@ImportCompartments`

Value	Description
on	Import both compartments in the IGES files, and compartments defined by “Faces”. The former by importing the geometry, the latter just by creating a metadata; also reference planes are created, because they can be referenced by compartments.
off	Do not import any compartments (default).

@ImportIGESCompartments

Value	Description
on	Import compartments in the IGES files.
off	Do not import any compartments in the IGES files (default).

@ImportPlateSystemMetadata

Value	Description
on	Create physical properties assigned in the XML to plate systems and bracket children as metadata.
off	Do not import physical properties assigned in the XML to plate systems and bracket children (default).

@ImportStiffenerMidSurfaces

Value	Description
on	Create stiffener/beam/ER midsurfaces (instead of solid geometry) in both global and local analysis; also create landing curves in Global Analysis.
off	Create stiffener/beam/ER solid geometry in local analysis, and landing curves in Global Analysis (default).

@ImportTopologicalProperties

Value	Description
on	Create reference planes and sketch contours, and attach metadata to the systems which own these properties (see Intergraph Metadata Suppport).

Value	Description
`off`	Do not import topological property (default).

@LegacyHierarchyAsMetadata

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

@MetadataPrefix

Value	Description
`string`	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@PropertyName

Value	Description
`string`	Name of the properties created in HyperMesh. It can contain "special" strings that will be replaced with their meaning: `<COMPONENT_NAME>` is the name of the component to which the property is associated, `<PROPERTY_CARD_NAME>` is the name of the property card. Default is `<COMPONENT_NAME>`

@ScaleFactor

Value	Description
`double`	Define the model scaling factor during import. Default is 1.0.

@Solver

Value	Description
string	Use to choose the solver for which materials and properties are created; currently this string can be "nastran" or "ansys"; the default solver is Nastran.

@SplitComponents

Value	Description
Part	Generate part-based component (only allowed value if CreationType=Parts).
General	Keep component as in CAD (only allowed value if CreationType=TreeOfComponents).

@StiffenerJustCurves

Value	Description
on	Create only landing curves for stiffeners.
off	Create the complete stiffener geometry (default).

@TargetUnits

Value	Description
CAD units	Keep the units of the CAD files (default for GUI).
MKS [m kg N s]	Convert to the corresponding units system.
MMKS [mm kg N s]	Convert to the corresponding units system.
MPA [mm t N s]	Convert to the corresponding units system.
CGS [cm g dyn s]	Convert to the corresponding units system.
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.
IPS Std [in pounds lbf s]	Convert to the corresponding units system.

Value	Description
`IPS Grav [in slinch lbf s]`	Convert to the corresponding units system.
`FPS Std [ft pounds lbf s]`	Convert to the corresponding units system.
`FPS Grav [ft slug lbf s]`	Convert to the corresponding units system.
`Scale factor`	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

Supported Metadata

Metadata generated from the Intergraph reader.

BODY_ID

Type	Entities	Description
`string`	points lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

BRACKET SUPPORT

Type	Entities	Description
`string`	assems	Created for each bracket system and associated to the bracket system assembly. Its value is "Name = <name_1> ID = <id_1>; Name = <name_2> ID = <id_2>; …; Name = <name_n> ID = <id_n>”, where <name_1>, <name_2>, … <name_n> are the support names and <id_1>, <id_2>, …<id_n> are the support ids.

COMPARTMENT <name>

Type	Entities	Description
`string`	assems	Created for each compartment which has a child of type "Faces" and is attached to the generic system assembly to which the compartment belongs to. The <name> field is the name of the compartment; the value is "Faces = <id_1> <id_2>… <id_n> Compartment father = <id>", where <id_1>, <id_2>…<id_n> are the ids of the faces, and <id> is the id of the compartment father if it exists (there could be a hierarchy of compartments). Generated when `@ImportCompartments = on`.

END NORMAL

Type	Entities	Description
`string`	comps	Created for ER systems in any case, and also for stiffener/beam systems in Global Analysis. Its value is the normal direction of the stiffener at the landing curve end point. Generated when `@CreateMetadataForStiffenerNormals = on`.

END POINT

Type	Entities	Description
`string`	comps	Created for ER systems in any case, and also for stiffener/beam systems in Global Analysis. Its value is the normal direction of the stiffener at the landing curve end point. Generated when `@CreateMetadataForStiffenerNormals = on`.

LEGACY_HIERARCHY

Type	Entities	Description
string	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
integer	comps parts	Model units specified in the CAD file. 6 = meters This is always generated.

MOUNTING RULE

Type	Entities	Description
string	comps	Created for ER systems in any case, and also for stiffener/beam systems in Global Analysis. Its value is the mounting rule value as found in the XML. Generated when `@CreateMetadataForStiffenerNormals = on`.

<name>

Type	Entities	Description
double string	assems	The attributes that are in the XML under "Details" are attached to the main ship assembly. The <name> field is the attribute name and the double/string is the value of the attribute.

Type	Entities	Description
`double` `string`	comps	**Physical Properties** Physical properties assigned in the XML to plate systems and bracket child are created as metadata. The <name> field is the property name and the double/ string is the property value. Generated when `@ ImportPlateSystemMetadata = on`. **Molder Properties** Molded properties assigned in the XML to plate systems and bracket child are created as metadata. The <name> field is the property name and the double/ string is the property value.

NamingCategory

Type	Entities	Description
`string`	assems comps	Generated for root system assemblies, bracket system assemblies, stiffener/beam/ER components, and member part components. Its value is the the XML field NamingCategory -> LongDesc.

OBJECT TYPE

Type	Entities	Description
`string`	assems comps	Generated for root system assemblies, bracket system assemblies, and stiffener/beam/ER components. Its value can be: • RootSystem • BracketSystem • RootStiffenerSystem • RootBeamSystem • RootERSystem

PLANE NAME

Type	Entities	Description
`string`	surfaces	Created for reference planes. Its value is the name of the reference plane as found in the XML. Generated when `@ImportTopologicalProperties = on` or `@ ImportCompartments = on`.

REFERENCE PLANE <n>

Type	Entities	Description
`string`	assems	Created for root systems, for each XML child of type TopologicalProperties -> Boundaries -> RefPlane (the number <n> is just the index of the child). Its value is the value of this field. Generated when `@ImportTopologicalProperties = on`.

REFERENCE STRUCTURE <n>

Type	Entities	Description
`string`	assems	Created for root systems, for each XML child of type TopologicalProperties -> Boundaries -> ReferenceStruct (the number <n> is the index of the child). Its value is the value of this field. Generated when `@ImportTopologicalProperties = on`.

RootSystemID

Type	Entities	Description
`string`	assems	Generated for root system assemblies. Its value is the XML field OID.

START NORMAL

Type	Entities	Description
`string`	comps	Created for ER systems in any case, and also for stiffener/beam systems in Global Analysis. Its value is the normal direction of the stiffener at the landing curve start point. Generated when `@CreateMetadataForStiffenerNormals = on`.

START POINT

Type	Entities	Description
`string`	comps	Created for ER systems in any case, and also for stiffener/beam systems in Global Analysis. Its value is the landing curve start point. Generated when `@CreateMetadataForStiffenerNormals = on`.

STIFFENER TYPE

Type	Entities	Description
`string`	comps	Created for each stiffener/beam/ER system component. Its value is the XML field Molded -> Type.

XSECTION DATA

Type	Entities	Description
`string`	comps	Created for ER systems and member parts in any case, and also for stiffener/beam systems in Global Analysis. Its value contains the xsection geometrical data (different for each xsection type) in the form "<data name 1> = <data_value 1> <data name 2> = <data value 2>… <data name n> = <data value n>".

Inventor Reader

Supported Entities

Entities supported by the Inventor reader.

- B-rep geometry

Import Options

The Inventor reader uses the `ct_reader.ini` file.

`@AttributesAsMetadata`

Value	Description
`on`	Import all generic attributes (global and related to single entities) as metadata.
`off`	Do not import attributes related to single entities. Only import global attributes.
`%attribute1%attribute2%`	Import specified generic attributes as metadata. The attribute names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '%' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '%'. HyperMesh will recognize it as it is the first character of the string.

`@AvoidThirdPartyConversion`

Value	Description
`on`	Avoid using the conversion functions provided by third-party (default).
`off`	Use the conversion functions provided by third-party.

@BodyIdAsMetadata

Value	Description
`on`	Assign body identifier as metadata. `BODY_ID`
`off`	Do not assign body identifier as metadata (default).

@CleanupTol

Value	Description
`double`	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
`integer`	Read color attributes of geometric entities as metadata with integer values. `COLOR_RGB`
`on`	Read color attributes of geometric entities as metadata. `COLOR_RGB`
`off`	Do not read color attributes (default).

@CreationType

Value	Description
`Parts`	Generate a full part-based hierarchy.
`BOMOnly`	Generate an empty part-based hierarchy.
`TreeOfComponents`	Generate an assemblies/components based hierarchy.
`PackIntoSinglePart`	Consolidate hierarchy into a single part.

@DensityAsMetadata

Value	Description
`on`	Read density value as metadata (default). `DENSITY`
`off`	Do not read density value.

@DoNotMergeEdges

Value	Description
`on`	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
`off`	Merge edges together during the import cleanup phase (default).

@ImportBlanked

Value	Description
`on`	Import entities that are hidden, blanked, or no show.
`off`	Do not import entities that are hidden, blanked, or no show (default).

@ImportCoordinateSystems

Value	Description
`on`	Import CAD coordinate systems as system collectors (default).

Value	Description
`off`	Do not import CAD coordinate systems.

`@ImportForVisualizationOnly`

Value	Description
`on`	Import the model for visualization purposes only. This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
`off`	Import the model in the normal fashion (default).

`@ImportFreeCurves`

Value	Description
`on`	Import free curves (wireframe entities) into the model (default).
`off`	Do not import free curves.

`@ImportFreePoints`

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

`@LayerAsMetadata`

Value	Description
`on`	Read layer value as metadata. `LAYER`
`off`	Do not read layer value (default).

@LegacyHierarchyAsMetadata

Value	Description
on	Generate metadata with the original CAD hierarchy within the part. LEGACY_HIERARCHY
off	Do not generate metadata with the original CAD hierarchy within the part (default).

@MetadataPrefix

Value	Description
string	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@OriginalIdAsMetadata

Value	Description
on	Import original CAD entity IDs as metadata. ORIGINAL_ID
off	Do not import original CAD entity IDs as metadata (default).

@ScaleFactor

Value	Description
double	Define the model scaling factor during import. Default is 1.0.

@ShrinkSurfaces

Value	Description
on	For trimmed surfaces, shrink the corresponding base surfaces to only enclose the extents of the external trimming contour, plus a narrow frame.

Value	Description
`off`	Do not modify the base surfaces (default).

@SkipCreationOfSolid

Value	Description
`on`	Read surfaces, but do not create solid entities.
`off`	Create solid entities (default).

@SplitComponents

Value	Description
`Body`	Generate body-based component (default if `CreationType=Parts`).
`Layer`	Generate layer-based component.
`Part`	Generate part-based component.
`General`	Keep component as in CAD (default if `CreationType=TreeOfComponents`).

@StitchingAcrossBodies

Value	Description
`on`	Stitch surfaces belonging to different components.
`off`	Do not stitch surfaces belonging to different components (default).

@TargetUnits

Value	Description
`CAD units`	Keep the units of the CAD files (default for GUI).
`MKS [m kg N s]`	Convert to the corresponding units system.
`MMKS [mm kg N s]`	Convert to the corresponding units system.
`MPA [mm t N s]`	Convert to the corresponding units system.
`CGS [cm g dyn s]`	Convert to the corresponding units system.

Value	Description
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.
IPS Std [in pounds lbf s]	Convert to the corresponding units system.
IPS Grav [in slinch lbf s]	Convert to the corresponding units system.
FPS Std [ft pounds lbf s]	Convert to the corresponding units system.
FPS Grav [ft slug lbf s]	Convert to the corresponding units system.
Scale factor	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

Supported Metadata

Metadata generated from the Inventor reader.

BODY_ID

Type	Entities	Description
string	points lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

COLOR_RGB

Type	Entities	Description
string	points lines surfs solids	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when `@ColorsAsMetadata = on`

DENSITY

Type	Entities	Description
`double`	solids	The value of the density of a solid. Generated when `@DensityAsMetadata = on`

LAYER

Type	Entities	Description
`integer`	points lines surfs solids	Value of the layer of an entity. Generated when `@LayerAsMetadata=on`

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles 6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters

Type	Entities	Description
		11 = microinches 12 = decimeters 13 = yards This is always generated.

ORIGINAL_ID

Type	Entities	Description
ORIGINAL_ID	points lines surfs solids parts	The original CAD entity ID. Generated when `@OriginalIdAsMetadata = on`

FULL_IDENTIFIER

Type	Entities	Description
string	points lines surfs solids comps assems	A string indicating the name in the following format: `"part_name/name"` Generated when `@FullNameAsMetadata = on`

JT Reader

Supported Entities

Entities supported by the JT reader.

- Free points
- Free curves

- Surfaces
- Solid bodies (JT B-rep)
- Embedded Parasolid (XT B-rep)
- Facets (triangular only)
- Parts
- Assemblies

Import Options

The JT reader uses the `jt_reader.ini` file.

@ApplyLayerFilters

Value	Description
`string`	Import only the specified layer filters. The string should be in the form of "`%filter1%filter2%filter3`". The filter names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '`%`' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '`%`'. HyperMesh will recognize it as it is the first character of the string. A special value of `DEFAULT` can be used to indicate the default layer filter.

@AttributesAsMetadata

Value	Description
`all`	Import all generic attributes as metadata (default). `<name>`
`%attribute1%attribute2%`	Import specified generic attributes as metadata. The attribute names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '`%`' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '`%`'. HyperMesh will recognize it as it is the first character of the string. `<name>`

@BodyIdAsMetadata

Value	Description
`on`	Assign body identifier as metadata. `BODY_ID`
`off`	Do not assign body identifier as metadata (default).

@BrepAndTessLoadOption

Value	Description
`0`	Import B-rep or tessellation, with B-rep given preference. If B-rep is present, tessellation is not imported (default).
`1`	Import both the B-rep and the tessellation.
`2`	Import only the B-rep. If B-rep is not present, nothing is imported.
`3`	Import only the tessellation. If tessellation is not present, nothing is imported.

@CleanupTol

Value	Description
`double`	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
`on`	Read color attributes of geometric entities as metadata. `COLOR_RGB`
`off`	Do not read color attributes (default).

@CreationType

Value	**Description**
`Parts`	Generate a full part-based hierarchy.
`BOMOnly`	Generate an empty part-based hierarchy.
`TreeOfComponents`	Generate an assemblies/components based hierarchy.
`PackIntoSinglePart`	Consolidate hierarchy into a single part.

`@DoNotMergeEdges`

Value	**Description**
`on`	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
`off`	Merge edges together during the import cleanup phase (default).

`@FilterByJTOpenParts`

Value	**Description**
`string`	The string should be in the form of "`%part1%part2%part3`". The part names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '%' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '%'. HyperMesh will recognize it as it is the first character of the string.

`@ImportForVisualizationOnly`

Value	**Description**
`on`	Import the model for visualization purposes only. This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
`off`	Import the model in the normal fashion (default).

`@ImportFreeCurves`

Value	Description
`on`	Import free curves (wireframe entities) into the model (default).
`off`	Do not import free curves.

@ImportFreePoints

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

@LegacyHierarchyAsMetadata

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

@MaterialName

Value	Description
`string`	Name of the CAD attribute containing the material name info of the current part. Default is "P_MAT".

@MeshFlag

Value	Description
`string`	Name of the CAD attribute containing the PDM mesh flag name info of the current part. Default is "MeshFlag".

@MetadataPrefix

Value	Description
`string`	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@MID

Value	Description
`string`	Name of the CAD attribute containing the PDM material ID info of the current part. Default is "MID".

@PartNumber

Value	Description
`string`	Name of the CAD attribute containing the PDM part number info of the current part. Default is "DB_PART_NO".

@PID

Value	Description
`string`	Name of the CAD attribute containing PDM property ID info of the current part. Default is "PID”.

@Revision

Value	Description
`string`	Name of the CAD attribute containing PDM major revision info of the current part. Default is "Revision”.

@SkipCreationOfSolid

Value	Description
`on`	Read surfaces, but do not create solid entities.
`off`	Create solid entities (default).

`@SplitComponents`

Value	Description
`Body`	Generate body-based component (default if `CreationType=Parts`).
`Part`	Generate part-based component.
`General`	Keep component as in CAD (default if `CreationType=TreeOfComponents`).

`@SplitPeriodicFaces`

Value	Description
`on`	Split periodic surfaces to improve the quality and robustness of the import (default).
`off`	Import periodic surfaces as a single surface with a seam edge.

`@TagsAsMetadata`

Value	Description
`on`	Create entity name as metadata (default). `TAG`
`FromBody`	Take entity from its parent body and create as metadata. `TAG`
`off`	Do not read tags.

`@TargetUnits`

Value	**Description**
`CAD units`	Keep the units of the CAD files (default for GUI).
`MKS [m kg N s]`	Convert to the corresponding units system.
`MMKS [mm kg N s]`	Convert to the corresponding units system.
`MPA [mm t N s]`	Convert to the corresponding units system.
`CGS [cm g dyn s]`	Convert to the corresponding units system.
`MMKNMS [mm kg kN ms]`	Convert to the corresponding units system.
`MMGNMS [mm g N ms]`	Convert to the corresponding units system.
`IPS Std [in pounds lbf s]`	Convert to the corresponding units system.
`IPS Grav [in slinch lbf s]`	Convert to the corresponding units system.
`FPS Std [ft pounds lbf s]`	Convert to the corresponding units system.
`FPS Grav [ft slug lbf s]`	Convert to the corresponding units system.
`Scale factor`	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

@ThicknessName

Value	**Description**
`string`	Name of the CAD attribute containing the PDM thickness info of the current part. Default is "P_GAUGE".

@UID

Value	**Description**
`string`	Name of the CAD attribute containing the PDM UID info of the current part.

Value	Description
	Default is "UID".

Supported Metadata

Metadata generated from the JT reader.

BODY_ID

Type	Entities	Description
`string`	lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

COLOR_RGB

Type	Entities	Description
`string`	lines surfs comps	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when `@ColorsAsMetadata = on`.

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet

Type	Entities	Description
		5 = miles 6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

<name>

Type	Entities	Description

`integer`
`double`
`string`
	comps
	assems
		The <name> field is the JT attribute name and the integer/double/string is the value of the JT attribute.
		Generated when `@AttributesAsMetadata = all` or `<name>` is one of the selected attributes.

TAG

Type	Entities	Description
`string`	points lines surfs	The tag (name) of the entity as read from the CAD model, if one exists. Generated when `@TagsAsMetadata = on`

Type	Entities	Description
	solids	

NX Reader

NX import is available through two different readers: UGOpen native reader and a third-party non-native reader.

NX Native Reader

The NX reader utilizes the UGOpen library to read files from NX. The reader relies on a valid NX installation and license to access these libraries. Environment variables must be set appropriately to ensure proper access to these libraries. Any NX file formats not supported by the available NX installation are not supported.

Supported Entities

Entities supported by the Native reader.

- UF_coordinate_system
- UF_point
- UF_line
- UF_circle
- UF_conic
- UF_spline
- UF_faceted_model
- UF_solid
- When reading an NX assembly or part file with material information, the material information is read into HyperMesh as Nastran MAT1 material collectors. When `@CreationType = TreeOfComponents` if there is more than one material associated to the entities in a given part file, HyperMesh splits the part into multiple component collectors. When `@CreationType = Parts` this happens depending on the split type, that is a material is created and associated to the component only if there is just one material associated to the entities of that component. A property collector is always created when importing material information and assigned to the component (see note below), and the material collector is then associated to the respective property collector.
- The NX Native reader also recognizes midsurface thickness information for each part of an assembly. After the part is imported, the thickness information is stored in Nastran PSHELL property collectors. If no thickness information is present but material information does exist, an empty PSHELL property collector is created and the material is assigned to the property collector. The property collector is then assigned to the component collector.

- When `@creationType = TreeOfComponents`, midsurface geometry is organized into a separate component from the solid geometry it is associated to, using a similar name as the solid component. When `@CreationType = Parts` this happens depending on the split type.

Environment Variables

Since the NX reader needs to use the UGOpen library during the run time, it requires that a valid NX installation and NX license be present and available to the user with the assemblies, gateway and solid_modeling modules.

Set the following environment variables prior to starting:

Windows

UGII_BASE_DIR

Must point to the NX installation directory[1].

UGII_ROOT_DIR

Must point to the NX installation UGII directory[1].

PATH

Must include the `%UGII_BASE_DIR%\UGII\` directory, and for NX11 and NX12 the `%UGII_BASE_DIR%\NXBIN` directory[2].

SPLM_LICENSE_SERVER

Must point to the NX license server.

UGS_LICENSE_BUNDLE

Must specify the NX license bundle.

Example: NX installation located at `C:\Program Files\Siemens\NX 12.0`
UGII_BASE_DIR: `C:\Program Files\Siemens\NX 12.0`
UGII_ROOT_DIR: `%UGII_BASE_DIR%\UGII\`
PATH: `%UGII_BASE_DIR%\UGII\`
SPLM_LICENSE_SERVER: `28000@licsrv`
UGS_LICENSE_BUNDLE: `NXPTNR100`

LINUX

UGII_BASE_DIR

Must point to the NX installation directory.[1]

UGII_ROOT_DIR

Must point to the NX installation bin directory.[1]

SPLM_LICENSE_SERVER

Must point to the NX license server.

UGS_LICENSE_SERVER

UGS_LICENSE_SERVER

Must specify the NX license bundle.

Example: NX installation located at `/soft/usr/ugs120`
UGII_BASE_DIR: `/soft/usr/ugs120`
UGII_ROOT_DIR: `/soft/usr/ugs120/bin/`

UGS_LICENSE_SERVER: `28000@licsrv`
UGS_LICENSE_BUNDLE: `NXPTNR100`

1. UGII_ROOT_DIR is needed only for NX10. NX is very sensitive about the environment variables. You should NOT have '/' at the end of UGII_BASE_DIR path and you MUST have '/' at the end of UGII_ROOT_DIR path.
2. It is advisable to specify %UGII_BASE_DIR%\UGII\ at the beginning of the PATH environment variable to avoid DLL version conflicts.

Import Options

The NX Native reader uses the `ug_reader.ini` file. The NX reader also includes a browser to control the import of assemblies, the sorting of entities by component, and various other import details. The NX Part browser will come up just for GUI import when Creation type is set to **Assemblies**.

@AssignThicknessToSolids

Value	Description
`on`	Assign thickness to a solid if the owning part has an attribute with the name THICKNESS. The thickness value is the value of this attribute.
`off`	Do not assign thickness to solids (default).

@AttributesAsMetadata

Value	Description
`on`	Import all generic attributes (global and related to single entities) as metadata. `<name>`
`off`	Do not import attributes related to single entities. Only import global attributes.
`%attribute1%attribute2%.`	Import specified generic attributes as metadata. The attribute names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '%' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '%'. HyperMesh will recognize it as it is the first character of the string. `<name>`

@BodyIdAsMetadata

Value	Description
on	Assign body identifier as metadata. BODY_ID
off	Do not assign body identifier as metadata (default).

@CleanupTol

Value	Description
double	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
on	Read color attributes of geometric entities as metadata. COLOR_RGB
off	Do not read color attributes (default).

@ComponentAttributes

Value	Description
string	In batch mode, this corresponds to the NX Part Browser Component attributes option. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

@ComponentNameFormat

Value	Description
string	In batch mode, this corresponds to the NX Part Browser Assign components by name option; the default value is "default", that is the

Value	Description
	standard component naming; to have custom component name one should set: `@ComponentNameFormat = "custom <custom1><custom2>...<customnn>"` where `<customi>` could be one of the following: `<Part Name>`, `<Part Number>`, `<RID-PDI>`, `<Layer>`, `<Category>`, `<Instance Name>`, `<Material Name>`, `<Thickness>`, `<UG Body ID>`, `<UG Tag>`, `<Solid Name>`. In GUI import when Creation type” is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

@CoordinateSystem

Value	Description
`global`	In batch mode, individual parts in an assembly are imported into the global coordinate system (default). In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.
`local`	In batch mode, individual parts in an assembly are imported into their local coordinate system. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

@CreateMatsAndProps

Value	Description
`on`	Create materials and properties in HyperMesh if material and thickness data are found in the model (default).
`off`	Do not create materials and properties in HyperMesh.

@CreateWeldsInHypermesh

Value	Description
`on`	In batch mode, welds are created in HyperMesh.

Value	Description
	In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.
`off`	In batch mode, welds are not created in HyperMesh (default). In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

@CreationType

Value	Description
`Parts`	Generate a full part-based hierarchy.
`BOMOnly`	Generate an empty part-based hierarchy.
`TreeOfComponents`	Generate an assemblies/components based hierarchy.
`PackIntoSinglePart`	Consolidate hierarchy into a single part.

@DensityAsMetadata

Value	Description
`on`	Read density value as metadata (default). `DENSITY`
`off`	Do not read density value.

@DisableLayers

Value	Description
`string`	This corresponds to the NX Part browser Disable option in Layer filtering mode. The layers to disable in the import should be listed with the same rules as for the NX Part browser entry.

@Display

Value	Description
`categories`	In batch mode, this corresponds to the NX Part browser Categories selection. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.
`layerfiltering`	In batch mode, this corresponds to the NX Part browser Layer filtering selection (default). In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

`@DoNotMergeEdges`

Value	Description
`on`	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
`off`	Merge edges together during the import cleanup phase (default).

`@EnabledLayers`

Value	Description
`string`	This corresponds to the NX Part browser Enable option in Layer filtering mode. The layers to enable in the import should be listed with the same rules as for the NX Part browser entry.

`@ExportToMasterWeldFile`

Value	Description
`on`	In batch mode, a master weld file is created. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.
`off`	In batch mode, no master weld file is created (default).

Value	Description
	In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

@ImportBlanked

Value	Description
on	In batch mode, imports entities that are hidden, blanked, or no show. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.
off	In batch mode, do not import entities that are hidden, blanked, or no show (default). In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

@ImportCoordinateSystems

Value	Description
on	Import CAD coordinate systems as system collectors (default).
off	Do not import CAD coordinate systems.

@ImportForVisualizationOnly

Value	Description
on	Import the model for visualization purposes only. This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
off	Import the model in the normal fashion (default).

@ImportFreeCurves

Value	Description
`on`	Import free curves (wireframe entities) into the model (default).
`off`	Do not import free curves.

`@ImportFreePoints`

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

`@ImportParameters`

Value	Description

`on`	Import parameters.
	Parameters are imported even if they are not associated to any geometry. See also `@ParametersPrefix`.
	For parameters that have relationships to other parameters, parameter expressions are created.
	In addition, `@DoNotMergeEdges` is automatically set to "`on`" when using this option, to give the best results for edge parameter association.
	`CADIO_FEAT_PARAM_IDS`
	`FULL_NAME`

`off`	Do not import parameters (default).

`@ImportWaveLinkedGeometry`

Value	Description
`on`	Import geometry created by the WAVE Geometry Linker.
`off`	Do not import geometry created by the WAVE Geometry Linker (default).

`@IncludeWireFrame`

Value	Description
`on`	In batch mode, import free curves and points (default). In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.
`off`	In batch mode, do not import free curves and points. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

`@LayerAsMetadata`

Value	Description
`on`	Read layer value as metadata. `LAYER`
`off`	Do not read layer value (default).

`@LegacyHierarchyAsMetadata`

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

`@MatchUGAssemblyHierarachy`

Value	Description
`on`	The assembly hierarchy created in HyperMesh is the same as seen in NX.
`off`	An assembly level is created in HyperMesh for parts contained in an assembly (default).

`@MaterialName`

Value	Description
string	Name of the CAD attribute containing the material name info of the current part. Default is "P_MAT".

@MeshFlag

Value	Description
string	Name of the CAD attribute containing the PDM mesh flag name info of the current part. Default is "MeshFlag".

@Merge

Value	Description
on	In batch mode, the MERGE metadata is generated. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.
off	In batch mode, the MERGE metadata is not generated. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

@MetadataPrefix

Value	Description
string	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@MID

Value	Description
string	Name of the CAD attribute containing the PDM material ID info of the current part.

Value	Description
	Default is "MID".

@Midsurface

Value	Description
`on`	In batch mode, and when `@Display= layerfiltering`, midsurfaces are imported. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.
`off`	In batch mode, and when `@Display= layerfiltering`, midsurfaces are not imported (default). In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

@ModelAttributes

Value	Description
`string`	In batch mode, this corresponds to the NX Part browser Model attributes option. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

@OriginalIdAsMetadata

Value	Description
`on`	Import original CAD entity IDs as metadata. `ORIGINAL_ID`
`off`	Do not import original CAD entity IDs as metadata (default).

@ParametersPrefix

Value	Description
`string`	Only import parameters with this prefix. All others will not be imported. Default is "HW_"; meaningful only if `@ImportParameters` = "`on`".

@PartNumber

Value	Description
`string`	Name of the CAD attribute containing the PDM part number info of the current part. Default is "PartNumber”.

@PID

Value	Description
`string`	Name of the CAD attribute containing PDM property ID info of the current part. Default is "PID”.

@PropertyName

Value	Description
`string`	Name of the properties created in HyperMesh. It can contain “special” strings that will be replaced with their meaning: `<COMPONENT_NAME>` is the name of the component to which the property is associated, `<PROPERTY_CARD_NAME>` is the name of the property card. Default is `<COMPONENT_NAME>`

@Revision

Value	Description
`string`	Name of the CAD attribute containing PDM major revision info of the current part. Default is "Revision”.

@SaveSettingsAsDefault

Value	Description
on	In batch mode, the ug_reader.ini file is saved with the current settings. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.
off	In batch mode, the ug_reader.ini file is not saved (default). In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

@ScaleFactor

Value	Description
double	Define the model scaling factor during import. Default is 1.0.

@SelectedCategories

Value	Description
string	This corresponds to the NX Part browser option Categories; they should be listed as they are written in the command file string: "<name1_length> <name1> <name2_length> <name2>…", where <name<i>_length> is a number representing the string length of the ith category name.

@SelectedParts

Value	Description
string	This corresponds to the NX Part browser Parts selection; they should be listed as they are written in the command file string: "<handle1_length> <handle1> <handle2_length> <handle2>…", where <handle<i>> is a string representing the handle of the part as retrieved by UGOpen libraries and <handle<i>_length> is the string length.

@ShowAllCategories

Value	Description
`on`	In batch mode, when `@Display= categories`, all categories will be retrieved and displayed. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.
`off`	In batch mode, when `@Display= categories`, only categories in the top assembly will be retrieved and displayed (default). In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

`@SkipCreationOfSolid`

Value	Description
`on`	Read surfaces, but do not create solid entities.
`off`	Create solid entities (default).

`@Solid`

Value	Description
`on`	In batch mode, when `@Display= layerfiltering`, solids are imported (default). In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.
`off`	In batch mode, when `@Display= layerfiltering`, solids are not imported. In GUI import when Creation type is set to **Assemblies**, this initializes the corresponding option in the NX Part browser.

`@SplitComponents`

Value	Description
`Body`	Generate body-based component (default if CreationType=Parts).
`Category`	Generate category-based component.

Value	Description
`Layer`	Generate layer-based component.
`Material`	Generate material-based component.
`InstanceName`, `Part`, `PartName`, `PartNumber`, `RID-PDI`, `UGTag`	Generate part-based component (and the name is given based on the specific split value).
`Solid`	Generate solid-based component.
`Thickness`	Generate thickness-based component.

@SplitPeriodicFaces

Value	Description
`on`	Split periodic surfaces to improve the quality and robustness of the import (default).
`off`	Import periodic surfaces as a single surface with a seam edge.

@StitchDifferentSheets

Value	Description
`on`	Stitching across different sheet bodies belonging to the same NX part/ instance is enabled (default).
`off`	Stitching across different NX sheet bodies is disabled.

@TagsAsMetadata

Value	Description
`on`	Create entity name as metadata. `TAG`
`FromBody`	Take entity from its parent body and create as metadata. `TAG`

Value	Description
off	Do not read tags (default).

@TargetUnits

Value	Description
CAD units	Keep the units of the CAD files (default for GUI).
MKS [m kg N s]	Convert to the corresponding units system.
MMKS [mm kg N s]	Convert to the corresponding units system.
MPA [mm t N s]	Convert to the corresponding units system.
CGS [cm g dyn s]	Convert to the corresponding units system.
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.
IPS Std [in pounds lbf s]	Convert to the corresponding units system.
IPS Grav [in slinch lbf s]	Convert to the corresponding units system.
FPS Std [ft pounds lbf s]	Convert to the corresponding units system.
FPS Grav [ft slug lbf s]	Convert to the corresponding units system.
Scale factor	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

@ThicknessName

Value	Description
string	Name of the CAD attribute containing the PDM thickness info of the current part. Default is "P_GAUGE".

@UID

Value	Description
string	Name of the CAD attribute containing the PDM UID info of the current part. Default is "UID".

@UniqueIdAsMetadata

Value	Description
on	Create metadata to track the unique CAD ID. When a Parasolid body is present in the Parasolid assembly, the Parasolid instance ID is appended to the ORIGINAL_ID using "-". For each assembly level, its instance ID will be appended in the same way. For example: No body entities present in the assembly: 20 Body entities present in a single level assembly: 1-20 Body entities present in a multiple level assembly: 1-4-2-20 The same identifier can be obtained for two entities when a single entity is split during import. UNIQUE_ID
off	Do not create metadata to track the unique CAD ID (default).

Supported Metadata

Metadata generated from the NX Native reader.

BODY_ID

Type	Entities	Description
string	points lines surfs	Identifier of the CAD body containing the entity. Generated when @BodyIDAsMetadata=on

CADIO_FEAT_PARAM_IDS

Type	**Entities**	**Description**
entity array	points lines surfs solids	This metadata is assigned to the geometric objects for which the imported parameters apply. One or more parameters may control a particular object. This association is only for parameters that are directly assigned to the object in NX. The association is not done for any parameters that may be part of an expression, since that relationship can be determined from the parameter expression itself. Generated when `@ImportParameters = on`

COLOR_RGB

Type	**Entities**	**Description**
`string`	points lines surfs solids	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when `@ColorsAsMetadata = on`

DENSITY

Type	**Entities**	**Description**
`double`	solids	The value of the density of a solid. Generated when `@DensityAsMetadata = on`

FULL_NAME

Type	**Entities**	**Description**
`string`	params	Parameters are named with the NX default name. This contains the full name of the parameter, as defined in NX.

Type	Entities	Description
		Generated when `@ImportParameters = on`

LAYER

Type	Entities	Description
`integer`	points lines surfs solids	Value of the layer of an entity. Generated when `@LayerAsMetadata=on`

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MERGE

Type	Entities	Description
`integer`	comps	Set to 1 if the Merge assemblies option is chosen in the Part browser. Generated when `@Merge = on`

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles

Type	Entities	Description
		6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

NX_MIDSURFACE

Type	Entities	Description
`integer`	comps	Generated and set to 1 when `@CreationType="TreeOfComponents"`, if the component contains midsurface geometry.

ORIGINAL_ID

Type	Entities	Description
`string`	points lines surfs solids parts	The original CAD entity ID. Generated when `@OriginalIdAsMetadata = on`

TAG

Type	Entities	Description
`string`	points lines	The tag (name) of the entity as read from the CAD model, if one exists.

Type	Entities	Description
	edges surfs solids	Generated when `@TagsAsMetadata = on`

UG_DESCRIPTIVE_PARTNAME

Type	Entities	Description
`string`	assems comps	The NX descriptive part name of the assembly/comp. For example, its attribute DB_PART_DESC. This is always generated, if present.

UG_PARTNAME

Type	Entities	Description
`string`	assems comps	The NX part name of the assembly/comp. This is always generated.

UNIQUE_ID

Type	Entities	Description
`string`	assems points lines surfs solids	For assemblies the name of the NX assembly. For geometrical entities a combination of the body identifiers and the entity ID. Generated when `@UniqueIdAsMetadata = on`

<name>

Type	Entities	Description
`integer`	points	The `<name>` field is the attribute name and the integer/double/string is the value of the attribute.

Type	Entities	Description
`double` `string`	lines surfs solids comps assems	Generated when `@AttributesAsMetadata = on`

NX Part Browser

Use the NX Part Browser to filter and select options for importing NX parts and assemblies. The browser opens when you import an NX file from the Import browser when Creation type is set to **Assemblies**.

On the left of the browser is the model tree. The parts listed in the model tree are shown either using the File name or Instance name options. To select a part import, select its check box. If the root is selected, all parts are considered selected. Root assembly/part is selected by default.

The NX Part Browser searches all part files for weld information as well as mid-surface definitions. Parts in the assembly that have welds assembled to them are marked with . Parts with mid-surface definitions are marked with .

On the right of the browser are the available import options.

Display by

Method used to import parts specified in the model tree.

- Choose **Categories** to import the selected Categories for the parts specified in the model tree. Some categories may be available in multiple parts/assemblies and some may be available in only one. You can choose multiple categories to import. All the categories for the selected parts are imported.
- Choose **Layer filtering** to import only selected layers for the parts specified in the model tree. You can specify specific layers to Enable or Disable for import. To designate more than one layer or a range of layers, use commas and dashes. For example, 1,4-7 designates layers 1, 4, 5, 6 and 7. An asterisk (*) matches everything. Only one of these two options may be specified. If any layers are specified for Disable, all other layers containing entities are imported. The default is to enable all layers.

  You can additionally choose to import the Midsurface geometry and/or the Solid geometry. There are three options for Midsurface import.

  - Choose **Off** (default) to not import any midsurface geometry.
  - Choose **on** to import only the midsurface geometry that is visible.
  - Choose **always** to import all midsurface geometry regardless of visibility.

Include wireframe geometry

Import free curves and points. The default is to import these entities.

Include invisible geometry

Import geometry contained in invisible layers. The default is to not import these entities.

Assign components by name

Customizes the organization and names of components (NX parts). The naming options are specified using the Format option.

- Choose **<Part Name>** to use the string attribute `DB_PART_NAME` from the part. If that attribute does not exist, the part file name is used.
- Choose **<Part Number>** to use the string attribute `DB_PART_NUMBER` or `DB_PART_NO` from the part. If that attribute does not exist, the part filename is used.
- Choose **<Category>** to creates and name components based off of the NX categories associated with the geometry.
- Choose **<RID-PDI>** to use the string attribute `DB_PART_REV` from the part.
- Choose **<Layer>** to create and name components based off of the layers of the geometry being imported.
- Choose **<Instance Name>** to use the name of the part's instance in an assembly. If the part is the root of an assembly, the part filename is used.
- Choose **<Material Name>** to use the name of the material if a material is specified for the geometry being imported. Otherwise, the name of the component is unaltered.
- Choose **<Thickness>** to use the thickness value if the geometry being imported is a midsurface with thickness information. Otherwise, the name of the component is unaltered.
- Choose **<UG Body ID>** to use the internal numerical ID of each geometric body. This value is unique for each geometric body imported.
- Choose **<UG Tag>** to use the internal numerical tag of the part instance. This value is unique for each part instance imported.

Save settings as default

Save the settings into a customized `ug.ini` file in the current working directory for future use.

Access additional options for coordinate system, weld creation, merging assemblies, and importing of attributes by clicking **Options** at the bottom of the browser.

Co-ordinate system

Import an individual part from an assembly in the global coordinate system or its local coordinate system. If the whole assembly file is selected, it can only be imported in the global coordinate system.

Create welds in HyperMesh

Create welds, and organized them in a component named ^weld.

- If the part file has mid-surface definition and the **Midsurface** option is selected while reading the part, the welds are created between two mid-surfaces.
- If there is no mid-surface definition or the **Midsurface** option is not selected while importing the part, the welds are created as they were initially created in the NX part file.

Export to master weld file

Write a master weld file from the weld data that can later be used to create the welds.

Merge Assemblies

Merge parts with the existing assembly if some parts of an assembly have been previously loaded and you want to add additional parts from the same assembly.

Model and component attributes

Select the attributes in the NX part file that are to be imported. Currently, the NX readers have the ability to import any user-defined attributes attached to a NX part, and attach that information as metadata to the component corresponding to the NX part. Specify the naming options by clicking **Format**.

The values setup and used to populate the NX Part Browser are stored in a file named `ughm16.txt`, located in the current working directory.

Load Options

The NX Native reader provides additional customization options for loading of NX models. The `ug_load_options.def` file can be used for this purpose.

Behavior of the `ug_load_options.def` file.

- This file can be setup and written out of NX using the **File** > **Options** > **Assembly Load Options** dialog.
- The NX Native reader passes the options in this file to UGOpen during import, and NX utilizes the options accordingly

A default version of the `ug_load_options.def` file is located in the directory `[Altair Home]/io/afc_translators/bin/[platform]`. When the NX Native reader is activated, it first checks the current working directory for the `ug_load_options.def` file. If the file is not found, the translator uses the `default ug_load_options.def` file in the above directory. In this way the `ug_load_options.def` file can have "global" or "local" user scope. For instance, "local" user changes for a current job can be made by copying and modifying the `ug_load_options.def` file in the local current working directory.

NX Third Party Reader

The NX Third Party reader does not need a valid NX installation or license to access these libraries.

Supported Entities

Entities supported by the NX Third Party reader.

- Points
- All Types of Curves
- Sheet Bodies
- Solid Bodies
- Assemblies
- Coordinate Systems

Import Options

The NX Third Party reader uses the `ct_reader.ini` file.

`@AttributesAsMetadata`

Value	Description
on	Import all generic attributes (global and related to single entities) as metadata.
off	Do not import attributes related to single entities. Only import global attributes.
%attribute1%attribute2%.	Import specified generic attributes as metadata. The attribute names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '%' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '%'. HyperMesh will recognize it as it is the first character of the string.

@AvoidThirdPartyConversion

Value	Description
on	Avoid using the conversion functions provided by third-party (default).
off	Use the conversion functions provided by third-party.

@CleanupTol

Value	Description
double	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@BodyIdAsMetadata

Value	Description
on	Assign body identifier as metadata. BODY_ID
off	Do not assign body identifier as metadata (default).

@ColorsAsMetadata

Value	Description
integer	Read color attributes of geometric entities as metadata with integer values. COLOR_RGB
real	Read color attributes of geometric entities as metadata with real values. COLOR_RGB
on	Same as integer. COLOR_RGB
off	Do not read color attributes (default).

@CreationType

Value	Description
Parts	Generate a full part-based hierarchy.
BOMOnly	Generate an empty part-based hierarchy.
TreeOfComponents	Generate an assemblies/components based hierarchy.
PackIntoSinglePart	Consolidate hierarchy into a single part.

@DensityAsMetadata

Value	Description
on	Read density value as metadata (default). DENSITY
off	Do not read density value.

@DoNotMergeEdges

Value	Description
`on`	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
`off`	Merge edges together during the import cleanup phase (default).

@FullNameAsMetadata

Value	Description
`on`	Generate the full CAD name, as retrieved from the CAD part, as metadata. This consists of assembly name/part name/feature name/entity name. `FULL_IDENTIFIER`
`off`	Do not generate full name metadata (default).

@ImportBlanked

Value	Description
`on`	Import entities that are hidden, blanked, or no show.
`off`	Do not import entities that are hidden, blanked, or no show (default).

@ImportCoordinateSystems

Value	Description
`on`	Import CAD coordinate systems as system collectors (default).
`off`	Do not import CAD coordinate systems.

@ImportForVisualizationOnly

Value	Description
`on`	Import the model for visualization purposes only.

Value	Description
	This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
`off`	Import the model in the normal fashion (default).

`@ImportFreeCurves`

Value	Description
`on`	Import free curves (wireframe entities) into the model (default).
`off`	Do not import free curves.

`@ImportFreePoints`

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

`@LayerAsMetadata`

Value	Description
`on`	Read layer value as metadata. `LAYER`
`off`	Do not read layer value (default).

`@LegacyHierarchyAsMetadata`

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`

Value	Description
off	Do not generate metadata with the original CAD hierarchy within the part (default).

@MaterialName

Value	Description
string	Name of the CAD attribute containing the PDM material name info of the current part. Default is "Material".

@MeshFlag

Value	Description
string	Name of the CAD attribute containing the PDM mesh flag name info of the current part. Default is "MeshFlag".

@MetadataPrefix

Value	Description
string	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@MID

Value	Description
string	Name of the CAD attribute containing the PDM material ID info of the current part. Default is "MID".

@OriginalIdAsMetadata

Value	Description
on	Import original CAD entity IDs as metadata. ORIGINAL_ID
off	Do not import original CAD entity IDs as metadata (default).

@PartNumber

Value	Description
string	Name of the CAD attribute containing the PDM part number info of the current part. Default is "PartNumber”.

@PID

Value	Description
string	Name of the CAD attribute containing PDM property ID info of the current part. Default is "PID”.

@Revision

Value	Description
string	Name of the CAD attribute containing PDM major revision info of the current part. Default is "Revision”.

@ScaleFactor

Value	Description
double	Define the model scaling factor during import. Default is 1.0.

@SkipCreationOfSolid

Value	Description
on	Read surfaces, but do not create solid entities.
off	Create solid entities (default).

@SplitComponents

Value	Description
Body	Generate body-based component (default if CreationType=Parts).
Layer	Generate layer-based component.
Part	Generate part-based component.
General	Keep component as in CAD (default if CreationType=TreeOfComponents).

@SplitPeriodicFaces

Value	Description
on	Split periodic surfaces to improve the quality and robustness of the import (default).
off	Import periodic surfaces as a single surface with a seam edge.

@StitchingAcrossBodies

Value	Description
on	Stitch surfaces belonging to different components.
off	Do not stitch surfaces belonging to different components (default).

@TagsAsMetadata

Value	Description
on	Create entity name as metadata (default).

Value	Description
	`TAG`
`FromBody`	Take entity from its parent body and create as metadata. `TAG`
`off`	Do not read tags.

@TargetUnits

Value	Description
`CAD units`	Keep the units of the CAD files (default for GUI).
`MKS [m kg N s]`	Convert to the corresponding units system.
`MMKS [mm kg N s]`	Convert to the corresponding units system.
`MPA [mm t N s]`	Convert to the corresponding units system.
`CGS [cm g dyn s]`	Convert to the corresponding units system.
`MMKNMS [mm kg kN ms]`	Convert to the corresponding units system.
`MMGNMS [mm g N ms]`	Convert to the corresponding units system.
`IPS Std [in pounds lbf s]`	Convert to the corresponding units system.
`IPS Grav [in slinch lbf s]`	Convert to the corresponding units system.
`FPS Std [ft pounds lbf s]`	Convert to the corresponding units system.
`FPS Grav [ft slug lbf s]`	Convert to the corresponding units system.
`Scale factor`	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

@ThicknessName

Value	Description
`string`	Name of the CAD attribute containing the PDM thickness info of the current part. Default is "Thickness".

@UID

Value	Description
`string`	Name of the CAD attribute containing the PDM UID info of the current part. Default is "UID".

Supported Metadata

Metadata generated from the NX Third Party reader.

BODY_ID

Type	Entities	Description
`string`	points lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

COLOR_RGB

Type	Entities	Description
`string`	points lines surfs solids	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when `@ColorsAsMetadata = on`

DENSITY

Type	Entities	Description
`double`	solids	The value of the density of a solid. Generated when `@DensityAsMetadata = on`

FULL_IDENTIFIER

Type	Entities	Description
`string`	points lines surfs solids comps assems	A string indicating the name in the following format: `"part_name/name"` Generated when `@FullNameAsMetadata = on`

LAYER

Type	Entities	Description
`integer`	points lines surfs solids	Value of the layer of an entity. Generated when `@LayerAsMetadata=on`

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles 6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

ORIGINAL_ID

Type	Entities	Description
`string`	points lines surfs solids parts	The original CAD entity ID. Generated when `@OriginalIdAsMetadata = on`

TAG

Type	Entities	Description
`string`	points lines edges surfs	The tag (name) of the entity as read from the CAD model, if one exists. Generated when `@TagsAsMetadata = on`

Type	Entities	Description
	solids	

Parasolid Reader

Supported Entities

Entities supported by the Parasolid reader.

- Free points
- Free curves
- Surfaces
- Quilt bodies
- Solid bodies
- Assemblies

Import Options

The Parasolid reader uses the `parasolid_reader.ini` file.

`@AttributesAsMetadata`

Value	Description
`on`	Import all generic attributes (global and related to single entities) as metadata.
`off`	Do not import attributes related to single entities. Only import global attributes.
`%attribute1%attribute2%`	Import specified generic attributes as metadata. The attribute names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '%' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '%'. HyperMesh will recognize it as it is the first character of the string.

`@AvoidDisjointShells`

Value	Description
`on`	Stitch disjoint sheet body surfaces together (default).
`off`	Do not stitch disjoint sheet body surfaces together.

`@BodyIdAsMetadata`

Value	Description
`on`	Assign body identifier as metadata. `BODY_ID`
`off`	Do not assign body identifier as metadata (default).

`@CleanupTol`

Value	Description
`double`	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

`@ColorsAsMetadata`

Value	Description
`integer`	Read color attributes of geometric entities as metadata with integer values. `COLOR_RGB`
`on`	Read color attributes of geometric entities as metadata. `COLOR_RGB`
`off`	Do not read color attributes (default).

`@CreationType`

Value	Description
`Parts`	Generate a full part-based hierarchy.
`BOMOnly`	Generate an empty part-based hierarchy.
`TreeOfComponents`	Generate an assemblies/components based hierarchy.
`PackIntoSinglePart`	Consolidate hierarchy into a single part.

`@DensityAsMetadata`

Value	Description
`on`	Read density value as metadata (default). `DENSITY`
`off`	Do not read density value.

`@DoNotMergeEdges`

Value	Description
`on`	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
`off`	Merge edges together during the import cleanup phase (default).

`@ExtractManifoldFromGeneralBody`

Value	Description
`on`	Extract the manifold body from a Parasolid general body that is non-manifold and/or of mixed topological dimensionality (default).
`off`	Extract Parasolid general bodies as defined.

`@ImportBlanked`

Value	Description
`on`	Import entities that are hidden, blanked, or no show.

Value	Description
`off`	Do not import entities that are hidden, blanked, or no show (default).

`@ImportForVisualizationOnly`

Value	Description
`on`	Import the model for visualization purposes only. This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
`off`	Import the model in the normal fashion (default).

`@ImportFreeCurves`

Value	Description
`on`	Import free curves (wireframe entities) into the model (default).
`off`	Do not import free curves.

`@ImportFreePoints`

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

`@LegacyHierarchyAsMetadata`

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`

Value	Description
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

`@MaterialName`

Value	Description
`string`	Name of the CAD attribute containing the PDM material name info of the current part. Default is "Material".

`@MetadataPrefix`

Value	Description
`string`	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

`@OriginalIdAsMetadata`

Value	Description
`on`	Import original CAD entity IDs as metadata. `ORIGINAL_ID`
`off`	Do not import original CAD entity IDs as metadata (default).

`@PartNumber`

Value	Description
`string`	Name of the CAD attribute containing the PDM part number info of the current part. Default is "PartNumber”.

`@ScaleFactor`

Value	Description
double	Define the model scaling factor during import. Default is 1.0.

@SkipCreationOfSolid

Value	Description
on	Read surfaces, but do not create solid entities.
off	Create solid entities (default).

@SplitComponents

Value	Description
Body	Generate body-based component (default if CreationType=Parts).
Part	Generate part-based component.
General	Keep component as in CAD (default if CreationType=TreeOfComponents).

@SplitComponentsByBody

Value	Description
on	Create one component per third-party body. There might be one or more third-party bodies per third-party component. This option cannot be used when @ReadCompositeData is enabled, or in conjunction with @SplitComponentsByPart.
off	Do not consider the division of third-party components into third-party bodies while creating components (default).

@SplitPeriodicFaces

Value	Description
on	Split periodic surfaces to improve the quality and robustness of the import (default).

Value	Description
off	Import periodic surfaces as a single surface with a seam edge.

@StitchingAcrossBodies

Value	Description
on	Stitch surfaces belonging to different components.
off	Do not stitch surfaces belonging to different components (default).

@TagsAsMetadata

Value	Description
on	Create entity name as metadata (default). TAG
FromBody	Take entity from its parent body and create as metadata. TAG
off	Do not read tags.

@TargetUnits

Value	Description
CAD units	Keep the units of the CAD files (default for GUI).
MKS [m kg N s]	Convert to the corresponding units system.
MMKS [mm kg N s]	Convert to the corresponding units system.
MPA [mm t N s]	Convert to the corresponding units system.
CGS [cm g dyn s]	Convert to the corresponding units system.
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.

Value	Description
`IPS Std [in pounds lbf s]`	Convert to the corresponding units system.
`IPS Grav [in slinch lbf s]`	Convert to the corresponding units system.
`FPS Std [ft pounds lbf s]`	Convert to the corresponding units system.
`FPS Grav [ft slug lbf s]`	Convert to the corresponding units system.
`Scale factor`	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

@ThicknessName

Value	Description
`string`	Name of the CAD attribute containing the PDM thickness info of the current part. Default is "Thickness".

@UID

Value	Description
`string`	Name of the CAD attribute containing the PDM UID info of the current part. Default is "UID".

@UniqueIdAsMetadata

Value	Description
`on`	Create metadata to track the unique CAD ID (default). When a Parasolid body is present in the Parasolid assembly, the Parasolid instance ID is appended to the `ORIGINAL_ID` using "-". For each assembly level, its instance ID will be appended in the same way. For example:

Value	**Description**
	No body entities present in the assembly: 20 Body entities present in a single level assembly: 1-20 Body entities present in a multiple level assembly: 1-4-2-20 The same identifier can be obtained for two entities when a single entity is split during import. `UNIQUE_ID`
`off`	Do not create metadata to track the unique CAD ID.

Supported Metadata

Metadata generated from the Parasolid reader.

`BODY_ID`

Type	**Entities**	**Description**
`string`	points lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

`COLOR_RGB`

Type	**Entities**	**Description**
`string`	points lines surfs solids	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when `@ColorsAsMetadata = on`

`DENSITY`

Type	**Entities**	**Description**
`double`	solids	The value of the density of a solid.

Type	Entities	Description
		Generated when `@DensityAsMetadata = on`

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles 6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

<name>

Type	Entities	Description
`integer` `double`	points lines	The `<name>` field is the attribute name and the integer/double/string is the value of the attribute.

Type	Entities	Description
`string`	surfs solids comps assems	Generated when `@AttributesAsMetadata = on`

`ORIGINAL_ID`

Type	Entities	Description
`ORIGINAL_ID`	points lines surfs solids parts	The original CAD entity ID. Generated when `@OriginalIdAsMetadata = on`

`TAG`

Type	Entities	Description
`string`	points lines surfs solids	The tag (name) of the entity as read from the CAD model, if one exists. Generated when `@TagsAsMetadata = on`

`UNIQUE_ID`

Type	Entities	Description
`string`	lines solids	A combination of the body identifiers and the Parasolid entity ID. Generated when `@UniqueIdAsMetadata = on`

Type	Entities	Description
	surfs	

PDGS Reader

Supported Entities

Entities supported by the PDGS reader.

- Entity #5

Import Options

The PDGS reader uses the `pdgs_reader.ini` file.

@CleanupTol

Value	Description
double	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
integer	Read color attributes of geometric entities as metadata with integer values. COLOR_RGB
on	Read color attributes of geometric entities as metadata. COLOR_RGB
off	Do not read color attributes (default).

@CreationType

Value	Description
Parts	Generate a full part-based hierarchy.

Value	Description
`BOMOnly`	Generate an empty part-based hierarchy.
`TreeOfComponents`	Generate an assemblies/components based hierarchy.
`PackIntoSinglePart`	Consolidate hierarchy into a single part.

`@LegacyHierarchyAsMetadata`

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

`@MetadataPrefix`

Value	Description
`string`	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

`@ScaleFactor`

Value	Description
`double`	Define the model scaling factor during import. Default is 1.0.

`@SplitComponents`

Value	Description
`Part`	Generate part-based component (only allowed value if `CreationType=Parts`).

Value	Description
`General`	Keep component as in CAD (only allowed value if `CreationType=TreeOfComponents`).

@TargetUnits

Value	Description
`CAD units`	Keep the units of the CAD files (default for GUI).
`MKS [m kg N s]`	Convert to the corresponding units system.
`MMKS [mm kg N s]`	Convert to the corresponding units system.
`MPA [mm t N s]`	Convert to the corresponding units system.
`CGS [cm g dyn s]`	Convert to the corresponding units system.
`MMKNMS [mm kg kN ms]`	Convert to the corresponding units system.
`MMGNMS [mm g N ms]`	Convert to the corresponding units system.
`IPS Std [in pounds lbf s]`	Convert to the corresponding units system.
`IPS Grav [in slinch lbf s]`	Convert to the corresponding units system.
`FPS Std [ft pounds lbf s]`	Convert to the corresponding units system.
`FPS Grav [ft slug lbf s]`	Convert to the corresponding units system.
`Scale factor`	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

Supported Metadata

Metadata generated from the PDGS reader.

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines	Hierarchy of an entity within a part.

Type	Entities	Description
	surfs	Generated when `@LegacyHierarchyAsMetadata=on`

Rhino Reader

Supported Entities

Entities supported by the Rhino reader.

- B-rep geometry

Import Options

The Rhino reader uses the `ct_reader.ini` file.

@AttributesAsMetadata

Value	Description
`on`	Import all generic attributes (global and related to single entities) as metadata.
`off`	Do not import attributes related to single entities. Only import global attributes.
`%attribute1%attribute2%.`	Import specified generic attributes as metadata. The attribute names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '%' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '%'. HyperMesh will recognize it as it is the first character of the string.

@AvoidThirdPartyConversion

Value	Description
`on`	Avoid using the conversion functions provided by third-party (default).
`off`	Use the conversion functions provided by third-party.

@BodyIdAsMetadata

Value	Description
on	Assign body identifier as metadata. BODY_ID
off	Do not assign body identifier as metadata (default).

@CleanupTol

Value	Description
double	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
integer	Read color attributes of geometric entities as metadata with integer values. COLOR_RGB
on	Read color attributes of geometric entities as metadata. COLOR_RGB
off	Do not read color attributes (default).

@CreationType

Value	Description
Parts	Generate a full part-based hierarchy.
BOMOnly	Generate an empty part-based hierarchy.
TreeOfComponents	Generate an assemblies/components based hierarchy.

Value	Description
PackIntoSinglePart	Consolidate hierarchy into a single part.

@DensityAsMetadata

Value	Description
on	Read density value as metadata (default). DENSITY
off	Do not read density value.

@DoNotMergeEdges

Value	Description
on	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
off	Merge edges together during the import cleanup phase (default).

@ImportBlanked

Value	Description
on	Import entities that are hidden, blanked, or no show.
off	Do not import entities that are hidden, blanked, or no show (default).

@ImportCoordinateSystems

Value	Description
on	Import CAD coordinate systems as system collectors (default).
off	Do not import CAD coordinate systems.

@ImportForVisualizationOnly

Value	Description
`on`	Import the model for visualization purposes only. This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
`off`	Import the model in the normal fashion (default).

@ImportFreeCurves

Value	Description
`on`	Import free curves (wireframe entities) into the model (default).
`off`	Do not import free curves.

@ImportFreePoints

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

@LayerAsMetadata

Value	Description
`on`	Read layer value as metadata. `LAYER`
`off`	Do not read layer value (default).

@LegacyHierarchyAsMetadata

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part.

Value	Description
	LEGACY_HIERARCHY
off	Do not generate metadata with the original CAD hierarchy within the part (default).

@MetadataPrefix

Value	Description
string	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@OriginalIdAsMetadata

Value	Description
on	Import original CAD entity IDs as metadata. ORIGINAL_ID
off	Do not import original CAD entity IDs as metadata (default).

@ScaleFactor

Value	Description
double	Define the model scaling factor during import. Default is 1.0.

@ShrinkSurfaces

Value	Description
on	For trimmed surfaces, shrink the corresponding base surfaces to only enclose the extents of the external trimming contour, plus a narrow frame.
off	Do not modify the base surfaces (default).

@SkipCreationOfSolid

Value	Description
on	Read surfaces, but do not create solid entities.
off	Create solid entities (default).

@SplitComponents

Value	Description
Body	Generate body-based component (default if CreationType=Parts).
Layer	Generate layer-based component.
Part	Generate part-based component.
General	Keep component as in CAD (default if CreationType=TreeOfComponents).

@StitchingAcrossBodies

Value	Description
on	Stitch surfaces belonging to different components.
off	Do not stitch surfaces belonging to different components (default).

@TargetUnits

Value	Description
CAD units	Keep the units of the CAD files (default for GUI).
MKS [m kg N s]	Convert to the corresponding units system.
MMKS [mm kg N s]	Convert to the corresponding units system.
MPA [mm t N s]	Convert to the corresponding units system.
CGS [cm g dyn s]	Convert to the corresponding units system.
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.

Value	**Description**
`IPS Std [in pounds lbf s]`	Convert to the corresponding units system.
`IPS Grav [in slinch lbf s]`	Convert to the corresponding units system.
`FPS Std [ft pounds lbf s]`	Convert to the corresponding units system.
`FPS Grav [ft slug lbf s]`	Convert to the corresponding units system.
`Scale factor`	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

Supported Metadata

Metadata generated from the Rhino reader.

BODY_ID

Type	**Entities**	**Description**
`string`	points lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

COLOR_RGB

Type	**Entities**	**Description**
`string`	points lines surfs solids	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when `@ColorsAsMetadata = on`

DENSITY

Type	Entities	Description
`double`	solids	The value of the density of a solid. Generated when `@DensityAsMetadata = on`

LAYER

Type	Entities	Description
`integer`	points lines surfs solids	Value of the layer of an entity. Generated when `@LayerAsMetadata=on`

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles 6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches

Type	Entities	Description
		12 = decimeters 13 = yards This is always generated.

ORIGINAL_ID

Type	Entities	Description
ORIGINAL_ID	points lines surfs solids parts	The original CAD entity ID. Generated when @OriginalIdAsMetadata = on

<name>

Type	Entities	Description
integer double string	points lines surfs solids comps assems	The <name> field is the attribute name and the integer/double/string is the value of the attribute. Generated when @AttributesAsMetadata = on

SolidWorks Reader

Supported Entities

Entities supported by the SolidWorks reader.

- Free points
- Free curves
- Surfaces

- Solid bodies
- Assemblies

Entities with non-latin characters in their names are supported.

Import Options

The SolidWorks reader uses the `ct_reader.ini` file.

@AttributesAsMetadata

Value	Description
on	Import all generic attributes (global and related to single entities) as metadata.
off	Do not import attributes related to single entities. Only import global attributes.
%attribute1%attribute2%	Import specified generic attributes as metadata. The attribute names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '%' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '%'. HyperMesh will recognize it as it is the first character of the string.

@AvoidThirdPartyConversion

Value	Description
on	Avoid using the conversion functions provided by third-party (default).
off	Use the conversion functions provided by third-party.

@BodyIdAsMetadata

Value	Description
on	Assign body identifier as metadata. BODY_ID
off	Do not assign body identifier as metadata (default).

@CleanupTol

Value	Description
double	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
integer	Read color attributes of geometric entities as metadata with integer values. COLOR_RGB
on	Same as integer.
off	Do not read color attributes (default).

@CreationType

Value	Description
Parts	Generate a full part-based hierarchy.
BOMOnly	Generate an empty part-based hierarchy.
TreeOfComponents	Generate an assemblies/components based hierarchy.
PackIntoSinglePart	Consolidate hierarchy into a single part.

@DensityAsMetadata

Value	Description
on	Read density value as metadata (default). DENSITY
off	Do not read density value.

@DoNotMergeEdges

Value	Description
on	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
off	Merge edges together during the import cleanup phase (default).

@ImportBlanked

Value	Description
on	Import entities that are hidden, blanked, or no show.
off	Do not import entities that are hidden, blanked, or no show (default).

@ImportCoordinateSystems

Value	Description
on	Import CAD coordinate systems as system collectors (default).
off	Do not import CAD coordinate systems.

@ImportForVisualizationOnly

Value	Description
on	Import the model for visualization purposes only. This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
off	Import the model in the normal fashion (default).

@ImportFreeCurves

Value	Description
on	Import free curves (wireframe entities) into the model (default).

Value	Description
`off`	Do not import free curves.

@ImportFreePoints

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

@LayerAsMetadata

Value	Description
`on`	Read layer value as metadata. `LAYER`
`off`	Do not read layer value (default).

@LegacyHierarchyAsMetadata

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

@MaterialName

Value	Description
`string`	Name of the CAD attribute containing the PDM material name info of the current part. Default is "Material".

@MeshFlag

Value	Description
string	Name of the CAD attribute containing the PDM mesh flag name info of the current part. Default is "MeshFlag".

@MetadataPrefix

Value	Description
string	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@MID

Value	Description
string	Name of the CAD attribute containing the PDM material ID info of the current part. Default is "MID".

@OriginalIdAsMetadata

Value	Description
on	Import original CAD entity IDs as metadata. `ORIGINAL_ID`
off	Do not import original CAD entity IDs as metadata (default).

@PartNumber

Value	Description
string	Name of the CAD attribute containing the PDM part number info of the current part.

Value	Description
	Default is "PartNumber”.

@PID

Value	Description
`string`	Name of the CAD attribute containing PDM property ID info of the current part. Default is "PID”.

@Revision

Value	Description
`string`	Name of the CAD attribute containing PDM major revision info of the current part. Default is "Revision”.

@ScaleFactor

Value	Description
`double`	Define the model scaling factor during import. Default is 1.0.

@SkipCreationOfSolid

Value	Description
`on`	Read surfaces, but do not create solid entities.
`off`	Create solid entities (default).

@SplitComponents

Value	Description
`Body`	Generate body-based component (default if `CreationType=Parts`).
`Layer`	Generate layer-based component.
`Part`	Generate part-based component.
`General`	Keep component as in CAD (default if `CreationType=TreeOfComponents`).

@SplitComponentsByBody

Value	Description
`on`	Create one component per third-party body. There might be one or more third-party bodies per third-party component.
`off`	Do not consider the division of third-party components into third-party bodies while creating components (default).

@SplitPeriodicFaces

Value	Description
`on`	Split periodic surfaces to improve the quality and robustness of the import (default).
`off`	Import periodic surfaces as a single surface with a seam edge.

@StitchingAcrossBodies

Value	Description
`on`	Stitch surfaces belonging to different components.
`off`	Do not stitch surfaces belonging to different components (default).

@TargetUnits

Value	Description
`CAD units`	Keep the units of the CAD files (default for GUI).

Value	Description
MKS [m kg N s]	Convert to the corresponding units system.
MMKS [mm kg N s]	Convert to the corresponding units system.
MPA [mm t N s]	Convert to the corresponding units system.
CGS [cm g dyn s]	Convert to the corresponding units system.
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.
IPS Std [in pounds lbf s]	Convert to the corresponding units system.
IPS Grav [in slinch lbf s]	Convert to the corresponding units system.
FPS Std [ft pounds lbf s]	Convert to the corresponding units system.
FPS Grav [ft slug lbf s]	Convert to the corresponding units system.
Scale factor	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

@ThicknessName

Value	Description
string	Name of the CAD attribute containing the PDM thickness info of the current part. Default is "Thickness".

@UID

Value	Description
string	Name of the CAD attribute containing the PDM UID info of the current part.

Value	Description
	Default is "UID".

Supported Metadata

Metadata generated from the SolidWorks reader.

BODY_ID

Type	Entities	Description
`string`	points lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

COLOR_RGB

Type	Entities	Description
`string`	points lines surfs solids	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when `@ColorsAsMetadata = on`

DENSITY

Type	Entities	Description
`double`	solids	The value of the density of a solid. Generated when `@DensityAsMetadata = on`

LAYER

Type	Entities	Description
`integer`	points lines	Value of the layer of an entity.

Type	Entities	Description
	surfs solids	Generated when @LayerAsMetadata=on

LEGACY_HIERARCHY

Type	Entities	Description
string	points lines surfs	Hierarchy of an entity within a part. Generated when @LegacyHierarchyAsMetadata=on

MODELUNIT

Type	Entities	Description
integer	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles 6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

ORIGINAL_ID

Type	Entities	Description
string	points	The original CAD entity ID.

Type	Entities	Description
	lines surfs solids parts	Generated when `@OriginalIdAsMetadata = on`

TAG

Type	Entities	Description
`string`	points lines edges surfs solids	The tag (name) of the entity as read from the CAD model, if one exists. Generated when `@TagsAsMetadata = on`

STEP Reader

Supported Entities

Entities supported by the STEP reader.

- Coordinate systems
- Free points
- Free curves
- Surfaces
- Quilt bodies
- Solid bodies
- Facets/triangles
- Assemblies

Entities with non-latin characters in their names are supported.

Import Options

The STEP reader uses the `ct_reader.ini` file.

@AttributesAsMetadata

Value	Description
`on`	Import all generic attributes (global and related to single entities) as metadata.
`off`	Do not import attributes related to single entities. Only import global attributes.
`%attribute1%attribute2%`	Import specified generic attributes as metadata. The attribute names are listed, separated by a character that you choose, and inserted as the first element of the string. The example uses '%' as a separator. You can choose another character as a separator, in case one of the listed attribute names contains '%'. HyperMesh will recognize it as it is the first character of the string.

`@AvoidThirdPartyConversion`

Value	Description
`on`	Avoid using the conversion functions provided by third-party (default).
`off`	Use the conversion functions provided by third-party.

`@BodyIdAsMetadata`

Value	Description
`on`	Assign body identifier as metadata. `BODY_ID`
`off`	Do not assign body identifier as metadata (default).

`@CleanupTol`

Value	Description
`double`	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@ColorsAsMetadata

Value	Description
integer	Read color attributes of geometric entities as metadata with integer values. COLOR_RGB
real	Read color attributes of geometric entities as metadata with real values. COLOR_RGB
on	Same as integer. COLOR_RGB
off	Do not read color attributes (default).

@CreationType

Value	Description
Parts	Generate a full part-based hierarchy.
BOMOnly	Generate an empty part-based hierarchy.
TreeOfComponents	Generate an assemblies/components based hierarchy.
PackIntoSinglePart	Consolidate hierarchy into a single part.

@DensityAsMetadata

Value	Description
on	Read density value as metadata (default). DENSITY
off	Do not read density value.

@DoNotMergeEdges

Value	Description
on	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
off	Merge edges together during the import cleanup phase (default).

@FullNameAsMetadata

Value	Description
on	Generate the full CAD name, as retrieved from the CAD part, as metadata. This consists of assembly name/part name/feature name/entity name. `FULL_IDENTIFIER`
off	Do not generate full name metadata (default).

@ImportBlanked

Value	Description
on	Import entities that are hidden, blanked, or no show.
off	Do not import entities that are hidden, blanked, or no show (default).

@ImportCoordinateSystems

Value	Description
on	Import CAD coordinate systems as system collectors (default).
off	Do not import CAD coordinate systems.

@ImportForVisualizationOnly

Value	Description
on	Import the model for visualization purposes only.

Value	Description
	This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
`off`	Import the model in the normal fashion (default).

`@ImportFreeCurves`

Value	Description
`on`	Import free curves (wireframe entities) into the model (default).
`off`	Do not import free curves.

`@ImportFreePoints`

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

`@LayerAsMetadata`

Value	Description
`on`	Read layer value as metadata. `LAYER`
`off`	Do not read layer value (default).

`@LegacyHierarchyAsMetadata`

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`

Value	Description
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

@MaterialName

Value	Description
`string`	Name of the CAD attribute containing the PDM material name info of the current part. Default is "Material".

@MeshFlag

Value	Description
`string`	Name of the CAD attribute containing the PDM mesh flag name info of the current part. Default is "MeshFlag".

@MetadataPrefix

Value	Description
`string`	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@MID

Value	Description
`string`	Name of the CAD attribute containing the PDM material ID info of the current part. Default is "MID".

@OriginalIdAsMetadata

Value	Description
`on`	Import original CAD entity IDs as metadata. `ORIGINAL_ID`
`off`	Do not import original CAD entity IDs as metadata (default).

`@PartNumber`

Value	Description
`string`	Name of the CAD attribute containing the PDM part number info of the current part. Default is "PartNumber”.

`@PID`

Value	Description
`string`	Name of the CAD attribute containing PDM property ID info of the current part. Default is "PID”.

`@Revision`

Value	Description
`string`	Name of the CAD attribute containing PDM major revision info of the current part. Default is "Revision”.

`@ScaleFactor`

Value	Description
`double`	Define the model scaling factor during import. Default is 1.0.

@ShrinkSurfaces

Value	Description
on	For trimmed surfaces, shrink the corresponding base surfaces to only enclose the extents of the external trimming contour, plus a narrow frame.
off	Do not modify the base surfaces (default).

@SkipCreationOfSolid

Value	Description
on	Read surfaces, but do not create solid entities.
off	Create solid entities (default).

@SplitComponents

Value	Description
Body	Generate body-based component (default if CreationType=Parts).
Layer	Generate layer-based component.
Part	Generate part-based component.
General	Keep component as in CAD (default if CreationType=TreeOfComponents).

@SplitComponentsByBody

Value	Description
on	Create one component per third-party body. There might be one or more third-party bodies per third-party component. This option cannot be used when @ReadCompositeData is enabled, or in conjunction with @SplitComponentsByPart.
off	Do not consider the division of third-party components into third-party bodies while creating components (default).

@StitchingAcrossBodies

Value	Description
on	Stitch surfaces belonging to different components.
off	Do not stitch surfaces belonging to different components (default).

@TagsAsMetadata

Value	Description
on	Create entity name as metadata (default). TAG
FromBody	Take entity from its parent body and create as metadata. TAG
off	Do not read tags.

@TargetUnits

Value	Description
CAD units	Keep the units of the CAD files (default for GUI).
MKS [m kg N s]	Convert to the corresponding units system.
MMKS [mm kg N s]	Convert to the corresponding units system.
MPA [mm t N s]	Convert to the corresponding units system.
CGS [cm g dyn s]	Convert to the corresponding units system.
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.
IPS Std [in pounds lbf s]	Convert to the corresponding units system.
IPS Grav [in slinch lbf s]	Convert to the corresponding units system.
FPS Std [ft pounds lbf s]	Convert to the corresponding units system.

Value	Description
`FPS Grav [ft slug lbf s]`	Convert to the corresponding units system.
`Scale factor`	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

@ThicknessName

Value	Description
`string`	Name of the CAD attribute containing the PDM thickness info of the current part. Default is "Thickness".

@UID

Value	Description
`string`	Name of the CAD attribute containing the PDM UID info of the current part. Default is "UID".

Supported Metadata

Metadata generated from the STEP reader.

BODY_ID

Type	Entities	Description
`string`	points lines surfs	Identifier of the CAD body containing the entity. Generated when `@BodyIDAsMetadata=on`

COLOR_RGB

Type	Entities	Description
`string`	points lines surfs solids	Three RGB values, ranging from 0 to 255, indicating the color of the entity in the CAD model. Generated when `@ColorsAsMetadata = on`

DENSITY

Type	Entities	Description
`double`	solids	The value of the density of a solid. Generated when `@DensityAsMetadata = on`

FULL_IDENTIFIER

Type	Entities	Description
`string`	points lines surfs solids comps assems	A string indicating the name in the following format: `"part_name/name"` Generated when `@FullNameAsMetadata = on`

LAYER

Type	Entities	Description
`integer`	points lines surfs	Value of the layer of an entity. Generated when `@LayerAsMetadata=on`

Type	Entities	Description
	solids	

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

MODELUNIT

Type	Entities	Description
`integer`	comps parts	Model units specified in the CAD file. 1 = inches 2 = millimeters 4 = feet 5 = miles 6 = meters 7 = kilometers 8 = mils 9 = microns 10 = centimeters 11 = microinches 12 = decimeters 13 = yards This is always generated.

ORIGINAL_ID

Type	Entities	Description
`string`	points lines	The original CAD entity ID. Generated when `@OriginalIdAsMetadata = on`

Type	Entities	Description
	surfs solids parts	

TAG

Type	Entities	Description
`string`	points lines edges surfs solids	The tag (name) of the entity as read from the CAD model, if one exists. Generated when `@TagsAsMetadata = on`

VDAFS Reader

Supported Entities

Entities supported by the VDAFS reader.

- POINT
- LINE
- PLANE
- PSET
- MDI
- CIRCLE
- CURVE
- SURF
- FACE

Import Options

The VDAFS reader uses the `vdafs_reader.ini` file.

@BreakAngle

Value	Description
`double`	A negative value (default) indicates that the curves are being split at C1 discontinuities. A positive value indicates that curves are being split at G1 discontinuities with an angle greater than the given value (expressed in radians).

@CleanupTol

Value	Description
`double`	A negative value (default) indicates to use the auto cleanup tolerance. A value greater than the calculated auto cleanup tolerance indicates to use that specific tolerance. Refer to the CAD Cleanup Tolerance.

@CreationType

Value	Description
`Parts`	Generate a full part-based hierarchy.
`BOMOnly`	Generate an empty part-based hierarchy.
`TreeOfComponents`	Generate an assemblies/components based hierarchy.
`PackIntoSinglePart`	Consolidate hierarchy into a single part.

@DoNotMergeEdges

Value	Description
`on`	Preserve the original geometry edges, instead of merging them together during the import cleanup phase.
`off`	Merge edges together during the import cleanup phase (default).

@ImportForVisualizationOnly

Value	Description
`on`	Import the model for visualization purposes only.

Value	Description
	This skips many of the import steps (cleanup, stitching, solid creation, and so on) to provide a faster import. The resulting model may not be suitable for other uses.
`off`	Import the model in the normal fashion (default).

@ImportFreeCurves

Value	Description
`on`	Import free curves (wireframe entities) into the model (default).
`off`	Do not import free curves.

@ImportFreePoints

Value	Description
`on`	Import free points into the model (default).
`off`	Do not import free points.

@LegacyHierarchyAsMetadata

Value	Description
`on`	Generate metadata with the original CAD hierarchy within the part. `LEGACY_HIERARCHY`
`off`	Do not generate metadata with the original CAD hierarchy within the part (default).

@MetadataPrefix

Value	Description
`string`	The string is prefixed to all metadata names. No prefix is used by default. See CAD Metadata Naming for more details.

@ParameterTolerance

Value	Description
`Any positive real number.`	The reader considers this tolerance as the parametric tolerance used to process the CAD data. The default value is 1.00E-06.

@Planes

Value	Description
`on`	Treat each surface as found in the CAD file (planes/NURBS).
`off`	Convert planes into NURBS surfaces.
`preferred`	Convert NURBS surfaces to planes if they are within the object space tolerance of being planar (default).

@ScaleFactor

Value	Description
`double`	Define the model scaling factor during import. Default is 1.0.

@SplitComponents

Value	Description
`Body`	Generate body-based component (default if `CreationType=Parts`).
`Part`	Generate part-based component.
`General`	Keep component as in CAD (default if `CreationType=TreeOfComponents`).

@StraightPolynomials

Value	Description
`on`	Treat each polynomial segment as a straight line segment. This is necessary for reading some VDAFS files, COMPUTERVISION CADDS 4X in particular.

Value	Description
off	Treat each polynomial as found in the CAD file (default).

@TagsAsMetadata

Value	Description
on	Read tags of supported entities as metadata (default). TAG
off	Do not read tags.

@TargetUnits

Value	Description
CAD units	Keep the units of the CAD files (default for GUI).
MKS [m kg N s]	Convert to the corresponding units system.
MMKS [mm kg N s]	Convert to the corresponding units system.
MPA [mm t N s]	Convert to the corresponding units system.
CGS [cm g dyn s]	Convert to the corresponding units system.
MMKNMS [mm kg kN ms]	Convert to the corresponding units system.
MMGNMS [mm g N ms]	Convert to the corresponding units system.
IPS Std [in pounds lbf s]	Convert to the corresponding units system.
IPS Grav [in slinch lbf s]	Convert to the corresponding units system.
FPS Std [ft pounds lbf s]	Convert to the corresponding units system.
FPS Grav [ft slug lbf s]	Convert to the corresponding units system.
Scale factor	Apply the corresponding scale factor (default for no GUI to allow supporting legacy scripts).

@Tolerance

Value	Description
`Any positive real number.`	The reader considers this tolerance as the object space tolerance used to process the CAD data. The default value is 0.01.

Supported Metadata

Metadata generated from the VDAFS reader.

LEGACY_HIERARCHY

Type	Entities	Description
`string`	points lines surfs	Hierarchy of an entity within a part. Generated when `@LegacyHierarchyAsMetadata=on`

TAG

Type	Entities	Description
`string`	points lines surfs	The tag (name) of the entity as read from the CAD model, if one exists. Generated when `@TagsAsMetadata = on`

CAD Import Options

CAD readers provide options for processing data during import.

You can access some of these options from the Import browser, while other options are accessed from a reader's `_reader.ini` file.

Hierarchy as

Type of hierarchy to generate.

- Choose **Assemblies** to generate an assemblies/components based hierarchy.
- Choose **BOM Only** to generate an empty part-based hierarchy.
- Choose **Parts** to generate a full part-based hierarchy.
- Choose **Single part only** to consolidate the hierarchy into a single part.

Split components by

Strategy used to split components, which is dependent on the CAD format.

- Choose **Body** to generate a body-based component.
- Choose **Layer** to generate a layer-based component.
- Choose **Part** to generate a part-based component.

Target units

Target unit system to be adopted for the imported model.

- Choose a specific unit.
- Choose **CAD units** to match the unit system to the CAD file being imported. If the target unit system does not match the CAD file being imported, than the corresponding numbers are modified to account for the unit change.
- Alternatively, choose **Scale Factor** to adopt the CAD unit system and scale the entities by the indicated factor.

Cleanup to

Refer to CAD Cleanup Tolerance.

Import hidden (blanked/no show) entities

Import entities that are hidden, blanked, or no show for relevant formats.
See each format's available import options for details. Not all CAD formats supported.

Import composite data

Read Composite Part Design (CPD) data from CATIA files into HyperMesh.

> **Restriction:** Only available in CATIA format.

Read hidden ply base surfaces

Read hidden ply base surfaces.

> **Restriction:** Only available when Import composite data is enabled for the CATIA format.

Trim ply surfaces

Trim ply base surfaces during import.

> **Restriction:** Only available when Import composite data is enabled for the CATIA format.

Ply contour gap tolerance

Ply contour gap tolerance (default 0.3).

> **Restriction:** Only available when Import composite data is enabled for the CATIA format.

Import publication data

Create regions from the geometry referenced in the publication.

> **Restriction:** Only available in CATIA format.

Do not merge edges

Preserve the original breakdown of edges.

By default edges are merged together, when possible, during import as part of the cleanup phase.

Global Analysis

Create just landing curves for stiffener/beam/ER systems, unless the ini option `@ImportStiffenerMidSurfaces` is "`on`" in which case both landing curves and stiffener/beam/ER geometry midsurfaces will be created.

When disabled, stiffener/beam/ER geometry is created (complete solid or just midsurfaces depending on the ini option), while landing curves are not created.

> **Restriction:** Only available for Intergraph format.

Split periodic faces

Split periodic faces during CAD import (default).

> **Restriction:** Only available for Inspire, JT, NX native, NX third-party, Parasolid, and SolidWorks readers.

Use native reader

Enable a native reader, requiring original CAD software and licenses, to be used.

> **Restriction:** Only available for NX format.

This method of import, along with feature- based meshing technology, provides a robust way to create high quality CAE solid tetrahedral mesh models without a lot of geometry cleanup.

User defined

Use this GUI field to enter additional options not explicitly exposed in the GUI, in the form `option1_name=option1_value`, `option2_name=option2_value`, and so on. If an option also exposed in the GUI is mentioned in this field, the value of this field prevails.

Default versions of the `_reader.ini` files are included in the directory `[Altair Home]/io/afc_translators/bin/[platform]` directory and its children. When a CAD reader is activated, each reader first checks the current working directory for the appropriate `_reader.ini` file. If the file is not found, the translators looks in all directories pointed by the HW_CONFIG_PATH environment variable.

As a last resort, the translator uses the default `_reader.ini` file in the above directory. In this way the `_reader.ini` file can have "global" or "local" user scope. For instance, "local" user changes for a current job can be made by copying and modifying the `_reader.ini` file in the local current working directory.

Options can take on only one value at a time. Options can also be commented out (ignored) by placing a `#` in front of an option, in which case the default value for that option will be used.

The available `_reader.ini` options are explained in detail within the Import Options sections for each reader.

Many CAD translators also import other relevant information as metadata attached to specific entities (assemblies, components, points, lines, surfaces, solids). Some metadata is generated by default while other metadata is generated by enabling/disabling certain options in the `_reader.ini` files. Metadata is stored in the database and can be used for review or to perform process automation. For example, you can obtain the tag (name) of a surface from the CAD file and apply certain mesh criteria to that surface inside HyperMesh. Refer to the Import Options topics for each format and CAD Metadata Naming for specific details about metadata.

CAD Import Message Files

Each reader reads the CAD file and sends the geometry information to HyperMesh. When a CAD file is read, a `.msg` file is created (or appended to) in the current working directory.

For example, the file `iges_reader.msg` is created for the IGES reader.

Three types of messages appear in the `.msg` file.

Info
Includes information about the file being read.

Warning
Indicates that data was modified.

Error
Indicates when a geometric entity could not be created.

These files can be useful for debugging errors found during import.

For CATIA and JT formats, those files (respectively `ct_raedr.msg` and `jt_reader.msg`) will contain detailed information about imported assembly files.

CAD Import Difficulties

Learn about possible difficulties you may encounter during CAD import.

If a line does not import correctly and has gaps, the gaps are filled with straight lines. If necessary, pieces of composite curves are reversed to make the entire line continuous. Lines shorter than the object space tolerance are rejected.

A line cannot intersect itself. This condition is not detected, and results may be incorrect if this kind of line is imported. A closed loop can have coincident starting and ending points.

Untrimmed surfaces are corrected if they do not import correctly. Internal gaps (C0-discontinuity) are removed by blending each edge of the gap together. Breaks at C1-discontinuities are controlled internally and may accept lower-level continuities depending on the reader's internal tolerance. While surfaces with one or two dimensions smaller than the object space tolerance are not prohibited, it is likely that they may be rejected.

Each point in the interior of the surface's parameter space should map to a distinct point in object space. This condition is not detected and incorrect results may occur if such a surface is imported. A surface may be closed in one or both directions.

Trimmed surfaces are a combination of untrimmed surfaces and one or more lines. The above restrictions apply to trimmed surfaces. If a surface cannot be trimmed, as much geometry information as possible is supplied. In most cases, both the underlying surface and its lines are created. It is then possible to use the Surface Edit panel to trim the surface manually.

It is also useful to refer to the `.msg` files generated by each CAD reader to find warnings/errors that may help in resolving import difficulties.

CAD Metadata Naming

The metadata names generated by the CAD readers are composed of a general prefix followed by a metadata specific name. The prefix is specified using the `@MetadataPrefix` option in each `_reader.ini` file.

The `@MetadataPrefix` option works as follows:

- The string you specify is appended as a prefix to the metadata name generated by that reader. The prefix is used explicitly with the metadata name. No . (period) is automatically added if it is not specified by the option. For example:

  `@MetadataPrefix = ".ALTAIR.HW"`

  .ALTAIR.HW<metadata name>

  `@MetadataPrefix = ".ALTAIR.HW."`

  .ALTAIR.HW.<metadata name>

- If the option is commented out, or the value is set to null "", no prefix is prepended and only the names of the metadata are used. For example:

  `@MetadataPrefix = ""`

  <metadata name>

- There are some pre-defined strings that have specific meaning. These pre-defined strings are automatically substituted with specific data where applicable. These strings are as follows:

  `<FORMAT>` - the format of the CAD file (CATIA, STEP, NX, and so on)

  `<VERSION>` - the version of the CAD file. For CATIA, this is V4 or V5. For NX, this is any NX version (for example, NX 10.0, NX 11.0, and so on). No other format currently supports this option. In the case where there is no `<VERSION>` possible, an empty string is substituted.

- All other strings are used verbatim. For example:

  `@MetadataPrefix = "<MYSTRING>"`

  <MYSTRING><metadata name>

- By default, the prefix is empty string for all readers. To maintain consistency with metadata names used by previous versions, a value of `".ALTAIR.HW.<FORMAT>."` is appropriate.

Additional Examples:

`@MetadataPrefix = ".ALTAIR.HW.<FORMAT>."`

.ALTAIR.HW.CATIA.<metadata name>

`@MetadataPrefix = "/ALTAIR/HW/<FORMAT>/"`

/ALTAIR/HW/CATIA/<metadata name>

```
@MetadataPrefix = "<FORMAT><VERSION>."
```

CATIAV5.<metadata name>

```
@MetadataPrefix = "<FORMAT>.<VERSION>."
```

CATIA.V5.<metadata name>

CAD Export

Learn about the CAD readers supported by HyperMesh and the options available for exporting CAD geometry data.

CAD writers are dynamically loaded upon demand.

Supported CAD Writers

CAD writers supported by HyperMesh.

Format	Latest Version Supported	File Extensions Supported
CATIA Composites Link		`.h5`
FiberSim		`.h5`
IGES	v6.0 JAMA-IS	`.iges` `.igs`
Inspire	2018.1	`.stmod`
JT	10.2	`.jt`

Parasolid	v30.0.185	`.x_t`
		`.x_b`
		`.xmt_txt`
		`.xmt_bin`

STEP	AP214	`.step`

Format	Latest Version Supported	File Extensions Supported
		.stp

CATIA Composites Link Writer

Use the CATIA Composites Link option to export composite drape data.

FiberSim Writer

Use the FiberSim option to export composite data.

IGES Writer

Supported Entities

Entities supported by the IGES writer.

- Subfigure Definition (308)
- Singular Subfigure Instance (408)
- Composite curve (102)
- Plane (108)
- Line (110)
- Point (116)
- Rational B-spline curve (126)
- Rational B-spline surface (128)
- Curve on a parametric surface (142)
- Trimmed (parametric) surface (144)
- Form 7 group without back pointers (402)
- Transformation Matrix (124)
- Name Property (406 Form 15)
- Color Definition (314)

Fixed points (those associated with lines or surfaces) are not supported by the IGES standard and are not exported. In order to export fixed points, it is necessary to convert them to free points. Then, after import, those free points must be projected onto the appropriate lines or surfaces in order to generate fixed points with the proper association.

Export Options

When IGES data is exported, it writes lines in a form resembling the database. Each segment on the line becomes a separate curve in IGES. If there is more than one segment in the line, the resulting curves in IGES are joined together with a composite curve entity. NURBS curves and ellipses are written as rational b-spline curves (IGES's rational B-splines are general NURBS).

Surface faces that comprise each surface are written to allow for output type options. NURBS surfaces are output as rational B-spline surfaces in IGES are cones, spheres, and tori. If necessary, curves on parametric surfaces and trimmed surface entities are used to trim the NURBS surface.

The IGES writer uses the `iges_writer.ini` file with the following available options:

@AssemblyMode

Value	Description
`Parts`	Create subfigure definition entity (308) and singular subfigure instance entity (408) from the part entity (default).
`Assemblies`	Create subfigure definition entity (308) and singular subfigure instance entity (408) from the assembly entity.
`Layers`	Send each component as a different layer (level) in the IGES file. This is the most efficient format.
`Groups`	Send each component as an associative instance entity (type 402, form 7).
`LayersAndGroups`	Send each component as a different layer (level), and send each set of faces comprising one surface as an associativity instance (group).

@AuthorName

Value	Description
`string`	String to write in the IGES author name field. Default is `Unknown`.

@AuthorOrganization

Value	Description
`string`	String to write in the IGES author organization field. Default is `Unknown`.

@Export

Value	Description
`All`	Export all geometry (default).
`Displayed`	Export displayed geometry only.

@NameFromRepresentation

Value	Description
`on`	Assign part name in CAD file from the representation file name and attach it to the part entity. Meaningful only if `AssemblyMode` = "`Parts`".
`off`	Assign part name in CAD file from the part entity name in HyperMesh. Meaningful only if AssemblyMode = "`Parts`".

@OptimizeForCAD

Value	Description
`on`	Fix gaps between stitched edges and vertices to make the actual geometry of the surfaces consistent with the model topology before export, and check the topology of the model for gaps that are larger than a given tolerance between shared, non-manifold, or suppressed edges. If the gap is found to be bigger than the tolerance, the surface and edge geometries are morphed parametrically and in 3D, if necessary, to make the gap smaller than the tolerance. In addition, non-essential degenerate edges are removed. This also modifies the geometry within HyperMesh (default).
`off`	Do not perform the optimization steps.

@OuterLoop

Value	Description
`Optional`1	The IGES 6.0 standard allows you to default to the natural outer loop of the surface being trimmed. If you are trimming a NURBS surface with several holes and no exterior trimming, the outer loop can be left out of the IGES file, making the surface representation shorter and more accurate. Use this option if it is supported by your IGES post-processor (default). This is not available with the JAMA-IS standard.

Value	Description
Mandatory2	For those post-processors that do not allow an optional outer loop, this option forces an outer loop to be written for trimmed surfaces.

@SourceUnits

Value	Description
MKS (m kg N s) MMKS (mm kg N s) MPA (mm t N s) CGS (cm g dyn s) MMKNMS (mm kg kN ms) MMGNMS (mm g N ms) IPS Std (in pounds lbf s) IPS Grav (in slinch lbf s) FPS Std (ft pounds lbf s) FPS Grav (ft slug lbf s)	Assign unit system to the data present in HyperMesh during export.

@TargetUnits

Value	Description
Microns Millimeters Centimeters Meters Kilometers Microinches Mils Inches Feet	Units system written in the IGES file.

Value	Description
Miles	

1. The most efficient form is JAMA planes and the Outer Loop Optional option; however, this choice may cause compatibility difficulties.
2. To be JAMA-IS compatible, select the JAMA planes and the Outer Loop Mandatory options.

Inspire Writer

Supported Entities

Entities supported by the Inspire writer.

- Model
- Assembly
- Part
- Solid
- Sheet
- Curve
- Point
- Axis System

Export Option

The Inspire writer uses the `inspire_writer.ini` file with the following available options:

@AssemblyMode

Value	Description
Parts	Create part entity hierarchy as assembly and part entity in Inspire.
Assemblies	Create assembly and component entity as assembly and part entity in Inspire.
Flatten	Write entities in all components into a single Inspire part.

@Export

Value	Description
All	Export all geometry (default).

Value	Description
`Displayed`	Export displayed geometry only.

@GeometryMode

Value	Description
`Standard`	Write CAD geometry with standard types such as cylinder, cone, circle, and so on (default).
`BSpline`	Convert all surface geometry data into NURBS. This option may increase the output file size and be time consuming depending on the actual geometry.

@NameFromRepresentation

Value	Description
`on`	Assign part name in CAD file from the representation file name attached to part entity. Meaningful only if `AssemblyMode` = "`Parts`".
`off`	Assign part name in CAD file from the part entity name in HyperMesh. Meaningful only if `AssemblyMode` = "`Parts`".

@OptimizeForCAD

Value	Description
`on`	Fix gaps between stitched edges and vertices to make the actual geometry of the surfaces consistent with the model topology before export, and check the topology of the model for gaps that are larger than a given tolerance between shared, non-manifold, or suppressed edges. If the gap is found to be bigger than the tolerance, the surface and edge geometries are morphed parametrically and in 3D, if necessary, to make the gap smaller than the tolerance. In addition, non-essential degenerate edges are removed. This also modifies the geometry within HyperMesh (default).
`off`	Do not perform the optimization steps.

@SourceUnits

Value	Description

`MKS (m kg N s)`
`MMKS (mm kg N s)`
`MPA (mm t N s)`
`CGS (cm g dyn s)`
`MMKNMS (mm kg kN ms)`
`MMGNMS (mm g N ms)`
`IPS Std (in pounds lbf s)`
`IPS Grav (in slinch lbf s)`
`FPS Std (ft pounds lbf s)`
`FPS Grav (ft slug lbf s)`
	Unit system assigned to the data present in HyperMesh during export

@TargetUnits

Value	Description

`MKS (m kg N s)`
`MMKS (mm kg N s)`
`MPA (mm t N s)`
`CGS (cm g dyn s)`
`MMKNMS (mm kg kN ms)`
`MMGNMS (mm g N ms)`
`IPS Std (in pounds lbf s)`
`IPS Grav (in slinch lbf s)`
`FPS Std (ft pounds lbf s)`
`FPS Grav (ft slug lbf s)`
	Units system in the Inspire file.

@TopologyMode

Value	Description
`Solid/Shell`	Export topology data similar to HyperMesh topology data, keeping the distinction between solids and shells (default).
`Surface`	Export each surface, including solid faces, as a single surface, effectively neglecting connectivity.

@Version

Value	Description
`string`	Inspire version to use for export. Valid versions are 2018.1 (default), 2017.1

JT Writer

Supported Entities

Entities supported by the JT writer.

- Point
- Line
- Surface
- Solid
- Part
- Assembly
- Elements are exported as tessellations as follows:
    - tria3 elements
    - quad4 elements, each split into two triangles
    - tetra4 element faces
    - penta6 element faces, with each quad face split into two triangles
    - hex8 element faces, with each face split into two triangles

Export Options

The JT export option is extensively used inside CAD modeling and PLM systems to quickly visualize a products structure and to compare changes between revisions of CAD and FE models. Select the JT export option to export all of the model's geometry and FE element information, component organization, and property color.

The JT writer uses the `jt_writer.ini` file with the following available options:

@AllowDisjointShells

Value	Description
`on`	Store non-manifold geometry in one disjoint feature.
`off`	Split and store non-manifold geometries as separate joint shell features (default).

@AssemblyMode

Value	Description
`Parts`	Create part entity hierarchy as assembly and part entity in JT.
`Assemblies`	Create assembly and component entity as assembly and part entity in JT.
`Flatten`	Write entities in all components into a single JT Part.

n

@AttributeForMaterialName

Value	Description
`string`	Attribute created with the name contains PDM metadata-material name information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "Material".

@AttributeForMeshFlag

Value	Description
`string`	Attribute created with the name contains PDM metadata-mesh flag information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "MeshFlag".

@AttributeForMID

Value	Description
`string`	Attribute created with the name contains PDM metadata-material ID information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "MaterialId".

`@AttributeForPartNumber`

Value	Description
`string`	Attribute created with the name contains PDM metadata-part number information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "PartNumber".

`@AttributeForPID`

Value	Description
`string`	Attribute created with the name contains PDM metadata-property ID information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "PID".

`@AttributeForRevision`

Value	Description
`string`	Attribute created with the name contains PDM metadata-revision information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "Revision".

`@AttributeForThickness`

Value	Description
`string`	Attribute created with the name contains PDM metadata-thickness information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "Thickness".

`@AttributeForUID`

Value	Description
`string`	Attribute created with the name contains the PDM metadata-UID information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "UID".

`@CreateTessellation`

Value	Description
`on`	Create tessellations even when no mesh is exported (default).
`off`	Create just the tessellations corresponding to the exported mesh.

`@Export`

Value	Description
`all`	Export all geometry (default).
`displayed`	Export displayed geometry only.

`@FileMode`

Value	Description
`Monolithic`	Create a single JT file.
`PerPart`	Create a JT file “Per part”.

Value	Description
	For example, there is a JT file containing the main assembly and each part is stored in an individual JT file in a subdirectory with the same name as the main JT file (default).

@GeometryMode

Value	Description
`Standard`	Write CAD geometry with standard types such as cylinder, cone, circle, and so on (default).
`BSpline`	Convert all surface geometry data into NURBS. This option may increase the output file size and be time consuming depending on the actual geometry.

@MetadataPrefixFilter

Value	Description
`string`	Prefix to use to find metadata to export. Only metadata with the specified prefix is exported. Any metadata without this prefix is not exported. Default is empty, meaning all metadata will be exported.

@NameFromRepresentation

Value	Description
`on`	Assign part name in CAD file from the representation file name attached to part entity. Meaningful only if `AssemblyMode` = "`Parts`".
`off`	Assign part name in CAD file from the part entity name in HyperMesh. Meaningful only if `AssemblyMode` = "`Parts`".

@OptimizeForCAD

Value	Description
`on`	Fix gaps between stitched edges and vertices to make the actual geometry of the surfaces consistent with the model topology before export, and

Value	Description
	check the topology of the model for gaps that are larger than a given tolerance between shared, non-manifold, or suppressed edges. If the gap is found to be bigger than the tolerance, the surface and edge geometries are morphed parametrically and in 3D, if necessary, to make the gap smaller than the tolerance. In addition, non-essential degenerate edges are removed. This also modifies the geometry within HyperMesh (default).
`off`	Do not perform the optimization steps.

`@RemoveMetadataPrefix`

Value	Description
`on`	Remove the metadata prefix for export. Meaningful only if `@MetadataPrefixFilter` is enabled and non-empty.
`off`	Do not remove the metadata prefix for export (default). Meaningful only if `@MetadataPrefixFilter` is enabled and non-empty.

`@SourceUnits`

Value	Description

`MKS (m kg N s)`
`MMKS (mm kg N s)`
`MPA (mm t N s)`
`CGS (cm g dyn s)`
`MMKNMS (mm kg kN ms)`
`MMGNMS (mm g N ms)`
`IPS Std (in pounds lbf s)`
`IPS Grav (in slinch lbf s)`
`FPS Std (ft pounds lbf s)`
`FPS Grav (ft slug lbf s)`
	Unit system assigned to the data present in HyperMesh during export

@TargetUnits

Value	Description
Microns Millimeters Centimeters Decimeters Meters Kilometers Mils Inches Feet Yards Miles	Units System in the exported JT file.

@TopologyMode

Value	Description
Solid/Shell	Export topology data similar to HyperMesh topology data, keeping the distinction between solids and shells (default).
Surface	Export each surface, including solid faces, as a single surface, effectively neglecting connectivity.

@Version

Value	Description
string	The JT version to use for export. Valid versions are 10.2 (default), 10.1, 10.0, 9.5, 9.4, 9.3, 9.2, 9.1, 9.0, 8.1, 8.1, 8.0, 7.0 or 6.4.

@WriteColorFrom

Value	Description
string	Specify colors for the geometrical entities during export. Colors are supported at the component level only, whereas individual entity colors (points, lines, surfaces, solids) are not supported.

Value	Description
	Valid values are Component (default) and Metadata.

@WriteMetaDataAsColor

Value	Description
`string`	Speciify the metadata to use for coloring entities. Default is `COLOR_RGB`. Meaningful only if `@WriteColorFrom` = "`Metadata`".

@WriteMetaDataAsName

Value	Description
`string`	Specify the metadata to use for naming entities. Default is `TAG`. Meaningful only if `@WriteNameFrom` = "`Metadata`".

@WriteNameFrom

Value	Description
`string`	Specify names for the geometrical entities during export. Names are supported at the component level only, whereas individual entity names (points, lines, surfaces, solids) are not supported. Valid values are Component and Metadata (default).

Parasolid Writer

Supported Entities

Entities supported by the Parasolid writer.

- Assembly
- Part
- Solid
- Surface

- Curve
- Free Point

Entities not supported by the Parasolid writer.

- Axis system
- Vector

Export Options

The Parasolid reader uses the `parasolid_writer.ini` file with the following available options:

@AllowDisjointShells

Value	Description
on	Store non-manifold geometry in one disjoint feature.
off	Split and store non-manifold geometries as separate joint shell features (default).

@AssemblyMode

Value	Description
`Parts`	Create part entity hierarchy as assembly and part entity in Parasolid.
`Assemblies`	Create assembly and component entity as assembly and part entity in Parasolid.
`Flatten`	Write entities in all components into a single Parasolid part.

@AttributeForMaterialName

Value	Description
`string`	Attribute created with the name contains PDM metadata-material name information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "Material".

@AttributeForMeshFlag

Value	Description
`string`	Attribute created with the name contains PDM metadata-mesh flag information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "MeshFlag".

`@AttributeForMID`

Value	Description
`string`	Attribute created with the name contains PDM metadata-material ID information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "MaterialId".

`@AttributeForPartNumber`

Value	Description
`string`	Attribute created with the name contains PDM metadata-part number information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "PartNumber".

`@AttributeForPID`

Value	Description
`string`	Attribute created with the name contains PDM metadata-property ID information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "PID".

`@AttributeForRevision`

Value	Description
`string`	Attribute created with the name contains PDM metadata-revision information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "Revision".

`@AttributeForThickness`

Value	Description
`string`	Attribute created with the name contains PDM metadata-thickness information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "Thickness".

`@AttributeForUID`

Value	Description
`string`	Attribute created with the name contains the PDM metadata-UID information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "UID".

`@Export`

Value	Description
`all`	Export all geometry (default).
`displayed`	Export displayed geometry only.

`@GeometryMode`

Value	Description
`Standard`	Write CAD geometry with standard types such as cylinder, cone, circle, and so on (default).

Value	Description
`BSpline`	Convert all surface geometry data into NURBS. This option may increase the output file size and be time consuming depending on the actual geometry.

`@HMOrignalIdAsAttribute`

Value	Description
`on`	Write HyperMesh entity Identifier as HM_ORIGINAL_ID attribute
`off`	Do not export HyperMesh entity Identifier (default).

`@MetadataPrefixFilter`

Value	Description
`string`	The prefix to use to find metadata to export. Only metadata with the specified prefix is exported. Any metadata without this prefix is not exported. Default is empty, meaning all metadata will be exported.

`@NameFromRepresentation`

Value	Description
`on`	Assign part name in CAD file from the representation file name attached to part entity. Meaningful only if `AssemblyMode` = "`Parts`".
`off`	Assign part name in CAD file from the part entity name in HyperMesh. Meaningful only if `AssemblyMode` = "`Parts`".

`@OptimizeForCAD`

Value	Description
`on`	Fix gaps between stitched edges and vertices to make the actual geometry of the surfaces consistent with the model topology before export, and check the topology of the model for gaps that are larger than a given tolerance between shared, non-manifold, or suppressed edges. If the gap is found to be bigger than the tolerance, the surface and edge geometries

Value	Description
	are morphed parametrically and in 3D, if necessary, to make the gap smaller than the tolerance. In addition, non-essential degenerate edges are removed. This also modifies the geometry within HyperMesh (default).
`off`	Do not perform the optimization steps.

@RemoveMetadataPrefix

Value	Description
`on`	Remove the metadata prefix for export. Meaningful only if `@MetadataPrefixFilter` is enabled and non-empty.
`off`	Do not remove the metadata prefix for export (default). Meaningful only if `@MetadataPrefixFilter` is enabled and non-empty.

@SourceUnits

Value	Description

`MKS (m kg N s)`
`MMKS (mm kg N s)`
`MPA (mm t N s)`
`CGS (cm g dyn s)`
`MMKNMS (mm kg kN ms)`
`MMGNMS (mm g N ms)`
`IPS Std (in pounds lbf s)`
`IPS Grav (in slinch lbf s)`
`FPS Std (ft pounds lbf s)`
`FPS Grav (ft slug lbf s)`
	Unit system assigned to the data present in HyperMesh during export

@TargetUnits

Value	Description
`Meters`	Units system in the exported Parasolid file. Parasolid stores data in meters.

`@TopologyMode`

Value	Description
`Solid/Shell`	Export topology data similar to HyperMesh topology data, keeping the distinction between solids and shells (default).
`Surface`	Export each surface, including solid faces, as a single surface, effectively neglecting connectivity.

`@Version`

Value	Description
`string`	Parasolid version to use for export. Valid versions are 30.0 (default), 28.1, 28.0, 27.0, 26.1, 26.0, 25.1, 25.0, 24.0, 23.0, 22.0 or 21.0.

`@WriteColorFrom`

Value	Description
`string`	Specify colors for the geometrical entities during export. Colors are supported at the component level only, whereas individual entity colors (points, lines, surfaces, solids) are not supported. Valid values are Component (default) and Metadata.

`@WriteMetaDataAsColor`

Value	Description
string	Specify the metadata to use for coloring entities. Default is `COLOR_RGB`. Meaningful only if `@WriteColorFrom` = "`Metadata`".

`@WriteMetaDataAsName`

Value	Description
`string`	Specify the metadata to use for naming entities. Default is `TAG`. Meaningful only if @WriteNameFrom = "`Metadata`".

@WriteNameFrom

Value	Description
`string`	Specify names for the geometrical entities during export. Names are supported at the component level only, whereas individual entity names (points, lines, surfaces, solids) are not supported. Valid values are Component and Metadata (default).

STEP Writer

Supported Entities

Entities supported by the STEP writer.

- Point
- Line
- Surface
- Solid
- Part
- Assembly

Entities not supported by the STEP writer.

- Axis system
- Fixed points. In order to export fixed points, it is necessary to convert them to free points. Then, after import, those free points must be projected onto the appropriate lines or surfaces in order to generate fixed points with the proper association.
- Vector

Export Options

The STEP writer uses the `step_writer.ini` file with the following available options:

@AllowDisjointShells

Value	Description
`on`	Store non-manifold geometry in one disjoint feature.
`off`	Split and store non-manifold geometries as separate joint shell features (default).

@AssemblyMode

Value	Description
`Parts`	Create part entity hierarchy as assembly and part entity in STEP file.
`Assemblies`	Create assembly and component entity as assembly and part entity in STEP file.
`Flatten`	Write entities in all components into a single STEP part.

@AttributeForMaterialName

Value	Description
`string`	Attribute created with the name contains PDM metadata-material name information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "Material".

@AttributeForMeshFlag

Value	Description
`string`	Attribute created with the name contains PDM metadata-mesh flag information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "MeshFlag".

@AttributeForMID

Value	Description
`string`	Attribute created with the name contains PDM metadata-material ID information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "MaterialId".

`@AttributeForPartNumber`

Value	Description
`string`	Attribute created with the name contains PDM metadata-part number information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "PartNumber".

`@AttributeForPID`

Value	Description
`string`	Attribute created with the name contains PDM metadata-property ID information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "PID".

`@AttributeForRevision`

Value	Description
`string`	Attribute created with the name contains PDM metadata-revision information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "Revision".

`@AttributeForThickness`

Value	Description
`string`	Attribute created with the name contains PDM metadata-thickness information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "Thickness".

@AttributeForUID

Value	Description
`string`	Attribute created with the name contains the PDM metadata-UID information associated with the part entity. Meaningful only if `AssemblyMode` = "`Parts`". Default is "UID".

@Export

Value	Description
`All`	Export all geometry (default).
`Displayed`	Export displayed geometry only.

@GeometryMode

Value	Description
`Standard`	Write CAD geometry with standard types such as cylinder, cone, circle, and so on (default).
`BSpline`	Convert all surface geometry data into NURBS. This option may increase the output file size and be time consuming depending on the actual geometry.

@HMOrignalIdAsAttribute

Value	Description
`on`	Write HyperMesh entity Identifier as HM_ORIGINAL_ID attribute

Value	Description
`off`	Do not export HyperMesh entity Identifier (default).

@LayerMode

Value	Description
`None`	No special layer handling is performed (default).
`ComponentID`	The entities in each HyperMesh component will be assigned to the `layernumber` in the STEP file corresponding to that component ID.

@MetadataPrefixFilter

Value	Description
`string`	The prefix to use to find metadata to export. Only metadata with the specified prefix is exported. Any metadata without this prefix is not exported. Default is empty, meaning all metadata will be exported.

@NameFromRepresentation

Value	Description
`on`	Assign part name in CAD file from the representation file name attached to part entity. Meaningful only if `AssemblyMode` = "`Parts`".
`off`	Assign part name in CAD file is assigned from the part entity name in HyperMesh. Meaningful only if `AssemblyMode` = "`Parts`".

@OptimizeForCAD

Value	Description
`on`	Fix gaps between stitched edges and vertices to make the actual geometry of the surfaces consistent with the model topology before export, and check the topology of the model for gaps that are larger than a given tolerance between shared, non-manifold, or suppressed edges. If the gap is found to be bigger than the tolerance, the surface and edge geometries

Value	Description
	are morphed parametrically and in 3D, if necessary, to make the gap smaller than the tolerance. In addition, non-essential degenerate edges are removed. This also modifies the geometry within HyperMesh (default).
`off`	Do not perform the optimization steps.

`@RemoveMetadataPrefix`

Value	Description
`on`	Remove the metadata prefix for export. Meaningful only if `@MetadataPrefixFilter` is enabled and non-empty.
`off`	Do not remove the metadata prefix for export (default). Meaningful only if `@MetadataPrefixFilter` is enabled and non-empty.

`@SourceUnits`

Value	Description

`MKS (m kg N s)`
`MMKS (mm kg N s)`
`MPA (mm t N s)`
`CGS (cm g dyn s)`
`MMKNMS (mm kg kN ms)`
`MMGNMS (mm g N ms)`
`IPS Std (in pounds lbf s)`
`IPS Grav (in slinch lbf s)`
`FPS Std (ft pounds lbf s)`
`FPS Grav (ft slug lbf s)`
	Unit system assigned to the data present in HyperMesh during export

`@StepHeaderAuthorInfo`

Value	Description
`string`	String to use for the AUTHOR field in the STEP header.

@StepHeaderAuthorizationInfo

Value	Description
`string`	String to use for the AUTHORIZATION field in the STEP header.

@StepHeaderOrganizationInfo

Value	Description
`string`	String to use for the ORGANIZATION field in the STEP header.

@TargetUnits

Value	Description

`Microns`
`Millimeters`
`Centimeters`
`Meters`
`Kilometers`
`Microinches`
`Mils`
`Inches`
`Feet`
	Units System in the exported STEP file.

@TopologyMode

Value	Description
`Solid/Shell`	Export topology data similar to HyperMesh topology data, keeping the distinction between solids and shells (default).
`Surface`	Export each surface, including solid faces, as a single surface, effectively neglecting connectivity.

@Version

Value	Description
string	The STEP version to use for export. Valid versions are AP214 (default).

@WriteColorFrom

Value	Description
string	Specify colors for the geometrical entities during export. Colors are supported at the component level only, whereas individual entity colors (points, lines, surfaces, solids) are not supported. Valid values are Component (default) and Metadata.

@WriteMetaDataAsColor

Value	Description
string	Specify the metadata to use for coloring entities. Default is `COLOR_RGB`. Meaningful only if `@WriteColorFrom` = "`Metadata`".

@WriteMetaDataAsName

Value	Description
string	Specify the metadata to use for naming entities. Default is `TAG`. Meaningful only if `@WriteNameFrom` = "`Metadata`".

@WriteNameFrom

Value	Description
string	Specify names for the geometrical entities during export. Names are supported at the component level only, whereas individual entity names (points, lines, surfaces, solids) are not supported.

Value	Description
	Valid values are Component and Metadata (default).

CAD Export Options

CAD writers provide options for processing data during export.

Some of these options are available from the Export browser, while other options are accessed only from each writer's `_writer.ini` file.

Version
Corresponds to the @Version option.

File mode
Corresponds to the @FileMode option.

Export
Corresponds to the @Export option.

Source units
Corresponds to the @SourceUnits option.
Set a source unit before exporting by manually selecting a unit system from a list of units, or click **From Metadata** to select a source unit from the MODEL_UNIT metadata attached to the component/part. If the components/parts have differing MODEL_UNIT metadata values, the most commonly used value is adopted and a warning message is displayed.

To prevent the frequent, unintended, selection of units, the source units selection is cleared anytime the export CAD file type is changed and you must reselect the units before exporting.

Target units
Corresponds to the @TargetUnits option. When Target units value is selected as Source units (default), Target units is assigned a value from Source units and used during export.

Output Loop
Corresponds to the @OuterLoop option.

Layers
Corresponds to the @LayersMode options.

Geom mode
Corresponds to the @GeometryMode option.

Topo mode
Corresponds to the @TopologyMode option.

Assem mode
Corresponds to the @AssemblyMode option.

Naming mode
Corresponds to the @WriteMetaDataAsName and @WriteNameFrom options.

User defined

Additional options not explicitly exposed in the GUI, in the form option1_name=option1_value, option2_name=option2_value, and so on. If an option is also exposed in GUI is mentioned in this field, the value of this field prevails.

Name from representation

Corresponds to the @NameFromRepresentation option.

Optimize for CAD

Corresponds to the @OptimizeForCAD option.

Prompt before overwrite

Prompt the user if the specified file name already exists and will be overwritten.

CAD Export Message Files

When a CAD file is exported from HyperMesh, a `.msg` file is created (or appended to) in the current working directory.

For example, the file `iges_writer.msg` is created for the IGES writer. Three types of messages appear in the `.msg` file:

info

Includes information

warning

Indicates all warnings that occurred during the export.

error

Indicates when geometric entities could not be created in the CAD file.

These files can be useful for debugging errors that occured during export.

Browsers

5

Browsers supply a great deal of view-related functionality in HyperMesh by listing the parts of a model in a tabular and/or tree-based format, and providing controls inside the table that allow you to alter the display of model parts.

This chapter covers the following:

Basic Browser Operations

Browsers display information in a Treeview. In Treeviews, collectors such as components or groups appear at the top level of the hierarchy, while collected entities such as elements or surfaces display as "children" nested within the collector to which they belong. Each item in a Treeview is commonly referred to as a "node", regardless of whether it is a Parent Node or a Child Node.

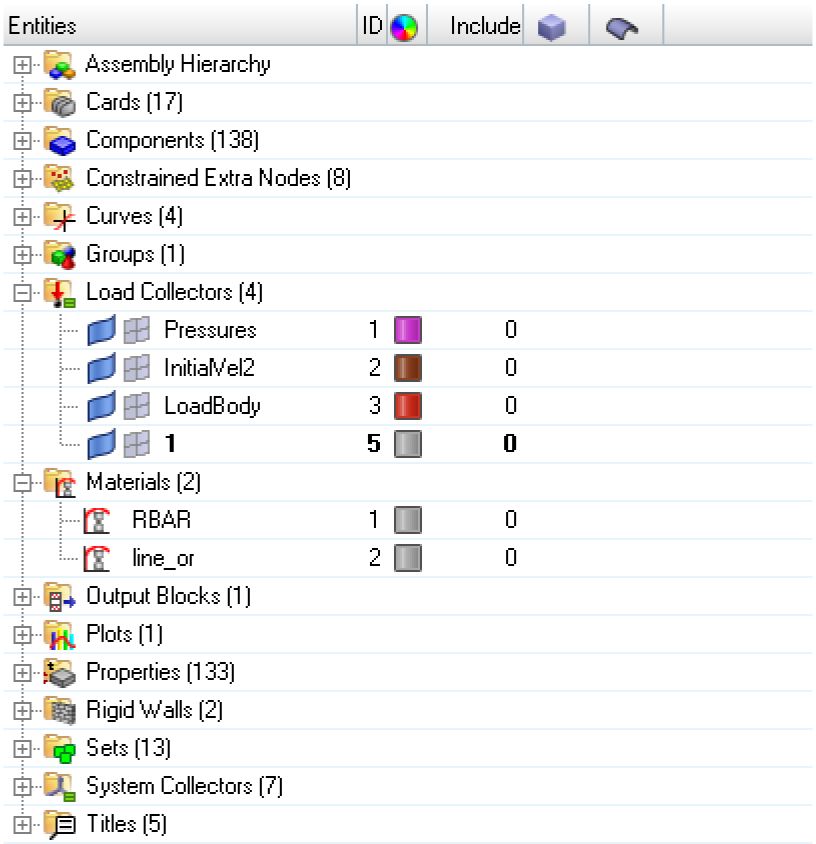


Figure 148: Hierarchy of Entities in the Model Browser

Performing an action on a child node affects only that item, be it a single load or the entire collection of elements in the model, which, in the example above, are collectively a child node of the components parent node. However, performing the same action on a parent node applies it to all children of that node as well. For example, in the image above, turning off the display of elements will not affect the display of connectors, geometry, or components (the parent node of the elements). However, turning off the display of components will turn off the display of connectors, elements, and geometry because they are children of the components node.

Different browsers are customized for usage with regard to the types of parts that you want to work with. Most browsers have similar basic functionality for sorting entities, filtering entities, and finding entities. However, most browsers also include a right-click context menu and sets of control buttons, similar to toolbars, but unable to be detached to re-docked, that are specific to the browser in which they appear.

Sorting Entities in the Model Browser

To sort the entities in a folder within the browser:

1. Click the heading of each column.
2. Click **Entities** to sort alphabetically by name, or click **ID** to sort numerically by entity ID.

   In either case, repeated clicks toggle between ascending and descending order.

Query Builder

Use the query builder to find and filter entities by building advanced filters for attributes listed in browser columns.

Recent filters are saved and can be quickly accessed by clicking .

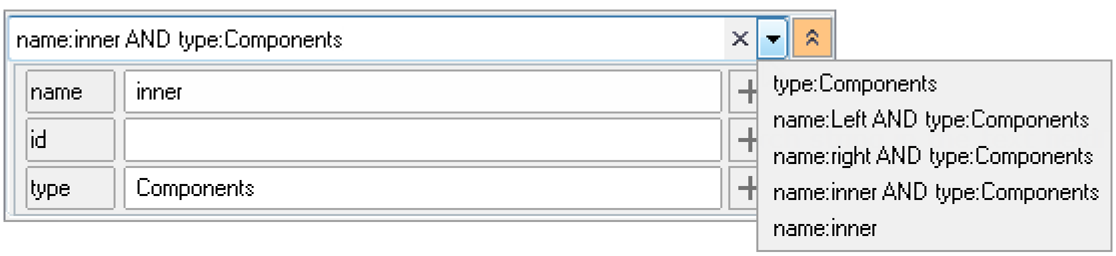


Figure 149:

Interactively Build Queries

1. To expose the interactive query builder, click the Expand/Collapse icon .

   By default, the name, id, and type attribute fields are exposed.
2. Filter via name and/or id by typing a string into the text field and pressing Enter.
3. Change the entity type by clicking on the type field and selecting a desired entity type.
4. Apply your selection by clicking .

   An example of an interactively defined query is shown in the image below. In this example, four additional attribute fields were added.

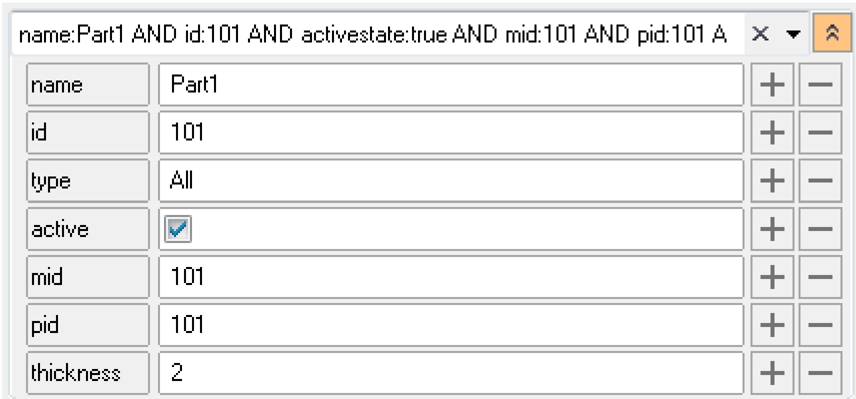


Figure 150:

> **Tip:**
> - To select multiple entity types, press and hold Control.
> - Add additional attribute fields by clicking [+]. Remove attribute fields by clicking [—].

Manually Build Queries

Manually build queries by typing into the text field.

The manual query builder syntax follows this basic form: <attribute>:<search pattern>. Use AND, OR, <, >, operators to refine your search.

Figure 151:

The following table provides examples of query builder syntax and results.

Entering this string...	Returns this result...
2	All entities that contain the number 2 in their ID.
"2"	All entities with the ID 2.
shell	All entities that contain shell in their name.
"shell"	All entities with the name shell.
name: shall AND id: 2	All entities that contain shell in their name and 2 in their ID.
id:>10 AND id:<20 AND name:cent	All entities that contain cent in their name and have an ID greater than 10 and less than 20.

Entering this string...	**Returns this result...**
1-4	All entities with the IDs 1, 2, 3, and 4.
type:comp AND id:1-4	All components with the IDs 1, 2, 3, and 4.

Display Controls and Browser Modes

These controls affect which entities display in the graphics area, and how they display, such as shaded or wireframe.

Global Display Controls

Use the global display controls to control the graphical display state of entities listed in the browser.

These controls reside at the left side of the browser, just below the Model Browser View Modes.

Figure 152:

By default, the global display controls will work on all entities listed in the browser, however, these controls can also work at the folder and individual named entity level. For example, to turn off all components, highlight the Component folder and click **Display None**. If an individual component is highlighted within the Component folder then the global display controls will only work on that specific entity. To enable the Display All, Display None, and Display Reverse controls to work on all of the entities listed in the browser, click on the white space within the browser (de-selecting any selected entities).

A shaded entity icon in the browser indicates that the entity is visible in the graphics area while an opaque icon indicates that the entity is hidden in the graphics area.

Because these controls work in combination with the filters, only the items displayed in the browser are affected. The hidden items that are also filtered out of the browser will not be displayed.

Elements and Geometry

Determines whether the other global display controls will act on elements, geometry, or elements and geometry.

Display All

Displays all of the entities listed in the browser.

Display None

Hides all of the entities listed in the browser.

Display Reverse

Reverses the display of all the entities listed in the browser (displays hidden entities, hides displayed entities).

> **Note:** These controls only affect the display state of entities; they do not actually remove entities from the model.

Local Display Controls

Use the Local Display Controls to change visual appearance of individual entities, such as shaded or wireframe.

These controls are located within the tree list, and each affects the specific entity that it appears beside.

Entity Display Icons

You can display or hide entities by toggling the corresponding icon that is located on the left-hand side of the entities name. The following rules apply:

- A bold icon next to an entity (components, multibodies, load collector, and so on) represents that the entity is currently displayed; a dimmed icon next to an unchecked entity represents that the entity is turned off from display.
- Assemblies containing components or multibodies are considered displayed only when all of the contents are displayed.
- Activating an assembly's display control icon displays all of its contents.
- Activating an assembly's display control icon displays all its components and multibodies.
- Deactivating the display control icon check box for an assembly hides all of its components and multibodies.
- Deactivating the display control icon for an item hides all of its parent assemblies.
- Deactivating the display control icon for an item does not affect the state of its parent assembly.
- An empty assembly never displays.

Colors

Assemblies, beam section collectors, blocks, components, contact surfaces, curves, groups, load collectors, materials, properties, shapes, system collectors, tags, titles, and vector collectors can all be colored individually. In the Model Browser, the column displays each entity's assigned color. To change an entity's color, click its color icon and select a new color from the palette.

Display Mode

Components have several display states, based on a combination of their elements and their geometry. In the Model Browser, the column displays the display mode assigned to each component, assembly, or load collector. To change the display state, click the entities display mode icon and then select a new style. Depending on which option you select, the entity displays differently:

FE Styles

Wireframe mesh

Shaded elements (no mesh)

Shaded elements with mesh lines

Shaded elements with feature lines (no mesh)

Transparent shaded elements without mesh

Geometry Styles

Wireframe geometry

Wireframe geometry with surface lines

Shaded geometry

Shaded geometry with feature lines

Action Mode Tools

Use the action mode tools, on the right side of the Model Browser, to control both entity selection and the display of the model.

The first two tools determine the type of entity that you wish to manipulate, while the rest perform specific actions.

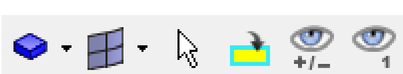


Figure 153:

Entity Type

The Entity Type tool ( ) sets functions as a "mode" selector for the rest of the tools by determining what type of entity the remaining tools will act on.

For example, if you set the Entity Type to "components", the Selector will only select or deselect components.

Click ( ) to open the Entity Type drop-down menu and then select the desired entity type.

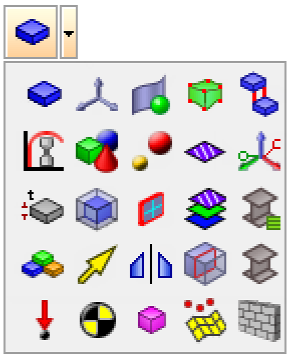


Figure 154:

Elements/Geometry

The Elements/Geometry tool ( ) sets functions as a "mode" selector for the rest of the tools, by determining what type of contained entities the remaining tools will act on.

For example, if you set the mode to "elements", the Selector will only select or deselect elements.

Click ( ) to open the Elements/Geometry drop-down menu and then select the desired contained entity type. The entity types available from the drop-down include: Elements, Geometry, and Elements and Geometry.

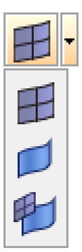


Figure 155:

Selector

Use the selector ( ) to interactively select supported entities from either the Model Browser or from within the graphics area.

The type of entity you are able to select is determined by the entity selection that you made in the Entity Type drop-down menu.

> **Note:** You are only able to select one entity type at a time, but you can select multiple entities without losing your selection. This is also an efficient way of selecting multiple entities at once – such as when changing color.

HyperMesh outlines the selected entities with a thick white line in the graphics area, and highlights the entity names in grey in the Model Browser.

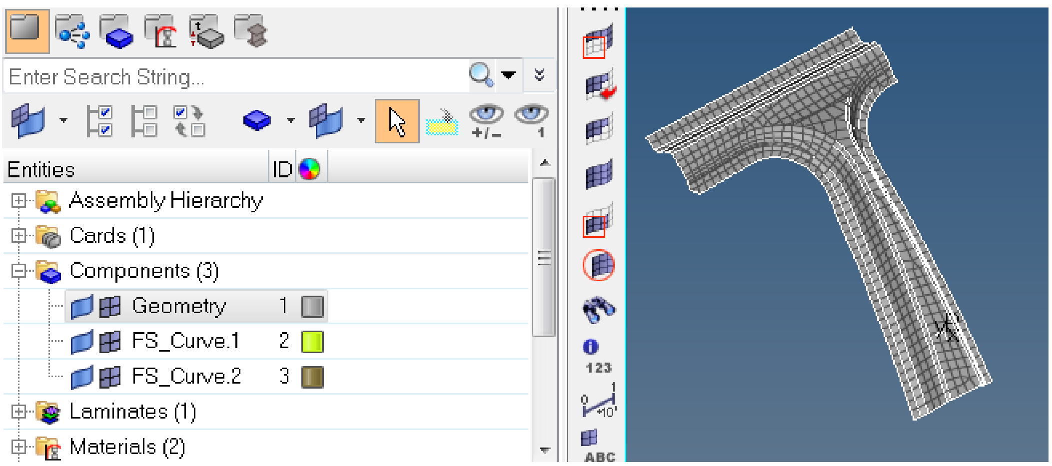


Figure 156:

The selection behavior of ply entities acts differently than other entities. Plies often contain both geometry and FEs that nearly match. In order to distinguish between the two in the graphics area, HyperMesh outlines selected geometry with a thick white line and selected FEs with the color that it is assigned.

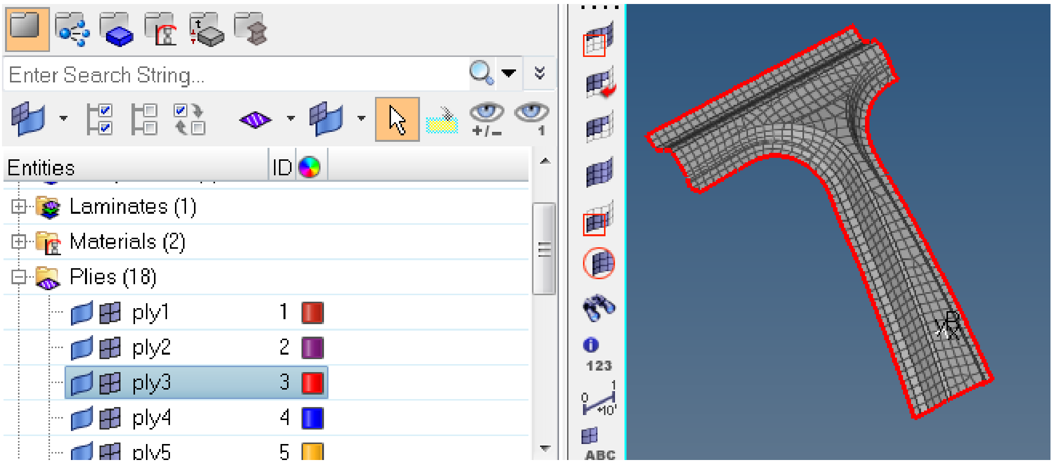


Figure 157:

When you select materials and properties, the visualization mode changes to by Mat or by Prop, respectively.

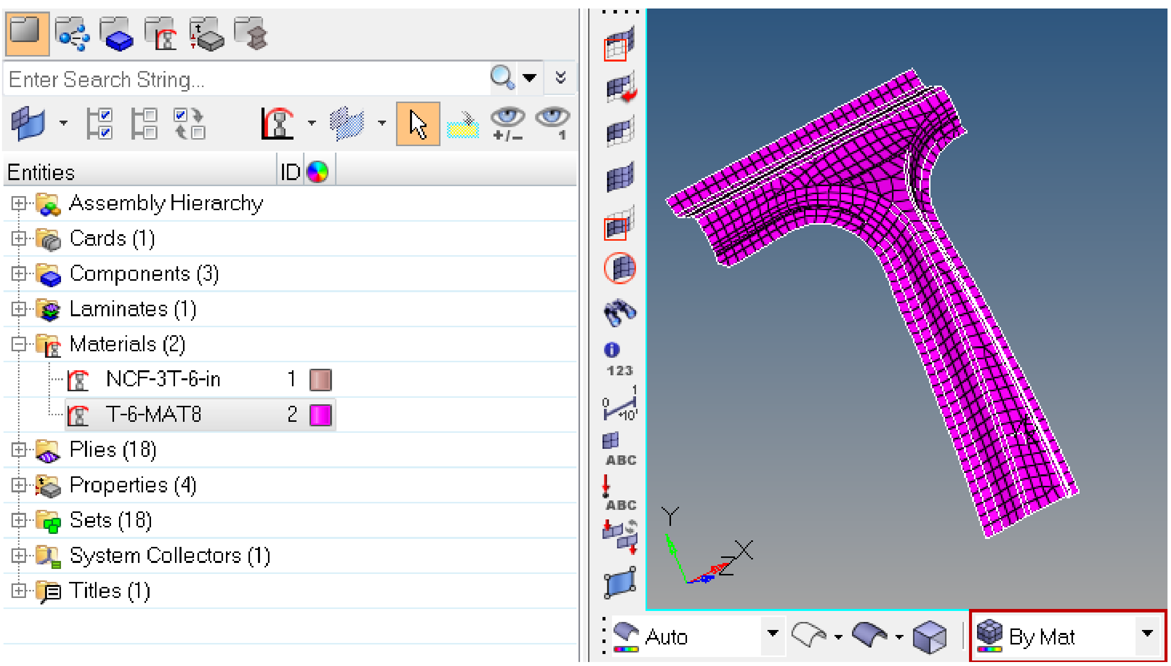


Figure 158:

Change the Selection in the Graphics Area Using the Mouse Scroll Wheel

In a Microsoft Windows environment, you can also use a scroll-wheel-equipped mouse to "drill down" through the model and select parts that may not be immediately visible.

1. To move deeper into the field of view and away from you, hold the left mouse button down while rolling the scroll wheel forward.

2. To move the selection closer to you, hold the left mouse button down while rolling the scroll wheel backward.

Highlighting indicates the component that will be selected once you release the left button.

> **Note:** This only works in a Windows environment; it will not work in UNIX, Linux, and so on.

3. You can also use the Selector tool in conjunction with the Hide or Isolate functions to hide or isolate the entities selected in the graphics area. Once you have made a selection in the Model Browser or graphics area, using the selector, click the Hide icon (right mouse button) to hide that selection or click the Isolate icon (left mouse button) to isolate that selection.

Select Entities in the Graphics Area

To select entities in the graphics area, you can:

- Left-click to select entities.
- Left-click and hold to pre-highlight entities; the entity under the selector at any given moment highlights, but is not selected until you release the mouse button.
- Shift + left-click to use window selection to highlight or select multiple entities.
- Right-click an entity to deselect it.
- Control + right-click to use window selection to deselect multiple entities.

When using a panel with an active collector, each entity selected gets added to the collector, while each one de-selected gets removed from the collector.

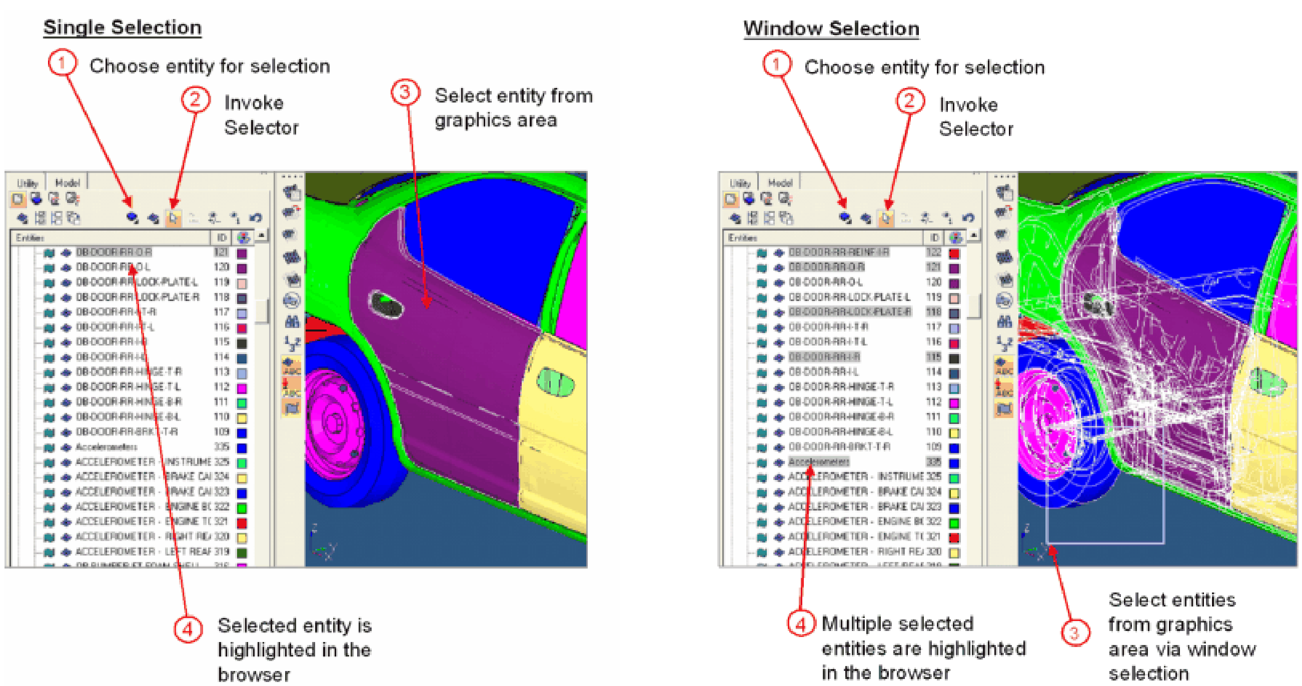


Figure 159:

Select Entities in the Model Browser Tree List

To select entities using the Model Browser, you can:

- Left-click to select and highlight an entity in the browser – the entity is also highlighted in graphics area.
- Control + left-click to select multiple entities in both the browser and graphics area. Multiple entities of the same type can be appended to the selection. Left-click can be used to add/remove components from an active collector on a pane.
- Shift + left-click highlights multiple entities in the browser and the graphics area. You can append and remove entities from a panel's active collector list depending on which entity entry is selected.
- Right-clicking highlights components in the list and invokes the context menu – no highlighting of the entity in graphics area will occur with this operation.

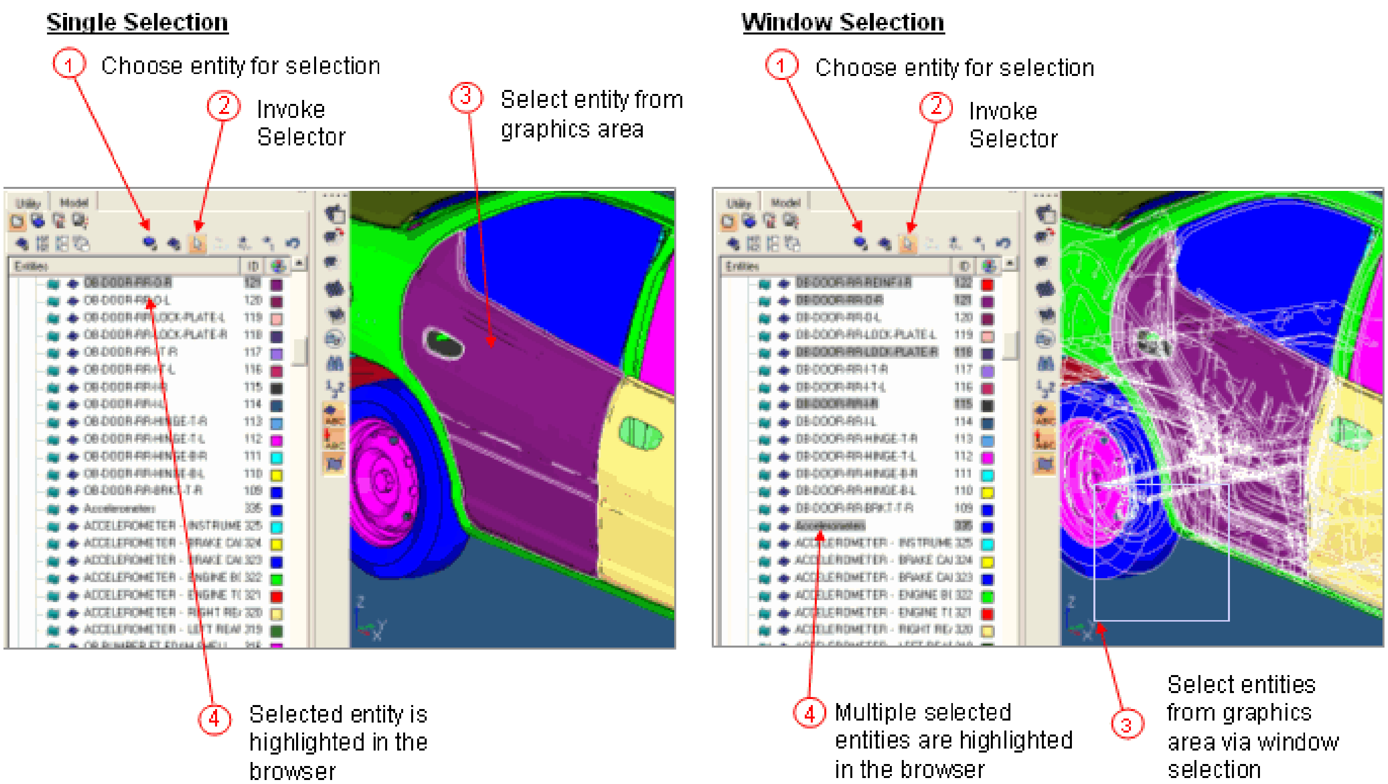


Figure 160:

Add to Panel Collector

The Add to Panel Collector ( ) is a function whereby the browser can be used to select and add entities to the panel collectors.

This is an alternative method to using the advanced selection capabilities already available in each collector's extended entity selection menu. This button is only available when you have a panel open that contains at least one entity collector.

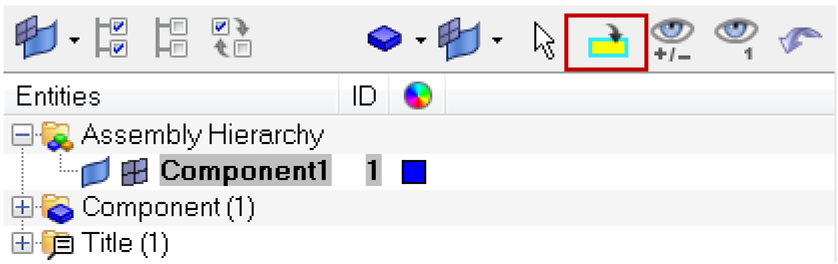


Figure 161:

> **Note:** The selected entities are only added to the panel's active (blue halo) collector. Additionally, only entities of the correct type will be added to the active collector; for example, you cannot add lines to an Elements collector.

There are two ways of using this tool:

1. Use the selector to choose a set of entities beforehand, and then click **Add to Panel Collector** to add them to the panel's collector.

   In this method, the selector effectively gives a preview of the selection, because the selected entities are highlighted but only added to the active panel collector when you click **Add to Panel Collector**. For example:

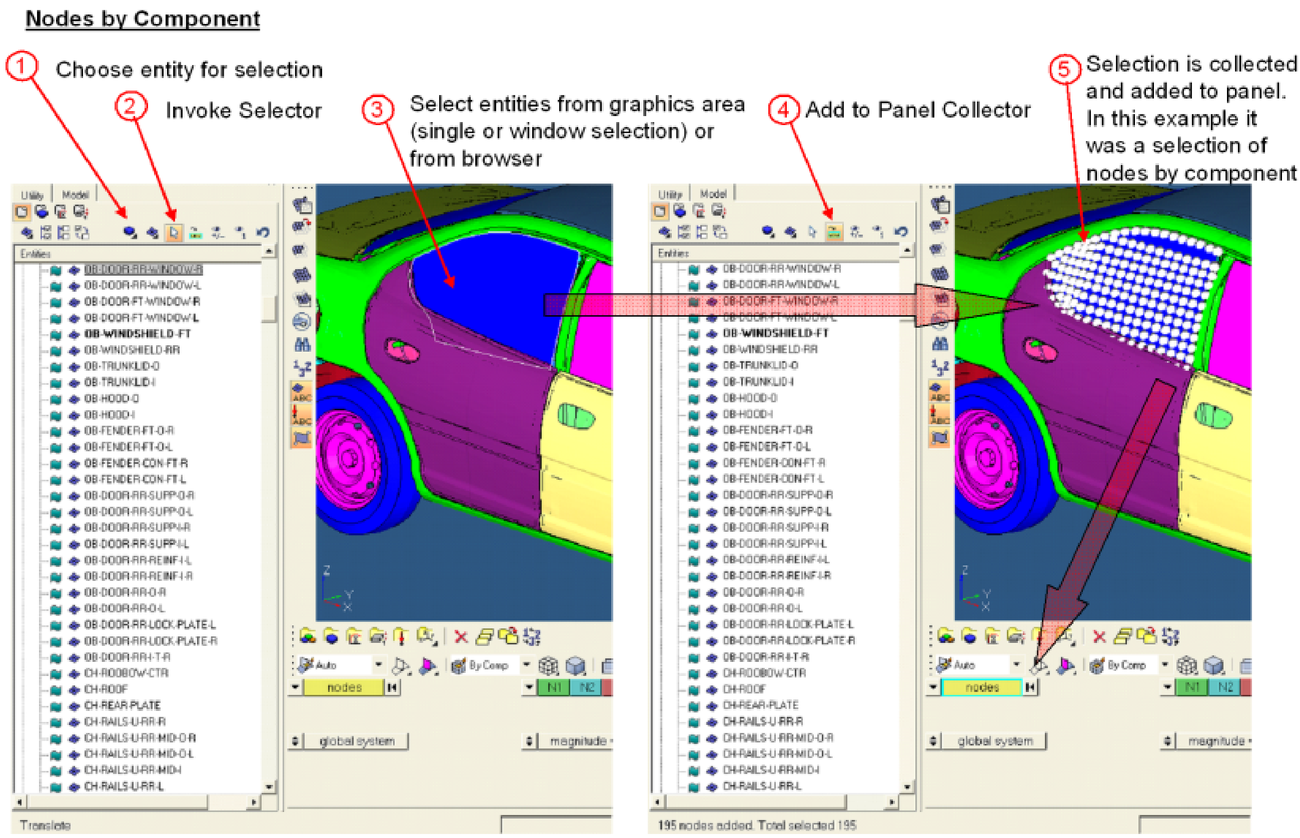


Figure 162:

2. Click **Add to Panel Collector** first to activate it, then select the entities you wish to add to the collector. HyperMesh adds each selection to the collector immediately after you release the mouse button.

You can use either method in the graphics area or the browser tree list. Control-clicks and Shift-clicks are supported in the browser list; window selections (Shift+click-and-drag) are supported in the graphics area.

You can use either method in the graphics area or the browser tree list. Control-clicks and Shift-clicks are supported in the browser list; window selections (Shift+click-and-drag) are supported in the graphics area.

Remove Entities From a Panel Collector

To remove entities from a panel collector, you can either:

- Click the collector's reset button .
- Right-click (with full shift-, control-, or window-based selection) on individual entities when the Add to Panel Collector button is active.

Show/Hide

Use Show/Hide () to control the model display by interactively selecting entities within the graphics area.

This tool is not designed to operate within the browser; therefore, you should only use this tool to select entities in the graphics area. To control the browser display, use the local entity controls found inside the browser's tree structure.

> **Note:** When using window selection (Shift-click-and-drag), an entity is considered selected if any portion of it falls within the window; you do not need to encompass the entire entity with the window, only a small portion of it. Also remember that only entities of the types determined by the Entity Types and Elements/Geometry buttons will be hidden or revealed.
>
> When you select materials and properties, the Visualization mode will change to by Mat and by Prop respectively. However, no graphics highlighting of material or properties occurs when using the selector.
>
> Similar to the selector, you can change your selection in the graphics area using the mouse scroll-wheel for the Show/Hide tools. To show your selection, left-click, hold, and scroll. To hide your selection, right-click, hold, and scroll.

Pick in Graphics

- Left-click turns on entities to the display that are currently turned off.
- Left-click-and-drag pre-highlights only the entities that are currently turned off in the display (entities already turned on do not highlight).
- Right-click turns off entities in the display.
- Right-click-and-drag pre-highlights only the entities that are currently turned on in the display (entities already turned off do not highlight.)

Single Selection

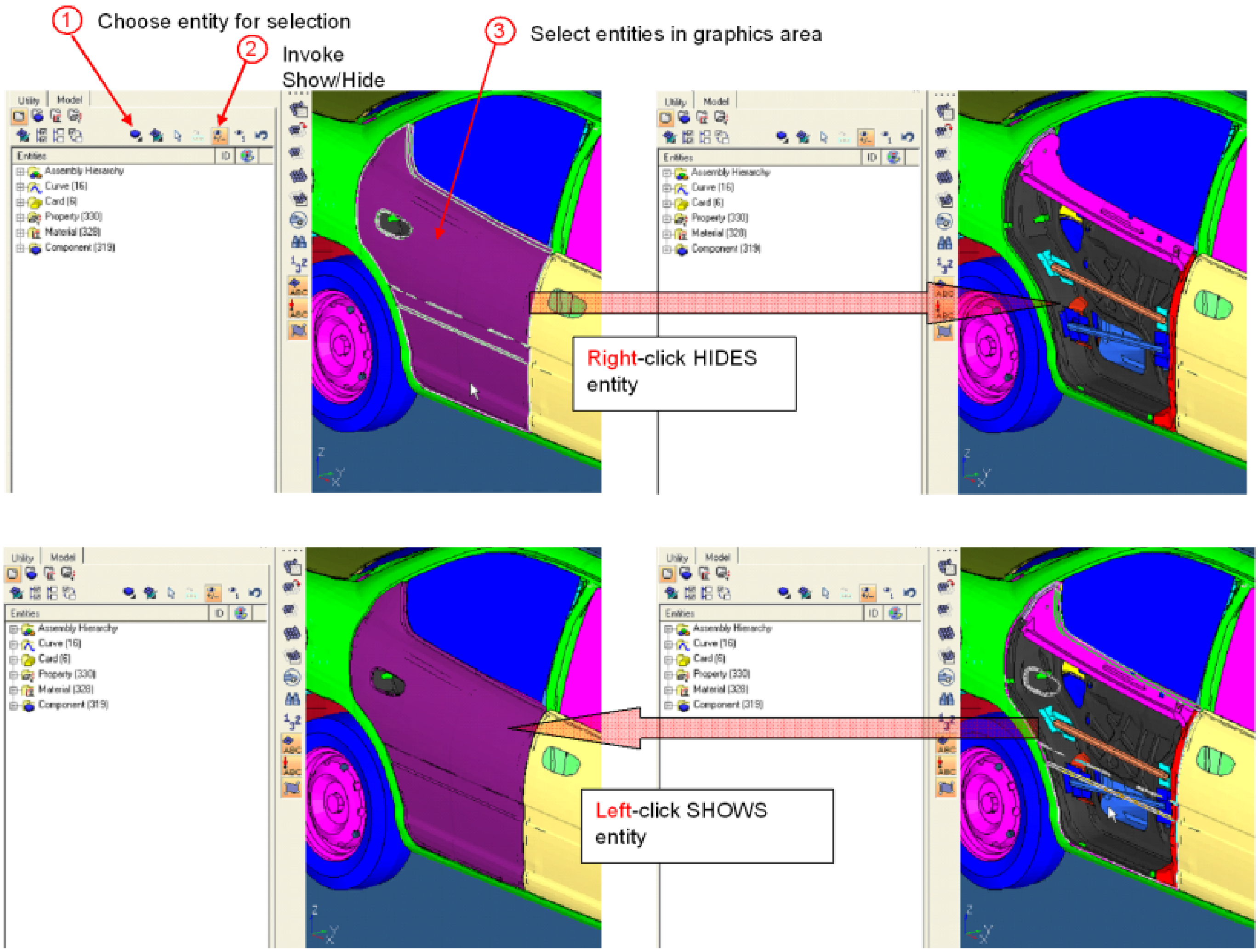


Figure 163:

- Shift + left-click-and-drag uses window selection to turn on multiple entities in the display.
- Shift + right-click-and-drag uses window selection to turn off multiple entities in the display.

Window Selection

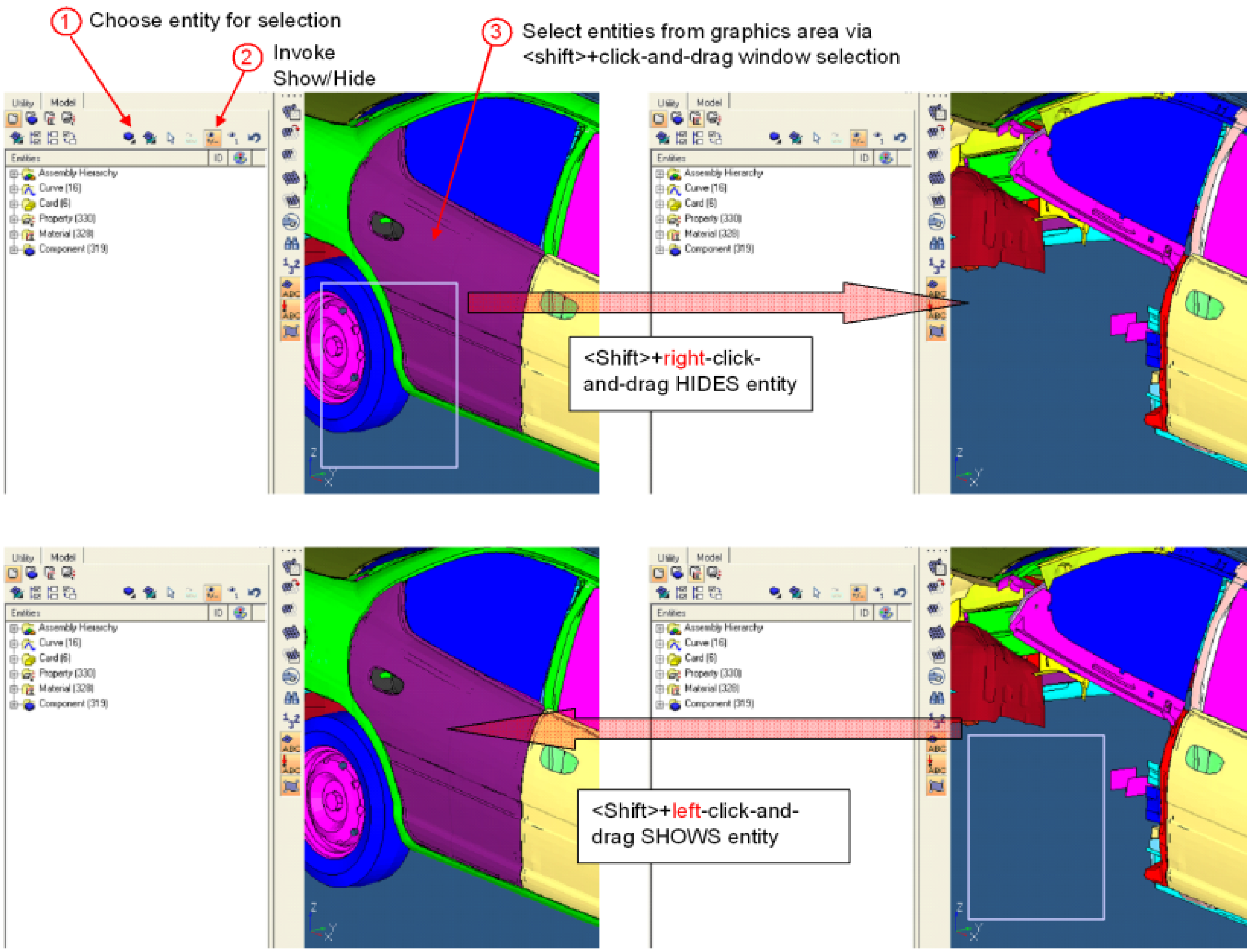


Figure 164:

Pick in Browser

This tool does not function on line items within the browser's tree list. To control the display of the listed entities use All/None/Reverse, the individual local display control inside the tree structure, or the show/hide/isolate functions from the context menu.

isolate that component, while leaving the load collectors untouched in the display. In other words, all of the other components will be turned off, but the isolated component and the load collectors will still display.

> **Note:** Isolate can be used in conjunction with the selector. In such a case, after you have selected the desired entities, clicking Isolate hides everything else except for the selected entities. However, as described above only entities of the chosen type are hidden – so connectors and similar entities will still remain visible.
>
> Using the selector may give you more precise control over which entities to retain – but for simple isolation tasks, direct usage of the Isolate button is generally quicker.
>
> Similar to the selector tool, you can change your selection in the graphics area using the mouse scroll-wheel for the Isolate Shown/Isolate Not Shown tools. To isolate entities that are shown, left-click, hold, and scroll. To isolate entities that are not shown, right-click, hold, and scroll.

Pick in Graphics

In general, use the left mouse button to isolate visible entities, and use the right mouse button to isolate entities that are already hidden (thus turning the hidden ones back on):

- Left-click will isolate the selected entity from those on display (single-click selection).
- Left-click-and-drag will pre-highlight entities that are currently displayed. It will not highlight entities that are currently turned off in the display. Upon release, the pre-highlighted entity will be isolated.
- Shift + left-click-and-drag uses window selection to isolate multiple entities (but only entities currently visible).
- Right-click will isolate entities from all available entities, whether currently on or off in display.
- Right-click-and-drag will pre-highlight an entity that is displayed or turned off from the display in the graphics area. Upon release, the pre-highlighted entity will be turned on and isolated.
- Shift + right-click-and-drag uses window selection to isolate entities from all available entities, whether displayed or turned off from the display.

Single and Window Selection

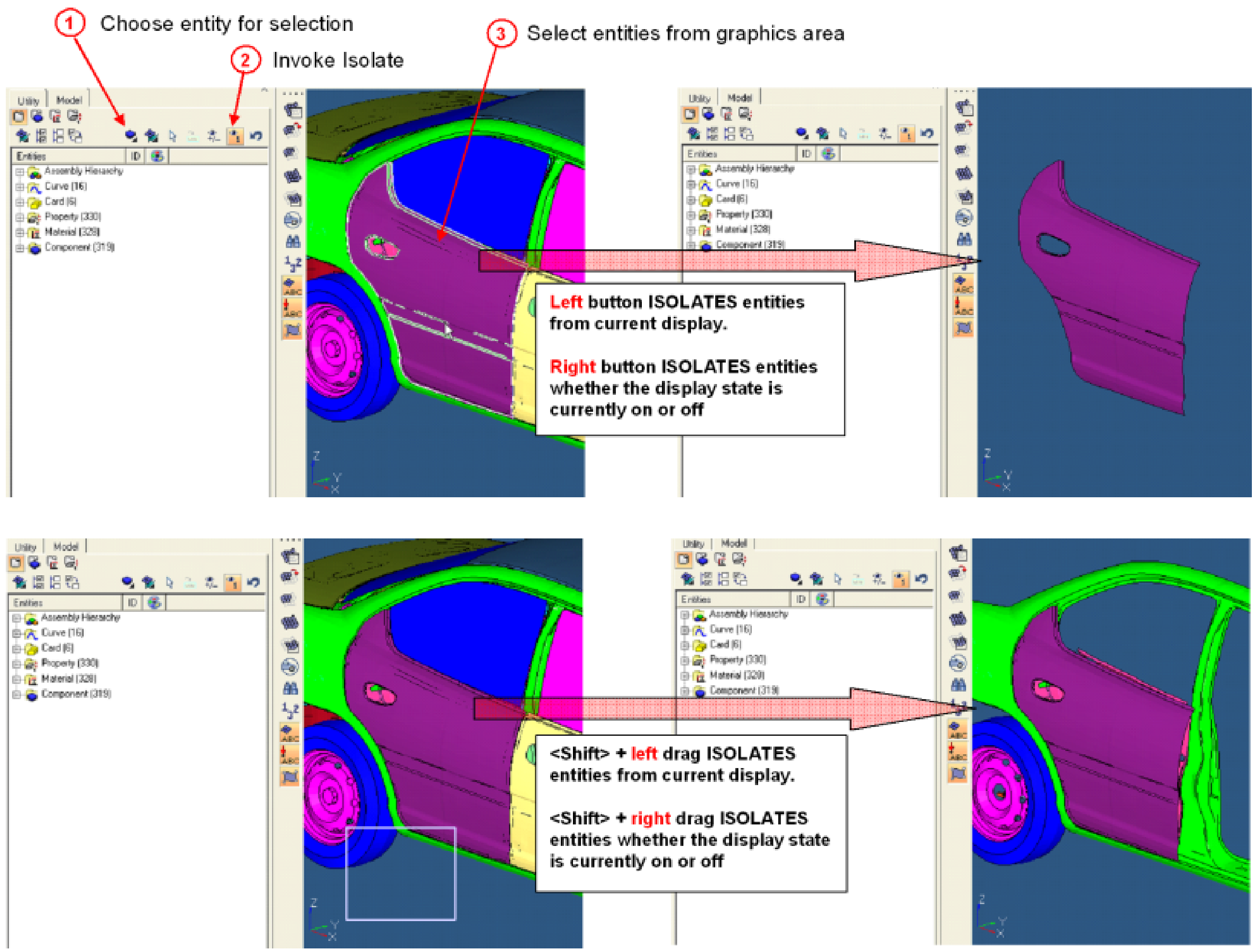


Figure 165:

Note: When using window selection (Shift+click-and-drag), an entity is considered selected if any portion of it falls within the window; you do not need to encompass the entire entity with the window, only a small portion of it. Also remember that only entities of the types determined by the Entity Types and Elements/Geometry buttons will be hidden or revealed.

Pick in Browser

- In Isolate mode, clicking on an entity folder, such as the Components folder, will isolate all components, therefore turning every component on.
- Left-click will select/highlight an entity in the list – the entity is isolated and displayed in graphics area.
- Control + left-click highlights an entity name in the browser and isolates it in the graphics area. Multiple entities of the same type, components for example, can be appended to the selection, thus displaying more than one entity but still hiding all non-selected ones. Selected/isolated entities can be de-selected by Control + left-clicking on them a second time.
- Shift + left-click highlights all entities of the same type, components for example, in the browser between the first click and the most recent click, and displays the selected entities isolated from the non-selected ones in the graphics area. You can use additional Shift-clicks or Control-clicks to modify the selection of displayed entities.

Context Menu

A context menu of actions is available for any selected item in the browser.

To open the context menu, right-click on either an entity folder, an individual entity, or the white-space of the browser.

Review

In Review mode you can review selected entities irrespective of their display state, masked, active state (Entity State Browser), but not outside of the spherical clipping, if enabled.

Review mode is currently supported in the following browsers: Model, Solver, Part, Mechanism, and Entity State.

Review is currently supported for: Assemblies, Beamsection collectors, Beamsection, Blocks, Bodies, Boxes, Components, Configuration, Constrained extra nodes, Constrained rigid bodies, Constraints, Contact surfaces, Control volumes, Cross sections, Design variables, Design variable links, Design Objective Reference, Design Variable Property Relationship, Elements, Groups, Joints, Laminates, Load collectors, Loads, Loadsteps, Materials, Mechanisms, Objectives, Optimization constraints, Optimization responses, Output blocks, Part, Part Assembly, Part Set, Plies, Properties, Regions, Seatbelts, Sensors, Sets, System collectors, Systems, Vector collectors, and Vectors.

Invoke Review Mode

1. Select valid entities to review.

> **Note:** You are only able to review one type of entity at a time.

   - Select entities in the browser.
   - From the action mode tools in the browser, select an entity type and enable the Selector ( ). In the graphics area, select entities.

2. Invoke review mode.
   - Right-click on selected entities in the browser and select **Review** from the context menu.
   - Press **Q** (only when a supported browser is active or the graphics area is active).

> **Note:** When the graphics area is active and you have two browsers which support Review mode open simultaneously in the left and right tab areas, pressing **Q** will invoke Review mode for the entities selected in the last browser you accessed. If the Entity State Browser is one of the two browsers opened, then pressing **Q** will invoke Review mode for the entities selected in the other browser.

Tip:

- Append entities to your selection by left-clicking. Remove entities from your selection by right-clicking.
- Switch your review selection from one entity to another or from one entity type to another by selecting a new entity. You can also switch your review selection from one browser to another when you have two browsers, which support Review mode, open simultaneously in the left and right tab areas. The graphics area will automatically display your new selection.
- In the browser, use the up and down keys to review.

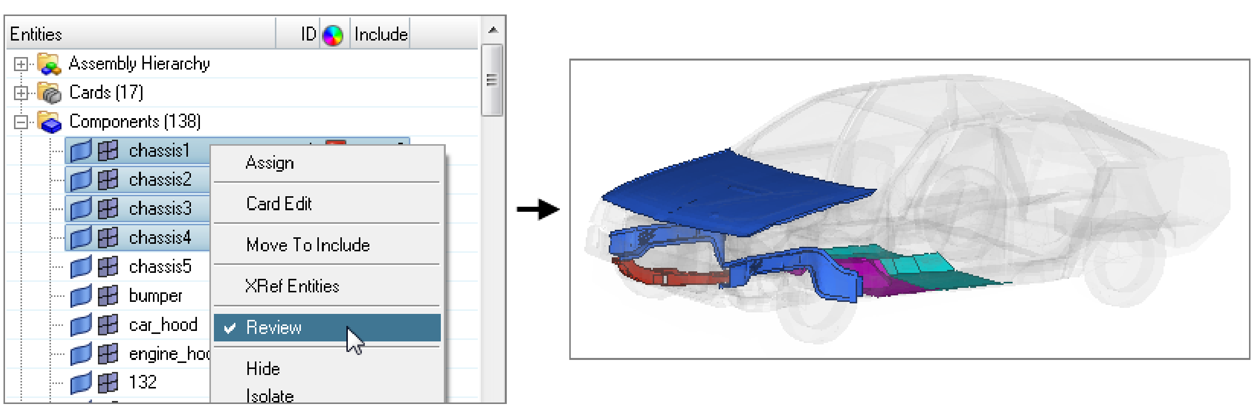


Figure 166:

Disable Review Mode

Manually exit review mode by right-clicking on a selected entity in the browser and selecting **Review** from the context menu, or press **Q**.

When other interaction modes may conflict, Review mode will be automatically turned off.

Warning:

- Review is reset when:
    - Switching the Visualization Color mode to any mode except by Thickness or by Element Quality
    - Changing Visualization FE and Geom styles
    - Switching browsers or browser views
    - Undocking browsers from the tab area
    - Clicking anywhere in a browser that does not support Review
    - Enabling an action mode tool
    - Changing data
- Review is reset and disabled (when Review is on) or disabled (when Review is off) when:
    - Any panel or pull-down menu referring to a panel is opened and remains open
    - Switching the Visualization Color mode to by Thickness or by Element Quality
    - The Spherical Clipping panel is opened

Color of Reviewed Entities in the Graphics Area

Entities being reviewed will appear with their current visualization style in the graphics area, while all other non-reviewed entities will display transparent.

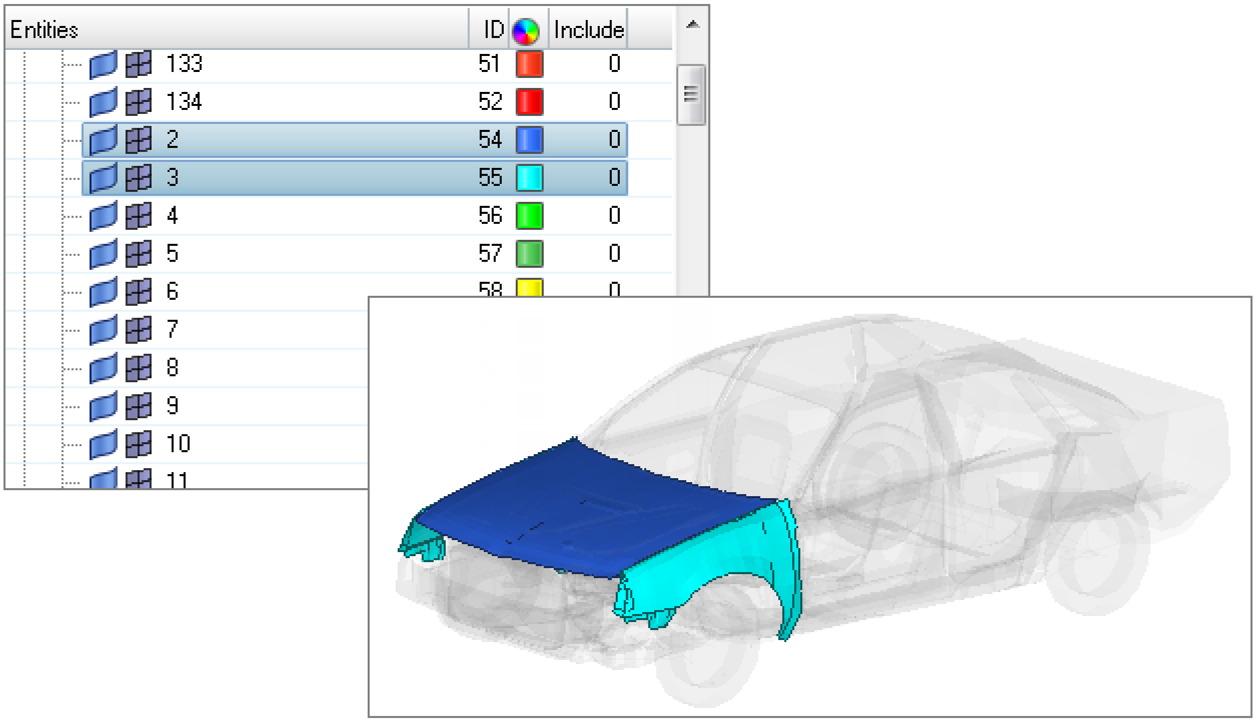


Figure 167: Two Components Selected in Review Mode

Entities being reviewed that do not have a visualization style assigned will display yellow.

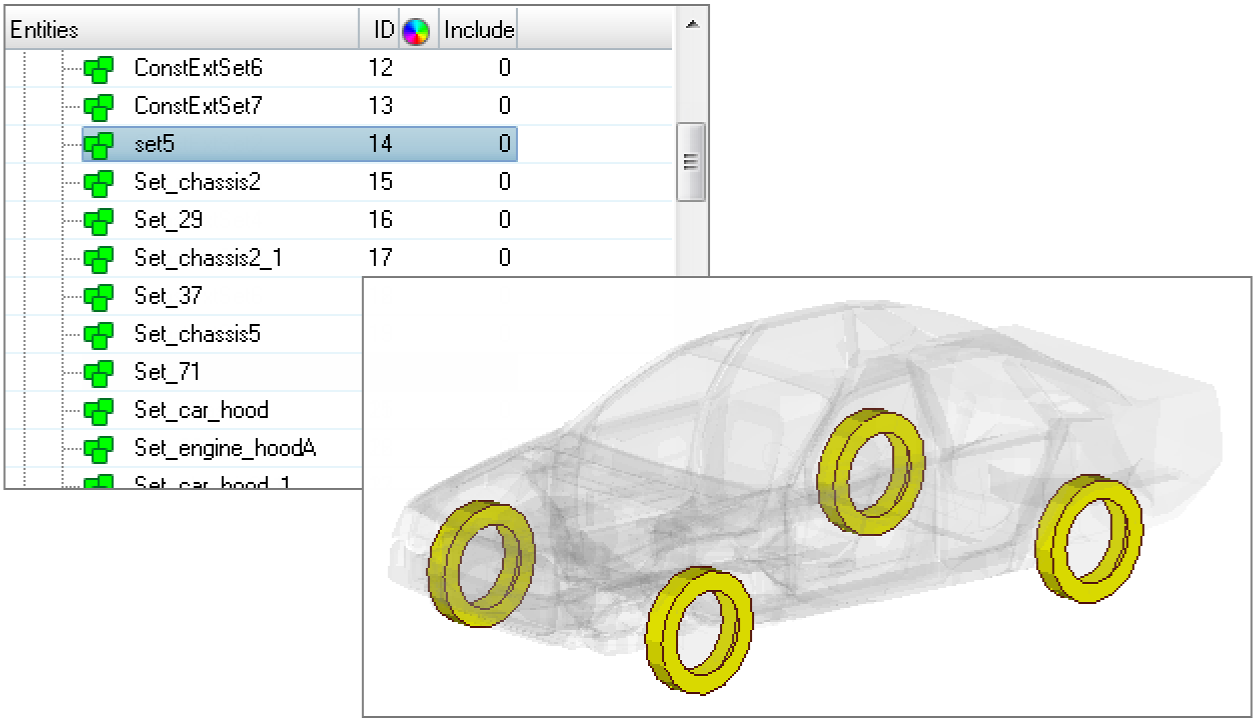


Figure 168: Sets Selected in Review Mode

When a single group entity is being reviewed, the contact pair's master surface will display blue and the slave surface will display red. When there is only one surface definition being reviewed, it will display red.

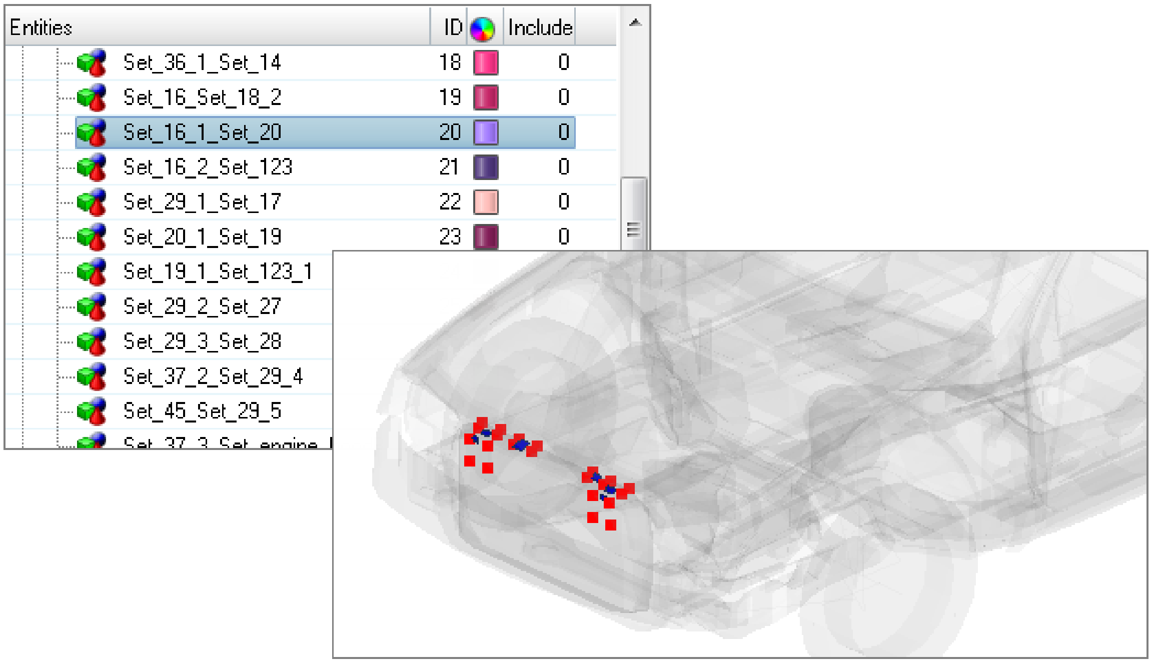


Figure 169: Single Contact Pair Selected in Review Mode

When multiple group entities are being reviewed, they will display in their own entity color.

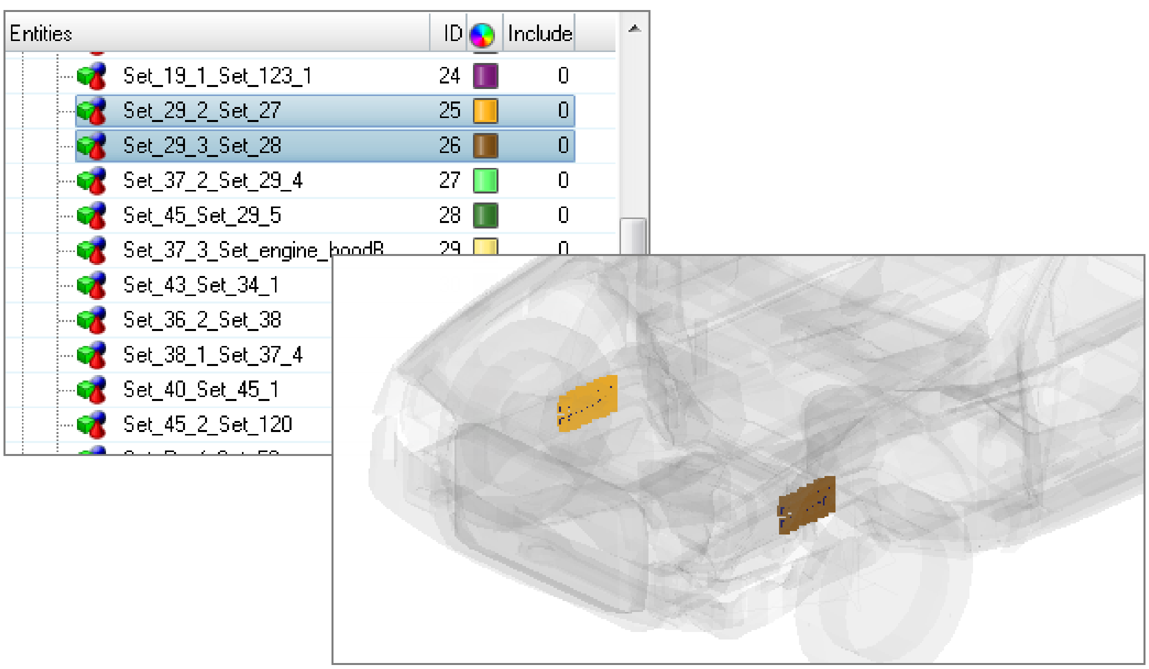


Figure 170: Multiple Contact Pairs Selected in Review Mode

Assembly Browser

Use the Assembly Browser to manage parts and modules during the assembly building process.

This browser can be accessed from the View menu when the Radioss, OptiStruct, Nastran, LS-DYNA and Abaqus user profiles are loaded.

Currently, Bill-of-Material (BOM) file support is limited to PLMXML assembly files from Teamcenter. For more details on the required Teamcenter configuration, supported PLMXML file formats, and supported transfer modes, contact your local Teamcenter support. Once the necessary configuration is made in Teamcenter, you can import the file in HyperMesh for representation management, batch meshing, connectors, and exporting the meshes and solver files.

The Assembly Browser structure is laid out in a table format. The different modules are displayed in the Entities column, and each module's representation, material, PID, and thickness information is displayed in the columns that follow.

The Assembly Browser contains advanced filtering tools to filter BOMs based on their properties, existence of representation, and so on. To display the filtering tools, click .

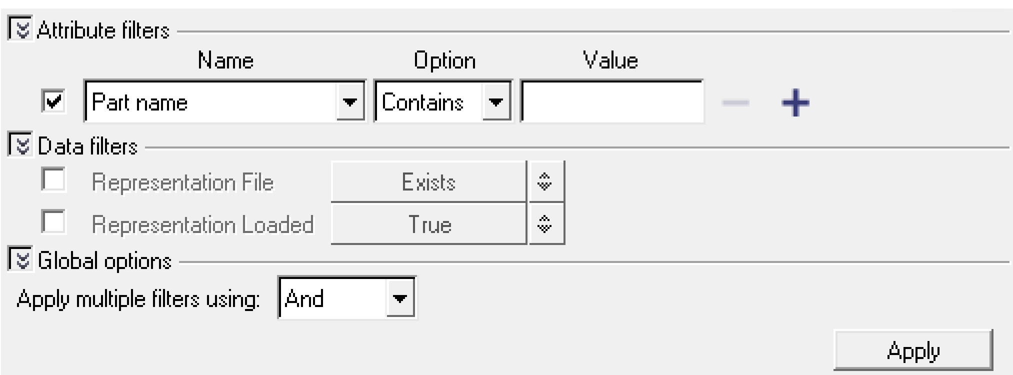


Figure 171:

To filter BOMs, apply the following advanced filters:

Apply Attribute Filters

Use these filters to determine, which Attribute(s) to filter upon.

1. To select which Attribute(s) to filter upon, select the check box next to the desired Attribute(s).
2. From the Name list, select the attribute you wish to filter upon.

   The options available in this list include: Part name, Material, PID, or Thickness.
3. From the Option list, select the one of the following options:
   - **Contains**
   - **==**
   - **!=**

- **REGEXP**

4. In the Value column, specify specific information about the Attribute(s).
5. Optional: To add an additional attribute filter, click .
6. Optional: To remove an attribute filter, click .

Apply Data Filters

Use these filters to determine what data appears in the Assembly Browser.

1. To activate the Representation File filter, select its checkbox.
2. Click the Representation File toggle and select:
   - **Exists**
   - **Does Not Exist**
3. To activate the Representation Loaded filter, select its checkbox.
4. Click the Representation Loaded toggle and select:
   - **True**
   - **False**

Apply Global Filters

From the Apply multiple filters using list, select:

- **And:**
- **Or:**

Assembly Browser Context Menu

Access advanced Assembly Browser options from the context menu when you right-click on a module in the browser.

Load BOM

PLMXML is the exchange format that transports the assembly information between Teamcenter and HyperMesh.

To load a PLMXML (`.xml`) file:

1. Right-click in the Assembly Browser and select **Load BOM** from the context menu.
   The **XML Import** dialog appears.

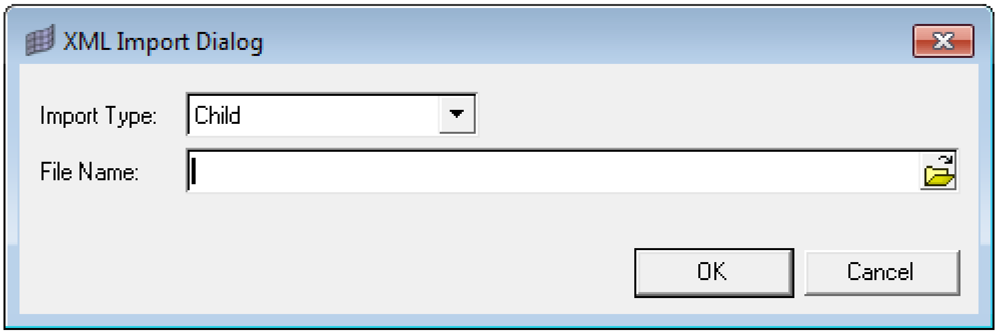


Figure 172:

2. In the File Name field, navigate to your working directory and open the `.xml` file.
3. Click **OK**.
   HyperMesh imports the `.xml` file into the Assembly Browser.

> **Note:** In the Assembly Browser, many operations are recursive, meaning that when you select a module and perform an operation on it, the same applies to all the children modules.

Representations are various versions associated with any module. You can use any of these at a given time. Basically, each representation has a file associated with it, and that file for the current representation will be imported into HyperMesh when you import a module.

A representation cannot have more than one file attached to it.

When you select a representation, it is set as the current representation for that particular module. When that module is imported into HyperMesh, the file associated with that particular representation is imported into HyperMesh. If you want to import a different file, change the representation to the one that contains the desired file and then import that particular module.

Save WIP

To save your work-in-progress data, select **Save WIP** from the right-click context menu.

Once you select Save WIP, HyperMesh saves a UDMXML file in the assembly working directory. This file captures the assembly structure state, including the current representations and all metadata values. After you have saved your work-in-progress to the UDMXML file, you can safely close the Assembly Browser or HyperMesh.

> **Note:** Saving the HyperMesh file by clicking **File** > **Save** from the menu bar will not save the assembly structure or state.

To continue working with a saved UDMXML assembly BOM, load the WIP BOM using the Load BOM process. Select the saved WIP UDMXML BOM instead of the original PLMXML BOM.

Save PLMXML

In order to import the meshed parts into Teamcenter, an updated PLMXML `.xml` file is needed. This file captures any changes that were made to the assembly after it was imported into HyperMesh. If the original `.xml` file contained only CAD parts with thickness and material attributes, and if you meshed these parts in HyperMesh and made changes to the thickness for certain parts, then the delta PLMXML file will capture the changes made.

To create a delta PLMXML file and a monolithic HyperMesh or solver file, select the **Module Model** module and then select **Save PLMXML** from the right-click context menu.

The type of monolithic file HyperMesh creates is determined by the settings you selected within Teamcenter prior to exporting the original PLMXML file.

Teamcenter uses the delta PLMXML file to update metadata and files when you upload the updated assembly file. You cannot use the delta PLMXML file to save work-in-progress information, or load it into the Assembly Browser.

> **Note:** You cannot save the HyperMesh file by clicking **File** > **Save** from the menu bar because it will not save the assembly structure or state. To save your work-in-progress data, select **Save WIP** from the right-click context menu.

Set Rep

1. To change the representation of a module, right-click on the representation and then select **Set Rep** from the context menu.

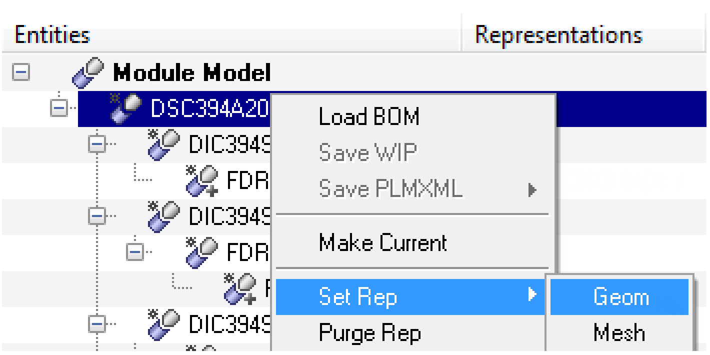


Figure 173:

2. Select a representation category.
   - Choose **Geom** to set the geometry as the current representation. The type of CAD that is set as the current representation is based on the order of preference as set in the config file, which is provided by Teamcenter.

- Choose **Mesh** to set the mesh file as the current representation. The last representation in the list of mesh representations is set to current.

A module can have multiple representations associated with it. The representations are basically categorized into two types, Geom and Mesh. The Geom representations have CAD files associated with them and Mesh representations have mesh files associated with them.

Generally, CAD representations are provided along with the BOM file and the representations are added based on a failing order. The format that is on top of failing order is associated first, and so on.

Most of the time, the mesh representations are created after operations likes Batchmeshing, mesh editing, penetration checks, and so on. The last representation of Mesh type is generally the latest representation and is of primary concern.

When you select **Set Rep** > **Geom**, HyperMesh searches for a list of Geom representations and sets the first one in the list as the current representation. This is in alignment with the failing order as the representation on top of the failing order is set as current.

When you select **Set Rep** > **Mesh**, HyperMesh searches for a list of Mesh representations and sets the last one in the list as the current representation. This way, you set the last created representation as current.

There might be representations that do not have any files associated with them, which may come with a `.xml` file. HyperMesh ignores these representations when searching for the appropriate Geom/Mesh representations. In a case where a representation of a selected type is not present, HyperMesh does not change the current representation. For example, if the current representation is of type Mesh and there is no Geom representation associated with the module, when you select **Set Rep** > **Geom**, the current representation does not change and HyperMesh leaves it as type Mesh. If the current representation does change, HyperMesh purges the module, as well.

Purge Rep

To remove all of the contents under the modules from the current HyperMesh session, select **Purge Rep** from the right-click context menu.

Before you are able to remove all of the contents under a module, the **Confirm delete** dialog appears asking if you want to purge the selected modules.

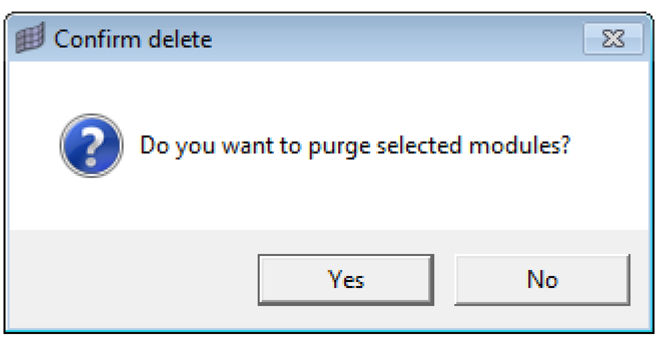


Figure 174:

HyperMesh will still maintain the representation in the hierarchy, and you can import when desired. When you purge a module, all of the changes you have made to the contents of the module will be lost permanently. HyperMesh will not automatically update the files in the current representation with the updated content. To retain the changes, you must save the modules as a HM file (or Solver file) before purging.

Import Rep

To import the associated representation into HyperMesh, right-click on a representation and then select **Import Rep** from the context menu.

If there are children modules in the module you select, then all of the children will be imported. In general, the operations in the right-click context menu work on everything under the selected module.

Set Param-Criteria

1. To set individual mesh parameter and criteria for selected module(s), select **Set Param-Criteria** from the right-click context menu.
   The **Select Mesh Parameters & Criteria** dialog appears.

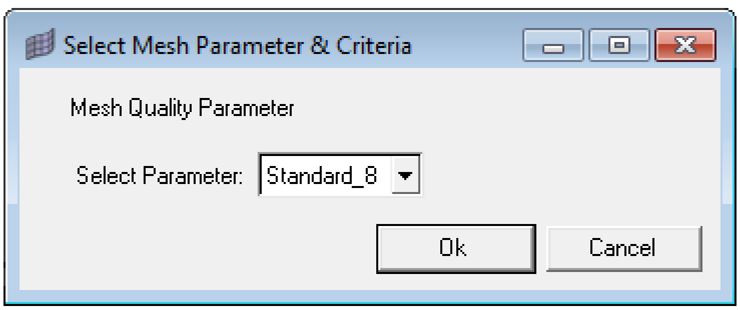


Figure 175:

2. From the Select Parameter list, select a parameter to assign to the selected module.

Batchmesh

1. To batchmesh a module or modules, right-click on a module or modules and then select **Batchmesh** from the context menu.
   The **Batchmesh Representation** dialog appears.

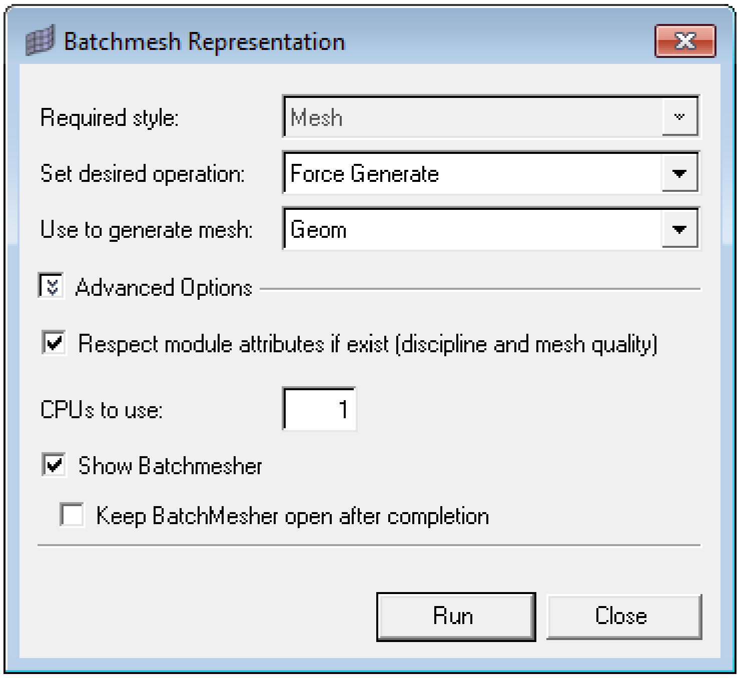


Figure 176:

2. Set the desired operation.
   - Choose **Generate, if not present** to create a new mesh for the given module using the selected parameter and criteria files, as long as a mesh for the given module and selected parameter and criteria files does not already exist.
   - Choose **Force Generate** to create a new mesh for the given module using the selected parameter and criteria files, even if a mesh for the given module and selected parameter and criteria files already exists.
3. Make a selection for generating mesh.
   - Choose **Use Current** to use the current representation as input to the batch mesher for creating the new mesh.
   - Choose **Geom** to use the geometry representation as input to the batch mesher for creating the new mesh.
   - Choose **Mesh** to use the existing mesh representation as input to the batch mesher for creating the new mesh.
4. Click **Advanced Options** to edit the following:

Option	Description
Respect module attributes if exist (discipline and mesh quality)	If module attributes exist and you would like to leave them as is during batchmeshing, select this checkbox.

Option	Description
CPUs to use	In order to run the Batchmesher faster on a system with multiple processors, enter the number of CPU values. This will run that many instances of Batchmesher.
Show Batchmesher	To show the Batchmesher GUI instead of running the batchmesh in the background, select this checkbox.
Keep Batchmesher open after completion	To keep the Batchmesher open after the batch meshing process is finished, select this checkbox.

5. To start Batchmesher, click **Run**.
The following messages appear during the batch mesh process:

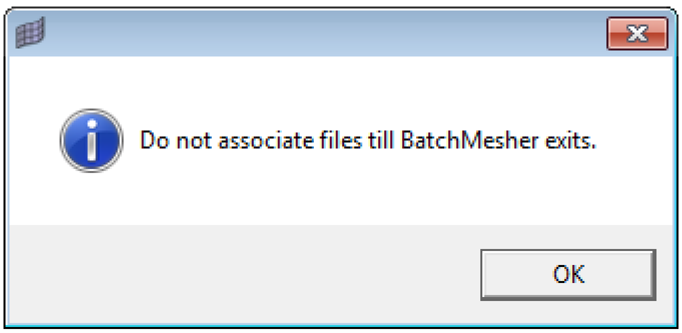


Figure 177:
Message displayed when Batchmesher is running in GUI mode

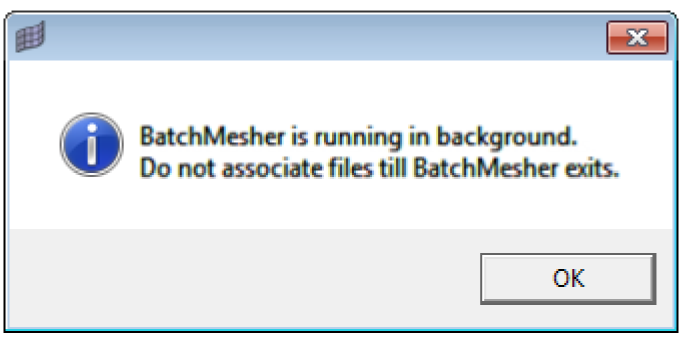


Figure 178:
Message shown when Batchmesher is running in background

Associate Mesh Files

To associate mesh files for modules shown in the Assembly Browser, right-click on a module and then select **Associate Mesh Files** > **All files** or **Latest only** from the context menu.

- Choose **All files** to assign all files to the module, if several representations exist for a particular module.
- Choose **Latest only** to set the latest file, based on time stamp, as the only representation associated with the module.

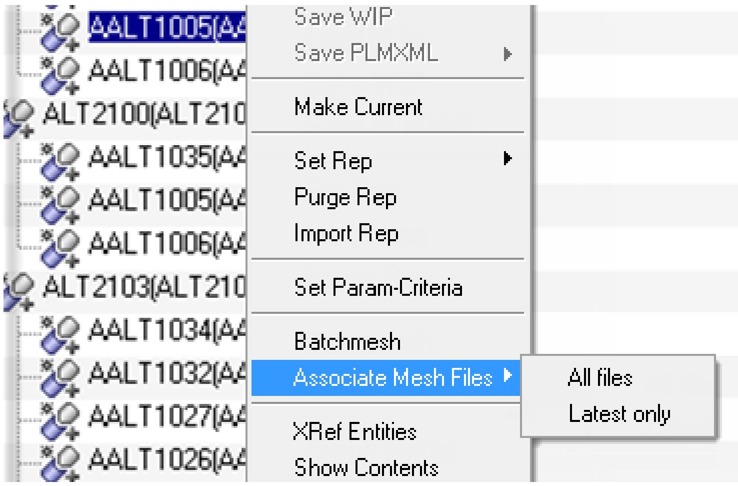


Figure 179:

Associate mesh files search the mesh files based on their CAD file name. If the module does not have any CAD files associated with it, then no file will be associated. When HyperMesh is associating mesh files, a CAD representation is necessary. For example, when the initial PLMXML file contains only solver/ hm files, nothing can be associated.

The associate mesh file also searches the mesh files based on their extension and CAD file name. If HyperMesh associates the wrong mesh file, select **Associate Mesh Files** > **All files** and then locate the appropriate file.

Show Contents in Module

To display all of the contents in a selected module, select **Show Contents** from the right-click context menu.
HyperMath opens a new Content Browser, which displays all the contents in the selected module.

Export a Model

To export a model for a given module, select **Export Model** > **HM File** or **Solver File** from the right-click context menu.

- Choose **HM File** to save all of the information about a particular module into an HyperMesh file and associates that file as a representation for the selected module.
- Choose **Solver File** to save all of the information about a particular module into a Solver file and associates that file as a representation for the selected module. The file format is determined by the current user profile.

You are only able to save a file when entities are organized under the module and the current representation of the module is not empty. After HyperMesh creates the new file, it is set as the current representation to the module.

Show, Hide and Isolate Modules

To show, hide, or isolate a module in the graphics area, right-click on the module and then select **Show**, **Hide** or **Isolate** from the context menu.

Show will bring the module to the display; Hide will hide the module; Isolate will turn off all other modules and display the selected one.

> **Note:** This operation is recursive, meaning that when you select a module and perform an operation on it, the same applies to all the children modules.

Realize Connectors

1. To import and realize the connectors from a `.mcf` file, click in the Assembly Browser.

   The **Mcf Import** dialog opens.

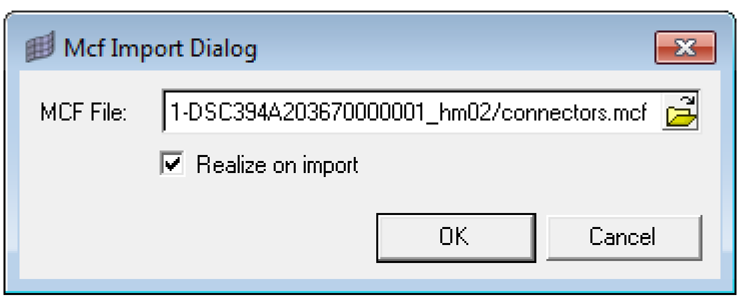


Figure 180:

2. In the MCF File field, navigate to your working directory and open the connector file.

> **Tip:**
> - To automatically realize the connectors when you import the connector file, select the **Realize on import** checkbox.
> - To view the connectors that you have imported, click **View** > **Connector Browser** from the menu bar.

Show All Attributes in a Module

To display all of the attributes in a selected module, select **Show All Attributes** from the right-click context menu.
HyperMesh opens a dialog that displays all the attributes in the selected module.

Collapse and Expand All Modules in the Assembly Browser

To collapse or expand all of the modules models in the Assembly Browser, select **Collapse All** or **Expand All** from the right-click context menu.

To close all of the modules models in the tree structure and only display the top-most module, select **Collapse All**. To open all of the modules models in the entire tree structure and expose every item nested at every level, select **Expand All**.

Configure the Assembly Browser

1. To change what columns display in the Assembly Browser, select **Configure Browser** in the right-click context menu.
   The **Column Visibility** dialog opens.
2. From this dialog, select the check boxes of the columns you wish to display.

Connector Browser

Use the Connector Browser to view and modify connectors in the current model.

To invoke the Connector Browser in the tab area, click **View** > **Connector Browser** from the menu bar.

The Connector Browser consists of:

- The Link Entity Browser, located in the first pane of the tab, displays information about all of the linked entities in the model and Connectors groups.
- The Connector Entity Browser, located in the middle pane of the tab, displays a tree view of all the connections the model contains. HyperMesh organizes and displays the connectors in different folders based on their respective realization type. The names of the folders are obtained from the FE configuration names that are specified for respective solvers in the `feconfig.cfg` file.
- The Connector Entity Editor, located in the last pane of the tab, displays attributes assigned to the connector(s) selected in the Connector Entity Browser.

•

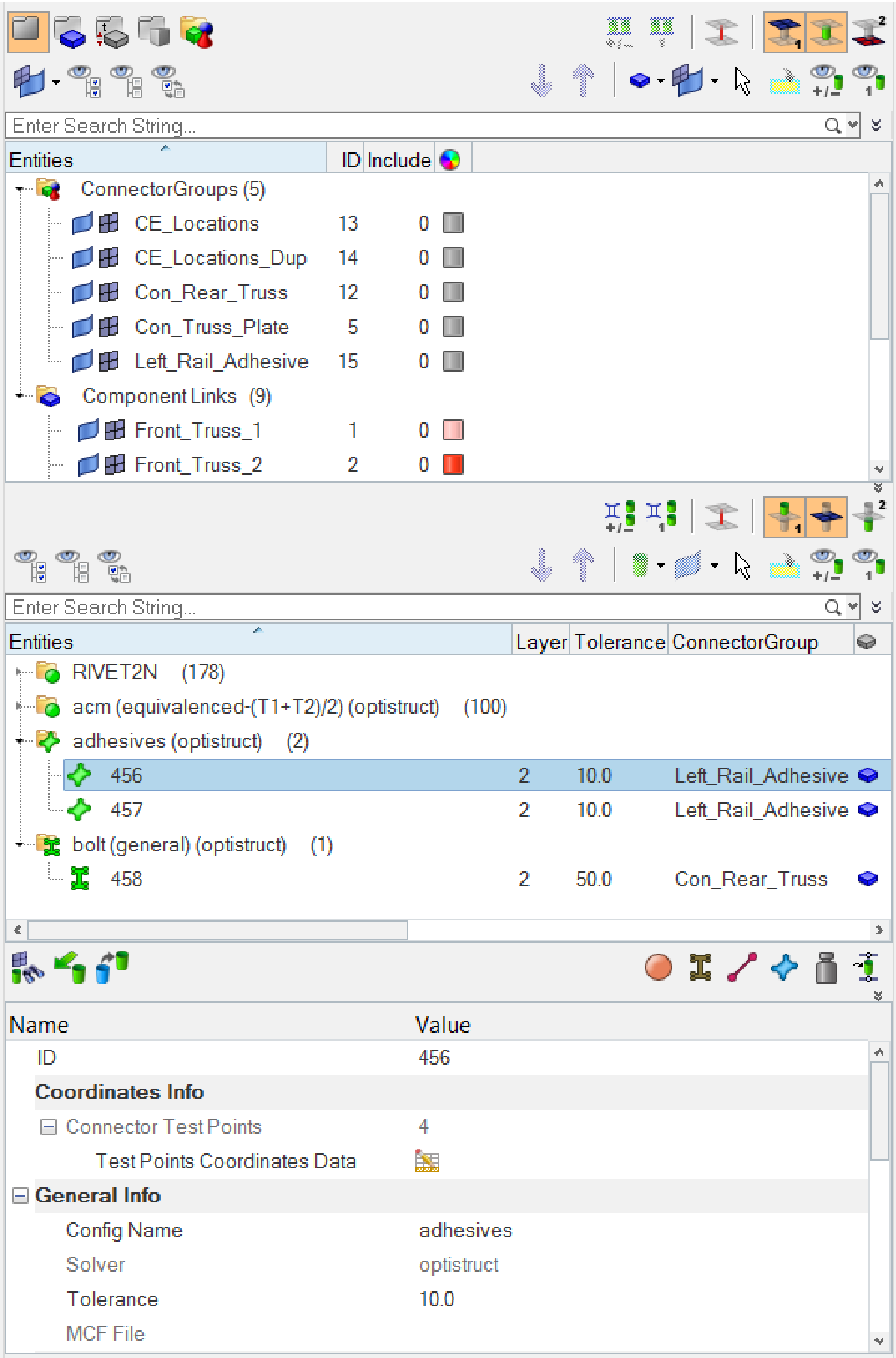


Figure 181:

Use the Connector Browser to:

- Add Links
- Remove links
- Update links

- Find Connectors from Parts or Links
- Find Connectors from Realizations
- Find Links from Connectors

The actions performed in the Connector Browser, such as selecting, highlighting, showing, or hiding supported entities, are reflected in the graphics area. For example, when you select an entity in the Connector Browser, HyperMesh outlines it in white in the graphics area.

The browser and the HyperMesh database synchronize with each other to ensure that all the changes made to the connector or the supported entities information in the database are reflected correctly in the browser at all times.

The browser can be configured to only display the information you wish to see. The current configuration is always saved, so that when the browser is opened in the future it will contain the same configuration as the last time that it was used.

Connector Browser Tool Sets and Context Menus

Use the different tool sets and context menus in the Connector Browser to access additional options. The Link Entity Browser and the Connector Entity Browser each contain their own set of tools.

The Link Entity Browser consists of a context menu, global display tool set, and action modes tool set (rimmed in blue below). Click the toggle buttons to access additional view options (red) and advanced action options (yellow). A query builder allows to filter down the links to be listed (green).

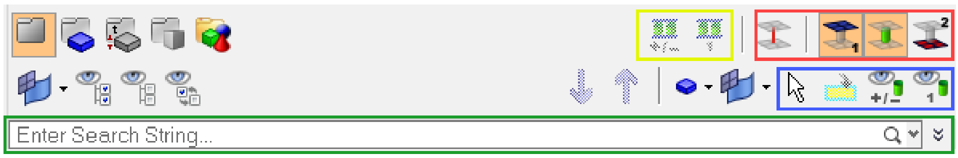


Figure 182:

The Connector Entity Browser consists of a context menu, global display tool set, and action modes tool set (rimmed in blue below). Click the toggle buttons to access additional view options (red) and advanced action options (yellow). A query builder allows to filter down the connectors to be listed (green).

Figure 183:

Use the Utility tools, at the bottom of the Connector Browser, to access various Connector panels, as well as export connector data in XML format.

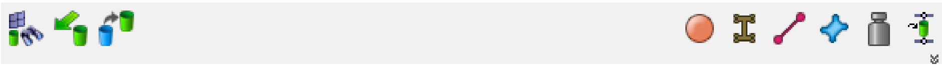


Figure 184:

Link Entity Browser

The Link Entity Browser displays all of the links that are connected using the connectors in the model.

Predefined browser view modes available in this browser include: Common Top view, Component Link view, Property Link view, Part Link view and Connector Group view.

Figure 185:

> **Note:** If a connector uses an element-type link, the component containing the element is also listed in the Link Entity Browser. Isolating this link displays the whole component instead of the single element. In the browser configuration window, activating the **Show Primary Links Only** option, for example, will list only real component links in the component link view.

The information displayed in this browser can be used to quickly locate certain links and appropriate connectors in the graphics area.

The global display tools, action mode tools, and context menu options (show, hide, isolate, and isolate only) work exactly as they do in the Model Browser, when all of the view options are inactive (contain a grey background). The only difference is that they will only work on component, property, and part links.

When one or more of the view options are active (contain an orange background), the show, hide, isolate, and isolate only options behavior and appearances will change. To indicate a change in their behavior, a connector symbol will appear in the icon.

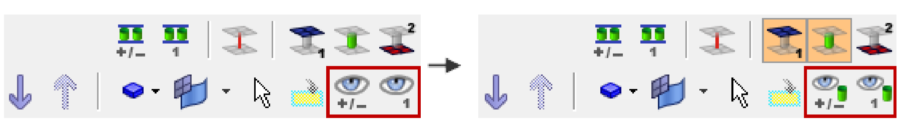


Figure 186:

The action mode tools take into account the connector-link-relations, and the display results depend on the active view options. Therefore, parts of the model are separated into the following three categories that are based on the link selection:

1st link entity
: Selected links

linked connectors
: Connectors which reference at least one of the selected links

2nd link entity

Links which are referenced by the linked collectors

> **Note:** In this case, connectors and their realizations are treated as being a separate category from their links, in order to prevent unpredictable cross-references.

HyperMesh only uses this type of categorical separation when one of the view options are active and you select the show, hide, isolate, and isolate only options from the context menu or from the action mode tools. No other functionality uses this categorization at all.

The base features in the Link Entity Browser can be changed from the Link Entity Browser configuration window.

Action Mode Tools

Use the series of action mode tools on the right-hand side of the Link Entity Browser to control both entity selection and the display of the model in the graphics area.

Figure 187:

The behavior of the action mode tools depends on the settings of the view options. When all of the view options are inactive (gray background), the core behavior of the action mode tools work exactly as they do in the Model Browser. If you select one or more of the view options, the behavior of the show, hide, isolate, and isolate only options change and a connector symbol appears in the icon. In this mode, you are able to select entity links in the graphics area instead of entities. The entity links you are able to select depends on the view option you select.

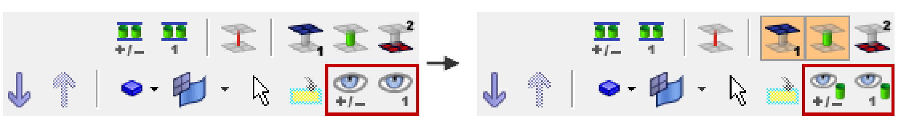
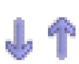
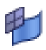


Figure 188:

The table below explains the different action mode tools.

Icon	Description
	Use the next and previous tools to cycle through the selected links in the Link Entity Browser. To activate these tools, select link entities in the browser. These tools will be especially helpful to you when not all of the selected links can be seen in the partial view of the list.
	Use this tool to select a type of entity that the rest of the action mode tools will act on. For example, when you set this tool to "components", the Selector tools (described below) will only be able to select or deselect component links.

<table>
<tr><th>Icon</th><th>Description</th></tr>
<tr><td></td><td>This icon is always disabled and set to the entity type that is defined by the view mode you have selected.</td></tr>
<tr><td></td><td>Use this tool to determine whether the rest of the action modes tools will act on:

 elements only
 geometry only
 both

If you select elements only, you can still perform actions on connectors (geometric entities) by selecting their link entities. However, any actions taken, such as Isolate, will only affect the entity types specified by this tool. In the case of geometry only, only the connectors with at least one link state defined as "geometry" are taken into account.
Since you can define connectors as linking elements, surfaces, or a combination of both, this tool affects actions taken on them in a similar manner. For example, if you set the tool to "elements only" but select a connector that only connects geometry, the actions you might take on that connector will not affect it. However, if the connector links both geometry and elements, then any actions you might take will still apply to the connector due to the element links. </td></tr>
<tr><td></td><td> Use the Selector tool to interactively select and find any type of supported link entity in the browser or graphics area. For example, when you select a link entity in the graphics area, HyperMesh highlights it in the browser. This is an efficient way of selecting and finding multiple link entities at once.
You can use this tool in conjunction with the show, hide, isolate, and isolate only action mode tools, as well as with the show/hide connectors between and isolate/isolate only connectors between advanced action tools. To use the Selector tool in conjunction with the other tools listed above, select the link entities in the browser or graphics area and then click the desired action mode tool.
Note: The advanced action tools are only active when you select at least two link entities.
In other respects, the Selector tools behaves exactly as it does in the Model Browser. </td></tr>
<tr><td></td><td>The Add to Panel Collector is a function whereby the browser can be used to select and add entities to the panel collectors within HyperMesh. This is an alternative method to using the advanced selection capabilities already available in each panel collector's extended entity selection menu. This button only becomes available when you have a HyperMesh panel open that includes at least one entity collector.</td></tr>
<tr><td></td><td>Use the Show/Hide tool to display or hide selected entities. The behavior of this tool will depend on the view option settings.</td></tr>
</table>

Icon	Description
	Click this option to activate it, and then select the desired links in the graphics area. HyperMesh hides the selected links and any other entities found by the view option settings. **Note:** If you are selecting from the graphics area, this option will only work on visible links.
	Use this option to isolate/isolate only selected entities. The behavior of this tool will depend on the view options settings. Click this option to activate it, and then select the desired link entity in the graphics area. **Isolate** Displays only the selected entities which match the view option toggles, turning their display state to on and turning all other entities of the same type off. Isolate works locally within a specific entity type – for example, if component(s) are isolated, then all display states of other displayable entities, such as load collectors, remain untouched. **Isolate Only** Works like Isolate, except that it also affects entity types different from the matching, selected entities. Thus, it turns off ALL displayable entities, regardless of type, except for the selected one(s) that match the view option settings.

View Option Toggle Buttons

The Link Entity Browser is a tool to easily examine the connections between the different parts of a model. One strategy to do that is to start the investigation on one part or a certain group of parts, so each action in the Link Entity Browser starts with a selection of parts. Parts can be considered as links.

These view options affect the Link Entity Browser action modes tools, and thus determine the entities that display when you select and then show, hide, or isolate a link.

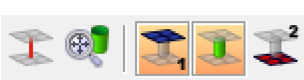


Figure 189:

Each button is modal, that is, you click it once to activate it, and click it again to deactivate it. Active buttons remain active until you specifically deactivate them, so you do not need to worry about them "resetting" after you perform an action such as isolate.

Figure 190: Active

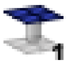


Figure 191: Inactive

The following model illustrates the effects of these search options.

> **Note:** The highlighted component; this component is the starting point for the following examples.

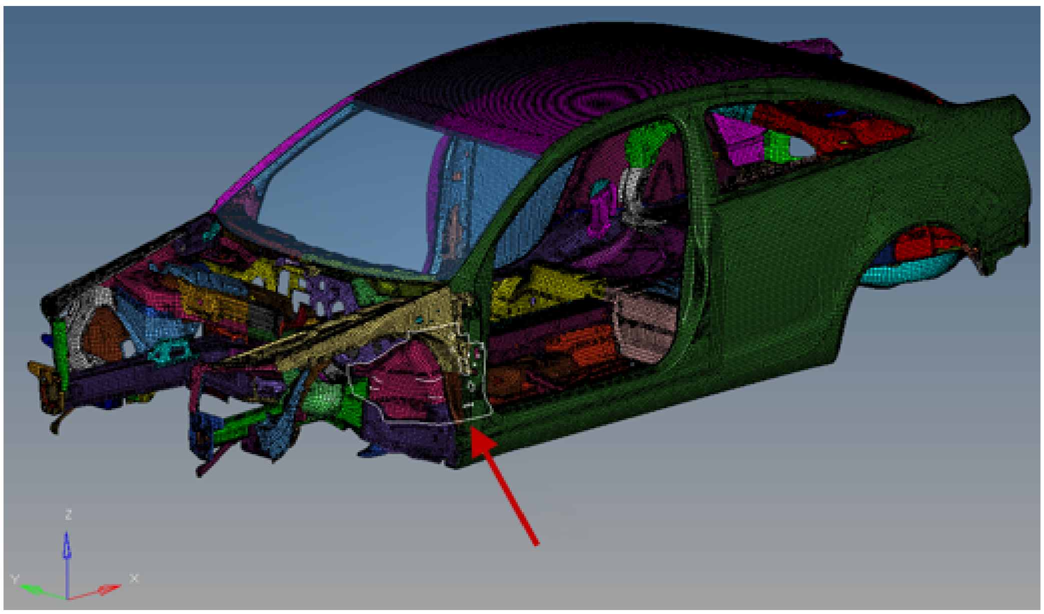
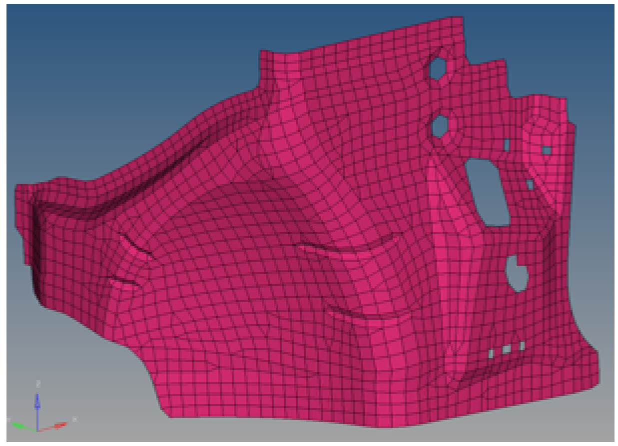
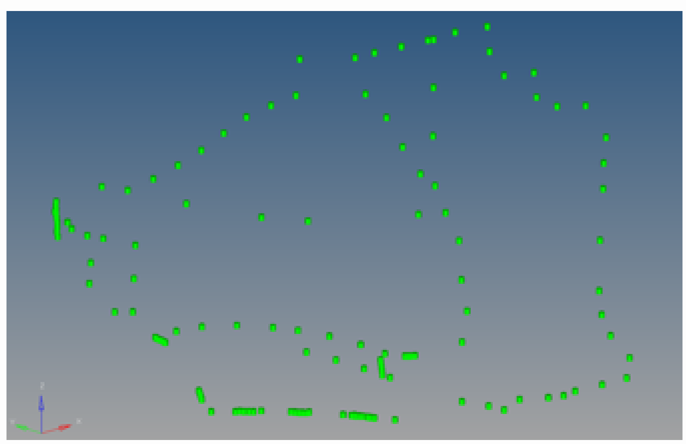
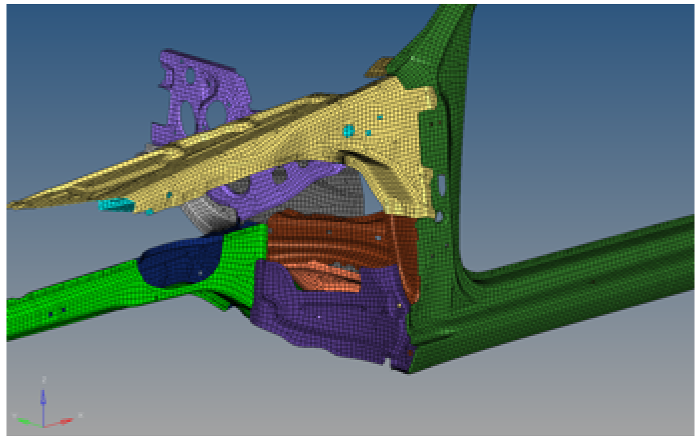
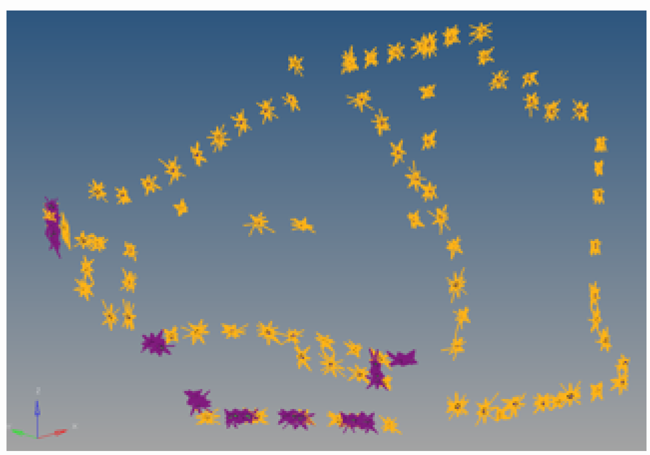


Figure 192:
Model provided courtesy of Audi.

In each example below, the selected component has been isolated using only the relevant view option.

Option	Name	Effect
	1st link entity	The selected link entity can always be seen as the 1st link entity, even if the selected part is not referenced by any connector at all. If the 1st link entity view option button is active, the selected links will be taken into account for the action regardless. This means that in case of the isolation, all selected links will be isolated.

Option	Name	Effect
		 *Figure 193:*
	Linked connector entity	If this linked connector entity view option button is active all connectors linked to the 1st link entities will be taken into account for the action. It does not matter if the 1st link entity view option button is active or inactive, its connectors will still be located. This determination has nothing to do with any display states. This means that in case of the isolation, only the connectors which are referenced by the selected links (1st link entity) are isolated.  *Figure 194:*
	2nd link entity	If the 2nd link entity view option button is active, all link entities referenced by the determined connectors except the (selected) 1st link entities will be taken into account for the action. It does not matter if the 1st link entity or the linked connector entity view option buttons are active

Option	Name	Effect
		or inactive; this determination has nothing to do with any display states. In the case of isolation, this means that only the links that share connectors with the selected entities are isolated.  *Figure 195:* *The selected link is hidden because 1st link entity was not active.*
	Realization	When this option is active, all realizations belonging to the determined linked connector entities will be taken into account for the action.  *Figure 196:*

Cumulative Effect of Multiple Options

These options work accumulatively, for example, when both the 1st link entity and 2nd link entity buttons are active, then selecting and isolating a component link displays both it and all of the components connected it. If you had realization, 1st link entity, and 2nd link entity active when you isolated the same component link, then the model would display all of the component links connected to the selected one, as well as graphical representation of the realizations of each connector linking those component links together.

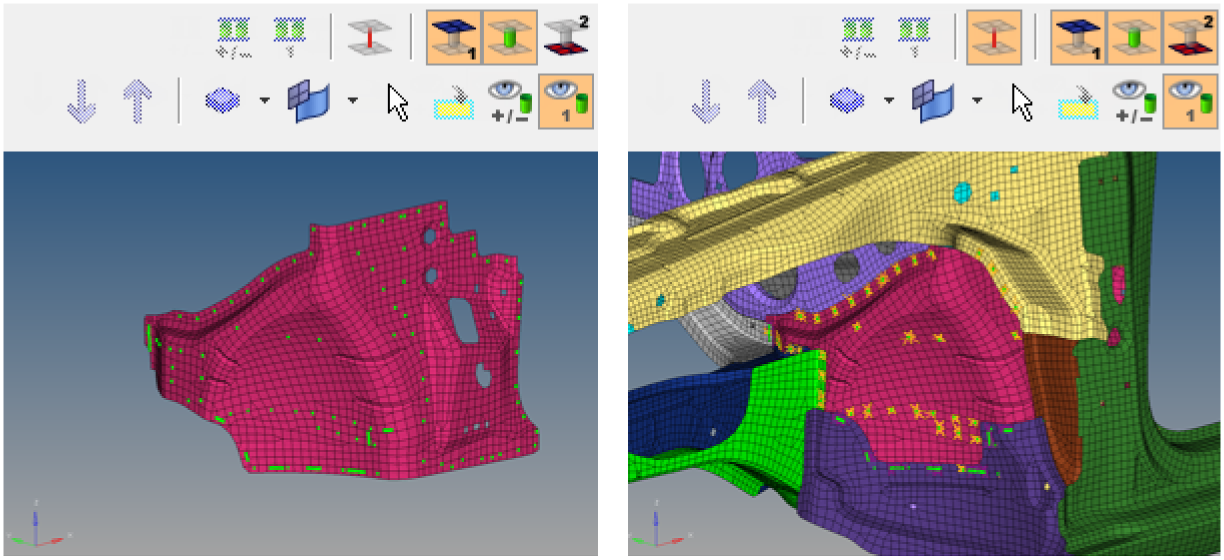


Figure 197:

The first image shows an isolated view with only 1st link and connectors. The second includes 1st and 2nd links, connectors, and realizations.

Using the 2nd Link Option to Expand the Selection

You can use the 1st link and 2nd link options together to gradually add more and more links to your viewable model by starting with a small area, such as a single supported entity, and then selecting additional supported entities. The example below starts with a single component that has been isolated using the 1st link and linked connector entity options. Then, using the 1st link, linked connector entity, and 2nd link options, the Show action mode tool is activated. In each subsequent image, one component (highlighted) is selected to be shown. Because the 2nd link option is active, all components connected to the selected one are revealed.

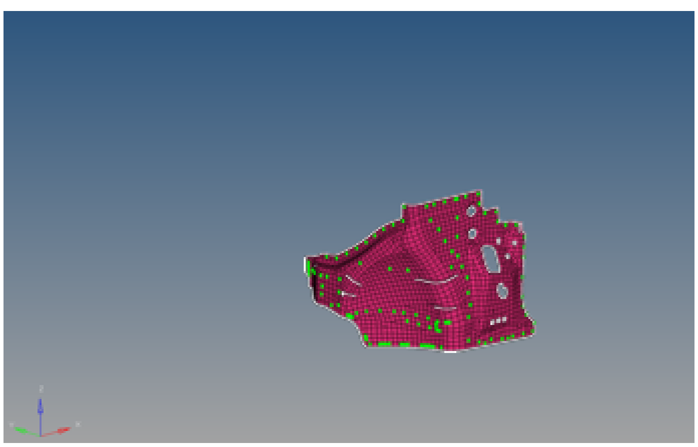


Figure 198:

The initial isolated component is selected.

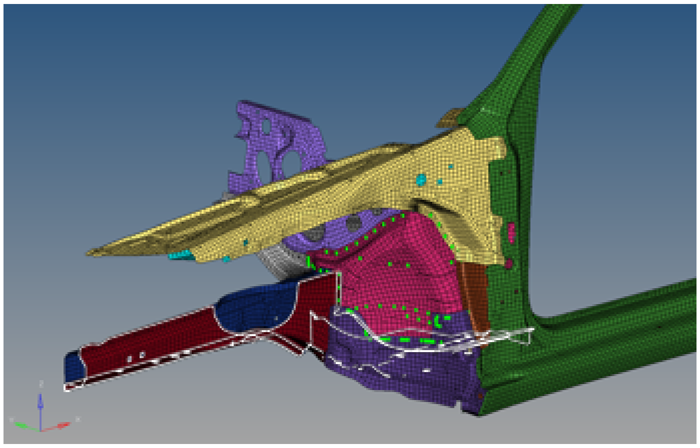


Figure 199:
All entities attached to the component are revealed. Another is selected.

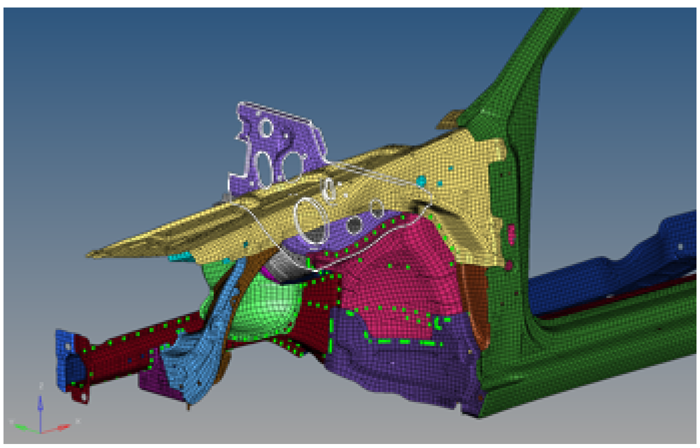


Figure 200:
Link entities attached to the selected one are revealed. A third is selected.

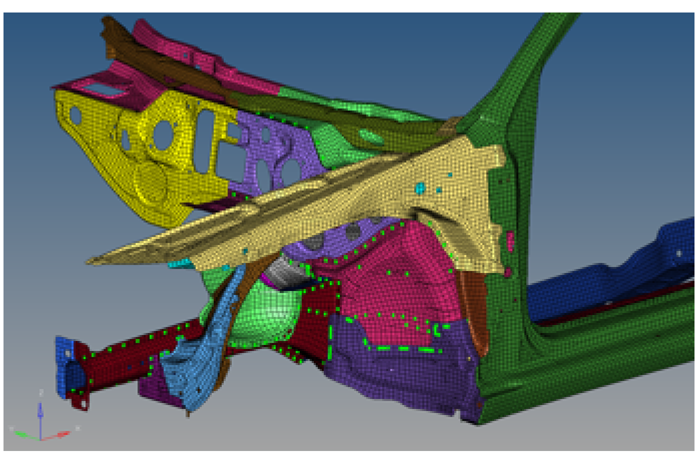


Figure 201: Entities now have direct connection to the original part, display.

Advanced Option Buttons

Located adjacent to the view option toggle buttons, these two buttons allow you to show, hide, isolate or isolate only connectors shared by two or more link entities, by picking the links, or other supported link entities, that they connect. These advanced actions then select the connectors referenced by the selected link entities and perform the desired action (show, hide, and so on) upon them.

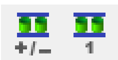


Figure 202:

These buttons only enable when you select at least two entities from the graphics area or the Link Entity browser list, and they perform different actions depending on the mouse button used:

Button	Name	Left-Click Behavior	Right-Click Behavior
	Show/Hide	Show connectors between the selected entities.	Hide connectors between the selected entities.
	Isolate/Isolate Only	Isolate the selected entities and the connectors between them.	Isolate only the connectors between the selected entities.

Unlike other action-type buttons, these two are not affected by the link view option toggle buttons (1st link, 2nd link, or linked connector). However, they do work in conjunction with the realization view option button.

You can specify how these shared connectors are determined by changing settings in the Link Entity Browser configuration window.

Global Display Tools

Use this tool set to display or hide the graphics for supported entities in the browser list.

Figure 203:

Icon	Description
	Elems/Geom/Both (filter for All/None/Reverse and local display control)

Icon	Description
	Display All
	Display None
	Display Reverse

The Elements/Geometry/Both button determines what the other buttons act on; left-click the small triangular downward arrow to reveal a drop-down menu of options. You can select Elements, Geometry, or Both.

Use the Display All, Display None, and Display Reverse buttons at the top of the tab to change the display state of all assemblies, multibodies, components, groups, system cols, load cols, and vector cols shown in the list. All displays and None hides all of the items shown in the list/tree. Display Reverse reverses the state of all the entities (displaying the hidden, and hiding the displayed). If you have multiple link entities selected in the browser list, then HyperMesh will only perform these actions on the selected entities. To deselect all currently selected entities, left-click in any empty "white space" within the browser list, such as between columns. To clear the selected entities in the browser, make sure the browser is active, and then press Esc.

> **Note:** These buttons only affect the display state. They do not actually remove entities from the model, but only show or hide them in the graphics area.

Context Menu

In the Common Top View, right-click on an empty space or right-click on an entity to access the context menu. The options in context menu varies accordingly.

Right-clicking in empty space displays the following:

Option	Description
Create	Create a new Connector Group.
Reorganize	**Auto Reorganize** Organize connectors into connector groups based the settings defined in the **Auto Reorganize Settings** dialog. **Auto Reorganize Settings** Access settings used to reorganize connectors into Connector groups. **Organize Only Connector(s) from Model** Only reorganize connectors from the Model. Connectors that are present in other Connector Groups will not be reorganized.

Option	Description
	Delete Cleared ConnectorGroup(s) Delete previous Connector Groups which become empty after connectors are organized into new Connector Groups based on the reorganization schema setting. **Reorganization schema** Choose one of the following: • Select **Create one group per link combination** to organize connectors by creating new Connector Groups based on their connector link information. For example, if connectors are connecting components with ID 1 and 2, then a new Connector Group will be created with the name "Comp1_Comp2", and all of the connectors that connect those two components will be organized in to this newly created Connector Group. • Select **Create one group per include** to organize connectors by creating new Connector Groups based on the Include file to which the connectors belong to. For example, if connectors belong to the Include file with ID '10', then a new Connector Group will be created with name "Include10", and all of the connectors will be organized in to this newly created Connector Group.
Show All	Turn on the display state of all entities supported in the Common Top view.
Reverse	Reverse the display state of all entities.
Configure Browser	Opens the Configuration Window dialog.

Context menu option change when you right-click on entities. The context menu options vary based if you right-click on a Connector Group or Component Link/Property Link/Part Link. Right-clicking in empty space displays the following:

Option	Description
Duplicate	Create a new, empty Connector Group. If Duplicate is used on a Connector Group that has the name "Connecotr_Group_1", a new, empty Connector Group with the name "Connecotr_Group_11" is created.
Rename	Rename a Connector Group.
XRef Entities	Open the Reference browser and review a list of all Connectors Groups and Connectors that belong to that Connector Group.
Delete	Delete selected entity.

Option	Description
	In the **Confirm** dialog, select the **Retain Connectors and move to Model** checkbox to only delete the Connector Group, and move the connectors that belong to the deleted Connector Group to the Model. By default, both the Connector Group and all the connectors that belong to the Connector Group are deleted.
Empty	Open the **Delete Entity** dialog, from which you can choose which empty Connector Groups to delete.
Configure Browser	Opens the Configuration Window dialog.

Right-clicking on a Component Link, Property Link, or Part Link opens the following context menu options:

Option	Description
Convert Links(s)	Convert existing link type on connectors to: Part, Comp, or Prop. Connector link conversion requires a clear 1:1 relation between a component and a property, otherwise the conversion is not unique and cannot be performed.
Remove From Connectors	Remove links from connectors. This option works on selected/displayed/all connectors. Use the **update layer** option to automatically update layers once links are removed.
Find / Find with FE	The selected links and all connectors referencing them are isolated in the graphics area. The links as well as the linked connectors are highlighted in their browsers. This Find operation considers only the realization and fit view buttons. When the realization button is active, Find changes to Find With FE.
Find Attached / Find Attached with FE	The selected links, all connectors referencing them, and all links referenced by these connectors are isolated in the graphics area. All the found links as well as the linked connectors are highlighted in their browsers. This Find Attached operation considers only the realization and fit view buttons. When the realization button is active, Find Attached changes to Find Attached With FE.
Find Between / Find Between with FE	The selected links and connectors that link them together are isolated in the graphics area. All the found links as well as their shared connectors are highlighted in their browsers.

Option	Description
	This Find Between operation considers only the realization and fit view buttons. When the realization button is active, Find Attached changes to Find Attached With FE. **Note:** The type of connector to find by this action can be set in the options tab of the **Link Entity Browser Configuration** dialog. By default, a connector which references at minimum two of the selected links is treated as a "between" connector.
Show	Works exactly like the action button Show. All view option button settings are considered.
Hide	Works exactly like the action button Hide. All view option button settings are considered.
Isolate	Works exactly like the action button Isolate. All view option button settings are considered. **Tip:** Only show the isolated links in the browser by selecting **(last_celink_review)** from the query build drop-down menu.
Isolate Only	Works exactly like the action button Isolate Only. All view option button settings are considered.
Column Visibility	Opens the **Column Visibility** dialog. Checkboxes are used to control, which columns should be shown in the browser.
Configure Browser	Opens the Configuration Window dialog.

Configuration Window

In the **Browser Configuration** dialog you can alter the columns that display in the Link Entity Browser, and change how special features, such as the find between, operate.

Access the **Browser Configuration** dialog by right-clicking in the Link Entity Browser and selecting **Configure Browser** from the context menu.

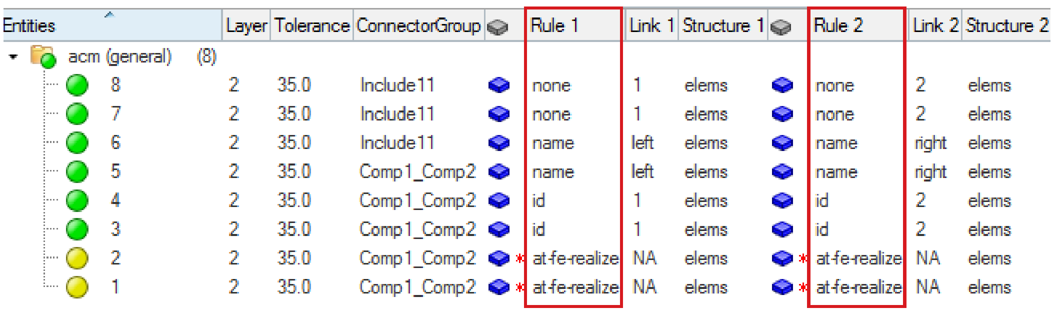
Local Options

Figure 204:

Show Link Rule Columns

When enabled, each link displays its reconnection rule (use ID, use name, or at fe realize) via the Rule column. The reconnect rule is set when the connector is created, and determines what the connector will automatically try to reconnect with when a part is deleted and then replaced.

- If the new part has the same part ID as the deleted one, then use ID will automatically reconnect.
- If the new part has the same name as the deleted one, then use name will automatically reconnect.

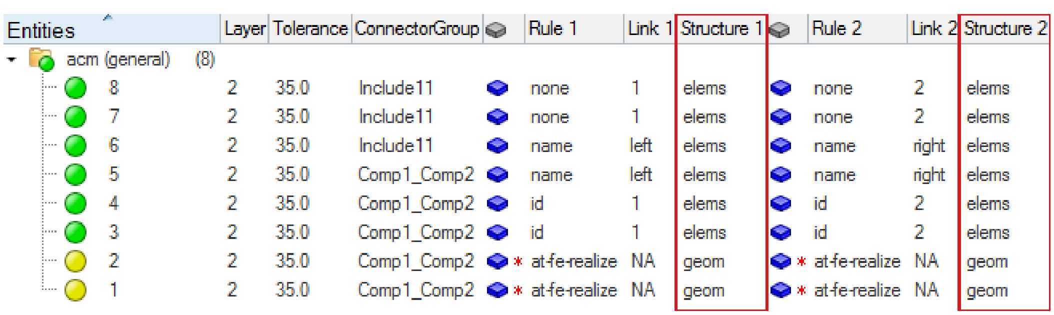


Entities	Layer	Tolerance	ConnectorGroup		Rule 1	Link 1	Structure 1		Rule 2	Link 2	Structure 2
acm (general) (8)											
8	2	35.0	Include11		none	1	elems		none	2	elems
7	2	35.0	Include11		none	1	elems		none	2	elems
6	2	35.0	Include11		name	left	elems		name	right	elems
5	2	35.0	Comp1_Comp2		name	left	elems		name	right	elems
4	2	35.0	Comp1_Comp2		id	1	elems		id	2	elems
3	2	35.0	Comp1_Comp2		id	1	elems		id	2	elems
2	2	35.0	Comp1_Comp2	*	at-fe-realize	NA	elems	*	at-fe-realize	NA	elems
1	2	35.0	Comp1_Comp2	*	at-fe-realize	NA	elems	*	at-fe-realize	NA	elems

Figure 205:

Show Link Elems/Geom Columns

When enabled, each link displays its link state (connect to mesh or geometry) via the Elems/ Geom column.



Entities	Layer	Tolerance	ConnectorGroup		Rule 1	Link 1	Structure 1		Rule 2	Link 2	Structure 2
acm (general) (8)											
8	2	35.0	Include11		none	1	elems		none	2	elems
7	2	35.0	Include11		none	1	elems		none	2	elems
6	2	35.0	Include11		name	left	elems		name	right	elems
5	2	35.0	Comp1_Comp2		name	left	elems		name	right	elems
4	2	35.0	Comp1_Comp2		id	1	elems		id	2	elems
3	2	35.0	Comp1_Comp2		id	1	elems		id	2	elems
2	2	35.0	Comp1_Comp2	*	at-fe-realize	NA	geom	*	at-fe-realize	NA	geom
1	2	35.0	Comp1_Comp2	*	at-fe-realize	NA	geom	*	at-fe-realize	NA	geom

Figure 206:

Max Link Column Viewed

Only display the specified number of columns, regardless of how many links are available in a connector. The maximum number of supported columns is 30.

Show Primary Links Only

By default, both primary and secondary links are displayed in the Connector Entity browser. To only display primary links, select the **Show Primary link only** checkbox.
By default, a node link is also displayed as a component link in the Component link view. For example, components are shown as a link when a node that belongs to an element belongs to a component, an element belongs to a component, or a surface belongs to a component.

When Show Primary Link Only is selected, the Part link view only displays real part links and the Component link view only displays real component links. Use the table below to determine which link types will be interpreted as a component or park link based on view you are using.

Link Type	Component Link View		Part Link View	
	Show Primary Links Only: off	Show Primary Links Only: on	Show Primary Links Only: off	Show Primary Links Only: on
Nodes	yes	no	yes	no
Elements	yes	no	yes	no
Components	yes	yes	yes	no
Assemblies	yes	no	no	no
Parts	no	no	yes	yes
Properties	no	no	no	no
Surfaces (geom)	yes	no	yes	no
Tags	no	no	no	no

Automatic Filter

Filter the link entity list and the connector list simultaneously when performing a show, isolate, or find operation.

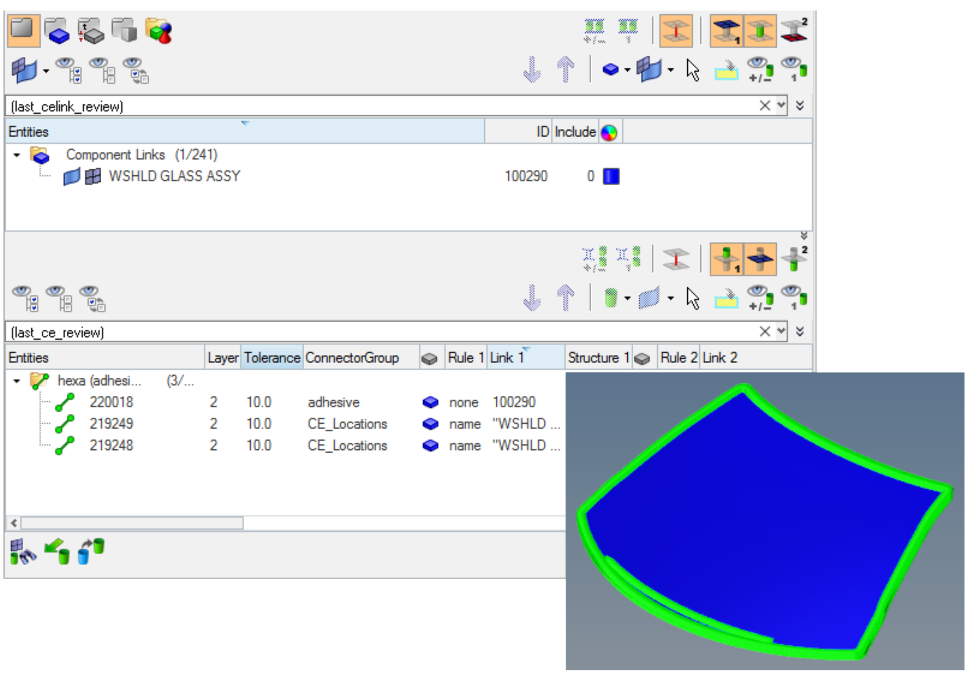


Figure 207: Isolation of the Windshield Glass Components with Attached Connectors

The strings entered in the Query Builder (last_celink_review and (last_ce_review illustrate that the lists have been filtered.

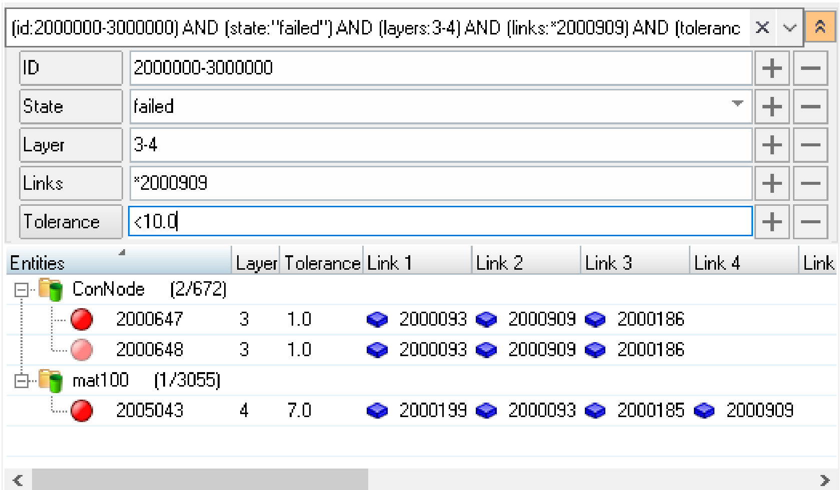


Figure 208: Query Build Filters for the Connector List

> **Tip:** For future filtering via the Query Builder, you can reuse previous filters and combine previous filters with other attribute filters.

Consider Geometry

Consider geometry along with elements while using Show/Hide/Isolate operations.

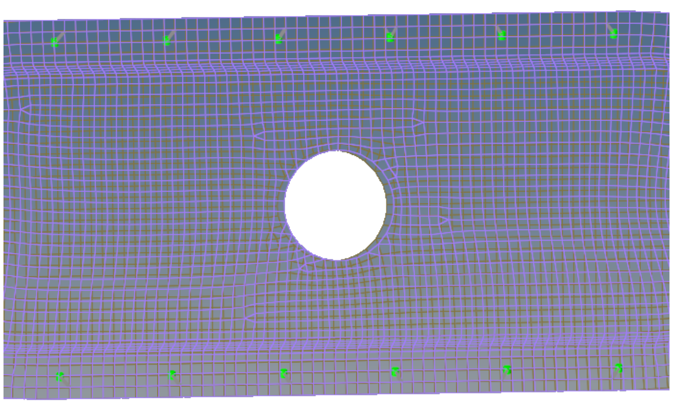


Figure 209: Consider Geometry: Off

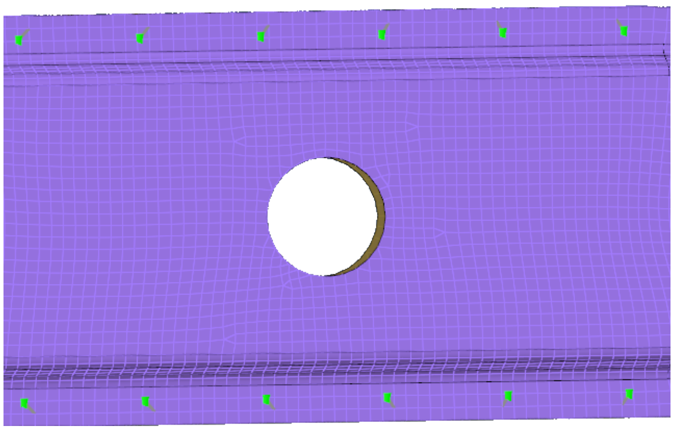


Figure 210: Consider Geometry: On

Consider HAZ Elements

Consider HAZ elements while using Show/Hide/Isolate operations. When this option is on, and if connectors are isolated, HAZ elements are also isolated along with connectors and their links. Available for Connector Configure browser options.

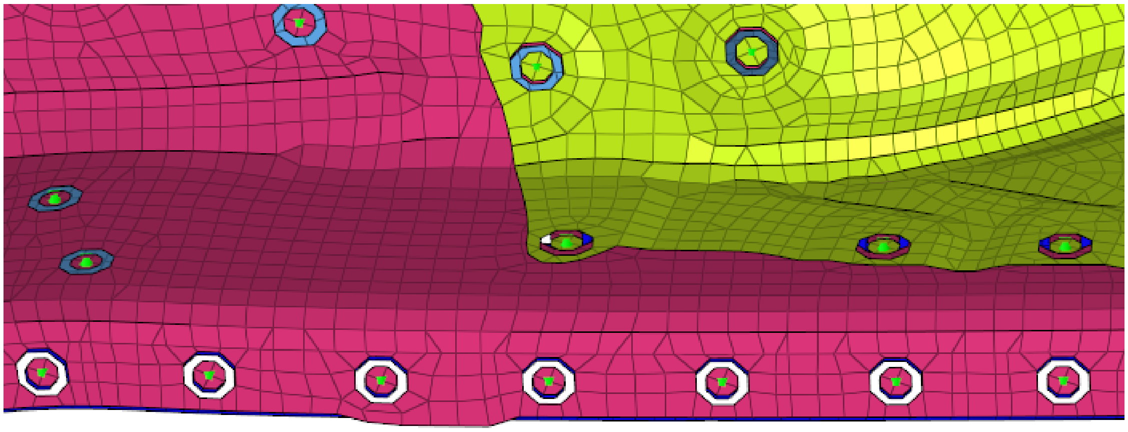


Figure 211: Consider HAZ Elements: Off

HAZ elements of hexa Nugget connectors are not isolated along with connectors and linked components.

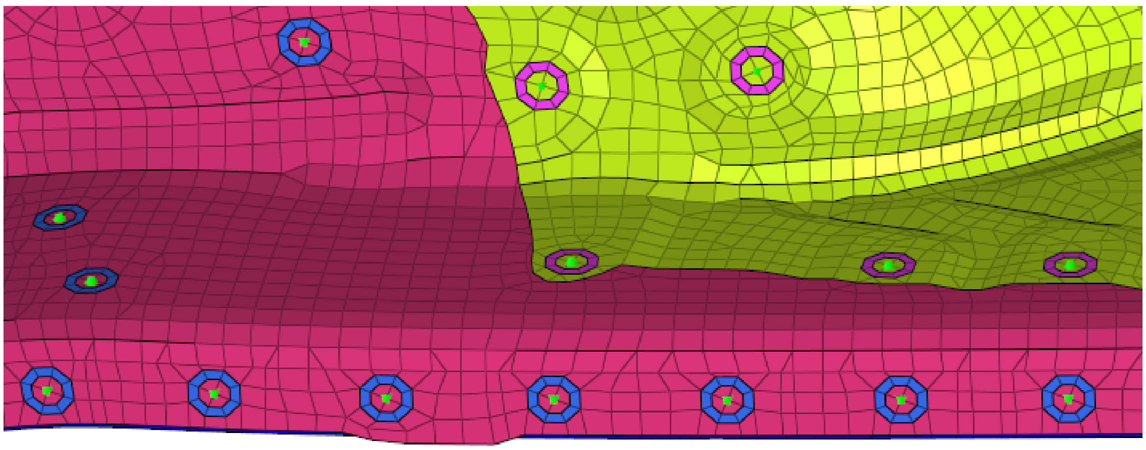


Figure 212: Consider HAZ Elements: On

HAZ elements of hexa Nugget connectors are isolated along with connectors and linked components.

Find Twin Connectors

Select how two connectors are found:

Minimum two links

Find only the connectors that have two or more matching links.

Exact links

Find only the connectors that have the same links. For example, if you start with a connector that has two links, than another connector with three links will not be found even if its first two links matched.

Find Connectors Between With

Select how connectors between selected links are found:

Minimum two selected links

Only connectors that link to at least two selected entities will be affected. Connectors with only one link to any of the selected entities will be ignored.

Exact selected links

Only connectors that link to the selected entities will be affected. This can vary from the Minimum two selected links option, because connectors with three or more links which link two selected entities with at least one unselected entity, would still be found by the Minimum two selected links option but not by this one.

All selected links

Any connector shared by the selected entities will be found.

> **Note:** Connectors which link selected entities to any unselected ones will not be found, as they are not located between the selected entities.

Filter Links To

Choose which links to filter when performing search/isolation functions.

Projection Components

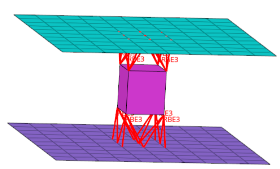
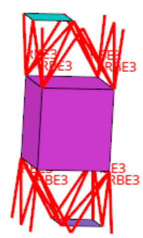
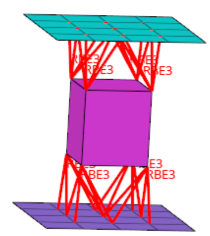
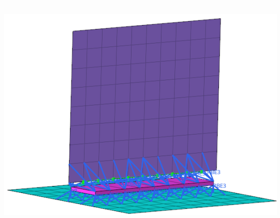
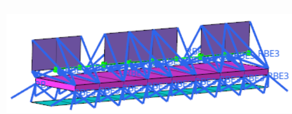
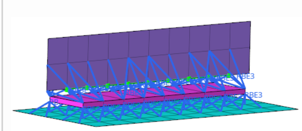
Isolate the entire link component.

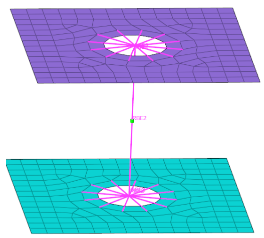
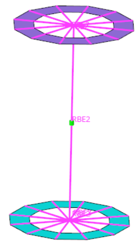
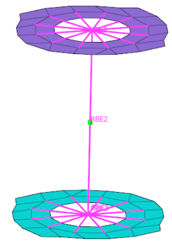
Projection Elements

Isolate only the elements on which a projection falls.

Projection and Attached Elements

Isolate only the elements on which the projection falls and the elements which the connector FE connects.

Filter Links To	Projection Components	Projection Elements	Projection and Attached Elements
Spots	 *Figure 213:*	 *Figure 214:*	 *Figure 215:*
Seams	 *Figure 216:*	 *Figure 217:*	 *Figure 218:*

Filter Links To	Projection Components	Projection Elements	Projection and Attached Elements
Bolts	 *Figure 219:*	 *Figure 220:*	 *Figure 221:*

Global Options

Figure 222:

Autofit

Use this option in combination with the other view option toggle buttons and show, hide, isolate, or isolate only, or in combination with the advanced action buttons. After an action is performed, the newly found connectors are placed in the middle of the screen. If this option is used in combination with one of the previously mentioned buttons, it works like a pure fit view.

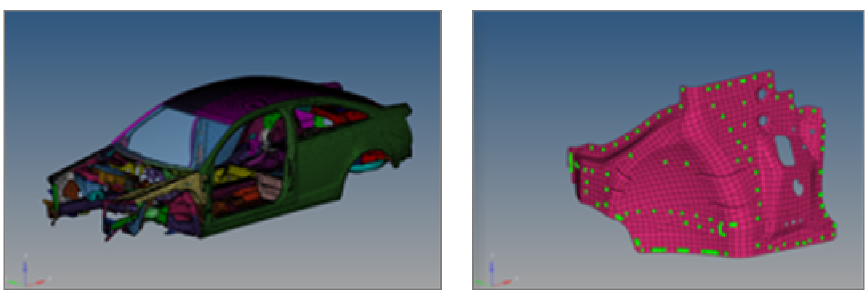


Figure 223:
Link entity isolated with the Autofit option activated.

Autocolor Visualization Mode

With this mode activated, the Connector Browser takes over the control of the element visualization mode from the Visualization panel. Elements will be colored by component, by property, or by part, depending on which connector link view is activated.

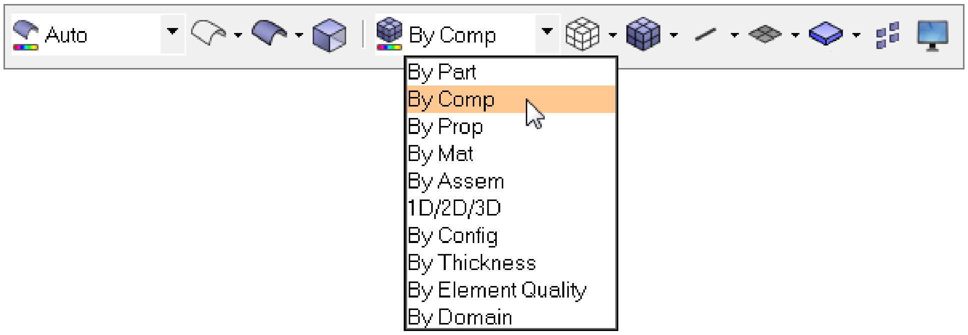


Figure 224:

Connector Entity Browser

The middle section of the Connector Browser contains a tree view of all the connections contained in the model, and has its own set of tools similar to the ones found in the Link Entity Browser.

All the connectors in the model are displayed in folders organized based on the respective realization types. The names of the folders are obtained from the FE configuration names specified for respective solvers in the `feconfig.cfg` file.

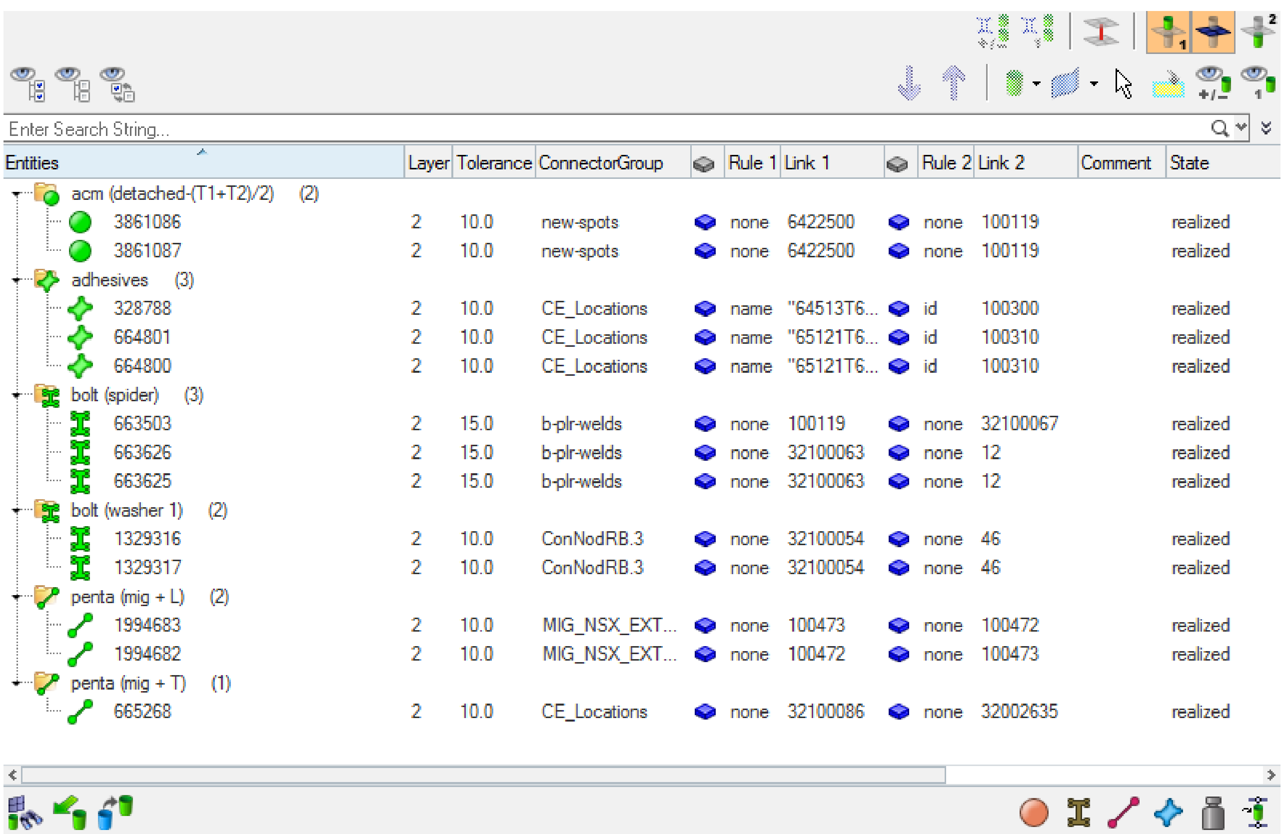


Figure 225:

The connector information in the tree can be used to find link entities connected by specific connectors, and also to modify certain connector attributes. The columns display a sub-set of connector information that is important for recognizing connection information easily.

Unless all view option toggle buttons in the global display tool set are inactive, the action mode tool set and the context menu actions (show, hide, isolate, isolate only) work similarly to their functions in the Model Browser's component view.

If at least one of the view option toggle buttons is active (appearing in orange), the behavior of show, hide, isolate and isolate only is different. The image on the action buttons changes to include a connector symbol, indicating the different behavior.

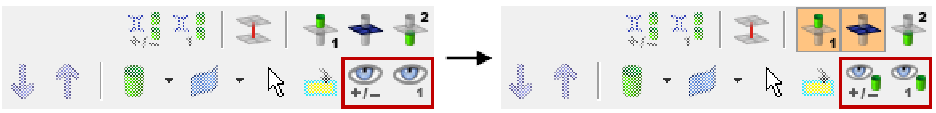


Figure 226:

The actions take into account connector-link relations, and the display result depends on the active view options. Therefore parts of the model are separated into three different categories based on the connector selection:

1st connector entity
Selected connectors.

linked entities
Links which are referenced by one of the selected connectors.

2nd connector entity
Connectors which reference at least one of the linked entities.

> **Note:** In this case connectors and their realizations are treated as being a separate category from their links, in order to prevent unpredictable cross-references.

This kind of categorical separation is only used for the actions show, hide, isolate and isolate only, and only when one of the view option toggle buttons is active, regardless of whether the actions are taken from the context menu or the action buttons. No other functionality uses this categorization at all.

You can also change the base features of the Connector Entity Browser in the Connector Entity Browser configuration window.

The following connector details are displayed as column data in the browser tree:

Option	Description
Entities	ID of the connector and an image that represents the respective connector's style (spot, seam, bolt, and so on).
Layer	Total number of link entities to be joined by the connector. This is also marked as thickness layers (2T/3T/4T, and so on).

<table>
<tr><th>Option</th><th>Description</th></tr>
<tr><td>Tolerance</td><td>Realization tolerance of the connector. You can change this by right-clicking the field and typing in a new value, then pressing Enter.</td></tr>
<tr><td>ConnectorGroups</td><td>Name of the Connector Group to which the connector belongs.</td></tr>
<tr><td>Component</td><td>Name of the component to which the connector belongs (this column is not displayed in default view).</td></tr>
<tr><td>(link type)</td><td>Type of entity that can be added to the selected connector(s) as a link reference. Supported entities are assemblies, components, surfaces, elements, tags, and nodes.

Tip: Left-click on a link to highlight the relevant entity (part) in the graphics area.

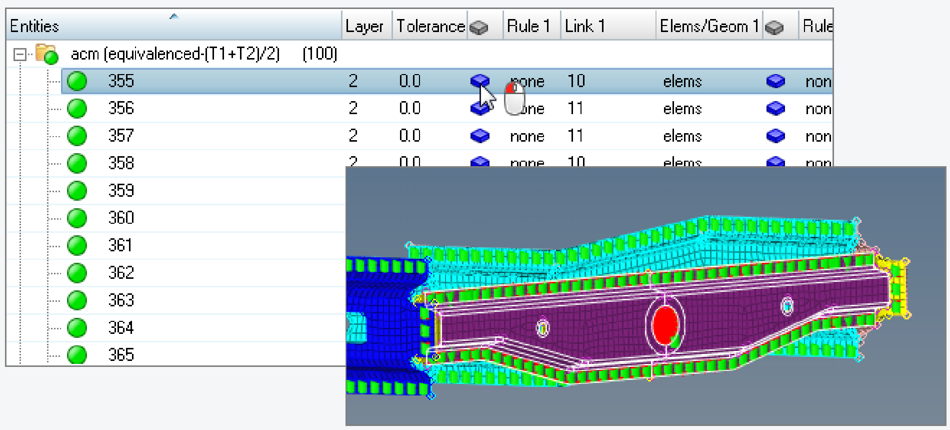


Figure 227:</td></tr>
<tr><td>Rule</td><td>Defines how a connector treats an entity added as a link. Adding a link with an ID or name rule forces the connector to retain the link's ID or name even if that link entity no longer exists in the database. This aids in part replacement when a new part replaces an old part and both share the same ID or name. Adding a link with the at fe-realize rule ensures that each time a connector is realized the closest entity of the correct type is found and connected. This is useful when connectors need to connect to a closest part in an assembly.</td></tr>
<tr><td>Link</td><td>ID or name of the entity added as a link to a connector.</td></tr>
</table>

Option	Description
Structure	Defines whether the weld created during connector realization connects to geometry or mesh on the link. This only applies to assembly, component, and surface entities that can contain geometry and/or mesh information.
Comment	Input comments for a connector.
State	Realization state of the connector entity: unrealized, realized, failed, or modified. **Note:** For those writing scripts instead of using the GUI, a more detailed report can be created by using the following lines in your script: set error_report [hm_ce_errorreport CE_ID 1].

Functionality is accessed from the global display tool set, an action modes tool set and a right-click context menu.

Action Mode Tools

The series of icons found on the right side of the Connector Entity Browser control both entity selection and the display of the model. These buttons behave in two different ways depending on the setting of the view option buttons. If no view option button is active, the core behavior of the action mode tools (show/hide/isolate) applies exclusively to the selected connectors, similarly to components in the component view of the Model Browser. For that reason, this topic focuses on their behavior when a view option is active.

In this mode the actions like show, hide, isolate or isolate only are not only done on the selection; based on the selection, the browser performs a process of finding and filtering so that further entities can be taken into account for the action depending on the setting of the view option toggle buttons.

Use these tool set buttons to advance or step back through multiple selected connectors, pick connectors from the graphics area, add connectors to an active panel's entity collector, turn the display of individual connectors on and off, visually isolate specific connectors, or undo any visual modifications (show/hide/isolate).

Figure 228:

Icon	Description
	The next and previous buttons cycle through the selected connectors in the tree, but are only active when entities in the tree are selected (highlighted). These are

Icon	Description
	especially helpful when not all selected connectors can be seen in the partial view of the tree.
	These buttons affect how the rest of the tools work, by determining what type of entity they will act on. For example, when set to "connectors", the Selector tool (described below) will only select or deselect connectors. **Note:** These icons are disabled and set to their default values ("connectors" and "both geometry and elements") since only connectors can be selected in the Connector Entity Browser, but the Connector Browser works with all kinds of connectors – both geometry and FE-based.
	Use the Selector tool to interactively select any type of supported connector entity via the browser, or by selecting within the graphics area. The Selector can be used to find connector entities from the graphics area which will then be highlighted in the list, and is also an efficient way of selecting multiple connector entities at once. Finally, the selector can be used in conjunction with the action buttons show, hide, isolate and isolate only as well as the advanced action buttons show/hide twin connectors and isolate/isolate only twin connectors; simply select connector entities from the browser or graphics area using the Selector, then click the desired action button. **Note:** These advanced action buttons only activate when at least two connector entities are selected. In other respects the Selector works exactly the same as described for the Model Browser.
	The Add to Panel Collector is a function whereby the browser can be used to select and add entities to the panel collectors within HyperMesh. This is an alternative method to using the advanced selection capabilities already available in each panel collector's extended entity selection menu. This button only becomes available when you have a HyperMesh panel open that includes at least one entity collector.
	Show/Hide the currently selected entities, depending on the currently active view option toggles. Alternatively, click this mode on and then pick the desired connectors from the graphics area; connectors and any other entities determined by the view option toggle settings are hidden as you click on them.

Icon	Description
	Note: When used to select from the graphics area, this button only works on visible connectors.
	Isolate/isolate only the currently selected entities. Alternatively, click this mode on and then pick the desired connector entity from the graphics area. **Isolate** Displays only the selected entities which match the view option toggles, turning their display state to on and turning all other entities of the same type off. Isolate works locally within a specific entity type, for example, if component(s) are isolated, then all display states of other displayable entities, such as load collectors, remain untouched. **Isolate Only** Works like Isolate, except that it also affects entity types different from the matching, selected entities. Thus, it turns off ALL displayable entities (regardless of type) except for the selected one(s) that match the view option settings. **Note:** Unlike the normal isolate button, when used in the graphics area, this button only works on visible connectors.

View Option Toggle Buttons

These options affect the action mode tools, and thus determine the entities that display when you select and then show, hide, or isolate a supported entity.

Figure 229:

Each button is modal, that is, you click it once to activate it, and click it again to deactivate it. Active buttons remain active until you specifically deactivate them, so you do not need to worry about them "resetting" after you perform an action such as isolate.

Figure 230: Active

Figure 231: Inactive

The following model illustrates the effects of these search options.

> **Note:** The highlighted connector; this connector is the starting point for the following examples.

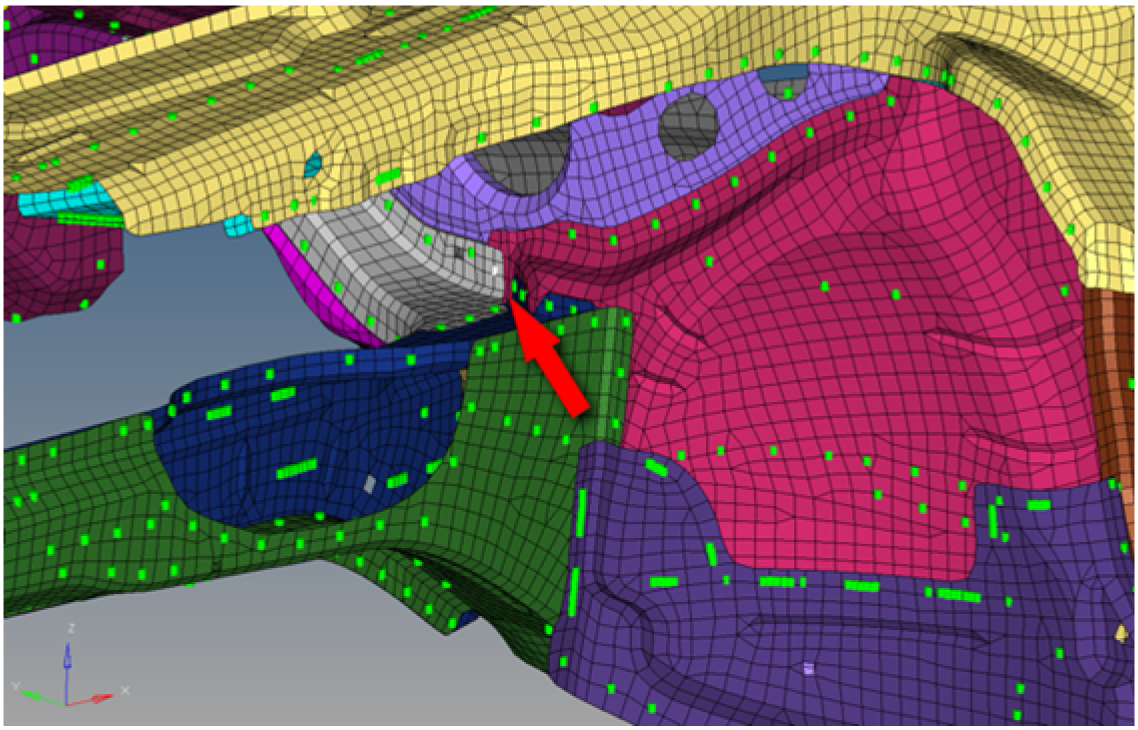
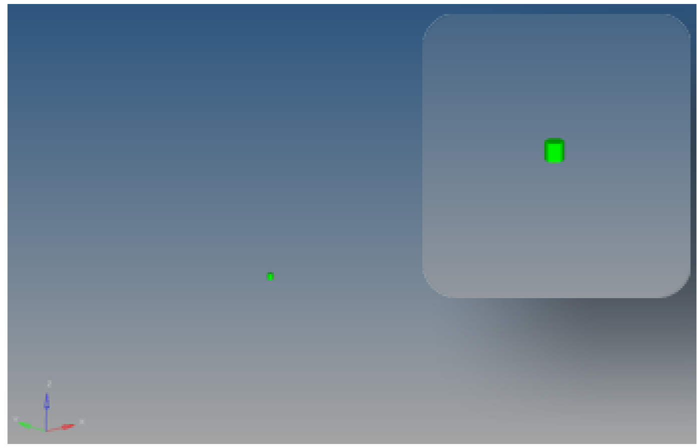
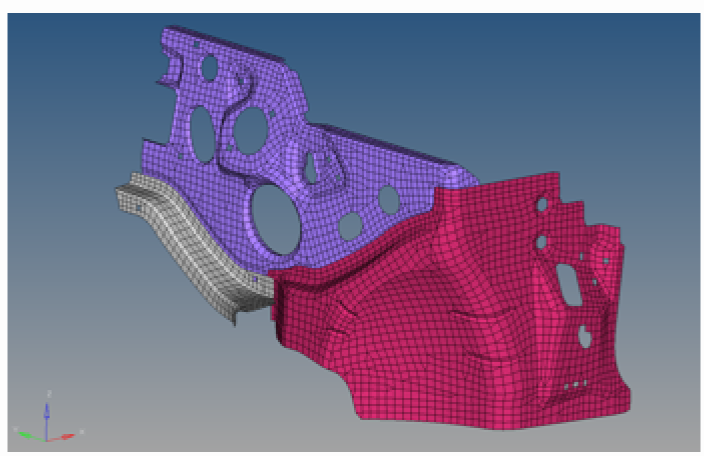
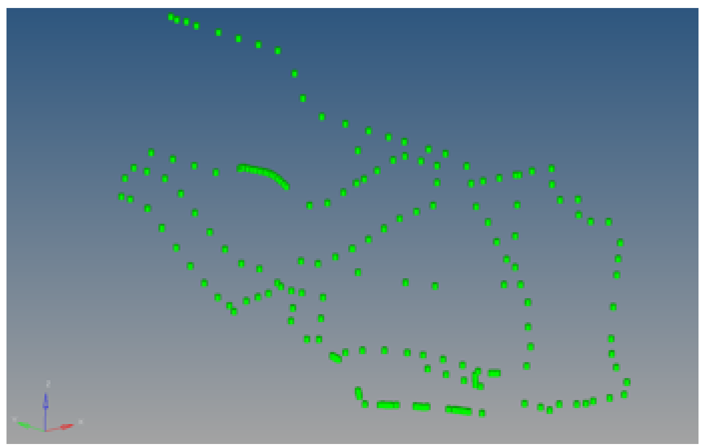


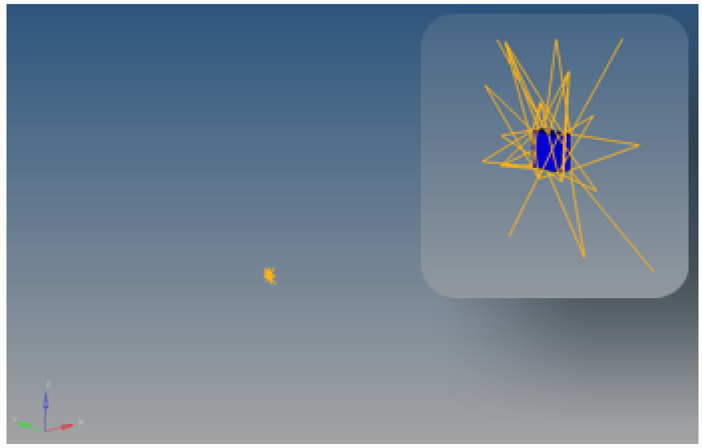
Figure 232:
Notice the small, white-highlighted connector entity on the grey-meshed part. (Model provided courtesy of Audi)

In each case below, the selected connector has been Isolated using only the relevant view option.

Option	Name	Effect
	1st connector entity	The selected connector entity can always be seen as the 1st connector entity, even if the selected connector doesn't reference any links at all. If the 1st connector entity view option button is active, the selected connectors will be taken into account for the action regardless. This means that in the case of isolation, all selected connectors will be isolated.

Option	Name	Effect
		 *Figure 233:* *The single selected connector is isolated (also shown in magnified view).*
	Linked entity	Finds the link entities/supported entities to which the selected connector connects. If this linked entity view option button is active, all entities linked to the 1st connector entities will be taken into account for the action. It does not matter if the 1st connector entity view option button is active or inactive, its entities will still be located. This determination has nothing to do with any display states. This means that in case of isolation, only the entities which are referenced by the selected connectors (1st connector entity) are isolated.  *Figure 234:*

Option	Name	Effect
		The connector does not display because "1st linked entity" is not active.
	2nd connector entity	Finds other connectors that are connected to the chosen connectors' linked entities. If this 2nd connector entity view option button is active, all connectors referenced by the determined linked entities except the originally-selected 1st connector entities will be taken into account for the action. It does not matter if the 1st connector entity or the linked entity view option buttons are active or inactive; this determination has nothing to do with any display states. In the case of isolation, this means that only the connectors that share links with the selected 1st connector entities are isolated.  *Figure 235:* *The entity does not display because "linked entity" is not active.*
	Realization	Finds and displays the realization for the selected connectors.

Option	Name	Effect
		 *Figure 236:*

.

Cumulative Effect of Multiple View Options

These options work accumulatively, for example, when both the Linked entity and 2nd connector entity buttons are active, then selecting and isolating a connector displays the component that it links to, and all the other connectors that link to that component. If you had Realization, Linked entity, and 2nd connector entity active when you isolated the same connector, then the model would display the component to which the connector links, all other connectors linking to that component, and the realizations of each displayed connector.

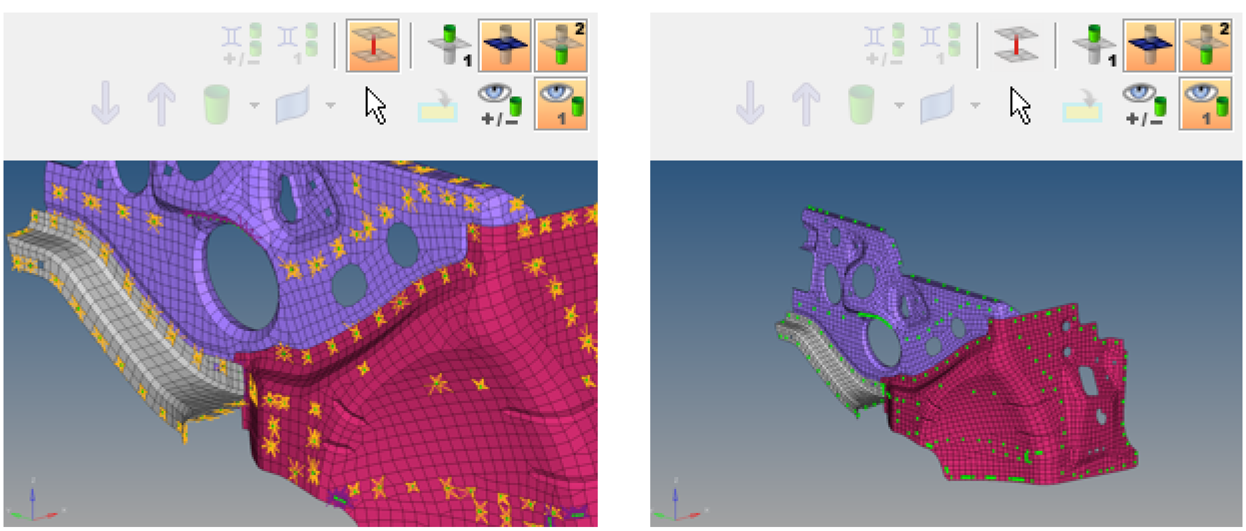


Figure 237:

The first image shows an isolated view with linked entity, 2nd connectors, and realizations. The second includes the same options, but without realizations.

Using Connector Links to Expand the Selection

You can use the Show feature to gradually increase the entities and connectors that display. In the following example, a single connector's link entities and 2nd connectors have been isolated. By activating the Show action mode, the components and connectors that display can be expanded by clicking on connectors that currently display; since the link entities and 2nd connectors view modes are still active, each clicked connector's link entities and 2nd connectors are added to the view.

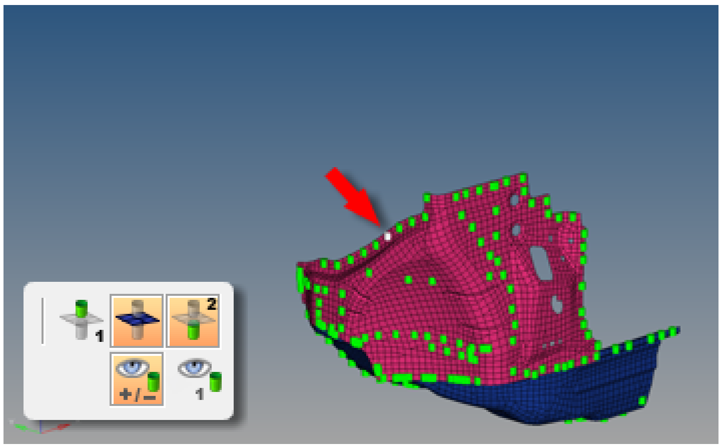


Figure 238:
The highlighted connector is selected for Show.

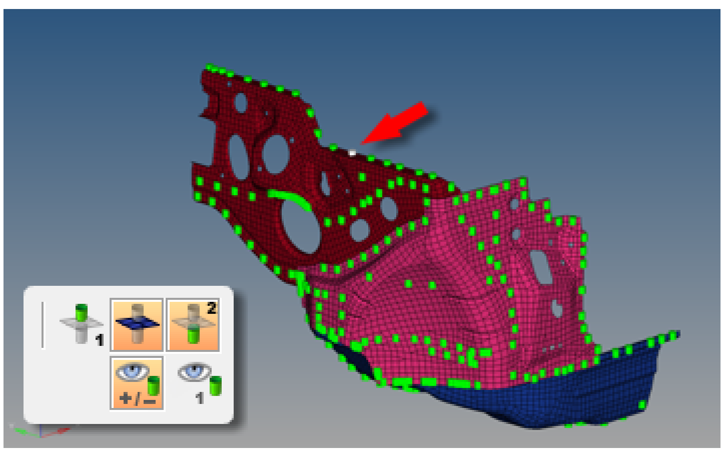


Figure 239:
A new connected component displays. Again, the highlighted connector is clicked...

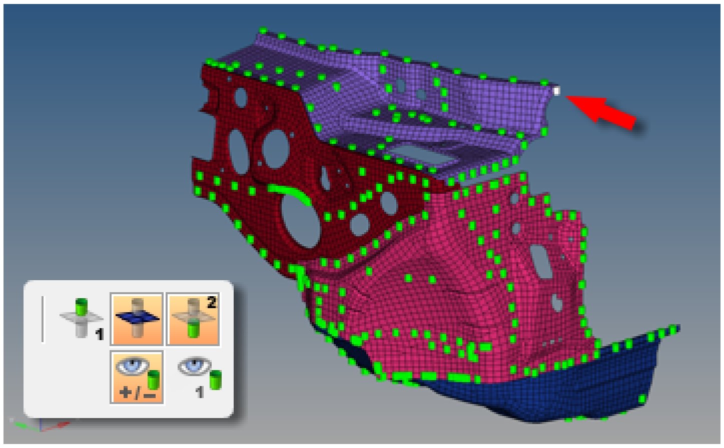


Figure 240:

Connected component displays. A third highlighted connector is clicked...

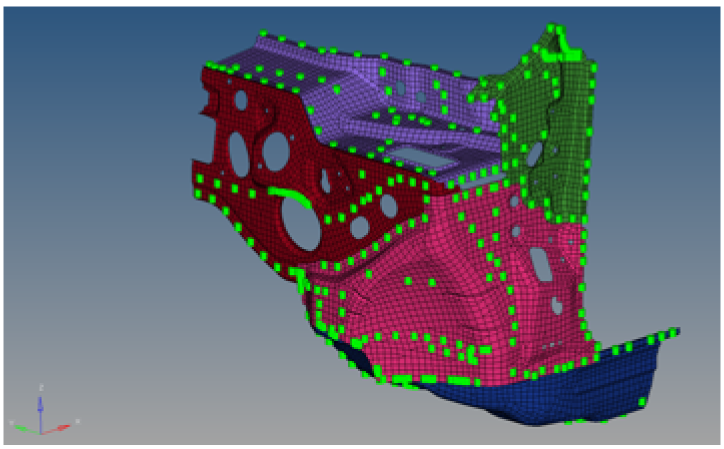


Figure 241:

Third connected component displays.

You can continue revealing more and more parts this way, theoretically eventually revealing the entire model. The reverse option is also true; by activating the Hide mode instead of Show, you could gradually "chip away" at the model, removing one connected component at a time – or multiple components in the case of multi-layer connectors.

Advanced Action Buttons

Located adjacent to the view option toggle buttons, these two buttons allow you to show, hide, isolate or isolate only "twin" connectors. Twin connectors are connectors which reference at least two matching link entities. The criteria for exactly how twin connectors must match can be increased in the Connector Entity Browser Configuration Window to require all links to match exactly, instead of only two or more matching ones. These advanced actions then select the connectors referencing the same links and perform the desired action (show, hide, and so on) upon them.

Figure 242:

These buttons only enable when you select at least one connector entity from the graphics area or the Connector Entity Browser tree, and they perform different actions depending on the mouse button used:

Button	Name	Left-Click Behavior	Right-Click Behavior
	Show/Hide	Show twin connectors	Hide twin connectors
	Isolate/Isolate Only	Isolate twin connectors	Isolate only twin connectors

Unlike other action-type buttons, these two are not affected by the view option toggle buttons (1st connector, 2nd connector, or linked entity). However, they do work in conjunction with the realization view option button.

Global Display Tools

Use this tool set to control the display of connectors.

Figure 243:

The functionality works at three levels; if nothing is selected, this is called the global level, the display action will work on all connectors. At the folder level, if the folder is highlighted then the action will only operate on the connectors within the folder. At the local entity level, if a single or multiple connectors are selected then the operation will operate only on those selected.

Icon	Description
	Display All
	Display None
	Display Reverse

> **Note:** These buttons only affect the display state. They do not actually remove entities from the model, but only show or hide them in the graphics area.

Context Menu

The Connector Entity Browser context menu includes all of the functionality available in the Connector Entity Browser, including connector creation and deletion, renumbering, updating, finding connectors, and much more.

Access the Connector Entity Browser's context menu by right-clicking in the tree list.

> **Note:** If you right-click the **Tolerance** or **Realize to** fields of a specific connector, you will access those fields for edit rather than opening the context menu.

The tools available in the context menu vary depending on what you right-click on; for example, right-clicking on a connector (in the Entities column) accesses the full menu, but right-clicking a link (in the Link 1 or Link 2 columns) only presents the Remove Link option.

Depending on the column clicked, the following options may or may not be available as appropriate to the item clicked:

Option	Description
Create	Choose the type of connector to create, then define connector settings in the related connector panel that opens in the HyperMesh panel area, as well as the FE Absorb user interface.
Delete	Delete connectors or connectors and their related FE elements.
Organize	Open the **Organize Connector(s)** dialog, from which you can organize connectors into different Connector Groups.
Convert	Convert connectors to a different type, for example a spot connector into a bolt connector. For more information, see Convert Style.
Combine	Combine single point connectors into line connectors, for example several spot connectors into a spotline connector. For more information, see Combine.
Split	Split line connectors into single point connectors, for example a spotline connector into several spots. For more information, see Split.

Option	Description
Position	Position connectors to the center between the farthest links or back to their source position. For more information, see Position.
Renumber	Renumber a single connector; the change is permanently recorded in the database and in the browser.
Update layer	Update the Layer field, which becomes editable in Connector Entity browser, to input the layer value that you wish to assign to the selected connectors (this function works for multiple selections). Enter the desired number of layers into the text box and press Enter. The layer value defines the number of thicknesses (2T/3T/and so on) a connector connects at its location. The layer value defined in a connector can be greater than or equal to number of links connected by the connector. The connectors will be unrealized after this update, but the change is recorded permanently in the database and in the browser. Alternatively, you can also update the layer directly by right-clicking on the field itself rather than using the context menu.
Update tolerance	Update the Tolerance field, which becomes editable in Connector Entity browser, to input the tolerance value that you wish to assign to the selected connectors (this function works for multiple selections). You can also do this directly by right-clicking on the field itself rather than using the context menu.
Update diafactor	Update the diafactor field, which becomes editable in Connector Entity browser, to input the diafactor (diameter factor) value that you wish to assign to the selected connectors. This function works for multiple selections. You can also do this directly by right-clicking on the field itself rather than using the context menu. **Note:** The diafactor field is shown for any connector of any type. This value is mainly used for specific bolt realizations; all others get "NA" by default for this field. Although you can update "NA" to any value, the value is ignored during realization.
Update diameter	Opens a dialog to enter the diameter value that you wish to assign to the selected connectors. This function works for multiple selections. You can also do this directly by right-clicking on the field itself rather than using the context menu.

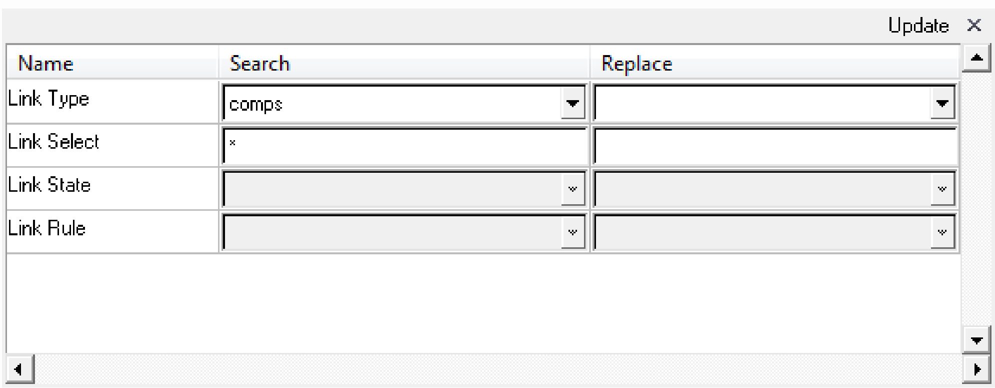
<table>
<tr><th>Option</th><th>Description</th></tr>
<tr><td></td><td>Note: The diameter field is shown for any connector of any type. This value is not necessary for many realization types; they get per default NA for this field, though you can update NA to any value. This value is ignored then during realization.</td></tr>
<tr><td>Add Link</td><td>Opens a new set of controls in the bottom of the tab to add links to the selected connectors.
For more information, see Add Link.</td></tr>
<tr><td>Update Link</td><td>Opens a new set of controls in the bottom of the tab to perform a part replacement.
Note: Editing a connector causes it to become unrealized.


Figure 244:</td></tr>
<tr><td>Remove Link</td><td>Remove the selected link(s) from the selected connectors or all connectors in the browser. The connectors will be unrealized and the change is recorded permanently in the database and in the browser.
This option is accessible only from the column(s) that display the link information.</td></tr>
<tr><td>Remove Links</td><td>Remove all of the links from the selected connectors. The connectors will be unrealized and the change is recorded permanently in the database and in the browser selected connectors or all connectors in the browser. The connectors will be unrealized and the change is recorded permanently in the database and in the browser.</td></tr>
<tr><td>Convert Links</td><td>Convert links to one of the following link types: Part, Comp, or Prop.</td></tr>
</table>

Option	Description
	For more information, see Convert Links.
Change Link Rule	The link rule defines how a connector treats an entity added as a link. Adding a link with the ID or name rule forces the connector to retain the link's ID or name even if that link entity no longer exists in the database.
Change Link Structure	The link state defines if the weld created during connector realization connects to geometry or mesh on the link. This is applicable to only components and surfaces entities that can contain geometry and/or mesh information.
Copy Attributes	Copy all attributes of selected connectors.
Paste Attributes	Paste the copied connector attributes on to another connector.
Rerealize	Calls the `*CE_Realize` command to realize connectors by accepting only a connector mark. The underlying assumption in the command is that each connector passed in the mark has the required information to be successfully realized. The required information such as tolerance, weld configuration, diameter, and so on is not defined for connectors created using the FE Absorb utility; hence the Rerealize feature in the Connector Browser works only for connectors that were realized through an HyperMesh Connector panel.
Unrealize	Unrealizes the selected connectors.
Find Components / Find Components with FE	The entities that are linked in the selected connectors are isolated in the display with the connectors. If the realization view option is turned on then the realized FE of the connectors is also displayed. The isolated entities are highlighted in the table.
Show	Works exactly like the action button Show. All view option button settings are considered.
Hide	Works exactly like the action button Hide. All view option button settings are considered.
Isolate	Works exactly like the action button Isolate. All view option button settings are considered. **Tip:** Only show the isolated connectors in the browser by selecting **(last_celink_review)** from the Query Build drop-down menu.

Option	Description
Isolate Only	Works exactly like the action button Isolate Only. All view option button settings are considered.
Collapse All	This option closes all the expanded folders in the browser.
Expand All	This option expands all the folders to display all the connectors. This operation may take some time for folders that contain thousands of connectors.
Configure Browser	Opens the Configuration Window which contains options to configure the Connector Browser's list display and button behavior.

Configuration Window

In the **Browser Configuration** dialog you can select which columns to display in the browser, and change the way special tools, such as find twin connectors, operate.

Open the **Browser Configuration** dialog by right-clicking in the Connector Entity Browser and selecting **Configure Browser** from the context menu.

Local Options

Show Link Rule Columns

When enabled, each link displays its reconnection rule (use ID, use name, or at fe realize) via the Rule column. The reconnect rule is set when the connector is created, and determines what the connector will automatically try to reconnect with when a part is deleted and then replaced.

- If the new part has the same part ID as the deleted one, then use ID will automatically reconnect.
- If the new part has the same name as the deleted one, then use name will automatically reconnect.

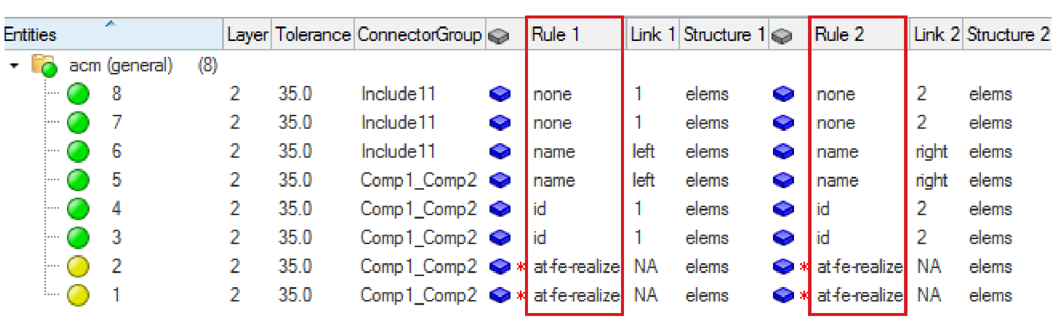


Figure 245:

Show Link Structure Columns

When enabled, each link displays its link state (connect to mesh or geometry) via the Structure column.

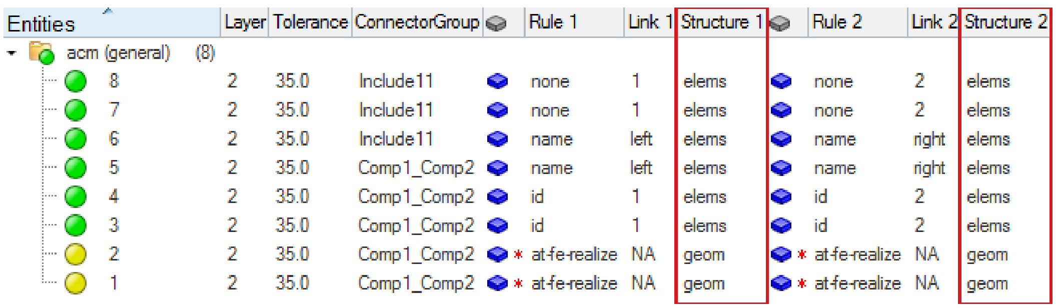


Figure 246:

Max Link Column Viewed

Regardless of how many links a connector might have, the browser will only display the specified number of columns.

Show Primary Links Only

By default, both primary and secondary links are displayed in the browser. To only display primary links, select the **Show Primary link only** checkbox.

By default, a node link is also displayed as a component link in the Component link view. For example, components are shown as a link when a node that belongs to an element belongs to a component, an element belongs to a component, or a surface belongs to a component.

When Show Primary Link Only is selected, the Part link view only displays real part links and the Component link view only displays real component links. Use the table below to determine which link types will be interpreted as a component or park link based on view you are using.

Link Type	**Component Link View**		**Part Link View**	
	Show Primary Links Only: off	Show Primary Links Only: on	Show Primary Links Only: off	Show Primary Links Only: on
Nodes	yes	no	yes	no
Elements	yes	no	yes	no
Components	yes	yes	yes	no
Assemblies	yes	no	no	no
Parts	no	no	yes	yes
Properties	no	no	no	no
Surfaces (geom)	yes	no	yes	no
Tags	no	no	no	no

Automatic Filter

Filter the link entity list and the connector list simultaneously when performing a show, isolate, or find operation.

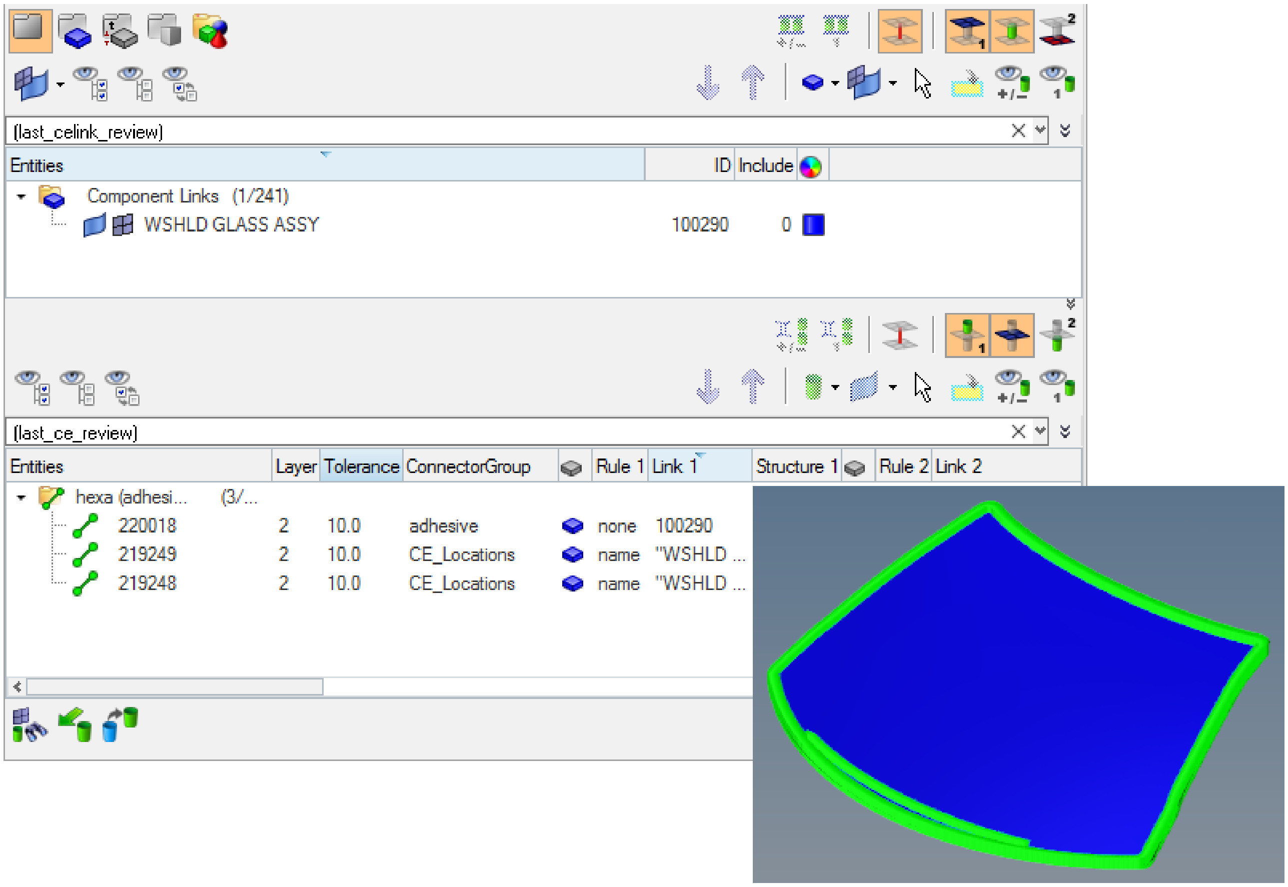


Figure 247: Isolation of the Windshield Glass Components with Attached Connectors

The strings entered in the Query Builder (last_celink_review) and (last_ce_review) illustrate that the lists have been filtered.

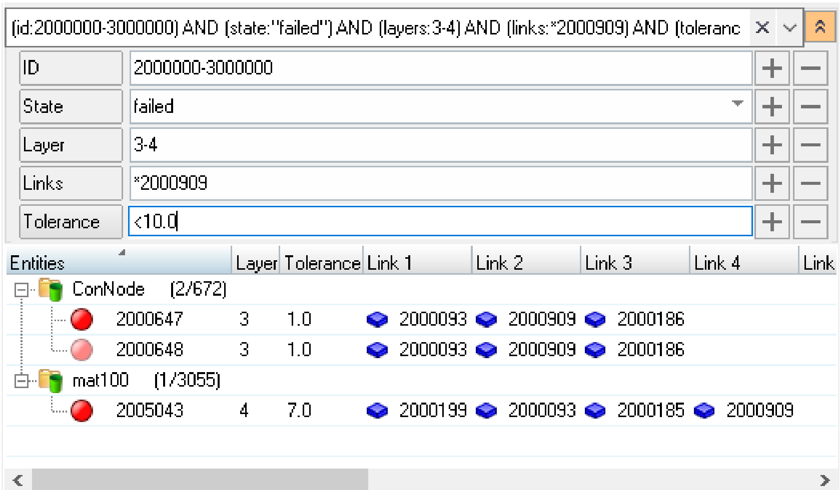


Figure 248: Query Build Filters for the Connector List

> **Tip:** For future filtering via the Query Builder, you can reuse previous filters and combine previous filters with other attribute filters.

Consider Geometry

Consider geometry along with elements while using Show/Hide/Isolate operations.

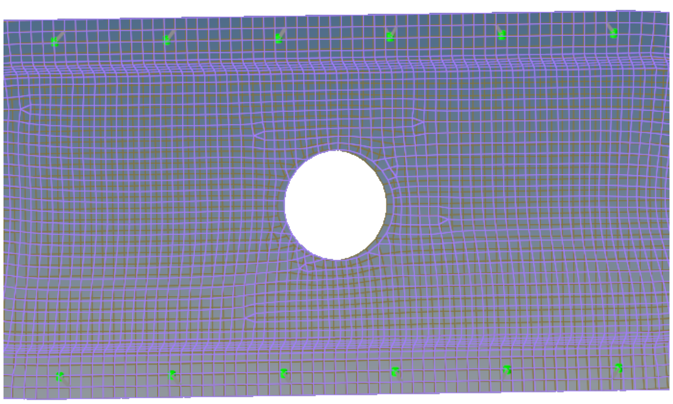


Figure 249: Consider Geometry: Off

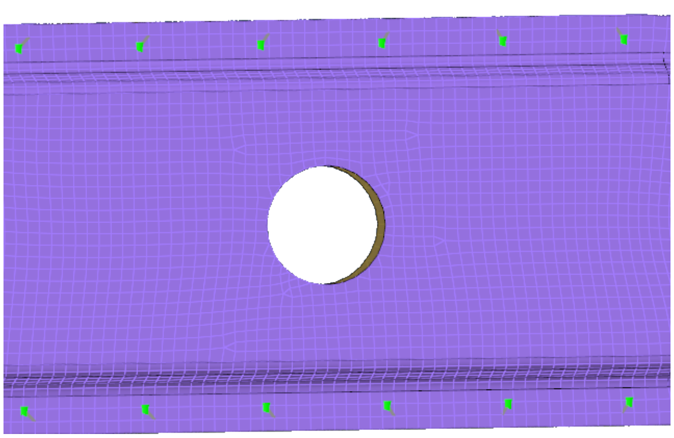


Figure 250: Consider Geometry: On

Consider HAZ Elements

Consider HAZ elements while using Show/Hide/Isolate operations. When this option is on, and if connectors are isolated, HAZ elements are also isolated along with connectors and their links. Available for Connector Configure browser options.

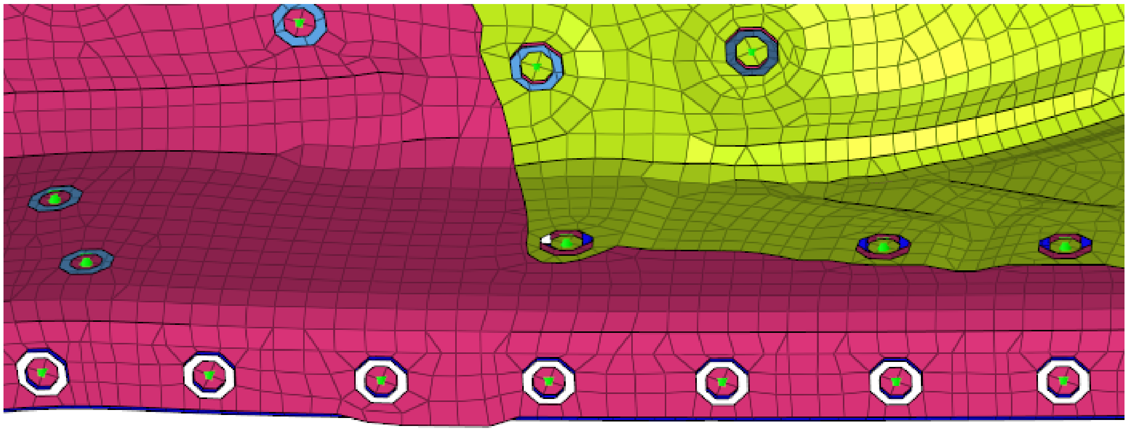


Figure 251: Consider HAZ Elements: Off

HAZ elements of hexa Nugget connectors are not isolated along with connectors and linked components.

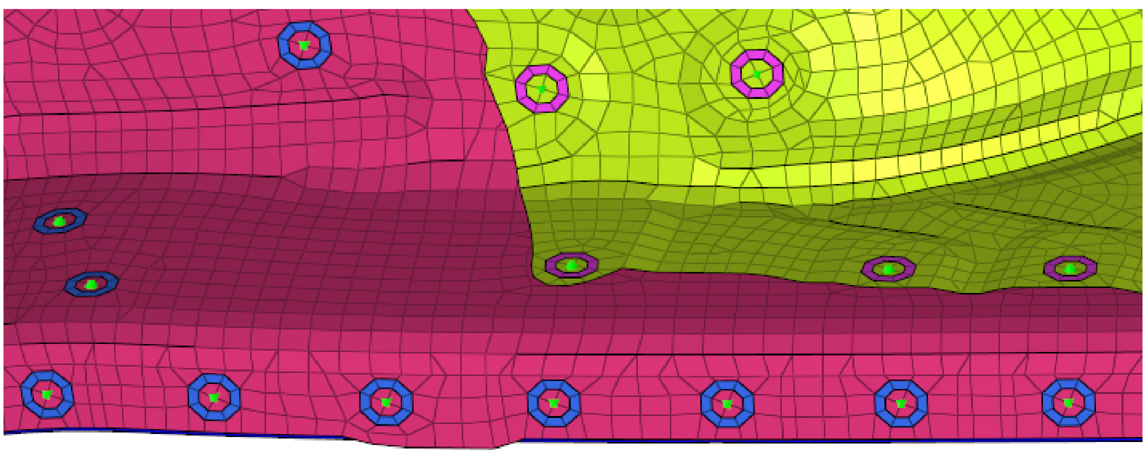


Figure 252: Consider HAZ Elements: On

HAZ elements of hexa Nugget connectors are isolated along with connectors and linked components.

Find Twin Connectors

Select how two connectors are found:

Minimum two links

Find only the connectors that have two or more matching links.

Exact links

Find only the connectors that have the same links. For example, if you start with a connector that has two links, than another connector with three links will not be found even if its first two links matched.

Find Connectors Between With

Select how connectors between selected links are found:

Minimum two selected links

Only connectors that link to at least two selected entities will be affected. Connectors with only one link to any of the selected entities will be ignored.

Exact selected links

Only connectors that link to the selected entities will be affected. This can vary from the Minimum two selected links option, because connectors with three or more links which link two selected entities with at least one unselected entity, would still be found by the Minimum two selected links option but not by this one.

All selected links

Any connector shared by the selected entities will be found.

> **Note:** Connectors which link selected entities to any unselected ones will not be found, as they are not located between the selected entities.

Filter Links To

Choose which links to filter when performing search/isolation functions.

Projection Components

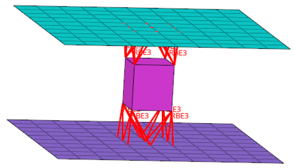
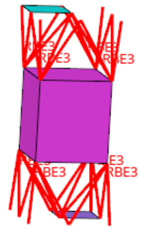
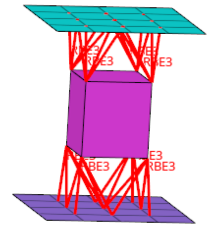
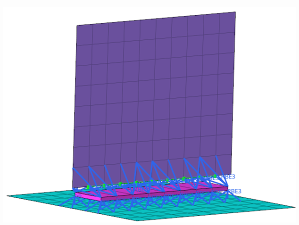
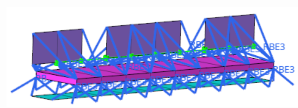
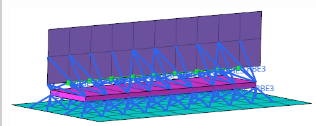
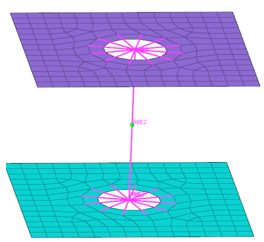
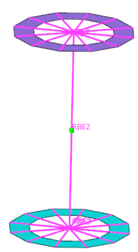
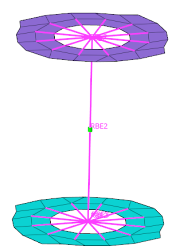
Isolate the entire link component.

Projection Elements

Isolate only the elements on which a projection falls.

Projection and Attached Elements

Isolate only the elements on which the projection falls and the elements which the connector FE connects.

Filter Links To	Projection Components	Projection Elements	Projection and Attached Elements
Spots	 *Figure 253:*	 *Figure 254:*	 *Figure 255:*
Seams	 *Figure 256:*	 *Figure 257:*	 *Figure 258:*
Bolts	 *Figure 259:*	 *Figure 260:*	 *Figure 261:*

Global Options

Autofit

Use this option in combination with the other view option toggle buttons and show, hide, isolate, or isolate only, or in combination with the advanced action buttons. After an action is performed, the newly found connectors and/or entities are placed in the middle of the screen. If this option is used in combination with one of the previously mentioned buttons, it works like a pure fit view.

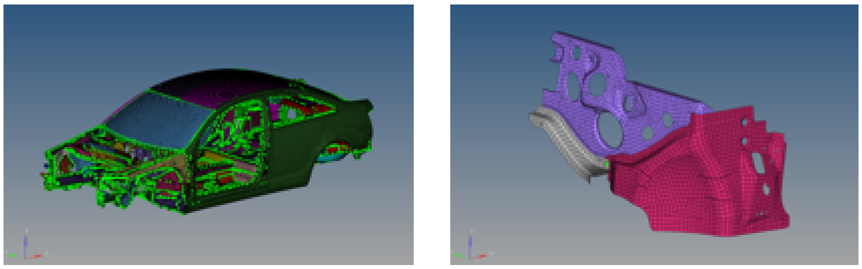


Figure 262:

In this example, the 1st connector and the link entity were Isolated with the fit view option.

Autocolor Visualization Mode

Automatically switch the element color mode when accessing different tabs in the Connector Link Entity Browser.

When enabled, switching to the Part Link view from the Component Link view automatically changes the element color to **by part** and vice versa.

Connector Entity Editor

Use the Connector Entity Editor to quickly view and modify attributes assigned to connector(s).

The Connector Entity Editor opens when you select single or multiple connectors from the Connector Entity Browser. Connector attributes are displayed in the Name column, and the values associated with each attribute are displayed in the Value column. The attributes that display in gray text, cannot be edited.

Once you modify a connector in the Connector Entity Editor, its state changes to modified (blue). A modified state indicates that the representation is not the exact same for the new representation, due to the changes you made. To apply these changes you must rerealize the connector.

For example, assume you have an acm (general) connector with a diameter of 10. If you modify this connector in the Connector Entity Editor to have a constant thickness, edited the diameter, and applied an RBE3 radius of 3, the changes will not be applied until the connector is rerealized.

If you modify the Config Name assigned to a connector in the Connector Entity Editor, a new set of attributes will be assigned. Such a modification will change the connector's state to unrealized (yellow), and the connector will be placed in a new folder.

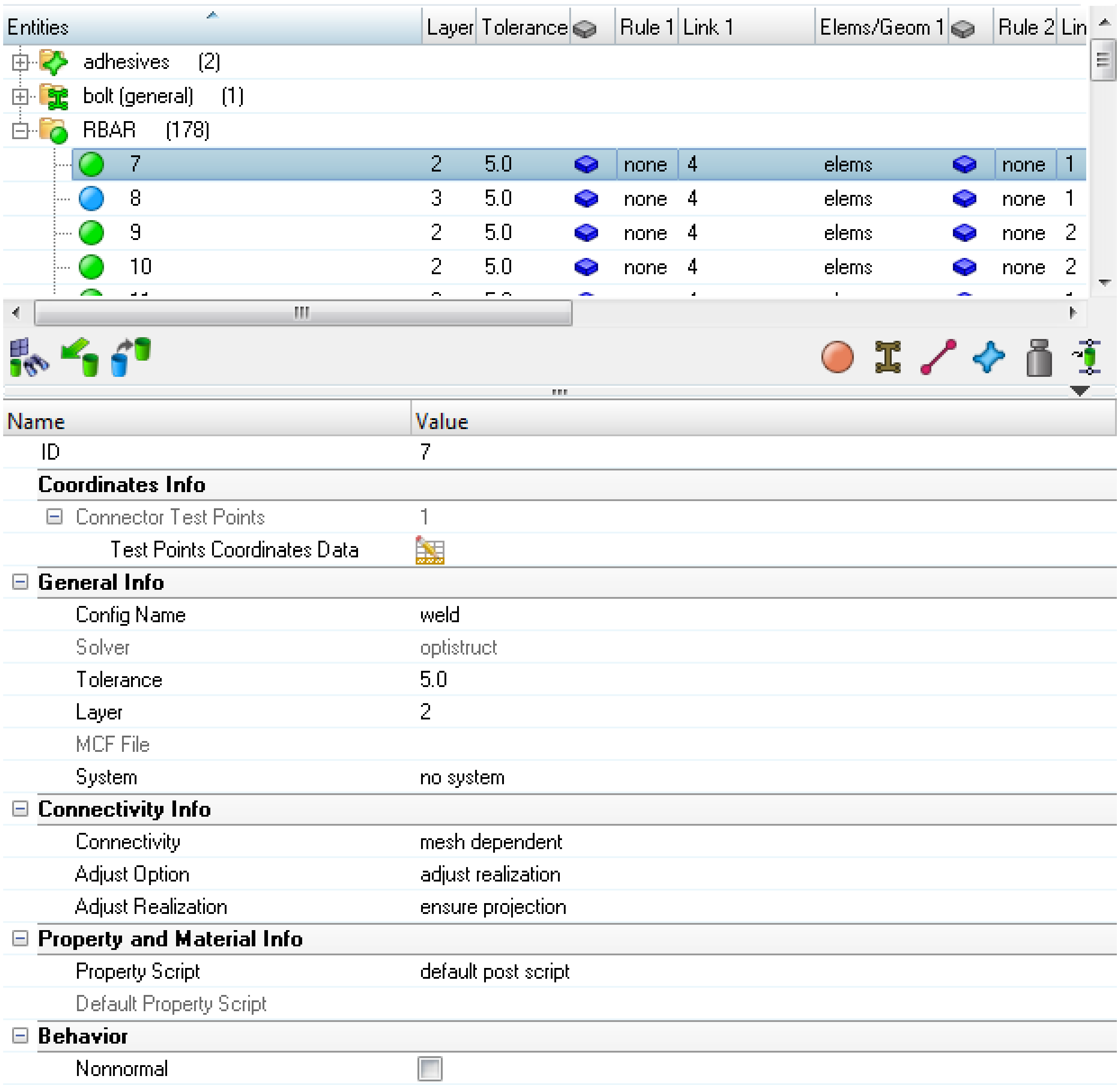

Entities
Layer
Tolerance
Rule 1
Link 1
Elems/Geom 1
Rule 2
Lin
adhesives (2)
bolt (general) (1)
RBAR (178)
7 2 5.0 none 4 elems none 1
8 3 5.0 none 4 elems none 1
9 2 5.0 none 4 elems none 2
10 2 5.0 none 4 elems none 2
Name
Value
ID
7
Coordinates Info
Connector Test Points
1
Test Points Coordinates Data
General Info
Config Name
weld
Solver
optistruct
Tolerance
5.0
Layer
2
MCF File
System
no system
Connectivity Info
Connectivity
mesh dependent
Adjust Option
adjust realization
Adjust Realization
ensure projection
Property and Material Info
Property Script
default post script
Default Property Script
Behavior
Nonnormal

Figure 263:

Utility Tool Set - Connector Browser

This tool set, located at the bottom of the Connector Browser, provides tools for output of a Master Connectors File, opening specific connector-related HyperMesh panels, or utilize specialized visualization options.

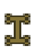
Exports the connector information to a Master Connectors File in XML format (`*.xml`). All the connectors in the browser or currently selected connectors can be exported.

Opens a temporary HyperMesh panel in which elements can be selected. Clicking the **proceed** button finds all the connectors that have the selected elements as their realized FE, and highlights them in the browser. This utility can be used to easily find connectors from their realized welds.

Imports mcf connectors.

Rerealize all of the connectors that have been modified in the Connector Entity Editor with their new settings.

Opens the Spot Panel Connector panel.



Opens the Bolt Panel Connector panel.

Opens the Seam Panel Connector panel.

Opens the Area Panel (Connectors) Connector panel.

Opens the Apply Mass Panel Connector panel.

Opens the FE Absorb Tool Connector GUI.

Link Definition

A link is a reference to a separate entity. One or more links are added to a connector. The entities to which the links refer are connected during realization.

The link definition consists of the following information:

Link Type
: Type of entity that can be added to the selected connector(s) as a link reference. Supported entities are assemblies, components, surfaces, elements, tags, and nodes.

Link Rule
: Defines how a connector treats an entity added as a link. Adding a link with an ID or name rule forces the connector to retain the link's ID or name even if that link entity no longer exists in the database. This aids in part replacement when a new part replaces an old part and both share the same ID or name. Adding a link with the at fe-realize rule ensures that each time a connector is realized the closest entity of the correct type is found and connected. This is useful when connectors need to connect to a closest part in an assembly.

Link ID/name
: The ID or name of the entity added as a link to a connector.
In the **Update Link** and **Add Link** dialog in the Connector Browser, the Link Select row corresponds to the link ID and the link name. Clicking into the fields in this row opens a temporary panel in which a specific entity can be selected. Clicking **proceed** returns to the **Update Link** or **Add Link** dialog. The selected entity is used to add or update a link reference.

Link State
: This defines whether the weld created during connector realization connects to geometry or mesh on the link. This only applies to assembly, component, and surface entities that can contain geometry and/or mesh information.

> **Note:** The link reconnect rule (use name, use id, and so on) and the link state (connect to mesh or geometry) can be viewed in the link column by selecting the extended link information checkbox in the Connector Entity Browser configuration window.

The Connector Browser permits the performance of different actions on the links.

Convert Style

Convert connectors of one specific style into a different style, for example a spot connector into a bolt connector using the **Convert** option.

Before you begin, define Convert Settings.
Highlight the connector(s) you want to convert, then right-click and select **Convert** > **Style** (spot, bolt, seamline) from the context menu.

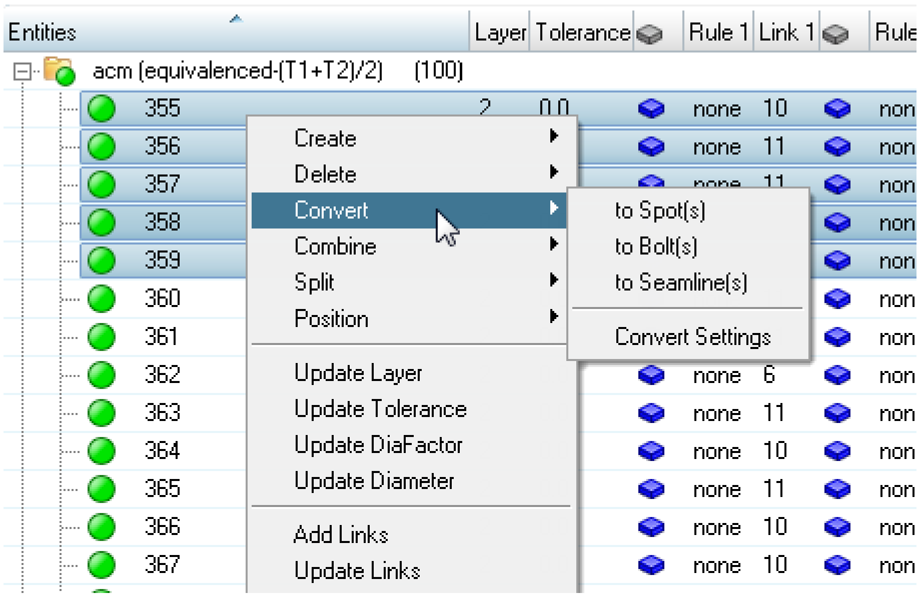


Figure 264:

The converted connector(s) will be unrealized and placed in the Undefined folder. Once you convert a connector, you will not be able to revert it back to its original state. The only information that will be retained includes: number of layers, tolerance, and link information.

Convert Settings

Settings used to define how connectors are converted.

Access **Convert Settings** by right-clicking on a connector and selecting **Convert** > **Convert Settings** from the context menu.

Table 8:

Option	Description
Maintain as Line	To keep line connectors as line connectors after the conversion, select the **Maintain as Line** checkbox. When this checkbox is off, line connectors will be converted to single point connectors per each connector point. **Note:** Only applicable when spot or boltlines are being converted to seamlines.

Combine

Combine aligned single point connectors into line connectors or combine several aligned seam line connectors into larger one(s) using the **Combine** option.

Before you begin, define Combine Settings.

Highlight the connectors you want to combine, then right-click and select **Combine** > **Style** option from the context menu.

Highlighted connectors of a different style are not considered for the combine operation.

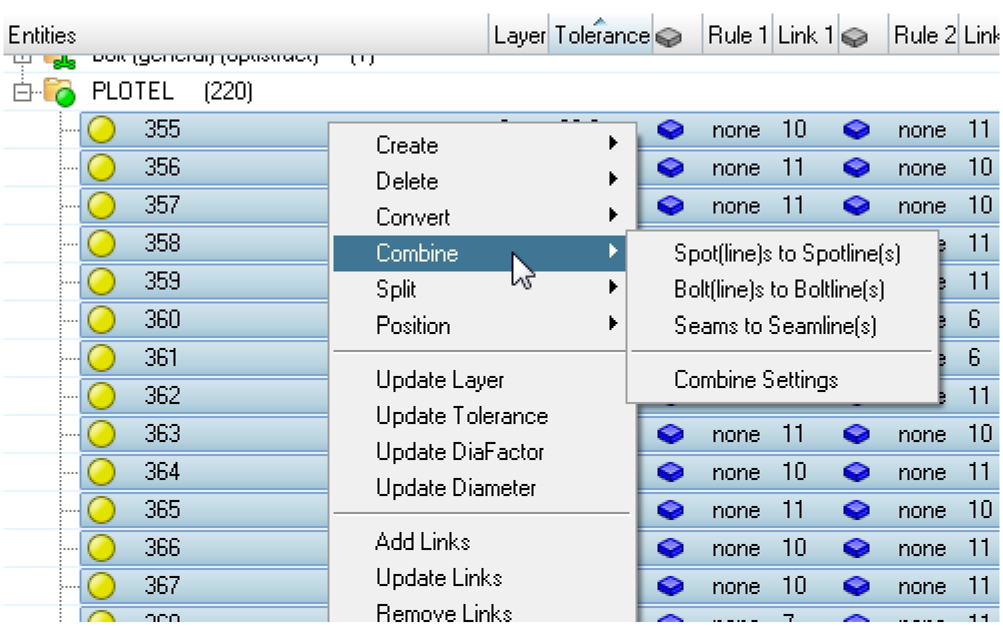


Figure 265:

When combining spots or bolts, the exact original connector positions are kept in the combined connector line. Combined seam connectors will not keep their original positions, instead they will be redistributed along the combined line. This functionality enables realizations to be retained in many cases, when combining realized connectors. Combining realized seam connectors will lead to a new realization of the combined seam. This behavior can be controlled in the **Combine Settings**.

Note: You can only combine connectors that have the same realization type, same link definition, and same number of layers. Further parameters of individual initial connectors cannot be retained, therefore a rerealization might defer from the initial realization. It is recommended that you combine unrealized connectors.

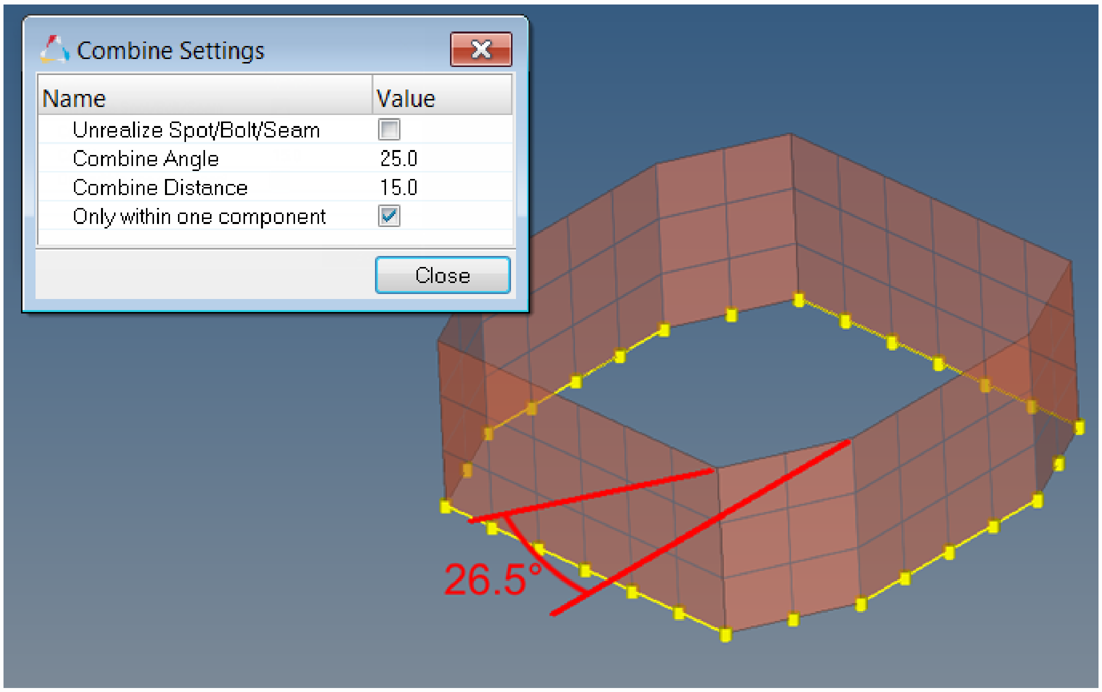
Combine Settings

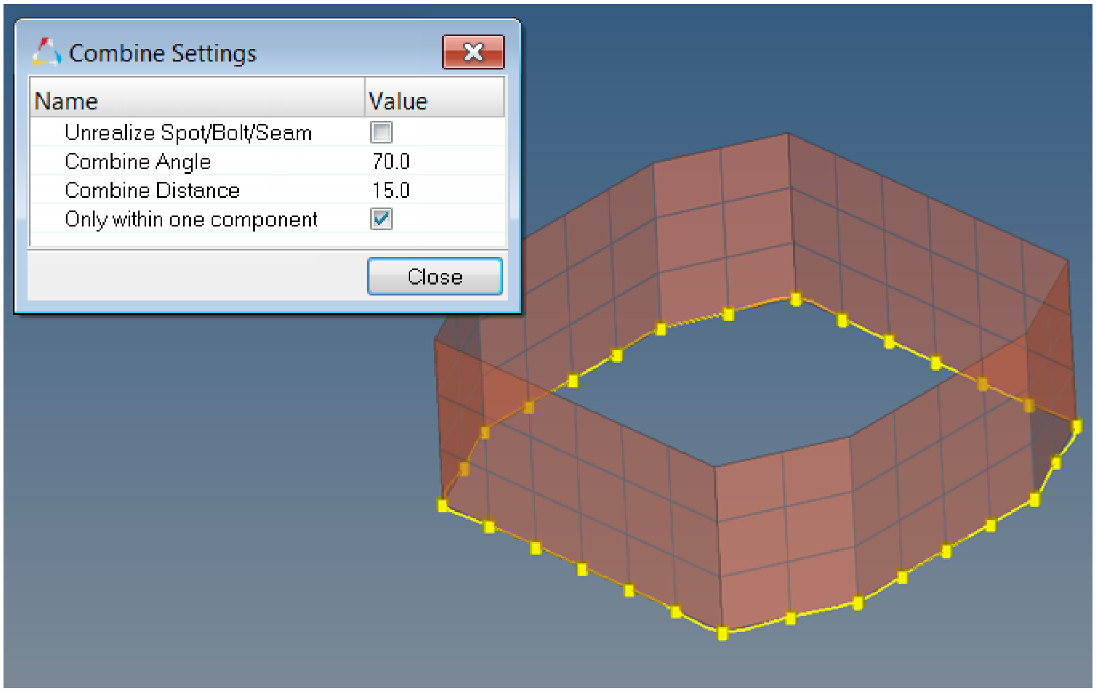
Settings used to define how connectors are combined.

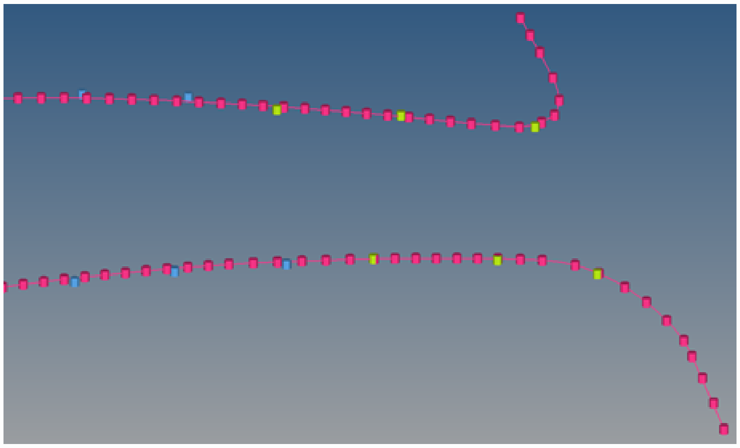
Access **Combine Settings** by right-clicking on a connector and selecting **Combine** > **Combine Settings** from the context menu.

Table 9:

Option	Description
Unrealize Spot/ Bolt/Seam	Unrealize initial connectors.
Combine Angle	Maximum allowed deviation angle measured between three consecutive connectors.

Option	Description
	A combination of these three connectors is allowed only when the angle is smaller than the defined one. This describes, why, at least, a minimum of three point connectors must be selected in order to be combined.  *Figure 266:* The creation of looped line connectors is supported.

Option	**Description**
	 *Figure 267:*
Combine Distance	Maximum distance two consecutive connectors must be from each other in order for the connectors to be combined.
Only within one component	Only combines connectors that are organized in the same component. The combined connector will remain in the same component. When combining connectors that are organized in different components, the new combined connector will be created in the current component. The image below demonstrates a problematic situation for combining connectors.

Option	Description
	 *Figure 268:*

Split

Split spot or bolt line connectors into individual single point connectors using the **Split** option.

Before you begin, define Split Settings.

Highlight the connector(s) you want to split, then right-click and select **Split** > **Style** option from the context menu.

Highlighted connectors of a different style are not considered for the split operation.

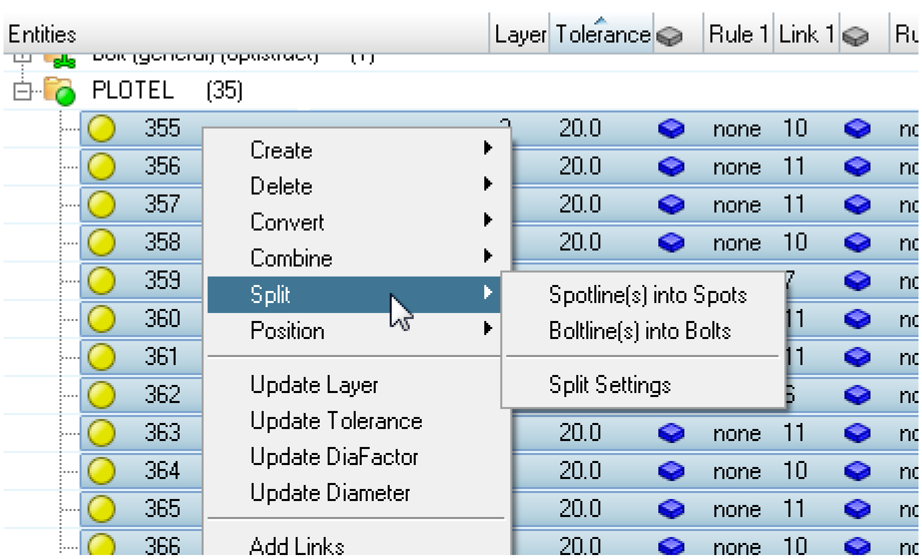


Figure 269:

Unless requested from the split settings, the split operation of a realized line connector will not unrealize the individual single point connectors. Newly created split connectors will be created in the component of the original connector.

Split Settings

Settings used to define how connectors are split.

Access **Split Settings** by right-clicking on a connector and selecting **Split** > **Split Settings** from the context menu.

Table 10:

Option	Description
Unrealize	During the Split operation of a realized line connector, the individual single point connectors will be unealized.

Position

Place a connector in-between the center of the two farthest defined links or its source position using the **Position** option.

Before you begin, define Position Settings.

Restriction: Bolt and area connectors are not supported for the positioning.

Highlight the connector(s) you want to position, then right-click and select **Position** > **<style>** from the context menu.
The source position is the position of the connector before it has been positioned in the center.

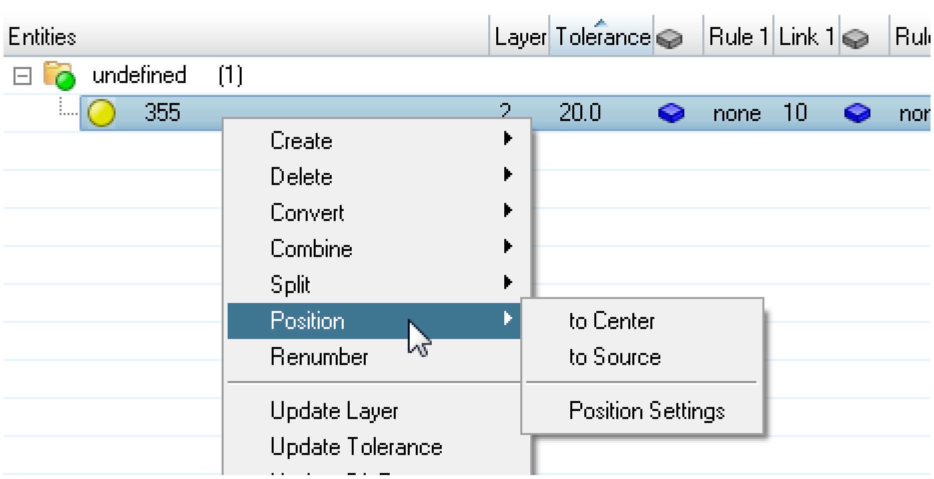


Figure 270:

In the case of line connectors, each individual connector point is centered.

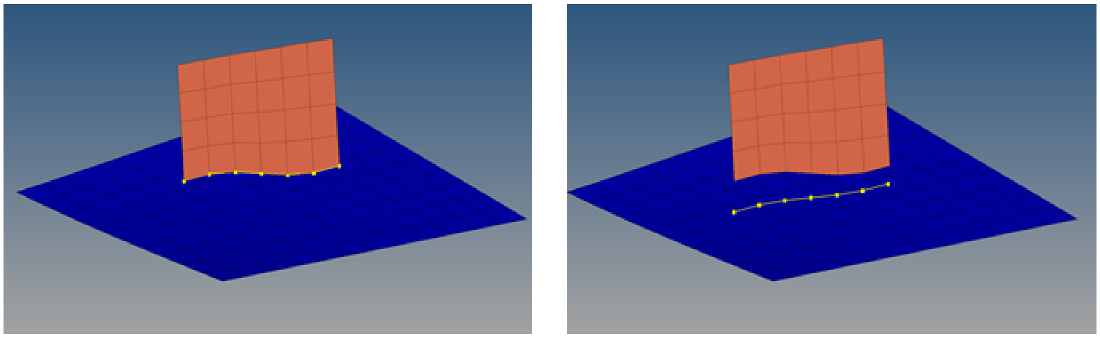


Figure 271:

Note: The source position of a connector is saved on the connector during the centering, therefore it is possible to go back to the origin position. There are certain actions, which reset the source position to the current one. This happens, for example, when splitting or translating a connector. Also, the source position is only available in the current session; it will not be saved in either the HyperMesh file or in the `.xml` file.

If a connector is defined with the link conservation option use extra links, it might be that positioning to the center provides unexpected results because all of the links with valid projections are considered, even if they are not needed for the final realization.

Position Settings

Settings used to define how connectors are positioned.

Access **Position Settings** by right-clicking on a connector and selecting **Position** > **Position Settings** from the context menu.

Table 11:

Option	Description
Consider only	These settings help to filter down the connector selection to those whom should be considered for the intended positioning. Exclusively connectors with the checked attributes are considered. **Spots** Consider spot connectors. **Seam** Consider seam connectors. **Unrealized** Consider unrealized connectors. **Realized** Consider realized connectors.

Option	Description
	Failed Consider failed connectors. **=2t** Consider connectors with a number of layers equal to 2. **>2t** Consider connectors with a number of layers greater than 2.
Upon Positioning	Functionality to use for FE representation during or after the positioning of connectors that have already been realized. **Unrealize** Unrealize all of the connectors being considered for positioning. Registered FE elements will be deleted. **Rerealize** Realize all of the connectors being considered for positioning, starting from their new position. **Accept wo Reconfirmation** Place all of the connectors being considered for positioning in their new location without doing anything to the realized FE. This option is only recommended for simple realization types with smaller extensiveness, since these use a similar projection logic than the center positioning. There is always the risk that a formerly successful realized connector fails after the center positioning. The risk increases with the complexity of the realization type.

Add Link

Add links (component, assembly, surface, element, tag, node) at a time to one or more connectors using the **Add Link** option.

1. Right-click on the connector(s) that you wish to add links to, and then select **Add Link** from the context menu.

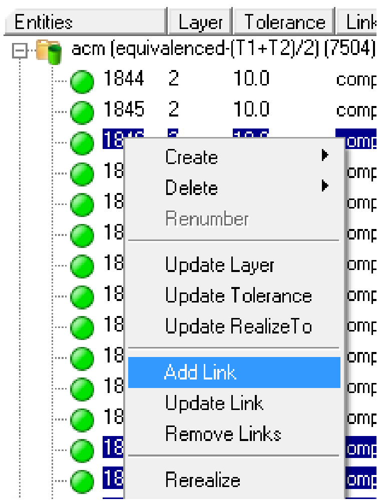


Figure 272:

The Add Link window opens at the bottom of the Connector Browser.

> **Tip:** All of the connectors highlighted in the connector list at the time the Add Link tool is enabled will be affected. If you wish to cancel the add link function, click **Reject**, located in the upper right-hand corner of the Add Link tool.

The Add Link window is laid out in a table format, with each Add Link option displayed in the Name column, and the controls and values associated with each option displayed in the Value column to the right.

Figure 273:

2. Edit the options in the Add Link window.

Option	Description
Add Link	Determine how you will be adding links. Click the **Value** column to select one of the two Add Link options from the pull-down menu: **From scratch** The connector gets into an unrealized state without any link information. When selected, previous link definitions have to be removed, but the Number of Layers and Tolerance can be reused. **To existing** A pure adding of links following the other settings. If no links are added, the connector will remain untouched and in a realized state, as long as it was realized before.
Layers	Select the number of thicknesses (layers) for the connectors. Click the **Value** column to select one of the following Layers options from the pull-down menu: **Keep previous** Allows you to work with a different number of layers at one time. **Auto by tol.** When this option is set, a tolerance needs to be defined. All link candidates found in the tolerance will be added as links. **Note:** This option is only available when From scratch is selected above. **Increment +** Activates the Value column in the following row, where you can enter the number of additional layers to be added to the current number of layers for each selected connector. **Note:** This option is only available when To existing is selected above. **Number** Activates the Value column in the following row, where you can enter the number of layers to display.
Tolerance	Select the tolerance to be used for link detection. If the link candidates are close enough to the connector (within the tolerance), then they could become links as long as other conditions are not fulfilled. Click the **Value** column to select one of the following Tolerance options from the pull-down menu:

Option	Description
	Keep previous Allows you to work with different tolerances at one time. **Value** Activates the Value column in the following row, where you can manually enter a tolerance. **None** Despite how far away the next links are, they will still be added. By default, the tolerance is set to 0.0.
Link Type	Select the type of entity that can be added to the selected connector(s) as a link reference.
Links	Select which entities in the model to which the connector will link to. The selection made from the Link Type option above will influence what is displayed in this option. Click the **Value** column to select one of the following options from the pull-down menu: **All** Selects all of the entities in the model to link to. **Displayed** Selects only the entities that are displayed in the model to link to. **Select** Opens the Entities selector in the following row, from which you can manually select entities to link to.
Link State	Determines whether the weld created during connector realization connects to geometry or mesh on the link. This option is significant for link detection. Only the entities of the right state, meaning elements or surfaces, will be taken into account. If the link state is set to geometry, then close elements will not be considered for link detection. Click the **Value** column to select one of the following Link State options from the pull-down menu: **elems** Specifies that the entity needs to be connected (welded) using its mesh. **geom** Specifies that the entity needs to be connected (welded) using its geometry (connect surfaces only).
Link Rule	Defines how a connector treats an entity added as a link.

Option	Description
	Click the **Value** column to select one of the following Link Rule options from the pull-down menu: **At fe realize** Ensures that each time a connector is realized the closest entity of the correct type is found and connected. This is useful when connectors need to connect to a closest part in an assembly. **By ID** Forces the connector to retain the link’s ID even if that link entity no longer exists in the database. This aids in part replacement, when a new part replaces an old part and both share the same ID. **By name** Forces the connector to retain the link's name even if that link entity no longer exists in the database. This aids in part replacement when a new part replaces an old part and both share the same name. **None** No link rule is created.

Remove Links

Remove links from one or more connectors at a time.

Remove Links is available from the Connector Entity Browser's context menu when you right-click in the **Entities** or **Link** columns.

Once links are removed, updated connectors will be unrealized and your changes will be permanently updated in the database and browser.

Remove All Links From Connectors

1. Select the connectors from the browser for which all the links needs to be removed.

   > **Tip:**
   >
   > The connectors can also be selected from graphics by using the Selector button in the Connector Entity Browser Action Mode Tools. The selected connectors in the graphics area will be highlighted in the browser tree.

2. Open the default context menu by right-clicking in the **Entities** column of the tree list.
3. Select **Remove Links** to confirm and execute the operation.

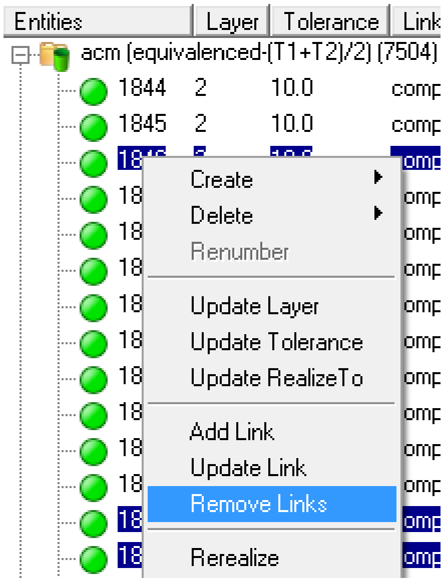


Figure 274:

Remove Specific Links From Connectors

1. Select the link(s) to remove, then right-click and select **Remove Links** from the context menu.
2. In the **Confirm** dialog, click **Remove link**.

Update Links

Update the attributes associated with a link, such as reconnection rule, link references, and link state.

Update link operations are frequently used for:

- Part replacement
- Modify link rules
- Modify link states

If a part replacement needs to be performed, the browser provides quick tools to update one or more connectors with the link referencing the new redesigned part. The following sections present steps to edit and update the link rule and the link state attributes as well as the part replacement.

You can update links in the following ways:

- Directly edit column data in the Connector Browser.
    a) In the Connector Browser, select the link(s) or connector(s) to update.
    b) Click a field to edit the current value or select a new value.

  In the Link Entity Browser, updates can be performed on a single or multiple selected links. When you update a link in the Link Entity Browser, link references will be automatically updated in the

Connector Entity Browser. In the Connector Entity Browser, updates can be performed for an individual link on a single or multiple selected connectors in the Connector Entity Browser.

For example, quickly change the reconnection rule from the Rule column, or change the link state from the Elems/Geom column. Update links for the purpose of part replacement by entering a new link ID or link name in the Link Entity Browser. Part replacement can also be performed on single or multiple selected connectors in the Connector Entity Browser by entering a new link ID or link name in the Link column.

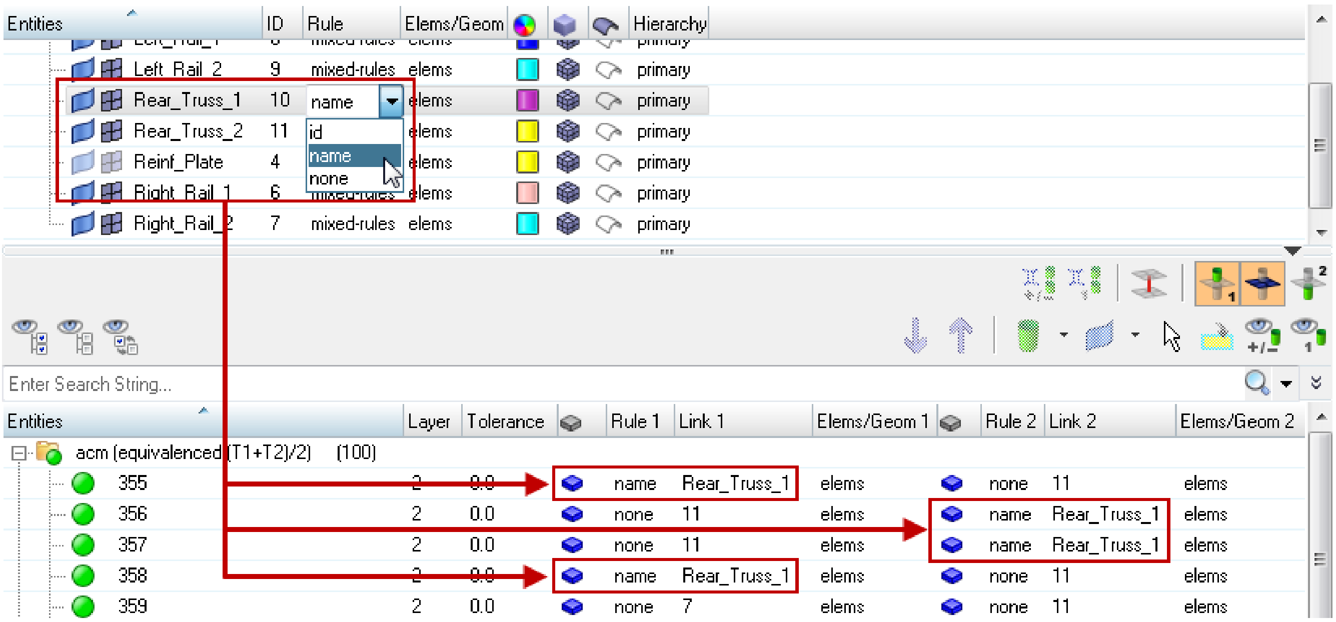


Figure 275:

- Using the **Update Link** option in the Connector Browser's right-click context menu.
  a) Select the connectors from the browser for which one or more links need to be updated.

     The connectors can also be selected from the graphics area by using the Selector button in the action modes tool set. The connectors selected in the graphics area will be highlighted in the browser tree.
  b) Right-click one of the highlighted connectors and select **Update Link** from the context menu. An **Update Link** dialog opens at the bottom of the Connector Browser. All highlighted connectors will be affected by the subsequent update link action.

     > **Note:** If the **Update Link** dialog is still open from a previous update link operation, all blue highlighted connectors will be affected when the update link function is performed again.

     The lines for Link State and Link Rule are grayed out by default in the **Update Link** dialog. When the extended information in the Connector Entity Browser configuration window is activated, these lines become active as well.

c) Fill in the fields in the Search column.

Due to this selection, the links of the selected (highlighted) connectors are filtered down to the links which fit to all given search attributes. Only the remaining links are taken into account for the subsequent attribute replacement.

> **Note:** Not all of the search attributes have to be defined; an asterisk can be used.

d) Fill in the fields in the Replace column.

All the remaining links will be updated with the attributes defined by these attributes.

> **Note:** Not all of the replace attributes have to be defined; an asterisk can be used. The prior attributes are maintained.

e) Click **Update** to acknowledge and execute the operation.

Note: Not every combination of search and replace attributes are valid. For example, it is not possible to select the asterisk for the Link Select field in the Search column and replace it with a concrete link. This helps to prevent global creation of unwanted modifications.

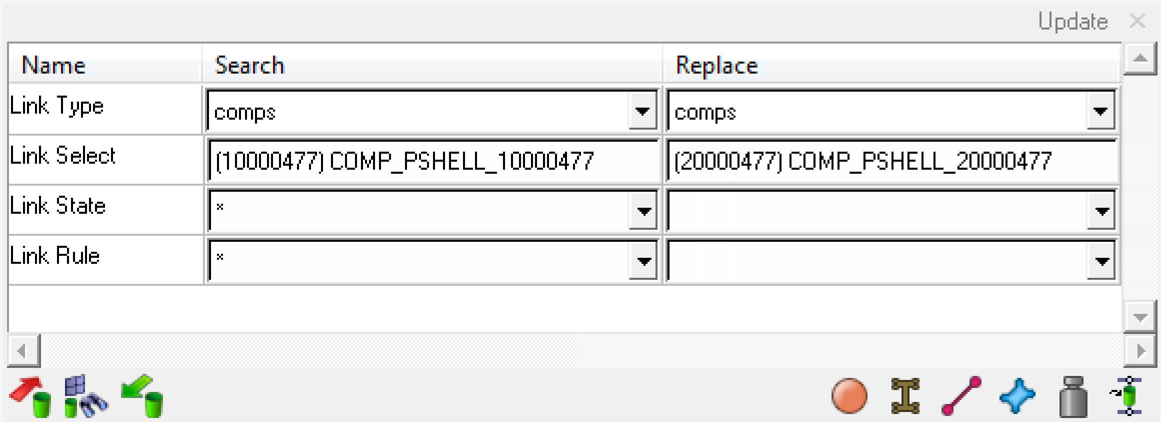


Figure 276:

Modify Link Rule

The link rule defines how a connector treats an entity added as a link.

Adding a link with the ID or name rule forces the connector to retain the link's ID or name even if that link entity no longer exists in the database. Adding a link with the at fe-realize rule ensures that each time a connector is realized the closest entity of the correct type is found and connected.

Update Link by Directly Editing Column Data in the Connector Browser

In the Connector Browser, select the link(s) or connector(s) to update, then click the **Rule** column and select a new link rule.

In the Link Entity Browser, updates can be performed on a single or multiple selected links. When you update a link in the Link Entity Browser, link references will be automatically updated in the Connector Entity Browser. In the Connector Entity Browser, updates can be performed for an individual link on a single or multiple selected connectors in the Connector Entity Browser.

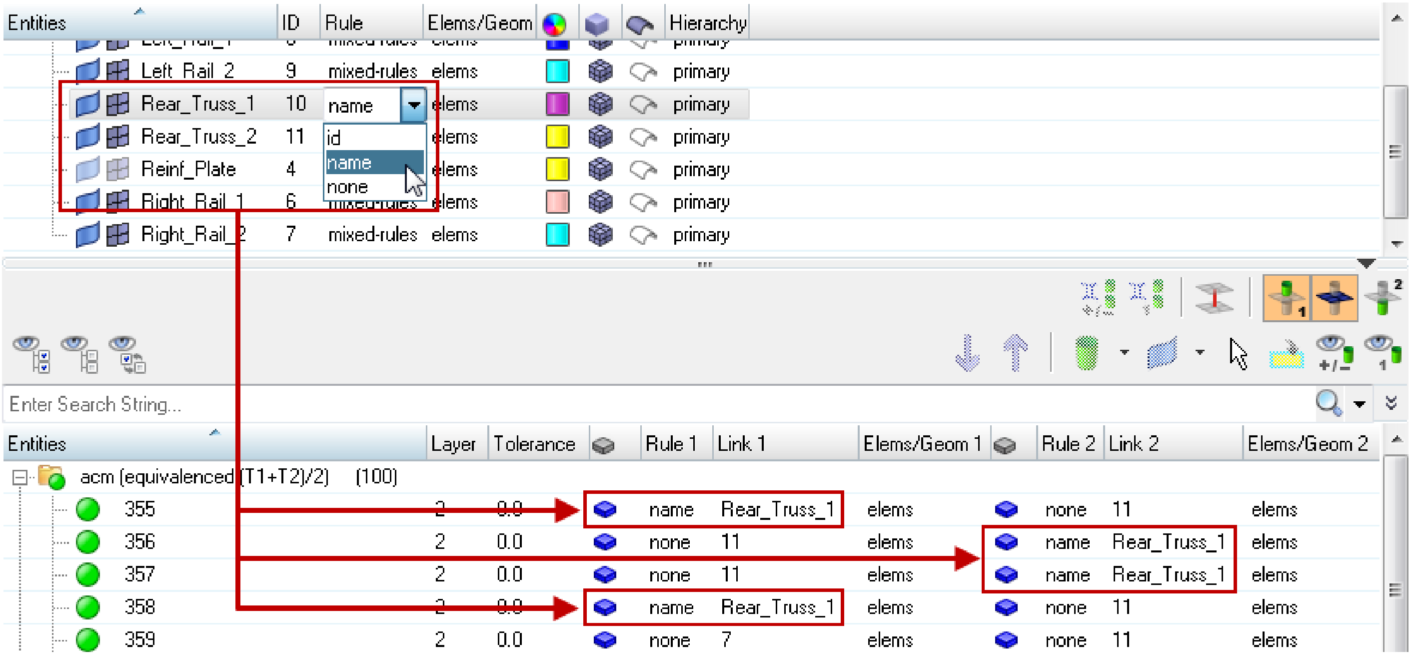


Figure 277:

Update Link Using the Update Link Option in the Connector Browser's Right-Click Context Menu

1. In the Options tab of the Connector Entity Browser configuration window, select the **extended information** checkbox to activate the lines for Link State and Link Rule in the **Update Link** dialog.
2. Select one or more connectors from the browser, then right-click and select **Update Link** from the context menu.

> **Tip:** The connectors can also be selected from graphics by using the Selector button in the Connector Browser tool set. The selected connectors in the graphics will be highlighted in the browser tree.

3. In the Search column's Link Rule list box, select the link rule to search.
4. Under the Replace column Link Rule list box, select the new rule.
5. Click **Update**.

> **Note:** The link column for the selected connectors will now display the modified information. The connector's state remains unchanged.

After performing the modification, the current link rule has changed for the selected links.

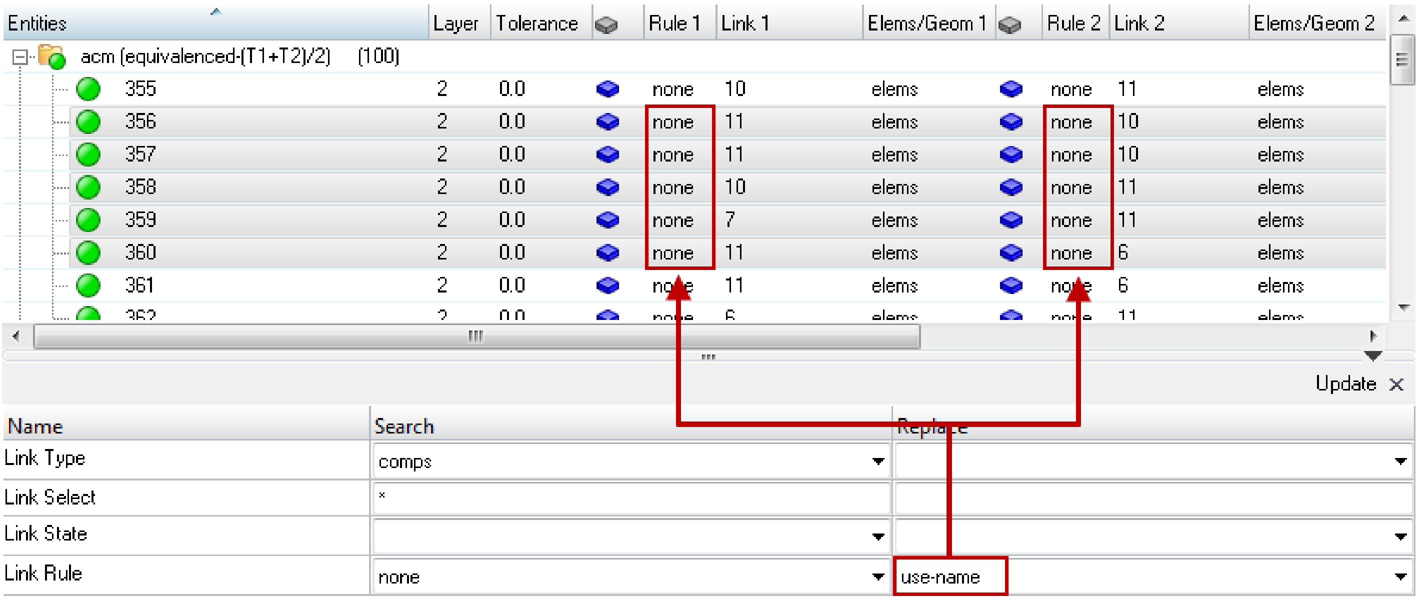


Figure 278: Starting Link Rule

The starting link rule is underlined in red. Note that use-id is specified as the replacement link rule.

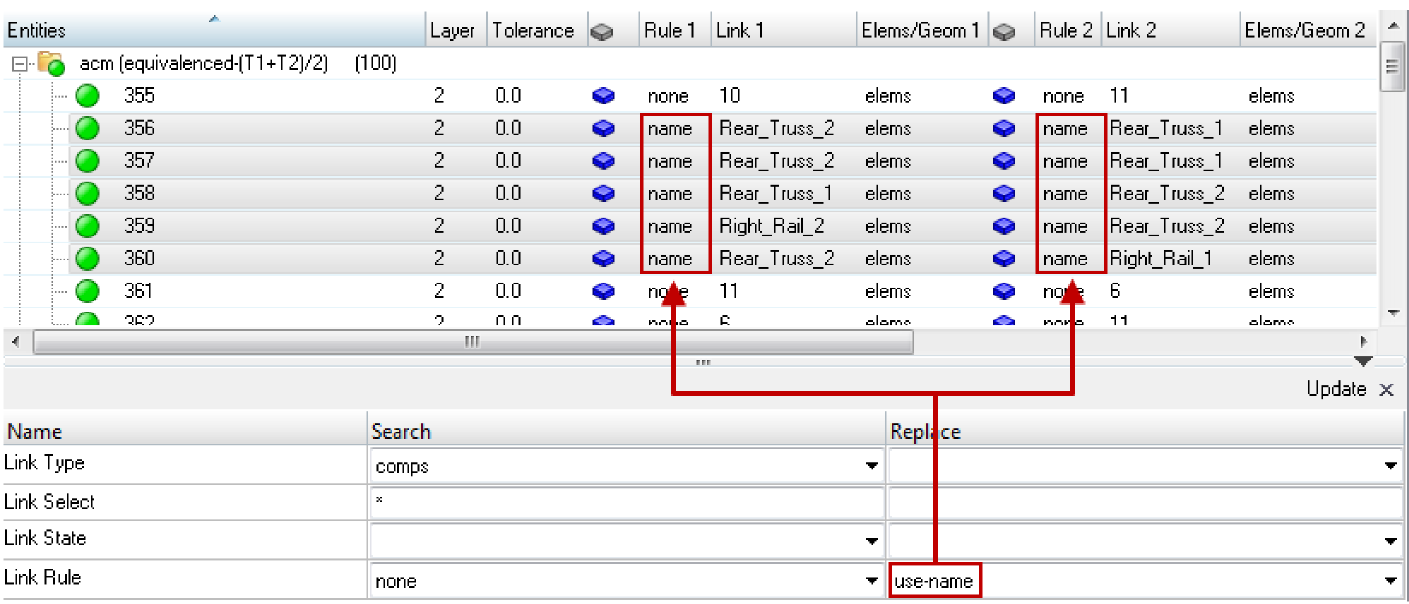


Figure 279: Updated Link Rule
The newly replaced link rule is underlined in blue.

Modify Link State

The link state defines if the weld created during connector realization connects to geometry or mesh on the link.

This is applicable to only components and surfaces entities that can contain geometry and/or mesh information.

Update Link by Directly Editing Column Data in the Connector Browser

In the Connector Entity Browser, select the link(s) or connector(s) to update, then click the **Structure** column and select a new link state.

Update Link Using the Update Link Option in the Connector Browser's Right-Click Context Menu

1. Select one or more connectors from the browser, then right-click and select **Update Link** from the context menu.

   > **Tip:** The connectors can also be selected from graphics by using the Selector button in the Connector Browser tool set. The selected connectors in the graphics will be highlighted in the browser tree.

2. In the Search column's Link State list box, select the link state to search.
3. Under the Replace column Link Sate list box, select the new state.
4. Click **Update**.

   > **Note:** The link column for the selected connectors will now display the modified information. The updated connectors will be unrealized and its realized welds will be permanently removed. If the link state was switched for a link from elems to geom, then realizing the connector again will result in the weld connecting to a surface (geometry) contained in that entity.

After performing the modification, the current link state has changed for the selected links.

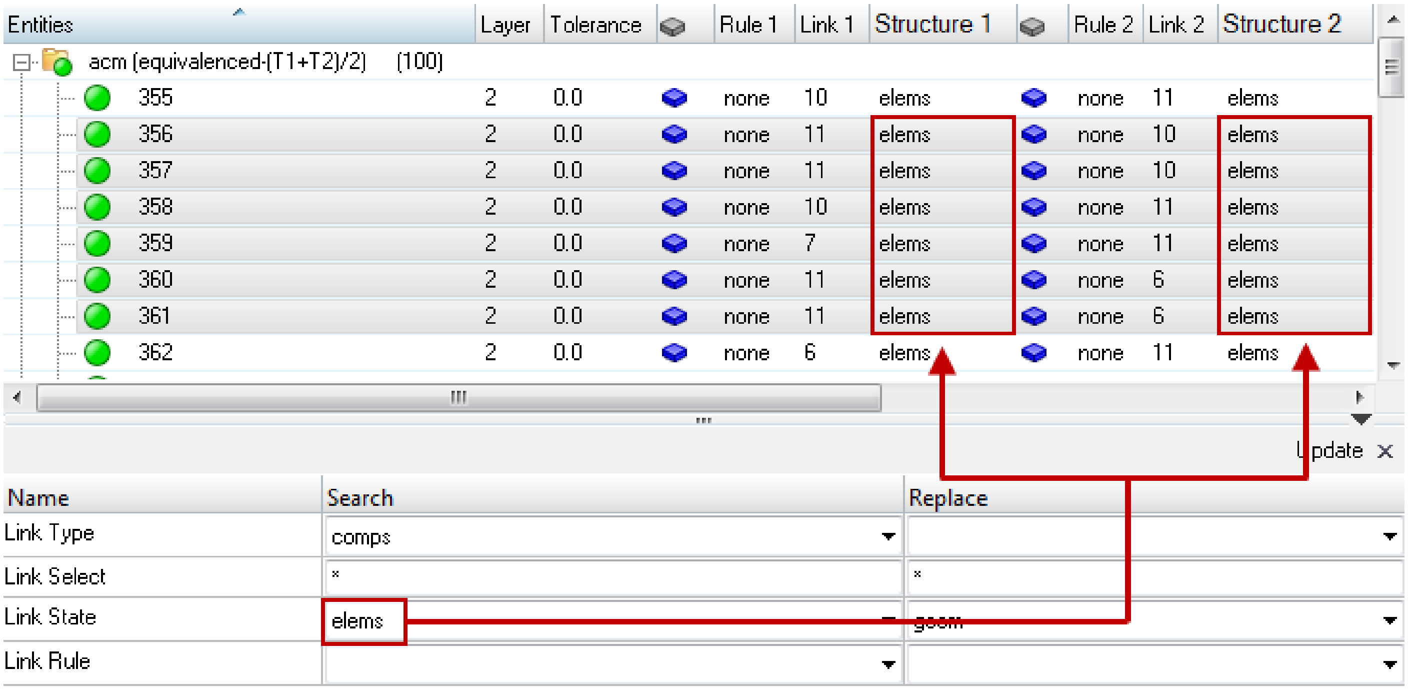


Figure 280: Starting Link State

The starting link state is underlined in red. Note that use-id is specified as the replacement link state.

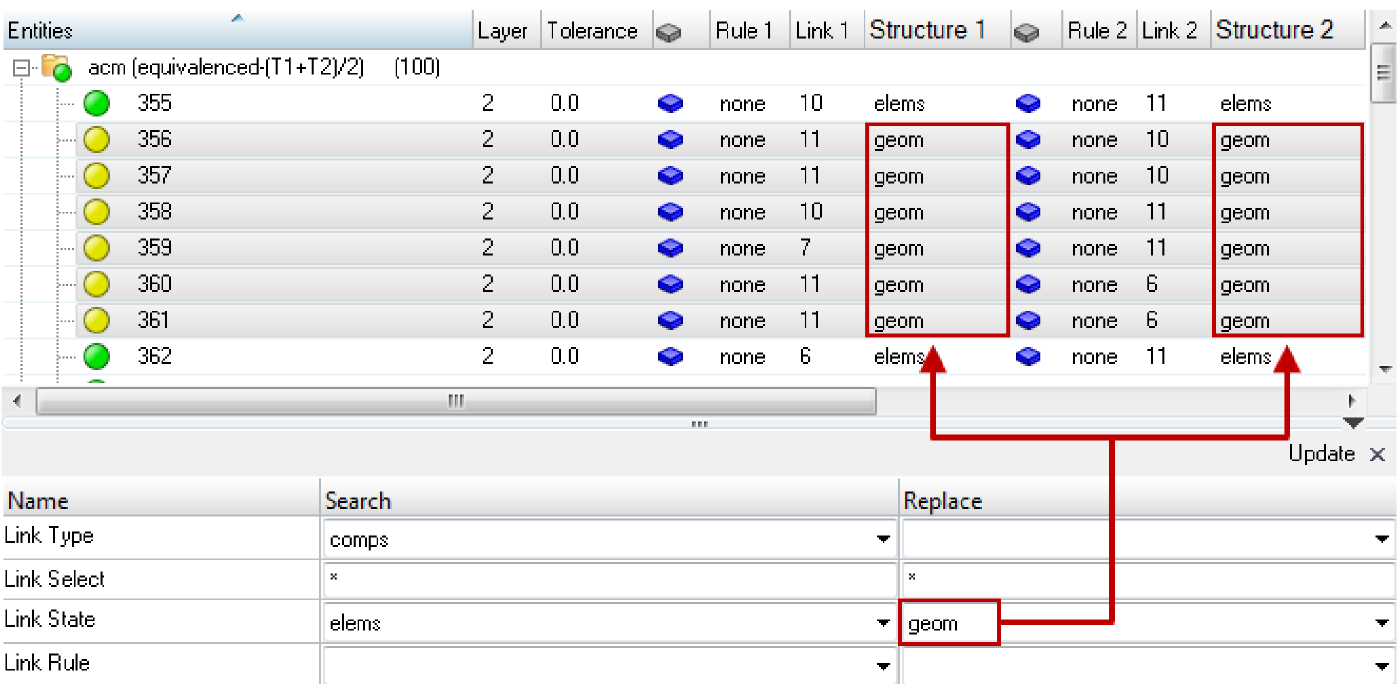


Figure 281: Updated Link State

The newly replaced link state is underlined in blue.

Replace Part

Update links for the purpose of part replacement (link reference).

Use the **Update Link** option available in the Connector Entity Browser's context menu to update a single link reference in one or more connectors. The connectors whose links were modified during this operation will be unrealized and the connector's realized elements will be permanently removed.

Update Link by Directly Editing Column Data in the Connector Browser

In the Connector Browser, select the connector(s) to update, then click the **Link** column and enter a new link ID or link name.

- In the Link Entity Browser, update links for the purpose of part replacement (link reference) by entering a new link name in the Entities field or link ID in the ID field.

  All link references to the old part will be updated to reference the new part in the Connector Entity Browser.

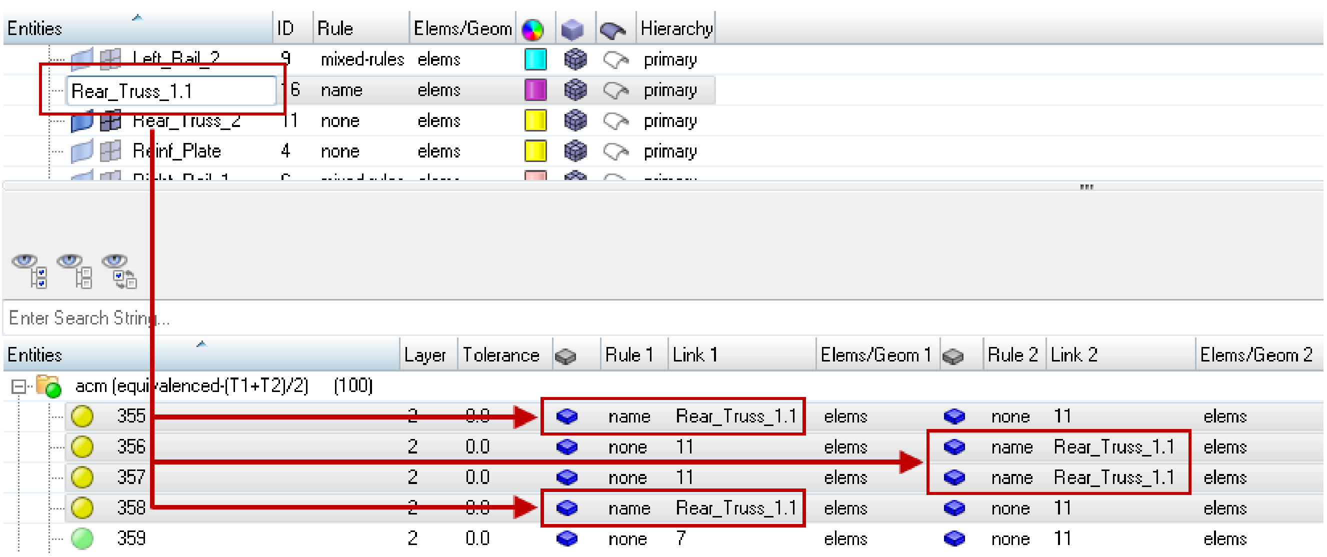
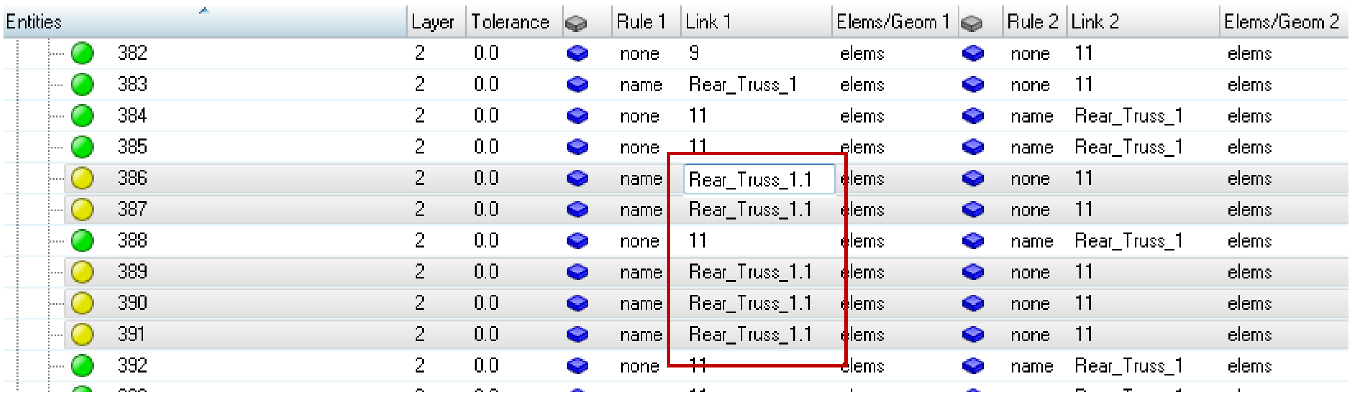
  

  Figure 282:

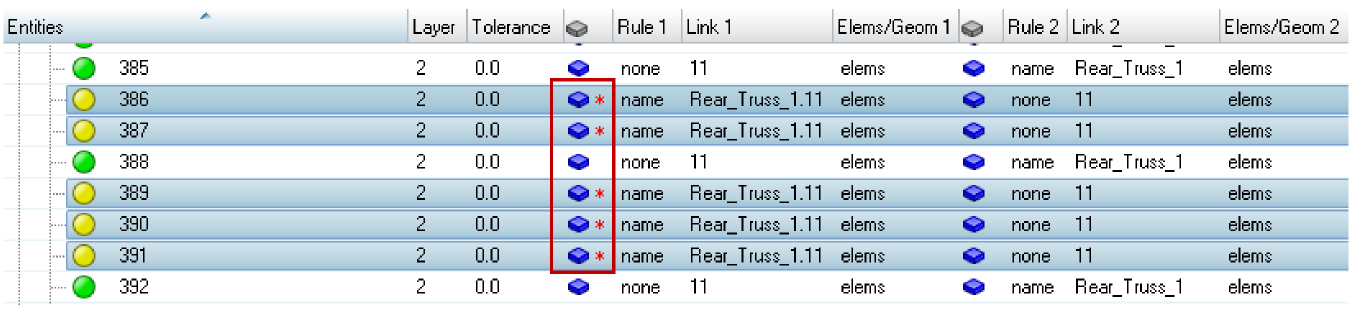
- In the Connector Entity Browser, perform part replacement on single or multiple selected connectors by entering a new link ID or link name in the Link column.



Entities	Layer	Tolerance		Rule 1	Link 1	Elems/Geom 1		Rule 2	Link 2	Elems/Geom 2
382	2	0.0		none	9	elems		none	11	elems
383	2	0.0		name	Rear_Truss_1	elems		none	11	elems
384	2	0.0		none	11	elems		name	Rear_Truss_1	elems
385	2	0.0		none	11	elems		name	Rear_Truss_1	elems
386	2	0.0		name	Rear_Truss_1.1	elems		none	11	elems
387	2	0.0		name	Rear_Truss_1.1	elems		none	11	elems
388	2	0.0		none	11	elems		name	Rear_Truss_1	elems
389	2	0.0		name	Rear_Truss_1.1	elems		none	11	elems
390	2	0.0		name	Rear_Truss_1.1	elems		none	11	elems
391	2	0.0		name	Rear_Truss_1.1	elems		none	11	elems
392	2	0.0		none	11	elems		name	Rear_Truss_1	elems

Figure 283:

Note: When unresolved entities are referenced in the Link field, a red asterisk will appear next. Once the entity is resolved, the asterisk will disappear.



Entities	Layer	Tolerance		Rule 1	Link 1	Elems/Geom 1		Rule 2	Link 2	Elems/Geom 2
385	2	0.0		none	11	elems		name	Rear_Truss_1	elems
386	2	0.0	*	name	Rear_Truss_1.11	elems		none	11	elems
387	2	0.0	*	name	Rear_Truss_1.11	elems		none	11	elems
388	2	0.0		none	11	elems		name	Rear_Truss_1	elems
389	2	0.0	*	name	Rear_Truss_1.11	elems		none	11	elems
390	2	0.0	*	name	Rear_Truss_1.11	elems		none	11	elems
391	2	0.0	*	name	Rear_Truss_1.11	elems		none	11	elems
392	2	0.0		none	11	elems		name	Rear_Truss_1	elems

Figure 284:

Update Link Using the Replace Link Option in the Link Entity Browser's Right-Click Context Menu

1. In the Link Entity Browser, right-click on a link and select **Replace Link** from the context menu.
2. Select a replacement part.
   - In the panel area, use the comps selector to select a replacement component.
   - In the graphics area, select a component.
3. Click **proceed**.
4. In the **Confirm** dialog, click **Replace**.

Update Link Using the Update Link Option in the Connector Browser's Right-Click Context Menu

1. Optional: Identify and select the connectors referencing to the link entity to be replaced.

   This step is not essential, but gives you a better overview; you could also select all connectors.

   Using these settings in the Link Entity Browser and right-clicking on the supported entity to be replaced will isolate the entity and all connectors referencing it in the graphics.

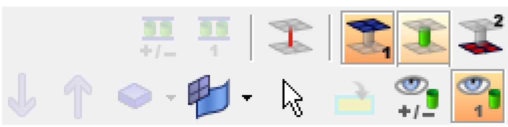
   

   Figure 285:

   By using the Selector button in the Connector Entity Browser you can chose all connectors in a window selection in the graphics. The selected connectors appear highlighted in the browser.

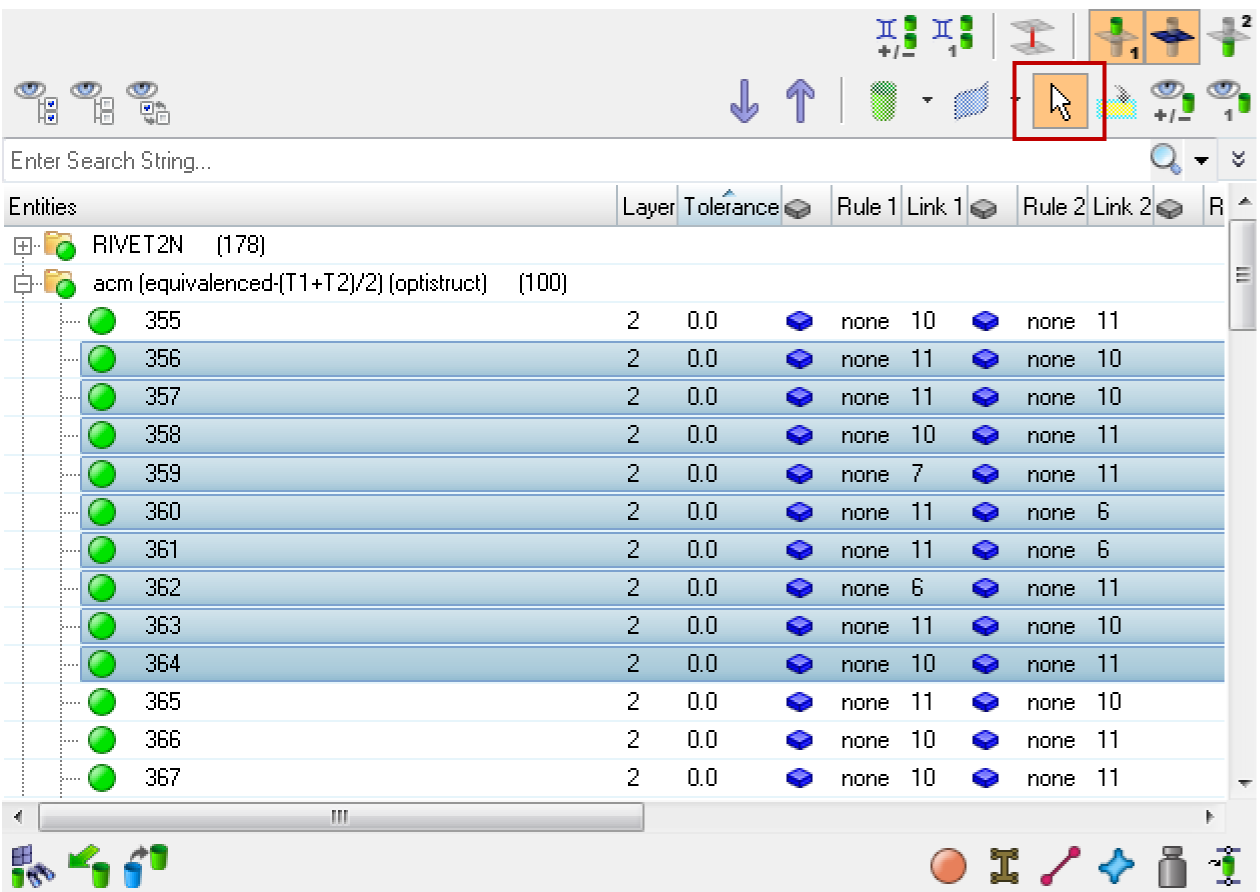
   

   Figure 286:

2. Right-click one of the highlighted connectors in the Entities column to get access to the full context menu and select the **Update Link** function.
   An **Update Link** dialog opens at the bottom of the Connector Browser. All highlighted connectors will be affected by the subsequent update link action.

> **Note:** The lines for Link State and Link Rule are grayed out by default in the **Update Link** dialog. When the extended information in the Connector Entity Browser configuration window is activated, these lines become active as well.

3. For pure part replacements, simply select the supported entity that needs to be replaced from the entity list by clicking on the Link Select field under the Search column, as shown below.

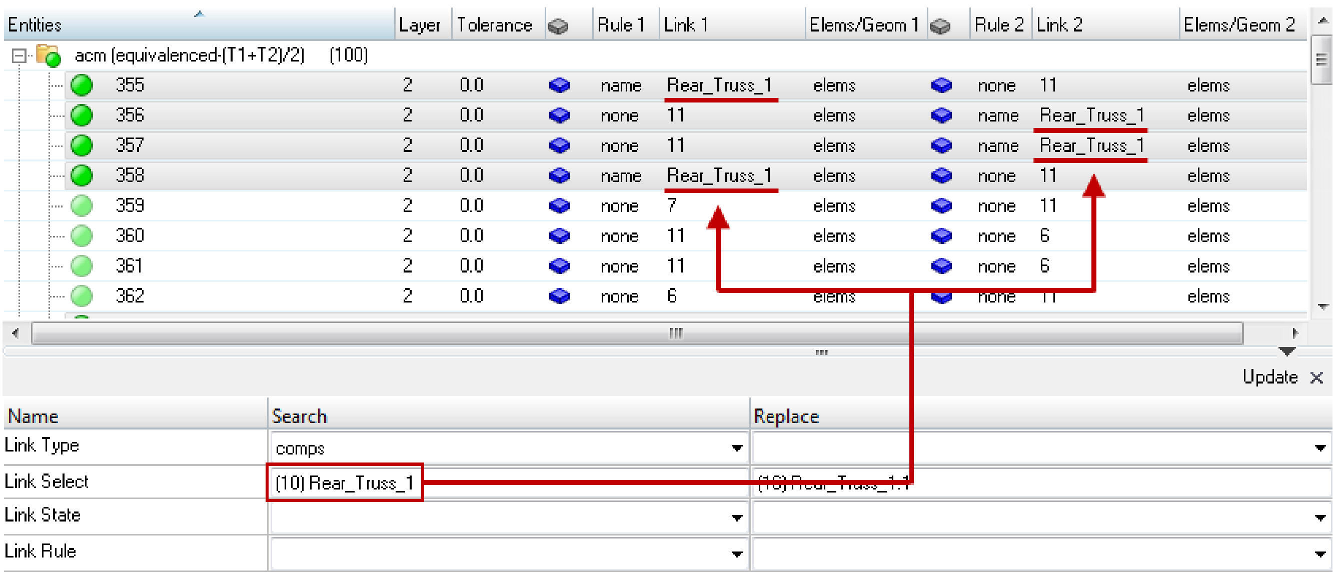


Figure 287:

4. After selecting the entity, click **proceed** to return to the **Update Link** dialog.

> **Note:** Due to the attributes given in the Search column the links of the selected (highlighted) connectors are filtered down to the links which fit all given search attributes. Only the remaining links are taken into account for the following attribute replacement.

5. Select the supported entity that needs to replace the previously selected one from the entity list by clicking on the **Link Select** field in the Replace column, as shown below.

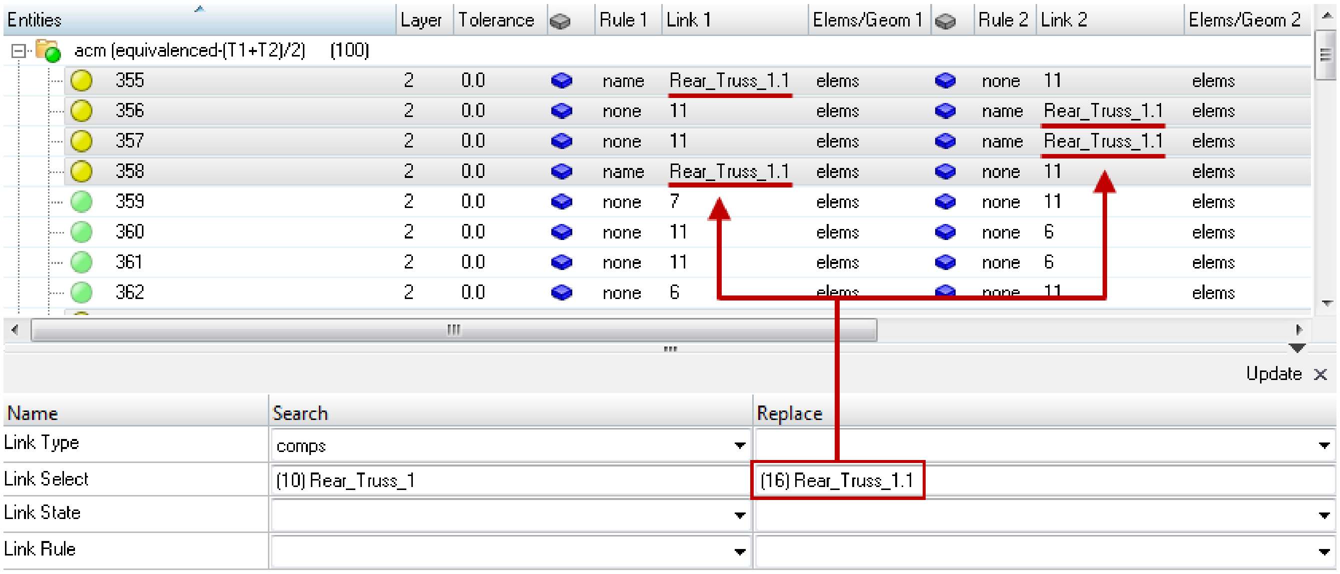


Figure 288:

6. After selecting the entity, click **proceed** to return to the **Update Link** dialog.
7. Click **Update** to execute the part replacement operation.

> **Note:** The updated connectors will be unrealized and its realized welds will be permanently removed. The connectors can be realized again without providing any inputs by using the Rerealize context menu operation.
>
> Rerealize calls the `*CE_Realize` command to realize connectors by accepting only a connector mark. The underlying assumption in the command is that each connector passed in the mark has the required information to be successfully realized.
>
> The required information such as tolerance, weld configuration, diameter, and so on is not defined for connectors created using the FE Absorb utility; hence the Rerealize feature in the Connector Browser works only for connectors that were realized through the Connector panel.

Convert Links

Existing connector links can be converted to one of the following link types: Part, Comp, or Prop. Connector link conversion requests a clear 1:1 relation between a component and a property, otherwise the conversion is not unique and will not be performed.

Links can be converted in the Connector Entity Browser.

Right-click on the connector icon and select **Convert Links** from the context menu to convert all of the links of all the selected connectors at once.

For a single link conversion right-click on the exact link and select **Convert Link** from the context menu.

Link to be converted		Link to Convert To					
		Part		Comp		Prop	
		If Single	If Multiple	If Single	If Multiple	If Single	If Multiple
	Part	NA	NA	Comp Link	Comp Group Link	If all Comp(s) hosted by this Part have the same Prop assigned, then this Prop is added as a Prop Link. SPECIAL CASE: If there are NO comps in this Part, but an exclusive Prop is hosted in this Part, then this Prop is added as a Prop Link.	If the Comps in this Part all have one Prop assigned (1:1 enforcement), then these Props are added as a Prop Group Link.
	Part Group	NA	NA	Comp Link (unlikely)	Comp Group Link	Same as above (just multiple parts).	Same as above (just multiple parts).
	Comp	Part Link	NA	NA	NA	If the Comp has one Prop assigned (1:1 enforcement), then this Prop is added as a Prop Link.	NA
	Comp Group	Part Link	Part Group Link	NA	NA	NA	If the Comp(s) all have one Prop assigned (1:1 enforcement), then these Props are added as a Prop Group Link.
	Prop	If Prop is assinged to Comp (1:1 indirectly only), then the Comp's Part is added as a Part Link. SPECIAL CASE: If the Prop is not assigned to a Comp, then the Module Part, which hosts the Prop, is added as a Part Link.	NA	If Prop is assinged to Comp (1:1 indirectly only), then this Comp is added as a Comp Link.	NA	NA	NA
	Prop Group	If Props are assigned to Comps (1:1 indirectly only), and there is exclusively one Module Part, which hosts all these Comps, then this Module Part is added as a Part Link.	If Props are assigned to Comps (1:1 indirectly only), and there are several Module Parts, which host all of these Comps, then these Module Parts are added as a Part Group Link.	NA	If Props are assigned to Comps (1:1 indirectly only), then these Comps are added as a Comp Group Link.	NA	NA

Figure 289:

Find connectors from components can be accessed from the Link Entity Browser's right-click context menu.

Find Connectors From Supported Entities/Parts

1. Ensure that the **Linked Connector** view option toggle ( ) is active.

   If the realized welds of the connectors also need to be displayed in graphics then activate the button.
2. Either:
   - Ensure that the **Show/Hide** button ( ) is active in Show mode, and then click the desired link entity, or
   - Use the selector to pick the desired link entities, and then left-click the Show/Hide button.

Find Connectors Between Two or More Supported Entities/Parts

1. Select two or more supported entities from the link entity table to find their connecting connectors.

   The entities can be easily located in the browser table list by using the Selector button in the action mode tools. Click the button and select the entities in the graphics area to highlight it in the table.

   If the realized welds of the connectors also need to be displayed in graphics then activate the button.
2. Use the Link Entity Browser advanced action buttons to show or isolate the shared connectors.

Find Connectors From Realizations

1. Select the desired link entities from the Link Entity Browser list or the graphics area.
2. Click the button in the utility tool set (at the bottom of the Connector Browser) to open an elements selection panel.
3. Select the desired elements.
4. Click **proceed** in the panel.

The connectors that contain the selected elements as their realized welds will be highlighted in the browser.

Find Links From Connectors

The find parts feature in the Connector Browser context menu finds the entities/part(s) connected by specific connector(s), and isolates them in the graphics area.

The entities found are also highlighted in the Link Entity Browser making it easy to perform further operations.

The find parts feature will also isolate the realized welds of a connector if the realization toggle button is activated in the browser.

1. Select one or more connectors in the Connector Entity Browser.

   The connectors can also be selected from the graphics area by using the selector button in the Connector Entity Browser action modes tool set. The connectors selected in the graphics area will be highlighted in the browser tree.
2. If the realized welds of the connectors also need to be isolated, activate the **realization** button.
3. Right-click in the browser tree to open the context menu, then click **find parts**.

> **Note:** The selected connectors, the entities that are connected by those connectors, and the realized welds of the selected connectors (if toggle is activated) will be isolated in the graphics area and the found entities are also highlighted in the Link Entity Browser.

Contact Browser

Use the Contact Browser to create, review and modify contact interfaces and surfaces in a model.

Contact interfaces and surfaces can be imported, created manually, or generated automatically using the AutoContact option. Upon importing a solver deck or HyperMesh model, existing contacts are created and populated in the Contact Browser. Contact interfaces and surfaces are organized into their respective folders in the browser.

To open the Contact Browser, click **View** > **Browser** > **HyperMesh** > **Contact** from the menu bar.

Contact Browser Interface

The Contact Browser consists of three panes; the first pane displays the entities needed for contact definition, the second pane displays the contact information, and the third pane displays the Entity Editor.

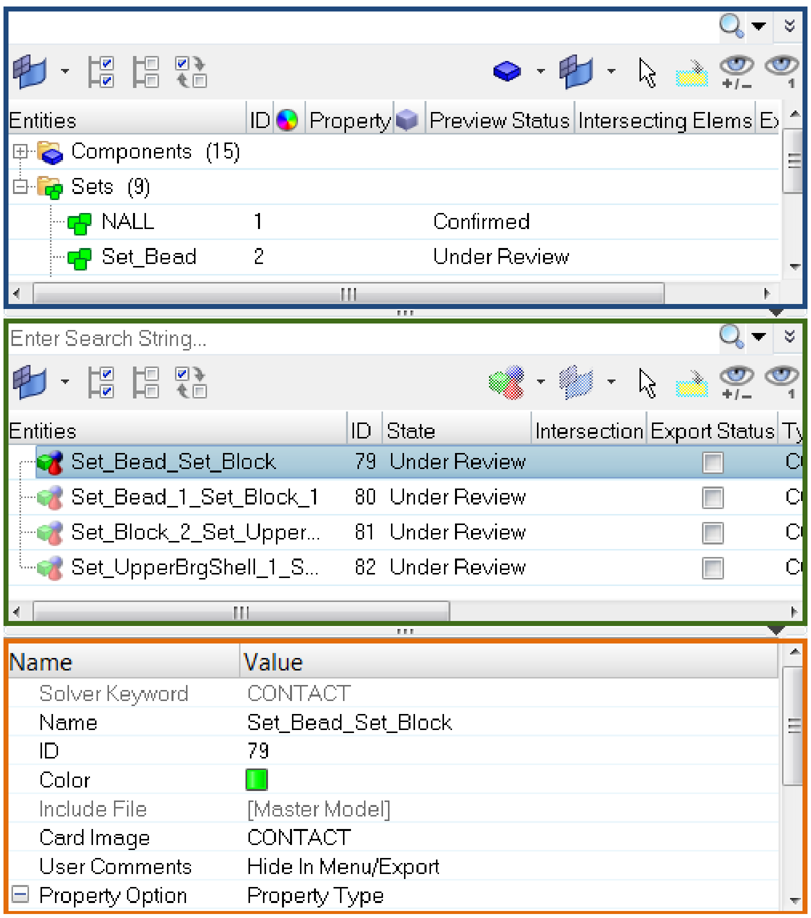


Figure 290:

Create Contacts

Create Contacts Manually

In the Contact Browser you can manually create contacts.

1. In the first pane of the Contact Browser, right-click and select **Create** from the context menu to create all of the entities needed for contact definition, such as contact surfaces.
2. In the second pane of the Contact Browser, right-click and select **Create** from the context menu to create contacts.
   The Entity Editor opens and displays the new contact.

   > **Note:** The entities available in the context menu will vary depending on the user profile loaded.

3. Assign Master and Slave entities when creating contact pairs and contact ties.
4. Define additional entity parameters and properties accordingly.

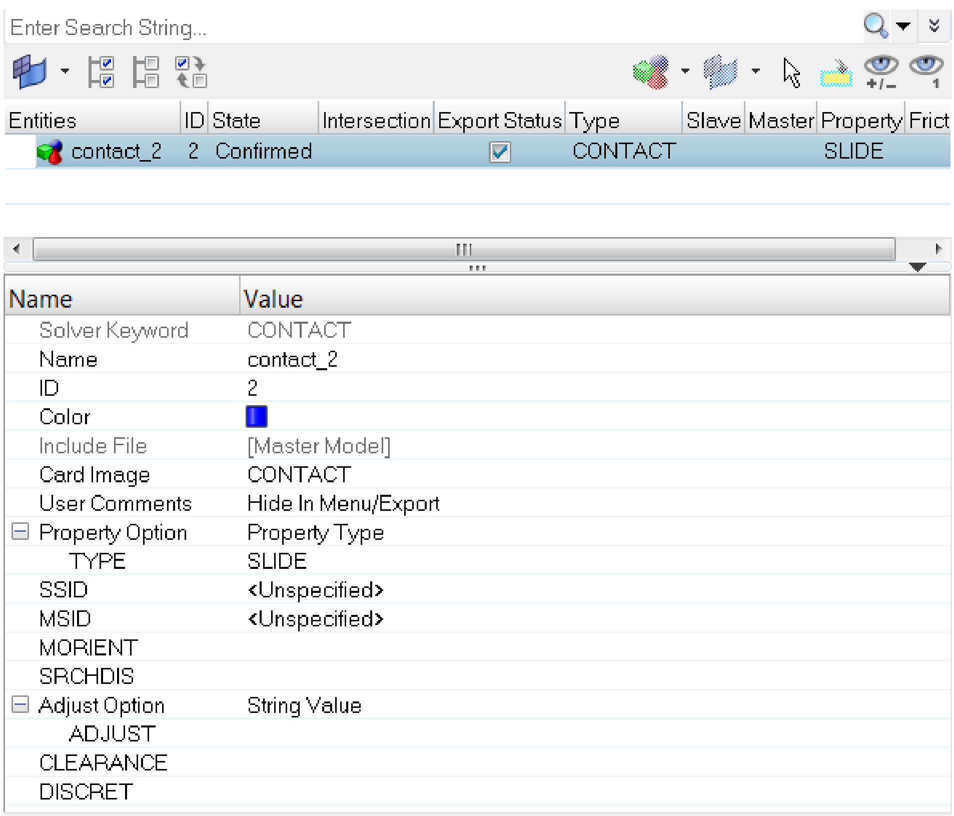


Figure 291:

Contact Browser and Entity Editor in the OptiStruct user profile.

Generate Contacts Automatically

Auto contact generation works independent of whether the model is FE or geometric. When you elect to generate contacts automatically, contact bodies are detected and contact pairs and contact sets are created.

1. In the Contact Browser, right-click and select **AutoContact** from the context menu.
   The **Create AutoContact** dialog opens.

   > **Tip:** To select all of the components in the model, right-click on the **Components** folder.

2. In the Pick application region field, select components to create contact interfaces and contact surfaces between.

   > **Note:** You must select a minimum of two components.

3. Define contact parameters and properties accordingly.
   a) Select a Contact tolerance type.
      If Vicinity tolerance is selected, you must enter a vicinity tolerance value.
   b) Enter a Reverse angle limit.
      This determines the maximum curve angle to identify contacts.
   c) Enable **Consolidate contact pair** to group multiple isolated surfaces into one surface.
   d) Select a Contact type to specify the type of contact to be generated. Select **Touch** to generate “Contacts”, and **Gap/Tie** to generate “Tie”.
   e) Create and assign, or pick an existing property to attach to the contact pair.

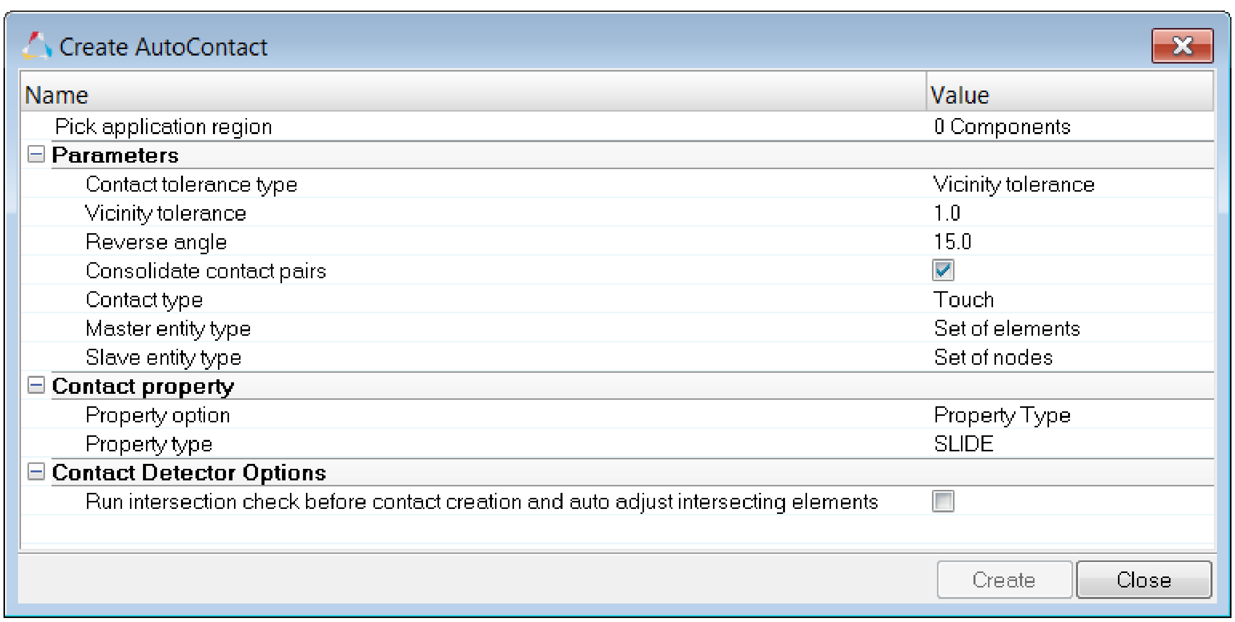


Figure 292:

4. Click **Create**.
   Based on a vicinity tolerance, the AutoContact tool searches throughout all of the selected components and automatically creates new Contact interfaces and contact surfaces between them. All the objects created are populated in the Contact Browser into their respective folders.

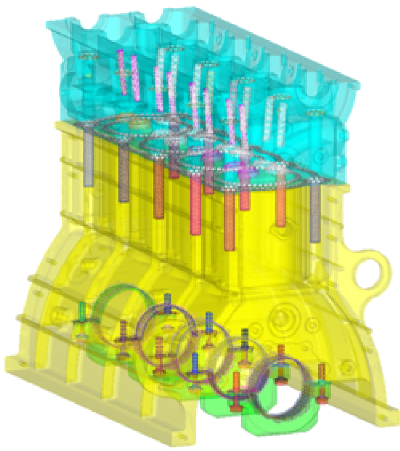


Figure 293:

Modify Contact Interfaces and Surfaces

The Contact Browser lists every contact interface and surface within a HyperMesh session and organizes them into their respective folders. You can modify these contact interfaces and surfaces using the Entity Editor.

1. In the Contact Browser, click a contact interface or surface.
   The Entity Editor opens and displays the entities corresponding data.

   > **Note:** If you select multiple entities in the browser, the Entity Editor displays the selected entities common corresponding data. The rows that contain ### indicate that these fields do not contain common data but can still be modified. When you modify the data, both common and ambiguous, HyperMesh applies the changes to all of the selected entities. To select multiple entities, left-click entities while pressing Control.

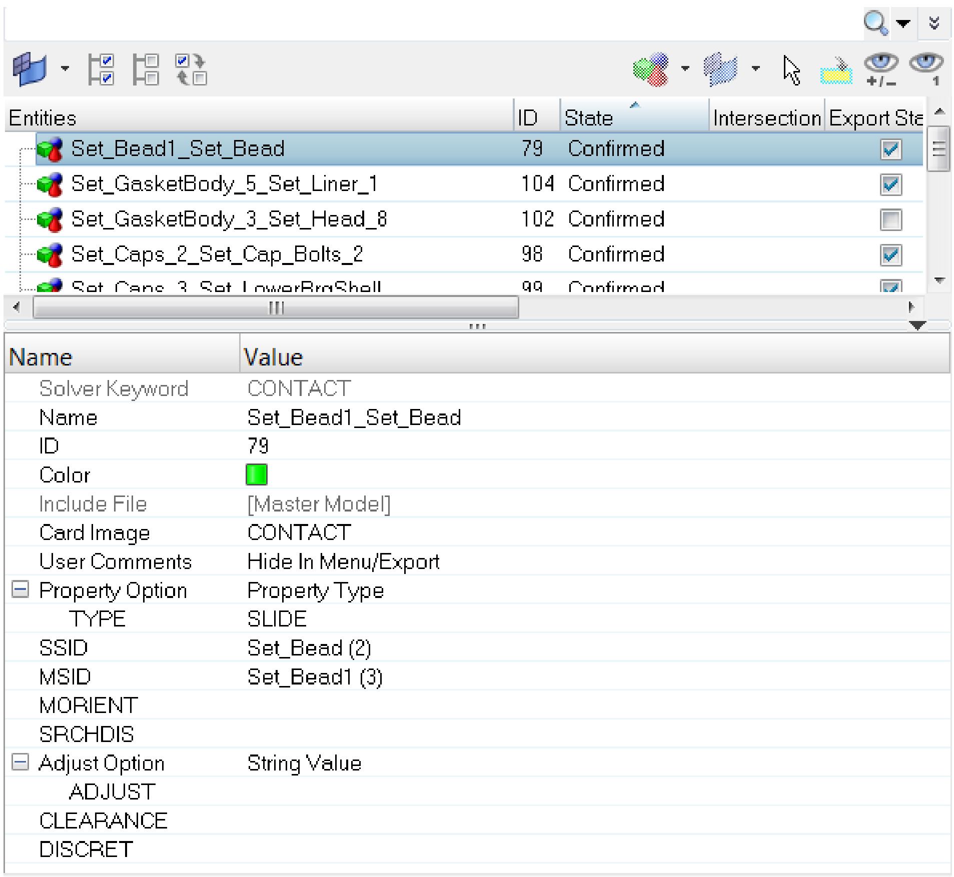


Figure 294:

2. In the Entity Editor, modify the contact parameters and properties accordingly.
   Modifications are automatically applied throughout the model.

Solver Specific Details

Abaqus

- The Contact Browser supports the following contact and constraint definitions for Standard and Explicit profiles of Abaqus:
    - AutoContact
    - Contact pair
    - General contact
    - Contact Tie
    - Shell to Solid coupling
- The AutoContact option creates contact pairs and tie constraints depending on whether you set Contact Type to touch or gap/tie.
- While using AutoContact, you can specify to have the slave surface be created as a node-based surface.
- All contact pair definitions in Abaqus will require a surface interaction definition.
    - AutoContact creates a surface interaction property with default solver behavior.
    - You can choose an existing surface interaction to be used for contact pairs recognized by AutoContact.
- AutoContact can create contact pair/tie with review status before confirming.
- In the contact information pane (middle pane in the browser), you can interactively add friction, adjust, and specify other optional parameters.
- Use Advanced delete to delete contact pair/tie. The corresponding master and slave surfaces and surface property are used for the contact pair unless the surface property is otherwise specified to be used in other contact pair definitions.

ANSYS

- Creating and managing contact surfaces and pairs is handled by the Contact Browser.
- Both manual contacts and AutoContact options are available. The AutoContact option can be used to create Surface to Surface contacts.
- Before creating contact pairs in HyperMesh, it is recommend that you know which ANSYS contacts are supported in HyperMesh.
- ANSYS contact regions are modeled like ordinary FE elements over the regions of FE models that represent the geometry. In ANSYS terms, both CONTACT and TARGET surfaces are sets of elements at contact regions. Prior to HyperMesh 14.0.130, the ANSYS interface supported the same principle as ANSYS, therefore both CONTACT and TARGET regions were elements over model geometry in HyperMesh.

- Starting in HyperMesh 14.0.130, both contact and target regions are identified as CONTACT SURFACES. A pair of contact and target surfaces which interact with each other in analysis are GROUPS. You need to create contact surfaces for both CONTACT and TARGET regions. They will then be paired together under groups to successfully define the contacts. ANSYS solver files (`*.cdb` files) with contacts are imported into HyperMesh, and identify the contact regions from the contact elements to show the regions as contact surfaces. Contact pairs can be seen in groups.
- During export, HyperMesh creates contact elements from the contact regions and exports them as elements. In ANSYS, they are explained above elements with contact element type.
- The Contact Browser currently supports the creation of edge to edge, surface to surface, and point o surface contacts with pilot node and symmetry contact options.

Figure 295: HyperMesh 14.0.120 and Earlier

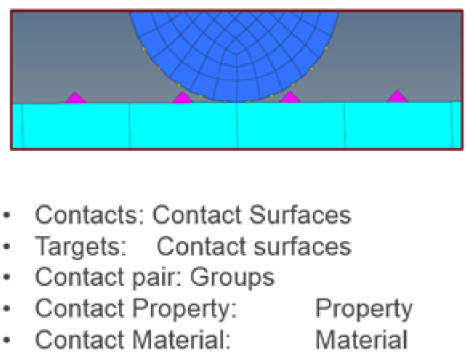


Figure 296: HyperMesh 14.0.130

Nastran

- The AutoContact option will create new contact cards using their default values.
- When reading legacy models (`.bdf`, `.dat` and `.hm`), old contacts will be mapped to new contact cards.
- Contact parameters in the BCTABLE card are mapped to equivalent contact properties (BCONPRP and BCONPRG) in new contact cards.
- Properties associated with BCPROP should be assigned to either elements or respective components in the contact card.
- You can export old contact cards by enabling the **Export old Contact Card** checkbox, which can be accessed in the in the Export-Solver Deck Browser when you click **Select Options**.
- The Contact group entity is mapped to the BCTABL1 solver card.
- The BCTABL1 solver card contains a list of contact pair (BCONECT) cards.
- The AutoContact option will create a BCONECT card for each pair, and group all pairs in the BCTABL1 card.

- BSURF and BCBODY1 solver cards are remapped from group entities to set entities.
- BSURF and BCBODY1 are combined into one entity. Input for BCBODY1 can be defined in the BSURF card with the Entity Editor.

OptiStruct

- The Contact type Touch creates a contact pair, and the Contact type Gap/Tie creates a Tie contact.
- Master ID and Slave ID types will be as per solver default Set of elements for master ID and Set of nodes for slave ID.
- Auto contact will honor the entity type in Master and Slave entity type.
- Master entity types have the following mutually exclusive options: Set of elements or SURF. Similarly Slave entity type will have Set of nodes, Set of elements, and SURF. You can only select one of the entities.
- The Contact property option is only available when the Contact type is Touch.
- You can assign Touch contacts a Property type (SLIDE, STICK, FREEZE), a Static Friction Coefficient, or an existing property (`PCONT`).

Samcef

- Automatically detect the contacts between selected parts.
- Create different type of contacts (.MCT/.STI).
- Modify the contacts already created: add/remove elements/nodes into the master/slave groups, change the type of the contacts (MCT to STI or STI to MCT).
- Review the contacts in the model.

Context Menu

The following options are available in the Contact Browser context menu:

Review

Highlights the slave and master surfaces in two different colors, while the rest of the model is transparent and gray.

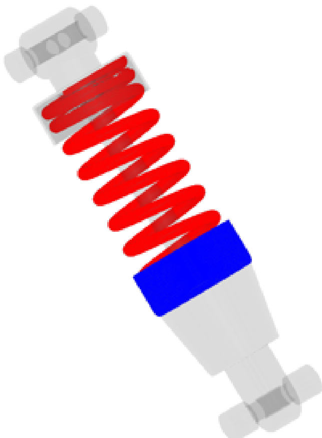


Figure 297:

Show

Displays the contact's master and slave surfaces in their assigned colors. The rest of the model display is left unchanged.

Hide

Turns off the display the contact's surfaces. The rest of the model display is left unchanged.

Isolate Only

Fits the selected contact to the graphics area, and displays the slave and master bodies in two different colors. The rest of the model is turned off.

Swap Master-Slave

Switches the contact surfaces identified as master and slave. Upon selecting Swap Master-Slave, the surfaces switch from places in the master to slave positions in the browser.
To select multiple entities, left-click entities while pressing Control.

Swap Cp-Tie

Changes the type of interface created. A surface interaction is required for Contact Pairs but not for a Tie, therefore any surface interaction identified earlier will be lost upon swapping from CP to TIE. If you switch back from TIE to CP, the surface interaction will not be retained.

Reverse Normals

Flips all the normals of selected elements or surfaces.

Display Normals

Displays the normals of selected elements or surfaces as vectors or in color mode.

Display Base Element Normals

Displays the normals for parent elements of the surface.

Normals Off

Turns off normals if they are displayed.

Entity Editor

The Entity Editor enables you to quickly view and edit entities in a model and correctly setup solver information.

The Entity Editor can be accessed from the Model, Reference and Solver browsers. In the Model Browser, the following view modes are supported: Model, Include, Component, Property and Material. When accessing the Entity Editor from the Reference Browser, you will only be able to view your entity selection's corresponding data in a non-editable form.

The Entity Editor opens when you create new entities, as well as duplicate, edit, or select single or multiple entities in the browser. To expand and collapse the Entity Editor, click in the top, right-hand corner. To adjust the height of the Entity Editor, drag up and down.

Entity parameters are displayed in the Name column, and the values associated with each parameter are displayed in the Value column. The parameters that display in gray text cannot be edited. Once you edit an entity in the Entity Editor, the changes will be automatically applied throughout your model.

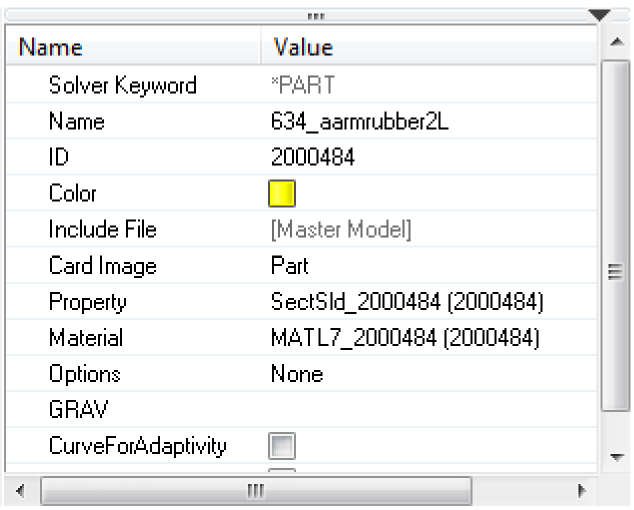


Figure 298:

Edit Multiple Entities

If you select multiple entities in the browser, the Entity Editor opens and displays the selected entities common corresponding data that can be modified simultaneously.

The rows that contain ### indicate that these fields do not contain common data but can still be modified. When you modify the data, both common and ambiguous, HyperMesh applies the changes to all of the selected entities.

To select multiple entities in the browser, press Shift and begin selecting entities.

In Figure 299, seven components are selected, and the Entity Editor displays their common corresponding data. The Color, Card Image, and Prop_Id are different, therefore you see ###. The Mat_Id is the same for all of the selected components, therefore you see the material name and its ID.

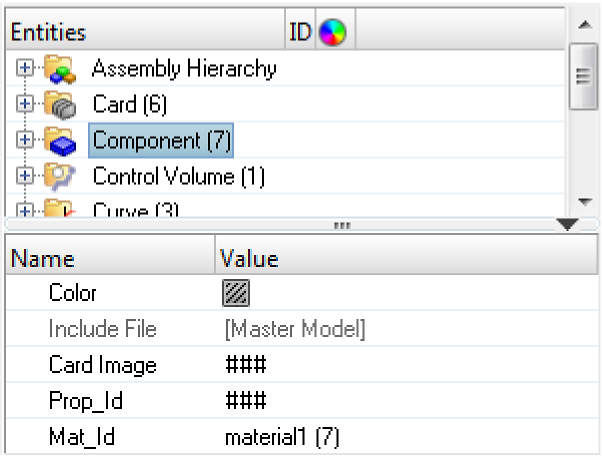


Figure 299:

Edit HyperMesh Specific Data

When you create an entity, it is assigned an unique ID, name and color. Use the Entity Editor to edit this data.

Open the Entity Editor for the entity to edit.

Option	Description
Edit ID and name	**1.** Click **Name** or **ID** and enter a new value for the entity in the editable field. **2.** Press Enter. HyperMesh updates the changes in the Entity Editor and browser automatically.

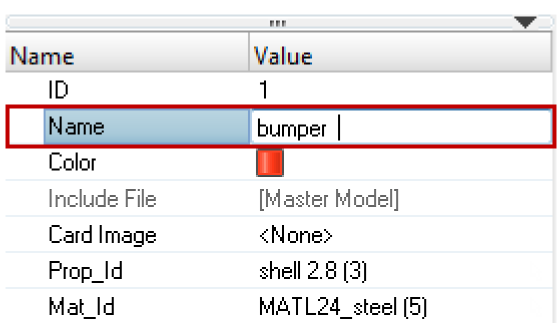


Figure 300:

Edit color

1. For Color, click the color box.
2. From the color pallet, select a new color to represent the entity.

The entities color changes in the Entity Editor, browser and graphics area.

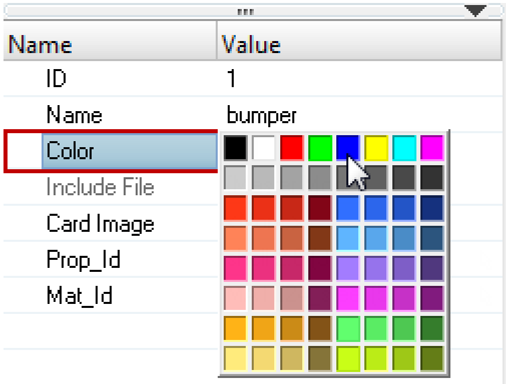


Figure 301:

Edit Solver Specific Data (Template Data)

Every entity is assigned solver specific data, which you can modify using the Entity Editor.

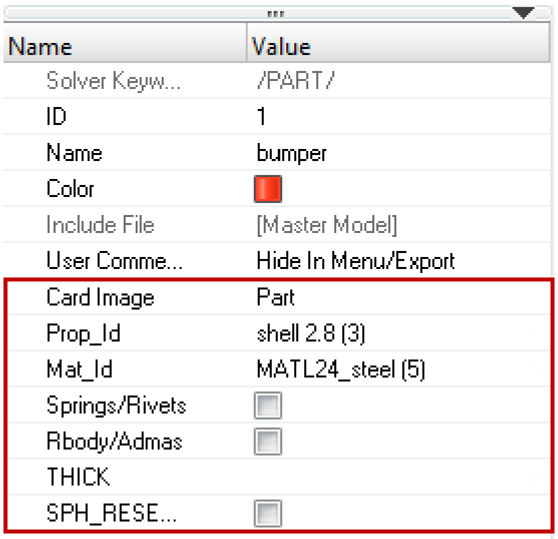


Figure 302:

Assign a Card Image

In the Entity Editor, you can assign a card image to an entity. The card images you are able to assign to an entity will depend on the user profile you have loaded and the entity type you have selected in the browser.

1. Click **Card Image**.
2. Select a new card image from the drop-down list.
   The entity's card image changes, and the Entity Editor displays additional solver related fields.

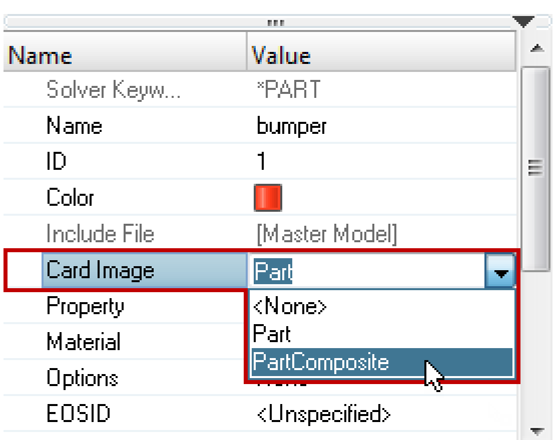


Figure 303:

Turn Fields ON or OFF

- Turn fields on and off in the Entity Editor.

Option	Description
Turn field off	Remove its corresponding value or select **OFF** from the drop-down list. If you hover over a field that is turned OFF, its default value will appear in the Value column.
Turn field on	Enter a new value or select a value from the drop-down list. When a field is turned ON, its corresponding value will always be displayed in the Value column.

In the example below, assume that initially the field ISOP is turned OFF and has a default value of FULL. To turn the field ON, click its corresponding **Value** field and select a value from the drop-down list. If you want to turn the field OFF, select **<OFF>** from the drop-down list.

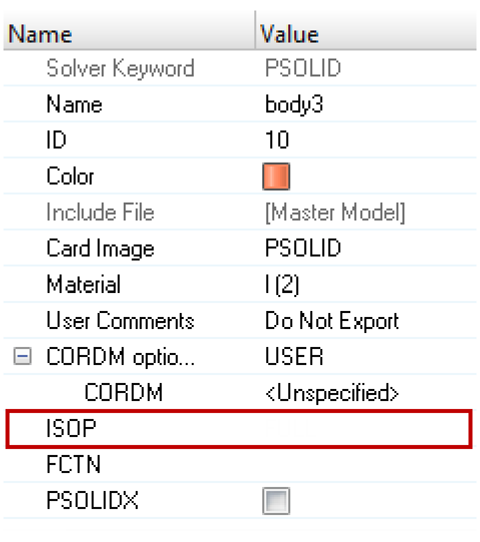


Figure 304: ISOP Field Off

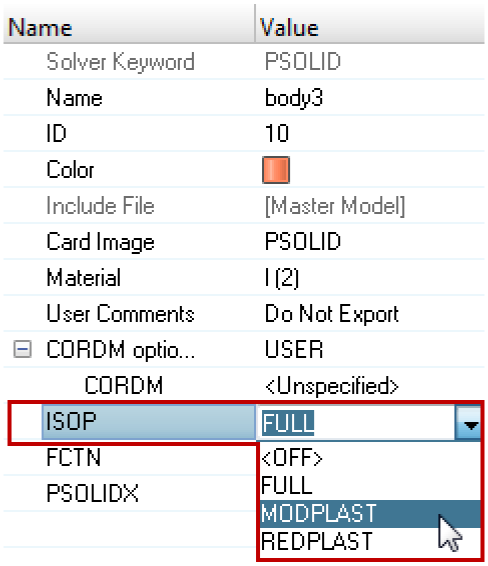


Figure 305: ISOP Field On

In the example below, assume that initially the field G is turned OFF and has a default value of 80769.2. To turn the field ON, click the **Value** field and press Enter. To turn the field OFF, delete the value and press Enter. If you turn the field OFF, the Entity Editor will retain the previous value you specified.

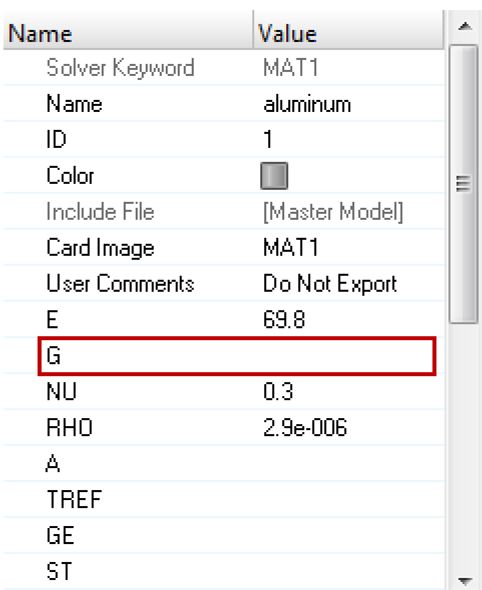


Figure 306: NU Field Off

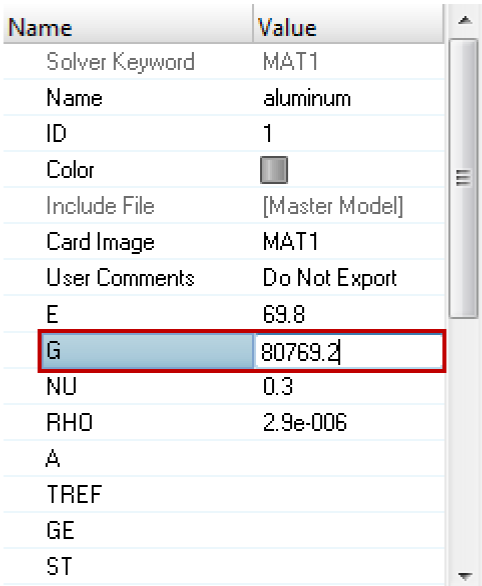


Figure 307: NU Field On

In the example below, assume that initially the field MID4 is turned OFF. You can turn the field on and then assign it a material. To turn the field ON, click its corresponding **Value** field and select **materials** from the drop-down list. You can then click the yellow selector button and select a material to assign to the field. If you want to turn the field OFF, select **<OFF>** from the drop-down list.

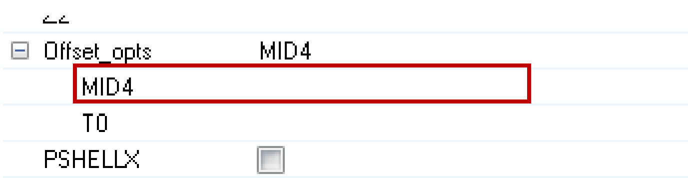


Figure 308: MID4 Field Off

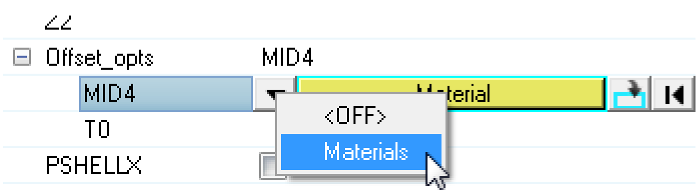


Figure 309: MID4 Field On

- Turn card image fields on and off in the Card Editor by clicking the field heading.

  When a field is turned ON, an additional field appears under the field heading, from which you can enter or select a value.

  In example below, the card image MAT1 is displayed for a material in OptiStruct. By default, the field G is turned OFF. To turn the field ON, click its field header. The default value is displayed under the field heading.

Figure 310: MAT1 Card Image with the Field G Off

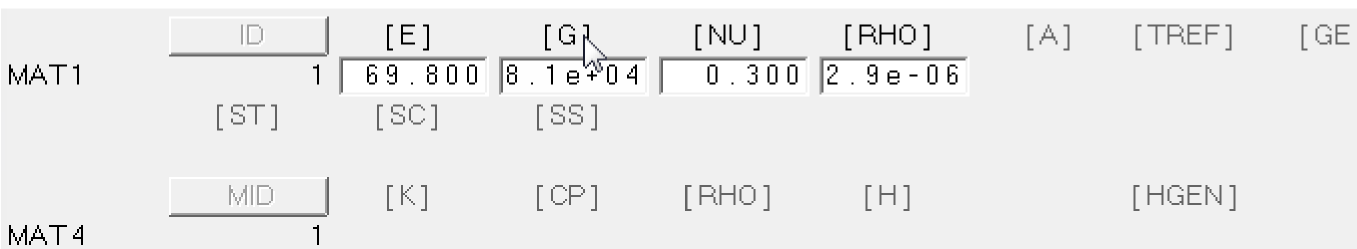


Figure 311: MAT1 Card Image with the Field G On

When defining data in tables, certain fields can be turned ON and OFF. To turn a field ON, right-click on the heading of the disabled field and select **Status**. In the image below, the NO field is being turned ON for the load collector TSTEP.

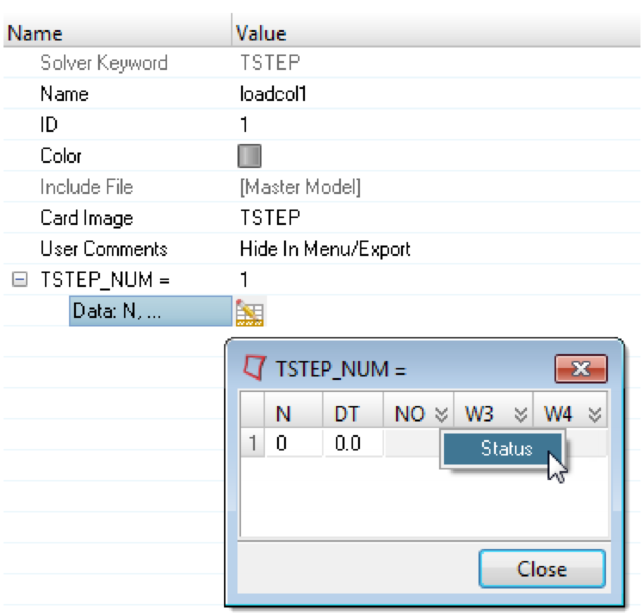


Figure 312:

Assign Entities

In the Entity Editor you can assign certain entities to another entity or a group of entities.

For example, you can assign a property to a component. When an entity has an entity assigned to it, the name and ID of the assigned entity will be displayed in the Value field. If an entity does not have an entity assigned to it, the Value field will display <Unspecified>.

In the image below, the Entity Editor is open for the component bottom_bracket. The material aluminum is assigned to the component, and a property has not yet been assigned.

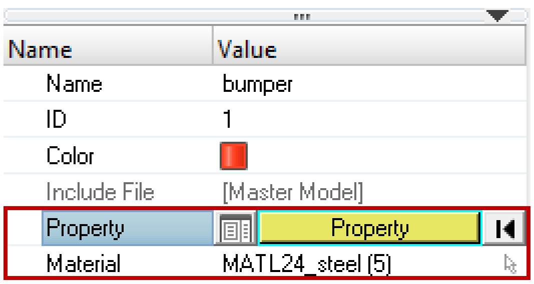


Figure 313:

In the Entity Editor, you can assign an entity using the **Select** dialog or the Entity Selector.

- Method 1: Assign Entities Using the **Select** Dialog
  a) Click the entity's corresponding **Value** field.
  b) Click the yellow selector.

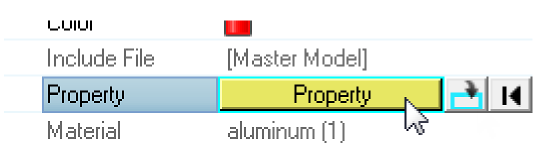


Figure 314:

  c) Select an entity in the following ways:
    1. In the **Select** dialog, select an entity.

> **Tip:** You can search for entities in the **Select** dialog by entering a name, ID, or card image in the search field. When you click or press Enter, the dialog only displays the entities that match your search string.

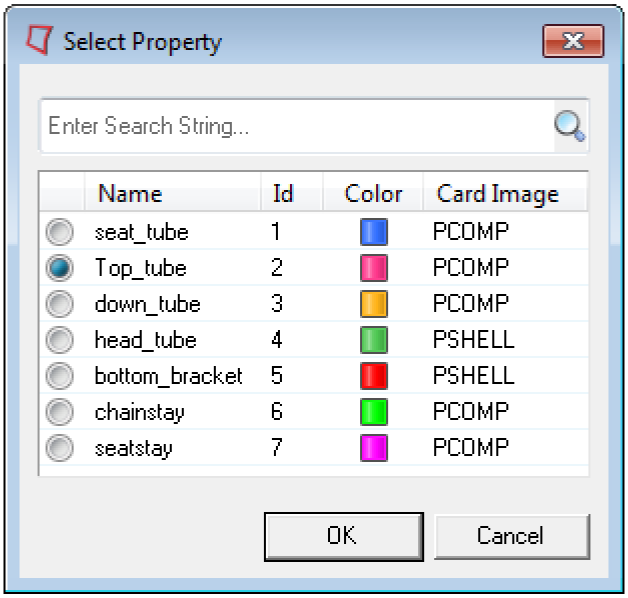


Figure 315:

2. In the graphics area, select an entity. HyperMesh automatically selects the entity in the **Select** dialog.

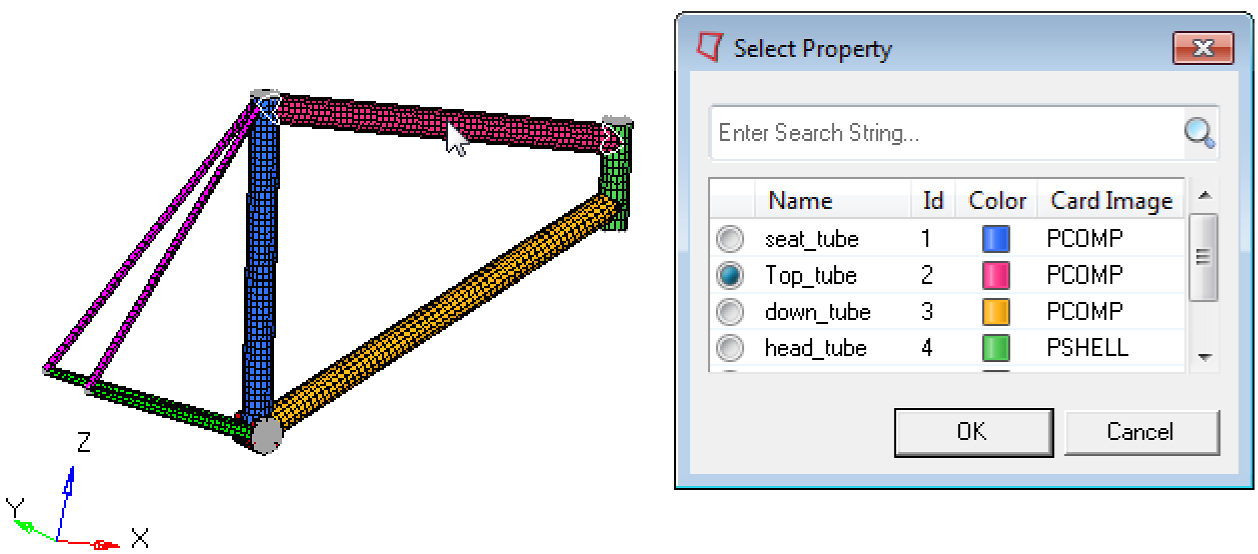
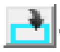


Figure 316:

d) Click **OK**.

- Method 2: Assign Entities Using the Entity Selector
  a) Click the entity's corresponding **Value** field.
  b) Click .
  c) In the panel area, click the yellow selector.

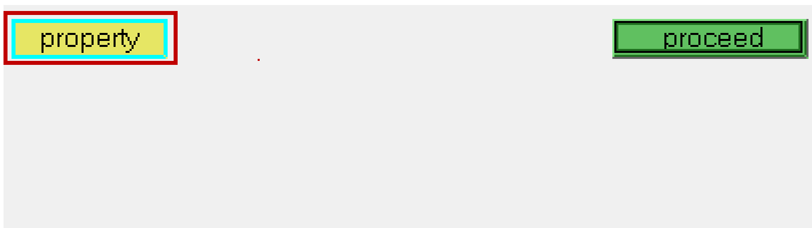


Figure 317:

  d) Select an entity in the following ways:
    1. In the panel area, select an entity and click **return**.

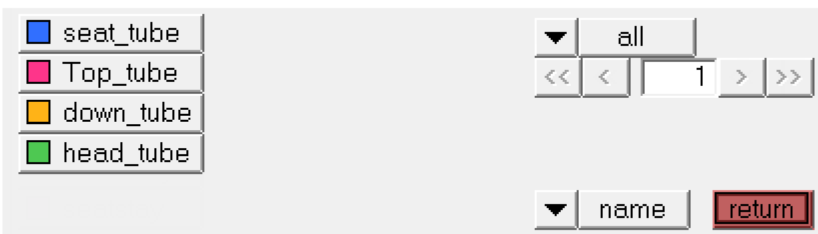


Figure 318:

    2. In the graphics area, select a entity. HyperMesh outlines your selection in white.

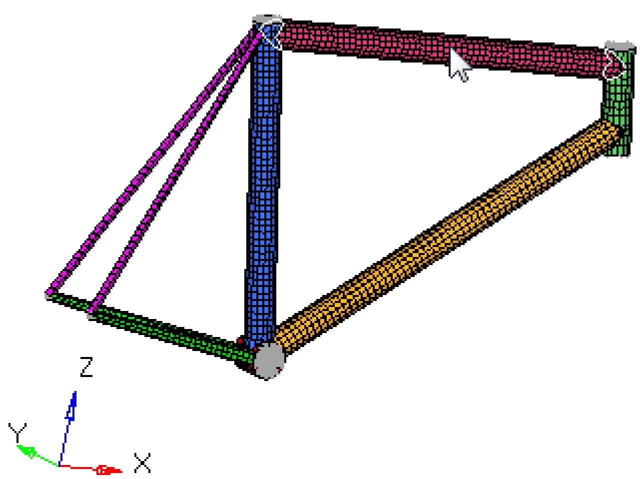


Figure 319:

  e) Click **proceed**.

Resolve Unresolved Entities

The import and export of unresolved entity IDs is supported, in a limited way. If you have an unresolved entity ID which is referenced by other entities, the Entity Editor displays the phrase "Unresolved Entity" in the Value field of the unresolved entity.

In the image below, the load collector Fatigue refers to a load collector with id=9, which is unresolved.

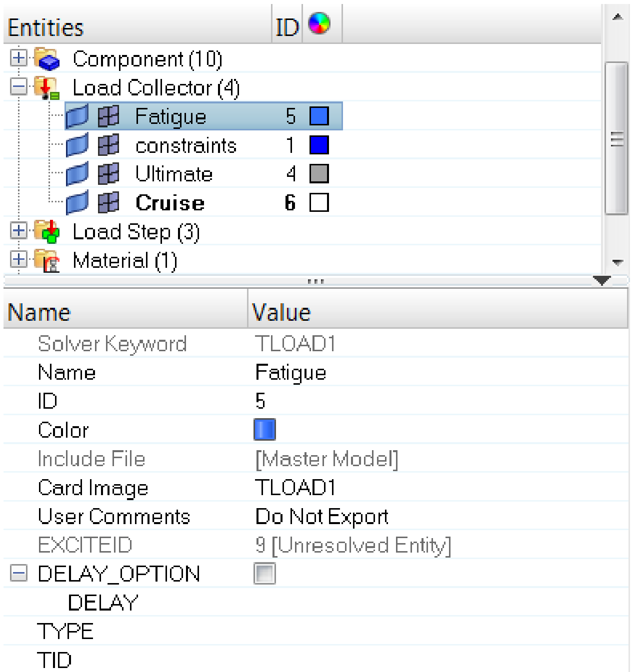


Figure 320:

1. Right-click on the **Name** field of the unresolved entity and select **Clear** unresolved entity from the context menu.
2. Click the entity's corresponding **Value** field.
3. Click the yellow selector.
4. In the **Select** dialog, select a new entity.
5. Click **OK**.

Keyboard Shortcuts

Function Key	Description
Tab	Moves from one row to the next in the Entity Editor.
Shift + Tab	Moves to the previous row in the Entity Editor.
Spacebar	Activates the Value field of the selected row.
↑ ↓	Move up and down in the Entity Editor, a list of options in drop-down menus, or in a right-click context menu.
Enter	Accepts the changes made to a parameter in the Value field, or selects an option in a right-click context menu.
Esc	Dismisses the changes made to a parameter in the Value field, or closes a right-click context menu.

Create and Edit Assigned Entities

Create and Assign a New Entity

To create and assign a new entity to an entity or a group of entities:

1. Right-click on the entity assignment field and select **Create** from the context menu.
2. In the **Create** dialog, define the new entity.
3. Click **Close**.
   The Entity Editor creates and assigns the new entity.

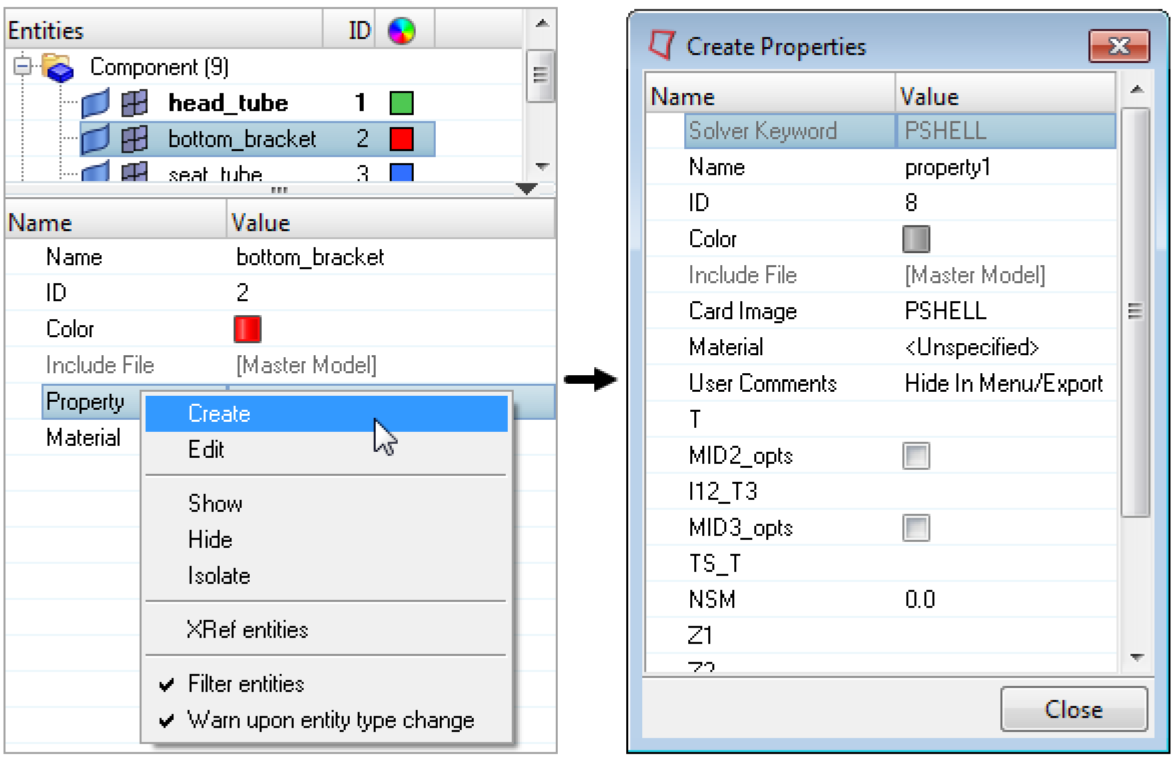


Figure 321:

Edit an Assigned Entity

To edit an entity that is assigned to an entity or a group of entities in the Entity Editor:

1. Right-click on the entity assignment field and select **Edit** from the context menu.
2. In the **Edit** dialog, modify the entity data.
3. When you are finished making changes, click **Close**.

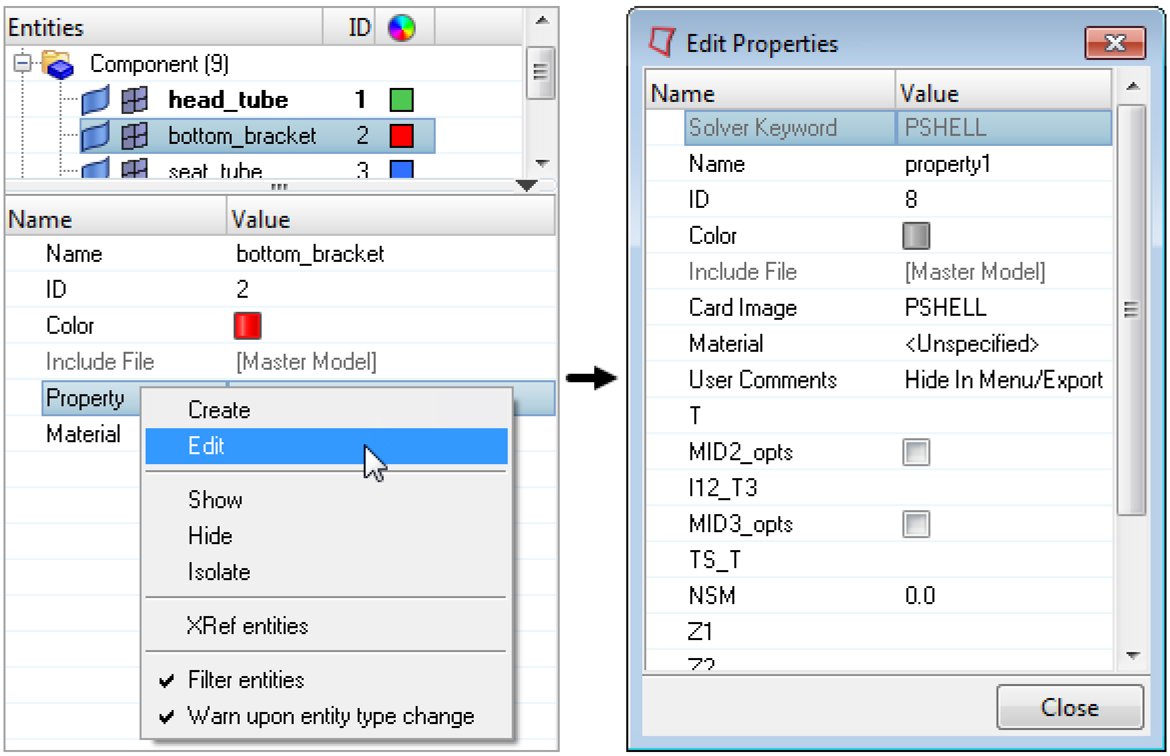


Figure 322:

Create and Assign Contact Surfaces Using Elements or Nodes

When defining contact entities in the Entity Editor, you can create and assign contact surfaces using elements or nodes to master and slave entities.

Only available in the Abaqus, ANSYS, and OptiStruct user profile.

1. In the Entity Editor, right-click on the Slave entity or Master entity field and select **Create Contact Surface using Elements** or **Create Contact Surface using Nodes** from the context menu.

    > **Note:** The options available depend on the solver keyword selected.

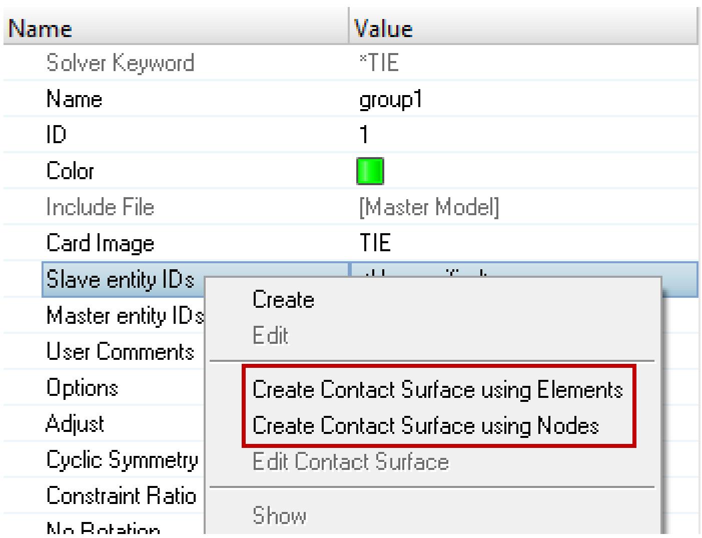
    

    Figure 323:

2. In the corresponding panel, select elements or nodes and click **add**.
3. Click **return** to exit the panel.

Create and Edit Parameters

Create and Assign a New Parameter

In the LS-DYNA and Radioss user profiles, certain entities can be defined with parameters.

1. Right-click on the parameter assignment field and select **Create and Assign Parameter** from the context menu.
2. In the **Create Parameter** dialog, define the new parameter by changing the Parameter type and Expression value.

   When the Card Image and Parameter type are set to expression, the Expression value field becomes active.

   > **Note:** The Entity Editor currently supports the following types of parameters: Double, Integer, Double Expression, and Integer Expression.

3. In the Expression value field, enter a valid expression value.
4. When you are finished defining the parameter, click **Close**.
   The Entity Editor creates and assigns the parameter to the selected entity.

   > **Note:** These parameters can be used in HyperStudy for design exploration and optimization.

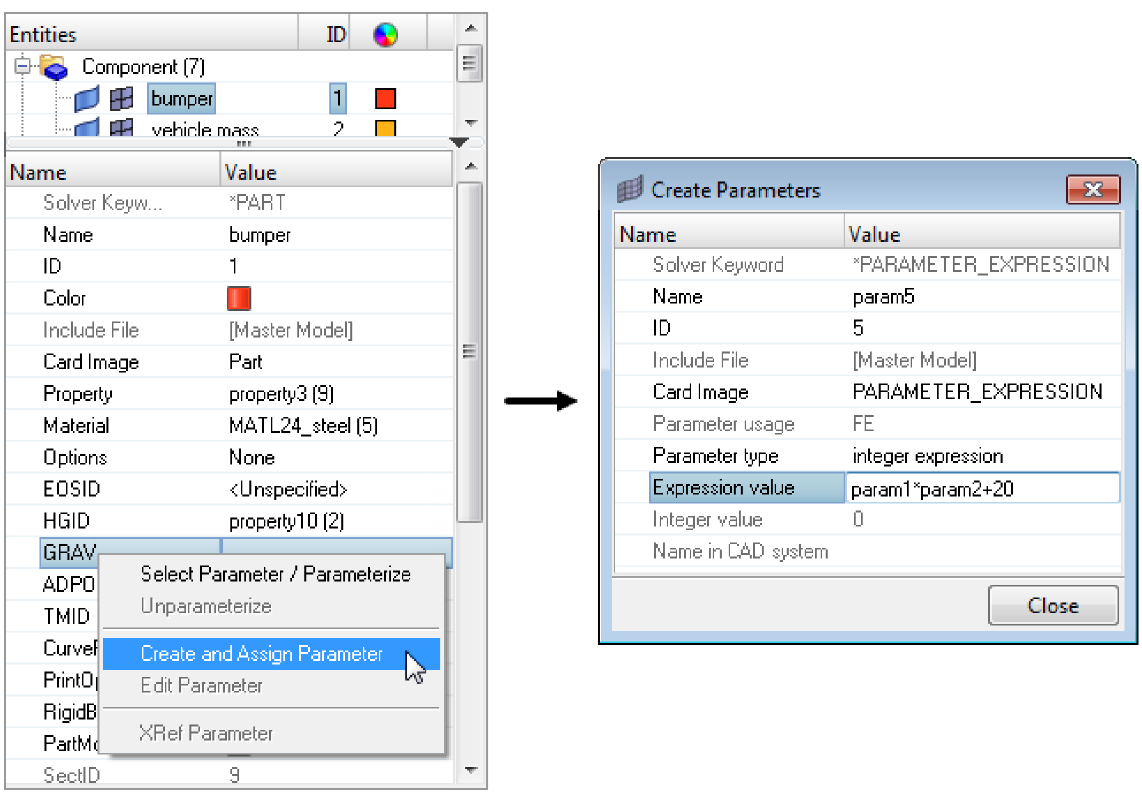


Figure 324:

Edit a Parameter

1. Right-click on the parameter assignment field and select **Edit Parameter** from the context menu.
2. In the **Edit Parameter** dialog, modify the parameter's data.
3. When you are finished making changes, click **Close**.

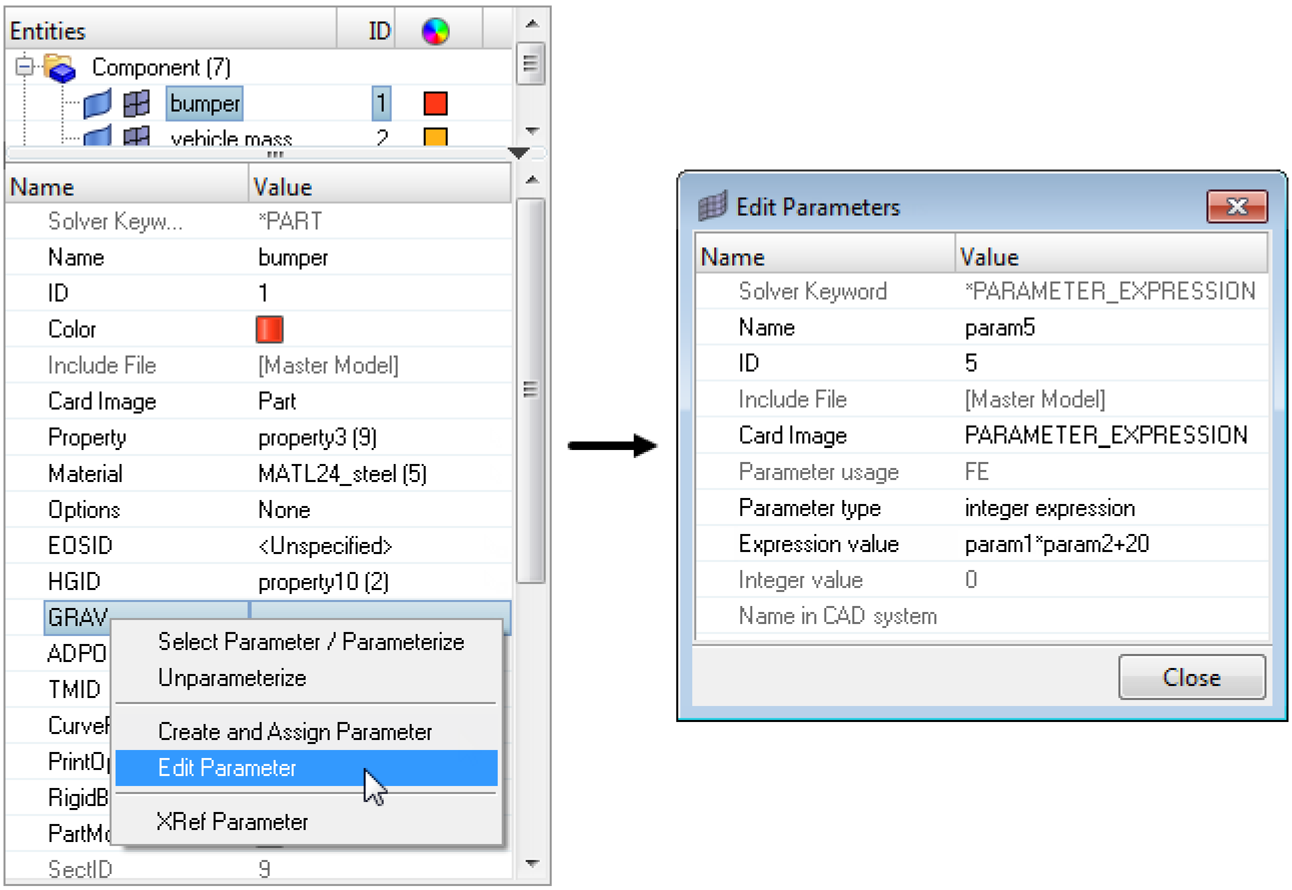


Figure 325:

Parameterize and Unparameterize Entities

Parameterize an Entity

For any solver entity, you can parameterize most of the solver template data of a numeric value.

1. Right-click on the solver data field and select **Select Parameter/Parameterize** from the context menu.
2. In the **Select Parameter** dialog, select a parameter from the list of valid types.
3. Click **OK**.

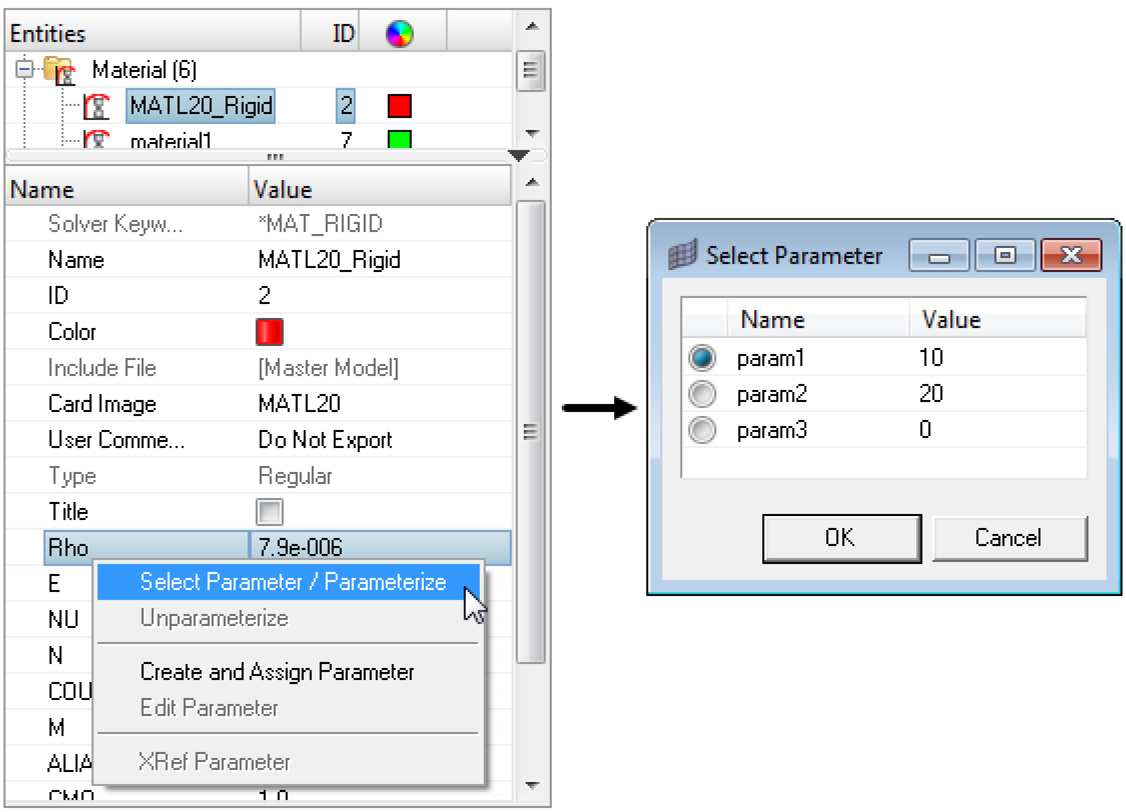


Figure 326: Parameter Rho is being Parameterized with the Parameter param1.

Unparameterize an Entity

When a solver data field is parameterized, you can unparameterize it or assign it a different parameter. Right-click on the field and select **Unparameterize** from the right-click context menu.

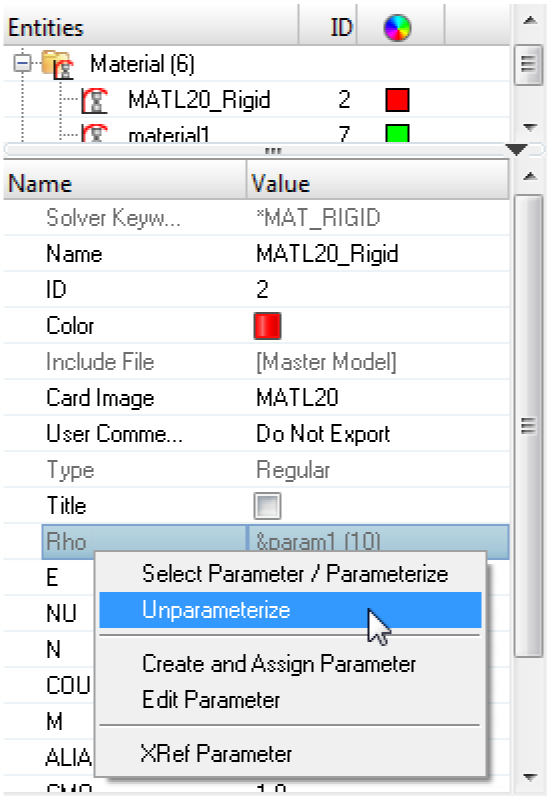


Figure 327: Parameter Rho is being Unparameterized with the Parameter param1.

Filter Entities

To filter the entities that are not applicable to the entity displayed in the Entity Editor, activate the entity filter in the right-click context menu.

When off, all of the entities available in the model will display in the **Select** dialog or Entity Selection panel.

In the image below, the Entity Editor is activated for the component bumper, which is of type *PART. For this example, assume that the Filter is activated. If you were to assign a property to this component using the **Select** dialog, seven properties would be available, even though the model contains a total of nine. The two properties that are filtered out cannot be assigned to a *PART of LS-DYNA. The status bar also displays a message that says seven of nine properties are being filtered using a filter criteria, and in this case it is an IdPool filter.

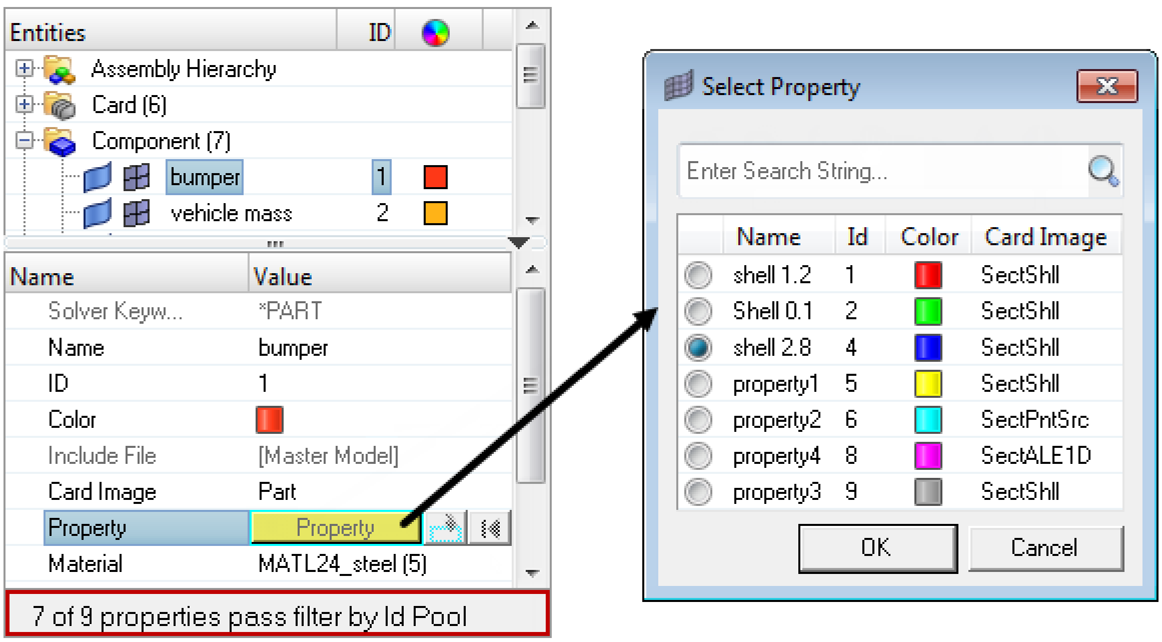


Figure 328:

If you deactivate the filter, the **Select** dialog will display all of the properties in the model.

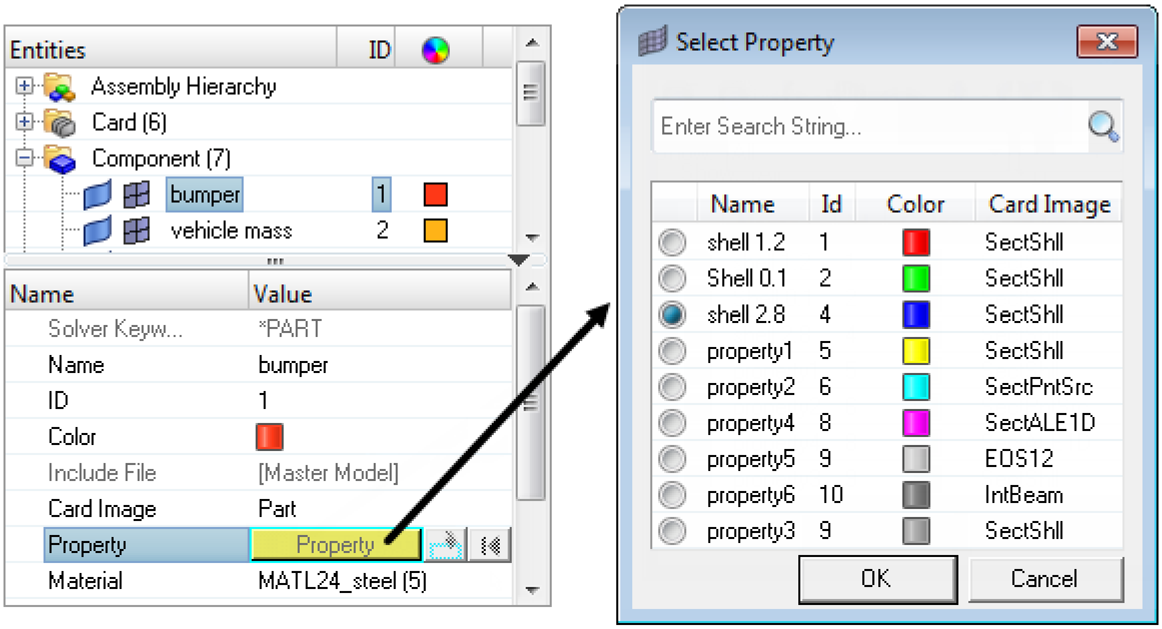

Entities
ID
Assembly Hierarchy
Card (6)
Component (7)
bumper
1
vehicle mass
2
Name
Value
Solver Keyw...
*PART
Name
bumper
ID
1
Color
Include File
[Master Model]
Card Image
Part
Property
Property
Material
MATL24_steel (5)
Select Property
Enter Search String...
Name
Id
Color
Card Image
shell 1.2
1
SectShll
Shell 0.1
2
SectShll
shell 2.8
4
SectShll
property1
5
SectShll
property2
6
SectPntSrc
property4
8
SectALE1D
property5
9
EOS12
property6
10
IntBeam
property3
9
SectShll
OK
Cancel

Figure 329:

Warn Upon Entity Type Change

Any changes that you make in the Entity Editor will be automatically applied. Some of the changes may be irreversible.

1. To confirm changes before they are made, activate the **Warn upon entity type change** option in the right-click context menu.
   When this option is activated, a **Confirmation** dialog will appear every time you make a change in the Entity Editor.
2. To proceed, you must click **Yes** or **No**.

   By default, the Warn upon entity type change option is activated. If you do not want to display this message every time you make a change, you can deactivate this option.

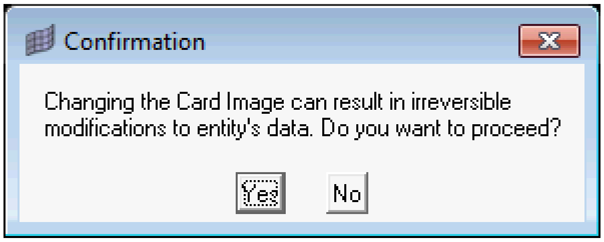


Figure 330:

View an ID List for Set Entities

Set entities are used to define and store lists of entity IDs for a specific entity.

Sets can be generated for nodes, elements, components, assemblies, properties, materials, ellipsoids, multibody planes, multibody joints, and multibodies which contain entity IDs for that specific entity. In the Entity Editor, you can view an ID list for set entities. The entity IDs are compacted using ranges, and segregated using ID pools. If some of the IDs are unresolved, than they will be listed separately under Unresolved IDs.

1. Right-click on the **Entity IDs** field and select **ID List** from the context menu.
2. In the **ID List** dialog, review a list of all of the entity IDs.
3. When you are finished, click **Close**.

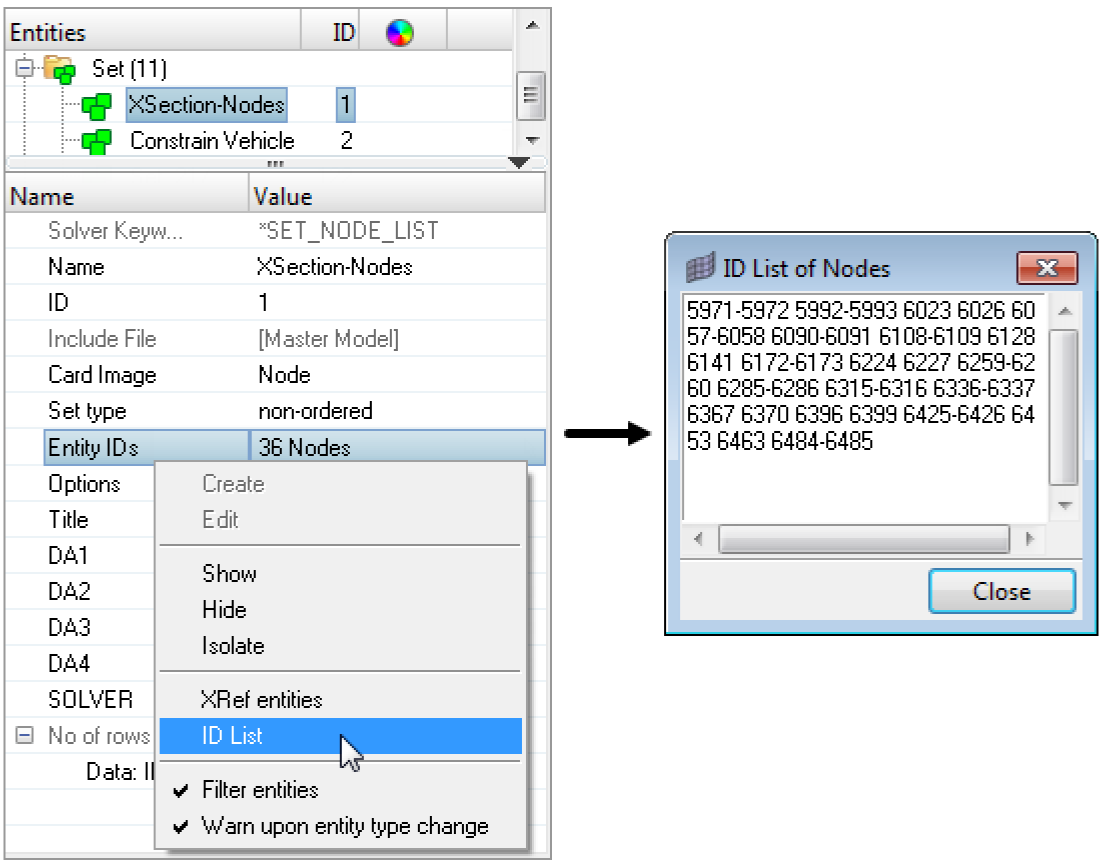


Figure 331:

View Xref Entities

To open the Reference Browser and view a hierarchical tree structure that displays the relationship of a selected entity in the Entity Editor to other entities and parameters in the model, right-click on a field and select **Xref entities** from the context menu.

In the image below, the Reference Browser displays the entities related to the Property entity of the component bumper. If you select an entity from the Reference Browser, the Entity Editor will open and display your entity selection's corresponding data in a non-editable form.

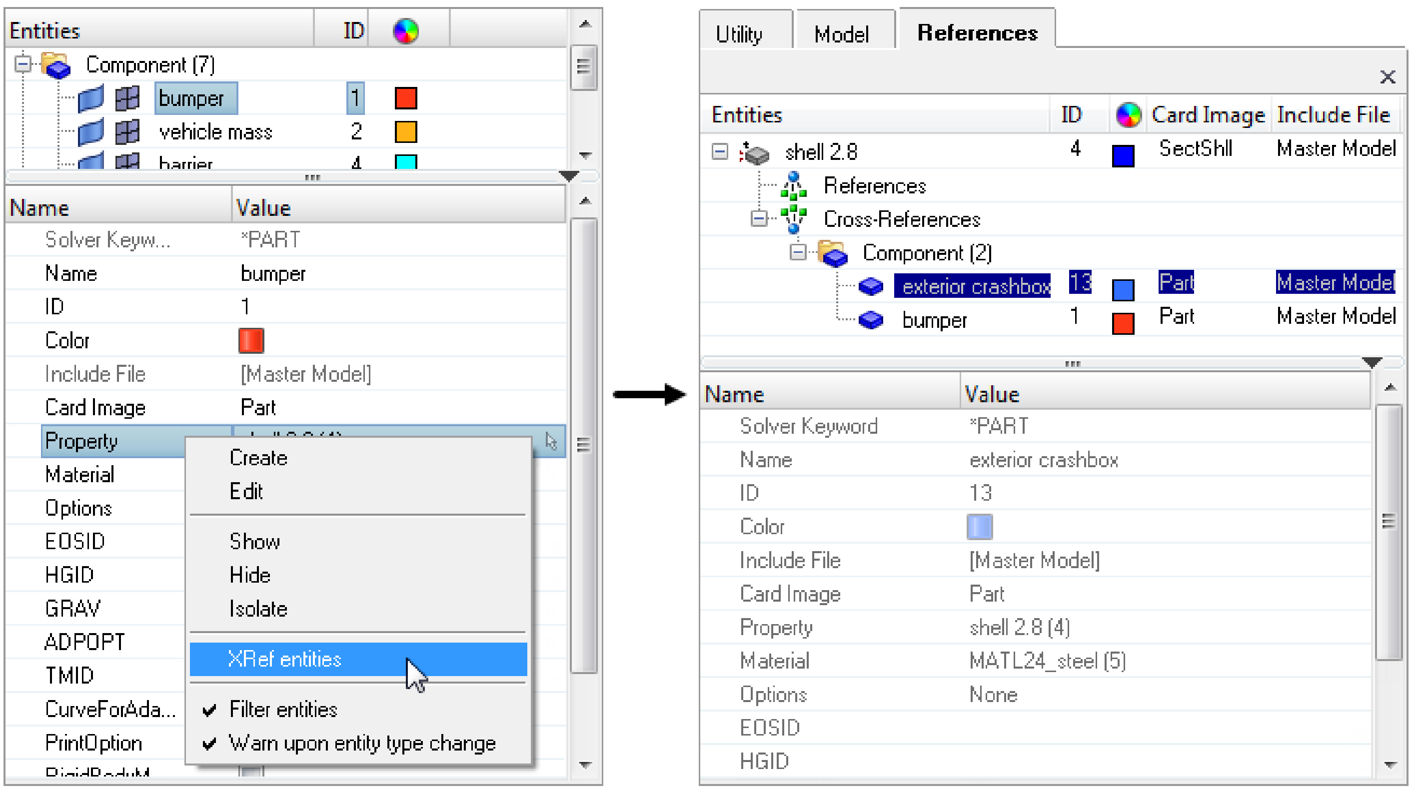


Figure 332:

Entity State Browser

Use the Entity State Browser to set various entity states for entities in a model.

To open the Entity State Browser, from the menu bar, click **View** > **Browsers** > **HyperMesh** > **Entity State**.

For example, from this browser you can set an entity's state to active or inactive.

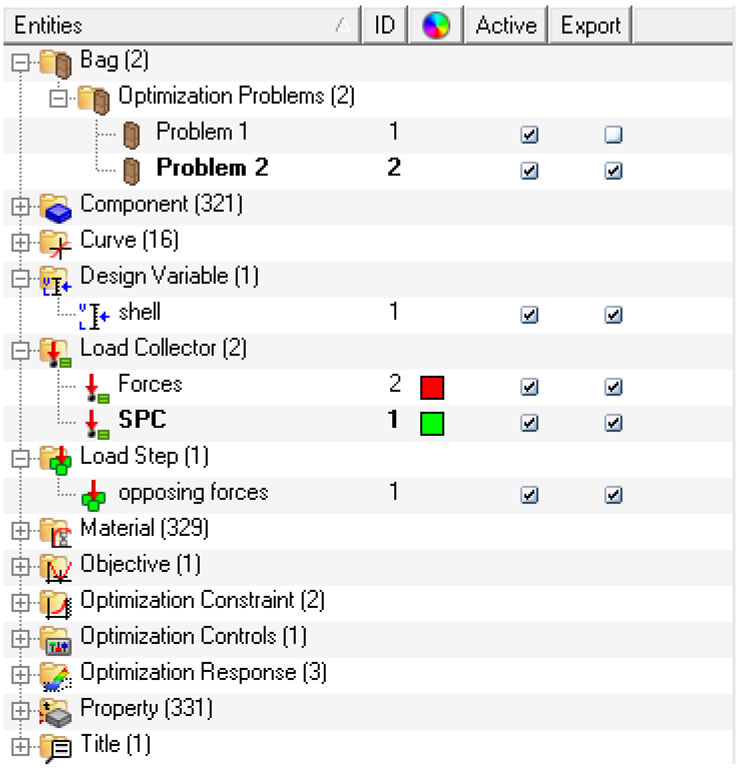


Figure 333:

The active/inactive state is a controllable state whereby the display of inactive entities will be turned off from the display in the graphics, the browsers, the display panel, and any panel entity collectors. It is designed to aid users who frequently work with large models and need be able to filter the list and display, to reduce the number of available or visible entities. Inactive are still present within the model but are removed from access until they are made active again.

The export/do not export state determines whether entities are exported when using the custom export option in the Export tab.

> **Note:** This state does not have any effect on the all and displayed export options.

Furthermore, the active/inactive and export/do not export states are independent of each other – one does not affect the other. Entities that are set inactive are still eligible for all and custom export. They are not output when using the displayed export option since they are, by definition, not eligible for display.

All entities in the current model that have active and export states are shown in the browser at all times. The check-boxes in the Active and Export columns indicate the current settings for those entities and can be clicked to change the state. Each entity is individually controlled via the browser, but all collected entities contained within a collector are subsequently set to the same state as the parent collector – control is not available at the individual collected entity level.

Changing the state of an assembly has two functions. First, it sets the state of that assembly directly. Secondly, it sets the state of all sub-assemblies, components and multibodies referenced by that assembly to the same state as the parent assembly.

Include files do not directly contain any states that can be controlled by the Entity State Browser. Operating on an include will, instead, operate on all supported entities that are referenced by that include.

The Entity State Browser's right-click context menu contains functionality unique to the Entity State Browser, but much of the basic browser functionality – such as sorting and filtering the tree list as well as the functions within the tool sets – are shared with the same features in the Model Browser:

- Action Mode Tools
- Query Builder
- Sorting Entities

Context Menu

The Entity State Browser's right-click context menu contains additional browser options:

Option	Available for:	Description
Set Active	Permanently	Sets the currently selected entities in the browser to the active state.
Set Active Only	Permanently	Sets the entities currently selected in each folder to the active state and sets the remaining entities in those folders to the inactive state.
Set Inactive	Permanently	Sets the currently selected entities in the browser to the inactive state.
Set Inactive Only	Permanently	Sets the entities currently selected in each folder to the inactive state and sets the remaining entities in those folders to the active state.
Set Export	Permanently	Sets the currently selected entities in the browser to the export state.
Set Export Only	Permanently	Sets the entities currently selected in each folder to the export state and sets the

Option	Available for:	Description
		remaining entities in those folders to the do not export state.
Set Do Not Export	Permanently	Sets the currently selected entities in the browser to the do not export state.
Set Do Not Export Only	Permanently	Sets the entities currently selected in each folder to the do not export state and sets the remaining entities in those folders to the export state.
Review	Assemblies, Beamsection collectors, Beamsection, Blocks, Bodies, Boxes, Components, Configuration, Constrained extra nodes, Constrained rigid bodies, Constraints, Contact surfaces, Control volumes, Crosssections, Design variables, Design variable links, Design Objective Reference, Design Variable Property Relationship, Elements, Groups, Joints, Laminates, Load collectors, Loads, Loadsteps, Materials, Mechanisms, Objectives, Optimization constraints, Optimization responses, Output blocks, Part, Part Assembly, Part Set, Plies, Properties, Regions, Seatbelts, Sensors, Sets, System collectors, Systems, Vector collectors, Vectors	Invokes Review mode, which displays selected entities irrespective of their display state, masked, active state (Entity State Browser), but not outside of the spherical clipping (if enabled).
Collapse All	All	Closes all of the folders in the tree structure, so that only the top-most level of items displays.
Expand All	All	Opens all of the folders in the entire tree structure, exposing every item nested at every level.

Option	Available for:	Description
Show Find	All	Turns the browser Find on/off functionality – see Find section for more information.
Show Filter	All	Turns the browser Filter functionality on/off – see Filter section for more information.
Columns	All	Hides or displays the various columns in the tree control.
Configure Browser	All	Opens the **Browser Configuration** dialog, from which you can select which entities to display in the tree as well as which columns the browser displays.

Loadsteps Browser

Use the Loadsteps Browser to create, manage and display loadsteps (sub-cases) and the associated control cards.

The information is arranged into a tree structure for ease of use, with controls for altering the display of the information and/or exporting it. A right-click menu accesses editing and advanced options, while popup forms allow you to quickly enter or select relevant information.

The Loadsteps Browser displays in its own tab in the tab area, but may not be active by default. Select it from the Tools menu to display its tab in the tab area.

The browser includes its own toolbar, used primarily to determine which loadsteps to export but also to sync the display between the browser and the graphics area.

Table 12: Toolbar Buttons

Function	**Description**
Select all, select none, reverse selection	Use these to select the items in the tree and mark them for export. You can also select individual items by clicking on them, or select multiple items by shift-clicking or control-clicking. When a loadstep is selected, the export icon next to its name is clear; when de-selected, the icon has a red "x" to indicate that it will not be exported. **Note:** Export state is independent of visibility in the graphics area. Only one loadstep can display in the graphics area at a time, but any number of loadsteps can be exported.
Sync browser	For large models, keeping the browser in sync with other actions taken can require considerable processing time. To alleviate this, the Loadsteps Browser does not automatically sync itself with the database. Instead, the Sync button becomes active whenever you make changes to the current database. This allows you to perform many operations without performance issues, and then sync the browser with one click.
Filter	Filter buttons allow for additional selection control, including a name filter that uses standard filtering syntax. Use this feature to limit the tree to display only loadsteps whose names match a specific text string, either partly or completely.

OptiStruct and Nastran Profiles

The browser's tree structure lists relevant control cards and loadstep information, organized into folders.

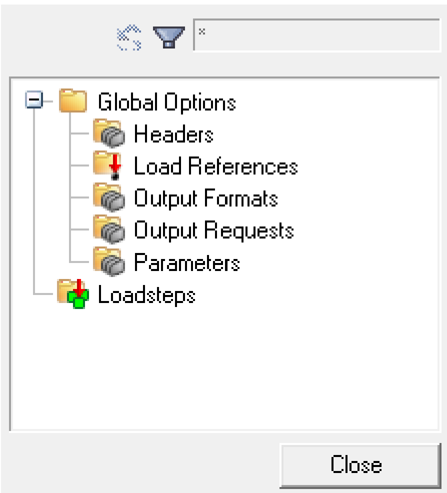


Figure 334:

There are many functions available, accessed by right-clicking on the tree background or on individual or multiple items. For the OptiStruct and Nastran profiles, these options include:

Function	Description
New loadstep	Create a new loadstep, either from scratch or by creating an exact copy of an existing loadstep.
Edit options	Depending on the entity selected, this will bring up an appropriate GUI for editing of the loadstep or control card information.
Edit card	Review the selected entity in the HM card editor.
Delete	Delete the selected entity or entities.
Rename	Rename the selected entity.
Renumber	Renumber the selected entity.
Summary table	Generates a summary table of the selected loadsteps.
Contour Loads	This launches the Contour Loads utility and automatically selects the loadcols associated with the selected loadstep.
Loads summary	This launches the Loads Summary utility and automatically selects the loadcols associated with the selected loadstep.

Function	Description
Collapse all/selection	Collapses all selected folders and subfolders, or all folders if none are selected.
Expand all/selection	Expands all selected folders and subfolders, or all folders if none are selected
Auto-manage load references	This option is for users who wish to have existing DLOAD, LOAD, MLOAD, MOTION, MPCADD and SPCADD cards auto-managed. This option creates a copy of loadcols with these card images and converts them into an auto-managed naming convention for easy editing/reviewing inside the Edit options popup.
OptiStruct	Opens the OptiStruct panel in HyperMesh.

In addition, every loadstep listed in the tree has a small checkbox next to it as well as an export state indicator. You can click these to toggle them back and forth:

The loadcols in the loadstep display in the HyperMesh graphics area.

The loadcols in the loadstep do not display in the HyperMesh graphics area.

This loadstep will not be exported.

This loadstep will be exported.

> **Note:** When you first open the Loadsteps Browser, all of the loadsteps in the model default to the blank (unchecked) state.

Create a New Loadstep

1. Right-click anywhere in the Loadsteps Browser and select **New loadstep**.
   A pop-up window opens, allowing you to:
   - Type in a loadstep name
   - Select the same as option, if desired, then pick an existing loadstep to base the new one on. When this option is active, the new loadstep is an exact copy of the existing one.
2. Click **create**.

Another pop-up window opens, allowing you to edit the loadstep.

Edit a Loadstep

1. Right-click on the desired loadstep folder, or any subfolder in the Loadsteps Browser, and select **Edit**.

   > **Note:** This step is skipped when you create a loadstep.

   A pop-up window opens, allowing you to edit the loadstep. The pop-up has several tabs to gather the relevant information.
2. To activate an option, check the box next to the desired option and fill in the required fields.

   Depending on the Loadstep Type, the list of appropriate Load References will change accordingly. A tree structure lists the load references that are available for the selected loadstep type.
   - A bold reference signifies that the load reference is defined.
   - A red indicator signifies that a load reference is mandatory for the loadstep type and requires attention.
   - A green indicator signifies that a load reference is mandatory for the loadstep type and is defined.
3. The table on the left lists the loadcols that are valid for a particular load reference, depending on the card image or types of loads contained within.
   - Depending on the load reference selected in the tree, the list will change accordingly.
   - You can sort the loadcols by clicking on the column heading that you wish to sort by (repeated clicks alternate between ascending and descending order).

     > **Note:** You cannot sort based on the display column.
   - Name, ID, Type, and Color filtering is available by using standard filtering syntax (color filtering is based on the HyperMesh color ID number).
4. The table on the right lists the loadcols currently selected for that load reference. To add a loadcol to the load reference, select the loadcol in the left table and use the right arrow to add the loadcol to the table on the right.
   - If a loadcol is assigned and that loadcol is not appropriate for that particular load reference, a warning message appears to notify you.
   - If a loadcol is assigned and that loadcol does not exist in the database, a warning message appears to notify you.
   - When importing a model, it is possible that the loadstep may reference loadcols that have not been imported (they are in a separate include file). In order to support this, use the Add <unavailable> load reference ID option to modify a loadstep and add in references to loadcols that do not exist in the current model. These references are also listed in the right table with a warning message to notify you that the loadcol does not exist in the database.
5. To remove a loadcol from the load reference, select the loadcol in the right table and use the left arrow to remove the loadcol.

6. To select multiple loadcols, use the all/none/reverse buttons where appropriate.

   These buttons select loadcols from the currently active table.
7. Right-click options allow for additional functionality depending on the current selection.

   Use the Add <unavailable> load reference ID option to add a reference to a loadcol ID that does not currently exist in the database.
8. Click **Accept** to apply your changes and close the editing window.

   Alternatively, click **cancel** to close the window, discarding your changes.

You can also edit multiple loadsteps simultaneously by selecting more than one loadstep in the browser, such as by shift-clicking, before you choose **edit options** from the right-click menu.

- For each of the options, if the option values are the same for all selected loadsteps, that option is checked "on" and shown with the appropriate values. If the option values are not the same for all selected loadsteps, that option is checked "off" and shown with the default values.
- If the loadstep type is not the same for all selected loadstep, it shows a blank value on the Loadstep Type tab.
- If the loadstep type is not the same, the Load References tab will only display the load reference types that are in common between all of the loadstep types for the selected loadsteps.

Any edits you make will be applied equally to all of the selected loadsteps. Any values that are checked "off" will not be modified, but options checked "on" will all be set to the same values.

Display a Loadstep

1. Check/uncheck the display checkbox next to the loadstep of interest.

   Additional control is also available at both the Global Options and Loadstep Load References level:

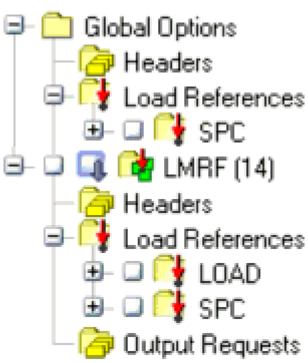
   

   Figure 335:

2. Click the display checkboxes for each desired loadstep to check (display) or clear (hide) it.
   - All of the loads contained in a loadcol display regardless of their relevance to the load reference they are assigned to. It is up to you to organize their loads for proper display.
   - Global load references are not checked on/off by selecting or deselecting a loadstep. You must determine the appropriate loadcols to check on/off depending on the loadstep type.

> **Note:** You can also launch the BCs Contour and Loads Summary utilities from the Loadsteps Browser. The selected utility launches with the loadcols associated with the selected loadstep automatically selected.

Rename, Renumber, Delete or Edit the Card of a Loadstep

1. Right-click on the appropriate loadstep or loadcol.
2. Select the desired option from the pop-up menu.
3. For renaming and renumbering, an entry box appears so that you can enter the appropriate information in the browser.

Edit the Global Options

Editing Global Options works exactly like editing a loadstep, except that the first step is to right-click the **Global Options** folder or any of its sub-folders, instead of clicking on a specific loadstep's folder or sub-folder.

Auto-Manage Load References

This option is recommended for all users. There has traditionally only been one way to create DLOAD, LOAD, MLOAD, MOTION, MPCADD and SPCADD loadcols: by creating a loadcol, assigning the appropriate card image, and selecting the appropriate loadcols. However, many users do not want to be responsible for managing these load collectors, nor are they always aware of their existence. To satisfy both types of users, the Auto-manage load references option is available. This option does the following:

1. Looks at each loadstep and at each load reference. If the load reference points to a loadcol with one of the card images indicated above, it will:
   - Create a copy of that loadcol and assign it a new name, based on a fixed naming convention (auto<CARD IMAGE NAME>_#). For example, if a load reference pointed to an SPCADD loadcol, a new copy would be created and named "autoSPCADD_1".
   - Assign that new loadcol to the original load reference. The original loadcol is not deleted or modified in any way.
2. Inside the Edit options pop-up, if a load reference points to a loadcol with one of the card images above and that loadcol has not been converted to the auto-managed naming convention, the loadcol will not be expanded or editable inside the GUI. The only way to modify the loadcol is via the card editor (right-click option from the editor GUI).

   Inside the Edit options pop-up, if a load reference points to a loadcol with one of the card images above and that loadcol has been converted to the auto-managed naming convention, the loadcol is expanded and editable inside the GUI.

If the loadcol selected for the load reference already has the card image assigned (for users wishing to manually manage their loadcols and point to an existing loadcol with one of the card images listed above) no additional action takes place. However, when appropriate, a loadcol is automatically created and assigned the correct card image when any of these conditions are met:

- More than one loadcol is selected for the load reference
- One loadcol is selected and the local scale factor is not 1.0 (DLOAD and LOAD)
- The global scale factor is not 1.0 (DLOAD and LOAD)

Mask Browser

In the Mask Browser, set the mask/unmask state for entities at the entity configuration level.

To open the Mask Browser, from the menu bar, click **View** > **Browsers** > **HyperMesh** > **Mask**.

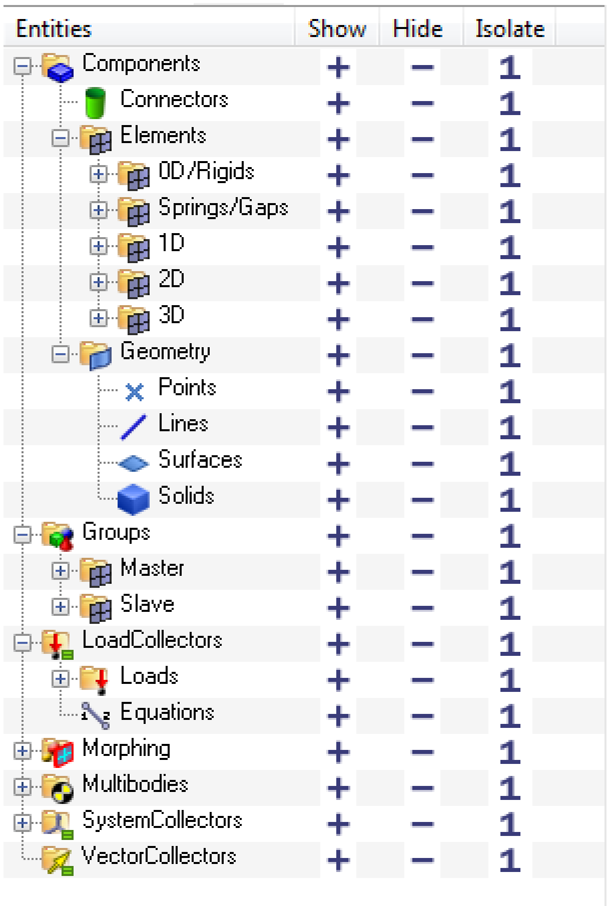


Figure 336:

The entities are logically organized in the browser to represent the collectors they belong to. Regardless of the current model, the entities listed in the browser remain the same. The Show/Hide/Isolate columns contain icons that can be clicked to perform the relevant masking operations. The buttons perform the masking operations at the selected entity and folder level, and for all entities and sub-folders that may be contained within that folder. These operations are only valid for entities contained in collectors that are currently displayed.

The Show column corresponds to the unmask operation. It unmasks the relevant entities for the current row and sub-folders. For example, the Show icon at the Geometry folder unmasks all points, lines, surfaces and solids within any displayed components.

The Hide column corresponds to the mask operation. It masks the relevant entities for the current row and sub-folders. For example, the Hide icon at the 1D folder masks all rod, bar2, bar3, weld, joint and plot elements within any displayed components.

The Isolate column corresponds to both a mask and an unmask operation. It performs a Hide on the top level folder and then a Show on the current row and sub-folders. For example, the Isolate icon at the 3D folder masks all connectors, geometry, 0D/rigid elements, spring/gap elements, 1D elements and 2D elements and unmasks all 3D elements within any displayed components. The exception to this rule is when the Isolate button is selected at a top-level folder; in this case, all sub-folders underneath the top-level folder are unmasked and all other top-level folders, and their contents, are masked. For example, the Isolate icon at the Components folder masks all supported entities in any displayed groups, load collectors, morphing, multibodies and system collectors, and unmasks all supported entities within any displayed components.

The Mask Browser's context menu contains functionality unique to the browser.

Context Menu

The Mask Browser's right-click context menu contains additional browser options.

Option	Available for:	Description
Collapse All	All	Closes all of the folders in the tree structure, so that only the top-most level of items displays.
Expand All	All	Opens all of the folders in the entire tree structure, exposing every item nested at every level.
Morph operates on all elements/Morph operates on displayed elements	All	Determines whether the masking operations for morph entities apply to all/displayed elements.

Mass Trimming Browser

Use the Mass Trimming tool to create, define and realize mass entities.

Mass entities are used to define mass distribution. A mass entity stores information like the mass value, method of distribution, NSM, Point Mass, or Rigid mass with a mass center, and attached locations and realization types. These definitions are not associated with physical FE entities. If all of the FE entities in the model are deleted, the mass information will still be maintained. Mass entities can be saved in an HyperMesh binary file which can be imported into a new FE model and realized.

Open the Mass Trimming tool from the menu bar by clicking **Tools** > **Mass Trimming**.

Mass Trimming Tool Interface

The Mass Trimming tool opens in the browser area.

The toolbar at the top of the Mass Trimming tool consists of tools used to create mass entities and control their display state.

The first pane displays all of the mass entities in your model, and their associated attributes. The second pane displays the Entity Editor, which can be used to define the attributes assigned to each mass entity.

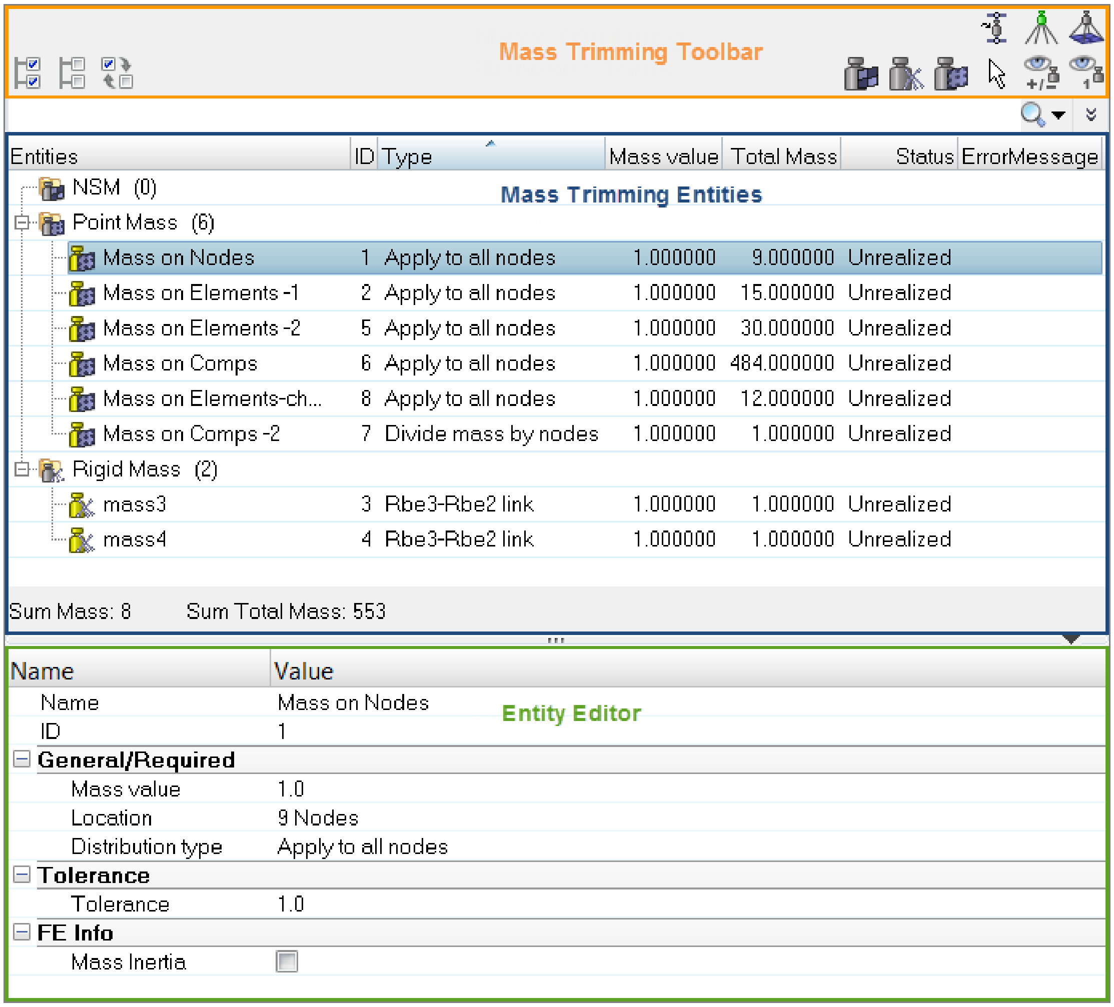


Figure 337:

Mass Trimming browser tools:

Tool	**Description**
	Creates an NSM mass.
	Creates a Rigid mass.
	Creates a point mass.
	Interactively select any type of supported mass entity via the browser, or by selecting within the graphics area. The Selector can be used to find mass entities from the graphics area which will then be highlighted in the list. This is an efficient way of selecting multiple mass entities at once.

Tool	Description
	This tool can be used in conjunction with the Show, Hide, Isolate, and Isolate Only tools.
	Show/hide selected mass entities. Alternatively, click this mode on and then pick the desired mass from the graphics area; masses and any other entities determined by the view option toggle settings are hidden as you click on them. **Note:** When used to select from the graphics area, this button only works on visible connectors.
	Isolate/isolate only selected mass entities. Alternatively, click this mode on and then pick the desired mass entity from the graphics area.
	Opens the **Perform FeAbsorb** dialog, from which you can add (absorb) certain FE information into mass entities.
	Finds and displays the realized FE for the selected mass entities.
	Finds the entities attached to the selected mass entity. When this tool is active, all entities attached to the selected mass entity will be taken into account when a display change is performed.

Context Menu

In the Mass Trimming Browser, right-click to access the context menu. The options available in the menu are dependent upon whether you right-click on a mass entity, or in the white space of the browser.

Option	Description
Create	Create a mass entity.
Delete	Delete the selected mass entity.
Rename	Rename the selected mass entity.
Xref Entities	Opens the References Browser and displays the relationship of the selected mass entities to other entities in the model in a hierarchical tree structure.

Option	Description
Realize	Realize the selected mass entity.
Re-Realize	Re-realize the selected mass entity.
Unrealize	Unrealize the selected mass entity.
	If certain attachments cannot be made using the tolerance that is set, then a reconcile will accept the current attachments, ignore the failed ones, and change the status to green.
Show, Hide, Isolate, Isolate Only	Control the display of the selected mass entities.
Review, Reset Review	Review will show the mass entity highlighted with the rest of the displayed content in a grey transparent mode. Reset will restore the display.
Settings	Apply a search order for failed mass attachment locations. Only applicable for rigid masses with spider connections, and RBE3-RBE2 and RBE3-RBE2 links.

Supported Entities

Supported mass entities include:

NSM ()
A non-structural mass that can be distributed to elements, properties, and components, and defines/updates the required solver cards. A group entity is created for each mass entity. The group entity can be assigned a NSM1 or NSML1 card depending on the distribution method chosen. Only supported in OptiStruct and Nastran user profiles.

Rigid ()
A mass that takes the form of a mass element connected to nodes with rigid elements (Multiple RBE2, RBE3/RBE2, and so on).

Point ( )

A single mass element that is created at each selected node. A value is computed based on the distribution method.

Mass Entity States

There are four states of mass entities. Each mass state is assigned a different color. Color coding provides an easier way to visualize and filter masses based on state.

unrealized ( )

The initial definition of the mass entity after it is created. Unrealized masses are displayed yellow.

realized ( )

The mass is considered realized only if weld creation at the mass was successful. Realized masses are displayed green.

modified ( )

The mass is considered modified when one or more of its corresponding attributes have been edited in the Entity Editor. Modified masses are displayed blue.

failed ( )

The mass is considered failed if the weld creation at the mass was not successful. Failed masses are displayed red.

Manage Mass Entities

Create, define and realize mass entities.

Create Mass Entities

Create Mass entities in the Mass Trimming Browser using one of the following methods:

- In the browser, right-click and select **Create** from the context menu.
- Click the **Create Mass** tool for the appropriate mass type on the toolbar.

Masses are organized by type in the browser.

Define Mass Entities

Define mass entities in the Entity Editor.

You can modify the distribution type, mass value, and total mass in the Mass Trimming Browser.

1. In the Mass Trimming Browser, select the mass to define.
2. In the Entity Editor, define the mass.

Mass Attributes

Supported attributes for Mass entities.

NSM Mass

Mass value

Mass value to be distributed.

NSM Distribution type

- Total Mass. A NSML1 card is created with either an element set or a property set that is based on the NSM entity type definition.
- Per unit area/length.

Property type

Type of elements or properties used to distribute the mass. Supported property types include PSHELL, PSHEAR, and PCOMP.

Location

- If NSM entity type is Elements, than elements can be selected in the model and element set will be created for the NSM solver card.
- If NSM entity type is Properties, than properties can be selected in the model and a property set will be created to realize the mass entity.

Reconnect rule

Rigid Mass

Calculate node

Calculates the mass location at the CG of the attachment points.

Mass value

Mass value to be put at the mass center.

Realization type

- RBE2. The mass takes the form of a mass element connected to nodes with rigid elements (multiple RBE2).
- RBE2 link. The mass takes the form of a mass element connected to nodes with a rigidlink element (single RBE2).
- RBE3. The mass takes the form of a mass element connected to nodes with a RBE3 element.
- RBE3-RBE2. The mass takes the form of a mass element connected to nodes with a RBE3 element connected to RBE2.
- RBE3-RBE2 link. The mass takes the form of a mass element connected to nodes with a RBE3 element connected to RBE2 link.

Location

Position at which the mass is created. If the calculate node option is not selected, then a location for mass center will need to be selected.

Attachment point

To define the attachment points, nodes have to be selected in the model. Node selection is not used as is. A search is performed to see what is attached to the nodes.

- If a node is selected on a hole, then the center of the hole will become the attachment location. If multiple nodes on a hole are selected, they will still treated as one attachment.
- If a node is selected on a mesh, then the nodes of the attached elements (shell or solid) will be selected.
- If two or more nodes are selected in close proximity and the attached elements are adjacent, then the entire patch will be selected.
- If an independent node of a rigid is selected, then the independent node is selected for the attachment. The second leg is not created for two level rigid masses, such as RBE3-RBE2 or RBE3-RBE2 links. The first leg is terminated at the independent node.
- If the free end of a CBUSH, with or without a tag, is attached to a rigid on the other end, then the second leg is not created and the rigid type mass is terminated at the first leg.

Attached Location Tolerance

Tolerance value used to search for nearby FE entities to attach.

Realization zone

Only available when the attachment is on a hole.

- Inside. Use the nodes of the holes.
- Outside. Use the node from the elements that form the hole, without the hole nodes.
- Inside/outside. Use the nodes of the elements at the hole.

Mass Inertia

Rbe2/Rbe2 link

Realized FE in

- Auto
- Realized FE to current comp
- Realized FE to selected comp

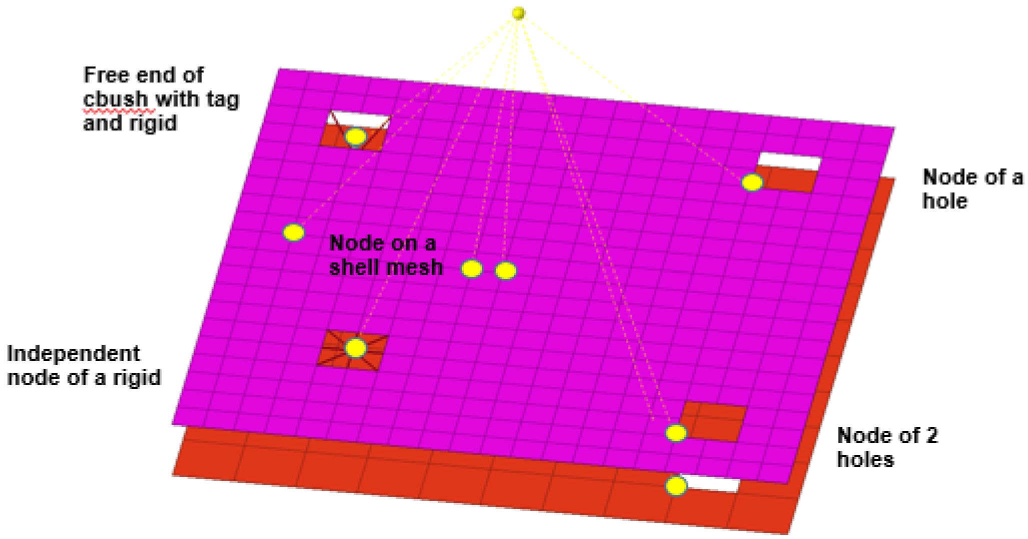


Figure 338: Nodes Selected for Attachment Points

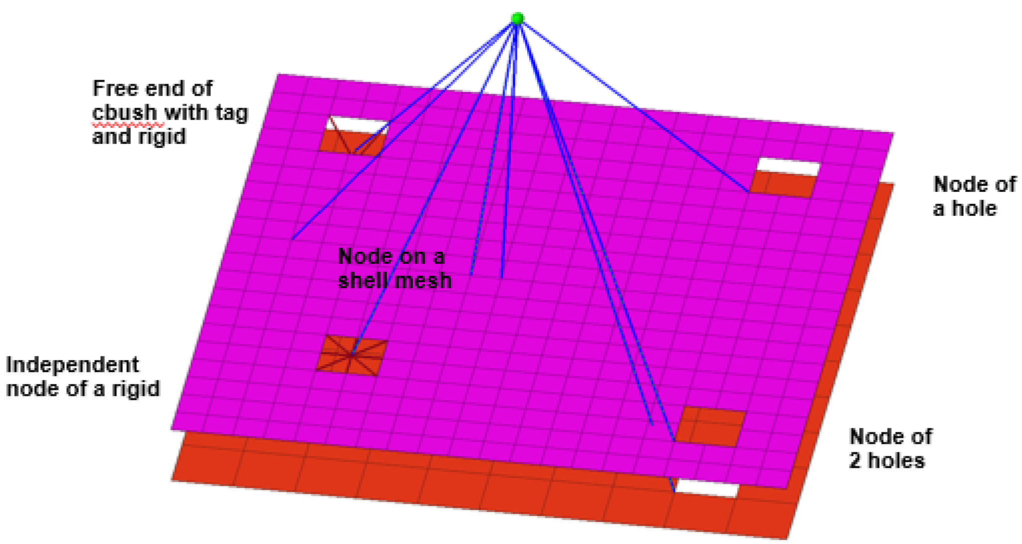


Figure 339: Manual Mode for Single Level Rigid Creation

In this mode, only the selected nodes are used to define the rigid.

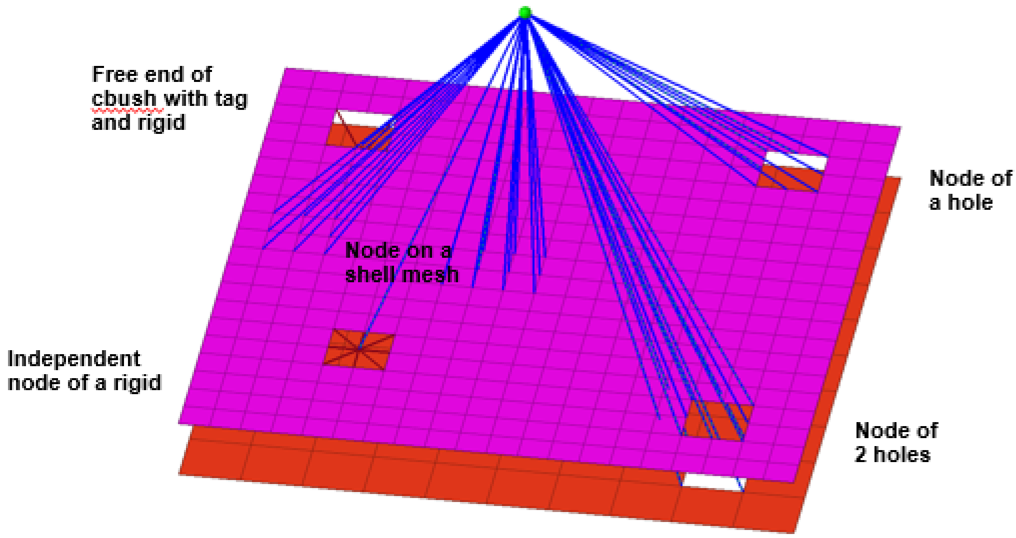


Figure 340: Automatic Mode for Single Level Rigid Creation

If the Manual option is not enabled in the Entity Editor, the nodes of adjacent elements will be used to define the rigid if the selected nodes are on a hole or are nodes on a mesh.

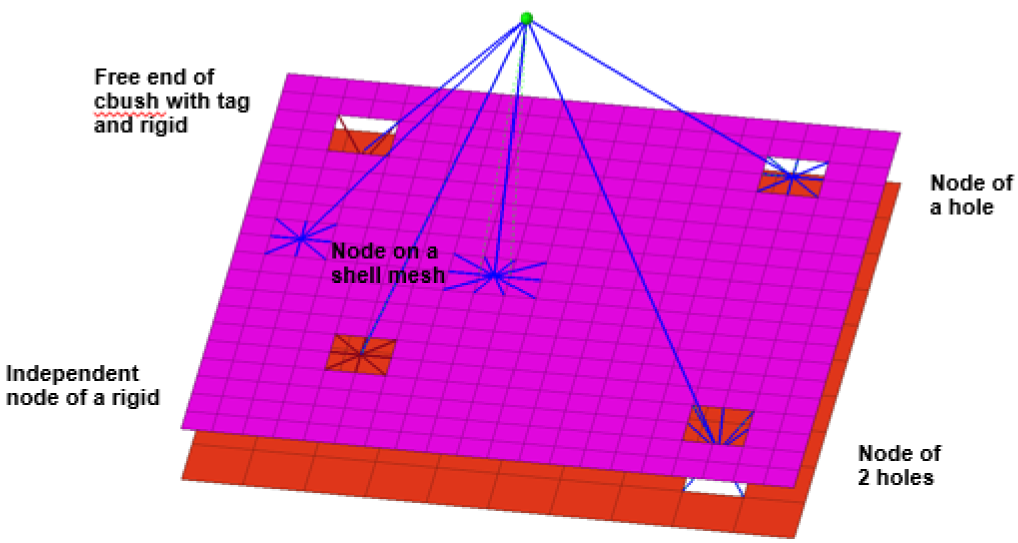


Figure 341: Two Level Rigid

In the case of a two level rigid, Automatic and Manual mode is not available. Instead, input nodes are processed. If input nodes are on a hole, then the nodes of the hole are used. If the realization is set to inside or inside/outside, then washer nodes are used. When the input node is an independent node or the free end of a cbush attached to a rigid, then the second level rigid will not be created. The RBE3 from the conm is terminated at these nodes.

Point Mass

Mass value

Mass value to be used for defining point masses.

Location

Position at which the mass is created. Can be nodes, elements, properties, or components. For elements, only shell elements are supported.

Distribution type

- Apply to all nodes
- Divide mass by nodes
- Divide mass by area (Not available when Location is defined by Components or Properties)
- Unit area distribution (Not available when Location is defined by Components or Properties)

Tolerance

Tolerance value used to search for nodes to create point mass elements. point mass on nodes, the tolerance value is used for a spherical search.

Threshold for Element Remapping(+-%)

When the old area and tolerance are used to search in the new mesh, the remapping may not always give 100% of the old area based on mesh size and pattern. A % value like +-10% will allow you to successfully remap and realize if the difference in area is within 10%.

Mass Inertia

Realized FE in

- Auto
- Realized FE to current comp
- Realized FE to selected comp

Limitations

- For NSM and Point masses defined on elements, if the mass entity is in an unrealized state and partial elements are deleted, then upon rerealization, the status will be failed. If you translate the nodes of the elements and try to realize again, the graphics may be misleading because the failed locations might not reflect the translated location. However, re-realization with appropriate threshold values or a reconcile will work properly.
- For NSM and Point masses defined on elements, if the mass entity is saved as an HyperMesh file and realized in a new design that has coincident elements in the same area, then both elements will be selected which will result in an increase of area of elements; hence the mass entity will fail, showing a new threshold value that is needed in order to realize properly. In such cases, the duplicate elements have to be deselected for those mass entities.
- For Rigid masses in automatic mode, if a node on a hole is selected, then the nodes of the hole will be automatically selected. However, if the node is on the boundary of a mesh, then the nodes of the entire boundary will be selected even though it is not a hole. In automatic mode, it is advised that you do not select boundary nodes.
- When performing FE-Absorb on Rigid masses, a rigid mass might be absorbed as a mass entity, and the realization might fail if the attachment point is not supported. For example, if the Rigid from a CONM2 mass is connected to a dependent node of another Rigid, FE-Absorb will absorb this as a mass entity. Re-realization will fail because only the independent node is supported as a valid node and not a dependent node.

- For Rigid masses, Rigids will be absorbed during FE-Absorb as a Mass entity when they are connected to spring elements of any type, and a Rigid will be created during re-realization. If the FE model is deleted from the current session, realization will fail when you try to reapply the Trim mass to a new model, and a graphics message that reads "CBUSH node" will be displayed. At this point, you will only be able to reconnect to CBUSH spring elements.

Realize Mass Entities

In the Mass Trimming Browser, right-click on a mass entity and select **Realize** from the context menu.

Resolve Failed Connections

Execute the search order to resolve failed connections.

1. In the browser, right-click on a Rigid mass and select **Search Order** from the context menu.

   When defining Rigid masses, the following attachments are defined:

   1. Node on a shell mesh
   2. Node of a hole on a shell mesh
   3. Independent node of a rigid
   4. Free end of a CBUSH with or without a tag that is attached to rigid on the other end

   Mass entities will fail during realization when:

   1. Current mesh is deleted
   2. A new model is imported that does not have a hole at the second location
   3. There is no independent node and free end of a CBUSH at #3 and #4
2. You can execute a search order to find alternate connection methods at failed locations. The search order provides the following options which can be selected and ordered in the sequence it is desired.

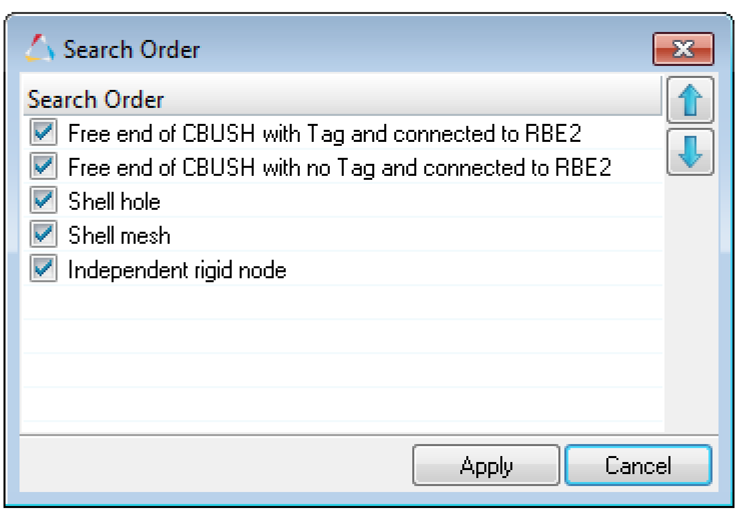


Figure 342:

If you execute the search order, an alternate attachment method will be searched based on the tolerance in the sequence defined and establish connection at the failed locations. At this point, the new method will be saved for sub-sequent realizations. If within the tolerance, the search order does not find an alternate attachment method, then the status will remain failed. However, if after the search order is executed, and alternate attachment methods are established within the tolerance, then the status will turn to yellow and the mass entity can be realized.

3. Click **Apply**.

Below is an example of reapplying Rigid Mass entities to a new mesh with mesh modifications, such that reapplying fails, but executing a search order finds alternate types of attachments.

1. In the original mesh, the attachment type is a node on a shell mesh which finds nodes of adjacent elements to create the second level rigid.

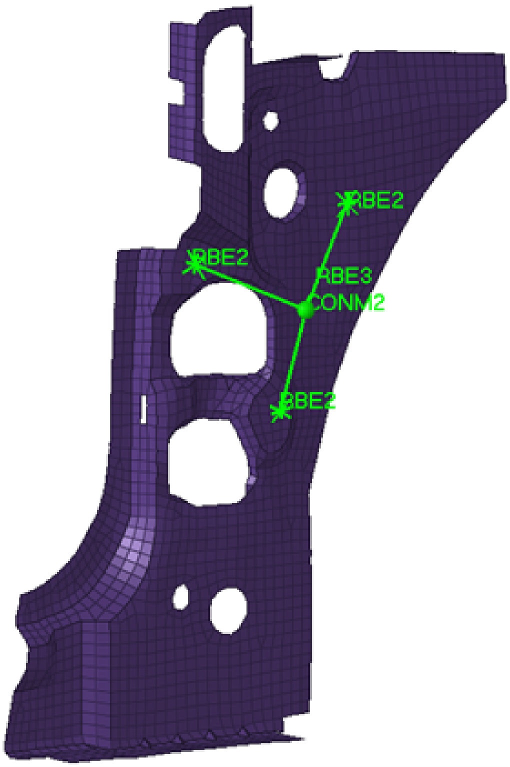


Figure 343:

2. When the mesh is deleted and realized, the realization fails. The modeling window indicates what it is looking for at those attachment locations.

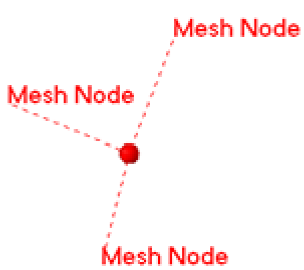


Figure 344:

3. When importing a new mesh, there is no shell mesh near the attachment within the tolerance at two locations; however, there is a hole and an independent node. After realizing, the mass fails at the two locations.

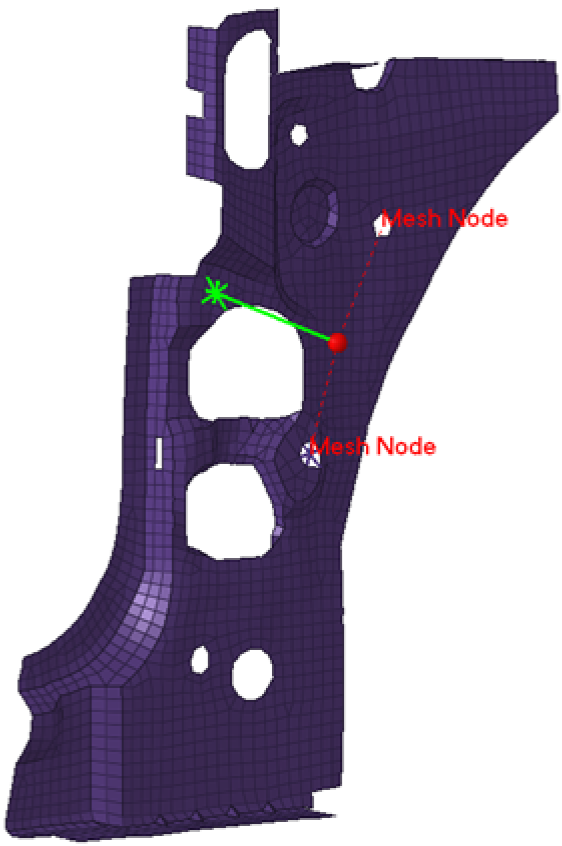


Figure 345:

4. Executing a search order with the following settings finds alternate attachment methods. The search order is set to first look for independent node. If no independent nodes are found it will then look for a shell mesh. The other options are not selected.

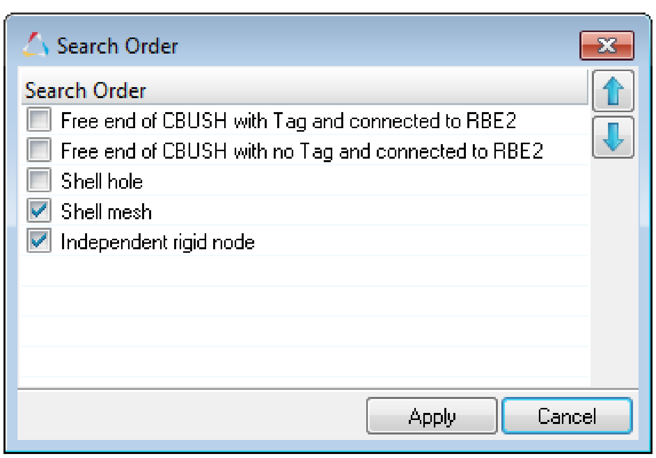


Figure 346:

5. The image below shows the alternate attachments that were found at those locations, which are realized.

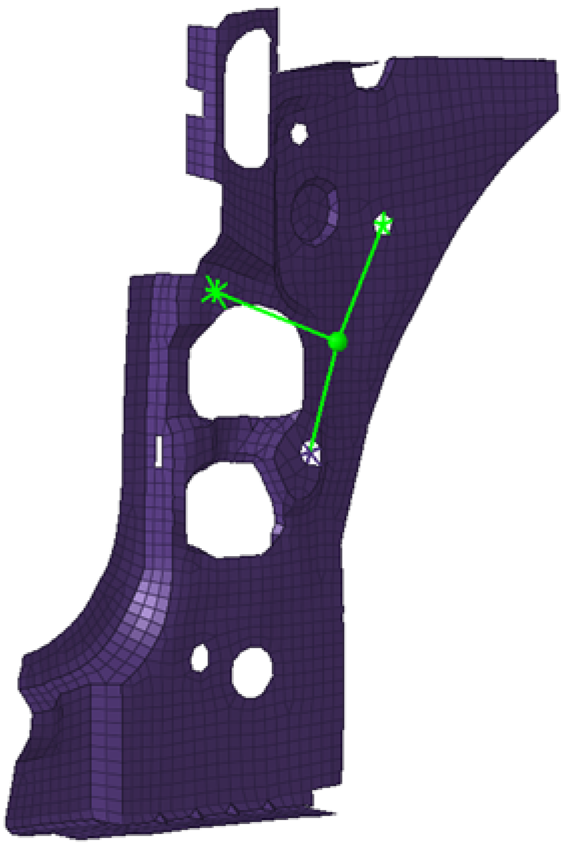


Figure 347:

6. If the mesh is deleted and realized again, the new attachment method will be used for subsequent realizations. The original attachment method was node on a mesh at all three locations, but after executing the search order on the new mesh, the attachment methods change.

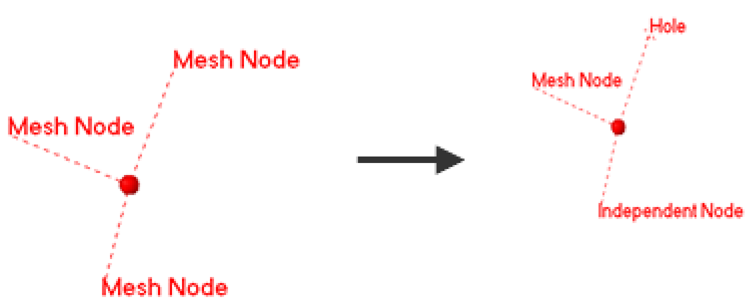


Figure 348:

Perform FE Absorption

For FE models, use FE-Absorb to absorb the Trim Mass into the Mass entity.

During FE Absorption, you can only use realization types that are supported by the Mass Trimming tool. For second level rigids that cannot be absorbed completely, the independent node will be used as the attachment node.

Rigid Mass

1. In the Mass Trimming Browser, click on the toolbar.

   The **Perform FEAbsorb** dialog opens.
2. For Mass Type, select **Rigid**.
3. Set the Entity filter.
   - Choose **All** to checks all elements to find FE.
   - Choose **Select** to open the entity selector, which you can use to select a portion of the model to find FE.
4. For Realized FE in, select a component to organize realized FE.

   During absorption, no reorganizing is done based on this setting.

   Only available for realizations
   - Auto
   - Realized FE to current comp
   - Realized FE to selected comp
   - Realized FE to original comp
5. Click **FeAbsorb**.

   In the image below, the second level leg is attached to the node surface of the solid hole, which is not supported during the realization process. The second leg will instead be used to absorb the attachment to the independent node. A second level spider will not created during realization.

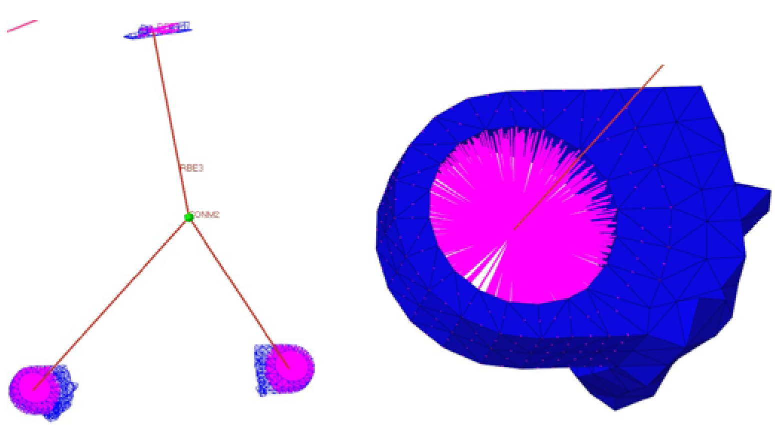


Figure 349:

Point Mass

1. In the Mass Trimming Browser, click on the toolbar.

   The **Perform FEAbsorb** dialog opens.
2. For Mass Type, select **Point**.
3. Set the Entity filter.
   - Choose **All** to check all elements to find FE.
   - Choose **Select** to open the entity selector, which you can use to select a portion of the model to find FE.
4. Select an Entity creation rule.
   - Choose **Based on magnitude** to group all CONM's that are located directly on a mesh based on the magnitude, and creates one mass entity.
   - Choose **Based on component** to group all CONM's that are located directly on a mesh based on the magnitude and component the mesh is in, and creates one mass entity. If several meshed components have CONMs with the same magnitude, multiple mass entities will be created.
5. Select an Apply rule.
   - Apply on all nodes
   - Total mass
6. Click **FeAbsorb**.

   Only CONM2 masses will be absorbed since that is the only mass type supported during realization.

NSM Mass

1. In the ,Mass Trimming Browser click on the toolbar.

   The **Perform FEAbsorb** dialog opens.
2. For Mass Type, select **NSM**.
3. Set the Entity filter.
   - Choose **All** to checks all elements to find FE.
   - Choose **Select** to open the entity selector, which you can use to select a portion of the model to find FE.
4. Select a Reconnect rule.

   This option is useful when the entities to be connected will be exchanged in a subsequent process. When rerealizing mass entities, HyperMesh looks for attachment entities based on the reconnect rule.

   - Choose **None** to not define a reconnect rule. If the attachment entity is not currently in the model, the mass with this reconnect rule will fail to realize. If no reconnect rule is defined for an attachment entity, the attachment will disappear from the mass when the attachment entity is deleted.
   - Choose **Use id** to reconnect using the IDs of the selected attachment entities. If the attachment entity is not currently in the model, the mass with this reconnect rule will search for entities with the same ID.
   - Choose **Use name** to reconnect using the names of the selected attachment entities. If the attachment entity is not currently in the model, the mass with this reconnect rule will search for entities with the same name.
5. Click **FeAbsorb**.

Matrix Browser

Use the Matrix Browser to explore the HyperMesh and HyperView data entities and export the entities to external files for further analysis.

To open the Matrix Browser, click **Tools** > **Matrix Browser** from the menu bar.

The Matrix Browser is organized into two main sections. The top half of the browser provides a table view similar to an Excel spreadsheet. The bottom half of the browser shows and allows you to browse the HyperMesh/HyperView (data source) and its data entity (object) names and their data names (attributes). The HyperMesh database entities are shown in green and the HyperView database entities are shown in yellow. When you select a column in the table it becomes an active column and is highlighted in blue. All of the following searches are conducted on the active column.

The Matrix Browser's context menu contains functionality unique to the browser.

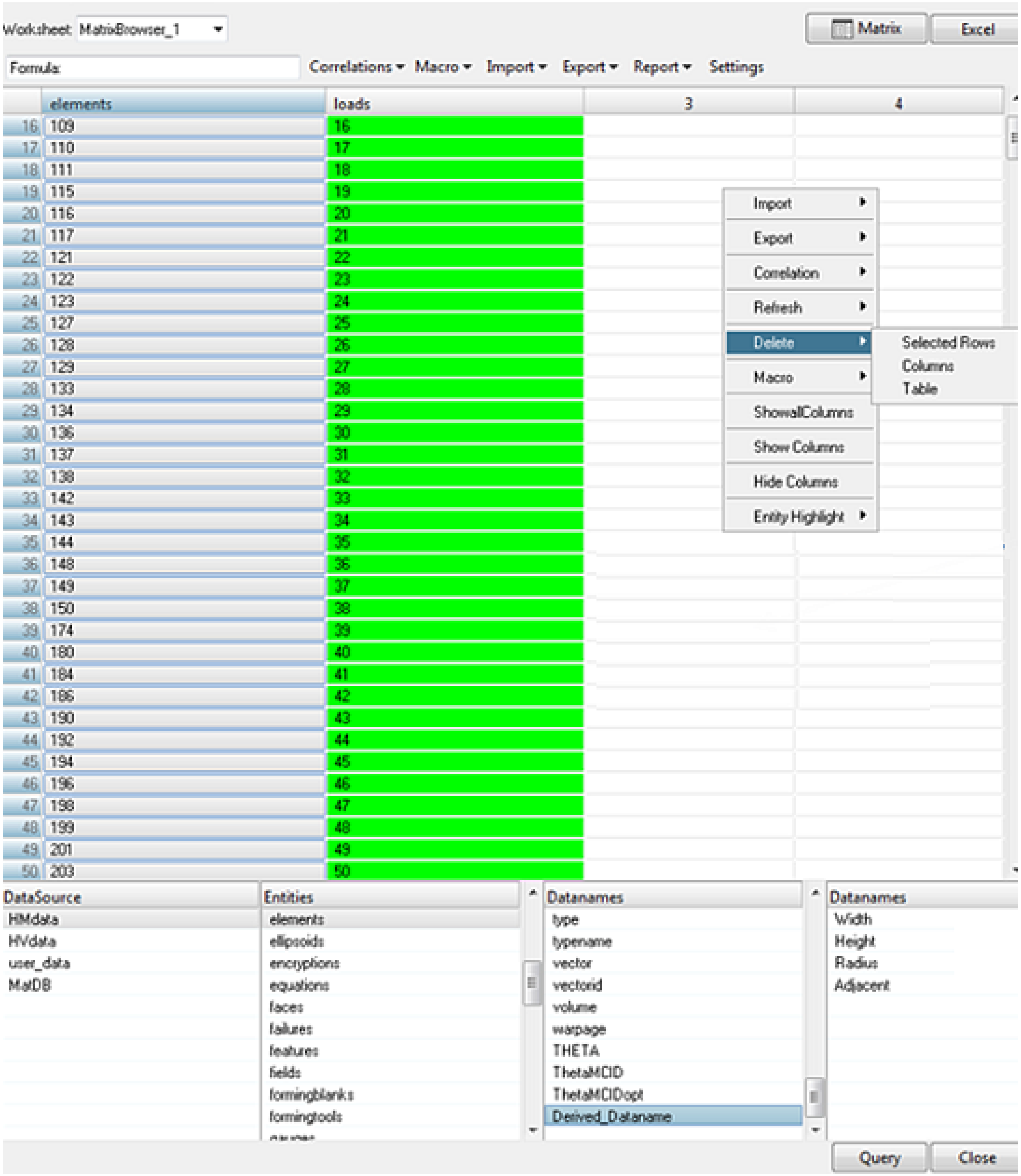


Figure 350:

Matrix Browser Functionalities

Below is a highlight of the functionalities supported by the Matrix Browser.

- Query and modify HyperMesh and HyperView data entities without writing cumbersome Tcl coding.
- Export HyperMesh and HyperView data to an Excel file for further post-processing and report generation.

- Establish a link between HyperMesh and HyperView in order to access model data, such as material, property, beam section, and results data in a single environment. The browser supports the selection of multiple load cases from the HyperView results database.
- Create new procedures (macros) to enable automation with minimum Tcl coding and also capture the automation process as Tcl scripts. These macros can be used external to the Matrix Browser, like the tools in the Altair script exchange.

> **Note:** The Matrix Browser does not allow for the creation of new HyperMesh entities. The tool queries and modifies existing entity data. The browser also does not allow extensive geometry (lines, surface, solids) data query. It is intended for FE data and results that are already in the HyperMesh/HyperView databases.

Prepare to Use the Matrix Browser

1. Open an existing HyperMesh model or import solver data.
2. Use the split screen and invoke HyperView.
3. Load the results file for the solved data into HyperMesh.
4. Click **Tools** > **Matrix Browser** from the menu bar in HyperMesh.

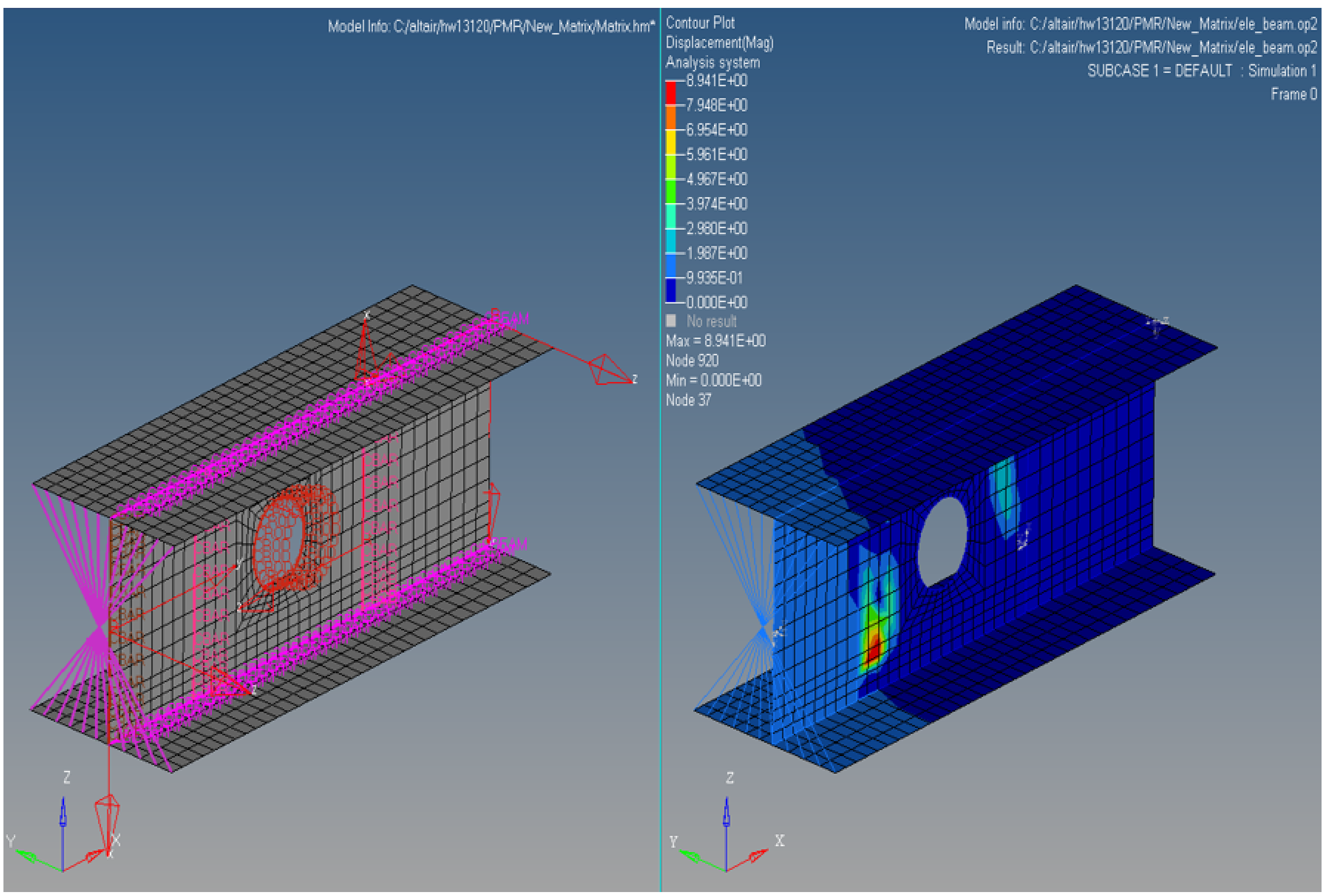


Figure 351:

Model opened in HyperMesh and HyperView.

> **Note:** HyperView is only needed if the results are accessed from the Matrix Browser. Use HyperWorks Desktop to access HyperView functions from HyperMesh in the browser.

Access HyperMesh Data Using the Matrix Browser

1. In the DataSource column, select **HMdata**.
2. In the Entities column, select an HyperMesh entity.
3. Click **Query**.
4. In the panel area, use the entity selector to select HyperMesh entities to add to the matrix.
   A new green colored column is added to the matrix.
5. In the matrix, click the **elements** header to access entity data.
   The elements column turns blue when clicked, indicating the entities are active for the search operation. The Datanames column is populated with entity attributes from the HyperMesh database for the element IDs shown in the elements column.

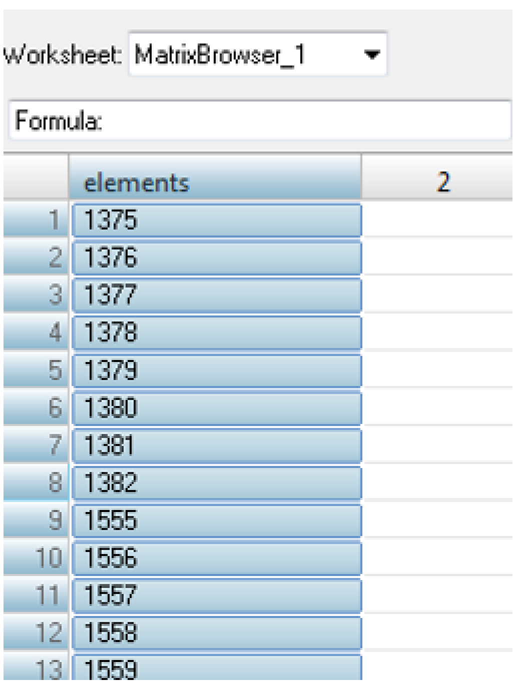
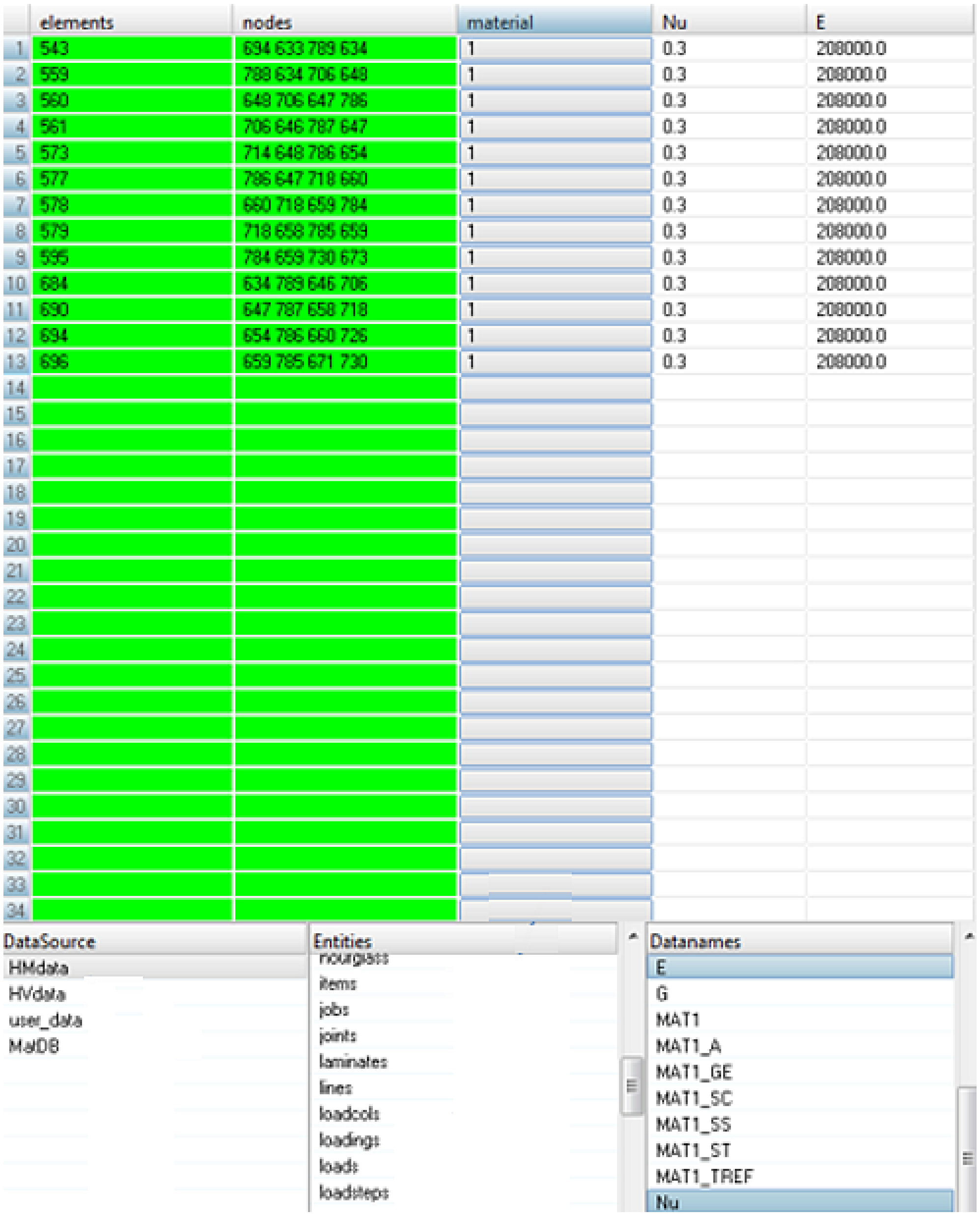


Figure 352:

6. Select the material, property, or any other attribute(s) and click **Query**.
   One of more columns will be added. If you selected material, the material IDs will be shown.
7. The next step is to get E, Nu from the material. Activate the **material** column, and then select **E** and **Nu** from the data names.



	elements	nodes	material	Nu	E
1	543	694 633 789 634	1	0.3	208000.0
2	559	788 634 706 648	1	0.3	208000.0
3	560	648 706 647 786	1	0.3	208000.0
4	561	706 646 787 647	1	0.3	208000.0
5	573	714 648 786 654	1	0.3	208000.0
6	577	786 647 718 660	1	0.3	208000.0
7	578	660 718 659 784	1	0.3	208000.0
8	579	718 658 785 659	1	0.3	208000.0
9	595	784 659 730 673	1	0.3	208000.0
10	684	634 789 646 706	1	0.3	208000.0
11	690	647 787 658 718	1	0.3	208000.0
12	694	654 786 660 726	1	0.3	208000.0
13	696	659 785 671 730	1	0.3	208000.0

Figure 353:

8. Click **Query**.
9. Repeat this process for property and get the element thickness.

Derived data is not stored in HyperMesh data. It is calculated by a procedure predefined by the Matrix Browser to calculate useful data from HyperMesh data. You can create these Tcl procedures and store them in the Matrix start up. They will show up in the browser as derived_datanames. The pre-installed derived_datanames are width, height, radius and adjacent.

Access HyperView Results From the HyperMesh Database

Access HyperView Results From the HyperMesh Database with Multiple Subcases

1. In HyperWorks Desktop, split the graphics area into two windows. In the first screen, use HyperMesh to load the model files. In the second screen, use HyperView to load results files.
2. From the menu bar, click **Tools** > **Matrix Browser**.
   The Matrix Browser opens.
3. In the DataSource column, select **HMdata**.
4. In the Entities column, select the required element directly or select components, material, property, or sets.
5. In the DataSource column, select **HVdata**.
6. In the Entities column, select **Results**.
7. In the Subcase Options column, select **multiple_subcases**.

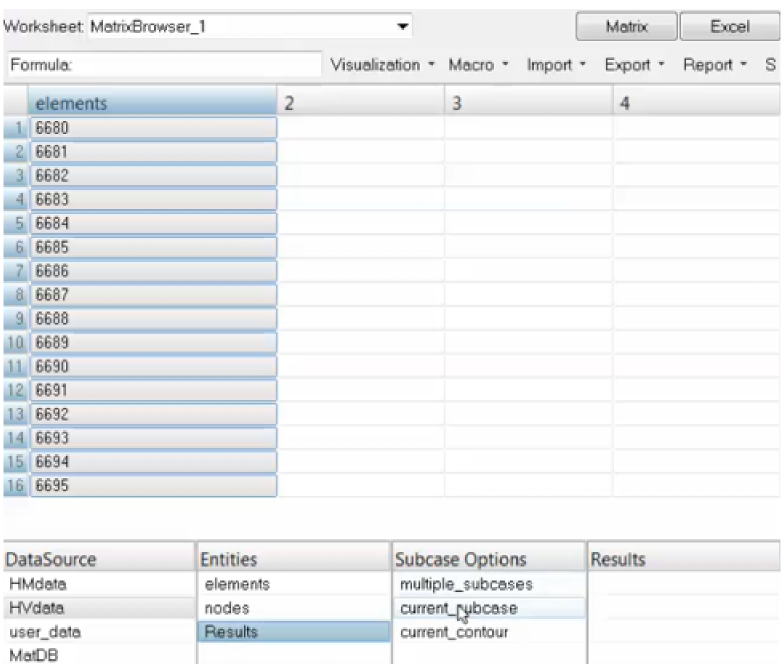


Figure 354:

8. Select the results type(s) available in the solver results.

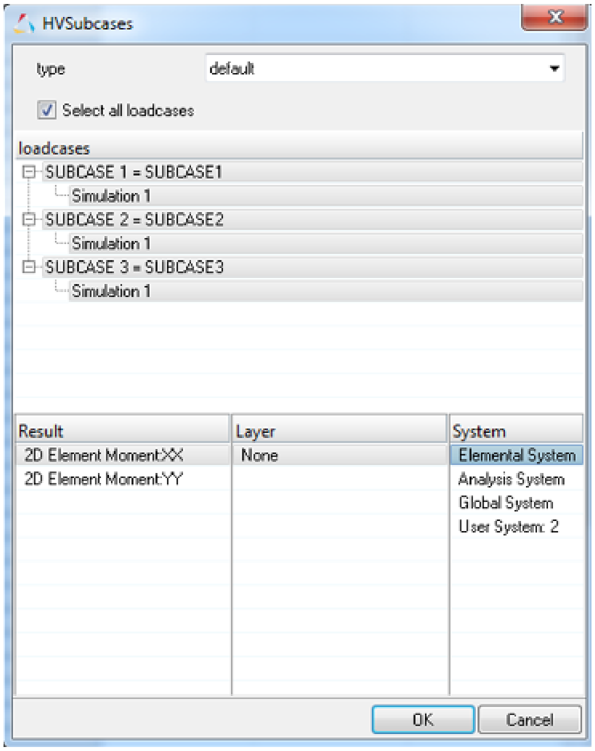


Figure 355:

9. Select layer information (if available) and the results system in which the results are to be interpreted.

   User-defined system results are available as long as a system is available in HyperView.
10. By default, the matrix only displays the results for each subcase in separate column. Retrieve additional information by right-clicking and selecting **Show** all columns from the context menu.

Access HyperView Results From the HyperMesh Database with the Current Subcase

Current_subcase selection is the same as multiple_subcase, except that it provides the ability to create multiple worksheets when you run the macros.

1. In HyperMesh Desktop, split the graphics area into two windows. In the first screen, use HyperMesh to load the model files. In the second screen, use HyperView to load results files.
2. From the menu bar, click **Tools** > **Matrix Browser**.
   The Matrix Browser opens.
3. In the DataSource column, select **HMdata**.
4. In the Entities column, select the required element directly or select components, material, property, or sets.
5. In the DataSource column, select **HVdata**.
6. In the Entities column, select **Results**.
7. In the Subcase Options column, select **current_subcase**.

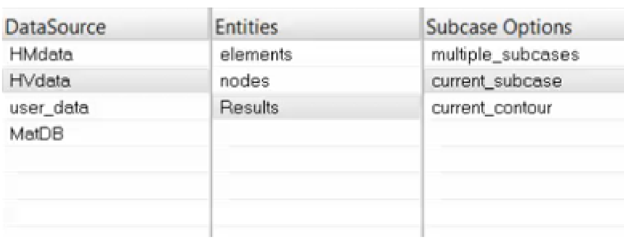


Figure 356:

8. Select the results type.
9. In the **HVSubcases** dialog, select layer and system information.
   Results for the current subcase display in the column.

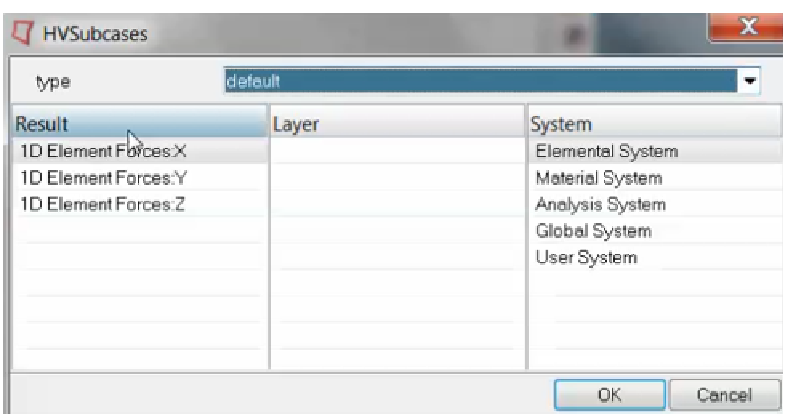


Figure 357:

10. From the Macro pull-down, click **Save**.

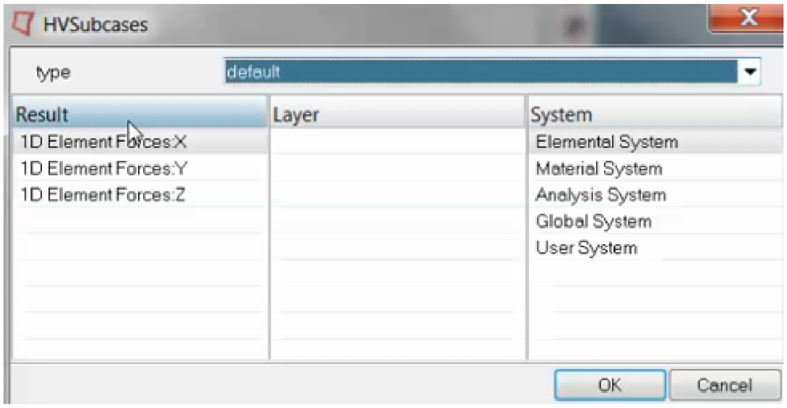


Figure 358:

11. From the Macro pull-down, click **Run**.

12. After the macro is finished running, you will be asked to select the load case. If you select multiple load cases, separate worksheets will be created for each load case as in the case of current load case.

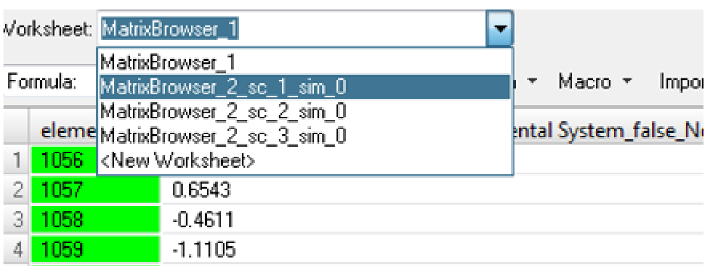


Figure 359:

13. Select the worksheet corresponding to the simulation.

Access HyperView Results From the HyperMesh Database with the Current Contour Results

1. In HyperView, select the required results and contour data.

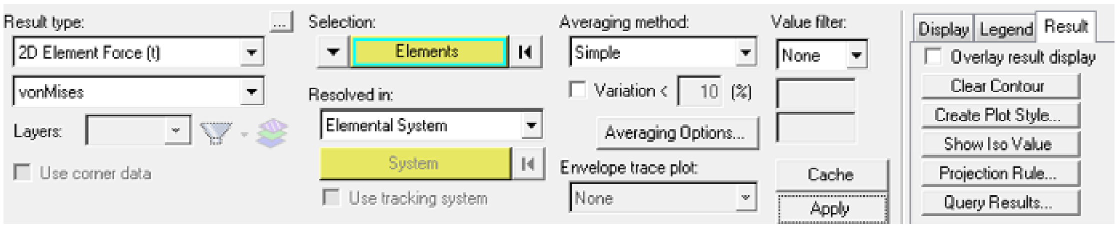


Figure 360:

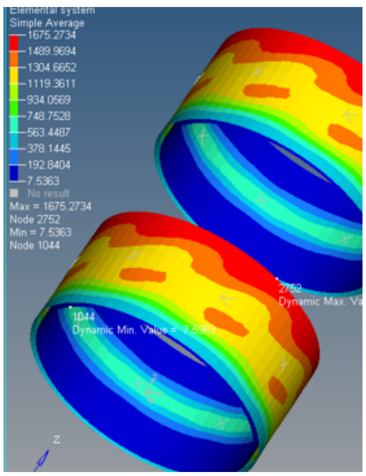


Figure 361:

2. From the menu bar, click **Tools** > **Matrix Browser**.
   The Matrix Browser opens.
3. In the DataSource column, select **HVdata**.
4. In the Entities column, select **Results**.
5. In the Subcase Options column, select **current_contour**.

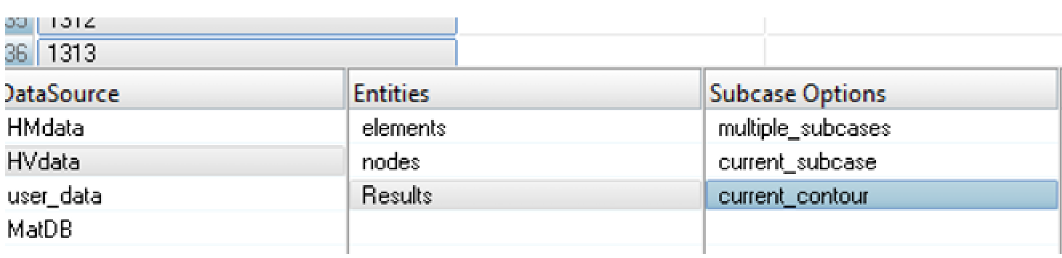


Figure 362:

6. The current contour selection does not prompt you for load case, layer, or system information. The displayed results are queried in HyperView and sent to the matrix.

Query Max/Min Results for Components/Sets/Materials/Property

When you query max/min results for component/sets/material /property, HyperMesh will search the max/min values for the elements in that component and provide single values and the element ID/load case ID where that values occurs.

1. In the DataSource column, select **HMdata**.
2. n the Entities column, select components, sets, materials, or property.

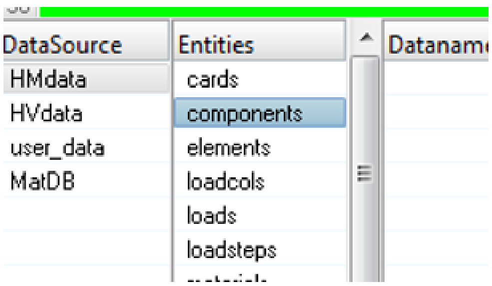


Figure 363:

3. In the DataSource column, select **HVdata**.
4. Select results and the required data type.
5. In the **HVSubcases** dialog, select layer, corner data type, system, and averaging methods.

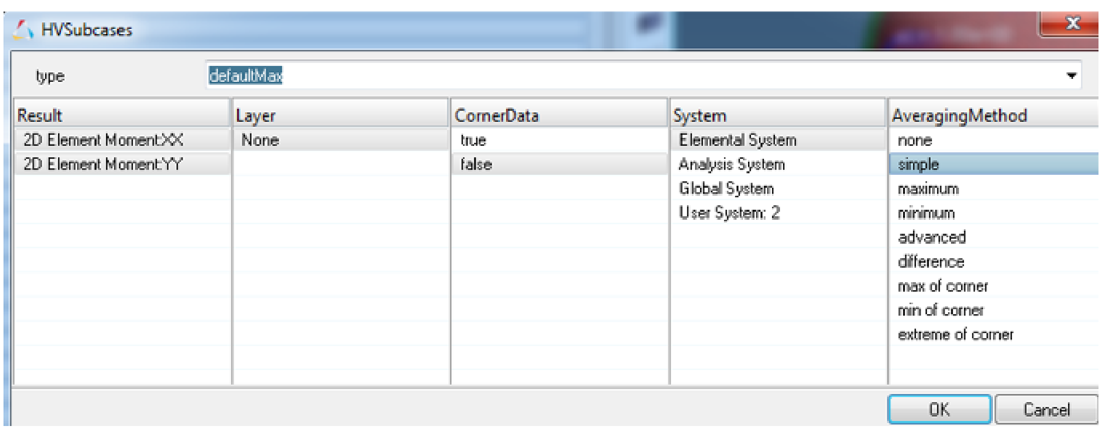


Figure 364:

The max/min results for that component display. In a separate column, additional data lines display the loads case and node/element information for the particular data.

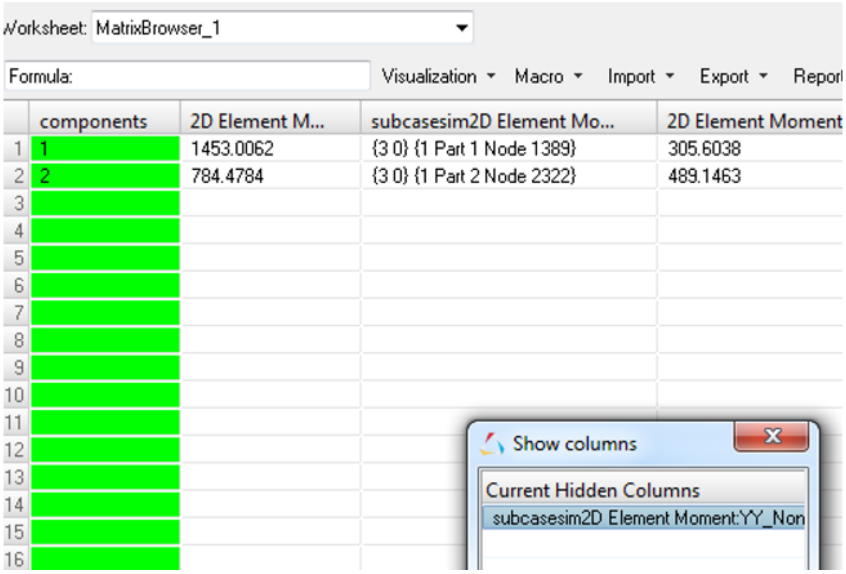


Figure 365:

Create Notes in HyperView

In the Matrix Browser, you can create and delete annotations (HV-notes) for selected column results using the right-click context menu option Notes to HV.

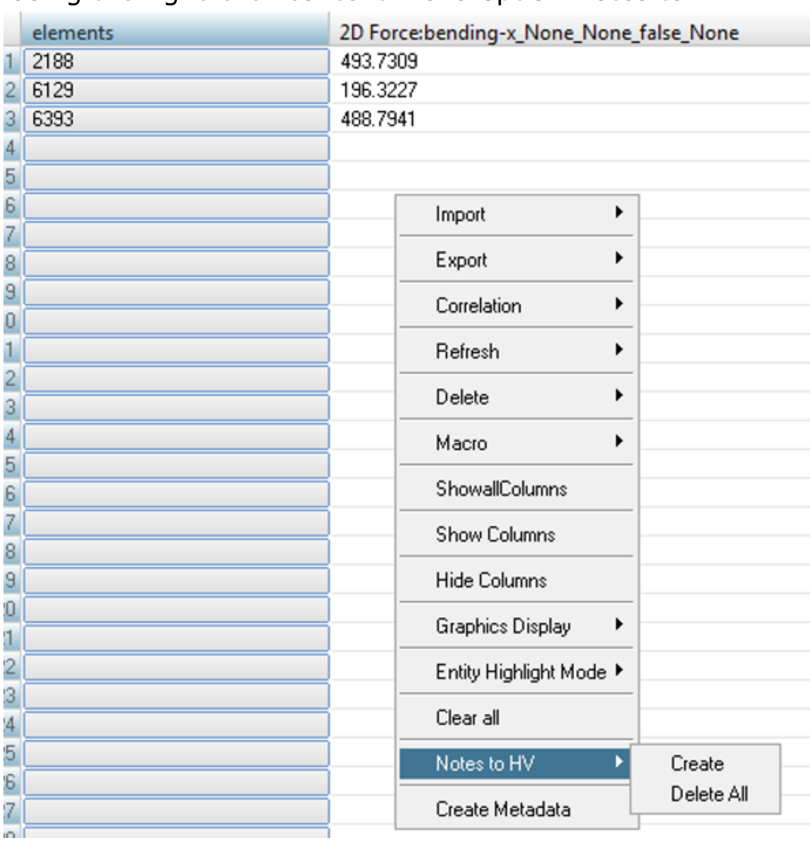


Figure 366:

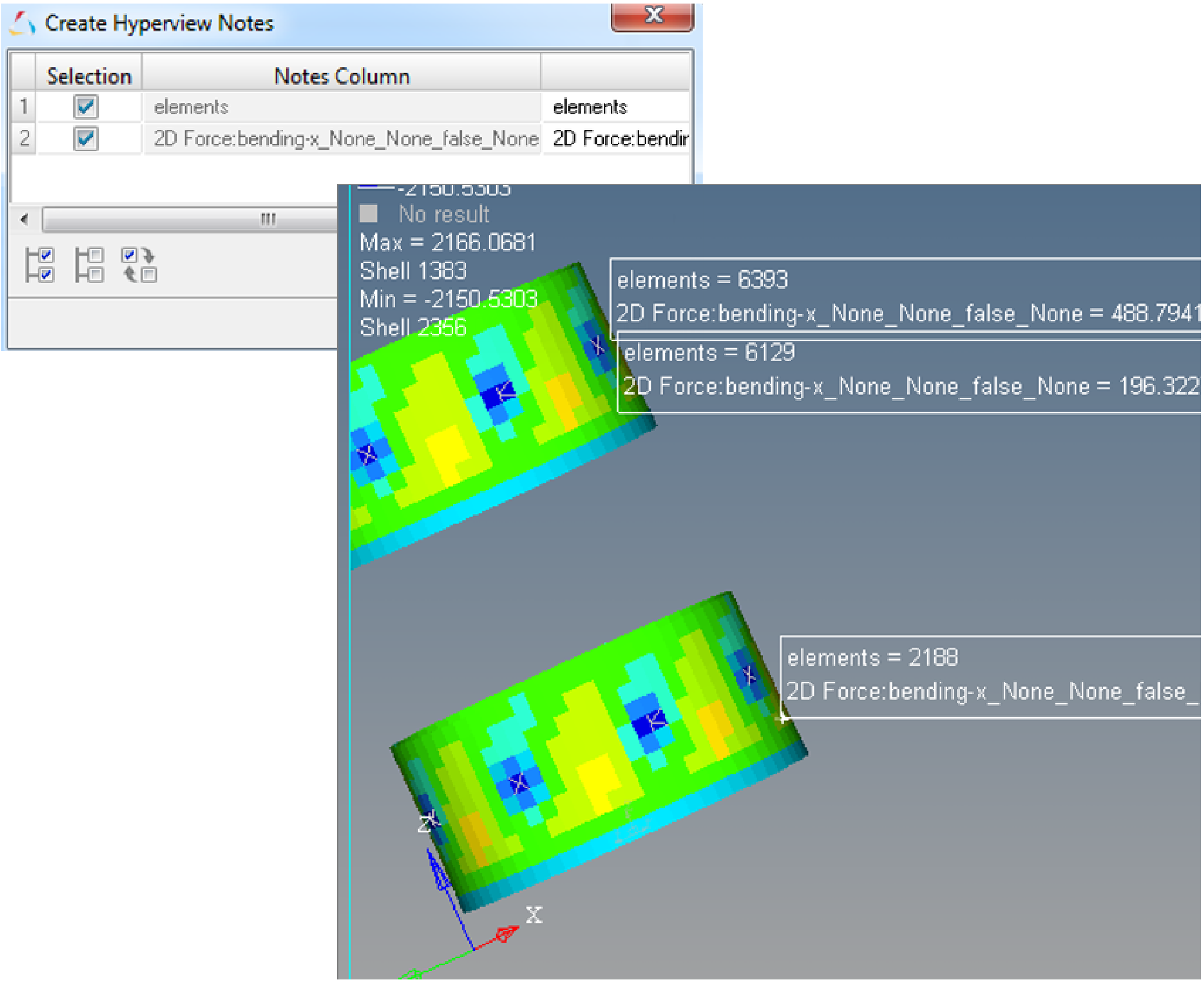


Figure 367:

Create and Retrieve User Data in the Matrix Browser

It is often useful to add additional data that is not in the HyperMesh or HyperView databases, which is known as user data.

1. Click **user_data** in the Data Source column.
2. Click **Create** in the user variables GUI column.
3. Click **double** in the variable types column.
4. Click **Query**.
5. In the dialog, provide the column label name, select **Multiple values**, and activate the **Create a column on OK** checkbox.

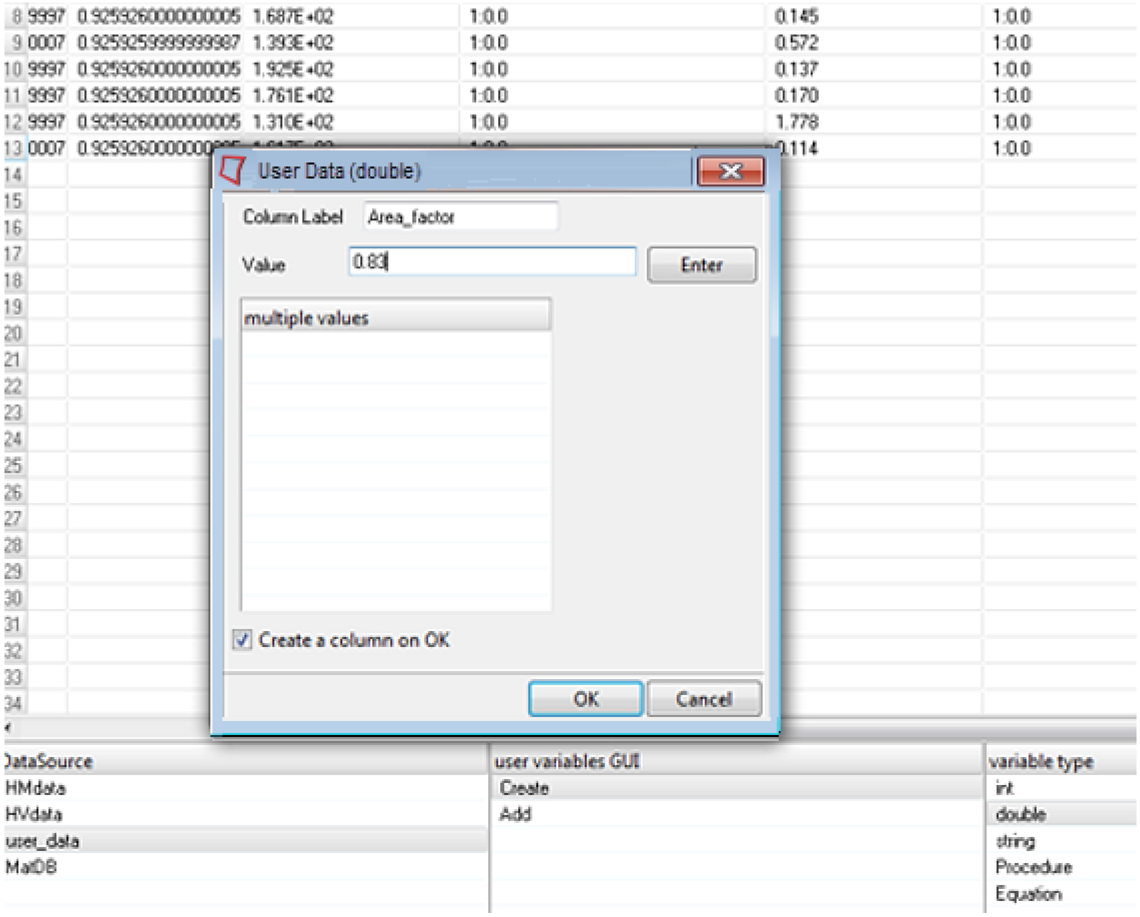


Figure 368:

6. Click **OK**.
   The user data is added to the Matrix Browser.
7. In the matrix, right-click on the user data you created and select **Create Metadata** from the context menu.
8. In the dialog, select entities to create metadata for.

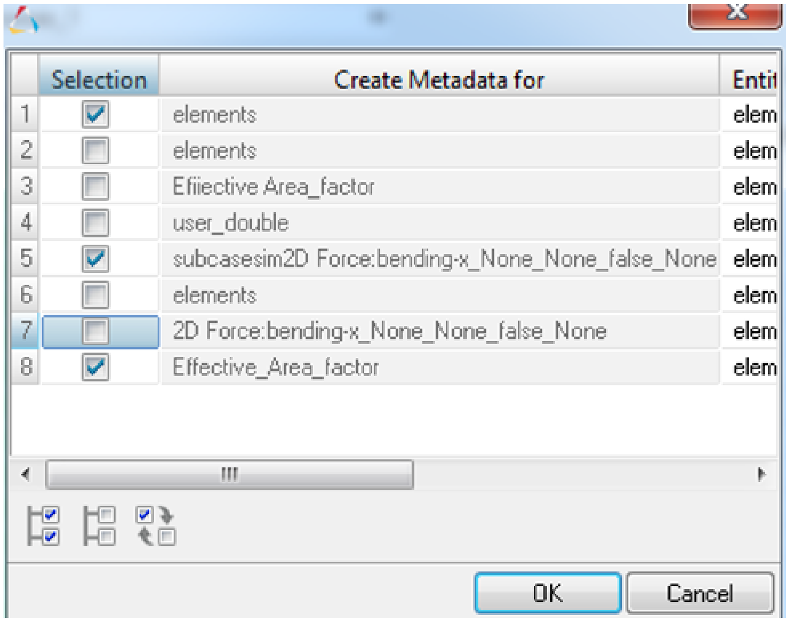


Figure 369:

9. Retrieve this data using entity based metadata.

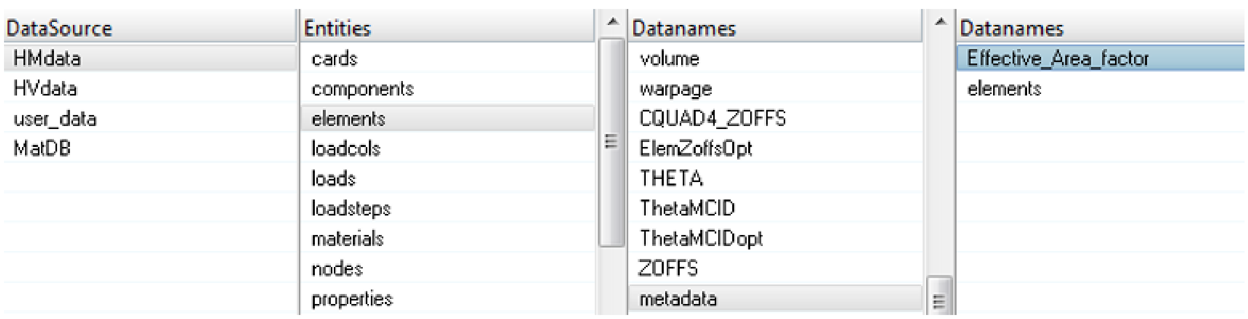


Figure 370:

Work with Microsoft Excel

1. Once the HyperMesh, HyperView and user data is gathered in the Matrix Browser, the data can be exported to Excel by clicking **Excel** in the top right-hand corner of the browser.
2. You can hide some of the columns that are not needed for export by right-clicking and selecting **Hide Columns** from the context menu.

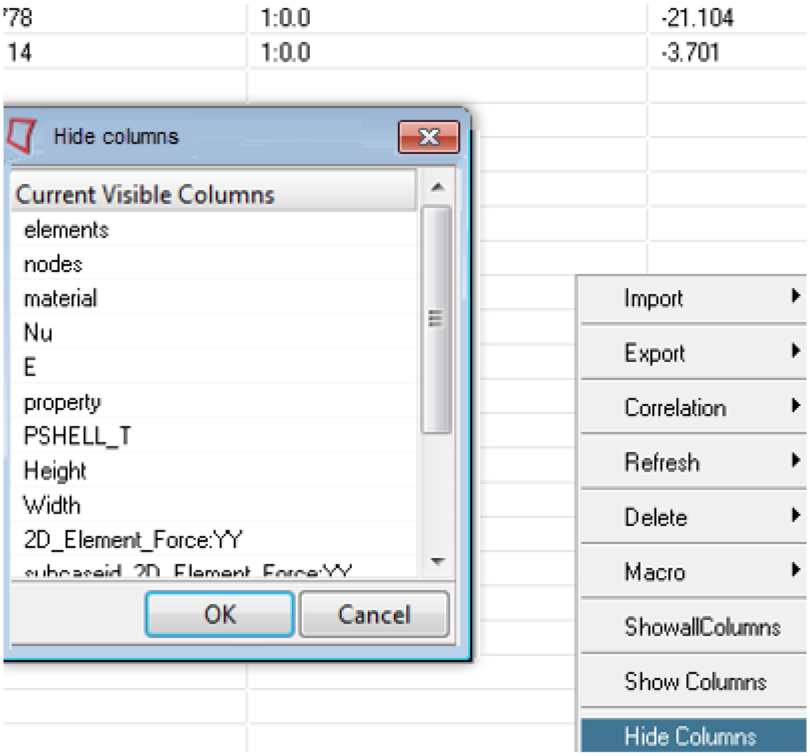


Figure 371:

3. You can also add more columns. To import the new column(s) from Excel into the Matrix Browser, click **Matrix**.

Display Data in HyperMesh or HyperView

1. Display data in HyperMesh.
   a) Click **Correlations** > **Contour**.
   A dialog opens.
   b) Select **elements** for the Entity Column label and **Net_area** for the Column on Y field.
   c) Click **OK**.
   The contour appears in HyperMesh.
2. Display data in HyperView.
   a) Click **Export** > **HyperView**.
   b) Select a column to export.

   The exported data will appear in HyperView. You can post-process this data in the same manner as other HyperView data types.

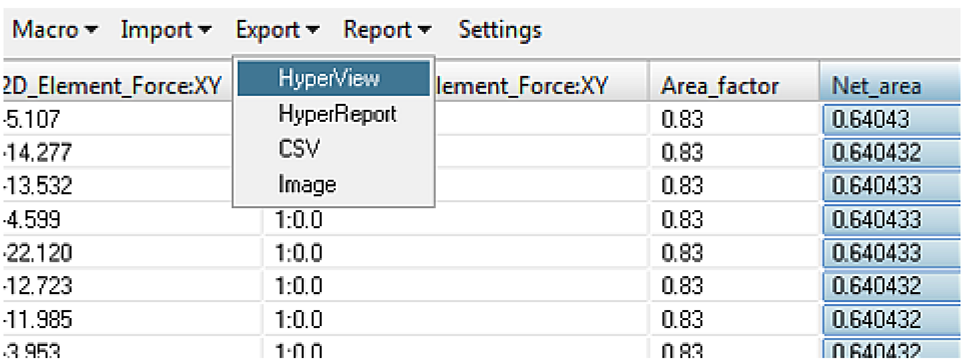


Figure 372:

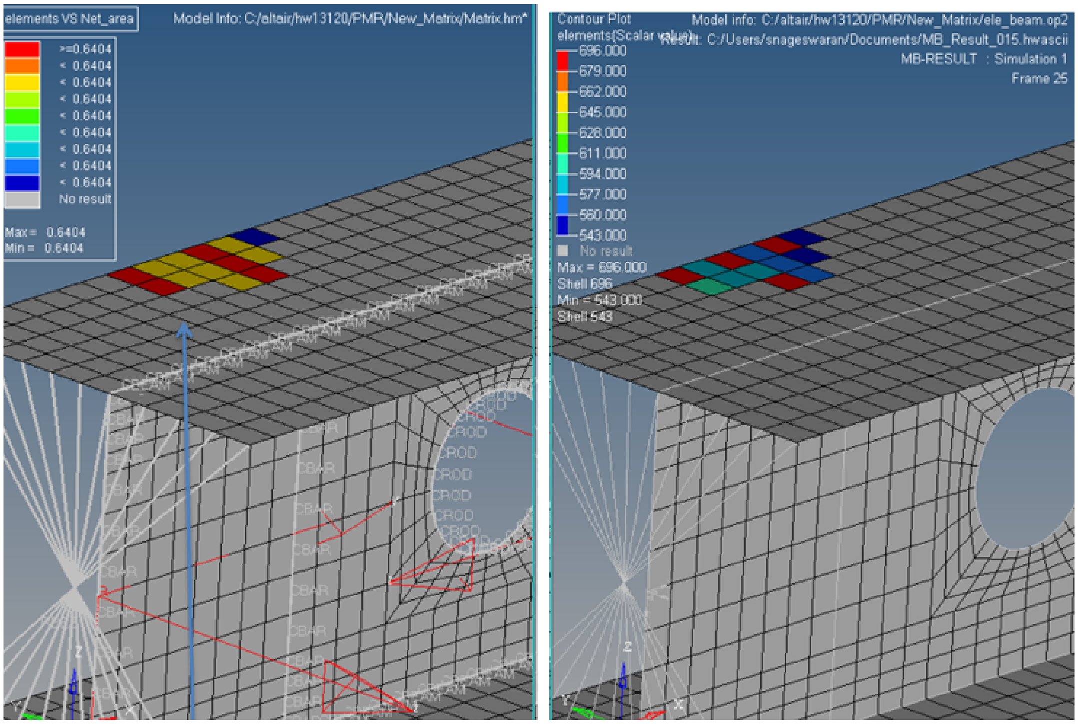


Figure 373: Contourin in both HyperMesh and HyperView.

Add TCL Calculation Procedure

HyperMesh has a built in scripting language based on Tcl programming language. Tcl is used to access HyperMesh and HyperView functionalities\methods as well as internal data. The following example calculates element strain (yy) from element 2D-Force-YY for a shell element.

1. Click **user_data** in the Data Source column.
2. Click **Create** in the user variables GUI column.
3. Click **Procedure** in the variable type column.
   A dialog opens.
4. Enter the column and function name.
5. Activate the **Create a column on OK** checkbox.
6. Add the following Tcl code, based on Matrix and column names, then click **OK**.

```
proc Element_strain_calc  { } {
  set force_fyy[::MatrixBrowser::getColumn2D_Element_Force:YY]
  set width [::MatrixBrowser::getColumn Width]
  set young_mod [::MatrixBrowser::getColumn E]
  set thickness [::MatrixBrowser::getColumn PSHELL_T]
 set Strain_yy ""
 set Strain_yy [expr $force_fyy/$width/$thickness/$young_mod]
 return $Strain_yy
}
```

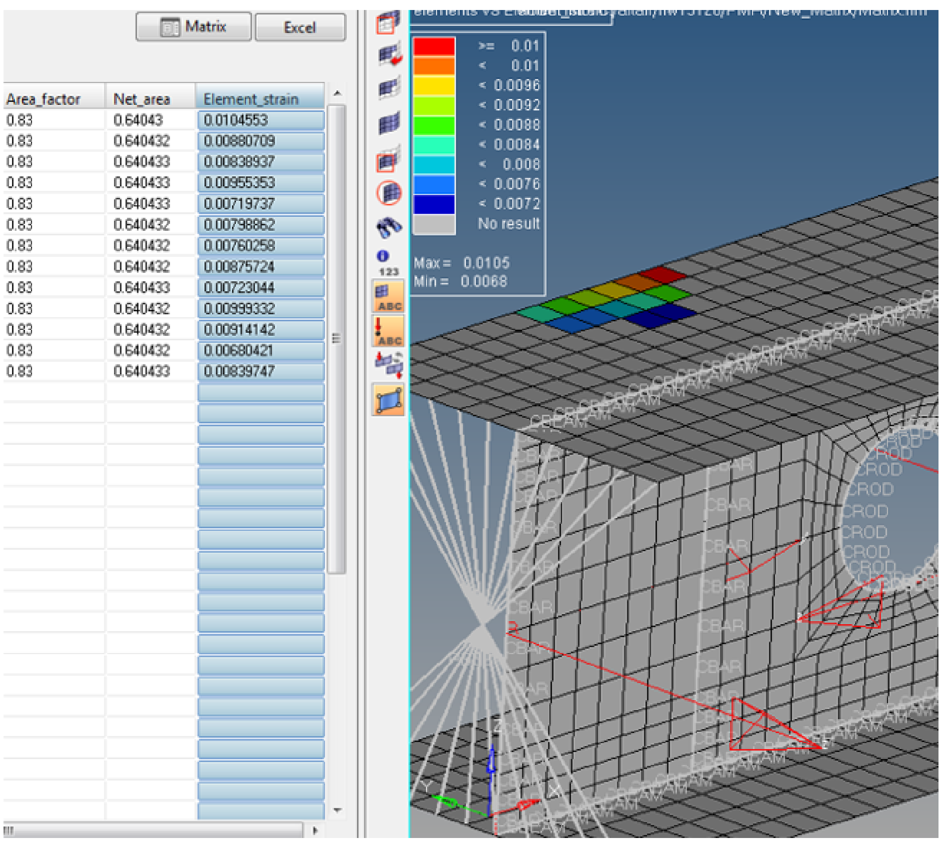


Figure 374: Element_strains are calculated and displayed.

Store and Reuse the Procedures Outside of the Matrix Browser

The process developed in the previous section can be stored as a macro (script) and can be reused with another HyperMesh data model or other HyperView databases.

1. Click **Macro** > **Save**.
2. Enter the Macro Name in the dialog and click **OK**.

   This macro can also be added to the menu bar and accessed and run without using the Matrix Browser. The scripts are stored and can be reused like other HyperWorks scripts without the Matrix tool.

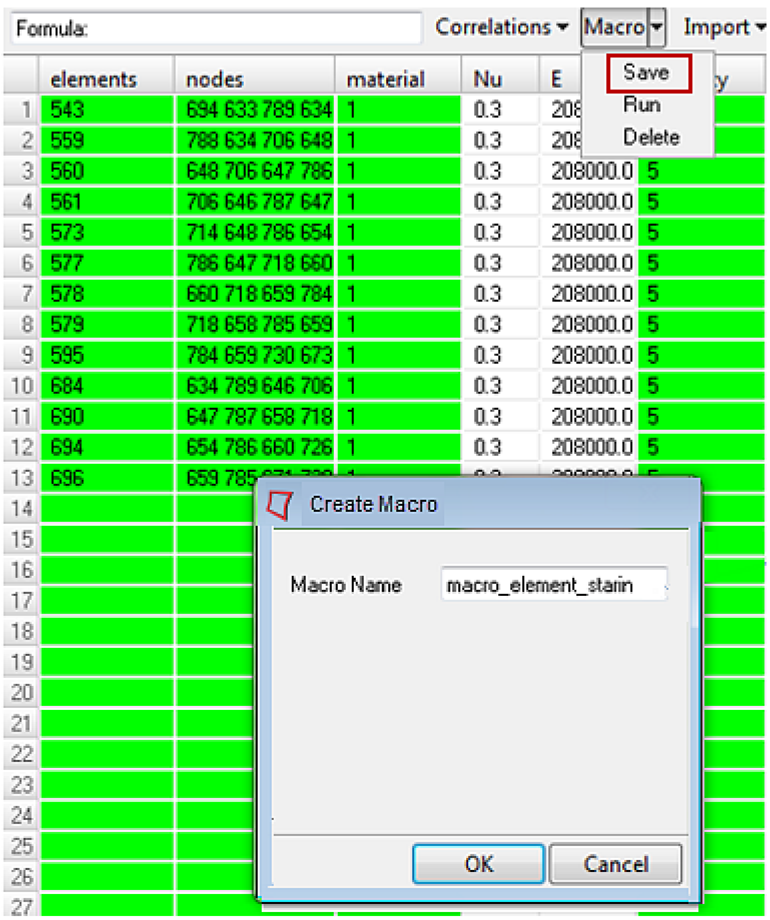


Figure 375:

Context Menu

The Matrix Browser's right-click context menu contains additional browser options.

Option	Description
Import	Import Matrix or CSV.
Export	Export HyperView or CSV.
Correlation	Select Contour, Shape, Vector, or Plot.
Refresh	Refresh the Matrix Browser or HyperMesh session.
Delete	Delete selected rows, columns, or the whole table.
Macro	Save or run the macro.
Show all Columns	
Show Columns	
Hide Columns	Hide some of the columns that are not needed for export.
Entity Highlight	Select Show all, Show, Hide, Isolate, or Isolate Only.
Clear all	Clear the table.
Notes to HV	Create or delete notes.
Create Metadata	Create and save metadata for user data selected in the matrix.

Model Browser

Use the Model Browser to view the model structure while providing full find, display and editing control of entities.

The model structure is viewed as a flat, listed tree structure within the browser. However, if the model has an assembly hierarchy then the Model Browser accommodates this hierarchical structure. The browser can list every named entity within the session and places those entities into their respective folders; however, it does not support non-named entities such as nodes and elements. Some of the more important entities within the model include: assemblies, components, multibodies, properties, materials, entity sets, groups, load collectors, system collectors, vector collectors, and beamsectcols, all of which are placed into a tree-like display.

To open the Model Browser, click **View** > **Model Browser** from the menu bar. The browser displays on one of the tab area sidebars.

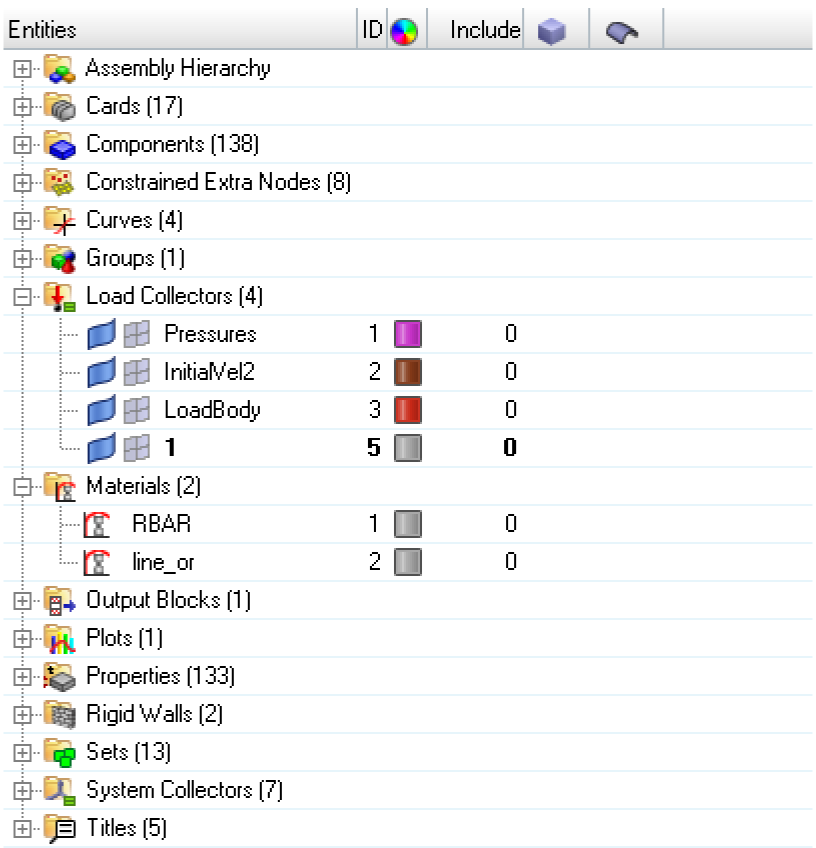


Figure 376: Entities Organized in the Model Browser

Multiple entities of the same type are collected into folders in the tree structure. You can expand or collapse each folder to display or hide its contents. Assemblies can also have sub-folders within the main Assembly folder, so that the items related to each assembly appear within that assembly's folder in the Assembly Hierarchy. Materials, properties, entity sets, groups, load cols, system cols, vector cols, and beamsectcols cannot be organized into assemblies and are all placed at the top level of the tree, each in their corresponding folder. For example, all sets are placed as a flat list in the Sets folder.

Components and Assemblies may appear in multiple places in the tree; for example, a specific component might appear under Components and again as a sub-item of a specific Assembly. When appropriate, the color and display style of entities also display in the Model Browser.

The Model Browser tools include:

- Toolbars provide the ability to change model views, show or hide entities within the model, and add entities to a panel collector. These abilities are collectively referred to as display controls and browser modes.
- The context menu includes most of the same functions as the toolbars, however only relevant context menus will appear based on entity(s), folder(s) or white space right-mouse click.
- You can sort, find, and filter entities in the Model Browser's tree list using column header sort controls and the query builder.
- The tree list within the browser is configurable, so that you can determine which columns and entity types that display in the tree.

Browser Attributes (Columns)

Each column in the browser displays a different browser attribute.

Add or remove browser attributes by right-clicking on a column header and checking/unchecking the appropriate attribute from the context menu.

Sort by a particular entity in the browser by clicking on the heading of each column. For fields that are numeric and/or alphanumeric, repeated clicks toggles between ascending and descending order.

Column	Description
Entity	Lists every named entity within the session; however, it does not support non-named entities such as nodes and elements.
ID	Unique entity ID
Color	Entity color
Include	ID of Include file the entity is stored within.
FE-Style	Lists the element style applied to each entity. Click the icon to change the element style.
Geometry Style	Lists the geometry style applied to each entity. Click the icon to change the element style.
Attributes	Append entity attributes for components, properties and materials to the browser as columns to facilitate fast and efficient review, editing, sorting and filtering in the Component, Material, and Property views. Right-click on a column heading and select **More** from the context menu to access a list of all entity attributes that can be appended to the browser.

Column	Description
	Tip: From the Entity Editor, right-click on an attribute and select **Add to Browser** from the context menu to append the attribute as a new column in the browser. Filter entity attributes in their respective columns by clicking in the column header and entering a search string. Edit the values displayed in each column by double-clicking the cell and changing the value. Create and assign parameters to attributes by right-clicking on the attribute in the browser and selecting the correct function from the context menu, such as **Create and Assign Parameters**.

Tool Button Groups

Many of the Model Browser functions are accessed via the View, Global Display, and Action Modes groups of buttons. Rest the mouse cursor over a tool set or call-out in Figure 377 to see the name, or click to jump to help for that tool set.

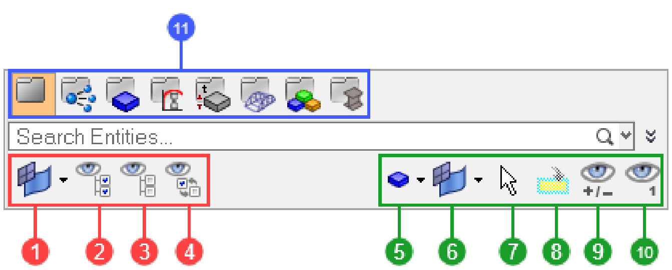


Figure 377: Model Browser Tool Button Groups

Blue call-out is the View toolbar; green is the Action Modes; and red is the Global Display.

1. elems/geom/both (filter for all/none/reverse and local display control)
2. display all
3. display none
4. display reverse
5. entity selector
6. elems/geom/both (filter for selection controls)
7. selector
8. virtual collector
9. show/hide
10. isolate
11. view modes

Drag and Drop

Components, multibodies, and assemblies can be dragged and dropped with the left and right mouse button. Use the left mouse button to move an item into another assembly; use the right mouse button to open the context-menu and select **remove** to delete an item from an assembly. If an assembly is moved, all the items in the assembly are moved to the new location. Items that are not seen in the tree due to filters are also moved. You can drag and drop multiple items at any time using the standard Shift and Control keys. When dragging-and-dropping entities, the number of entities you have selected to is displayed.

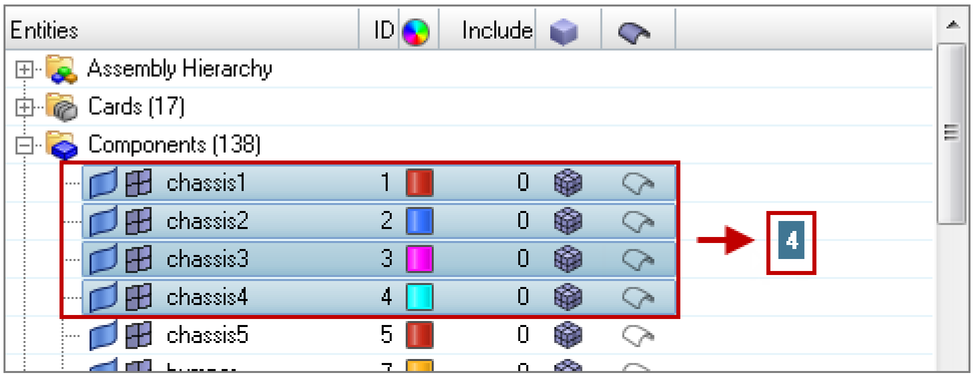


Figure 378:

> **Note:** If an item is dragged out of the tree and dropped onto empty space, it is deleted in all its parent assemblies and placed at the top level of the tree. A dragged item is added to the bottom of the list in an assembly.

Model Browser View Modes

The Model Browser consists of the following predefined browser view modes:

- Model
- Includes
- Components
- Properties
- Materials
- Optimization (OptiStruct user profile only)
- Assemblies
- HyperBeam

The different view modes are located within the first row of icons in the Model Browser.

Figure 379: Model Browser View Modes in OptiStruct User Profile

Quickly invoke a view mode by double-clicking on the entity folder or an entity in the browser. For example, double-clicking the Components folder or a component entity invokes the Components view. Double-clicking on the Beam Section Collectors folder opens the HyperBeam view.

> **Note:** The Optimization and HyperBeam views can only be invoked by clicking their respective view mode icon.

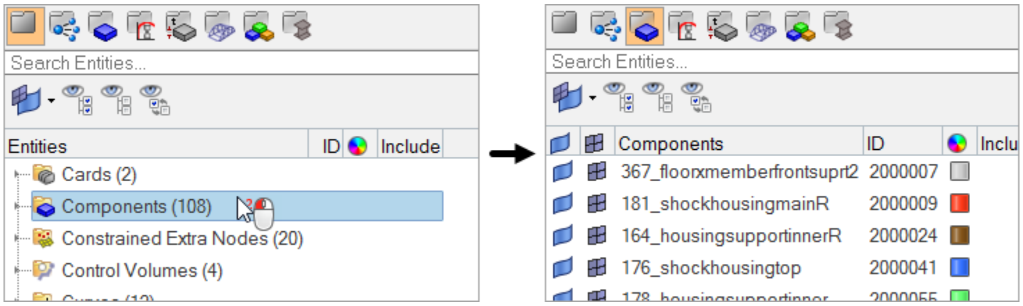


Figure 380:

Select the different view modes to quickly display specific entities in the Model Browser and graphics area. After you select a view mode, HyperMesh provides additional information associated with the entity in the Model Browser. Use the Optimization view mode to not only control the display, but to also create optimization problem definitions.

You can use the browser view modes in conjunction with the selector mechanism to easily find and query entities.

Model View

This is the standard view mode for the Model Browser. All entities within the session will be listed in the tree.

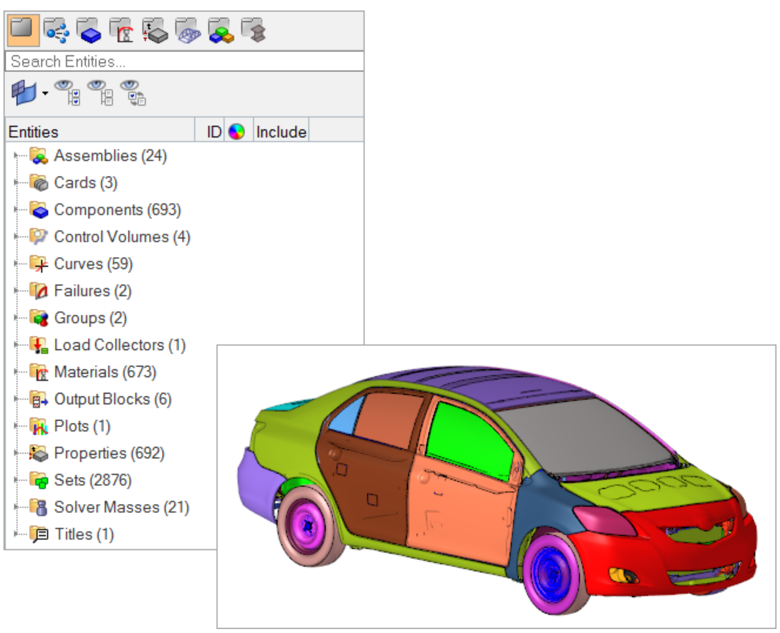


Figure 381:

Includes View

Lists all Include files in the model. The contents of each Include file is organized (grouped) into folders containing each entity type. Each of the folders can be expanded to review the individual entities in that folder. For more details, see Model Browser Include View.

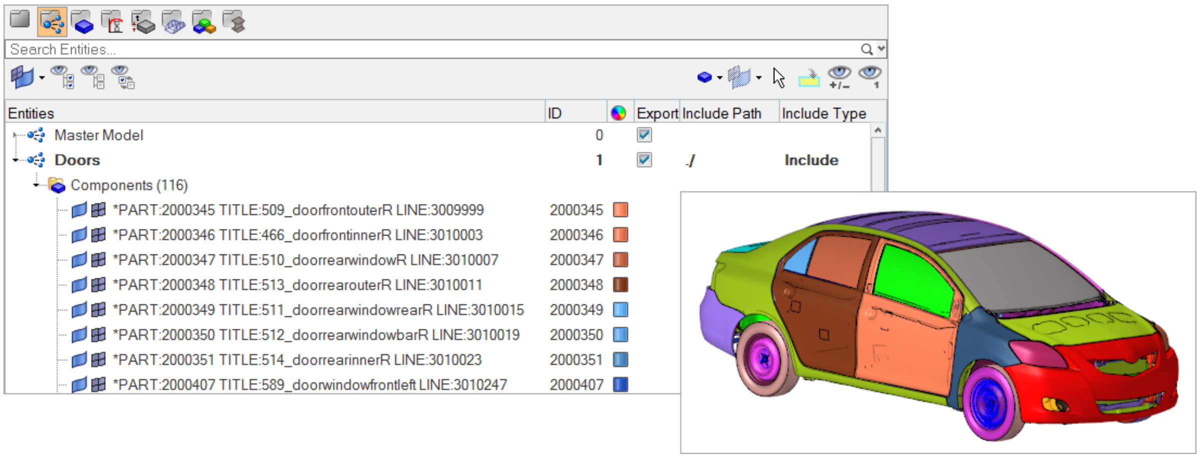


Figure 382:

Components View

- Lists only components in a flat list.
- Turns on the FE and Geometry style columns.
- Populates the Indirect Property and Material columns (dependant on user profile).
- Includes the Direct Property column, from which you can toggle between direct/indirect property assignment.
- Visualization mode is set to By Comp

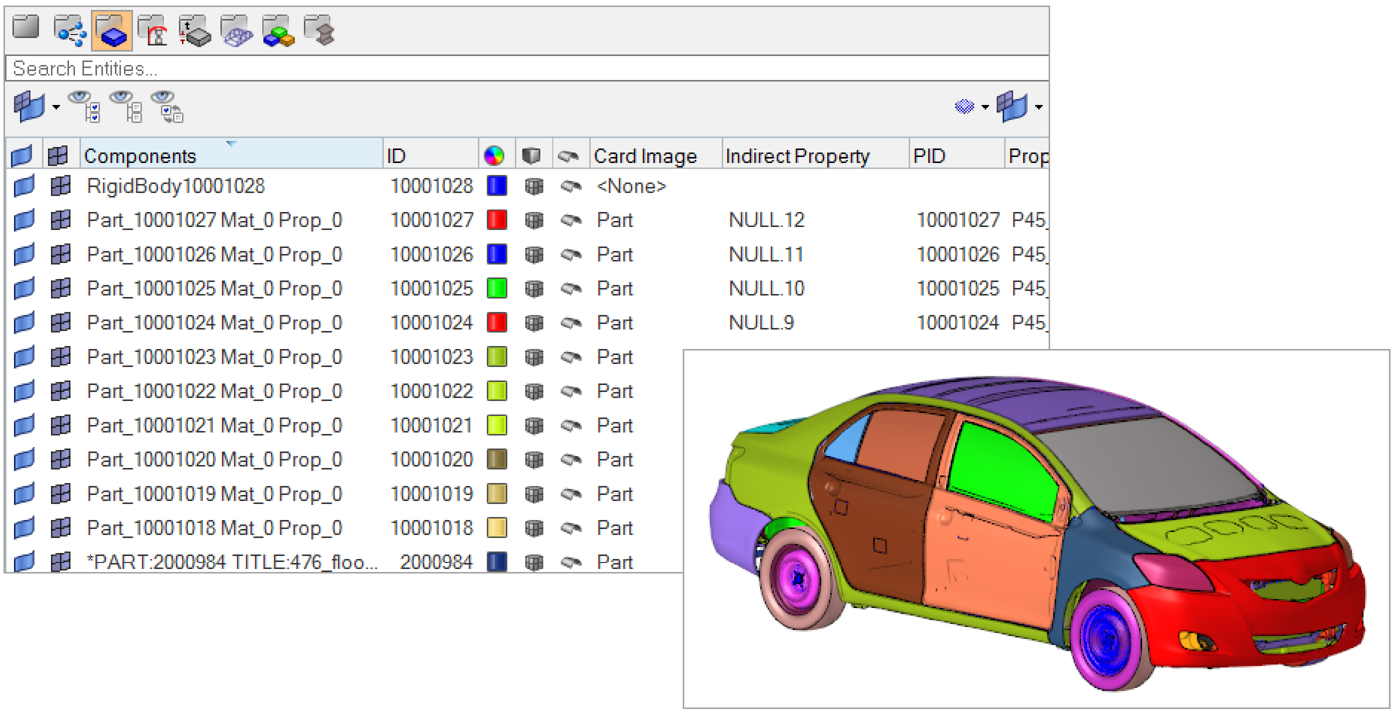


Figure 383:

Materials View

- Lists only materials in a flat list.
- Visualization mode is set to By Mat.

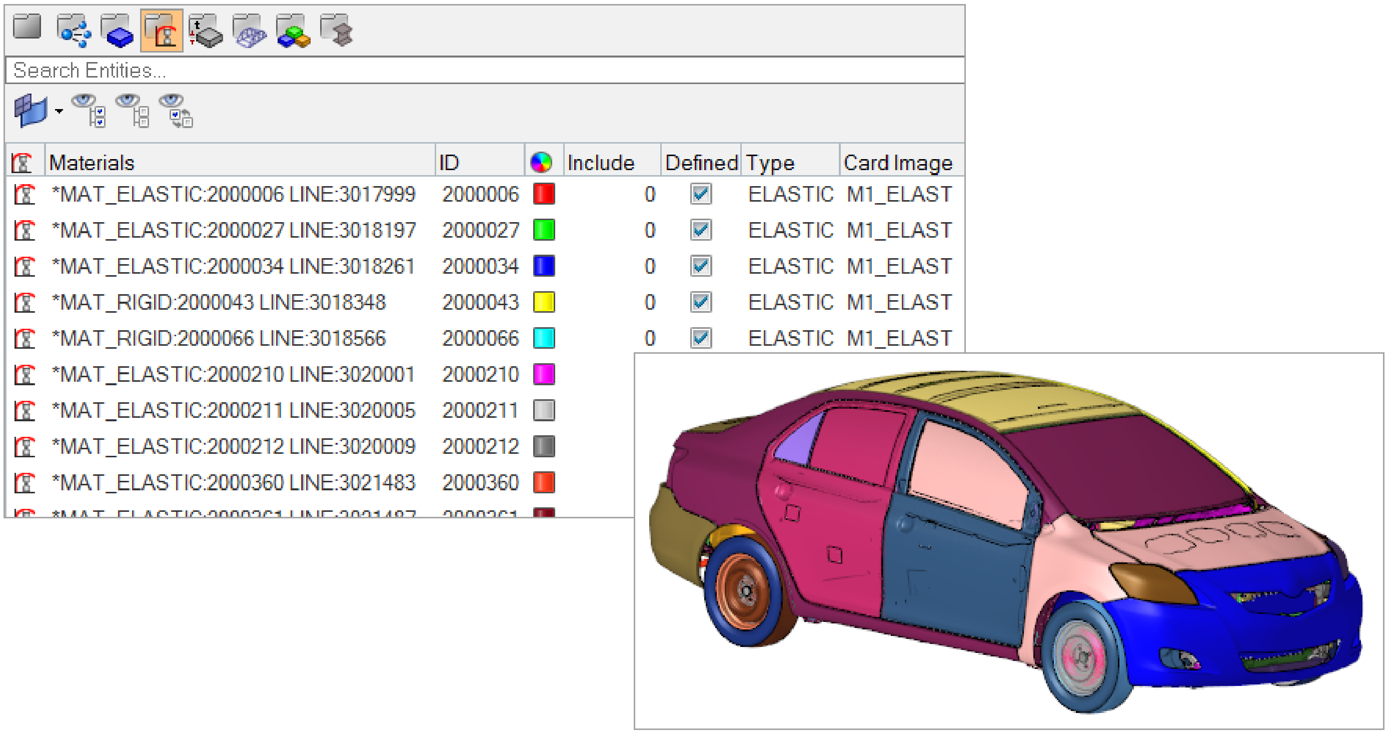


Figure 384:

Properties View

- Lists only properties in a flat list.
- Type and Card Image columns are turned on.
- Visualization mode is set to By Prop.
- A new button for element visualization by direct or indirect property is added.

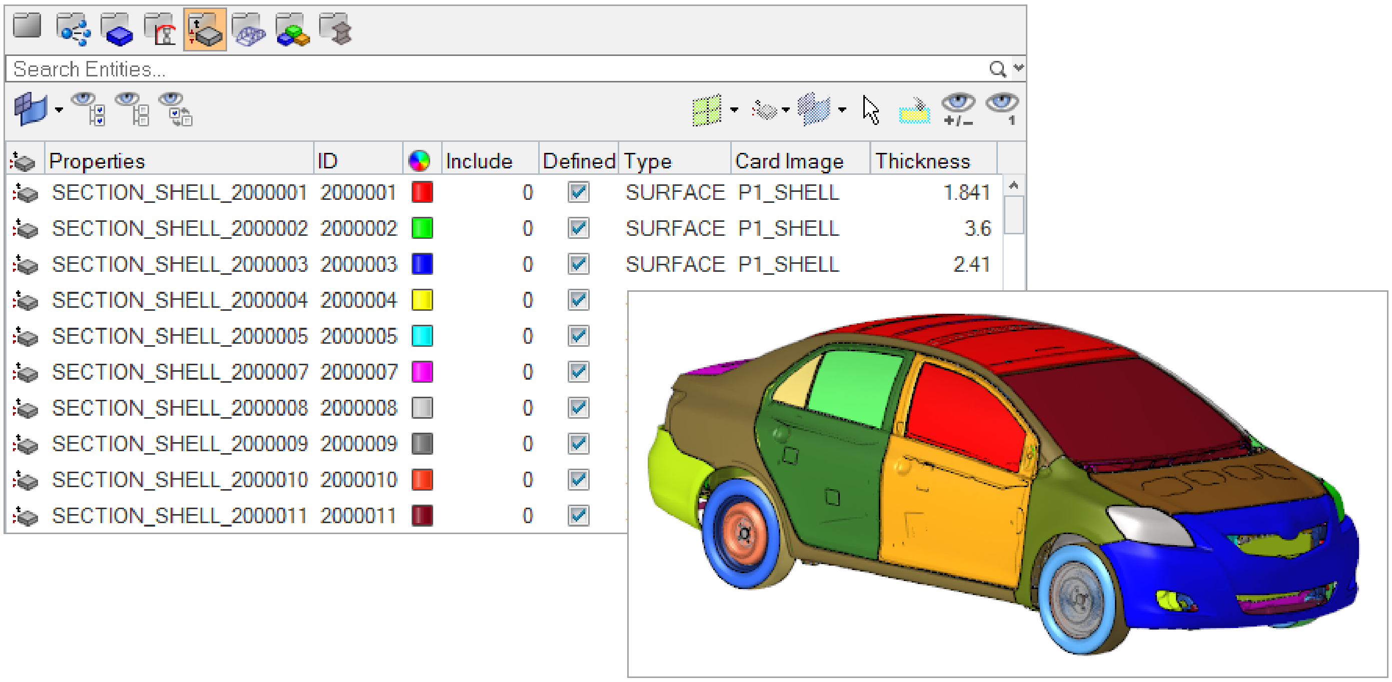


Figure 385:

Optimization View

- The optimization view is only available when the OptiStruct user profile is set.
- Lists only optimization related entities.
- Visualization mode is set to By Comp.

For more details, see Model Browser Optimization View.

The optimization view can be used to define optimization problems and objectives.

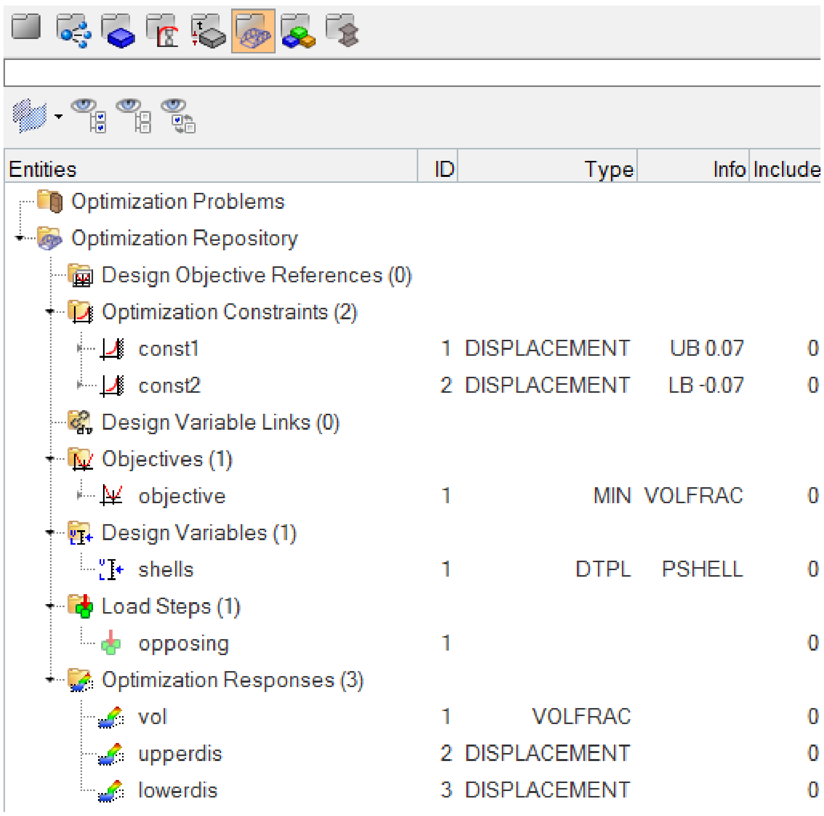


Figure 386:

Assemblies View

- Lists all Assemblies in the model. Components are organized (grouped) under the Assemblies they are assigned to. Components that are not assigned to an Assembly are displayed in a flat list under the Assemblies.
- Visualization mode is set to By Assembly.

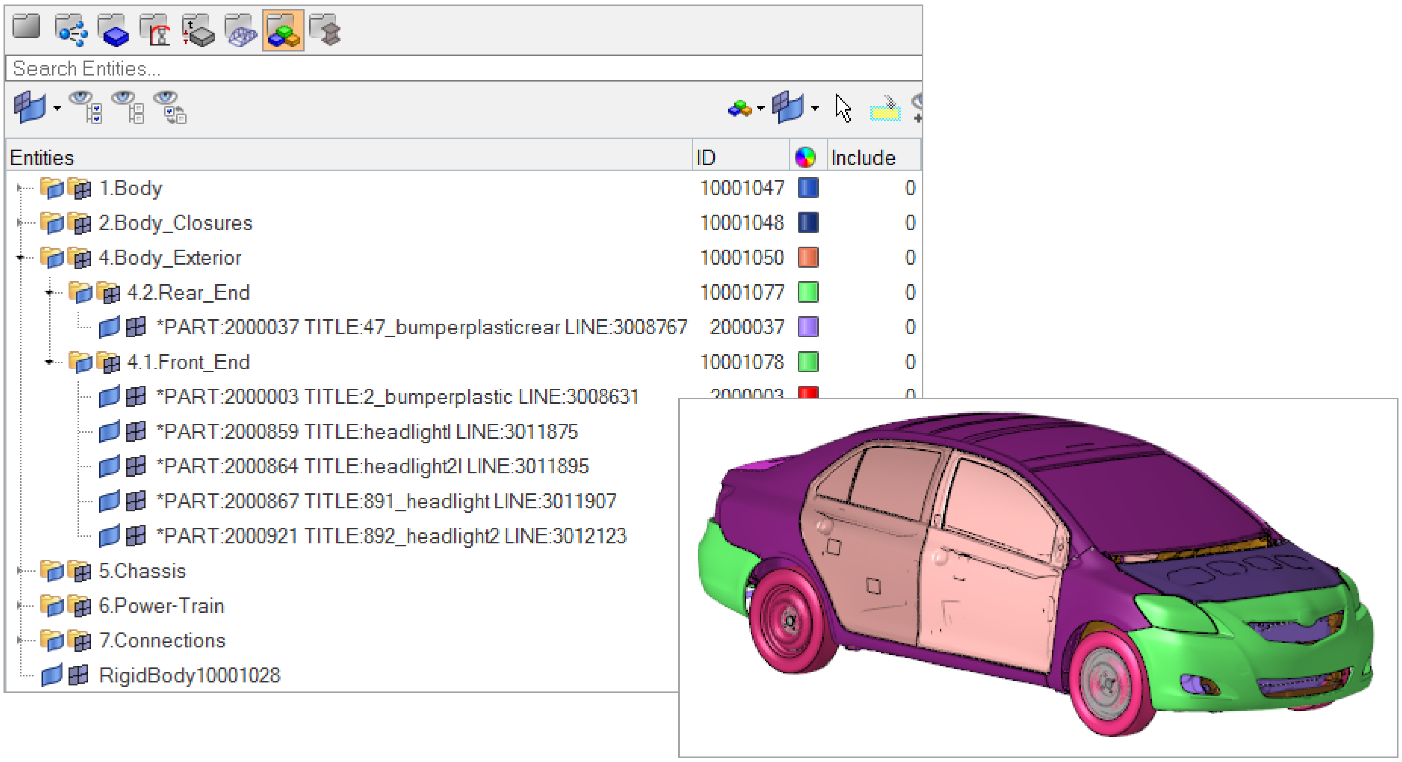


Figure 387:

HyperBeam View

Unlike other views, the HyperBeam view does not affect the overall model's display; instead, it focuses entirely on beam elements and enables the visualization, creation, and editing of beam sections and beamsection collectors. For more details, see HyperBeam View.

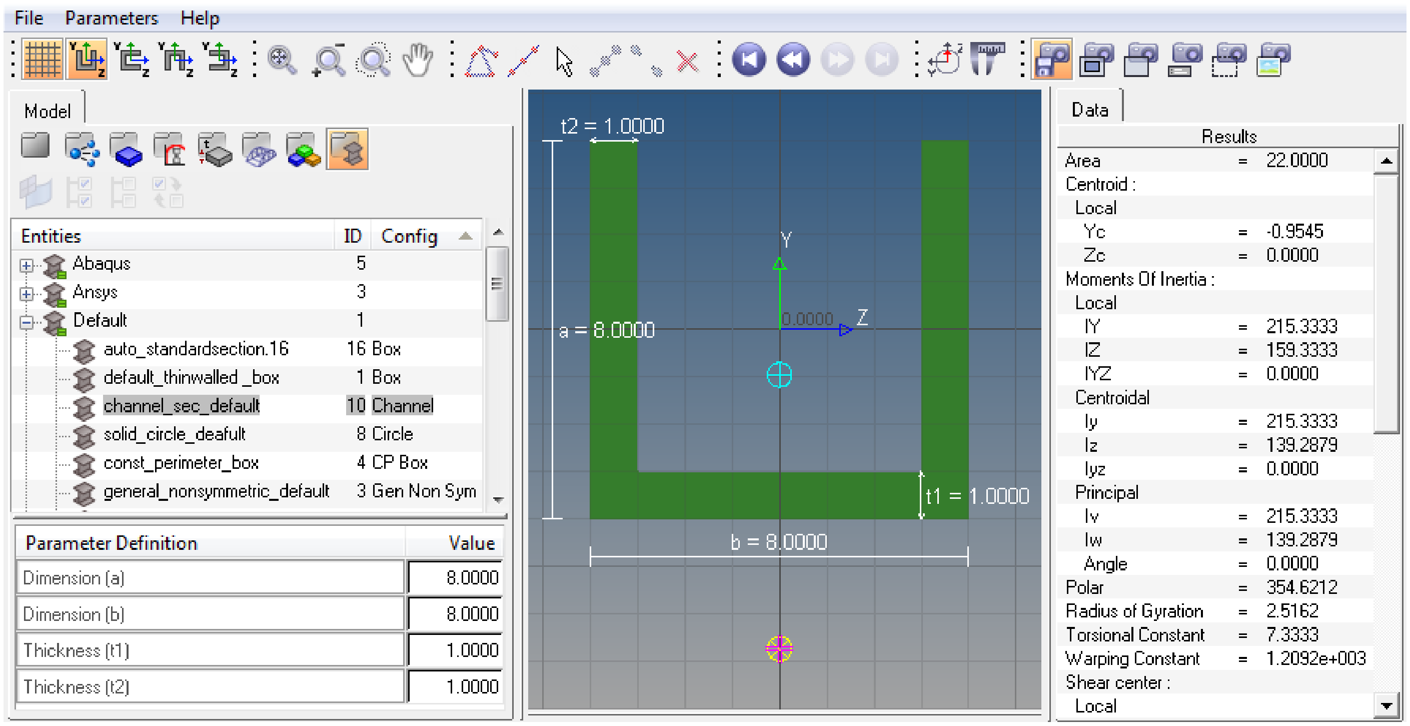


Figure 388:

Model Browser Optimization View

In the Model Browser, select the Optimization view to only display optimization related entities in the browser.

These entities include: Objectives, Objective References, Optimization Tables, Design Equations, Responses, Design Variables, Design Variable Relationships, Design Variable Links, Constraints, Loadsteps, Optimization Controls and Discrete Design Values.

The Optimization view displays two main folders: the Optimization Repository folder and the Optimization Problems folder. The Optimization Repository folder lists all the optimization related entities in the model, and the Optimization Problems folder lists all of the defined optimization problems.

To choose which problem will be included in the exported file, one (and only one) of the problems must be set to export.

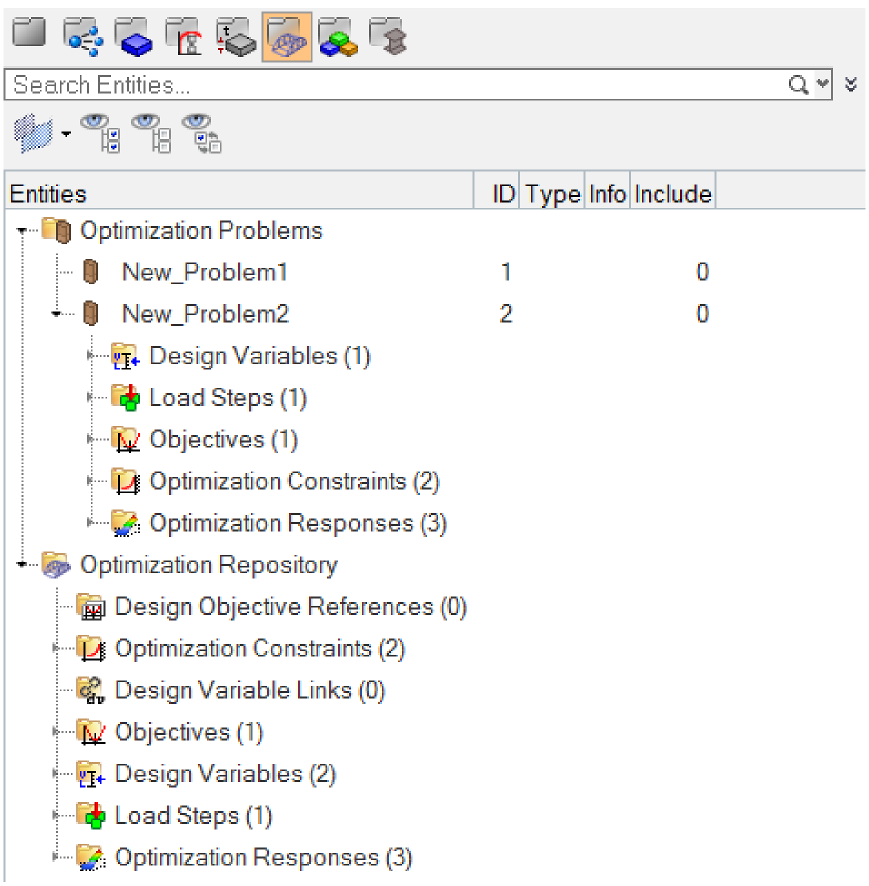


Figure 389:

Define a Problem

Use the tools in the context menu to create, delete, and rename optimization problems. To properly define a problem, you must drag and drop optimization entities into the problem. You can drag one or many entities from either the repository or a previously defined problem into a problem. There are no active problems; you must drag and drop to define problems.

Context Menu

The context menu contains tools to create, edit, and assign optimization entities in the same manner as the Optimization menu. All newly created optimization entities are placed in the repository and must be added to a problem to be considered. There is an option in the context menu to remove any optimization entities from a problem without deleting it from the repository. The delete option removes the entity from the database completely.

Export Problems

Although multiple problems can be defined with the optimization view, only one can be exported. Through the context menu, you can select which problem is set to export. The problem set to export is highlighted in bold type and gets written out to the input file. The export state can also be defined in the Entity State Browser, the export state is set by simply checking the checkbox next to the required problem in the export column. The optimization problems can be found under the Bag folder in the Entity State Browser.

Known Limitations

Only one objective and one opticontrol can be defined in one session.

Model Browser Include View

In the Model Browser, select the Include view to create, review, edit, organize and update the contents of a model into various include files.

An example of a model in the Include view is shown below.

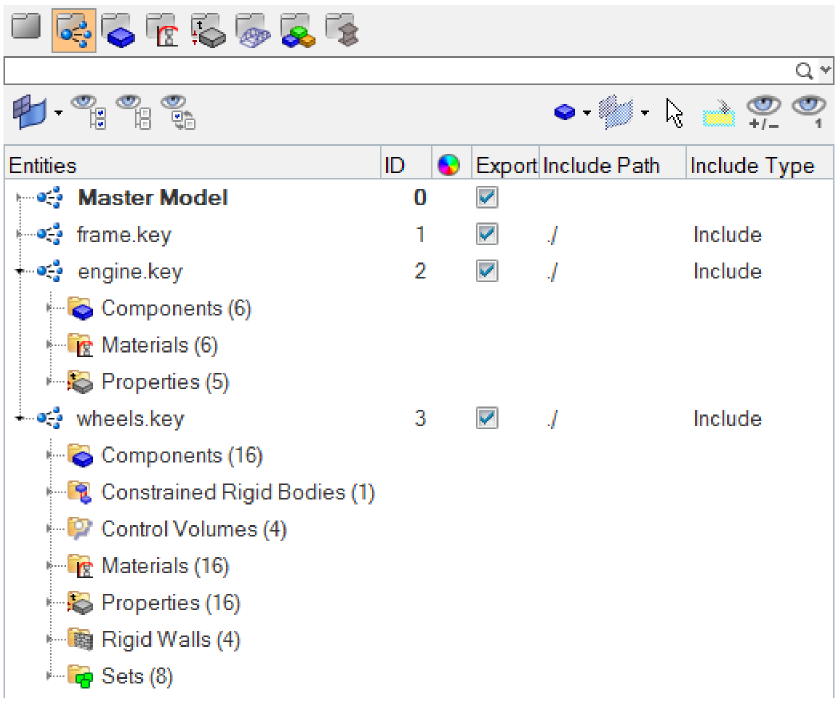


Figure 390:

The Master Model is at the top level of the Include view. Data which does not have any references to an include file is stored in the master model. Each include file is represented with an icon along with its name and internal HyperMesh ID. The attributes Export, Include Path and Include Type (if applicable, based on solver user profile) are listed in relevant columns. These extra columns can be turned on/off using column header context menu.

Export

This column has a checkbox that allows you to turn the export state of an include file on/off; each Include' s export state can be distinguished either by the checkbox status or also by a separate icon; that is the on icon () and the off icon ().

Include Path

This column allows you to define the export location for include files.

Include Type

This column lists the type of include file for applicable user profiles.

Each include can be expanded to reveal its contents. The contents of each include is organized (grouped) into folders containing each type, next to which appears the total number of entities of each type. Each of the folders can be expanded to review the individual entities in that folder. You can select entities (using the standard Shift and Control keys) and drag various entities between two includes or between the master model and an include. The browser can be configured to show only specific entities of interest.

You can drag-and-drop includes within the tree to nest them within other includes. In addition, when in Include view mode, the Model Browser context menu options Make Current and Move to Current become available when the menu is invoked by right-clicking on a valid include:

Make Current

Flags the highlighted include to be the default for subsequent Include operations such as Move to Current.

Move to Current

Organizes the highlighted include(s) to become part of the pre-designated current include. The selected includes are removed from their original location and added to the current one.

HyperBeam View

Use the HyperBeam View to create and control beamsection data.

HyperBeam view is activated by clicking  in the Model Browser. To exit the HyperBeam View and return to HyperMesh, click another view in the Model Browser, or click **File** > **Exit** from the menu bar.

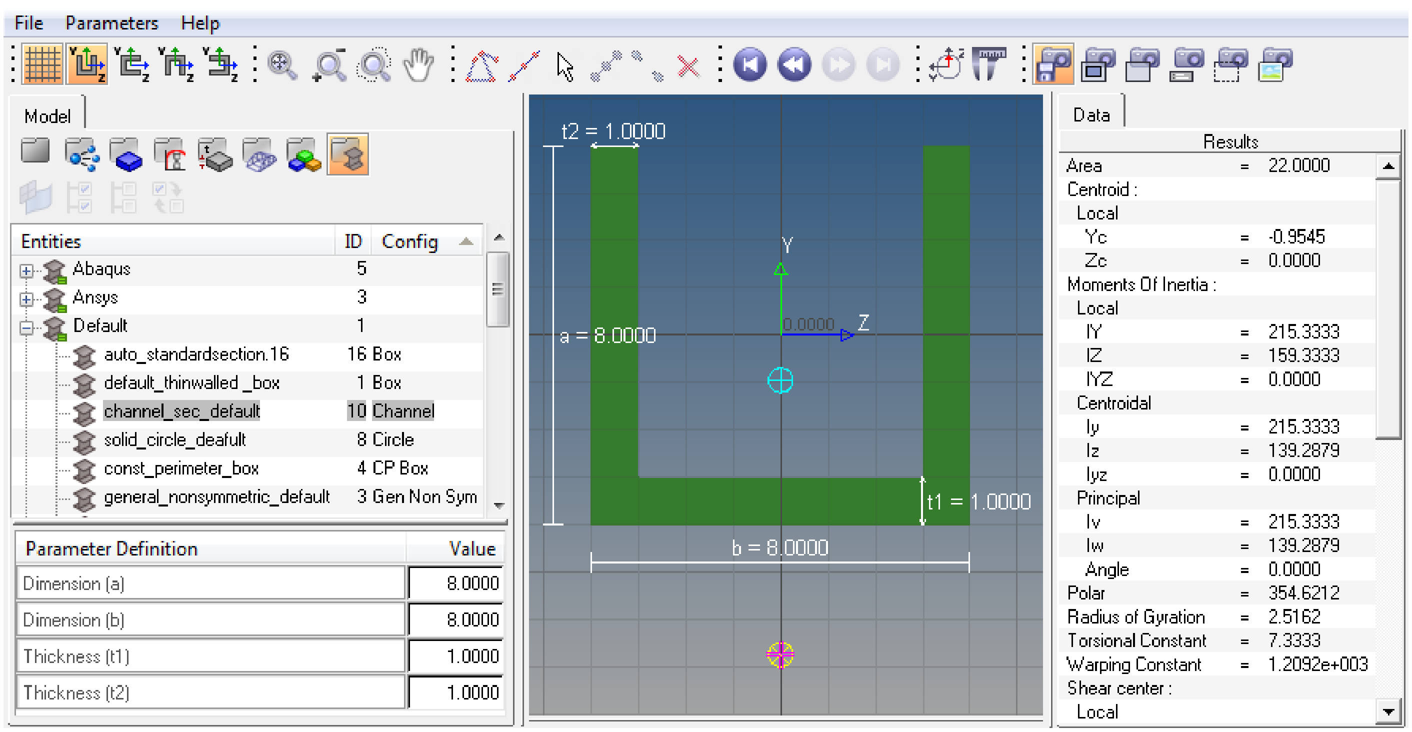


Figure 391:

References

The following sources were used in the creation of HyperBeam documentation:

1. *W.D. Pilkey, Analysis and Design of Elastic Beams, Wiley & Sons, New York, 2002.*
2. *H. Göldner, ed., Lehrbuch – Höhere Festigkeitslehre, Fachbuchverlag, Leipzig, 1979.*
3. *A. Gjelsvik, The Theory of Thin Walled Bars, Wiley & Sons, New York, 1981.*
4. *U. Schramm, V. Rubenchik, and W.D. Pilkey, Beam Stiffness Matrix based on the Elasticity Equations, International Journal for Numerical Methods in Engineering 40 (1997) 211-232.*

Section Browser and Parameter Definition

The Section Browser presents a hierarchical view of all of the beamsections and beamsection collectors in your database.

HyperBeam displays this hierarchy in a standard tree structure of beamsections residing inside of beamsection collectors. Click the **ID** or **config** column headings to sort the beamsections and beamsection collectors alphabetically. The Config column lists the type of section: Shell, Solid, Generic and various types of standard sections including Channel, I-Sect, L-Sect, and so on.

Use the Section Browser to find a particular section of your model for displaying or editing. When you click on a beamsection, it becomes highlighted and the results are displayed in the Results pane. Additionally, when you click on a beamsection, its parameters are listed and available for editing in the **Parameter Definition** window.

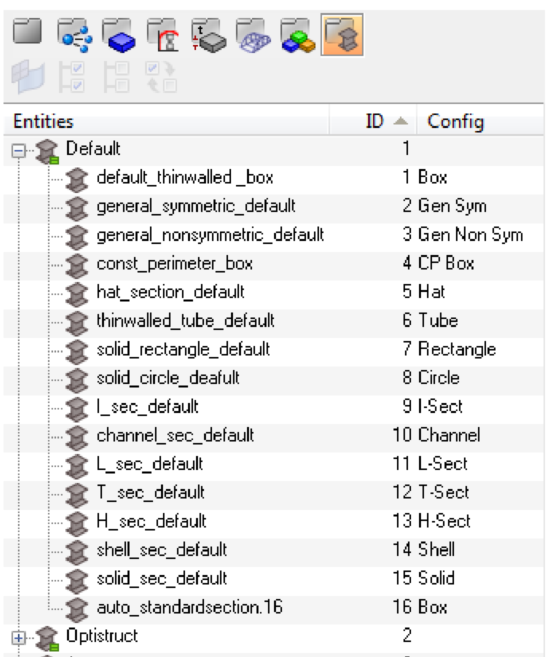


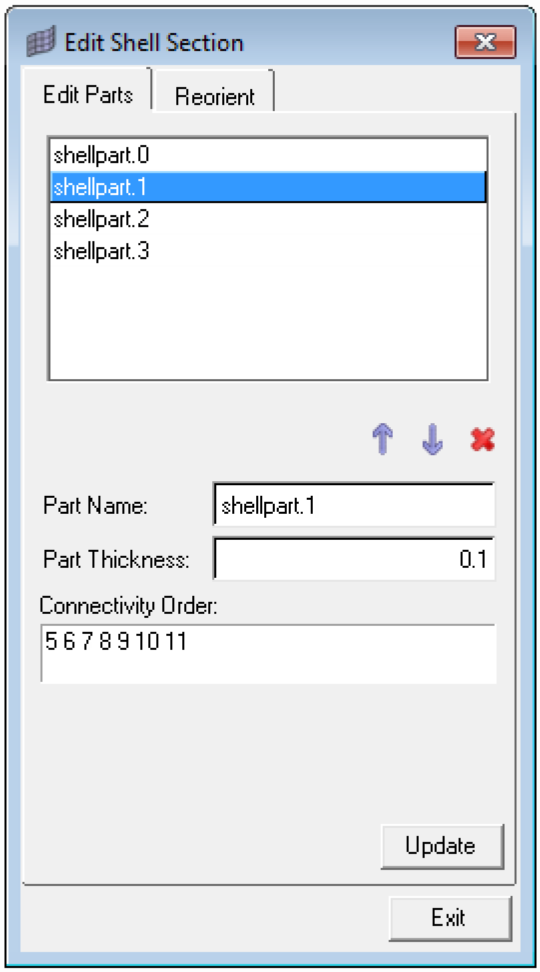
Figure 392:

Context Menu

The right-click context menu contains options for creating, editing, deleting, scaling, welding and exporting beamsections.

The following options are available from the HyperBeam View context menu:

Option	Description
Create	Create a new beam collector, as well as standard, shell, solid and generic beamsections. **Note:** The only standard section libraries listed are those of the current HyperMesh user profile and the HyperMesh general standard section library.
Edit	Opens the **Edit Shell Section** dialog. The dialog contains two tabs: Edit Parts and Reorient.

Option	Description
	• From the Edit Parts tab, you are able to control the naming, thickness, and connectivity order of each part in the beam section. The Connectivity Order simply lists the order of the vertices that define the highlighted part. • From the Reorient tab, you are able to reposition the section with respect to the local axis. 

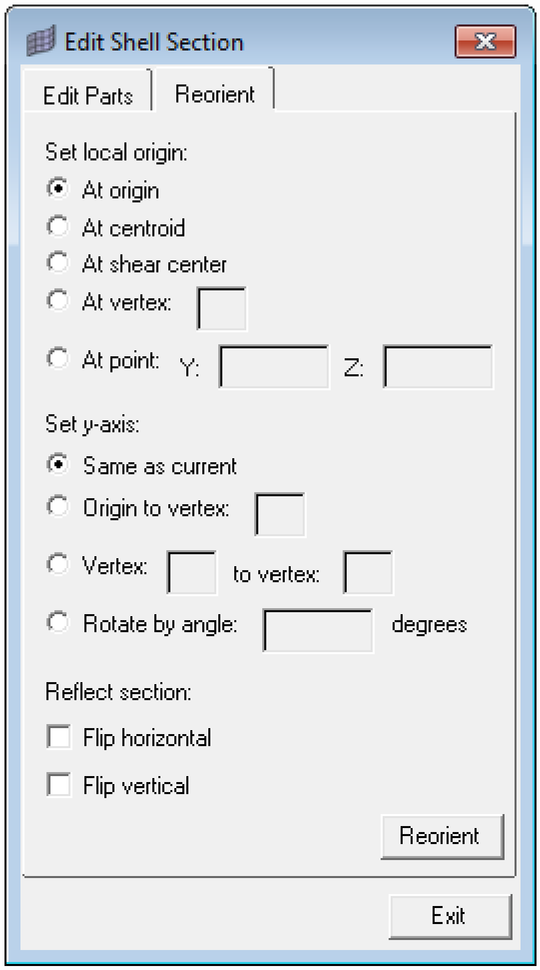
<table>
<tr><th>Option</th><th>Description</th></tr>
<tr><td></td><td>


Figure 393:

Note: This option is only available for shell sections.
</td></tr>
<tr><td>Scale</td><td>Opens the **Scale section** dialog, from which you can scale the entire beam section by a given scale factor.</td></tr>
<tr><td>Copy/Cut/Paste</td><td>Choose one or multiple beamsections to copy or cut, and then select a beamsection collector to paste the beamsections to.</td></tr>
<tr><td>Delete</td><td>Deletes the selected beam collector or standard, shell, solid and generic beam section.</td></tr>
<tr><td>Rename</td><td>Activates the Entities column of the selected beam collector or beamsection, allowing you to enter a new name.</td></tr>
<tr><td>Make Current</td><td>Flags which beam collector new beamsections will be created in.</td></tr>
</table>

Option	Description
	Note: This option is only available for beamsection collectors.
Collapse All	Closes all of the folders in the Section Browser, so that only the beamsection collectors are displayed.
Expand All	Opens all of the folders in the Section Browser, exposing every beamsection nested in every beamsection collector.
Auto-weld	Opens the **Auto Weld** dialog, from which you can set the tolerance and thickness for the welds. If the segments or any part are within the tolerance, Auto-weld will create a new part between them with the given thickness. New vertices might need to be created to make the connection, which should be centered at the area where the segments are the closest. If the thickness is given as 0.0, the connection will be set to a thickness equal to its length. Auto-weld can also be accessed by selecting the **auto-connect shell sections** option, from the HyperBeam panel > section utils subpanel. **Note:** This option is only available for shell sections.
Export CSV...	Captures the beamsection name, each part within the section, its thickness and each vertex number, and positions it all into a CSV file. Multiple shell sections can be selected and exported using Control+left-click, or by **selecting Export CSV - all shell sections** from the File menu. This option is particularly useful when you select **Import CSV** from the **File** menu, which will read the HyperBeam's CSV format and create new shell beamsections. The Import CSV and Export CSV options allow you to read and write to HyperBeam without using a `.hm` file. **Note:** This option is only available for shell sections.

Parameter Definition Window

The **Parameter Definition** window is located below the Section Browser. In this window you can edit the dimensions of standard, shell, and solid sections, and you can edit the Y and Z values for vertices of shell and solid sections. The graphics area is updated and the dimension values are automatically saved upon each entry. Use Tab and Shift+Tab to quickly navigate through the parameters.

Parameter Definition	Value
Dimension (a)	10.0000
Dimension (b)	10.0000
Thickness (t1)	1.0000
Thickness (t2)	1.0000

Figure 394: Standard Section Parameter Definition Window

Point	Y Value	Z Value
1	2.50000	-2.50000
2	-2.50000	-2.50000
3	2.50000	2.50000
4	-2.50000	2.50000

Figure 395: Shell Section Parameter Definition Window

HyperBeam View Toolbar

Access additional tools from the HyperBeam View toolbar.

The toolbar, located at the topic of the HyperBeam View, contains the following functions:

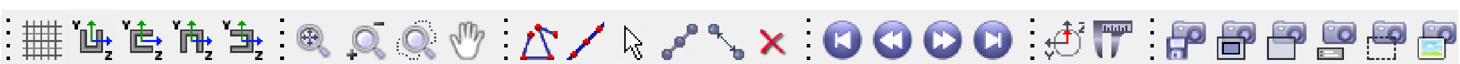
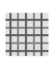


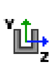
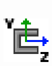
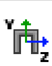
Figure 396:

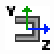
Grid Toggle Tool

Selecting and unselecting  turns the background grid in the graphics area on and off. When the background grid is turned on, you are able to quickly resolve the relative size of each section.

Beamsection Orientation Tools

Control the orientation with respect to the local origin of a standard section in the HyperBeam standard section library. Standard sections within the solver libraries already have a set orientation that maps to the solver; therefore, the Beamsection Orient options are only available for the sections that are defined in the HyperBeam library.

Button	Function	Behavior
	Orient 0.0	Sets the orientation to 0.0 degrees.
	Orient 90.0	Sets the orientation to 90.0 degrees.
	Orient 180.0	Sets the orientation to 1800.0 degrees.

Button	Function	Behavior
	Orient 270.0	Sets the orientation to 270.0 degrees.

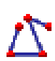
View Controls

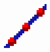
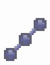
Controls the view. All these options are available through Control + mouse and follow the HyperMesh mouse controls.

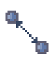
Button	Function	Behavior
	Fit to Window	Fits the model to the center of the graphics area.
	Zoom In/Out	Left-click: Zoom in Right-click: Zoom out
	Fit to Circle	Draw a circle around the area you would like to zoom in on in the graphics area.
	Move Center	This is a panning tool that allows you to grab the graphics area and move it to a different location.

Editing and Sketching Tools

Create and edit shell and solid sections, as well as, modify section parameters.

Button	Function	Behavior
	Create shell part or solid perimeter	This tool is activated automatically when defining a new shell or solid section canvas, and has two distinct functions: • For shell sections, this tool will sketch one shell section part at a time. Left-click to create a connected vertex, and right-click to remove it. Once the cursor leaves the screen the section will be initialized. • For solid sections, this tool will outline the perimeter of one continuous solid section at a time. It can also be used to cut holes in previously defined solid sections. Left-click to create a connected vertex, right-click to remove it. Once the cursor leaves the screen the section will be initialized.

Button	Function	Behavior
	Join parts / Split part at vertex	To join two parts that meet at a common point end, left-click on the first part, then left-click on the second part to merge it into the first part. To split a part at a vertex, left-click on the part that you wish to split, then right-click on the vertex that you wish to split the part at. The part splits into two parts.
	Move vertex / Edit dimension / Edit thickness	This tool is active for standard, shell and solid sections, and has three distinct functions: • For standard sections, this tool allows you to edit any of the parameters displayed on the section. Left-click and hold a dimension or thickness on the screen, and then drag; the section value and corresponding graphics will change as it is dragged. Double-click on a dimension, which is represented in the graphics area as text, to create a box around the text you want to edit. **Note:** The only available options to edit the text are backspace to delete characters, the ten number keys, and the period. Exponential notation is not supported. • For shell sections, this tool allows you to quickly snap any vertex to the predefined grid. It can also be used to edit the thickness of a shell part by left-clicking and dragging the thickness value. Double-clicking on a thickness, which is represented in the graphics area as text, will create a box around the text want can edit it. **Note:** The only available options to edit the text are backspace to delete characters, the ten number keys, and the period. Exponential notation is not supported. • For solid sections, this tool allows you to quickly snap any vertex to the predefined grid.
	Add vertex / Delete vertex	Left-clicking on or near any part (for a shell) or perimeter (for a solid) will create a new vertex on

Button	Function	Behavior
		the closest part or perimeter. The part or perimeter will be diverted to now run through the new vertex. If a clicked point is closer to the end of a part than a segment of it, the part will be extended. To prevent a bend in the part during vertex creation, select points on the shell parts themselves. If your cursor is near snap tolerance lines that cross existing shell parts, a "+" will appear. A relatively small snap size may make it difficult to tell whether the cursor is over the shell part or beside it. Increase the snap size so that it is more clear when the cursor has snapped to the shell part; or look at the point coordinates, if either the y or z value is not an even multiple of the snap size then the cursor is over the shell part and not beside it.
	Replace vertices / Detach a vertex	To replace vertices, left-click on the vertex that you wish to replace, then left-click on the vertex that you wish to replace the first vertex with. The second vertex replaces the first vertex. To split a vertex into two coincident vertices, right-click on a vertex.
	Delete shell part or solid perimeter	Left clicking on or near any part (for a shell) or perimeter (for a solid) will delete the nearest part or perimeter.

Undo/Redo Tools

Undo and redo changes you have applied to the currently displayed beam section (dimension changes, part additions, thickness changes, and so on). If you switch to a new beam, the undo/redo list will be cleared from the previous beam.

Button	Function	Behavior
	Undo all changes	Undo all of the changes applied to the currently displayed beam section.
	Undo last change	Undo the last change applied to the currently displayed beam section.
	Redo last change	Redo the last change applied to the currently displayed beam section.

Button	Function	Behavior
	Redo all changes	Redo the all of the changes applied to the currently displayed beam section.

Measuring Tools

Querying beamsection.

Button	Function	Behavior
	Find moments at mouse location	Selecting this icon activates an interactive mode. When you click on a point in the graphics area, the **Local moments of inertia** dialog appears. From this dialog you can review the following information: • The Y and Z coordinates of the point selected. • The Y and Z offsets from the shear center and centroid. • The moments of inertia Iyy, Izz, and Iyz for the section relative to the selected point. This tool is useful for determining beam element offsets or making custom beams. Click **Exit** to close the dialog select another point.
	Measure section with ruler	Selecting this icon activates an interactive mode. When you click two or more points in the graphics area, lines are created similar to the way a shell section is created. If two points are selected, the dy and dz distances between the points are displayed, as well as the total length of the line. If three or more points are selected, the total length of the line is displayed. Right-clicking removes the previously selected point.

Image Capture Tools

Take screen shots of various components in HyperBeam.

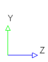
Button	Function	Behavior
	Save image to file/ clipboard	Use the toggle to determine whether to save an image to a file, or to a clipboard. When the toggle is on (highlighted orange), an image is saved to a file. When the toggle is off (gray), an image is saved to a clipboard. Images will be saved as a `.bmp`.

Button	Function	Behavior
	Capture full image	Captures and saves your entire application frame to your selected toggle setting.
	Capture graphics area	Captures and saves the graphics area to your selected toggle setting.
	Capture results area	Captures and saves the results area to your selected toggle setting.
	Capture rectangular area	Captures and saves a selected area to your selected toggle setting.
	Save graphics to jpg	Captures and saves the graphics area as `.jpg` to a location that you specify.

Graphics Window

The graphics area displays a representation of the geometric layout of the section.

All sections display the following items:

Item	Description
	Local origin of the beamsection
	Section Centroid
	Shear Center
	Element Axis

Standard sections display the geometric representation of the section based on the parameter values, which are also listed on the screen.

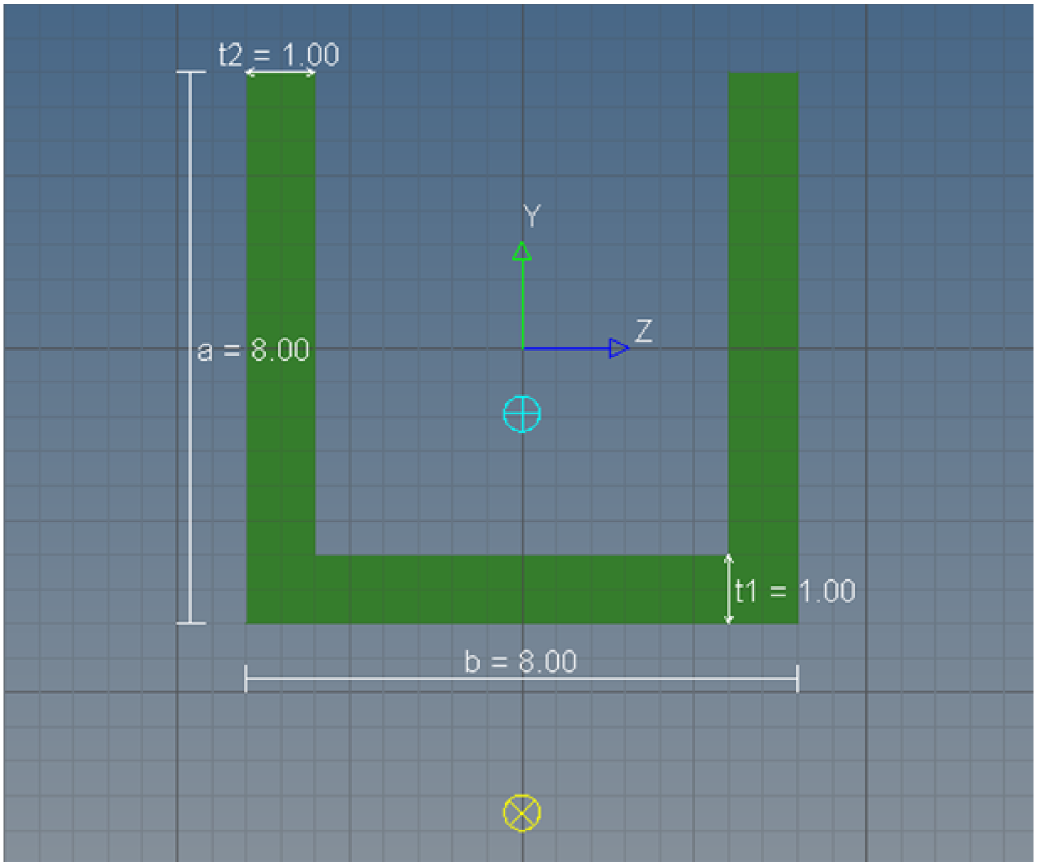


Figure 397:

For shell sections each part is drawn with lines connecting the dots that show the section's vertices.

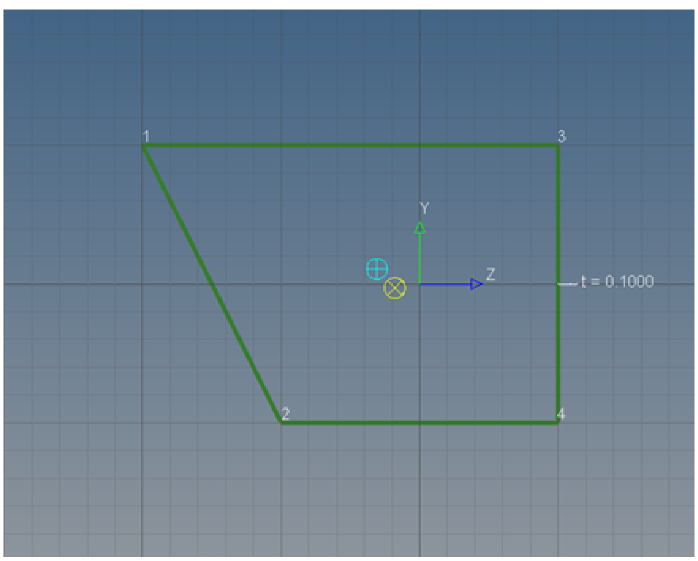


Figure 398:

Solid sections can be created and edited using the same tools as a shell section, and the mesh that is used for the section calculations is displayed.

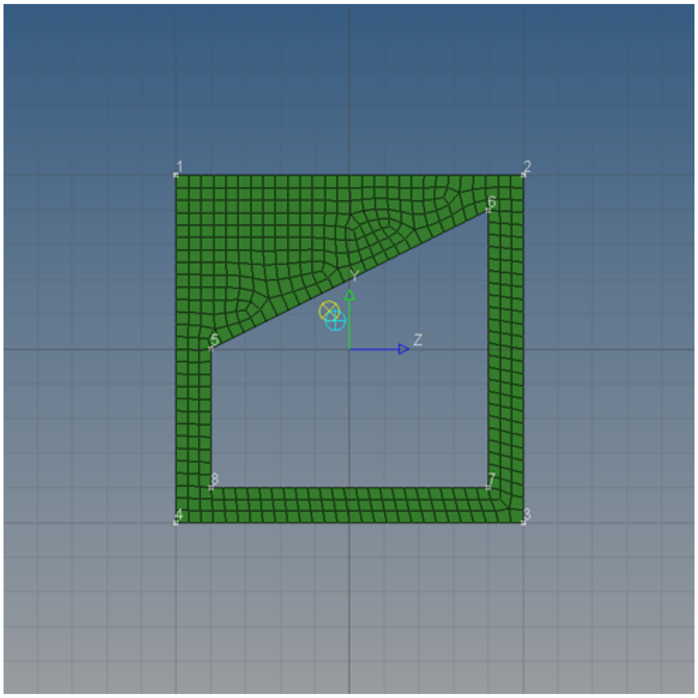


Figure 399:

Since generic sections do not inherently have any shape, only a grey box is shown. The centroid and shear center graphics location will update.

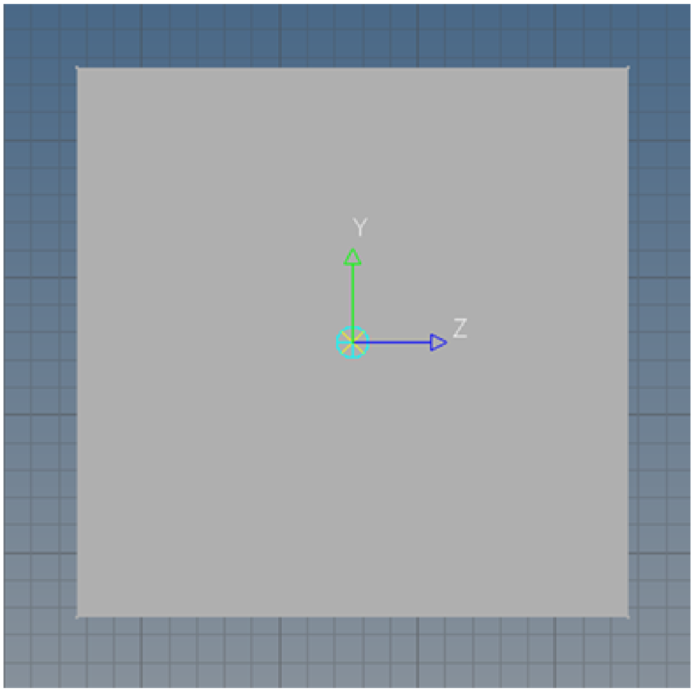


Figure 400:

Display Stress Recovery Locations

You can turn the display of stress recovery location's points and/or prop IDs, that are already defined within the property solver cards, on and off by modifying the display parameter.

Stress recovery points for shells, solids, and generic sections are only supported in the Nastran, Radioss and OptiStruct user profiles.

1. To access these parameters, click **Parameters** > **Display** from the menu bar.
2. To accept your changes, click **Update** before you exit the dialog.

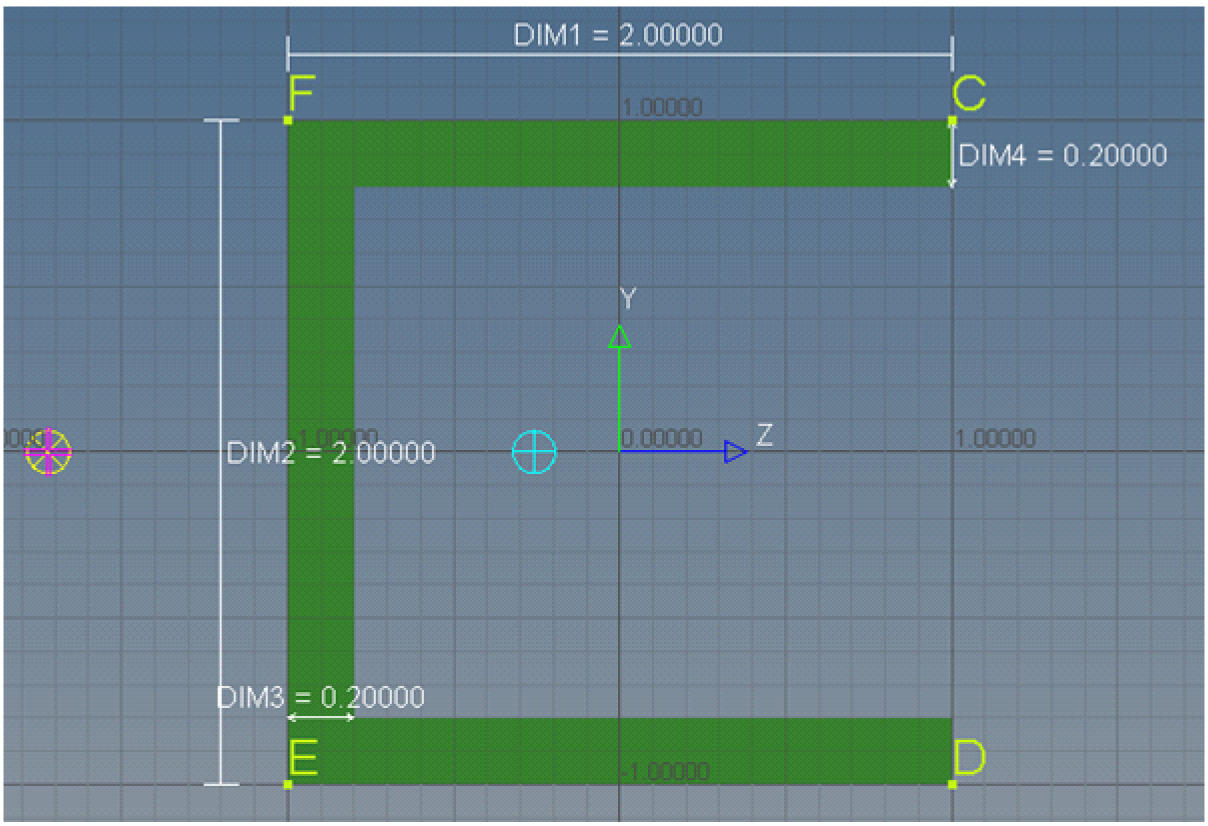


Figure 401:

Modify Grid Parameters in the Graphics Area

You can change the size of the background grid and vertices in the graphics area, as well as adjust the behavior of your cursor when it is near snap tolerance lines by modifying the grid parameters.

1. Click **Parameters** > **Grid** from the menu bar.
2. Modify the grid parameters.
3. In the Grid Size field, enter the size of the background grid in the graphics area.
4. In the Snap Size field, enter the incremental movement of your cursor over grid points in the graphics area. If your cursor is not snapping to the nearest grid points when you move your mouse in the graphics area, increase the snap size.
5. In the Point Size field, enter the size of the vertices in the graphics area. Increasing the point size makes it easier to select an existing grid point when you are creating a part.
6. To accept your changes, click **Update** before exiting the dialog.

HyperBeam Results Pane

Whenever HyperBeam computes the section properties of the current section, it displays them in the Results pane.

You cannot edit the text in this portion of the window for standard, shell and solid sections, but you can select it and copy/paste it into another application. The only way to change the parameters of a generic section is to edit the values in the Results pane.

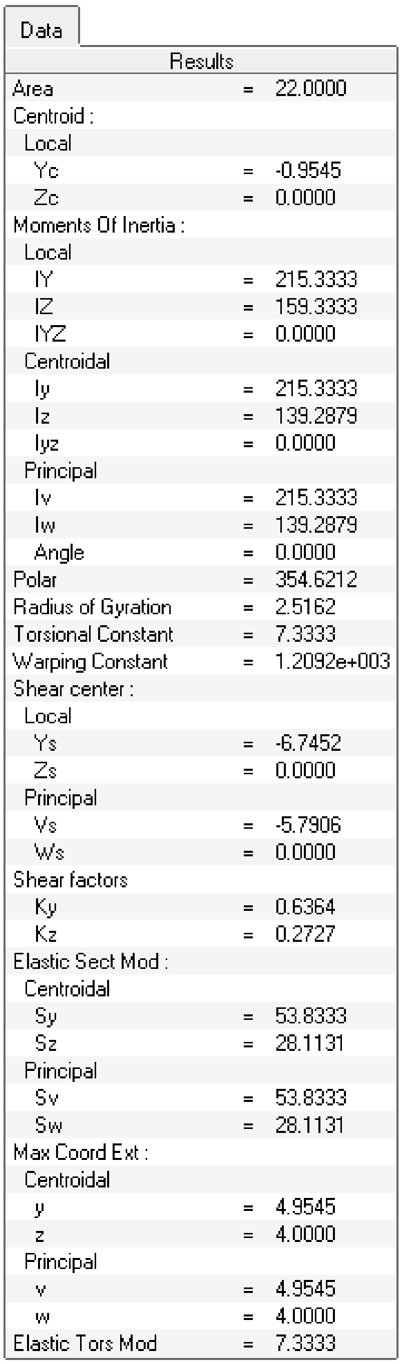


Figure 402:

HyperBeam Sections

Types of HyperBeam sections that can be defined.

Standard Section

- Use Standard sections to define solver supported sections. Each supported solver has a
- 3D visualization is available in HyperMesh.

Shell Section

- Use Shell sections to define thin cross-sections with geometric lines or 1D elements. To create shell sections, use the shell and solid creation and editing tools or bring in geometry and element data from HyperMesh.
- When you create a shell beam section by cutting a cross-section through a shell mesh, any thickness that was assigned directly to the elements will be given to the shell section. The thickness applied to the section is equal to the average thickness for all elements in the component with each element getting equal weight. The number of elements matters, but the area of the elements does not. For composites, the thickness will be extracted treating the composites as homogenous.
- 3D visualization is available in HyperMesh.

Solid Section

- Use Solid sections allow to define solid beam cross-sections with continuous 2D elements, connecting lines that have a closed loop, and continuous surfaces. To create solid sections, use the shell and solid creation and editing tools or bring in geometry and element data from HyperMesh.
- 3D visualization is available in HyperMesh.

Generic Section

- Use Generic sections to define sections without defining actual cross-section geometry. Areas, inertias, centroids, and other coefficients are supported.
- No 3D visualization is available in HyperMesh.

Example: Create Shell and Solid Sections Using the HyperBeam Sketcher

1. In the Section Browser, right-click and select **Create** > **Shell Section** or **Solid Section** from the context menu.
2. In the **Create** dialog define parameters and click **Create**.
   a) In the Thickness field, enter the default thickness for each new part created.
      Part thicknesses can be edited later.
   b) In the Canvas size field, enter the initial length and width of the sketching area.
   c) In the Grid size field, enter the default size of the background grids.
      This can be altered in the Parameters pull-down menu.
   d) In the Snap size field, enter the smallest interval at which you can define a vertex by clicking while sketching.
      This is also the snap size while moving existing vertices.
   e) In the Point size field, enter the size of the vertices in the graphics area.

> **Tip:** Increasing the point size makes it easier to select an existing grid point when you are creating a part.

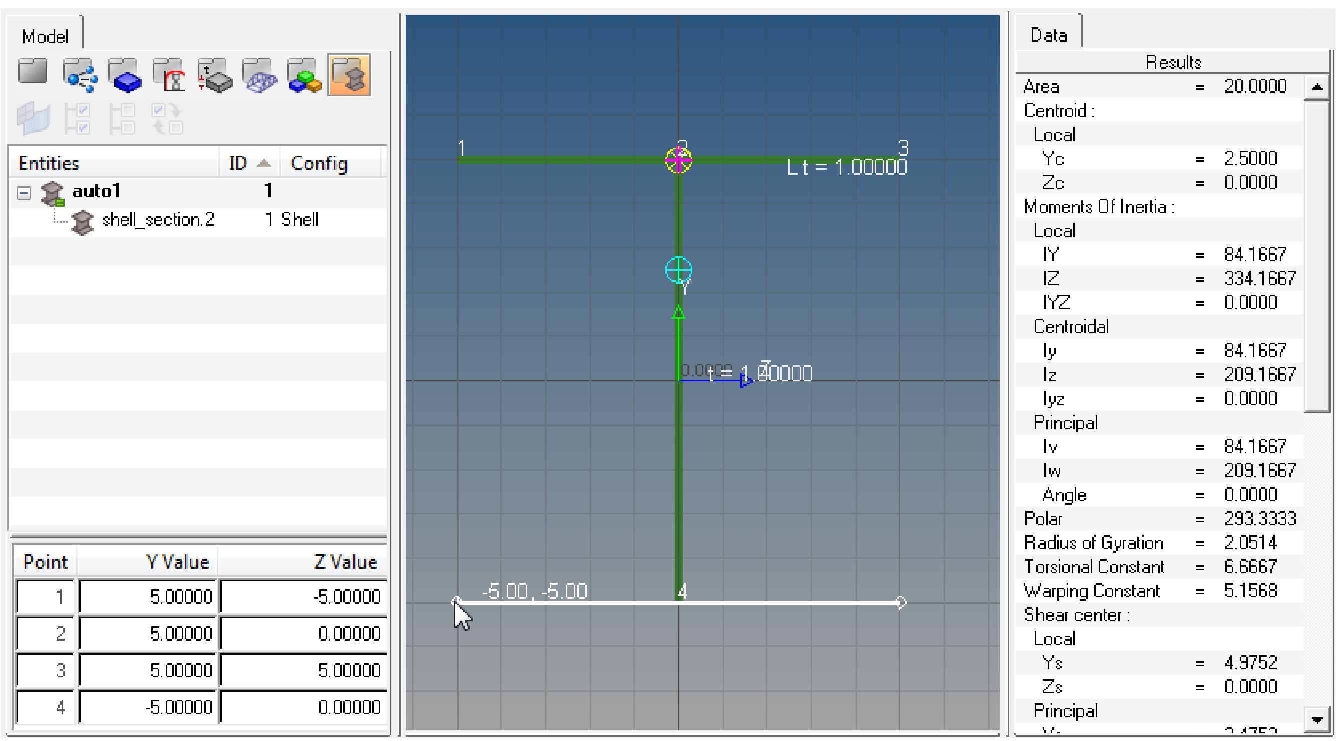


Figure 403:

3. Once the defaults are set, the canvas will appear and you can start creating parts with the shell part / solid perimeter tool ( ).

Once clicked, the button remains highlighted until you move the mouse off of the canvas.

- Left-click to create a new vertex.
- Right-click to remove vertices in the reverse order they were created.
- Once the part is complete, move the mouse off of the canvas to finalize it.

> **Note:** Each part has only one thickness, and it is common to have a section with multiple connecting parts.

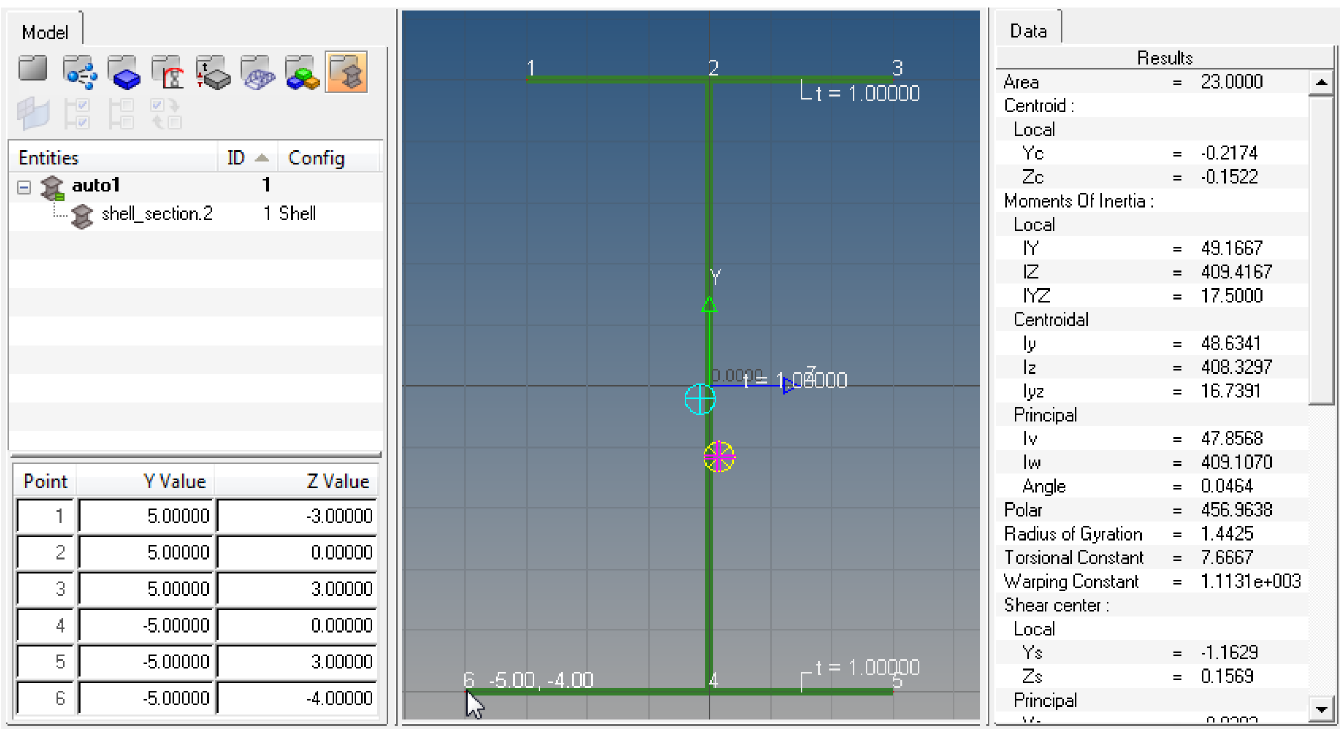


Figure 404:

4. Use the Move vertex / Adjust Dimension tool ( ) to click and drag vertices to snap points for shell and solid sections, or to adjust parameters such as shell thicknesses or standard section parameters.

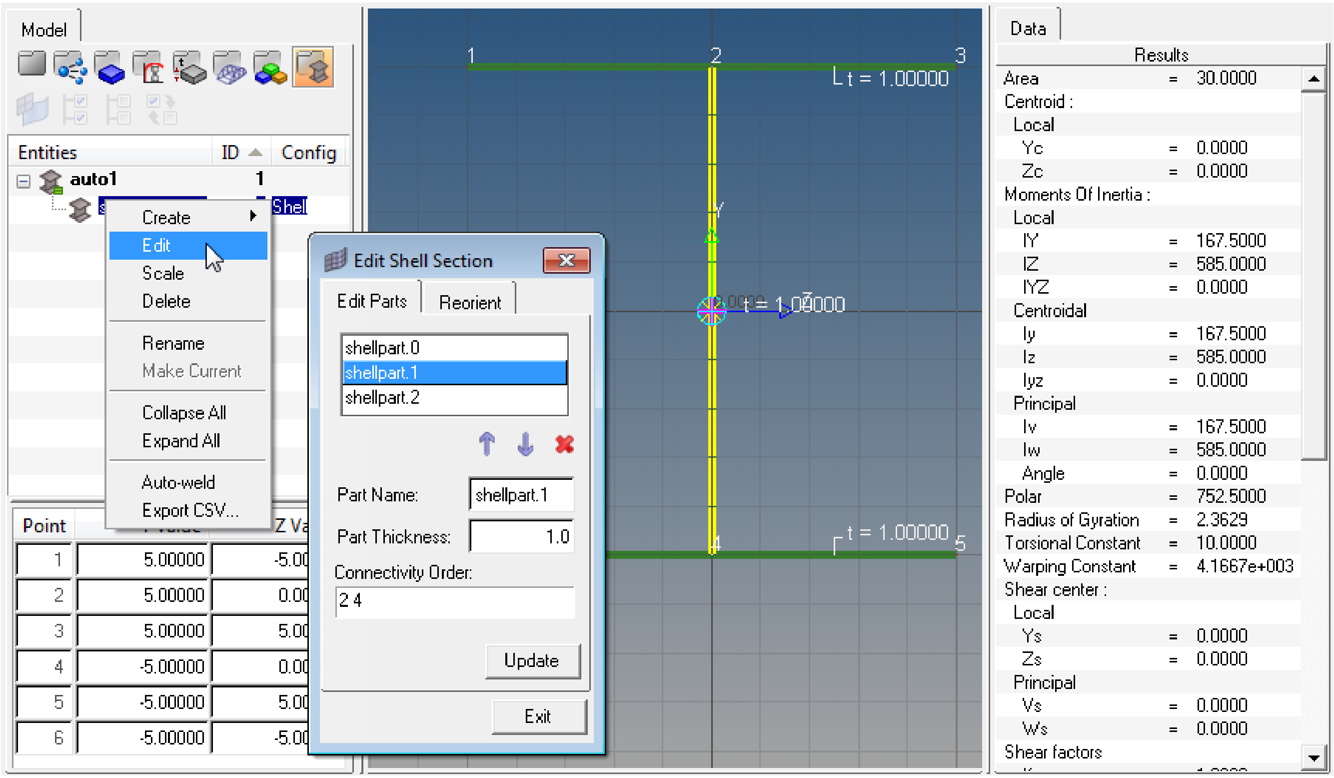


Figure 405:

5. After the parts are created and assembled it is sometimes necessary to revise the connectivity or change the part thicknesses.

This can be done in the part editor shown above. Each part is listed with its part name, thickness and connectivity, and as each part is selected it will highlight yellow on the canvas.

The connectivity order is simply the order in which the vertices are drawn for a given part. Remember to leave a space between each vertex when editing the connectivity order.

6. Redefine the local axis location and angle using the Reorient tab in the **Edit shell section** dialog.

This becomes useful when comparing HyperBeam shell section properties (results) to more rigidly-defined sections. Note that changing the local axis in HyperBeam is not the same as using element offsets in a finite element model.

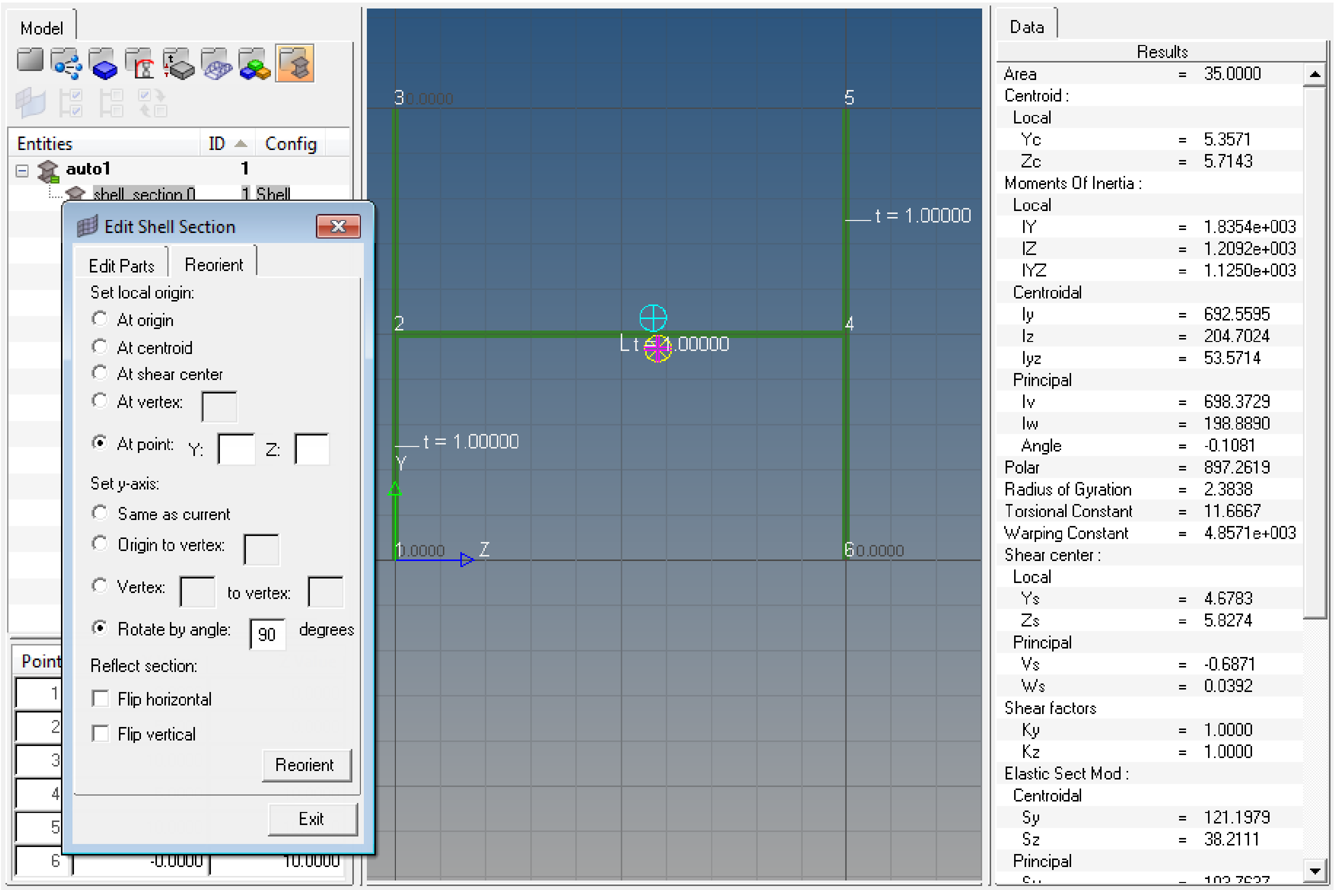


Figure 406:

The same toolset can be applied to sketching solid sections as well. You can start by right-clicking the browser on the left-hand side, creating a new solid section, and defining defaults for Canvas, Grid and Snap sizes.

The Shell part / Solid perimeter tool can now be used to trace the outline of a solid shape.

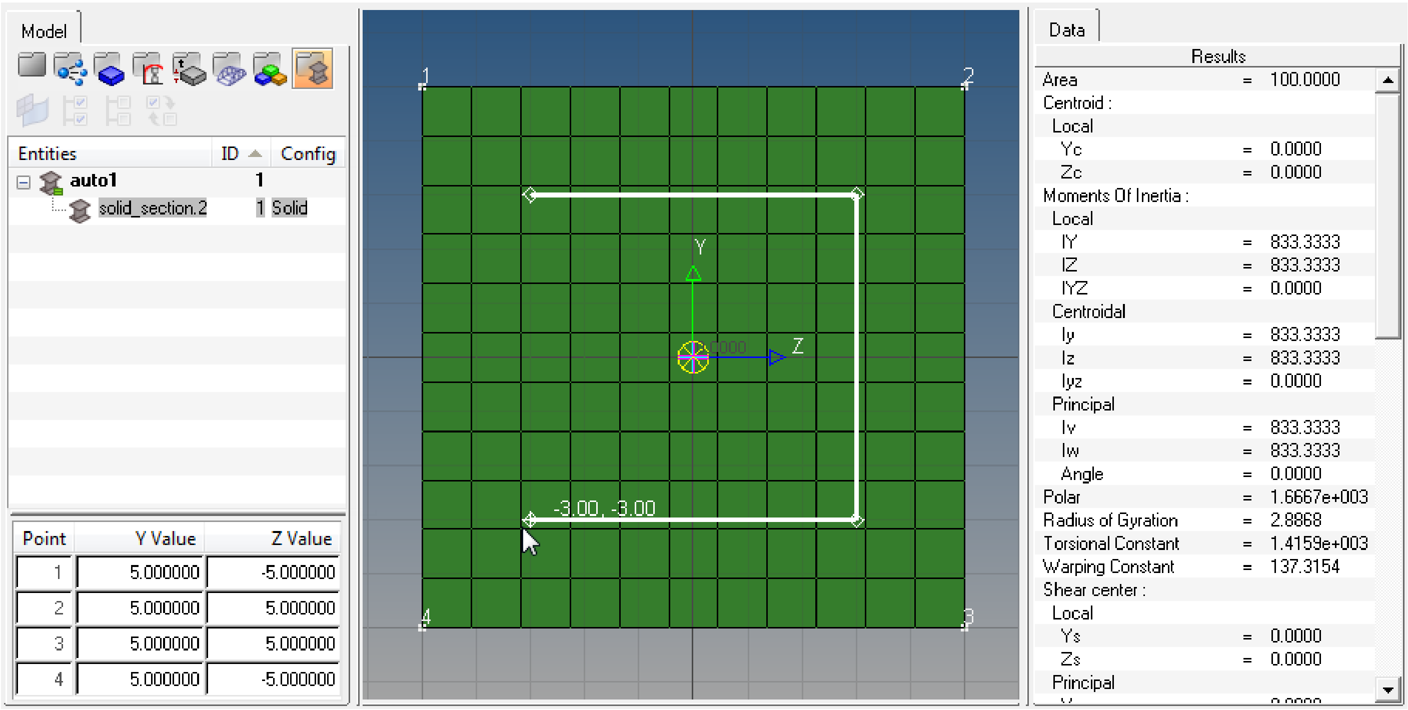


Figure 407:

As before, once the outline is sketched, move the mouse out of the canvas area to finalize the part. Unlike shell sections, solid sections can only have one part defined, although the same tool can be used to sketch out a cavity within the solid section.

Note: Only one cavity can be defined per section.

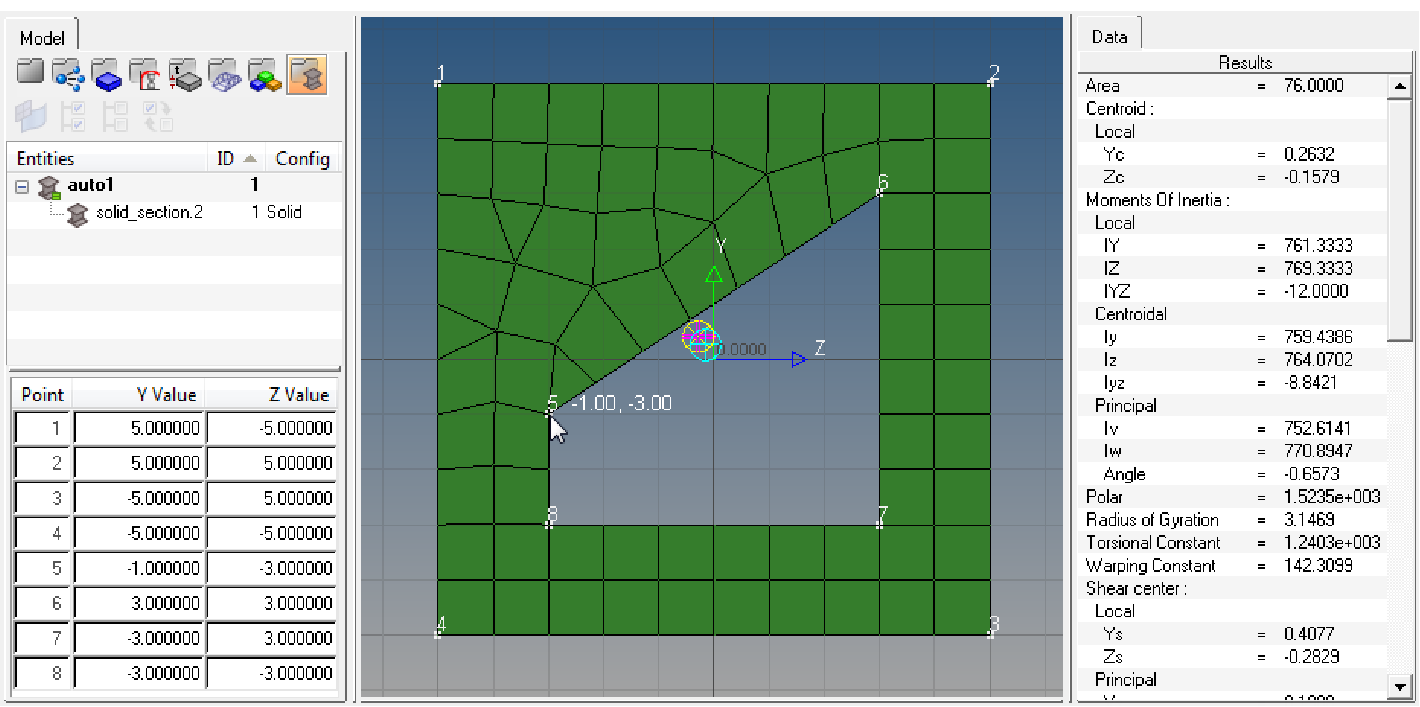


Figure 408:

As before, the Move vertex / Adjust dimension tool can be used to move vertices of solid sections. Each vertex movement requires a re-mesh of the solid section, so take care when moving vertices of complicated parts.

Example: Create and Assign a Standard Section

Create and assign a standard section to an OptiStruct `PBARL` property.

Before you begin, load the OptiStruct user profile and open the `standard_section.hm` model, located in <installation_directory>\tutorials\hm.

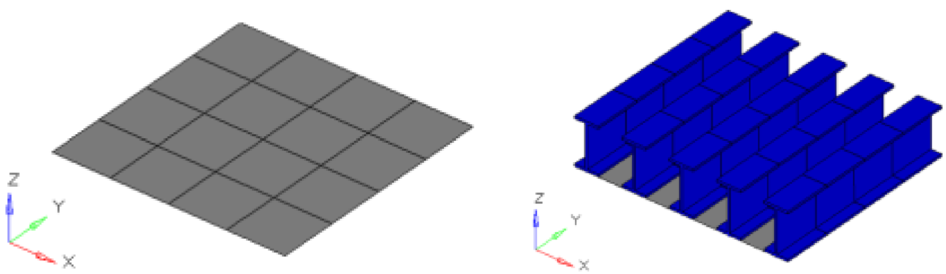


Figure 409:

The grey elements in the image to the left represent a structure that needs to be stiffened by adding I-beams down the length of it. The image to the right is the 3D visualization of 1D bar elements running along five separate node paths.

1. In the Model Browser, click  to activate the HyperBeam view.
2. Create a Standard Section by right-clicking on the browser and selecting **Create** > **Standard Section** > **OptiStruct** > **I** from the context menu.
3. Define the parameters as shown below:

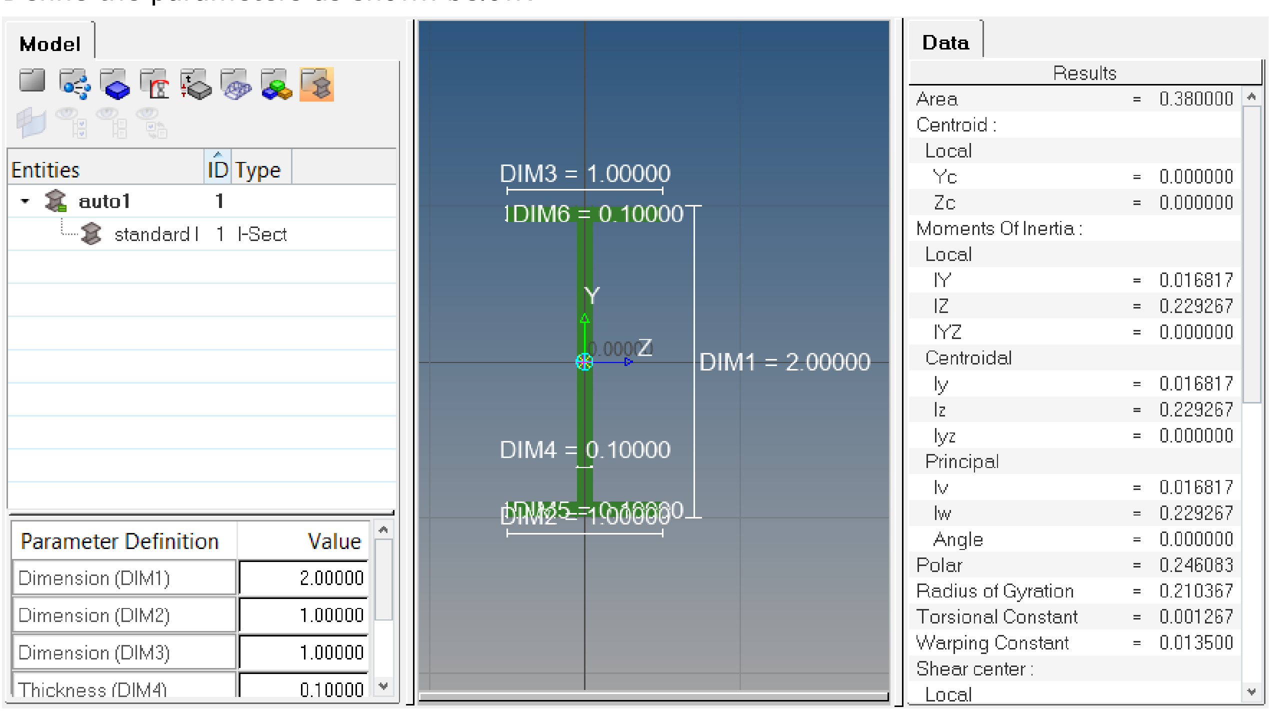


Figure 410:

Once HyperBeam solves the cross sectional properties, it is necessary to attach the beam section to a `PBARL` card image. This can be done by creating a component and assigning it a property that references the PBARL card image and beam section.

4. In the Model browser, right-click and select **Create** > **Component** from the context menu.
A new component is created, and opens in the Entity Editor.

5. In the Entity Editor, right-click on the **Property** field and select **Create** from the context menu.

6. Define the property in the **Create Properties** dialog and then click **Close**.

a) In the Name field, enter `pbarl`.

b) For Card Image, select **PBARL**.

c) For Beam Section, click **<Unspecified>** > **Beamsection**.

d) In the **Select Beamsection** dialog, select **standard I** and click **OK**.

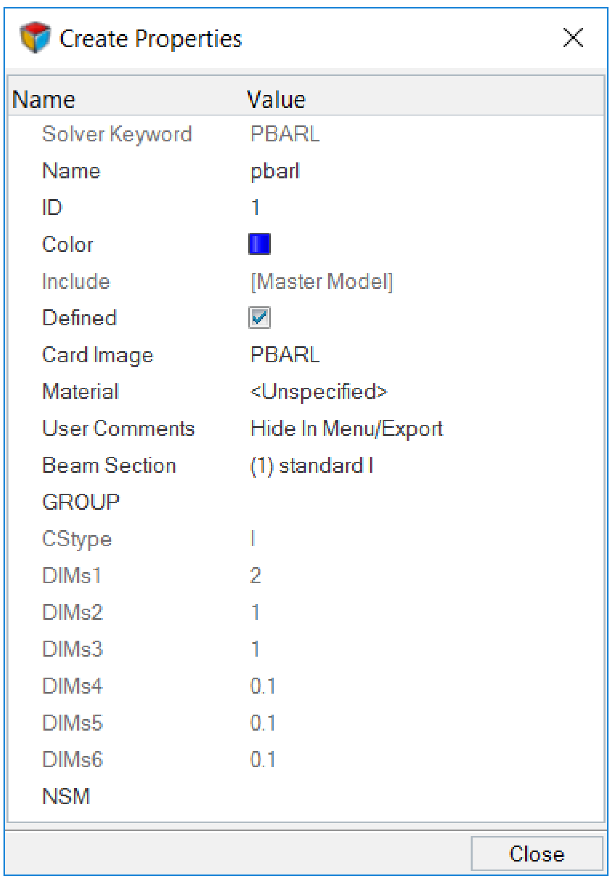


Figure 411:

7. After the new component and property are created with the beam section attached, the bar element can be defined in the bars panel in the 1D menu-page.

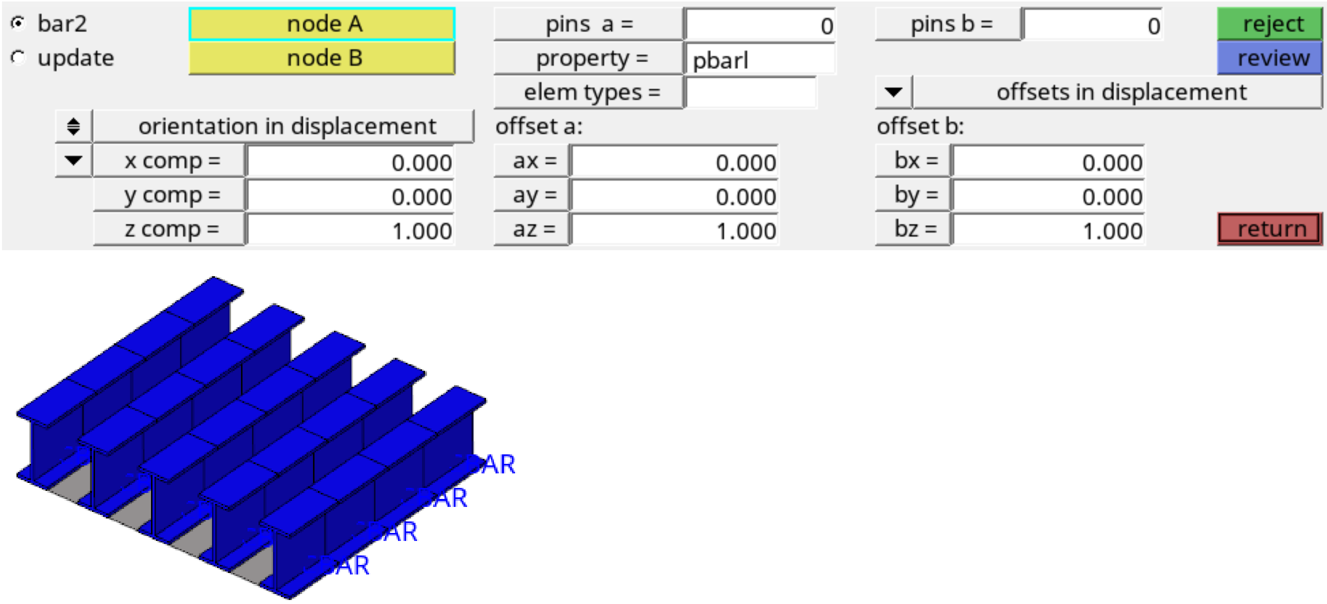


Figure 412: Bars Panel

Bar element alignment using HyperBeam sections is very straightforward since standard sections are defined using an absolute y-direction. The direction specified in the Bars panel defines the alignment of the beamsection's y-direction. In this case, the positive z-direction in the Bars panel will align with the y-direction of the HyperBeam section. Since the centerline of a 1D beam element is defined about the section's shear center, elemental offsets are frequently required. In this case, we have added z-offsets at both ends of each element to align the I-beams flush with the plate.

To fully visualize the 1D element in HyperMesh, find the display option in the Visualization toolbar ( ).

Example: Create and Assign a Shell Section

Create and assign a shell section to an OptiStruct `PBAR` property.

Before you begin, load the OptiStruct user profile and open the `shell_section.hm` model from `<installation_directory>\tutorials\hm`.

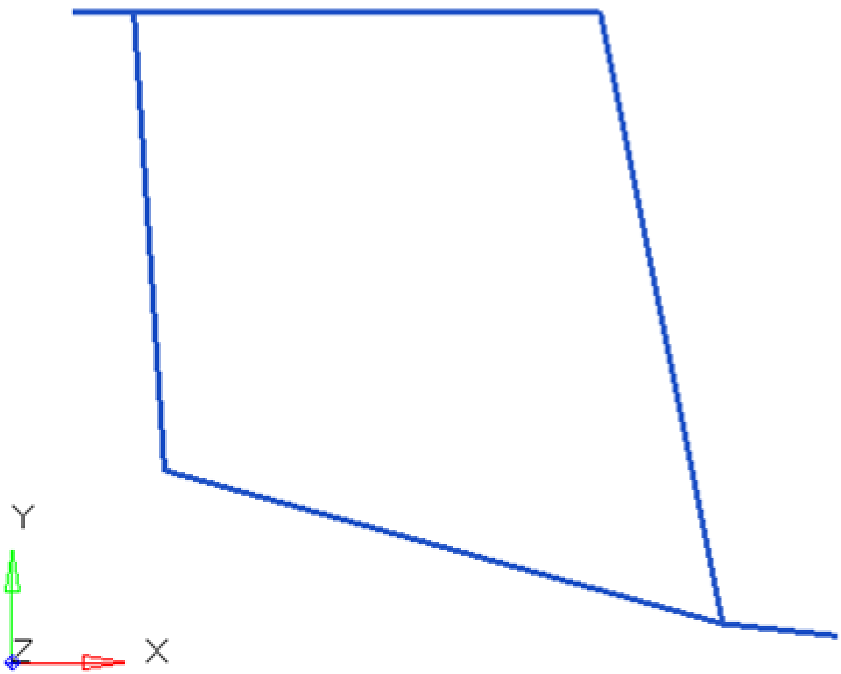
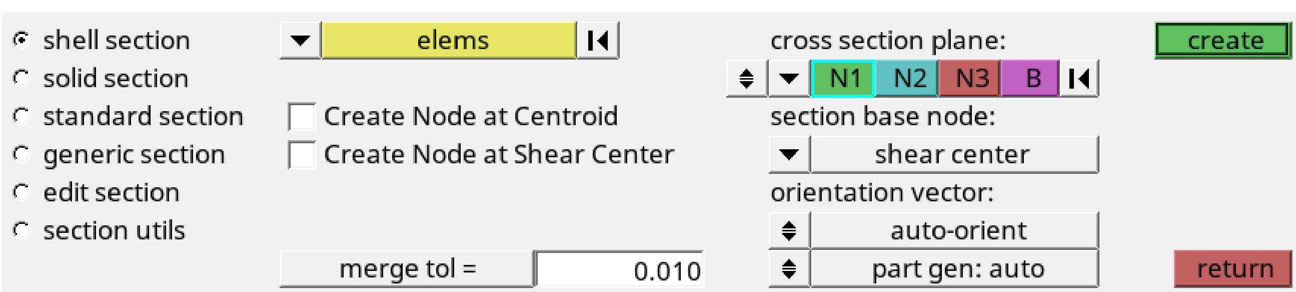


Figure 413:

The blue lines are plot elements denoting the beam section. Elements or lines can be used to describe a beam cross section. The purple lines are plot elements used to align the section within HyperBeam.

1. From the 1D page, select the **HyperBeam** panel, **shell section** subpanel.
2. Set the selector type to **use elems**, and then select the blue plot elements.
3. Under **cross section plane**, toggle to the plane and vector selector.



4. Select N1, N2, and N3 locations.

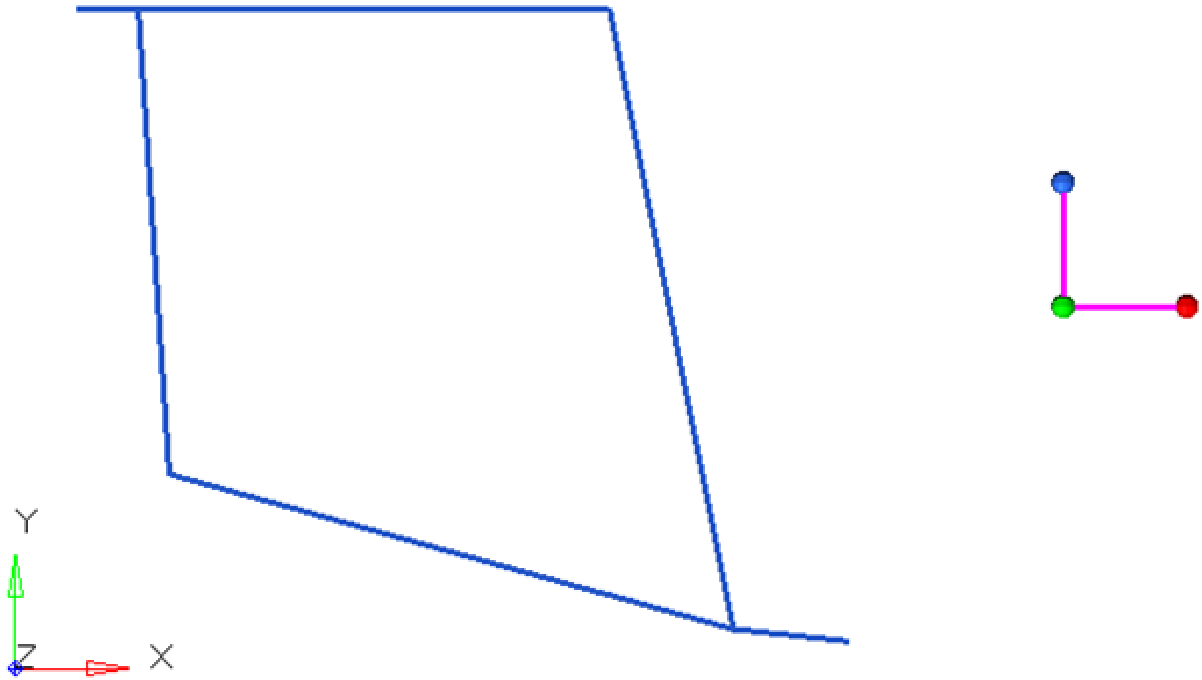


Figure 414:

The vector created by N1 to N2 describes the local y-axis used in HyperBeam. N3 describes the positive sense of the z-axis. It is important to note the alignment of the local axes at this point. Later, it will be necessary to know this when the beam section is aligned for bar elements.

5. Set section base node to **shear center**.
6. Click **create**.
   HyperBeam View is invoked.

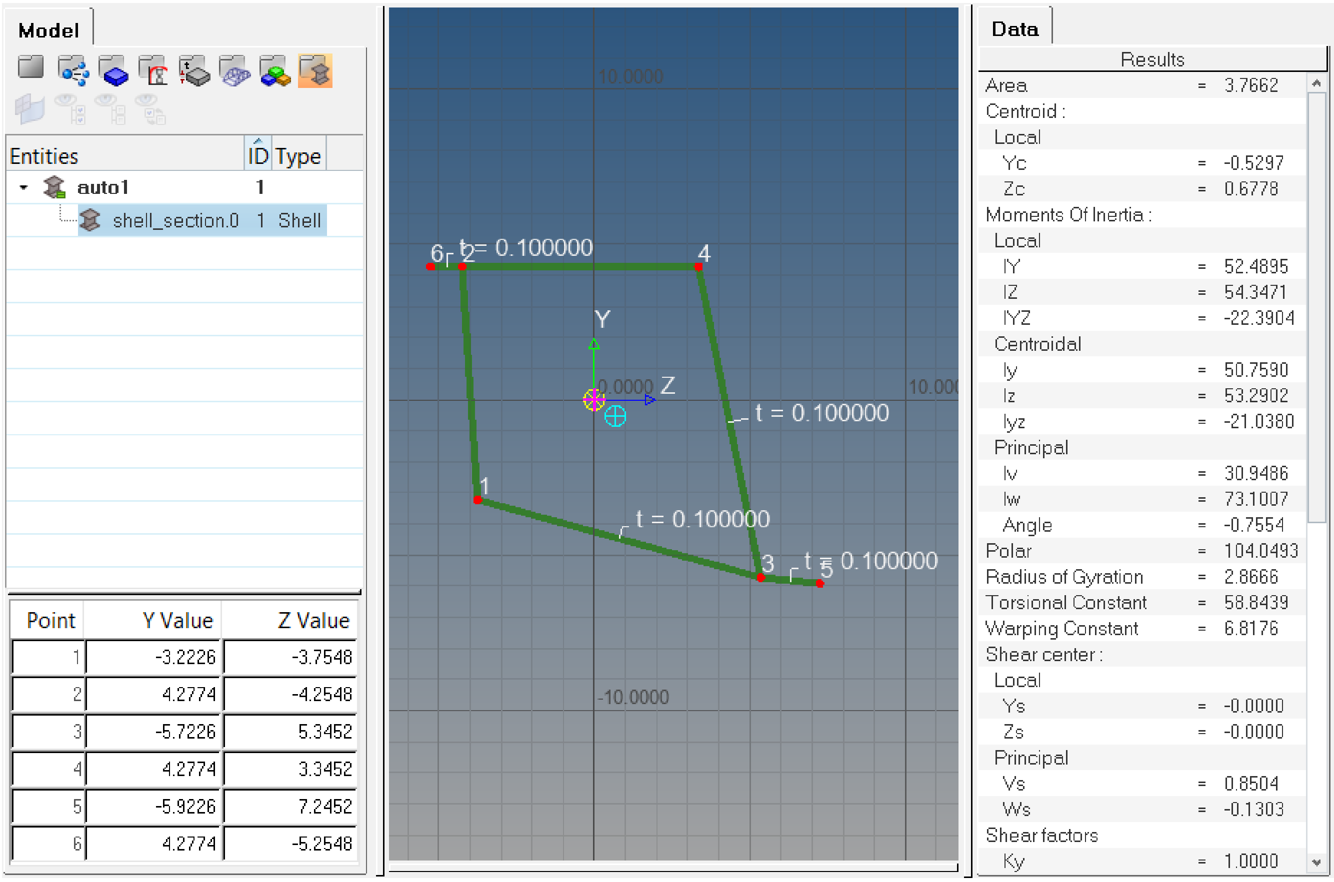


Figure 415:

Once selected solves the cross-sectional properties, it is necessary to attach the beam section to a `PBAR` card image. This can be done by creating a component and assigning it a property that references the PBAR card image and beam section.

7. In the Model browser, right-click and select **Create** > **Component** from the context menu.
   A new component is created, and opens in the Entity Editor.
8. In the Entity Editor, right-click on the **Property** field and select **Create** from the context menu.
9. Define the property in the **Create Properties** dialog and then click **Close**.
   a) In the Name field, enter `pbar`.
   b) For Card Image, select **PBAR**.
   c) For Beam Section, click **<Unspecified>** > **Beamsection**.
   d) In the **Select Beamsection** dialog, select **shell_section.0** and click **OK**.

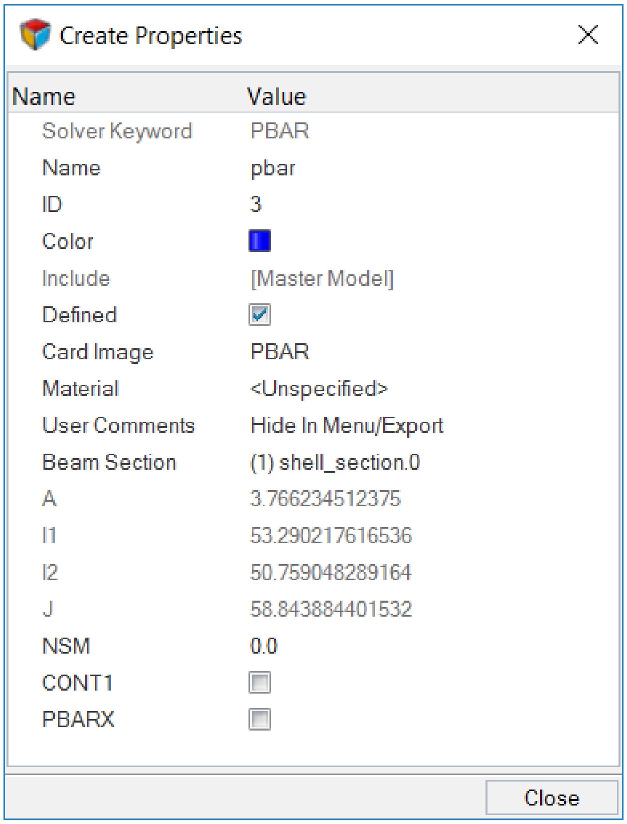


Figure 416:

10. After the new component and property are created with the beamsection attached, the bar element can be defined in the bars panel in the 1D menu-page.

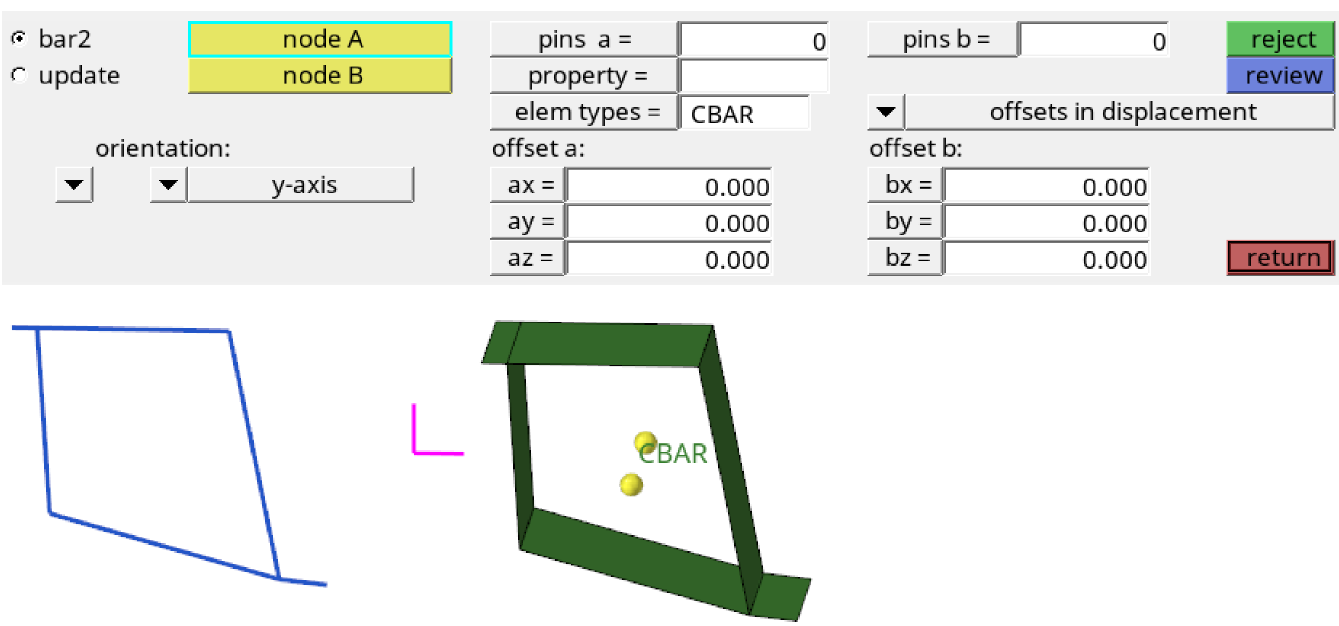


Figure 417:

Elastic Section Modulus

$$E_y = \frac{I_{yy}}{z_{\max}}$$

$$E_z = \frac{I_{zz}}{y_{\max}}$$

Bar element alignment using HyperBeam sections is very straightforward if the section has been defined using an absolute y-direction. The direction specified in the Bars panel defines the alignment of the beamsection's y-direction. In this case, the positive y-direction in the Bars panel will align with the y-direction of the HyperBeam section.

To fully visualize the 1D element in HyperMesh, find the display option in the Visualization toolbar ( ).

Cross Sectional Properties Calculated by HyperBeam

The beam cross section is always defined in a y,z plane.

The x-axis is defined along the beam axis. The coordinate system you define is called the local coordinate system; the system parallel to the local coordinate system with the origin in the centroid is called the centroidal coordinate system; the system referring to the principal bending axes is called the principal coordinate system.

For shell sections, only the theory of thin walled bars is used. This means that for the calculation of the moments and product of inertia, terms of higher order of the shell thickness t are neglected. Thickness warping is also neglected.

Area

$$A = \int dA$$

Area Moments of Inertia

$$I_{yy} = \int z^2 dA$$

$$I_{zz} = \int y^2 dA$$

Area Products of Inertia

$$I_{zz} = \int y^2 dA$$

Radius of Gyration

$$R_g = \sqrt{\frac{I_{\min}}{A}}$$

Elastic Section Modulus

$$E_y = \frac{I_{yy}}{z_{\max}}$$

$$E_z = \frac{I_{zz}}{y_{\max}}$$

Max Coordinate Extension

$$y_{\max} = \max|y|$$

$$z_{\max} = \max|z|$$

Plastic Section Modulus

$$P_y \int |z| dA$$

$$P_z \int |y| dA$$

Torsional Constant

Solid

$$I_t = I_{yy} + I_{zz} + \int \left(z\frac{\partial \omega}{\partial y} - y\frac{\partial \omega}{\partial z} \right) dA$$

ω - Warping function
(see below for warping function)

Shell open

$$I_t = \frac{1}{3}\int t^3 ds$$

t - Shell thickness

Shell closed

$$I_t = 2\sum A_{mi} F_{si}$$

A_{mi} - Area enclosed by cell i
F_{si} - Shear flow in cell i

Elastic Torsion Modulus

Solid

$$E_t = \frac{I_t}{\max}\left(y^2 + z^2 + z\frac{\partial \omega}{\partial y} - y\frac{\partial \omega}{\partial z} \right)$$

Shell open

$$E_t = \frac{I_t}{\max t}$$

Shell closed

$$E_t = \frac{I_t}{\max\left(\frac{F_{si}}{t}\right)}$$

Shear Center

$$y_s = \frac{I_{yz} I_{y\omega} - I_{zz} I_{z\omega}}{I_{yy} I_{zz} - I_{yz}^2} \quad I_{y\omega} = \int y\omega dA, I_{z\omega} = \int z\omega dA$$

$$z_s = \frac{I_{yz} I_{y\omega} - I_{yz} I_{z\omega}}{I_{yy} I_{zz} - I_{yz}^2}$$

Warping Constant (normalized to the shear center)

$$I_{\omega\omega} = \int \omega^2 dA$$

Shear deformation coefficients

$$\alpha_{zz} = \frac{1}{Q_y^2} \int \left(\tau_{xy}^2 \big|_{Q_z=0} + \tau_{xz}^2 \big|_{Q_z=0} \right) dA$$

$$\alpha_{zy} = \frac{1}{Q_y Q_z} \int \left(\tau_{xy} \big|_{Q_y=0} \tau_{xy} \big|_{Q_z=0} + \tau_{xz} \big|_{Q_y=0} \tau_{xy} \big|_{Qz=0} \right) dA$$

$$\alpha_{zz} = \frac{1}{Q_z^2} \int \left(\tau_{xy}^2 \big|_{Q_y=0} + \tau_{xz}^2 \big|_{Q_y=0} \right) dA$$

Shear stiffness factors

$$k_{yy} = \frac{1}{\alpha_{zz}}$$

$$k_{yz} = \frac{-1}{\alpha_{yz}}$$

$$k_{zz} = \frac{1}{\alpha_{yy}}$$

Shear stiffness

$$S_{ii} = k_{ii} G A$$

Warping Function

$$\nabla^2 \omega = 0$$

$$\left(\frac{\partial \omega}{\partial y} - z \right) n_y + \left(\frac{\partial \omega}{\partial z} + y \right) n_z = 0$$

For solid sections, the warping function is computed using a finite element formulation. This may lead to un-physically high stresses in geometric singularities (sharp corners) that get worse with mesh refinement. This may cause problems computing the elastic torsion modulus.

Nastran Type Notation

$/1 = I_{zz}$

$/2 = I_{yy}$

$/12 = I_{yz}$

$K1 = K_{yy}$

$K2 = K_{zz}$

Working with Beamseactions in HyperMesh

Beamsections are handled slightly differently in each HyperMesh user profile, although certain features remain constant.

Components, Properties, Elements, and Beamsections

HyperMesh offers a range of ways to organize a FEA model. Understanding the connection between components, properties, elements, and beamsections is important for 1D beam modeling.

The Model Browser allows you to create a component, property and material all at once and verifies that everything is appropriately assigned. It also allows you to assign an existing beamsection to the property. This is probably the simplest way to create and organize components for 1D modeling.

When a component is opened in the Entity Editor, right-clicking on the **Property** field and selecting **Create** from the context menu allows you to automatically create and assign a new property to the component in the **Create Properties** dialog. Selecting a beam section via the Beam Section attribute will automatically populate attributes related to the beam section.

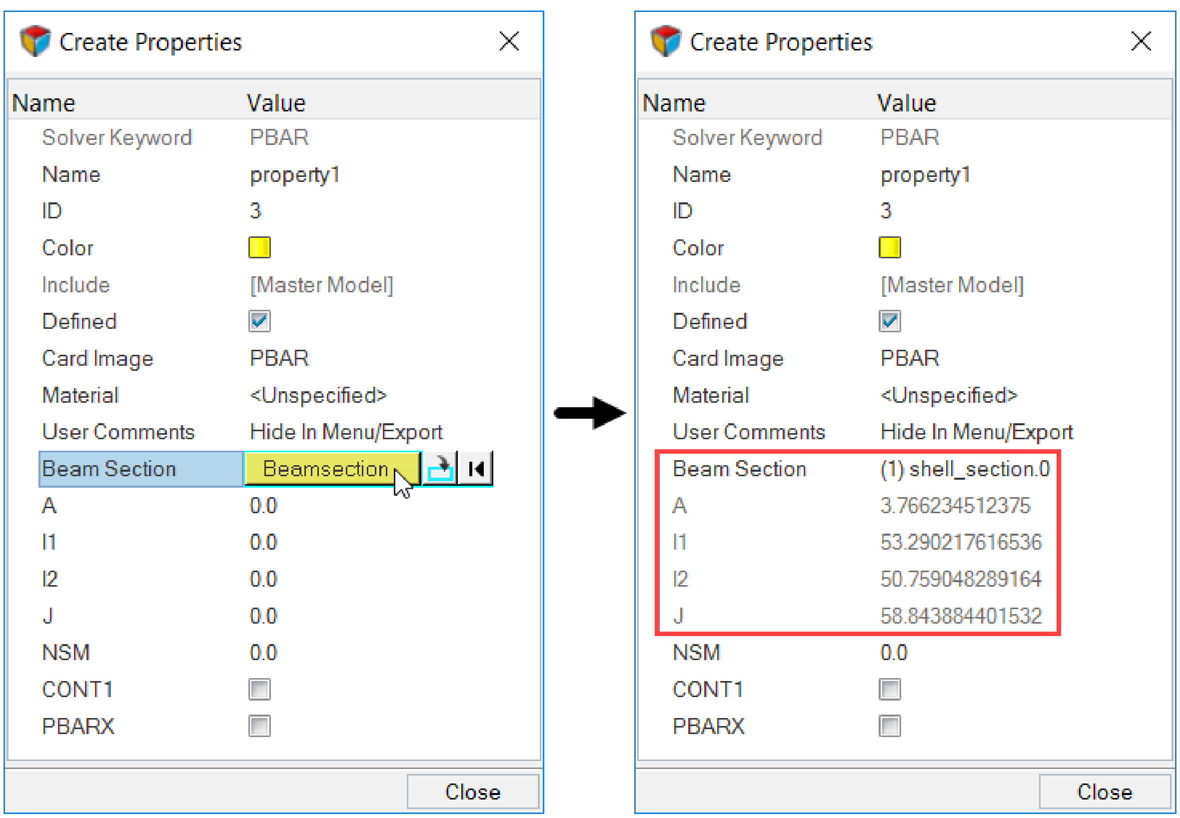


Figure 418:

Every element, including beam, bar and rod elements, must exist within a component. A property can be assigned to a component, or to an individual element. If there is a conflict with properties assigned directly to elements and properties assigned to components, direct element property assignment takes precedence. 1D properties hold the section information such as areas, inertias or even specific dimensions in the case of standard sections. 1D elements hold the orientation and connectivity information. Beamsections hold section geometry information and section calculation data, just the same as a 1D property. In fact when a beamsection is assigned to a property, it will automatically take over the property and fill in the necessary fields. The 3D visualization operates based on the beamsection's stored geometric data. To disassociate a beamsection from a property, right-click the beamsection selector in the property card image.

Import/Export HyperBeam Comments

Overview of how to import and export HyperBeam comments.

- Import HyperBeam comments.

  On import of a FEA model, beamsections are automatically created for all bar, beam, and rod properties that do not already have beamsections defined. These beam sections automatically populate bar, beam, and rod property cards and the cross-section should be edited with HyperBeam. 3D visualization of beams in HyperMesh is only made possible through HyperBeam beamsections and their association to properties.

  To import a FEA model without automatically creating beamsections for each 1D property, use the custom import feature with beamsections and beamsection collectors unchecked. The section information on the actual property cards will remain intact, but the geometric and section calculation data will be missing.

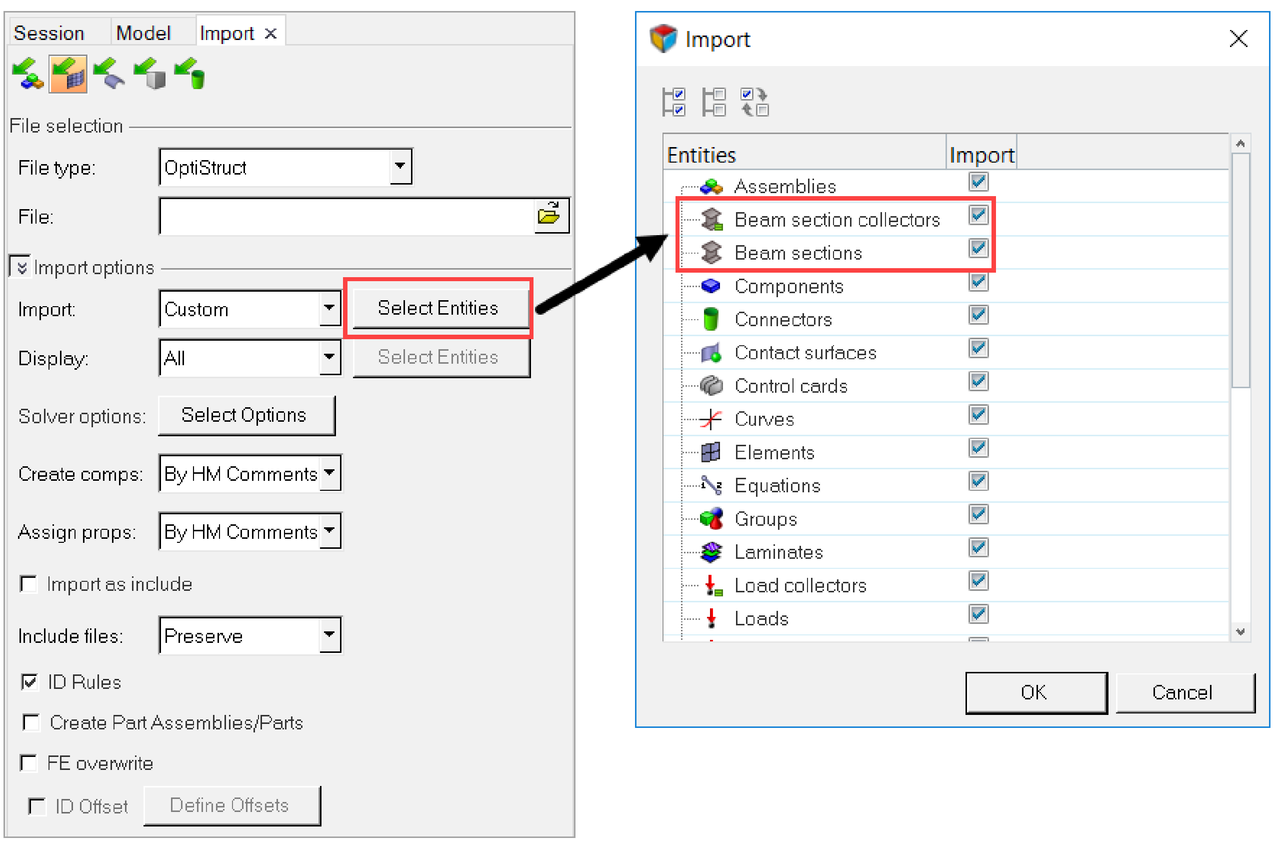


Figure 419:

- Export HyperBeam commetns.

  Exporting a FEA deck in HyperMesh operates in a similar fashion. HyperBeam comments are written out by default (as are all HyperMesh comments). In order to suppress HyperBeam comments from being exported, you must use the **Custom** export feature, or the **Solver option** to turn off all comments. Beamsections are stored as HyperBeam comments in an exported deck.

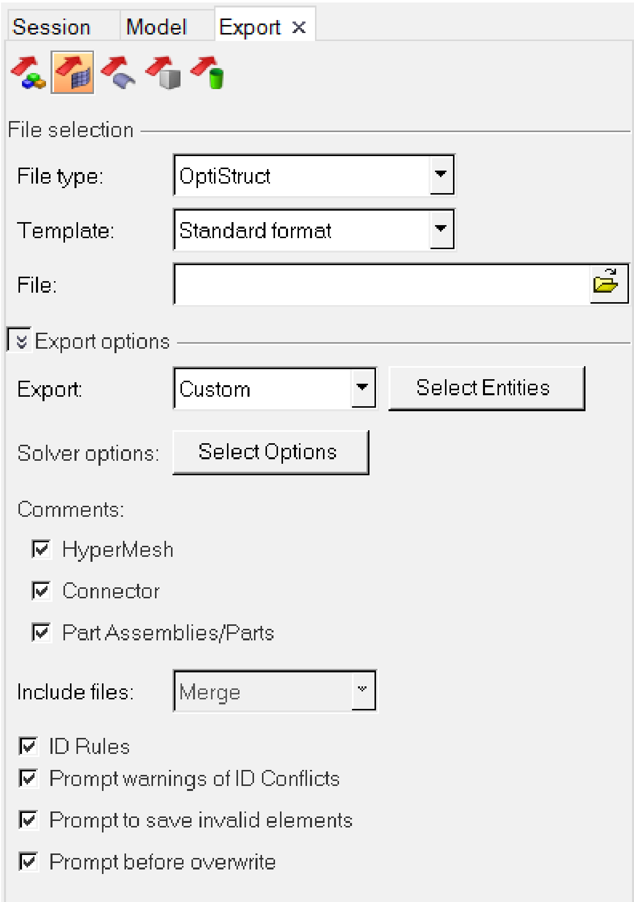


Figure 420:

Example: Import and Automatic Beamsection Creation

This example illustrates the automatic creation of beam sections on import and the visualization of these rods, bars and beams in HyperMesh.

Before you begin, load the OptiStruct user profile and open the `pbeaml.fem` input file from `<installation_directory>\tutorials\hm`.

1. Use a text editor to inspect the following file: `pbeaml.fem`.

```
$$
$$  PBEAMLData
$$
$HMNAME PROP                    1"ROD" 3
$HWCOLOR PROP                   1      11
PBEAML         1       0              ROD                                +
+       5.0     1.0
$$
$HMNAME PROP                    2"TUBE" 3
$HWCOLOR PROP                   2      11
PBEAML         2       0             TUBE                                +
+       10.0    5.0     1.0
```

Figure 421:

There are no beam sections or HyperBeam comments defined in this input file. When imported, HyperMesh automatically creates beam sections which allow you to visualize the sections in 3D.

2. Import `pbeaml.fem` into HyperMesh and make sure the beam visualization ( ) is turned on.

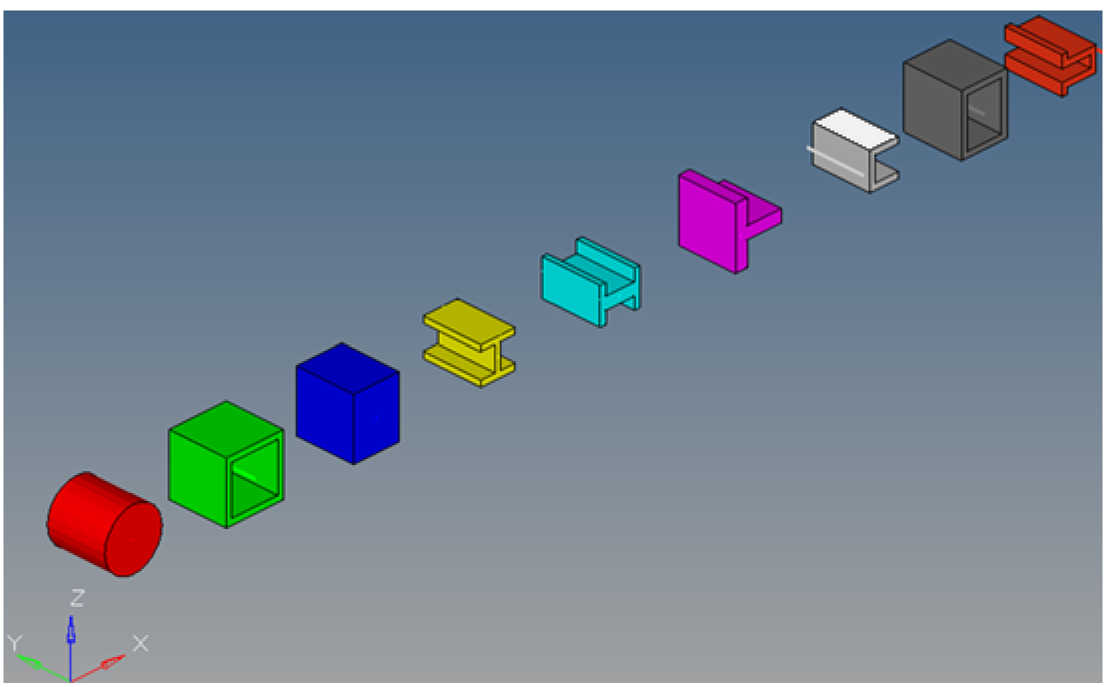


Figure 422:

Direct/Indirect Property View

Filter elements that display in the graphics area based on their property assignments.

When the Property view is active in the Model Browser, the Direct/Indirect Property view ( ) is enabled. From this view you can select one of the following:

Both direct and indirect properties

Direct properties only

Indirect properties only

> **Note:** This feature is not supported by the user profiles for ANSYS, LS-DYNA, Marc, PAM-CRASH 2G, Radioss, Samcef, and any profile in Manufacturing Solutions.

Selecting one of these options immediately filters the view in the graphics area. These filters are accumulative with the current component display state, so, for example, if you have only a few components displayed in the graphics area and the rest are hidden, selecting Direct Properties Only will filter out any elements from the currently displayed set, but will not cause previously-hidden elements to become visible again even if they have direct properties assigned. Similarly, Show, Hide, and Isolate functions work in conjunction with these controls rather than overriding them. If you switch to a different Model Browser view, the effects of your current direct/indirect property view remain.

Selecting any of these view modes automatically hides any non-element entities, such as boundary conditions or morphing domains.

> **Note:** Entities with no property assignments at all will be filtered out of the view by any of these options.

Examples

The simple model shown below (using the properties view) has elements organized into four components, each representing a property state: direct only, indirect only, mixed, and no property. The mixed component consists of three elements with indirect properties and one element with direct properties, but this only becomes apparent when using one of the property views.

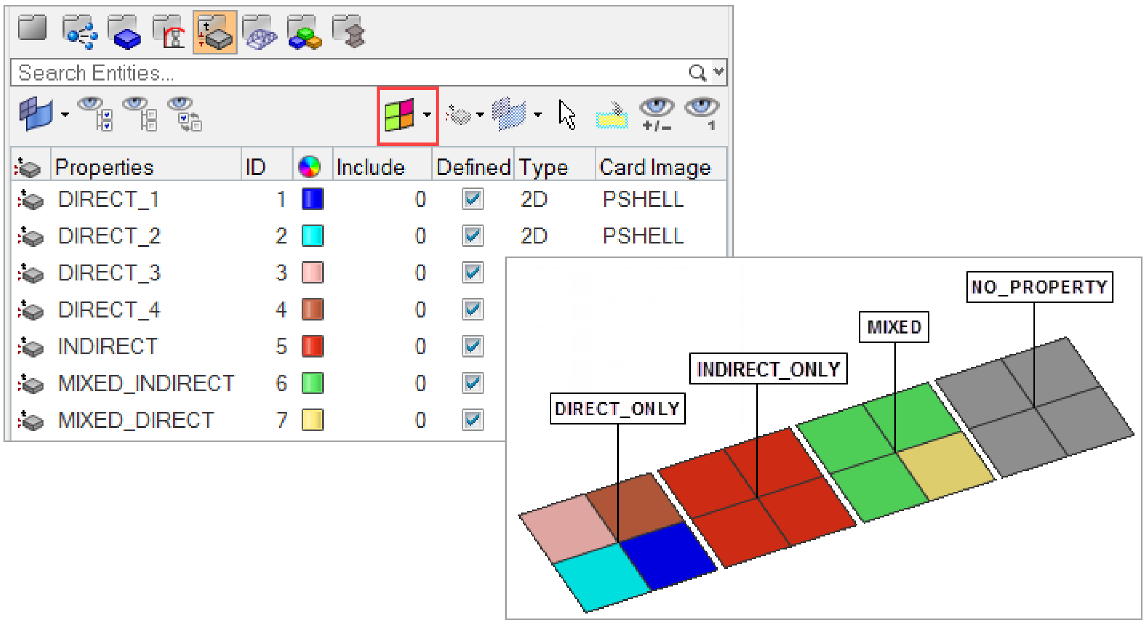


Figure 423:

Property View: Both

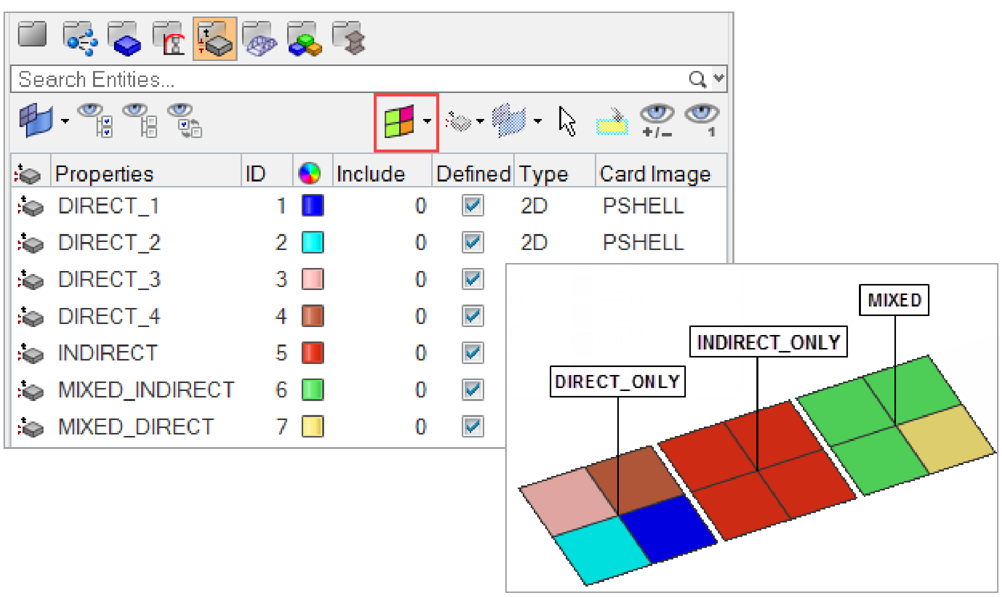


Figure 424:

Property View: Indirect Only

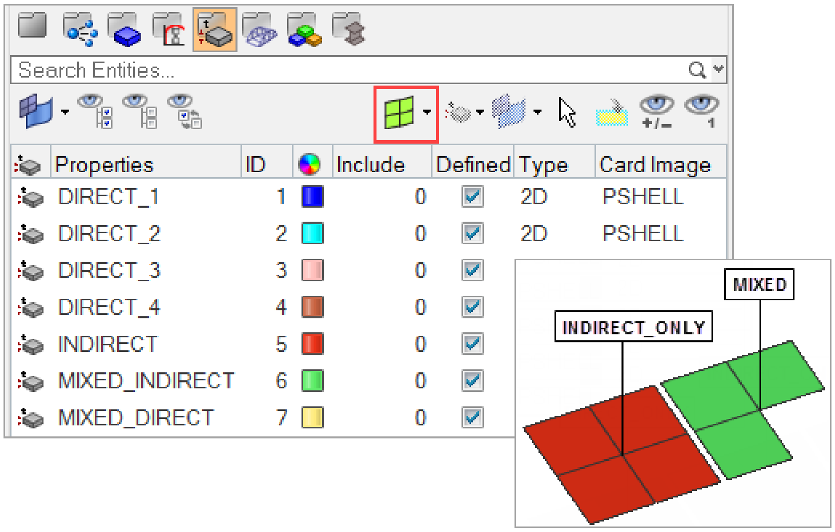


Figure 425:

Property View: Direct Only

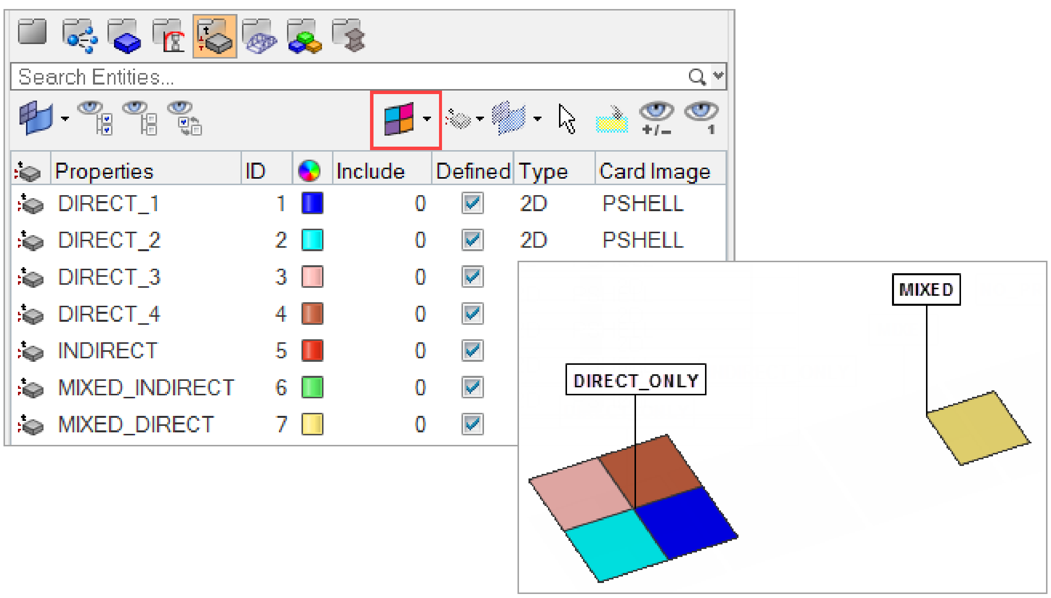


Figure 426:

Direct/Indirect Property Assignment

Define direct/indirect property assignment.

Many solver user profiles include a column called Direct Property when in Component view. This column holds checkboxes for each component.

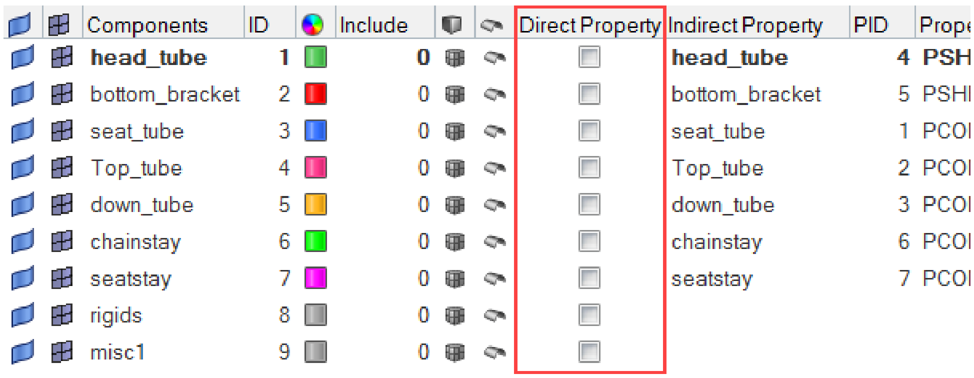


Figure 427:

- When checked, the component uses a direct property assignment.
- When unchecked, the component's direct property assignment is "unassigned" and the component will use indirect property assignment, if available.

The Direct Property column displays for all solver profiles except ANSYS, LS-DYNA, PAM-CRASH 2G, Radioss, and any profile in Manufacturing Solutions.

The Indirect Property column displays for all user profiles except PAM-CRASH 2G and Samcef.

The checkbox may be checked or unchecked based on the type of assignment already defined in the model, but you can change the assignment type by changing the state of the checkbox.

You can check or uncheck multiple components at a time, if you have multiple components selected before changing the state of the checkbox. The exact results depend on a number of factors:

- If you select more than one component and uncheck the DIRECT checkbox for one of them, then all selected components should have their property relationship unassigned.
- If you select more than one component and check one of the DIRECT checkboxes for that selection, then if and only if the INDIRECT properties are common they will be assigned. If there is a mixture of INDIRECT properties, the operation fails because multiple property assignments are not possible.
- If the component has no INDIRECT property, but does have DIRECT property assignment, and you uncheck the checkbox, then the component has NO property assignment. This means that if you then recheck the checkbox, you receive an error stating that no property is available, so automatic direct property assignment is not possible. The checkbox, in this instance, will be disabled until you make an indirect/direct property assignment for the relevant components.

  Like most browser columns, you can sort components by the state of their Direct Property flag.

Context Menu

Access additional Model Browser options from its right-click context menu.

These context menus will appear only where they are applicable hence making them smart and compact.

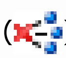
Table 13:

Option	Available for:	Description
Assign	Components	Opens a dialog from which you can assign properties and materials to the selected components. You can view properties by name, name and ID, or ID only.
Card Edit	All	You can edit any single item's card. You can edit multiple items provided that they use identical card images. This option displays the card image of the chosen entity for the current solver template; if a template is not loaded or if the entity does not have any card images associated with the loaded template, an error message displays in the status bar.
Collapse	All Folders	Closes the selected folder(s) and its children folders (if any), so that only the selected folder displays.
Collapse All	White space	Closes all of the folders in the tree structure, so that only the top-most level of items displays.
Configure Browser	All	Opens the Model Browser’s Browser Configuration window where you can specify the entities the browser displays in the tree as well as the columns the browser displays.
Create	White space and the respective folder; user profile valid entity from the list: Assembly, Beam Section Collector, Block, Box, Component, ConstrainedExtraNode, ConstrainedRigidBody, Contact, Contact Surface,	Opens the **Create** dialog, from which you can define and create a new entity. If you are creating a new entity in the Radioss, OptiStruct, Nastran, LS-DYNA, Abaqus, ANSYS, PAM-CRASH, Permas, Samcef, and EXODUS user profiles, HyperMesh creates the new entity and opens it in the Entity Editor. Use the Entity Editor to modify entity data.

Option	Available for:	Description
	Control Volume, Cross Section, Feature, Field, Group, Include File, Laminate, Load Collector, Load Step, Material, Multibody, Output Block, Parameter, Plot, Ply, Property, Region, Rigid Wall, Sensor, Set, System Collector, Transformation, Vector Collector, View, Accelerometer, Positions (Abaqus)	When you create a new Assembly or Include File, HyperMesh assigns it a unique, generic name and adds it to the Model Browser. When you create a View, HyperMesh saves the current model's graphics settings in the browser. Saved views include all of the visible effects of settings such as masks, zoom/pan, X/Y/Z orientation, and whether or not any given component's FE or Geometry is displayed. You can save multiple views (named view1, view2, and so on), and then switch back and forth between them by selecting each view from the browser.
Delete	All except the top-level of Assemblies	Deletes the selected entities. Most items can be deleted. • If a component or multibody is present in more than one assembly in the model, you have the choice of deleting that item from the database or deleting it from the present location only. • If you entirely delete an assembly, and that assembly has children that are not present anywhere else, those children move to the top level. Shortcut: To delete an entity that is selected in the Model Browser, press Delete.
Delete Advanced	Components in the Abaqus, OptiStruct, Nastran, LS-DYNA user profile.	Displays a comprehensive preview of the entities uniquely related to the selected component that can also be deleted.
Drape	Ply	Access to the Drape Estimator and Laminate Tool. • Drape Estimator estimates the draping angles for structural analysis of composite laminates. • Laminate Tool changes fiber angles with respect to an element material system, creates a distribution of drape angle changes, changes the thickness of each ply, and flattens the shape of a ply that needs to be fabricated before laying on the mold.

Option	Available for:	Description
Duplicate	User profile valid entity from the list: Assembly, Beam Section Collector, Block, Box, Component, ConstrainedExtraNode, ConstrainedRigidBody, Contact, Contact Surface, Control Volume, Cross Section, Feature, Field, Group, Include File, Laminate, Load Collector, Load Step, Material, Multibody, Output Block, Parameter, Plot, Ply, Property, Region, Rigid Wall, Sensor, Set, System Collector, Transformation, Vector Collector, View, Accelerometer, Position (Abaqus)	Duplicates a selected entity and opens it in the **Create** dialog. If you are duplicating an entity in the Radioss, OptiStruct, Nastran, LS-DYNA, Abaqus, ANSYS, PAM-CRASH, Permas, Samcef, and EXODUS, and EXODUS user profiles, HyperMesh duplicates the selected entity and then opens it in the Entity Editor. Use the Entity Editor to modify entity data. Shortcut: To duplicate an entity that is selected in the Model Browser, press **Control + D**.
Empty	Assembly, Beam Section Collector, Component, Control Volume (OptiStruct), Group, Load Collector, Load Step, Multibody, Output Block, Set, System Collector, Vector Collector, Accelerometer, Position(Abaqus), Ply, Feature, Field, Rigion, Include file, Cross-Section, Plot, ConstrainedRigidBody.	Preview and delete empty collectors. This operation can be performed on several entity types at the same time. To append entity types to the selection, left-click while pressing **Control**.
Expand	All folders	Opens the selected folders and children in the tree structure, exposing every item nested in the selected folder.
Expand All	White space	Opens all of the folders in the entire tree structure, exposing every item nested at every level.

Option	Available for:	Description
Expand All Includes	Model Browser Include View only	Exports all of the includes with their corresponding content, including the Master model.
Export	Model Browser Component, Material, and Property views.	Export all entity data and attributes display in the browser to a CSV or HTML file.
Export All Self Contained	Model Browser Include View only	Exports all of the model (master model and includes) as self-contained files, ensuring that each exported file contains all of the nodes and systems that are referenced by entities within the given include, as described above (see Export Self Contained option). **Tip:** Activate the **Delete duplicate nodes** and **Delete duplicate systems solver import** options to import a model that consists of multiple Self-Contained includes back into HyperMesh. This will ensure that any duplicate nodes and systems across different includes are merged together appropriately when the entire model is imported. Refer to the Importing Solver Options help for more details. This option is only available in the OptiStruct, Nastran, and Abaqus user profiles.
Export CSV	Model Browser Model View only, and Beam Section Collector folder	Exports a CVS file of all beam section names, each part within the section, corresponding thickness and vertex numbers, and positions. **Note:** Only available for shell sections.
Export an Include	Model Browser Include View only	Exports the contents of the selected include into the chosen file name.
Export Self Contained	Model Browser Include View only	Exports the contents of the selected include into the chosen file name, and ensures that the exported file contains all of the nodes and systems that are referenced by entities within the same

Option	Available for:	Description
		include, even if such nodes and systems belong to different includes. Because self-contained includes contain all of the nodes and systems that are used by entities within it, they can always be imported into HyperMesh on their own without requiring the rest of the model. This option is only available in the OptiStruct, Nastran, and Abaqus user profiles.
Hide	All except Beam Section, Curve, and Optimization Entities	Turns off the entity in the graphics area. This selection affects each item’s local display control, that is, will make the icon become ghosted indicating the display state is off. You can also use this on the entire folder. In such cases, this hides all of the items within that folder (for example all components, and so on).
Import CSV	Model Browser Model View only, and Beam Section Collector folder	Imports new shell beam sections from a HyperBeam CSV file, without having to use a `.hm` file.
Include File Options	Model Browser Include View only	These let you set the options for a selected include. The available options are: **File name** File name to be exported. **Do not export** Review the contents of an include but not export it). Includes that have this flag turned on display the off icon (). **File Path** File path to export the include to (absolute path or path relative to its parent include). **Flag representing the input deck in which the include belongs** This flag is specific to some solvers such as OptiStruct, Nastran, and so on, which subdivide their data deck into various sections such as Bulk Data, Executive Control, or Case Control. For the remaining solvers, this option is not available and does not display.

Option	Available for:	Description
		Include type This flag is specific to LS-DYNA, which subdivides their includes into various types such as Include, Include_Transform, and Include_Compensation_options. **Instance Option** Flag representing the instance relationship between the includes, allowing you to create a copy of your include files. This flag is specific to LS-DYNA, and only applies to Include files of the "Include_Transform" type.
Include XRef	Include file	Any single item or multiple items can be selected. This option opens a new References Browser that displays the relationship of the selected include to other includes in the model in a hierarchical tree structure.
Isolate Only	All except Beam Section, Curve, and Optimization Entities	Isolate Only works like Isolate, except that it affects all entities regardless of type. This option turns off the display of all entities that are not selected.
Isolate	All except Beam Section, Curve, and Optimization Entities	Isolate works locally within a specific entity type; for example, if component(s) are isolated then all display states of other entities, such as load collectors, are unaffected. Isolate displays only the selected entities, and turns off all other entities of the same type.
Make Current	BeamSection Collector, Components, Load Collector, Multibody	Using the pop-up menu, you can make any listed entities current. The currentcollector status is indicated in bold. Any new components, loads, beam sections or multibodies are created within the respective current collector.
Move to Include	Component	Opens the **Move To Include** dialog, from which you can create a self containing include out of selected components inside of an include. Available in the LS-DYNA and Radioss user profile.
Organize	All	Organizes the selected entities into the current include. This option is only enabled in the Include view.

Option	Available for:	Description
		The **Organize** dialog opens when system collectors, vector collectors, load collectors, beamsection collectors, groups, multibodies, and components are being organized into the current include. By default, any additional entities (nodes, elements, systems, vectors, loads and equations, or beamsections) will be moved along with selected parent collectors. If there are entities you do not want to organize along with the selected collector(s), clear their corresponding checkbox.
Organize Include	Include Files (Include View only)	Opens the **Organize Include** dialog, from which you can move selected entities into a new or existing include file. Available only in the LS-DYNA and Radioss user profile.
Part XRef	Include Files (Include View only)	Opens the Part References tab, from which you can review all of the part assemblies and parts in an Include file in a hierarchical view or in a flat list. Within the Part References tab there are two predefined view modes: Hierarchy view and Flat view. The different view modes are located in the top, left corner of the tab.
Realize	Plies	If ply contours are defined by lines, the realize algorithm will identify elements of an underlying mesh. If an element centroid lies within the ply shape defined by lines, it is added to the particular ply. You can manually define plies on lines in HyperMesh or import them from a CATIA Composite Parts Design (CPD) file. Along with this, if a ply has its shape defined in triangulation mesh (in the case of fibersim imported data), that data will be used in projecting on the actual mesh. Three realization/conversion methods are available: **Project Normal to target mesh for CPD/Geom data** If the element centroid projected along its normal lies within the geometrical ply definition, it is associated with this ply. This

Option	Available for:	Description
		method is valid for ply shapes defined in geometry (lines) coming from the CATIA CPD file. **FiberSim drape map by proximity method** This method is similar to Project Normal to target mesh for CPD/Geom data except that instead of ply geometry in terms of lines, it uses the triangulation data from fibersim which is stored in the related table entity. It also considers/maps drape data to the new ply elements/shapes. Once an element is selected for a given ply, drape data closest to its centroid from the ply table (where the drape data is stored) is considered. **CATIA Composites Link drape map by proximity method** Use this method to realize the plies. Advanced options: Three options are available for the search criteria: **Centroid** If the element centroid is inside the ply boundary, the element is added to that ply set boundary. **All nodes inside** The element will be added to a ply set, if all of the nodes are inside the ply boundary. **Shrinkage factor of border element edges** Based on the fraction of an element area intersected by the ply boundary, the element is added to the ply set. **Input Samle Point** Judging whether elements reside inside of a ply boundary is sometimes challenging, especially on a curved surface. Using correct sample points (or nodes) to indicate if the elements are inside helps ply realization. You should use this method for each ply, not for all of the plies. It is recommended to use ply realization for all the plies, and then inspect each ply to make sure it is correct. If the ply realization is wrong, correct that ply only using sample points or interactive nodes.

Option	Available for:	Description
		You can either import the core sample `.csv` file, or interactively pick nodes for a ply.
	Laminates	A laminate can also be realized. In this step, you create and assign properties to elements to translate the ply-based composite's definition into a zone-based one. A template composites property has to be assigned to the elements involved in the composites definition prior to this step. The algorithm copies all settings from this property to the newly generated ones. Each zone of elements with a unique set of plies receives its own property. If there is drape data available on every ply (such as fibersim drape data) then thickness and fiber orientation corrections are applied to each property layer/ply automatically. Currently Laminate Realize with drape data results into one property per element. To open the **Laminate Realize** dialog, right-click on a laminate in the Model Browser, and then select **Realize**.
	Fields	Field realization creates pressures and temperature loads, and maps properties IDs. In order to map the spatially varying values stored in a field entity to the element and node data of the new target mesh you must realize the field entity. Define field realization settings in the **Field Realization** dialog, which opens when you right-click on a field in the Model Browser and select **Realize** from the context menu.
Remove	Components	Removes the component from an assembly if the component is referenced in more than one assembly. **Note:** This does not delete the component, but merely removes one listing of it from the Model Browser tree.

Option	Available for:	Description
Rename	All	You can rename any item in the name text field, but the new name must be unique. All instances of the renamed item update automatically. You can cancel the rename operation by pressing **Esc**. The high-level entity folders are non-editable, but you can rename folders containing the assembly hierarchy.
Replace	Component	Opens the **Part Replace** dialog, from which you can quickly replace a part with either an existing part in your model or a part from an external file. This option is only available in the LS-DYNA user profile
Review	Assemblies, Beamsection collectors, Beamsection, Blocks, Bodies, Boxes, Components, Configuration, Constrained extra nodes, Constrained rigid bodies, Constraints, Contact surfaces, Control volumes, Crosssections, Design variables, Design variable links, Design Objective Reference, Design Variable Property Relationship, Elements, Groups, Joints, Laminates, Load collectors, Loads, Loadsteps, Materials, Mechanisms, Objectives, Optimization constraints, Optimization responses, Output blocks, Part, Part Assembly, Part Set, Plies, Properties, Regions, Seatbelts, Sensors, Sets, System collectors, Systems, Vector collectors, Vectors.	Invokes Review mode, which displays selected entities irrespective of their display state, masked, active state (Entity State Browser), but not outside of the spherical clipping (if enabled).

Option	Available for:	Description
Show	All except Beam Section, Curve, and Optimization Entities	Displays the item in the graphics area. The item's icon changes to bold indicating that the display state is on. You can use the Show option on a folder to display all items within a folder, for example, all components.
Unrealize	Laminates	Unassigns properties previously assigned to elements through the Realizefunction. These properties are not deleted, but merely disassociated from the elements.
Unused	System, Property, Curve, Material, Parameter, Block, Box, Encriptions, Features, Field, Rigion, Contact Surf, Ply, Include file, Set	Preview and delete unused property collectors, material collectors, curves, and so on. This operation can be performed on several entity types at the same time. To append entity types to the selection, left-click while pressing **Control**.
XRef Entities	Assembly, Beam Section Collector, Block, Box, Card, Component, ConstrainedExtraNode, Contact, Contact Surface, Control Volume, Cross Section, Curve, Field, Group, Include File, Laminate, Load Collector, Load Step, Material, Multibody, Output Block, Parameter, Plot, Ply, Property, Region, Rigid Wall, Sensor, Set, System Collector, Table, Transformation, Vector Collector	Any single item or multiple items can be selected. Opens the References Browser and displays the relationship of the selected entities to other entities in the model in a hierarchical tree structure.

Drape Estimator

Use the Drape Estimator to generate, directly into HyperMesh, draping angles and thickness variations resulting from the manufacturing process of fibers associated with plies in a composite laminate.

You can use the drape data generated by this tool, based on inverse mapping, to improve the accuracy of parts modeled with composite materials.

> **Note:** The Drape Estimator is available in the LS-DYNA, Nastran, OptiStruct and Radioss user profiles.

Use the Drape Estimator

To open the Drape Estimator, right-click on a single ply or multiple plies in the Model Browser, and then select **Drape** > **Drape Estimator** from the context menu.

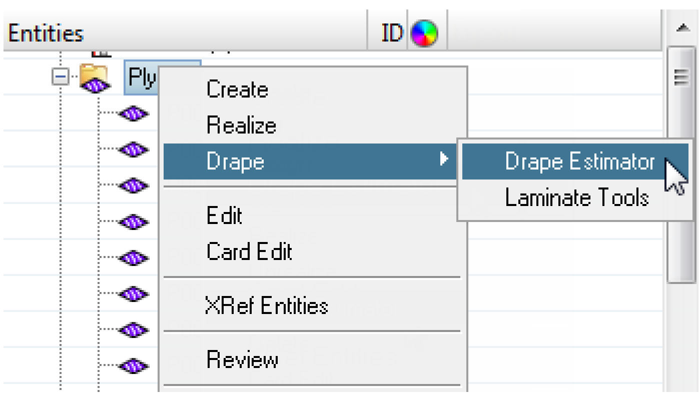


Figure 428: The Drape Estimator is being used to generate drape data for all 11 plies in the model

Once the Drape Estimator has finished generating the drape data, HyperMesh creates a drape table for each selected ply inside the Table folder in the Model Browser.

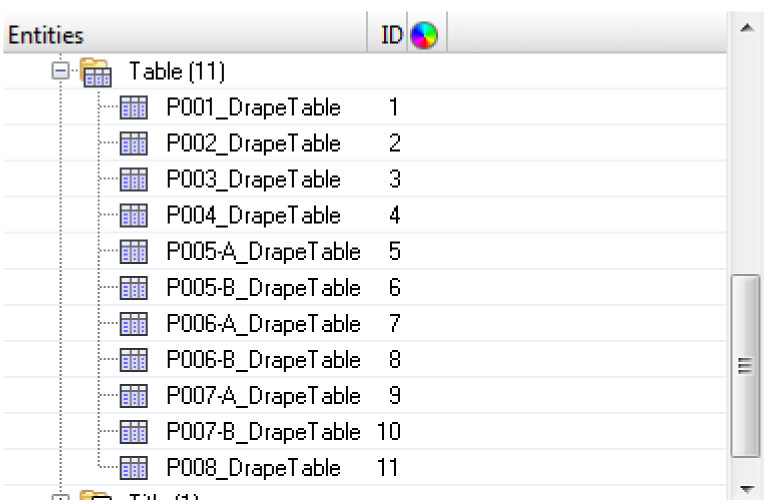


Figure 429: The Drape Estimator generated 11 drape tables for the 11 plies in the model

Review and Edit Data Generated by the Drape Estimator

1. Review and edit the data generated by the Drape Estimator in the Entity Editor by selecting a drape table in the Model Browser.

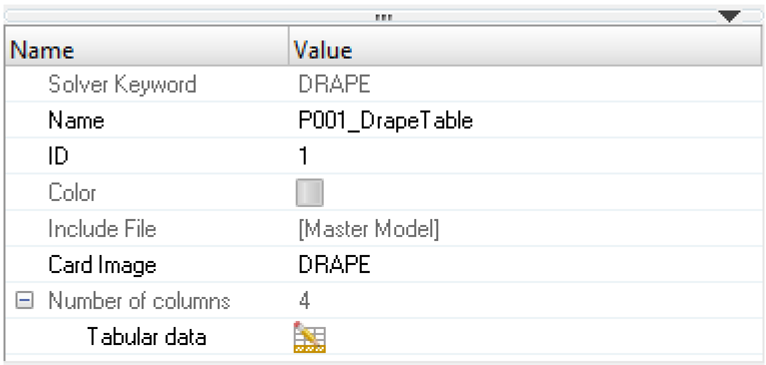


Figure 430: Entity Editor is Activated for the P001_DrapeTabe

2. To review the drape data for this table, click .

   The drape data available in this table include: DTYPE, DID, T, and THETA.

3. Modify this data by clicking a field and entering a new value.

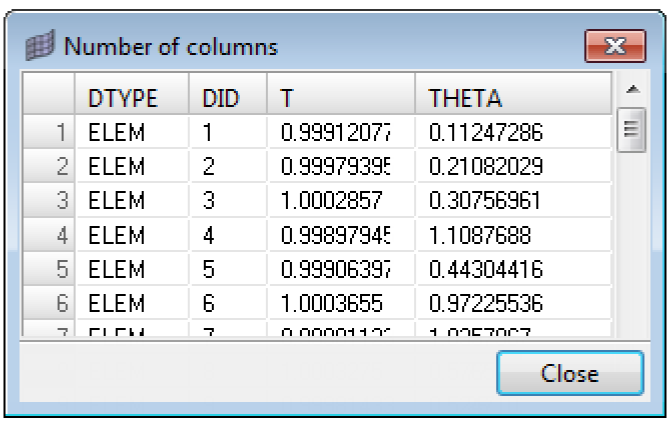


Figure 431:

Example: Drape Estimator

1. In the Model Browser, right-click on the **Ply** folder and select **Edit** from the context menu.

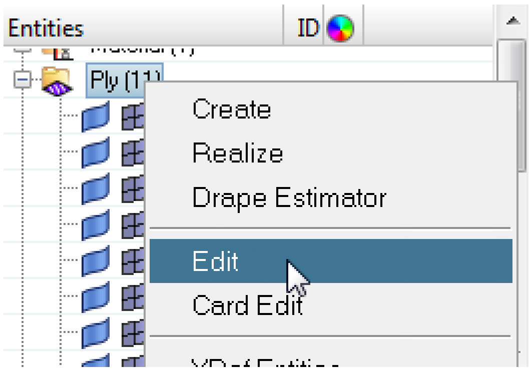


Figure 432:

2. In the **Edit Ply** dialog, select the **Update system** checkbox, and then activate the **systems** selector.
3. Click **Systems** then enter `1` in the id= field.

Figure 433:

4. Click **proceed**.
5. Click **Update**.
6. From the 2D page, click **composites**.
7. Go to the **material orientation** subpanel and activate the **elems** selector.
8. In the elements select window, click **displayed**.
9. Click **Systems** then enter `1` in the id= field.
10. Click **Assign**.
11. Click **Return**.
12. In the Model Browser, right-click on the **Ply** folder and select **Drape Estimator** from the context menu.
    The Drape Estimator generates drape data for the 11 plies in the model.
13. From the menu bar, click **File** > **Export** > **Solver Deck**.
14. In the File field, navigate to your working directory and save the drape data.
15. Click **Export**.
    HyperMesh creates a drape card of the data generated by the Drape Estimator and exports it to the solver deck.

```
$HMNAME OPTITABLEENTRS       10"P007-B_DrapeTable" 15
$
DRAPE          10
+       ELEM           11.000379-.446128
+       ELEM           21.000927.1082319
+       ELEM           31.000995.7028897
+       ELEM           41.000937-1.61453
+       ELEM           5  1.0007-.932679
+       ELEM           61.001489-.145031
+       ELEM           71.001564-.775223
+       ELEM           81.001251.4119438
+       ELEM           91.001256-.220736
```

Figure 434:

Laminate Tool

Use the Laminate tool to calculate fiber angles and thickness changes with respect to an element material system.

This tool creates a distribution table of drape angle changes, change the thickness of each ply, and flatten the shape of a ply that needs to be fabricated before laying on the mold.

Fiber angle changes with respect to a material system when a flat composite sheet is laid on surfaces of a part which is highly curved in bi-directions. This also changes the ply thickness. It is no longer the nominal ply angle (0,45,-45, 90). If the change in angle or thickness is significant, it will lead to a change in the stiffness of a part.

> **Note:** The Laminate tool is available in the OptiStruct and Abaqus user profiles.

1. Create a ply shape (elements) by realizing the plies.

   Each ply must have elements associated with it. If it has been draped already, delete the Table entity associated with that ply.
2. In the Model Browser, right-click on the ply/plies to be draped and select **Drape** > **Laminate Tools** from the context menu.
   The **Laminate Tools Drape** dialog opens.
3. In the Drape Calculation tab, define draping simulation settings.
   a) Using the Application Point selector, select a node to indicate where the ply is first placed on the surface.
   b) Using the Reference Direction selector, select two nodes to indicate the reference direction (zero degree ply direction).

      The starting/initial ply angle is automatically determined based on this reference direction.
   c) Using the Application Direction selector, select a vector to indicate how a ply is placed. If this direction (vector) is not selected, the element normal will be used as the application direction.

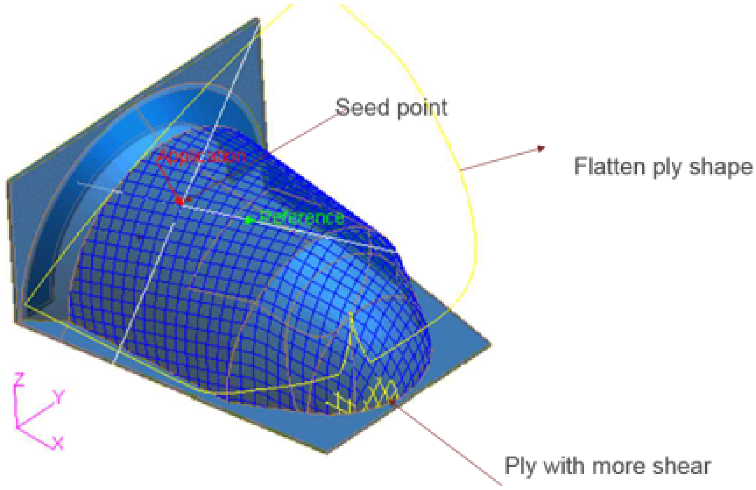


Figure 435:

d) In the Implicit Step field, enter a draping step length.

The average element size at that point could be a good step size.

e) In the Max strain field, enter a value to color the drape lines.

If the % shear is above this value, yellow and red contours will start appearing on the drape line, which indicates you may need to cut the fiber in order to relieve strain/wrinkling.

f) Click **Apply**.

4. In the Review tab, select a review option to read the results.

Option	Description
Drape Lines	Initially an unidirectional ply start with a 90 degree angle between the weft and wrap lines. As the ply is draped over a curved surface, this angle changes. If the shear is more than max strain, than the color changes from blue to yellow to red.

Option	Description

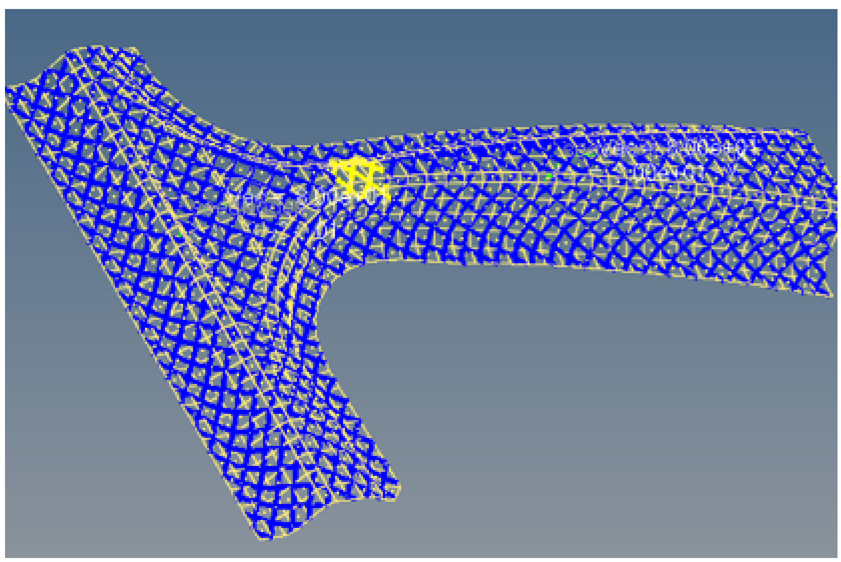


Figure 436:

Flat Lines

Review the flat ply shape that is needed to cover the surface.

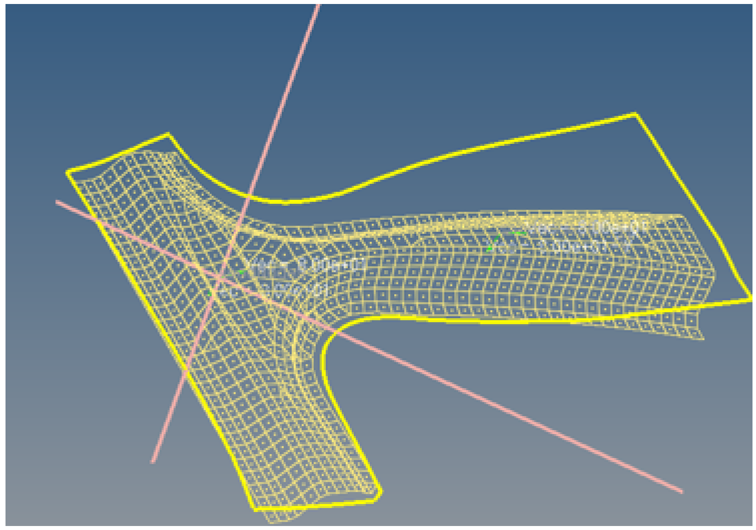


Figure 437:

Drape Thickness

Contour the distribution of thickness changes.

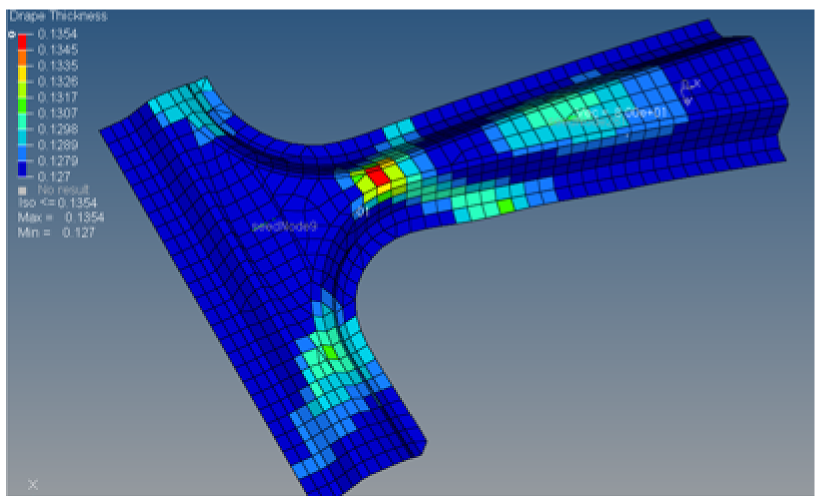


Figure 438:

Drape Orientation (shear)

Contour the shear (angle changes from 90 degrees).

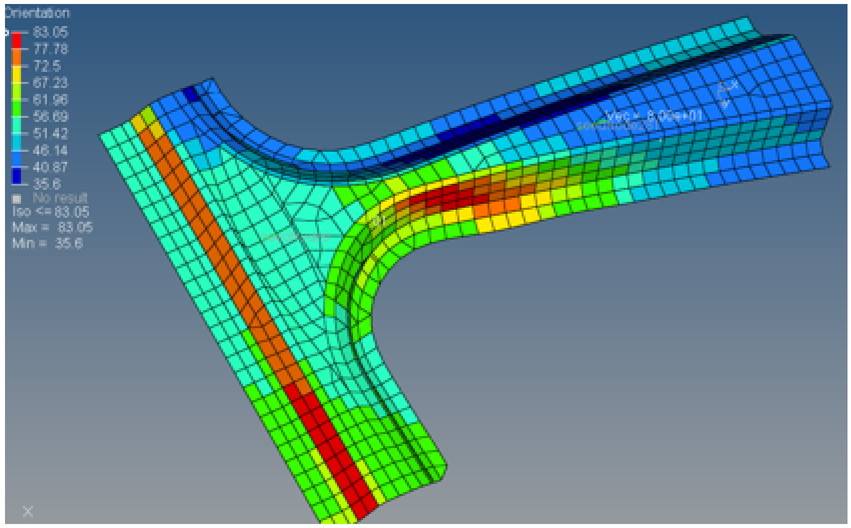


Figure 439:

5. In the Export tab, export the flatten ply shape as geometry (STEP format).

The Flat ply shape can be exported one ply at a time as STEP geometry.

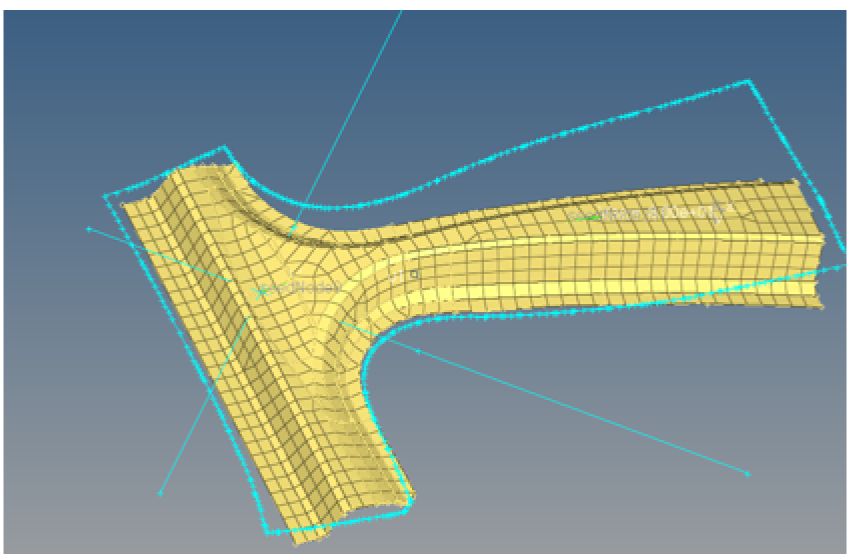


Figure 440:

Part Replacement

Use the Part Replacement tool replace one or more parts with an existing part in your model or a part from an external file.

In ANSYS, surface elements, if created on the component that is being replaced, will not be re-meshed to match the new mesh. Also, for edge to edge contacts on shell components, if those shell components are being replaced, then the contact edge surfaces will not be updated.

> **Note:** Only available for LS-DYNA, OptiStruct, Nastran, Radioss, PAM-CRASH 2G, Abaqus, ANSYS, and all Engineering Solutions user profiles.

Single Part Replacement

1. From the Model Browser, right-click on the component to replace and select **Replace** > **Manual** from the context menu.
   The **Part-Replace** dialog opens and lists all possible entities affected by the part replacement.
2. In the **Replace using** field, select the replacement part.
   - Choose Comp in Model to select a replacement part in the current model using the Component selector.
   - Choose Comp in File to identify the input file that contains the target, replacement part.
3. In the **Tolerance** field, enter a tolerance to search for closest nodes and elements to re-establish the connections and other references between the target part and the model.

   The default tolerance is set at 0.01.

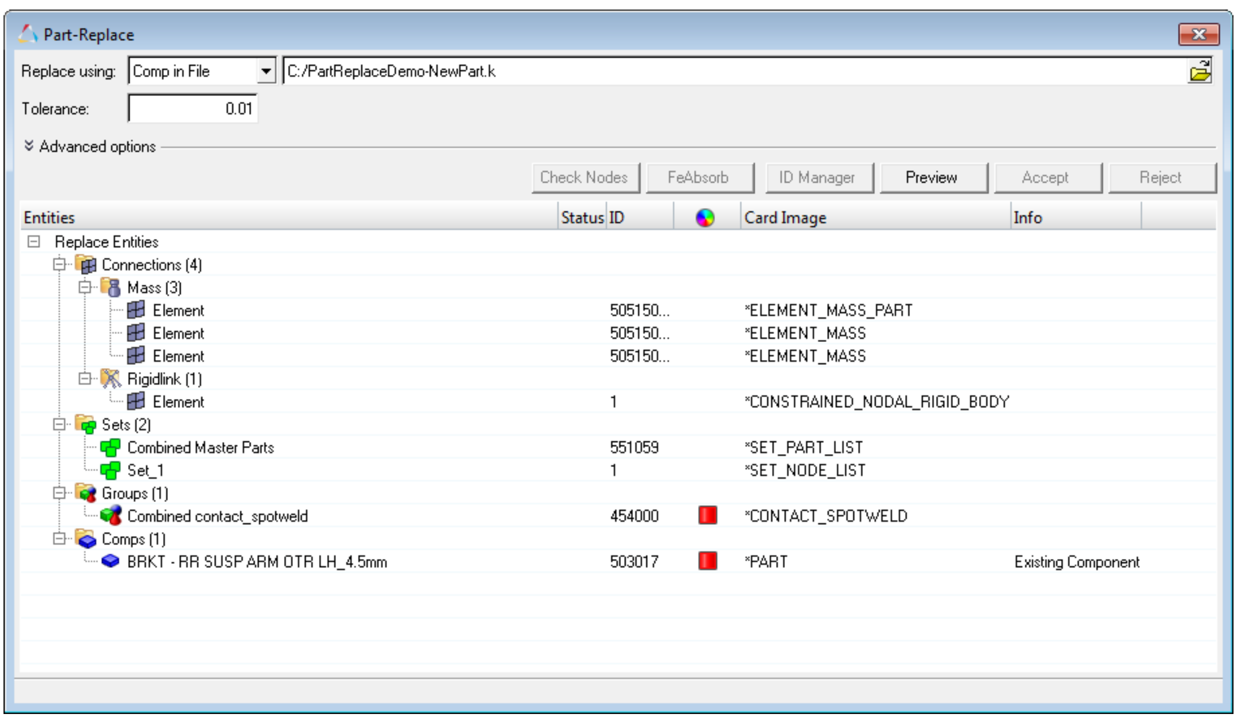


Figure 441:

4. Expand **Advanced options** to select advanced part replacement options.

Option	Description
Copy existing component card image to incoming component	copy the element type card from the source to the component that is being replaced.
Merge Nodes on incoming component boundary	Update node sets and/or boundary conditions when replacing a mesh with a very fine mesh.
Box Approach for Node Sets	**Note:** Available for OptiStruct, Nastran, ANSYS, and Abaqus.

Option	Description
Auto-Preserve the internal connections	Preserve internal connections automatically if there are not connections present in the target file.
Tolerance (bounding box) for connecting the 1D elements from incoming file to source components	Bounding box used to equivalence free 1D elements from the target file to the source components.
Write log file	Write all operations related to the part replacement in a log file.

5. Click **Preview** to show the status of the reconnection of the target part to the model.

> **Tip:** If you change the tolerance, you can click **Preview** to re-examine the effect of the change before accepting it.

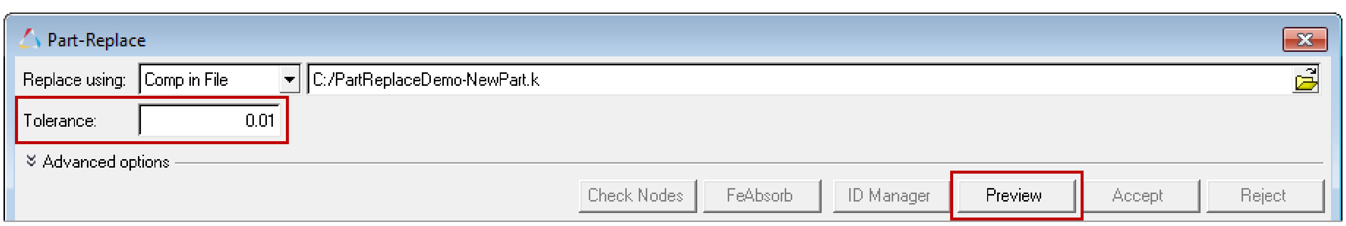
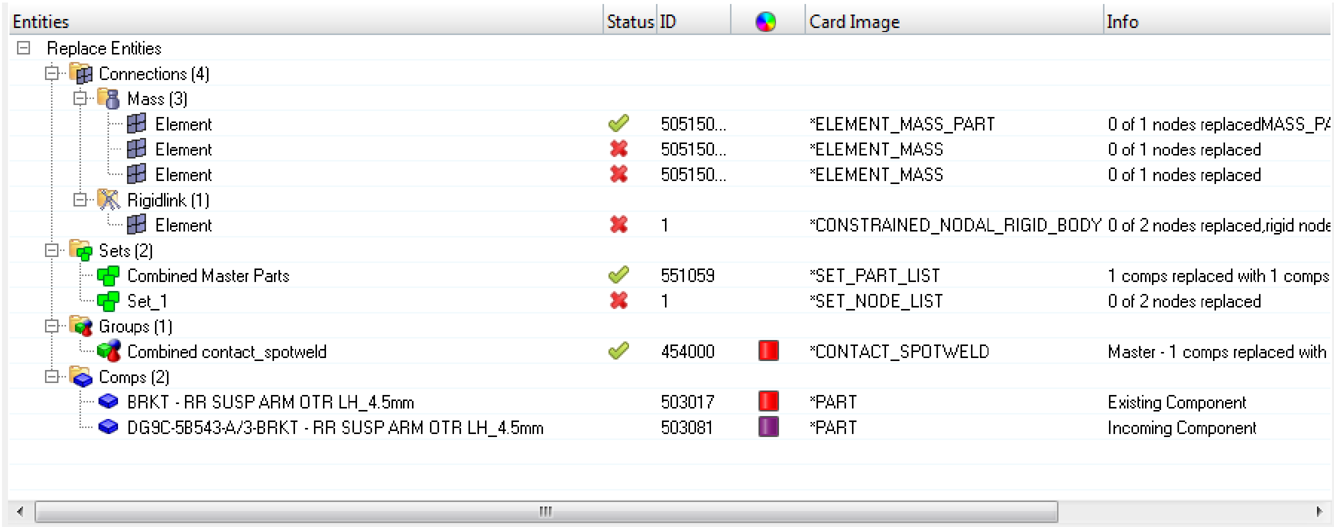


Figure 442:

6. To display connections, click **Check Nodes**.

   Connections are displayed in the Status column and in the graphics area.



Entities	Status	ID		Card Image	Info
Replace Entities					
Connections (4)					
Mass (3)					
Element	✔	505150...		*ELEMENT_MASS_PART	0 of 1 nodes replacedMASS_PA
Element	✖	505150...		*ELEMENT_MASS	0 of 1 nodes replaced
Element	✖	505150...		*ELEMENT_MASS	0 of 1 nodes replaced
Rigidlink (1)					
Element	✖	1		*CONSTRAINED_NODAL_RIGID_BODY	0 of 2 nodes replaced,rigid node
Sets (2)					
Combined Master Parts	✔	551059		*SET_PART_LIST	1 comps replaced with 1 comps
Set_1	✖	1		*SET_NODE_LIST	0 of 2 nodes replaced
Groups (1)					
Combined contact_spotweld	✔	454000		*CONTACT_SPOTWELD	Master - 1 comps replaced with
Comps (2)					
BRKT - RR SUSP ARM OTR LH_4.5mm		503017		*PART	Existing Component
DG9C-5B543-A/3-BRKT - RR SUSP ARM OTR LH_4.5mm		503081		*PART	Incoming Component

Figure 443:

In the graphics area, green connections are highlighted white and red connections are highlighted red.

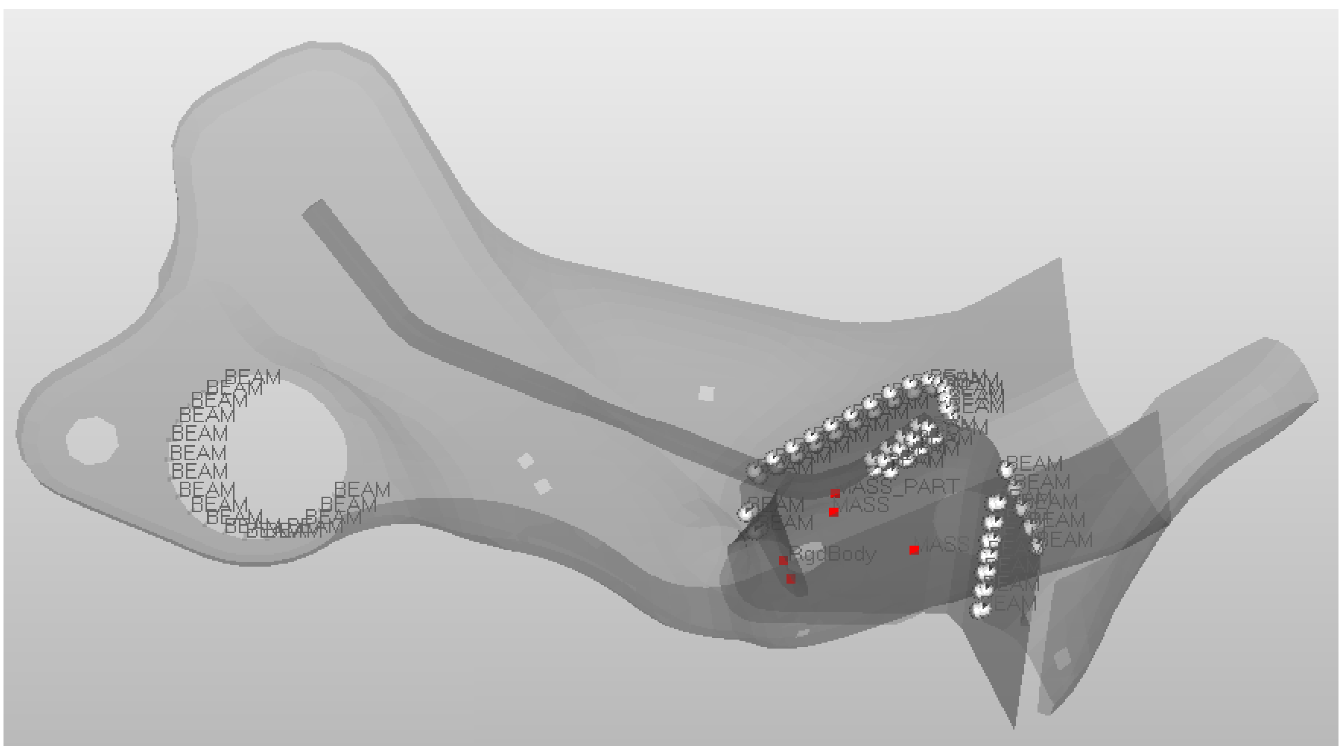


Figure 444:

7. Adjust the tolerance for the target part as needed, then click **Preview** to see the status.

In the example below, the tolerance is changed to `5` and all of the connections are re-established.

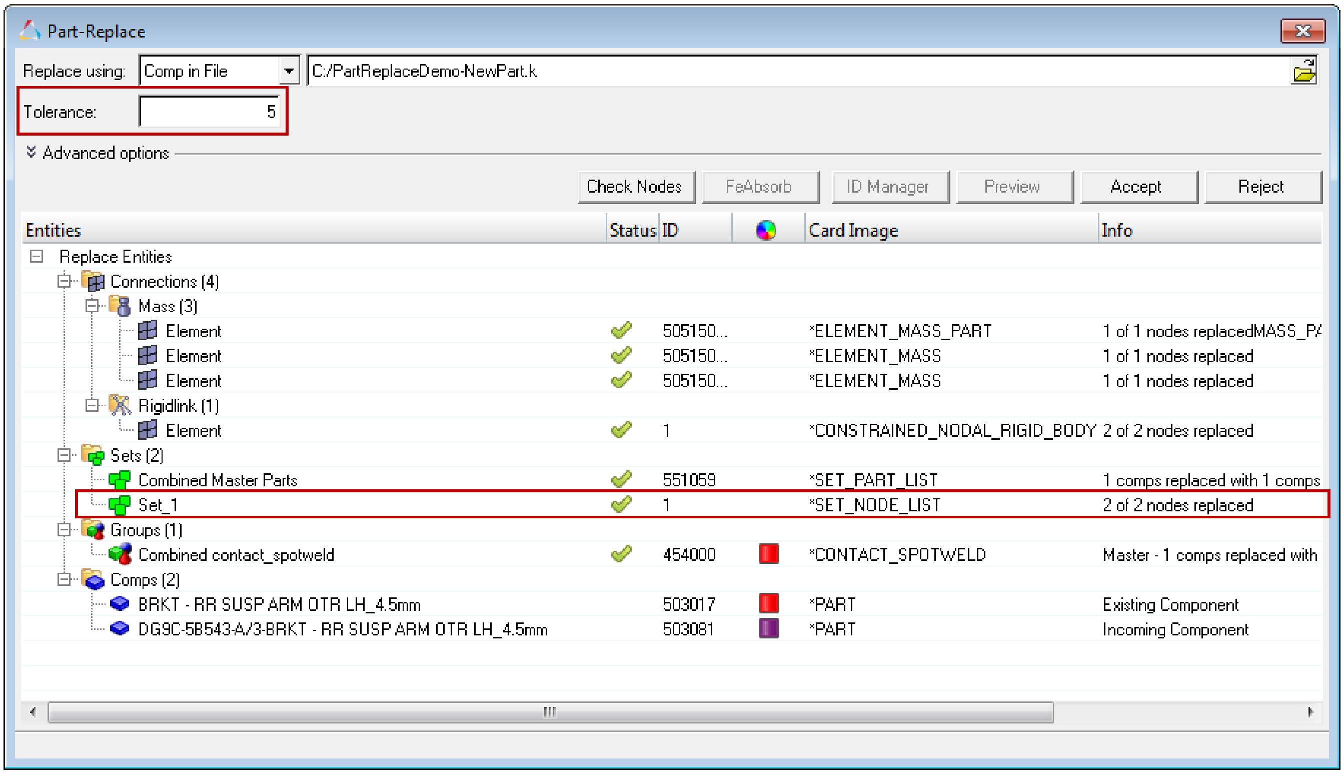


Figure 445:

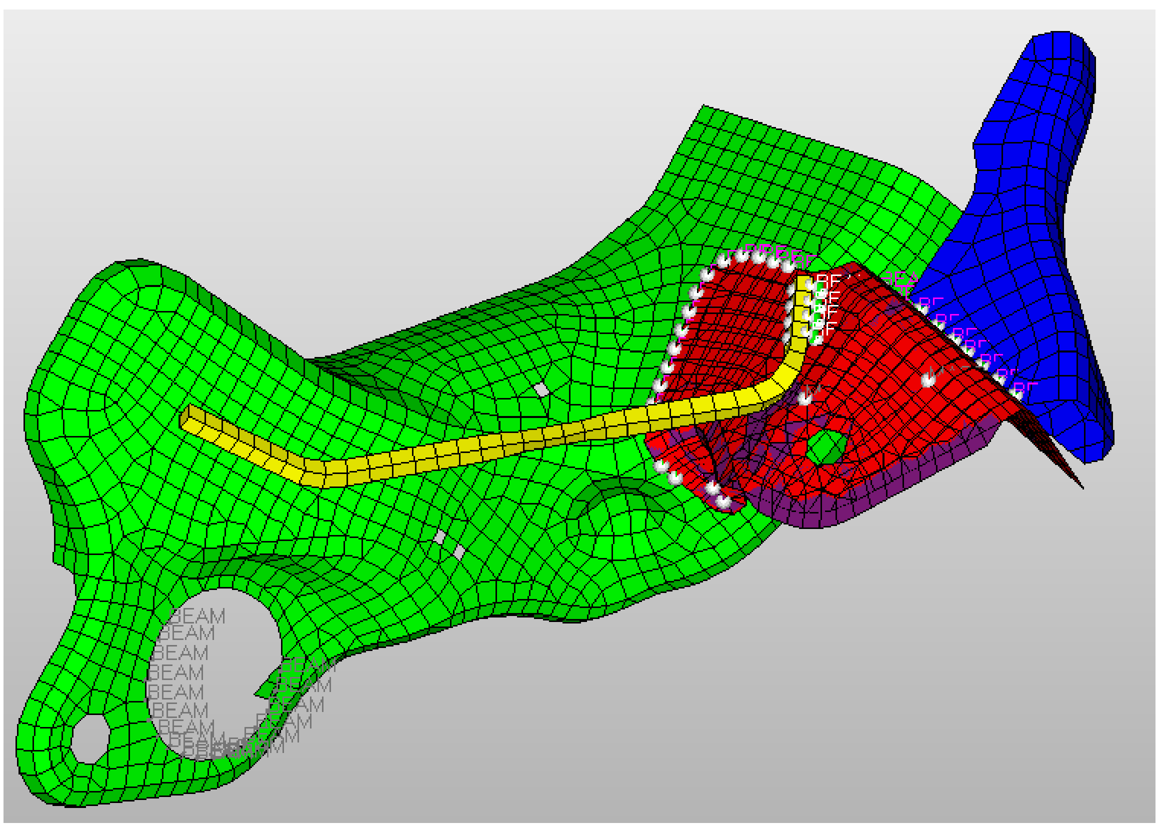


Figure 446:

8. After adjusting the tolerance, click **Accept**.
   The source part is deleted and the connection of the new part to the model is accepted.

   > **Note:** If the source part is connected by meshless welds, clicking **Reject** does not restore the model and welds to their original state prior to the part-replace operation.

Multicomponent Replacement

Multiple components and their corresponding entities can be replaced simultaneously using the Part Replacement tool.

Multicomponent replacement enables you to quickly replace components with a different mesh, replace components with a change in design, or replace components with a different design and internal connections.

After part replacement has completed, if both the source and target have the same name, the Assembly structure and Component names will be retained.

1. From the Model Browser, right-click on the components or Include file that contains the components to replace and select **Replace** > **Manual** or **Automatic** from the context menu.

   Note: When replacing more than one component at a time, all selected components should be organized in one Include file.

   - Choose **Manual** to automatically pair components. You can manually change the component pair and decide which entities to retain post replacement.
   - Choose **Automatic** to automatically pair components based on the bounding box or collision detection approach. You have no control over component pairing.

   The **Part-Replace** dialog opens and lists all possible entities affected by the part replacement.

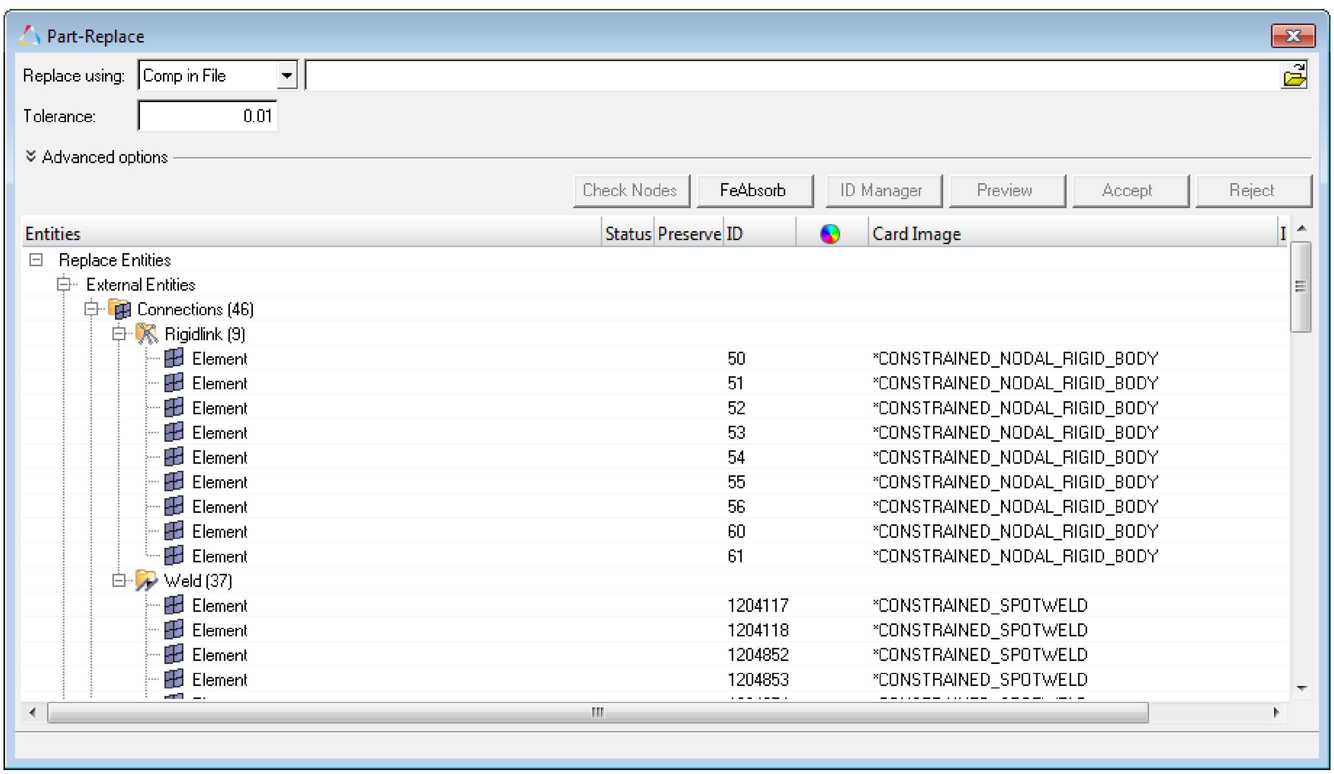


Figure 447:

2. In the **Replace using** field, select the replacement part.

- Choose **Comp in Model** to select a replacement part in the current model using the Component selector.
- Choose **Comp in File** to identify the input file that contains the target, replacement part.

3. In the **Tolerance** field, enter a tolerance to search for closest nodes and elements to re-establish the connections and other references between the target part and the model.

   The default Tolerance is set at 0.01.

4. Expand **Advanced options** to select advanced part replacement options.

Option	Description
Copy existing component card image to incoming component	copy the element type card from the source to the component that is being replaced.
Merge Nodes on incoming component boundary	Update node sets and/or boundary conditions when replacing a mesh with a very fine mesh.
Box Approach for Node Sets	**Note:** Available for OptiStruct, Nastran, ANSYS, and Abaqus.
Tolerance (bounding box) for connecting the 1D elements from incoming file to source components	Bounding box used to equivalence free 1D elements from the target file to the source components.
Auto-Preserve the internal connections	Preserve internal connections automatically if there are not connections present in the target file.
Write log file	Write all operations related to the part replacement in a log file.

5. Optional: To preserve internal entities so that they will not be deleted after part replacement, in the **Preserve** column, enable the checkbox.

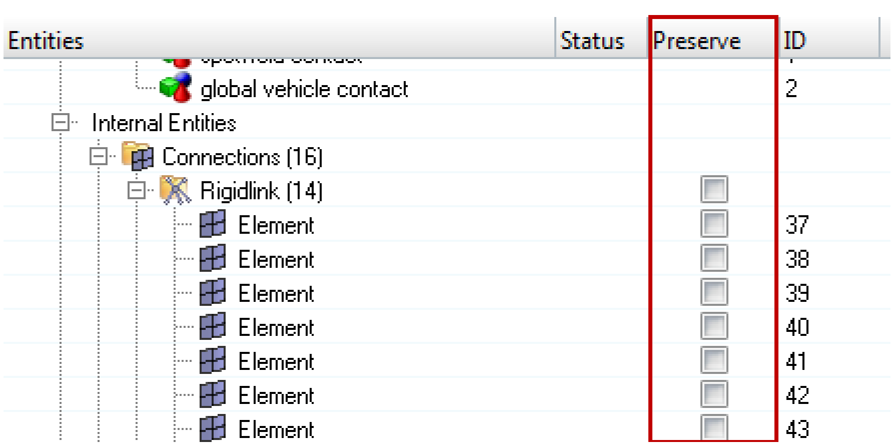


Figure 448:

6. Optional: External and internal preserved connections are preserved automatically during part replacement. To move the absorbed connectors to the current Include file, click **FeAbsorb**.
7. Click **Preview**.
8. If you are manually replacing components in the **Component Pairing** dialog, modify the component pairing and click **OK**.
   a) In the **Incoming** field, select whether to remove the pairing, add a new pairing, select a new pairing, or select to not replace the references.
   b) Review, show (show all), isolate, or reset the components in the pair using the options in the right-click context menu.

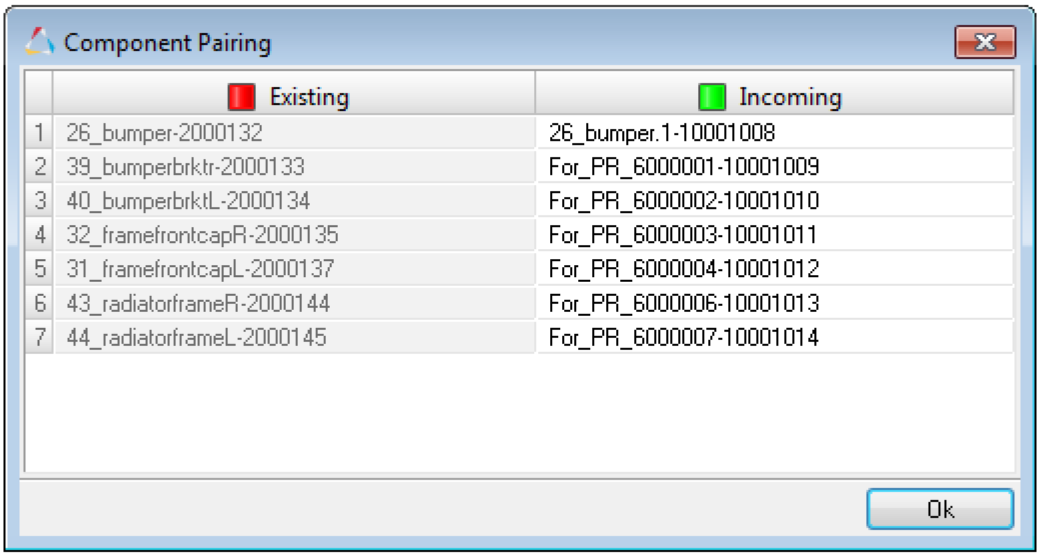


Figure 449:

9. In the **Entities Selection** dialog, specify replacement methods for incoming/existing entities and click **OK**.
   - Choose **Accept incoming entities (existing entities deleted)** to delete all of the internal entities on accept.
   - Choose **Accept existing entities (Incoming entities deleted)** to delete all of the incoming entities on accept.
   - Choose **Merge existing and incoming entities (no entities deleted)** to retain both existing and incoming entities on accept.

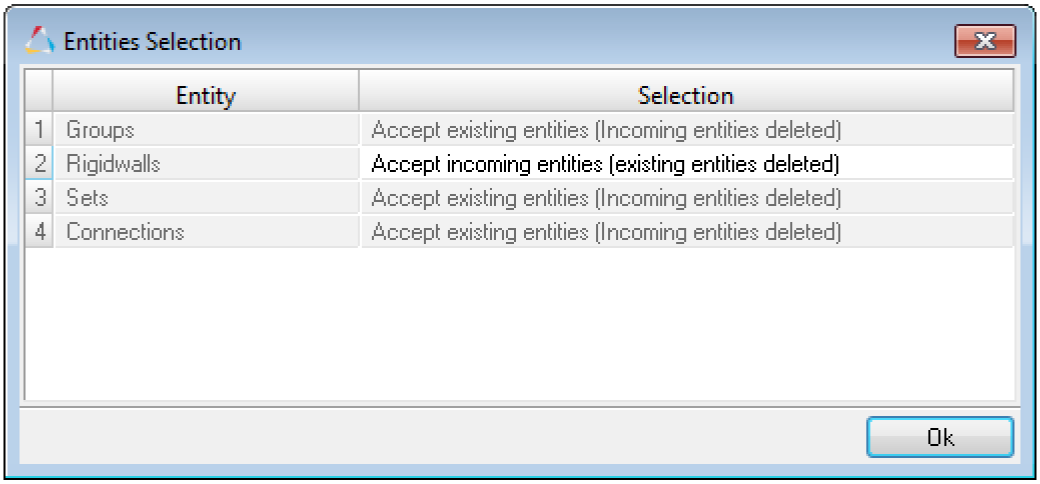
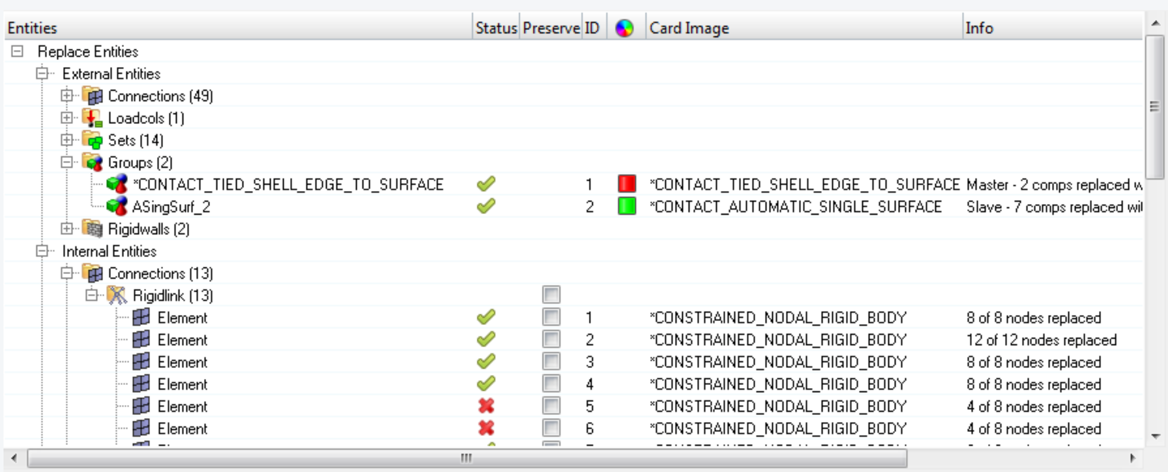


Figure 450:

10. Check the status of each entity.

> **Note:** All internal entities are deleted, and all external entities are updated.
>
> 
>
>
> *Figure 451:*

11. To display connections, click **Check Nodes**.

Connections are displayed in the Status column and in the graphics area. In the graphics area, green connections are highlighted white and red connections are highlighted red.

12. Adjust the tolerance for the target part as needed, then click **Preview** to see the status.

13. To review the ID ranges of existing components, click **ID Manager**.

14. After adjusting the tolerance, click **Accept**.

The source part is deleted and the connection of the new part to the model is accepted.

> **Note:** If the source part is connected by meshless welds, clicking **Reject** does not restore the model and welds to their original state prior to the part-replace operation.

Move to Include Dialog

This dialog opens when you select **Move To Include** from the Model Browser context menu.

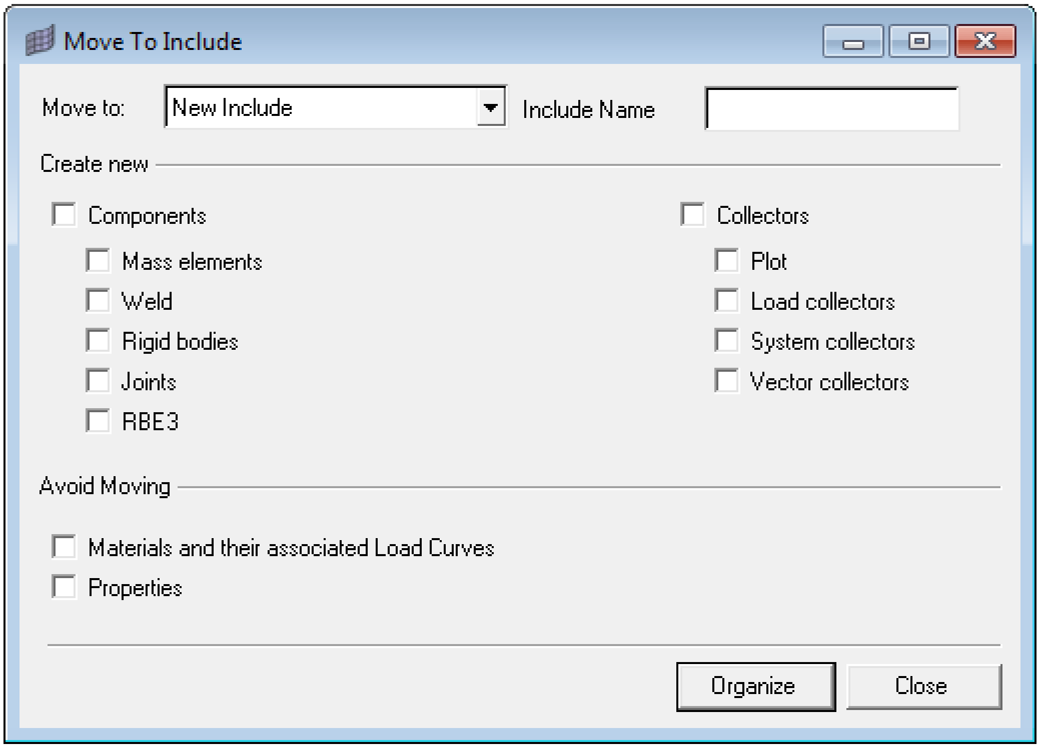


Figure 452: Move to Include Dialog

Several options allow you to specify the details of what is added to the include:

Move to

Use this drop down list box to either select one of the existing include files in the model, or create a new one with the name New Include.

Create New

When entities like mass elements or nodal rigid bodies are moved to the new include as part of selected self-contained entities, they may originate from different components. To help in visualization functions such as Show or Hide, additional options help organize the selected entities by creating new components or collectors to assign the moved entities to.

Choose **Components** to help in visualization for Show/Hide and similar functions when these entities originate from different components, they need to be grouped together in an HyperMesh component. This option creates new comps if only a portion of the content needs to be moved. A new component is created with a prefix Partially isolate. You can use the checkboxes to determine which types of entities will trigger the creation of a new component.

Choose **Collectors** to create new collectors if entities are being moved to the include. You can use the checkboxes to determine which types of entities will trigger the creation of a new component.

Avoid Moving

Use these options to address situations where the you may have a master include file for material and load curves, and do not want some cards to be moved to the include along with their associated entities.

Choose **Material and their associate Load Curves** to not move the materials to the new include even if they are used only by the components being moved.

Choose **Properties** to not move property cards to the include.

Organize Include Dialog

This dialog opens when you select **Organize Include** from the Model Browser context menu.

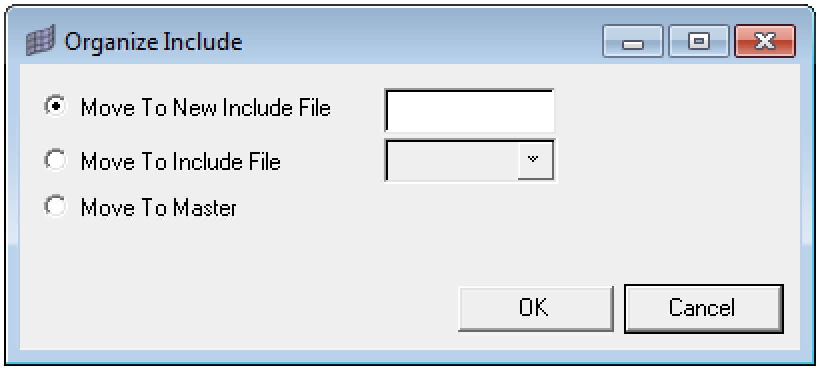


Figure 453: Organize Include Dialog

Use the following options to move selected entities into a new or existing include file.

Move To New Include File
: Create a new Include File to move the selected entities into.

Move To Include File
: Select an existing Include File from the list to move the selected entities into.

Move To Master
: Move the selected entities into the Master Include File.

Laminate Realize Dialog

Laminate realization is used for converting ply based models into zone based models and creating properties for each zone.

The **Laminate Realize** dialog opens when you select **Realize** when right-clicking on a laminate in the Model Browser.

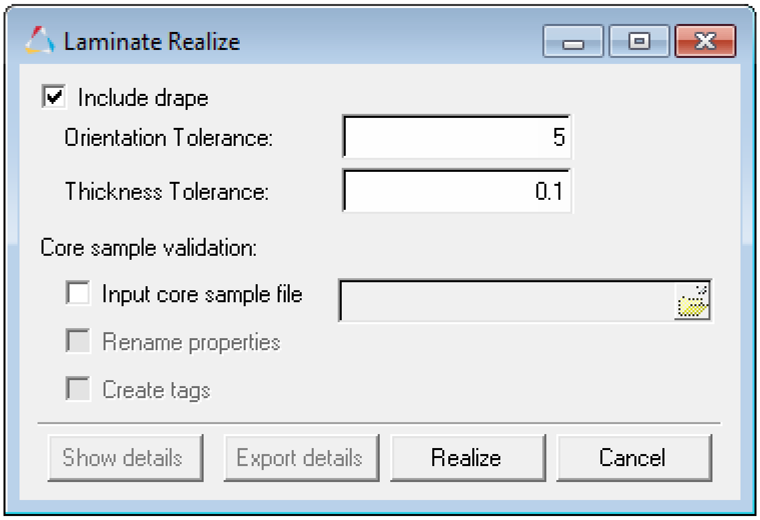


Figure 454: Laminate Realize Dialog

There are several options available in the dialog:

Include drape
: Currently only FiberSim CAD drape data is supported. CATIA –CPD drape data is not yet available. The drape data is exported to OptiStruct, Abaqus and ANSYS solvers.

Orientation and Thickness Tolerance
: When drape data is used from the CAD system, properties of each element can change from one to the other. This may create many properties (one for each element). It is possible to reduce the number of properties created by combining elements with similar properties with a given tolerance. These two tolerances are used in the reduction of the number of properties.

Core sample validation
: Many CAD systems provide core samples (simulating drilling and finding ply stacking sequences at discrete points) to validate composite modeling. You can export this data from CAD systems (mainly CATIA-CPD) and use it in HyperMesh to validate the zone property creation. Using this option, HyperMesh compares the core sample property from a `.csv` file to the actual zone property calculated at that specific location element. A report is also generated for the comparison. Each core sample has a name, and HyperMesh automatically displays it as HM: tag at that location. In order to view the tags, you need to activate **tools/tag/label:body**.

Rename properties
: When activated, this option assigns the property name to match the core sample name.

Configuring the Model Browser

Change the columns and entity types that display in the Model Browser and change browser options.

This dialog opens when you select the **Configure Browser** option from the Model Browser's context menu.

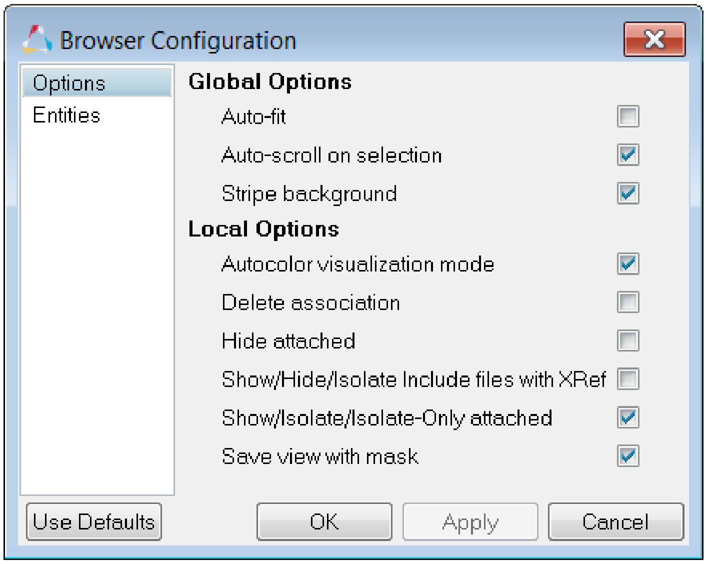


Figure 455: Browser Configuration Dialog

Entities Tab

To select entity types to display in the Model View, activate the checkboxes next to each desired entity type. A check mark indicates that the entity type will display in the browser. You can also use the select all, select none, and select reverse buttons in this mode.

Options Tab

To control various behaviors within the Model Browser, use the following Global and Local options.

Global Options

Autofit
Automatically fits the selected entities to the graphics area whether using the context menu or the Selector, Show/Hide, or Isolate functionality to control the display.

Autoscroll on selection
Automatically opens the folder in the Model Browser, highlights the selected entity in the browser list, and then adjusts the browser list so that the highlighted entity is shown automatically. This functionality is only available when you select an entity using the Selector functionality in the graphics area. If the Autoscroll on selection option is not active, then HyperMesh will continue to open the folders automatically and highlight the entity in the browser list, however, the browser list will not adjust to show the selected entity.

Stripe background
Causes the browser tree to display a gray grid lines in the background, making it easier to distinguish individual lines within the browser. When turned off, the browser background is flat white.

Local Options

Autocolor visualization mode

Change the graphics display based on the browser view mode you have selected. If **Model** or **Component View** is selected, then the visualization mode will change to "By Comp". If **Material View** is selected, then the visualization mode will change to "By Mat". If **Property View** is selected, then the visualization mode will change to "By Prop". If the **Autocolor** visualization mode is not active, then the visualization modes will not change automatically when you are in a certain browser view, instead, you will have to manually change the visualization mode from the Visualization toolbar.

Delete association

Open the **Delete component(s) and unique associations** dialog when **Delete Advanced** is selected from the context menu. When this box is checked off, the Delete component(s) and unique associations dialog does not open when Delete Advanced is selected from the context menu, and the entities uniquely related to the selected component are automatically deleted. Only available for components in the LS-DYNA user profile.

Hide attached

Mask the attached elements from the graphics area when you perform the Hide functionality on the following entities: loads, load collectors, load steps, groups and contact surfaces. This could be observed if the elements are currently displayed. This will not alter the display state to off, that is local display icon will not be Dimmed.

Show/hide/isolate Include files with Xref

Show, hide, isolate the contents of the Include files when you Show/Hide/Isolate the Include file. If you turn this option off, Show/Hide/Isolate will only affect the components and other HM entities that have graphics (vectors, systems, plies, laminates, loads) in the Include file of the Model, Solver, and ID Manger browsers.

Show/Isolate/IsolateOnly attached

Bring the attached elements to the graphics area when you perform the Show/Isolate/Isolate Only functionality on the following entities: loads, load collectors, load steps, groups and contact surfaces. This could be observed if the elements are currently masked or the elements belonging to the component's display state is off. This will also turn the local display icon to Bold if the component is currently with the display state off.

Save view with mask

Command Buttons

Once you finish configuring the browser, click one of the command buttons to close the dialog:

- Click **Apply** to append the new settings without closing the window.
- Click **OK** to keep the new settings including those that are not applied and close the window.
- Click **Cancel** to discard the changes made that are not applied (keeping previous applied settings) and close the window.

Model Checker

Check the solver validity of the model, identify modeling issues, and fix modeling issues in an automatic or manual way.

Using the Model Checker, you can manage the checks as per user requirements, by:

- Interactively creating/editing checks and corrections
- Changing the level of checks
- Organizing checks in custom folders
- Activating/deactivating checks
- Saving/reading Model Check configurations to/from an `.xml` file

The Model Checker can be accessed from the menu bar by selecting **Tools** > **Model Checker**. There are two levels of checks available:

Elements
Check the quality of elements.

Solver
Check your model for errors and warnings.

Create/Edit Checks and Corrections

Create new checks and corrections, and edit default checks and corrections.

HyperMesh supports default checks and corrections in the Model Checker. You can use the Check and Correction entities to create new checks and corrections.

Create Checks

1. In the Model Checker, right-click on an existing check or one of the check folders (ERROR, WARNING, INFO) and select **Create Check** from the context menu.
   The Entity Editor opens and displays entity attributes corresponding to the check.

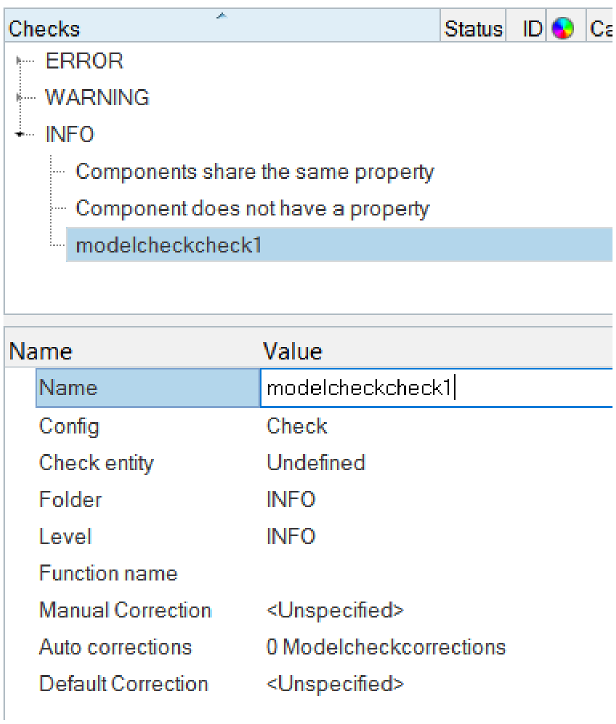


Figure 456:

2. For **Name**, enter a name for the check.
3. For **Config**, select a configuration type.
4. Define attributes.

   > **Note:** The attributes available are dependent on the type of Config selected.

   Common attributes to define include:

Attribute	Description
Check Entity	Type of entity on which the check is applied.
Folder	Directory that contains the definition of the grouping or folder name in which the check is located.
Level	Type of check to perform: ERROR, WARNING, INFO.
Function Name	Core check function to apply for the check. Core check function name can be found in Entity Editor of existing checks.
Auto Correction	Function to perform when automatically performing check correction.
Manual Correction	Function to perform when manually performing check correction.
Default Correction	Default automatic correction associated with the check. By default, the first chosen auto correction is considered the default value if you do not make a selection in this field.

   When Config = TclCheck, define the following attributes:

Attribute	Description
Tcl File	External Tcl script to perform the desired check.

   When Config = GenericCheck, define the following attributes:

Attribute	Description
Number of references	Numbers of references to be defined and exposed in the table entity in which references must be defined.
Number of filters	Number of filters to be defined and exposed in the table entity in which filter rules must be defined.
Number of Value definition	Number of value definition to be defined and exposed in the table entity in which value definition rules must be defined.

Create Corrections

1. Create a Correction in the following ways:
    - In the Model Checker, right-click and select **Create Correction** from the context menu.
    - In the Entity Editor of a Check entity, right-click on a Correction attribute and select **Create** from the context menu.

      Supported Correction attributes include: Manual Correction, Auto Correction, Default Correction.

   If you created a Correction from the Model Checker, a new Correction opens in the Entity Editor, and if you created a Correction from a Correction attribute in the Entity Editor, a new Correction opens in the **Create Modelcheckcorrections** dialog.

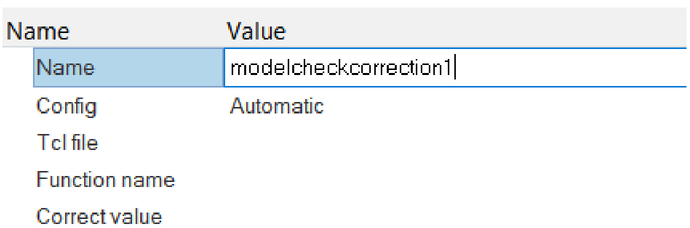


Figure 457:

2. For **Name**, enter a name for the correction.
3. For **Config**, select a configuration type.
4. Define attributes.

> **Note:** The attributes available are dependent on the type of Config selected.

When Config = Automatic, define the following attributes:

Attribute	Description
Tcl File	External Tcl script to perform the desired check.
Function name	Core check function to apply for the check. The core check function name can be found in Entity Editor of existing corrections.
Correct value	Value to be set when performing auto correction.

When Config = Manual, define the following attributes:

Attribute	Description
Option	Tool used to perform a manual correction.

Organize Checks

Organize new and existing checks inside of a user-defined Check Folder.

By default, checks are organized into the ERROR, WARNING, and INFO. You can create your own folder to organize checks in using the Check Folder entity.

ERROR
: Checks with ERROR check-level, identifying modeling issues that need to be fixed to ensure the model to be run by the solver.

WARNING
: Checks with WARNING check-level, identifying modeling recommendations or solver warnings to improve the quality of the model, but are not critical for the model to be run by the solver.

INFO
: Generic information on the content of the model.

1. In the Model Checker, right-click and select **Create Folder** from the context menu.
   A new Check Folder is created, and opens in the Entity Editor.
2. In the **Name** field, enter a folder name.
3. Organize checks inside the newly created Check Folder.
   - Create new checks directly in this folder by right-clicking on the folder and selecting **Create Check** from the context menu.
   - Move existing checks by dragging-and-dropping selected checks into the folder.

Configure Model Checker

Save configurations for future use, import existing configurations into your current HyperMesh session, or append configurations on top of existing configurations.

- Save configurations.
  a) In the white space of the Model Checker, right-click and select **Save Config File** from the context menu.
  b) In the **Save** dialog, specify a file name and directory to save the external `.xml` configuration file and click **Save**.
- Add configurations.
  a) In the white space of the Model Checker, right-click and select **Add File** from the context menu.
  b) In the **Add Config File** dialog, navigate to the configuration file to add and click **Open**.
- Import configurations.
  a) In the white space of the Model Checker, right-click and select **Load File** from the context menu.
  b) In the **Load Config File** dialog, navigate to the configuration file to import and click **Open**.

Run Model Check

Find all entities in your model that failed the active checks in the Model Checker.

1. Open the Model Checker.
2. In the **Active Status** column, select the checkbox of the checks to perform.

   Only active checks will be considered.
3. Run checks.
   - Run all active checks by right-clicking in the white-space of the Model Checker and selecting **Run** from the context menu.
   - Run all active checks in a Check Folder by right-clicking on the folder and selecting **Run** from the context menu.
   - Run a single check by right-clicking on a check and selecting **Run** from the context menu.

   A list of failed checks display, along with the entities in your model that failed the check.

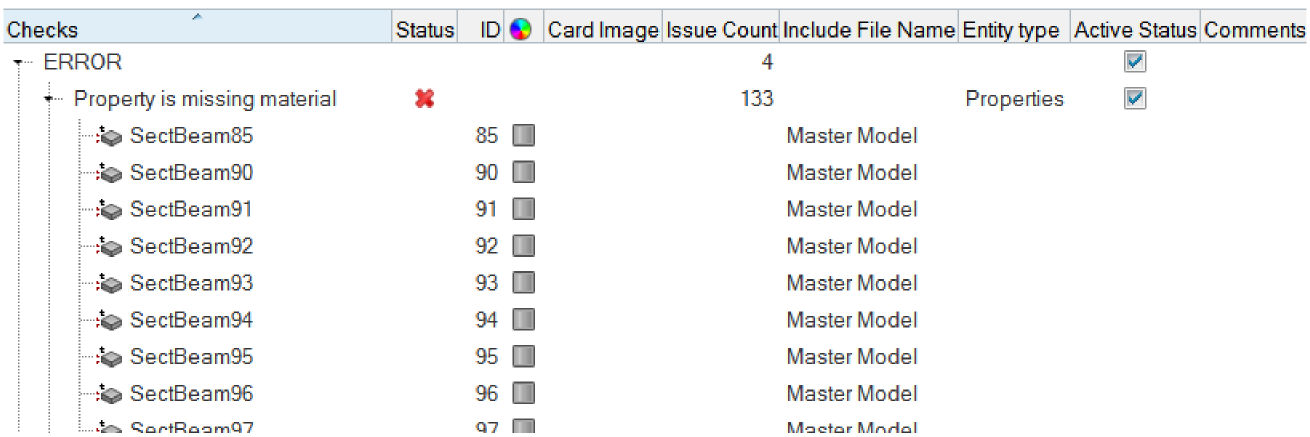
   

   Figure 458:
4. Perform correction.
   - Perform automatic correction by right-clicking on the failed check or failed entity and select **Apply Auto Correction** from the context menu.
   - Perform manual correction by right-clicking on the failed check or failed entity and select **Apply Manual Correction** from the context menu.
5. Optional: Export a detailed report of the checks that were run by right-clicking in the white-space of the Model Checker and selecting **Export Results** from the context menu.

Tip:

- Filter the checks that are displayed by right-clicking in the white-space of the Model Checker and selecting one of the following options from the context menu: **Show only run checks**, **Show only failed checks**, **Show only active checks**, **Show all checks**.
- Show, hide, and isolate failed entities in the graphics area by right-clicking on the failed entity and selecting **Show**, **Hide**, **Isolate only** from the context menu. In case of element/node checks, this action will be perform on all failed entities associated with the check.
- Cross reference failed entities by right-clicking on the failed entity and selecting **XRef Entites** from the context menu.
- Review failed entities in the graphics area by right-clicking on the failed entity and selecting **Review** from the context menu.
- View the failed nodes/elements in a new window provided the failed count is less than threshold limit(default 1000) by right-clicking on the failed elements/nodes and selecting **View** from the context menu.
- Delete selected entities or failed entities by right-clicking on the entity or check and selecting **Delete** from the context menu.

Supported Checks

Supported default checks organized by solver interface.

Abaqus

Errors

Name	Description	Result	Auto correction action	Manual correction action
Property card has NO material card	Checks for properties that are assigned to components or elements, but without a material reference.	Lists all properties that have no materials defined.	Not supported.	Opens the Properties panel
Common nodes of RBodies	Checks if the rigid entities have common nodes.	Lists all of the rigid entities with common nodes.	Not supported.	Opens the Rigid Link Creation panel.
Duplicate 1D Elements	Checks if any 1D elements share the same nodes in the same order.	Lists the duplicate 1D elements.	Deletes the failing elements.	Not supported.
Duplicate 2D Elements	Checks if any 2D elements share the same nodes in the same order.	Lists the duplicate 2D elements.	Deletes the failing elements.	Not supported.
Duplicate 3D Elements	Checks if any 3D elements share the same nodes in the same order.	Lists the duplicate 3D elements.	Deletes the failing elements.	Not supported.
Component with solid and shells	Checks if a component consists of both solid and shell elements.	Lists the failing components.	Not supported.	Opens the Organize panel.

Warnings

Name	Description	Result	Auto correction action	Manual correction action
Negative density	Checks if any materials have negative density defined.	Lists all of the materials that have negative density.	Not supported.	Opens the Card Image panel.
Negative Young's Modulus	Checks if any materials have negative Young's Modulus defined.	Lists all materials that have negative Young's Modulus	Not supported.	Opens the Card Image panel.
Negative Poisson Ratio	Checks if any materials have negative Poisson Ratio defined.	Lists all materials that have negative Poisson Ratio.	Not supported.	Opens the Card Image panel.
Poisson Ratio greater than 0.5	Checks if the Poisson Ratio value for any material is greater than 0.5.	Lists all materials that have Poisson Ratio greater than 0.5.	Not supported.	Opens the Card Image panel.
Unused materials	Checks for any materials that are not referenced by a property.	Lists the unused materials.	Deletes the unused materials.	Not supported.
Material Elasticity is not defined	Checks if Elasticity is defined for the material. This check is applicable for only Isotropic Elastic Material.	Lists materials with no Young's Modulus value.	Not supported.	Opens the Materials panel
Material density is not defined	Checks if density is defined for the material.	Lists materials with no density value.	Not supported.	Opens the Materials panel.
Material Piosson Ratio is not defined	Checks if Piosson Ratio is defined for the material.	Lists materials with no Piosson Ratio value.	Not supported.	Opens the Materials panel.
Material is missing a card image	Checks if the card image is selected for material.	Lists materials with no card image.	Not supported.	Opens the Materials panel.

Name	Description	Result	Auto correction action	Manual correction action
Unused properties	Checks for properties not used in the model.	Lists the properties that are not used	Deletes the unused properties.	Not supported.
Mass Property having zero mass	Checks for Mass elements with zero mass value.	Lists all Mass elements with zero mass value	Not supported.	Opens the Property panel.
Free mass elements	Checks for Mass elements with no property.	Lists all Mass elements with no property.	Not supported.	Opens the Property Creation panel.
Empty components	Checks for components that do not contain elements.	Lists the failing components.	Deletes the empty components.	Not supported.
Contact ghost elements have no base elements	Checks if the contact elements created for interface have a base element.	Lists all interface entities that have a contact element without its base element.	Not supported.	Opens the Interface panel, Add page.
Empty Group Surface	Checks if the surface in the group has elements.	Lists surface groups that are empty.	Not supported.	Opens the Interface panel.
Unused Sets	Checks if any sets are unused.	Lists the unused sets.	Deletes the empty sets.	Not supported.
Empty sets	Checks if the sets are empty.	Lists the empty sets.	Deletes the empty sets.	Not supported.
Unused systems	Checks if any systems in the model are not used.	Lists the unused systems.	Deletes the unused systems.	Not supported.
Unused contact surface	Checks if the contact surface defined in the model is unused.	Lists all unused contact surfaces.	Deletes unused contact surfaces.	Not supported.
	Checks if any beam sections are unused.	Lists all the unused beam sections.	Deletes the unused beam sections.	Not supported.

Info

Name	Description	Result	Auto correction action	Manual correction action
Number of Material	Checks for the number of material entities in the model.	Lists all of the materials.	Not supported.	Not supported.
Number of Properties	Checks for the number of property entities.	Lists all of the properties.	Not supported.	Not supported.
Number of Comps	Checks for the number of component entities.	Lists all of the components.	Not supported.	Not supported.
Number of Groups	Checks for the number of group entities.	Lists all of the groups.	Not supported.	Not supported.
Number of Sets	Checks for the number of set entities in the model.	Lists all of the steps in the model.	Not supported.	Not supported.
Number of Steps	Checks for number of step entities in the model.	Lists all of the steps in the model.	Not supported.	Not supported.

ANSYS

Errors

Name	Description	Result	Auto correction action	Manual correction action	Remarks
Component without HM_COMP	Checks and flags components that do not have a HM_COMP card image.	Lists the components that failed the check.	Supported.	Not Supported.	The HM_COMP card image should be attached to all components except those that have only RBE3 and RIGID.
Component without ET TYPE	Checks and flags components that have a card image, but do not have a sensor attached.	Lists the components that failed the check.	Not Supported.	Opens the Card Image panel.	
ET Type and Element config mismatch	Checks and flags components whose ET Type does not match with any of the element types in the component.	Lists the components that failed the check.	Not Supported.	Opens the Card Image panel.	
ET Type and Real sets or Sections	Checks and flags components that have a property type that does not match the ET Type. Targe169 and	Lists the components that failed the check.	Not Supported.	Opens the Components panel.	The GENERAL property can be attached to

Name	Description	Result	Auto correction action	Manual correction action	Remarks
are not matching	targ170 components can have a property type of any of the related contact types. Checks and flags components that have an ET Type and section that do not match.				componenets that have any ET Type.
Unpaired contacts	Checks and flags components of contact elements which are single. Components are identified based on the property attached to them.	Lists the components that failed the check.	Not Supported.	Not Supported.	
Mass elements without property	Checks and flags MASS21 and/or MASS71 elements if they have no property attached or if the attached property is not of type MASS21p or MASS71p, and if the values related to the mass elements are zero.	Lists the elements that failed the check.	Not Supported.	Not Supported.	It is necessary to define at least one of the mass values on the property.
Improper Entity Set Names	Checks and flags entity sets with a name that is longer than 32 characters, contains special characters other than an underscore, or that does not start with a letter in the alphabet.	Lists the entity sets that failed the check.	Renames the set by removing the special characters, reducing the length of the name to 32 characters, and/or removing the non alphabetic characters from the start of the name.	Not Supported.	Hypen and space are also considered special characters. The Model Checker currently does not check to see if the name already exists.

Name	Description	Result	Auto correction action	Manual correction action	Remarks
Free Nodes	Checks and flags nodes that are not connected to a valid ANSYS element.	Lists the nodes that failed the check.	Deletes the free nodes.	Not Supported.	RBE3, CERIG, and CP are considered equations by ANSYS.
Free 1D elements	Checks and flags 1D elements that are not attached to any structural components.	Lists the elements that failed the check.	Deletes the free 1D elements.	Not Supported.	
Duplicate 1D Elements	Checks and flags 1D elements that are connected between the same two nodes	Lists the elements that failed the check.	Deletes the duplicate 1D elements.	Not Supported.	The order of nodes does not matter.
Duplicate 2D Elements	Checks and flags 2D elements that share common nodes.	Lists the elements that failed the check.	Deletes the duplicate 2D elements.	Not Supported.	
Duplicate 3D Elements	Checks and flags 3D elements that share common nodes.	Lists the elements that failed the check.	Deletes the duplicate 3D elements.	Not Supported.	
Pretension elements without third node	Checks and flags pretension elements that do not have a direction node.	Lists the elements that failed the check.	Not Supported.	Not Supported.	
RBE3, CERIG and CP with free nodes	Checks and flags the RBE3, CERIG, and CPs with free, independent and dependent nodes.	Lists the RBE3, CERIG and CPs that failed the check.	Not Supported.	Not Supported.	The nodes need to be connected to a valid ANSYS element.
Equation with free nodes	Checks and flags equations that have nodes which are not connected to elements. RBE3 and	Lists the equations that failed the check.	Not Supported.	Not Supported.	The nodes need to be connected to a valid

Name	Description	Result	Auto correction action	Manual correction action	Remarks
	RIGIDS are considered equations in ANSYS.				ANSYS element.
Invalid pilot node sets	Checks for pilot node sets that have more than one node.	Lists all pilot node sets that have more than one node.	Not Supported.	Opens the Entity Edit: Sets dialog.	
Invalid contact surfaces	Checks for contact surfaces that have mixed surfaces. Cases of mixed surfaces includes: • Any shell edge + any solid face • First order shell edge + second order shell edge • First order solid face + second order solid face	Lists all contact surfaces that have mixed surfaces.	Not Supported.	Opens the Entity Edit: Contactsurfs dialog.	
Incomplete contact pairs	Checks for contact groups that have one or more of the following conditions: • Missing both contact and target surface attachments. • Missing a contact or target surface. • Missing an ET Type for either a contact or target surface. • No property card is assigned.	Lists all contact groups that fail the check.	Not Supported.	Opens the Entity Edit: Groups dialog.	

Name	Description	Result	Auto correction action	Manual correction action	Remarks
Invalid contact pairs	Checks for contact pairs that have the following: **1.** There is a mismatch of Et Type (sensor) and contact/ target surfaces, and the correct match of the sensor and contact/target surface are in the given below. **2.** The Contact sensor type and property type in the contact pair are different. **3.** The contact is a solid face, and the target is a shell edge or vice versa.	Lists all contact pairs that fail the check.	Not Supported.	Opens the Entity Edit: Groups dialog.	

Warnings

Name	Description	Result	Auto correction action	Manual correction action	Remarks
Unused ET Types	Checks and flags the ET Types that are not referenced by any components.	Lists the ET Types that failed the check.	Deletes the unused ET Types.	Not supported.	
Unused materials	Checks and flags the materials that are not referenced by any of the entities in the model.	Lists the materials that failed the check.	Deletes the unused materials.	Not supported.	
Material without E	Checks and flags materials if EX_FLAG, EY_FLAG, or EZ_FLAG is not activated, and if at least one of the array values is non zero.	Lists the materials that failed the check.	Not supported.	Opens the Card Image panel.	
Material without density	Checks and flags materials if DENS_FLAG is not activated, or if any of the array values are not assigned.	Lists the materials that failed the check.	Not supported.	Opens the Card Image panel.	
Unused Real Sets (properties)	Checks and flags the properties that are not referenced by any of the entities in the model.	Lists the properties that failed the check.	Deletes the unused properties.	Not supported.	

Name	Description	Result	Auto correction action	Manual correction action	Remarks
Shell properties or sections without thickness	Checks and flags properties that have the first thickness as zero. If it is a composite property, then the first thickness on each ply should be greater than zero. Checks and flags shell sections that do not have a thickness assigned on any of the plies.	Lists the properties and beam sections that failed the check.	Not supported.	Opens the Card Image panel.	
Mass properties without mass	Checks for properties with a mass card image, but no mass.	Lists all properties with a mass card image, but no mass.	Not supported.	Opens the Card Image panel.	
Empty components	Checks and flags the empty components.	Lists the components that failed the check.	Deletes the unused components.	Not supported.	
Component with missing material	Checks and flags the components that do not have a material attached. This is checked irrespective of whether there is a sensor attached to it or not.	Lists the components that failed the check.	Not supported.	Opens the Components panel.	
Empty sets	Checks and flags empty sets in the model.	Lists the sets that failed the check.	Deletes the unused sets.	Not supported.	

Name	Description	Result	Auto correction action	Manual correction action	Remarks
Bar Elements with improper orientation	Checks and flags bar elements with an orientation vector that is not perpendicular to the z axes.	Lists the elements that failed the check.	Not supported.	Not supported.	Bar elements in ANSYS are created so that the beam's y-axes is parallel to the global XY-axes.
Unused Sections	Checks and flags sections (beamsectcols) that are not referenced by any components.	Lists the beamsectcols that failed the check.	Deletes the unused beamsectcols.	Not supported.	
Unused Contact surfaces	Checks for contact surfaces that are not used in a contact pair.	Lists all contact surfaces that are not used in a contact pair.	Deletes unused contact surfaces.	Opens the Interfaces panel.	
Unused Contact masses	Checks for contact mass node sets that are not used in a contact pair.	Lists all contact mass node sets that are not used in a contact pair.	Deletes unused contact masses	Opens the Interfaces panel.	
Unused Pilot nodes	Checks for pilot node sets that are not used.	Lists all pilot node sets that are not used.	Deletes unused pilot nodes.	Opens the Interfaces panel.	

Info

Name	Description	Result	Auto correction action	Manual correction action
Components without property or section	Checks and flags components that have an ET Type attached which require, but do not have, a property or section.	Lists the components that failed the check.	Not supported.	Not supported.
SOLVE without /solu card	Checks and flags SOLVE cards to see if it is activated without the /solu card.	Lists the SOLVE control card that failed the check.	Not supported.	Not supported.
LSSOLVE without /solu card	Checks and flags the LSSOLVE card to see if it is activated without the /solu card.	Lists the LSSOLVE control card that failed the check	Not supported.	Not supported.
Properties without card image	Checks for properties that do not have a card image attached.	Lists all properties that do not have a card image attached.	Not supported.	Not supported.
Empty Contact Surfaces	Checks for contact surfaces that do not contain a single element surface.	Lists all contact surfaces that do not contain a single element surface.	Not supported.	Not supported.
Empty Contact Masses	Checks for node sets that have a contact mass card image, but do not contain nodes.	Lists all node sets that have a contact mass card image, but do not contain nodes.	Not supported.	Not supported.
Empty PilotNode Set	Checks for node sets that have a pilot node card image, but do not contain nodes.	Lists all node sets that have a pilot node card image, but do not contain nodes.	Not supported.	Not supported.

LS-DYNA

Errors

Name	Description	Result	Auto correction action	Manual correction action
Id of material out of bounds	Checks if there are any material ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the materials with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Sigy and Eps null in Matl24	Checks if SIGY is defined on Mat_024. If not, it checks to see if at least one EPS value is defined on it.	Lists all MAT_024 with zero SIGY and zero EPS values.	Not supported.	Opens the Entity Editor of the material entity.
Id of property out of bounds	Checks if there are any property ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the properties with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Pida or Pidb is not defined for Joint Stiffness	Checks if Pida and Pidb are defined for joint stiffness.	Lists all joint stiffness whose PIDA/PIDB is not defined.	Not supported.	Opens the Entity Editor of the property entity.
Pida and Pidb same in Joint Stiffness	Checks if Pida and Pidb are the same for joint stiffness.	Lists all joint stiffness whose Pida/Pidb are the same.	Not supported.	Opens the Entity Editor of the property entity.
Parts with non rigid material in Joint stiffness	Checks if both parts, PIDA, and PIDB in Joint Stiffness refer to the same component.	Lists all Joint stiffness properties with PIDA=PIDB.	Not supported.	Opens the Entity Editor of the property entity.

Name	**Description**	**Result**	**Auto correction action**	**Manual correction action**
Id of element out of bounds	Checks if there are any element ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the elements with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Duplicate 1D elements	Checks if two or more 1D elements are connected between the same nodes.	Lists all of the duplicate 1D elements.	Deletes failing elements.	Not supported.
Duplicate 2D elements	Checks if two or more 2D elements are connected between the same nodes.	Lists all of the duplicate 2D elements.	Deletes failing elements.	Not supported.
Duplicate 3D elements	Checks if two or more 3D elements are connected between the same nodes.	Lists all of the duplicate 3D elements.	Deletes failing elements.	Not supported.
Element mass with no mass	Checks if mass is defined on the element_mass.	Lists all element_mass with zero mass.	Not supported.	Opens the Entity Editor for the element.
Element mass with no Node id	Checks if element_mass is defined on a proper node ID.	Lists all element_mass with no node ID.	Not supported.	Opens the Entity Editor for the element.
Common nodes of RBodies	Checks if any keywords that are mapped to rigids share a common node.	Lists all all of the rigid elements that share the same node.	Not supported.	Opens the Rigid panel and updates the page.
Rbody connected to MATL20	Checks if any node of the component that is associated with MAT_RIGID is referred/ used by keywords mapped rigids.	Lists all of the rigid elements that share node(s) with the components associated with MAT_RIGID.	Not supported.	Opens the Rigid panel and updates the page.

Name	Description	Result	Auto correction action	Manual correction action
Rbody xtranode - common node	Checks if any keyword mapped to rigids in HyperMesh and *CONSTRAINED_EXTRA_NODE defined in the model share a common node.	Lists all of rigid elements that share node(s) with CONSTRAINED_	Not supported.	Opens the Rigid panel and updates the page.
0 or 1 slave nodes in Constrained nodal rigid body	Checks if a constrained_nodal_rigid_bo has atleast one master and slave defined.	Lists all onstrained_noda that have one or zero nodes.	Not supported.	Opens the Rigid panel.
Slavenodes in other Constrained nodal rigid bodies	Checks if constrained nodal rigid bodies have common slave nodes.	Lists all constrained nodal rigid bodies that have common slave nodes.	Not supported.	Opens the Rigid panel.
Joint nodes are non-coincident	Checks if nodes defined in any *CONSTRAINED_JOINTS that need to be coincident are actually coincident in the model.	Lists all of the joint elements whose node definitions are not coincident.	The nodes are made coincident.	Not supported.
Gear joint nodes 1-2 coincident	Checks if a gear joint's node 1 and node 2 are coincident.	Lists all gear joints whose node 1 and node 2 are coincident.	Not supported.	Opens the Joints panel.
Joint universal 1-3 and 2-4 not orthogonal	Checks if a segment formed by nodes 1-3 and 2-4 are orthogonal for a universal joint.	Lists all universal joints whose segments are not orthogonal.	Not supported.	Opens the Joints panel.

Name	**Description**	**Result**	**Auto correction action**	**Manual correction action**
Joint nodes not on rigid	Checks if nodes referred in *CONSTRAINED_JOINTS are not referred in *CONSTRAINED_NODAL_RI or *CONSTRAINED_EXTRA_NC or *CONSTRAINED_NODE_SET or in the component that is associated with MAT_RIGID.	Lists all joint elements whose node are not associated with rigids of LS-DYNA.	Not supported.	Opens the Fe joints panel.
Negative or zero volume of solids	Checks if a solid has volume associated with it.	Lists all of the solid elements with zero or negative volume.	Deletes failing elements.	Not supported.
Id of component out of bounds	Checks if there are any component ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the components with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Component has more than one element type	Checks if the component has more than one unique element type among shell, solids, beam, discrete and SPH.	Lists all components in the model that have multiple element types.	Not supported.	Opens the Organize panel.
Component does not have a part card	Checks if the components defined in HyperMesh are not assigned with card image *PART or *PART_COMPOSITE.	Lists all components that do not have a card image.	Not supported.	Opens the Components panel.

Name	Description	Result	Auto correction action	Manual correction action
Part has an incorrect property card	Checks if the components with the card image *PART are associated with the correct properties based on their elements content.	Lists all of the components that do not have the correct properties associated to them.	Not supported.	Opens the Components panel.
Part is not defined with a proper material	Checks if the components with card image *PART are associated with the correct material based on their element content and associated property.	Lists all components that do not have the correct materials associated to them.	Not supported.	Opens the Components panel.
No property in Part or Part composite	Checks if a property is attached to a part or part composite.	Lists all components with a card image, but with no property attached.	Not supported.	Opens the Entity Editor of the component entity.
No material in Part or Part composite	Checks if a material is attached to a part or part composite.	Lists all components with a card image, but with no material attached.	Not supported.	Opens the Entity Editor of the component entity.
Id of group out of bounds	Checks if there are any group ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the groups with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Master set not defined or empty	Checks if there are any interface entities defined in the model that have a valid master.	Lists all interface entities that do not have either a slave or master defined.	Not supported.	Opens the Entity Editor of the interface.

Name	Description	Result	Auto correction action	Manual correction action
Slave set not defined or empty	Checks if there are any interface entities defined in the model that have a valid slave.	Lists all interface entities that do not have either a slave or master defined.	Not supported.	Takes user to the interface EE.
Tied contacts: Slave nodes in master surface	Checks if the slave nodes of tied contacts are also present in its master.	Lists all of the tied contacts that have slave nodes that are also present its master.	Not supported.	Opens the Interfaces panel.
Parts with non rigid material in Interface	Checks if the entities of the group - ContRigidSurface, ConstRigidRbody, RgdNodeToRgdBody, and RgdBodyToRgdBody are rigid entities.	Lists all of the rigid groups with non rigid entities.	Not supported.	Opens the Entity Editor of the group entity.
Xtranode Pid is not defined	Checks whether the component ID is referred/assigned in the *CONSTRAINED_EXTRA_NODE keyword.	Lists all the *CONSTRAINED_ keywords that do not have part referred in them.	Not supported.	Opens the *CONSTRAINED_ EXTRA_NODE card image.
Nid or Nsid is not defined for Constrained extra node	Checks if NID or NSID is not defined on constrained extra nodes.	Lists all of the constrained extra nodes that do not have nid/ nsid defined.	Not supported.	Opens the Entity Editor of the constrained extra node entity.
Xtranode set is not a node set	Checks if the set referred/used in *CONSTRAINED_EXTRA_NODE_SET is not a node set.	Lists all the *CONSTRAINED_ keywords that are associated with a set that is not a node set.	Not supported.	Opens the add page in the Interface panel.

Name	Description	Result	Auto correction action	Manual correction action
Common nodes in Xtranode entities	Checks if any *CONSTRAINED_EXTRA_NODES defined in the model share a common node.	Lists all the *CONSTRAINED_ that share common node(s).	Not supported.	Opens the add page in the Interface panel
Xtranode connected to Matl20	Checks if any node of a component that is associated with MAT_RIGID is referred/used in *CONSTRAINED_EXTRA_NC	Lists all the *CONSTRAINED_ keywords that share node(s) with _RIGID.	Not supported.	Opens the add page in the Interface panel
Xtranode on non-rigid part	Checks if the component referred/used in *CONSTRAINED_EXTRA_NODE is not associated with *MAT_RIGID.	Lists all the *CONSTRAINED_ keywords that have part referred in them and are not associated with MAT_RIGID.	Not supported.	Opens the *CONSTRAINED_ EXTRA_NODE card image.
Constrained rigid bodies with empty master or slave	Checks if *CONSTRAINED_RIGID_BODIES has empty master/slave.	Lists all of the *CONSTRAINED_ for master/slave that are empty.	Not supported.	Opens the *CONSTRAINED_ RIGID_BODIES card image.
Id of load out of bounds	Checks if there are any load ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the loads with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Boundary prescribed motion rigid with non-rigid part	Checks if boundary_prescribed_motion_rigid is defined on a rigid part.	Lists all of the boundary_prescri on a non rigid part.	Not supported.	Opens the Entity Editor of the load entity.
Id of load collector out of bounds	Checks if there are any load collector ids that are either less than the min_limit or more than the	List all of the load collectors with ids outside of the limit.	Not supported.	Opens the Renumber panel

Name	**Description**	**Result**	**Auto correction action**	**Manual correction action**
	max_limit of allowable ids in the ID Manager.			
Parts with non rigid material in Load rigid body	Checks if the load_rigid_body is defined on a rigid part.	Lists all of the load_rigid_body defined on non rigid parts.	Not supported.	Opens the Entity Editor of the load collector entity.
Initvel_Gen part is not defined	Checks if part ID is not defined in the *INITIAL_VELOCITY_GENERATION keyword.	Lists the *INITIAL_VELOCI keyword where part is not defined.	Not supported.	Opens the *INITIAL_VELOCI TY_GENERATION card image.
Initvel_Gen part set is not defined	Checks if parts set ID is not defined in the *INITIAL_VELOCITY_GENERATION keyword.	Lists the *INITIAL_VELOCI keyword where the part set is not defined.	Not supported.	Opens the *INITIAL_VELOCI TY_GENERATION card image.
InitVel nodeset is empty	Checks if the nodeset referred to in the *INITIAL_VELOCITY keyword is empty.	Lists the *INITIAL_VELOCI keyword that is associated with an empty node set.	Not supported.	Opens the *INITIAL_VELOCITY card image.
Initvel_Gen nodeset is empty	Checks if the nodeset referred to in the *INITIAL_VELOCITY_GENERATION keyword is empty.	Lists *INITIAL_VELOCI keyword that is associated with an empty node set	Not supported.	Opens the *INITIAL_VELOCI TY_GENERATION card image.
Initvel and Initvel_Gen in the same model	Checks if both INITIAL_VELOCITY and INITIAL_VELOCITY_GENERA are defined	Lists *INITIAL_VELOCI *INITIAL_VELOCI defined in the model.	Deletes the entity.	Not supported.
Spc set is not a node set	Checks if the set referred in *BOUNDARY_SPC_NODE_S is not a node set.	Lists *BOUNDARY_SPC keywords that are associated with a set that	Not supported.	Opens the *BOUNDARY_SPC _NODE_SET card image.

Name	Description	Result	Auto correction action	Manual correction action
		is not a node set.		
Lcid has not been defined for LoadBody	Checks if the curve is not defined for the variable LCID in *LOAD_BODY.	Lists all *LOAD_BODY keywords that have no curve defined for the variable LCID.	Not supported.	Opens the *LOAD_BODY card image.
Boundary prescribed accel rigid with non-rigid part	Checks if a BoundPresAccRigid load collector is defined on a rigid part.	Lists all BoundPresAccRig load collectors not defined on a rigid part.	Not supported.	Opens the Entity Editor of the entity.
Constrained rigid body stopper with non-rigid part	Checks if a constrained rigid body stopper is defined on a rigid part.	Lists all of the constrained rigid body stoppers defined on non rigid part.	Not supported.	Opens the Entity Editor of the load collector entity.
Parts with non rigid material in InitVel	Checks if initial velocity is defined on a rigid part.	Lists all of the initial velocities that are not defined on rigid part.	Not supported.	Opens the Entity Editor of the load collector entity.
Parts with non rigid material in Dform2Rigid	Checks if PID, PSID, and MRB are defined on Dform2Rigid rigid parts	Lists all of the dform2rigids whose pid, psid, or MRB are non rigid	Not supported.	Opens the Entity Editor of the load collector entity.
Parts with non rigid material in Dform2RgdInert	Checks if PID is defined on Dform2RgdInertia a rigid parts.	Lists all of the dform2rigidinertia whose pid is non rigid.	Not supported.	Opens the Entity Editor of the load collector entity.
Id of curve out of bounds	Checks if there are any curve ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the curves with ids outside of the limit.	Not supported.	Opens the Renumbers panel.

Name	Description	Result	Auto correction action	Manual correction action
Repeated X values in curve	Checks if any ordinate values are repeated in *DEFINE_CURVE keyword.	Lists all curves defined in the model that have repeated absicca.	Not supported.	Opens the Curve Editor.
Number of curves not same as one defined by table	Checks if the number of curves on the table is equal to the curve ids defined on it.	Lists all of the tables with missing curve ids.	Not supported.	Opens the Entity Editor of the curve entity.
Id of set out of bounds	Checks if there are any set ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the sets with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
NodeAdd set is not referring to node set	Checks if *SET_NODE_ADD refers to a set other than Node sets.	Lists all of the *SET_NODE_ADD that refer to a set other than Node sets.	Not supported.	Opens the Sets panel.
Id of system out of bounds	Checks if there are any system ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the systems with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Id of system collector out of bounds	Checks if there are any system collector ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the system collectors with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Box boundary not properly defined	Checks if *DEFINE_BOX or *DEFINE_BOX_LOCAL have minimum X,Y, and Z values larger than maximum X, Y, and Z values.	Lists all of the *DEFINE_BOX; *DEFINE_BOX_LOCAL with incorrect definitions.	Switches the minimum X, Y, and Z values with the maximum X, Y, and Z values.	Opens the corresponding Block card image panel.

Name	Description	Result	Auto correction action	Manual correction action
Box definition with null length	Checks if there is a length defined for all three directions of the box, that is xmin!=xmax, and so on.	Lists all of the boxes with a null length.	Not supported.	Opens the Entity Editor of the block entity.
Id of contact surfaces out of bounds	Checks if there are any contact surface ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the contact surfaces with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Id of node out of bounds	Checks if there are any node ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the nodes with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Accelerometer nodes are not on a rigid	Checks if the nodes of the accelerometer belong to a part with MAT20, *CONSTRAINED_EXTRA_NODES_NODE/SET, or *CONSTRAINED_NODAL_RI	Lists all accelerometers whose nodes are not rigid.	Not supported.	Opens the Entity Editor.
Accelerometer nodes does not define system	Checks if the three nodes of the accelerometer defines a valid system, that is they are not co-incident or co-linear.	Lists all accelerometer which do not define a valid system.	Not supported.	Opens the Entity Editor.
Id of beamsection out of bounds	Checks if there are any beam section ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the beam sections with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Id of beamsection collector out of bounds	Checks if there are any beam section collector ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the beam section collectors with ids outside of the limit.	Not supported.	Opens the Renumbers panel.

Name	Description	Result	Auto correction action	Manual correction action
Id of control volumes out of bounds	Checks if there are any control volume ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the control volumes with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Invalid chamber id specified	Checks if an *AIRBAG_PARTICLE, *DEFINE_CPM_VENT, or *DEFINE_CPM_GAS_PROPE refer to a non-existing chamber ID.	Lists all of the *AIRBAG_PARTIC *DEFINE_CPM_VI *DEFINE_CPM_G referring a non-existing chamber ID.	Not supported.	Opens the corresponding control volumes Card Image panel.
Parts with non rigid material in Airbag	Checks if an airbag is defined on the rigid material.	Lists all of the airbags defined on non rigid parts.	Not supported.	Opens the Entity Editor of the control volume entity.
Sid is missing for the airbag definition	Checks if entities, that is, SID are associated with airbags.	Lists all of the airbags with an empty SID field.	Not supported.	Opens the Entity Editor of the control volume entity.
Id of parameters out of bounds	Checks if there are any parameter ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the paramters with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Id of output blocks out of bounds	Checks if there are any output block ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the output blocks with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Id of cross sections out of bounds	Checks if there are any cross section ids that are either less than the min_limit or more than the	List all of the cross sections with ids outside of the limit.	Not supported.	Opens the Renumbers panel.

Name	Description	Result	Auto correction action	Manual correction action
	max_limit of allowable ids in the ID Manager.			
Parts with non rigid material in cross section	Checks if database cross sections are defined on rigid parts (the ID attribute).	Lists all of the database cross sections defined on non rigid parts.	Not supported.	Opens the Entity Editor of the cross section entity.
ID of rigidwalls out of Bounds	Checks if there are any rigidwall ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the rigids with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Id of hourglasses out of bounds	Checks if there are any hourglass ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the hourglasses with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Id of pretensioners out of bounds	Checks if there are any pretensioner ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the pretensioner with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Seatbelt sensors are not defined for pretensioners	Checks if a pretensioner references any *ELEMENT_SEATBELT_SENSORS.	An error is displayed a pretensioner references any *ELEMENT_SEATI and lists all failing pretensioners.	Not supported.	Opens the Entity Editor of the pretensioner entity.
Id of retractors out of bounds	Checks if there are any retractor ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the retractors with ids outside of the limit.	Not supported.	Opens the Renumbers panel.

Name	Description	Result	Auto correction action	Manual correction action
Seatbelt sensors are not defined for retractors	Checks if a retractor references any *ELEMENT_SEATBELT_SENS	An error is displayed ractor references any *ELEMENT_SEATI and lists all failing retractors.	Not supported.	Opens the Entity Editor of the retractor entity.
Id of sliprings out of bounds	Checks if there are any slipring ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the sliprings with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Curve x axis values not ascending	Check if the value of x axis is not in ascending order	List all the curves whose x axis values are not in ascending order	Not supported	Opens Curve Editor
CRB referring to non rigid part	If PIDM or PIDS in Constrained Rigid Body is assigned with non rigid material	List all Constrained Rigid Body referring to non rigid part	Not supported	Opens Curve Editor
Number of curves referred by table should be more than 1	Check if *DEFINE_TABLE is referred with one or less curves	List all *DEFINE_TABLE which have one or less curves	Not supported	Opens Entity Editor

Warnings

Name	Description	Result	Auto correction action	Manual correction action
Mat id exceeds 8 characters	Check if any material id exceeds 8 characters.	List all of the materials with	Not supported.	Opens the Renumbers panel.
		ids greater than 8 digits.		
Negative or null density	Checks if any materials have negative or zero density defined.	Lists all materials that have negative density.	Not supported.	Opens the corresponding materials Card Image panel.
Negative or null young's Modulus	Checks if any materials have negative or zero Young's Modulus defined; also checks for EA, EB, and EC values.	Lists all materials that have negative Young's Modulus.	Not supported.	Opens the corresponding materials Card Image panel.
Negative or null Nu	Checks if any materials have negative or zero Poisson Ratio defined.	Lists all materials that have negative poisson ratio.	Not supported.	Opens the corresponding materials Card Image panel.
Nu greater than 0.5	Checks if any material is having Nu greater than 0.5.	Lists all materials in the model that have Nu greater than 0.5	Not supported.	Opens the Material panel.
Negative Lcss	Checks if any elastic plastic materials have negative curve IDs that describe its flow stress.	Lists all materials that have negative curve IDs.	Not supported.	Opens the corresponding materials Card Image panel.
Unused materials	Checks if any materials are not referred/used in any of the keywords.	Lists all unused materials.	Deletes the material.	Not supported.
Mat and referenced curves not in same include	Checks if materials and associated curves are in the same include file.	Lists all of the materials whose curves are not in same include.	Not supported.	Opens the Components panel.

Name	Description	Result	Auto correction action	Manual correction action
Material Ref value not compatible with Initial geometry	Checks if a material has a Ref value and corresponding nodes presents in a initial_foam_reference_geor	Lists all materials with a Ref value, but no nodes associated with initial geometry.	Removes the Ref value of the material.	Not supported.
Prop id exceeds 8 characters	Check if any property id exceeds 8 characters.	List all the properties with ids greater than 8 digits.	Not supported.	Opens the Renumbers panel.
Unused properties	Checks if any properties (*SECTION, *HOURGLASS,*EOS,*INTEG *DAMPING) are not used in any of the keywords.	Lists all unused properties.	Deletes the property.	Not supported.
Property assigned to more than one Part	Check if any property id is defined in more than one component.	List all of the properties assigned to more than one component.	Not supported.	Opens the Components panel.
Area is not defined for section beam	Checks if any section beam property area is not defined.	Lists all section beam properties that have an area less than or equal to zero.	Not supported.	Opens the Property panel.
ITT is not defined for section beam	Checks if any section beam property ITT is not defined.	Lists all section beam properties that have ITT less than or equal to zero.	Not supported.	Opens the Property panel.
ISS is not defined for section beam	Checks if any section beam property ISS is not defined.	Lists all section beam properties that have ISS less than or equal to zero.	Not supported.	Opens the Property panel.

Name	Description	Result	Auto correction action	Manual correction action
Section shell thickness is not defined	Checks if any section shell property thickness is less than or equal to zero.	Lists all section shell properties that have thickness less than or equal to zero.	Not supported.	Opens the Property panel.
Thickness Thic1s is not defined	Check if any section beam properties with ELFORM 1,4,5,7,8, 9, 11 have a thickness THIC1s less than or equal to zero.	Lists all section beam properties that have thickness THIC1s less than or equal to zero.	Not supported.	Opens the Property panel.
Thickness Thic2s is not defined	Check if any section beam properties with ELFORM 1,4,5,7,8, 9, 11 have a thickness THIC2s less than or equal to zero.	Lists all section beam properties that have thickness THIC2s less than or equal to zero.	Not supported.	Opens the Property panel.
Thickness Thic1t is not defined	Check if any section beam properties with ELFORM 1,4,5,7,8, 9, 11 have a thickness THIC1t less than or equal to zero.	Lists all section beam properties that have thickness THIC1t less than or equal to zero.	Not supported.	Opens the Property panel.
Thickness Thic2t is not defined	Check if any section beam properties with ELFORM 1,4,5,7,8, 9, 11 have a thickness THIC2t less than or equal to zero.	Lists all section beam properties that have thickness THIC2t less than or equal to zero.	Not supported.	Opens the Property panel.
Free 1D elements	checks if 1D elements in model are free, i.e. elements whose nodes are not connected to valid entities.	lists out the 1D elements whose nodes are not connected to any FE or to a constarined extra node.	Not supported.	Opens the Bars panel

Name	Description	Result	Auto correction action	Manual correction action
Id of elems greater than authorized	Checks if element id is greater than 8 characters.	Lists all of the elements whose id is greater than 8 digits.	Not supported.	Opens the Renumbers panel.
Rigid body has free slave nodes	Checks if a rigid body has slave nodes which are not attached to any other FE entity.	Lists all rigid elements with free slave nodes.	Not supported.	Opens the Rigids panel.
Rigid body connected to only one part	Checks if a rigid bodies slave nodes are connecting more than one component.	Lists all rigid bodies which are connected to only one part.	Not supported.	Opens the Delete panel.
Beam: 3 nodes aligned	Checks if the axis node is along the orientation direction for a beam element.	Lists all beam element with an aligned 3rd node.	Not supported.	Opens the Bars panel.
Incorrect seatbelt material	Checks if a seatbelt element (2 and 4 noded) is associated with a seatbelt material.	Lists all seatbelt elements not associated with a seatbelt material.	Not supported.	Opens the Components panel.
Check thickness and Initial shell stress of element	Checks if an element_shell_thickness is associated with an initial_shell_stress.	Lists all element_shell_th with no initial_shell_stres attribute.	Not supported.	Opens the Entity Editor for the element.
Element and component not in same include	Checks if an element and its associated component are in the same Include file.	Lists all elements of components that are not in the same Include file.	Not supported.	Opens the Organize panel.

Name	Description	Result	Auto correction action	Manual correction action
Constrained nodal rigid body and nodes not in same include	Checks if a constrained_nodal_rigid_bo and its associated nodes are in the same Include file.	Lists all constrained_noda whose nodes are not in the same Include file.	Not supported.	Opens the Entity Editor for the element.
Empty components	Checks if any components are empty (no Element and no Geometry).	Lists all empty components.	Deletes the component.	Not supported.
Component is referring to undefined entity	Checks for undefined entities that are referenced by components.	Checks for undefined entities that are referenced by components.	Not supported.	Opens the card image for the corresponding component.
Component is referring to unresolved entity	Checks if an unresolved property or material is attached to a component.	Lists all components attached to unresolved properties or materials.	Not supported.	Opens the Entity Editor for the component.
Component and property have different id	Checks if the component has the same id as the property attached to it.	Lists all of the components that fail the criteria.	Not supported.	Opens the Entity Editor of the component entity.
Component and material have different id	Checks if the component has the same id as the material attached to it.	Lists all of the components that fail the criteria.	Not supported.	Opens the Entity Editor of the component entity.
Part inertia Node id shared	Checks if the node id of part_inertia is attached to any other component other than itself.	Lists all of the part inertia whose node id is attached to any other component other than itself.	Not supported.	Opens the Entity Editor of the component entity.

Name	**Description**	**Result**	**Auto correction action**	**Manual correction action**
Component, material or property not in same include	Checks if the part, material, and section assigned to a component are all in the same include file.	Lists all of the components whose material and property are not in same include.	Not supported.	Opens the Entity Editor of the component entity.
Component hourglass not in same include	Checks if the hourglass assigned to a component are all in the same include file.	Lists all of the components whose hourglass is not in same include.	Not supported.	Opens the Entity Editor of the component entity.
Slave nodes in multiple interfaces	Checks if the same node is used as slave in more than one *CONTACT entity.	Lists all *CONTACT entities that share the same slave nodes.	Not supported.	Opens the add page in the Interface panel.
Common elements within the same Contact	Checks if the same element is used within the same contact.	Lists all interface entities that have the same element used with the same contact.	Not supported.	Opens the Interface panel.
Contact ghost elements have no base elements	Checks if the contact elements created for interface have a base element.	Lists all interface entities that have a contact element without its base element.	Not supported.	Opens the add page in the Interface panel.
Duplicate base elements within contact	Checks if there is any duplicate base element within contact.	Lists all contacts that have duplicate base elements.	Not supported.	Opens the Interfaces panel.
Un-Tied nodes in Contact tied	Checks for un-tied nodes in tied contact definitions.	Lists all contacts that have untied-nodes.	Not supported. Option to automatically create un-tied node sets.	Opens the card image for the corresponding *CONTACT.

Name	Description	Result	Auto correction action	Manual correction action
Undefined master or slave in Contact	Checks if a group contains undefined entities.	Lists all groups that reference undefined entities.	Not supported.	Opens the Entity Editor for the entity.
Unresolved master or slave in Contact	Checks if a group contains unresolved entities.	Lists all groups that contain unresolved entities.	Not supported.	Opens the Entity Editor for the entity.
Friction not well defined on groups	Checks the static and dynamic friction value associated with groups.	Lists all groups that have zero fs or zero fd values.	Not supported.	Opens the Entity Editor for the entity.
Boxes defined but not applicable	Checks if a group has a slave and master box only when Sstype and Mstype is 2/3 correspondingly.	Lists all groups that have invalid box definitions.	Removes invalid boxes.	Opens the Entity Editor for the entity.
Groups, referenced Box or other entities not in same include	Checks if a slave and master entities including sboxid/mboxid are in the same Include file as that of the set.	Lists all sets that have master/slave entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Empty Xtranode entities	Checks for each constrainedextranode entity if: • The PID field is unspecified or the specified component is empty. • The NID field is unspecified. • The NSID is unspecified or the specified set is empty.	Lists all constrainedextranode entities that are empty.	Deletes empty constrainedextranode entities.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Master or slave distance greater than 100	Checks if the minimum distance between master and slave nodes of each part of a constrained extra node is greater than 100.	Lists all constrained_extra whose master and slave nodes are greater than 100. You can change the max distance in the config file.	Not supported.	Not supported.
Pidm and Pids distance greater than 100	Checks if the minimum distance between master and slave nodes of each part of a constrained rigid body is greater than 100.	Lists all constrained_rigid whose master and slave nodes are greater than 100. You can change the max distance in the config file.	Not supported.	Not supported.
Multiple spcs on a node	Checks if more than one BOUNDARY_SPC are defined on the same node.	Lists all *BOUNDARY_SPC that are defined on the same node.	Deletes selected *BOUNDARY_SPC.	Not supported.
Common node between spcset and node	Checks if the same node is defined in *BOUNDARY_SPC_NODE and *BOUNDARY_SPC_NODE_S	Lists all *BOUNDARY_SPC that are defined on the same node as *BOUNDARY_SPC	Not supported.	Opens the Delete panel to delete *BOUNDARY_SPC _NODE.
Different types of thermal load in model	Checks if a model contains both *LOAD_THERMAL_CONSTAN and *LOAD_THERMAL_VARIABL thermal load types.	Checks if a model contains both LOAD_THERMAL and *LOAD_THERMAL thermal load types.	Deletes the thermal load of the second type.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Load node and node not in same include	Checks if the nodes of the load_node and the load_node are defined in the same include file.	Lists all of the loads of load_node type whose nodes are not in same include.	Not supported.	Opens the Entity Editor of the load entity.
Boundary prescribed motion, comp, curve not in same include	Checks if a prescribed motion load and its components/curves are in same include file.	Lists all boundary prescribed motions whose PID or curves are not in same include file.	Not supported.	Opens the Entity Editor of the load entity.
InitVel_Gen part set is empty	Checks if the part set associated with	Lists all *INITIAL_VELOCI	Not supported.	Opens the card image for

Name	**Description**	**Result**	**Auto correction action**	**Manual correction action**
	*INITIAL_VELOCITY_GENER is empty.	that have an empty part set.		*INITIAL_VELOCITY_GENERATION keyword.
InitVel or InitVel_Gen node set is not defined	Checks if node set ID is not defined in the *INITIAL_VELOCITY or *INITIAL_VELOCITY_GENER keyword.	Lists *INITIAL_VELOCI or *INITIAL_VELOCI keywords that do not have nodesets defined.	Not supported.	Opens the corresponding *INITIAL_VELOCITY or *INITIAL_VELOCITY_GENERATION card image.
Initial velocity set is not a node set	Checks if the set associated with *INITIAL_VELOCITY is node set.	Lists all *INITIAL_VELOCI keywords that have a set other than node set attached to it.	Not supported.	Opens the card image for *INITIAL_VELOCITY keyword.
InitVel or InitVel_Gen velocity is not defined	Checks if the velocity value is defined in *INITIAL_VELOCITY or *INITIAL_VELOCITY_GENER keywords.	Lists all *INITIAL_VELOCI keywords that have no velocity defined.	Not supported.	Opens the card image for *INITIAL_VELOCITY or INITIAL_VELOCITY_GENERATION keyword.
Part set has not been defined for LoadBodyPart	Checks if the part set is referred to in the *LOAD_BODY_PARTS keyword.	Lists all *LOAD_BODY_PA keywords that do not have a part referred to them.	Not supported.	Opens the card image for *LOAD_BODY_PARTS keyword.
Boundary prescribed accel rigid, comp not in same include	Checks if BoundPresAccRigid and its associated component are in the same include file.	Lists all boundary_prescri loadcols whose component is not in same include file.	Not supported.	Opens the Entity Editor of the load collector entity.
Load gravity, part or set not in same include	Checks if the loadcollector LOAD_GRAVITY and its associated part or set are defined in the same include file.	Lists all load_gravity whose part/ set is not in the same include.	Not supported.	Opens the Entity Editor of the load collector entity.

Name	Description	Result	Auto correction action	Manual correction action
Load node set and set not in same include	Checks if the loadcollector LOAD_NODE_SET and its associated set are defined in the same include file.	Lists all load_node_set whose set is not in the same include.	Not supported.	Opens the Entity Editor of the load collector entity.
Unused curves	Checks if the curves defined model are used/ referred to in other keywords.	Lists all unused curves *DEFINE_CURVE in the model.	Deletes unused curves.	Not supported.
Duplicate value in define table	Checks if curve IDs on a table are repeated.	Lists all tables that have repeated curve IDs.	Not supported.	Opens the Entity Editor for the curve.
Table and curve not in same include	Checks if tables and curves defined on it are in the same include file.	Lists all tables whose curves are not in the same include file.	Not supported.	Opens the Entity Editor of the curve entity.
Set with more than 100 rows	Checks if a set has more than 100 rows of data. The max length is customizable in the third row of the config file. Currently if it is a formula, then it checks for the number of formula clauses. The set doesn't expand even though it is expanded in card edit.	Lists all sets with more than 100 rows.	Not supported.	Not supported.
Unused sets	Checks if the sets defined model are used/referred to in other keywords or in others sets.	Lists all unused sets.	Deletes unused sets.	Not supported.
Empty sets	Checks if any sets defined in the model are empty.	Lists all empty sets.	Deletes empty sets.	Not supported.
Sets with no finite elements or nodes	Checks if a set contains valid finite elements/ nodes.	Lists all sets that do not contain valid fe entities, such	Not supported.	Opens the Entity Editor of the entity.

Name	Description	Result	Auto correction action	Manual correction action
	Sets listed under "Empty Sets" are not reported.	as elements or nodes.		
Entity set is referring to undefined entity	Checks if a set references undefined entities.	Lists all sets that contain undefined entities.	Not supported.	Opens the Entity Editor of the entity.
Entity set is referring to unresolved entity	Checks for unresolved entities that are referenced by sets.	Lists all sets that reference unresolved entities.	Not supported.	Opens the card image for the corresponding set.
Orphan child set collect	Checks for set collects that are not referenced by a master set collect.	Lists all orphan child set collects.	Not supported.	Opens the Set panel.
Child set collect referred in entities other than master	Checks for set collects that are referenced by entities other than master set collects.	Lists all child set collects that are referenced by other entities.	Not supported.	Opens the Set panel.
Set node and referenced entities not in same include	Checks if a set_node and its associated nodes or sets are in the same Include file.	Lists all set_node whose nodes/node sets are not in same Include file.	Not supported.	Opens the Entity Editor for the Set entity.
Set part and referenced entities not in same include	Checks if set_part and its associated components or sets are in the same include file.	Lists all set_part whose components/ part sets are not in same include file.	Not supported.	Opens the Entity Editor of the set entity.
Unused systems	Checks if any system defined in the model is unused/not referred to in other keywords.	Lists all unused systems.	Deletes unused systems.	Not supported.
Systems with axis badly defined	Checks if system axes are defined correctly.	Lists all systems with badly defined axes.	Not supported.	Opens the Systems panel and Update page.

Name	Description	Result	Auto correction action	Manual correction action
Define coordinate nodes and nodes not in same include	Checks if the system and nodes forming the axes are in the same include file.	Lists all define_corodinate whose nodes are not in same include file.	Not supported.	Opens the Entity Editor of the system entity.
Define coordinate vector and nodes not in same include	Checks if DEFINE_CORDINATE_VECT(and optional nodal point NID are in the same include file.	Lists all define_corodinate whose nodes are not in same include file.	Not supported.	Opens the Entity Editor of the system entity.
Unused blocks	Checks if the blocks (*DEFINE_BOX) defined in the model are unused / not referred to in other keywords.	Lists all unused blocks.	Deletes unused blocks	Not supported.
Unused contact surface	Checks if the contact surface (*set_segment) defined in the model is unused /not referred to in other keywords.	Lists all unused contact surfaces.	Deletes unused contact surfaces.	Not supported.
Id of node greater than authorized	Checks if id of node is greater than 8 characters.	Lists all nodes with ids greater than 8 digits.	Not supported.	Opens the Renumbers panel.
Free nodes in the model	Checks if the nodes in the model are free (not used by any elements).	Lists free nodes.	Deletes free nodes.	Not supported.
Accelerometer and referenced entities not in same include	Checks if an accelerometer and its nodes are in the same Include file.	Lists all accelerometers whose referenced entities are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Unused beamsections	Checks if any Beamsection defined in the model is unused.	Lists all unused Beamsections in the model.	Delete unused beamsections.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Airbag and referenced entities not in same include	Checks if an airbag and its referenced entities are in the same include file.	Lists all airbag control volumes whose referenced entities are not in same include file.	Not supported.	Opens the corresponding controlvols EE.
Initial foam reference geometry and nodes not in same include	Checks if intial_foam_reference_geon and its associated nodes are in the same include file.	Lists all initial_foam_refe whose nodes are not in the same include file.	Not supported.	Opens the corresponding controlvols EE.
Incompatible Material	Checks if initial_foam_reference_geor nodes are present in the component associated with the mat with ref value.	Lists all initial_foam_ref_ whose nodes are not present in the material with ref value.	Not supported.	Opens the Comps panel.
Unused parameters	Checks if there are unused parameters in the model.	Lists all parameters which are not referenced by any other entity.	Deletes unused parameters.	Not supported.
Duplicate parameter names in model	Checks if there are two or more parameters with the same name in the model.	Lists all parameters that have duplicate names.	Not supported.	Opens the Entity Editor for the parameter entity.
Database cross section, part or set not in same include	Checks if a part, set, or system of a database cross section are in the same Include file as the cross section.	Lists all database_cross_s whose part, set, or system are not in the same Include file.	Not supported.	Opens the Entity Editor for the cross section entity.
Rigidwall Set or Box not in same Include	Checks if the set and box of a rigid wall are in the same Include file as the rigid wall.	Lists all rigid walls whose referenced entities are not	Not supported.	Opens the Entity Editor for the entity.

Name	Description	Result	Auto correction action	Manual correction action
		in the same Include file.		
Rigidwall is referring to undefined entity	Checks if a rigidwall references undefined entities.	Lists all rigidwalls that contain undefined entities.	Not supported.	Opens the Entity Editor of the entity.
Rigidwall is referring to unresolved entity	Checks for unresolved entities that are referenced by rigidwalls.	Lists all rigidwalls that reference unresolved entities.	Not supported.	Opens the Entity Editor of the entity.
Unused hourglasses	Checks if there are unused hourglasses in the model.	Lists all hourglasses which are not referenced by any other entity.	Deletes unused hourglasses.	Not supported.
Pretensioner is referring to unresolved entity	Checks if a pretensioner references unresolved entities.	Lists all pretensioners that contain unresolved entities.	Not supported.	Opens the Entity Editor of the entity.
Pretensioner and referenced entities not in same include	Checks if a pretensioner and its referenced entities are in the same Include file.	Lists all pretensioners whose referenced entities are not in the same Include file.	Not supported.	Opens the Entity Editor of the entity.
Retractor is referring to unresolved entity	Checks if a retractor references unresolved entities.	Lists all retractors that contain unresolved entities.	Not supported.	Opens the Entity Editor of the entity.
Retractor and referenced entities not in same include	Checks if a retractor and its referenced entities are in the same Include file.	Lists all retractorw whose referenced entities are not	Not supported.	Opens the Entity Editor of the entity.

Name	Description	Result	Auto correction action	Manual correction action
		in the same Include file.		
Slipring is referring to unresolved entity	Checks if a slipring references unresolved entities.	Lists all sliprings that contain unresolved entities.	Not supported.	Opens the Entity Editor of the entity.
Slipring and referenced entities not in same include	Checks if a slipring and its referenced entities are in the same Include file.	Lists all sliprings whose referenced entities are not in the same Include file.	Not supported.	Opens the Entity Editor of the entity.
Translational mass is zero in Constrained nodal rigid body interia	Check if TM value is zero in Constrained nodal rigid body interia	List all the Constrained nodal rigid body interia with TM=0	Not supported	Opens Entity Editor
Translational mass is zero in Part inertia	Check if TM value is zero in Part Inertia	List all the Part Inertia with TM=0	Not supported	Opens Entity Editor
Translational mass is zero in Deformable to rigid inertia	Check if TM value is zero in Deformable to rigid inertia	List all the Deformable to rigid inertia TM=0	Not supported	Opens Entity Editor
PID and MID of hex spotweld assembly shared with hex spotweld			Supported to create separate assembly	Not Supported
Scale factor is zero for define curve feedback			Not supported	Opens Entity Editor
Scale factor is zero for curves			Not supported	Opens Entity Editor

Name	Description	Result	Auto correction action	Manual correction action
Scale factor is zero for define table	Check if SFA=0 for *DEFINE_TABLE, *DEFINE_TABLE_2D & *DEFINE_TABLE_3D	List the tables whose SFA value is zero	Not supported	Opens Entity Editor
Table rate is not in ascending order	Check if input in VALUE fields are not in ascending order	List the tables where input in VALUE is not in ascending order	Not supported	Opens Entity Editor
Last X-Axis point doesnt match with control termination	Check the curves whose X-Axis end value is not equal to ENDTIM value in *TERMINATION_TIME	List the curves whose X-Axis end value is not equal to ENDTIM value in *TERMINATION_T	Not supported	Opens Curve Editor
DB history with repeated entities				
DB history and referenced entities not in same include				
No db history defined on node N1 of accelerometer				

Info

Name	Description	Result	Auto correction action	Manual correction action
Number of defined materials	Checks for the number of defined materials.	Lists all defined materials.	Not supported.	Not supported.
Number of undefined materials	Checks for the number of undefined materials.	Lists all undefined materials.	Not supported.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Number of defined properties	Checks for the number of defined properties.	Lists all defined properties.	Not supported.	Not supported.
Number of undefined properties	Checks for the number of undefined properties.	Lists all undefined properties.	Not supported.	Not supported.
Number of Components	Checks for the number of components.	Lists all components.	Not supported.	Not supported.
Number of groups	Checks for the number of groups.	Lists all groups.	Not supported.	Not supported.
Number of defined curves	Checks for the number of defined curves.	Lists all defined curves.	Not supported.	Not supported.
Number of undefined curves	Checks for the number of undefined curves.	Lists all undefined curves.	Not supported.	Not supported.
Number of defined sets	Checks for the number of defined sets.	Lists all defined sets.	Not supported.	Not supported.
Number of undefined sets	Checks for the number of undefined sets.	Lists all undefined sets.	Not supported.	Not supported.
Number of output blocks	Checks for the number of output blocks.	Lists all output blocks.	Not supported.	Not supported.
Number of defined hourglasses	Check for number of defined HM hourglass entities in the model.	List all defined hourglasses.	Not supported.	Not supported.
Number of undefined hourglasses	Check for number of undefined hourglass entities in the model.	List all undefined HM	Not supported.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
		hourglass in the model		

Nastran

Errors

Name	Description	Result	Auto correction action	Manual correction action
Material ID exceeds 8 characters	HyperMesh does not write out more than eight characters for IDs, so if you have more HyperMesh will write out ***, which fails the solver. This check is disabled when using a long format template.	Lists the failing materials.	Not supported.	Opens the Renumber panel.
Property is missing material	Checks for properties without a material reference.	Lists the failing properties.	Not supported.	Opens the Properties panel.
Property ID exceeds 8 characters	HyperMesh does not write out more than eight characters for IDs, so if you have more HyperMesh will write out ***, which fails the solver. This check is disabled when using a long format template.	Lists the failing properties	Not supported.	Opens the Renumber panel.
Property with zero cross section area	Checks if the area of properties PBEAM, PBAR and PROD is less than or equal to zero.	Lists the failing properties.	Not supported.	Opens the Property panel.
Component with solids and shells	Checks if a component consists of both solid and shell elements.	Lists the failing components.	Not supported.	Opens the Organize panel.

Name	Description	Result	Auto correction action	Manual correction action
Node ID exceeds 8 character	HyperMesh does not write out more than eight characters for IDs, so if you have more HyperMesh will write out ***, which fails the solver. This check is disabled when using a long format template.	Lists the failing nodes.	Not supported.	Opens the Renumber panel.
Element ID exceeds 8 characters	HyperMesh does not write out more than eight characters for IDs, so if you have more HyperMesh will write out ***, which fails the solver. This check is disabled when using a long format template.	Lists the failing elements.	Not supported.	Opens the Renumber panel.
Free 1D elements	Checks for 1D elements that have one or more connection missing. This check does not include RBE2 or RBE3 elements.	Lists the failing elements.	Deletes the failing 1D elements.	Not supported.
RBE2 with constrained dependent nodes	Checks for RBE2 with constraints on dependent nodes.	Lists the failing elements.	Not supported.	Opens the Rigids panel.
Free Rigid elements	Checks for RBE2 elements that have one or more connection missing, typically a leg that is missing a connection.	Lists the failing elements.	Deletes the free arms of the Rigid.	Not supported.
Free RBE3 elements	Checks for RBE3 elements that have one or more connection missing, typically a leg that is missing a connection or an independent node that is not connected to structure.	Lists the failing elements.	Deletes the free arms of RBE3.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
RBE3 elements are collinear	Checks if the dependent and independent nodes of an RBE3 element are collinear.	Lists the failing elements.	Deletes the failing RBE3 elements.	Not supported.
RBE3 dof connected to solid	Checks for rotational DOFs on independent nodes of an RBE3 connected to a solid element.	Lists the failing elements.	Removes rotational DOFs from affected independent grids of failing RBE3 elements.	Not supported.
Zero-length CBUSH without CID	Checks if the CBUSH spring element of zero length has the element coordinate system, CID defined.	Lists the failing elements.	Not supported.	Opens the Springs panel.
Elements have a dependency	Checks for rigid elements that share dependent nodes with other rigid elements.	Lists the failing elements.	Deletes the failing rigid elements.	Not supported.
Element has rigid loops	Checks for rigid elements that form a closed loop; where the dependent node of one rigid serves as independent node of the next rigid in the loop.	Lists the failing elements.	Deletes the failing rigid elements.	Not supported.
Duplicate 1D Elements	Checks if any 1D elements share the same nodes in the same order.	Lists the duplicate 1D elements.	Deletes the failing elements.	Not supported.
Duplicate 2D Elements	Checks if any 2D elements share the same nodes in the same order.	Lists the duplicate 2D elements.	Deletes the failing elements.	Not supported.
Duplicate 3D Elements	Checks if any 3D elements share the same nodes in the same order.	Lists the duplicate 3D elements.	Deletes the failing elements.	Not supported.
NSM field not defined	Checks for invalid or missing entries on NSMADD loadcols.	Lists the failing loadcols.	Not supported.	Opens the Loadcols panel

Name	Description	Result	Auto correction action	Manual correction action
MPC with free independent nodes	Checks for MPC equations with free independent nodes.	Lists the failing equations.	Updates the MPC by removing the free independent node.	Not supported.

Warnings

Name	Description	Result	Auto correction action	Manual correction action
Unused materials	Checks for any materials that are not referenced by a property.	Lists the failing materials.	Deletes unused materials.	Not supported.
Material E is not defined	Checks if E of MAT1 or MAT8 is less than or equal to zero.	Lists the failing materials.	Not supported.	Opens the Materials panel.
Material Rho is not defined	Checks if the Rho of MAT1, MAT2,MAT4 ,MAT5, MAT8, MAT9, MAT10,MATHE and MATHP is less than or equal to zero.	Lists the failing materials.	Not supported.	Opens the Materials panel.
Material Nu is zero	Checks if the Nu of MAT1 and MAT8 is equal to zero.	Lists the failing materials.	Not supported.	Opens the Materials panel.
Material is missing a card image	Checks for defined materials with no card image.	Lists the failing materials.	Not supported.	Opens the Materials panel.
Unused properties	Checks for properties that are not assigned to a component or element.	Lists the failing properties.	Deletes unused properties.	Not supported.
Property is missing a card image	Checks for defined properties with no card image.	Lists the failing properties.	Not supported.	Opens the Properties panel.
PSHELL thickness is not defined	Checks if the property PSHELL thickness is less than or equal to zero.	Lists the failing properties.	Not supported.	Opens the Properties panel.

Name	Description	Result	Auto correction action	Manual correction action
PSHELL and PSHLN1 with different MID	Checks if the property PSHELL has a different material defined for PSHELL and PSHLN1.	Lists the failing properties.	Not supported.	Opens the Properties panel.
PCOMP thickness is not defined	Checks if the first thickness value of Property PCOMP array is less than or equal to zero. The first thickness value is mandatory.	Lists the failing properties.	Not supported.	Opens the Properties panel.
1D Prop beamsection mismatch	Checks if the 1D property and the beam section attached to it have the same area and area moments of inertia. A difference greater than 0.01 will fail the check.	Lists the failing properties.	Disassociates the beam section from the 1D property.	Not supported.
Empty components	Checks for components that do not contain elements.	Lists the failing components.	Deletes the empty components.	Not supported.
Empty assemblies	Checks if assemblies are empty.	Lists the empty assemblies.	Deletes empty assemblies.	Not supported.
Empty sets	Checks if sets are empty.	Lists empty sets.	Deletes empty sets.	Not supported.
Empty contact surfaces	Checks if contact surfaces are empty.	Lists empty contact surfaces.	Deletes empty contact surfaces.	Not supported.
Empty groups	Checks if groups are empty.	Lists empty groups.	Deletes empty groups.	Not supported.
Empty plys	Checks if plys are empty.	Lists empty plys.	Deletes empty plys.	Not supported.
Empty load collectors	Checks if load collectors are empty.	Lists empty load collectors.	Deletes empty load collectors.	Not supported.
Empty vector collectors	Checks if vector collectors are empty.	Lists empty vector collectors.	Deletes empty vector collectors.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Empty laminates	Checks if laminates are not having any ply.	Lists empty laminates.	Deletes empty laminates.	Not supported.
CONM1 with zero diagonal mass matrix values	Checks if the diagonal mass matrix values of CONM1 are less than or equal to zero.	Lists the failing CONM1.	Not supported.	Opens the Elements panel.
CONM2 with zero mass	Checks if the mass of CONM2 is less than or equal to zero.	Lists the failing CONM2.	Not supported.	Opens the Masses panel.
Orientation vector along axis vector	Checks if the orientation vector of bar elements CBEAM and CBAR is along axis vector.	Lists the failing bar elements.	Not supported.	Opens the Bars panel.
Element connectivity	Checks for shell elements connected by one or less nodes and solid elements connected by two or less nodes.	Lists the failing elements.	Deletes the failing elements.	Not supported.
Unused BeamSections	Checks if any beamsections defined in the model are unused.	Lists the failing beamsections.	Deletes the unused beamsecions.	Not supported.
Unused sets	Checks if any sets are not used.	Lists unused sets.	Deletes unused sets.	Not supported.
Unused curves	Checks if any curves are not used.	Lists unused curves.	Deletes unused curves.	Not supported.
Unused systems	Checks if any systems in the model are not used.	Lists unused systems.	Deletes unused systems.	Not supported.
Unused laminates	Checks if any laminates in the model are not used.	Lists unused laminates.	Deletes unused laminates.	Not supported.
Unused plys	Checks if any plys in the model are not used.	Lists unused plys.	Deletes unused plys.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Unused vectors	Checks if any vectors in the model are not used.	Lists unused vectors.	Deletes unused vectors.	Not supported.

Info

Name	Description	Result	Auto correction action	Manual correction action
Components share the same property	Checks for two or more components that have the same property assigned.	Lists the failing components.	Not supported.	Opens the Components panel.
Component does not have a property	Checks for components that contain elements but do not have any properties assigned.	Lists the failing components.	Not supported.	Opens the Components panel.

OptiStruct

Errors

Name	Description	Result	Auto correction action	Manual correction action
Material ID exceeds 8 characters	HyperMesh does not write out more than eight characters for IDs, so if you have more HyperMesh will write out ***, which fails the solver. This check is disabled when using a long format template.	Lists the failing materials.	Not supported.	Opens the Renumber panel
Property is missing material	Checks for properties without a material reference.	Lists the failing properties.	Not supported.	Opens the Properties panel.
Property ID exceeds 8 characters	HyperMesh does not write out more than eight characters for IDs, so if	Lists the failing properties.	Not supported.	Opens the Renumber panel

Name	Description	Result	Auto correction action	Manual correction action
	you have more HyperMesh will write out ***, which fails the solver. This check is disabled when using a long format template.			
Property with zero cross section area	Checks if the area of PBEAM, PBAR and PROD properties are less than or equal to zero.	Lists the failing properties.	Not supported.	Opens the Properties panel.
Component with solids and shells	Checks if a component consists of both solid and shell elements.	Lists the failing components.	Not supported.	Opens the Organize panel.
Node ID exceeds 8 character	HyperMesh does not write out more than eight characters for IDs, so if you have more HyperMesh will write out ***, which fails the solver. This check is disabled when using a long format template.	Lists the failing nodes.	Not supported.	Opens the Renumber panel.
Element ID exceeds 8 characters	HyperMesh does not write out more than eight characters for IDs, so if you have more HyperMesh will write out ***, which fails the solver. This check is disabled when using a long format template.	Lists the failing elements.	Not supported.	Opens the Renumber panel.
Free 1D elements	Checks for 1D elements that have one or more connection missing. This check does not include RBE2 or RBE3 elements.	Lists the failing elements.	Deletes the failing 1D elements.	Not supported.
RBE2 with constrained dependent nodes	Checks for RBE2 with constraints on dependent nodes.	Lists the failing elements.	Not supported.	Opens the Rigids panel.
Free Rigid elements	Checks for RBE2 elements that have one or more	Lists the failing elements.	Deletes the free arms of the Rigid.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
	connection missing, typically a leg that is missing a connection.			
Free RBE3 elements	Checks for RBE3 elements that have one or more connection missing, typically a leg that is missing a connection or an independent node that is not connected to structure.	Lists the failing elements.	Deletes the free arms of RBE3.	Not supported.
RBE3 elements are collinear	Checks if the dependent and independent nodes of an RBE3 element are collinear.	Lists the failing elements.	Deletes the failing RBE3 elements.	Not supported.
RBE3 dof connected to solid	Checks for rotational DOFs on independent nodes of an RBE3 connected to a solid element.	Lists the failing elements.	Removes rotational DOFs from affected independent grids of failing RBE3 elmements.	Not supported.
Zero-length CBUSH without CID	Checks if the CBUSH spring element of zero length has the element coordinate system, CID defined.	Lists the failing elements.	Not supported.	Opens the Springs panel.
Elements have a dependency	Checks for rigid elements that share dependent nodes with other rigid elements.	Lists the failing elements.	Deletes the failing rigid elements.	Not supported.
Element has rigid loops	Checks for rigid elements that form a closed loop; where the dependent node of one rigid serves as independent node of the next rigid in the loop.	Lists the failing elements.	Deletes the failing rigid elements.	Not supported.
Duplicate 1D Elements	Checks if any 1D elements share the same nodes in the same order.	Lists the duplicate 1D elements.	Deletes the failing elements.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Duplicate 2D Elements	Checks if any 2D elements share the same nodes in the same order.	Lists the duplicate 2D elements.	Deletes the failing elements.	Not supported.
Duplicate 3D Elements	Checks if any 3D elements share the same nodes in the same order.	Lists the duplicate 3D elements.	Deletes the failing elements.	Not supported.
NSM field not defined	Checks for invalid or missing entries on NSMADD loadcols.	Lists the failing loadcols.	Not supported.	Opens the Loadcols panel.
MPC with free independent nodes	Checks for MPC equations with free independent nodes.	Lists the failing equations.	Updates the MPC by removing the free independent node.	Not supported.

Warnings

Name	Description	Result	Auto correction action	Manual correction action
Unused materials	Checks for any materials that are not referenced by a property.	Lists the failing materials.	Deletes unused materials.	Not supported.
Material E is not defined	Checks if E of MAT1, MAT8 or MAT9ORT is less than or equal to zero.	Lists the failing materials.	Not supported.	Opens the Materials panel.
Material Rho is not defined	Checks if Rho of MAT1, MAT2,MAT4, MAT5, MAT8, MAT9,MAT9ORT and MAT10 is less than or equal to zero.	Lists the failing materials.	Not supported.	Opens the Materials panel.
Material Nu is zero	Checks if Nu of MAT1, MAT8 and MAT9ORT is equal to zero.	Lists the failing materials.	Not supported.	Opens the Materials panel.
Material is missing a card image	Checks for defined materials with no card image.	Lists the failing materials.	Not supported.	Opens the Materials panel.

Name	Description	Result	Auto correction action	Manual correction action
Unused properties	Checks for properties that are not assigned to a component or element.	Lists the failing properties.	Deletes unused properties.	Not supported.
Property is missing a card image	Checks for defined properties with no card image.	Lists the failing properties.	Not supported.	Opens the Properties panel.
PSHELL thickness is not defined	Checks if the property PSHELL thickness is less than or equal to zero.	Lists the failing properties.	Not supported.	Opens the Properties panel.
PCOMP thickness is not defined	Checks if the first thickness value of Property PCOMP array is less than or equal to zero. The first thickness value is mandatory.	Lists the failing properties.	Not supported.	Opens the Properties panel.
1D Prop beamsection mismatch	Checks if the 1D property and the beam section attached to it have the same area and area moments of inertia. A difference greater than 0.01 will fail the check.	Lists the failing properties.	Disassociates the beam section from the 1D property.	Not supported.
Empty components	Checks for components that do not contain elements.	Lists the failing components.	Deletes the empty components.	Not supported.
Empty assemblies	Checks if assemblies are empty.	Lists the empty assemblies.	Deletes the empty assemblies.	Not supported.
Empty sets	Checks if sets are empty.	Lists the empty sets.	Deletes empty sets.	Not supported.
Empty contact surfaces	Checks if contact surfaces are empty.	Lists the empty contact surfaces.	Deletes the empty contact surfaces.	Not supported.
Empty groups	Checks if groups are empty.	Lists empty groups.	Deletes empty groups.	Not supported.
Empty plys	Checks if plys are empty.	Lists the empty plys.	Deletes empty plys.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Empty load collectors	Checks if load collectors are empty.	Lists the empty load collectors.	Deletes empty load collectors.	Not supported.
Empty vector collectors	Checks if vector collectors are empty.	Lists the empty vector collectors.	Deletes the empty vector collectors.	Not supported.
Empty laminates	Checks if laminates are not having any ply.	Lists the empty laminates.	Deletes the empty laminates.	Not supported.
CONM1 with zero diagonal mass matrix values	Checks if the diagonal mass matrix values of CONM1 are less than or equal to zero.	Lists failing CONM1.	Not supported.	Opens the Elements panel.
CONM2 with zero mass	Checks if the mass of CONM2 is less than or equal to zero.	Lists failing CONM2.	Not supported.	Opens the Bars panel.
Element connectivity	Checks for shell elements connected by one or less nodes and solid elements connected by two or less nodes.	Lists the failing elements.	Deletes the failing elements.	Not supported.
Unused BeamSections	Checks if any beamsection defined in the model is unused.	Lists the failing beamsections.	Deletes the unused beamsecions.	Not supported.
Unused sets	Checks if any sets are not used.	Lists unused sets.	Deletes unused sets.	Not supported.
Unused curves	Checks if any curves are not used.	Lists unused curves.	Deletes unused curves.	Not supported.
Unused systems	Checks if any systems in the model are not used.	Lists unused systems.	Deletes unused systems.	Not supported.
Unused laminates	Checks if any laminates in the model are not used.	Lists unused laminates.	Deletes unused laminates.	Not supported.
Unused plys	Checks if any plys in the model are not used.	Lists unused plys.	Deletes unused plys.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Unused vectors	Checks if any vectors in the model are not used.	Lists unused vectors.	Deletes unused vectors.	Not supported.

Info

Name	Description	Result	Auto correction action	Manual correction action
Components share the same property	Checks for two or more components that have the same property assigned.	Lists the failing components.	Not supported.	Opens the Components panel.
Component does not have a property	Checks for components that contain elements but do not have any properties assigned.	Lists the failing components.	Not supported.	Opens the Components panel.

PAM-CRASH 2G

Errors

Name	Description	Result	Auto correction action	Manual correction action
Id of component out of bounds	Checks if there are any component ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the components with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Component missing material	Checks if the components with card image PART / are associated with the correct material based	Lists all components that do not have the correct materials associated to them.	Not supported.	Opens the Components panel.

Name	Description	Result	Auto correction action	Manual correction action
	on their element content.			
Component does not have a part card	Checks if the components defined in Altair HyperMesh are not assigned with card image PART / .	Lists all components that do not have a card image.	Not supported.	Not supported.
Id of material out of bounds	Checks if there are any material ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the materials with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Id of property out of bounds	Checks if there are any property ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the properties with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Id of group out of bounds	Checks if there are any group ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the groups with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Master entities of contacts/interfaces not defined or empty	Checks if there are any contact or interface entities defined in the model that do not have any entities referred or empty	Lists all contact or interface entities that do not have master entities defined.	Not supported.	Opens the Entity Editor of the contact.

Name	Description	Result	Auto correction action	Manual correction action
	in the place of master.			
Slave entities of contacts/interfaces not defined or empty	Checks if there are any contact or interface entities defined in the model that do not have any entities referred or empty in the place of slave.	Lists all contact or interface entities that do not have slave entities defined.	Not supported.	Opens the Entity Editor of the contact.
Slave nodes of contacts/interfaces in master surface	Checks if the slave nodes of contact or interface are also present in its master.	Lists all contacts that have slave nodes that are also present its master.	Not supported.	Opens the Entity Editor of the contact.
Id of element out of bounds	Checks if there are any element ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the elements with ids outside of the limit.	Not supported.	Opens the Renumbers panel.
Duplicate 1D elements	Checks if two or more 1D elements are connected between the same nodes.	Lists all of the duplicate 1D elements.	Deletes failing elements.	Not supported.
Duplicate 2D elements	Checks if two or more 2D elements are connected between the same nodes.	Lists all of the duplicate 2D elements.	Deletes failing elements.	Not supported.
Duplicate 3D elements	Checks if two or more 3D elements are connected between the same nodes.	Lists all of the duplicate 3D elements.	Deletes failing elements.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Negative or zero volume of solids	Checks if a solid has negative volume associated with it.	Lists all of the solid elements with zero or negative volume.	Deletes failing elements.	Not supported.
Beam: 3 nodes aligned	Checks if the beam element nodes IDNOD1, IDNOD2 & IDNOD3 are on same line or co-linear. Or, if the axis node is along the orientation direction for a beam element.	Lists all beam element with an aligned 3rd node.	Not supported.	Not supported.
Master node in other Rigid elements	Checks if node referred in master entity is also referred in rigid entity.	Lists all of the rigid entities that share the node.	Not supported.	Opens the Rigid panel
Slave nodes in other Rigid elements	Checks if node referred in slave entity is also referred in rigid entity.	Lists all of the rigid entities that share the node.	Not supported.	Opens the Rigid panel
Rigid element master shares slave nodes	Checks if same node referred in both master and slave entity in rigid entity.	Lists all of the rigid entities that share the same node.	Not supported.	Opens the Rigid panel
Degenerated 4 node shells	Checks if quad (shell) element's IDNOD2 and IDNOD3 shares same node IDs	Lists all of the quad (shell) elements that have IDNOD2 = IDNOD3	Not supported.	Not supported.
Zero slave nodes in Rigid elements	Checks if slave field of rigid entities have empty or zero nodes referred.	Lists all of the rigid entities that have empty slave field.	Not supported.	Opens the Rigid panel

Name	Description	Result	Auto correction action	Manual correction action
Id of node out of bounds	Checks if there are any node ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the nodes with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of system out of bounds	Checks if there are any system ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the systems with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of system collector out of bounds	Checks if there are any system collector ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the system collectors with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of curve out of bounds	Checks if there are any curve ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the curves with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Repeated X values in curve	Checks if any X coordinate (abscissa) values are repeated in FUNCT keyword.	Lists all curves defined in the model that have repeated X coordinate (abscissa) values.	Not supported.	Opens the Curve Editor.

Name	Description	Result	Auto correction action	Manual correction action
Curve X values not ascending	Checks if X coordinate (abscissa) values are not in ascending order for FUNCT keyword.	Lists all curves defined in the model that have X coordinate (abscissa) values which are not in ascending order.	Not supported.	Opens the Curve Editor.
Curve has no point	Checks if there are no X coordinate (abscissa) and Y coordinate (ordinate) values in FUNCT keyword.	Lists all curves defined in the model that doesn't have X (abscissa) & Y (ordinate) coordinates values.	Not supported.	Opens the Curve Editor.
Id of load out of bounds	Checks if there are any load ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the loads with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of loadcol out of bounds	Checks if there are any load collector ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the load collectors with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Component does not have a part card	Checks if the components defined in Altair HyperMesh are not	Lists all components that do not have a card image.	Not supported.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
	assigned with card image PART / .			

Warnings

Name	Description	Result	Auto correction action	Manual correction action
Empty components	Checks if any components are empty (no Element and no Geometry).	Lists all empty components.	Deletes the component.	Not supported.
Component with zero or negative thickness	Checks if the PART with card image type SHELL, TSHEL, MEMBR have zero or negative element thickness (H).	Lists all components with zero or negative element thickness (H).	Not supported.	Not supported.
Free nodes in the model	Checks if the nodes in the model are free (not used by any elements).	Lists free nodes.	Deletes free nodes.	Not supported.
Unused properties	Checks if any properties are not referred/ used in any of the keywords.	Lists all unused properties.	Deletes the property.	Not supported.
Unused curves	Checks if the defined curves (FUNCT & SECURE/ FUNCT) in the model are not used/referred to in other keywords.	Lists all unused curves (FUNCT & SECURE/FUNCT) in the model.	Deletes unused curves.	Not supported.
Rigid element connected to only one part	Checks if slave nodes of rigid entity not connecting	Lists all rigid entities which are connected to only one part.	Not supported.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
	more than one component.			
Cascades of Rigid elements	Checks if any keywords that are mapped to rigids (RBODY, MTOCO, LINCO, NODCO and MBYSY_RIGID) share a slave node of other rigid entity master node.	Lists all rigid entities where slave node is also connected or shared to master node of other rigids	Not supported.	Not supported.
Free 1D elements	Checks if 1D elements in model are free, i.e. elements whose nodes are not connected to valid entities.	Lists all 1D elements whose nodes are not connected to any FE entity.	Deletes failing elements.	Not supported.
Free Rigid elements	Checks if rigid entities in model are free, i.e. rigid (RBODY, MTOCO, LINCO, NODCO and MBYSY_RIGID) entities whose nodes are not connected to valid entities.	Lists all rigid entities whose nodes are not connected to any FE entity.	Deletes failing entities.	Not supported.
Free RBE3 elements	Checks if RBE3 (OTMCO) entities in model are free, i.e. rbe3 rigid entities whose nodes are not connected to valid entities.	Lists all RBE3 (OTMCO) rigid entities whose nodes are not connected to any FE entity.	Deletes failing entities.	Not supported.
Shell elements with negative thickness	Checks if the shell elements with card image type SHELL, TSHEL have zero or	Lists all shell elements with zero or negative	Not supported.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
	negative element thickness (H).	element thickness (H).		
Shell elements with thickness	Checks if the shell elements with card image type SHELL, TSHEL have non-zero elemental level thickness (H) assigned.	Lists all shell elements which have non-zero element thickness (H).	Not supported.	Not supported.
Spring elements with zero length	Checks if the length of SPRING entities is zero.	Lists all springs which have zero length.	Deletes failing springs.	Not supported.
One slave node in Rigid elements	Checks if a rigid (RBODY, MTOCO, LINCO, NODCO and MBYSY_RIGID) entities has only one slave node defined.	Lists all rigids that have one node.	Not supported.	Opens the Rigid panel.
Unused materials	Checks if any materials are not referred/used in any of the keywords.	Lists all unused materials.	Deletes the material.	Not supported.
Material with zero or negative density	Checks if any materials have zero or negative density defined.	Lists all materials that have zero or negative density.	Not supported.	Opens the corresponding materials Card Image panel.
Material with zero or negative Young's modulus	Checks if any materials have zero or negative Young's Modulus defined.	Lists all materials that have zero or negative Young's Modulus.	Not supported.	Opens the corresponding materials Card Image panel.
Material with Poisson's ratio less than 0.001	Checks if any materials have values less than 0.001 Poisson Ratio defined.	Lists all materials that have poisson ratio <0.001.	Not supported.	Opens the corresponding materials Card Image panel.

Name	Description	Result	Auto correction action	Manual correction action
Material Solid with IMPLICIT flags under ISINT	Checks if any solid materials have implicit flags or legal values assigned.	Lists all solid materials that have implicit flags.	Not supported.	Not supported.
Unused parameters	Checks if there are unused parameters (PYVAR, PYFINC) in the model.	Lists all parameters which are not referenced by any other entity.	Deletes unused parameters.	Not supported.
Unused systems	Checks if any system (FRAME) defined in the model is unused/ not referred to in other keywords.	Lists all unused systems.	Deletes unused systems.	Not supported.
Unused beamsections	Checks if any Beamsection defined in the model is unused.	Lists all unused Beamsections in the model.	Delete unused beamsections.	Not supported.
Contact with negative thickness (Hcont)	Checks if Hcont value is negative in CNTAC entity.	Lists all CNTAC entities that have negative Hcont value.	Not supported.	Not supported.
Slave nodes in multiple contacts/ interfaces	Checks if the same node is used as slave in more than one CNTAC entity.	Lists all CNTAC entities that share the same slave nodes.	Not supported.	Not supported.

Info

Name	Description	Result	Auto correction action	Manual correction action
Number of defined materials	Checks for the number of defined materials.	Lists all defined materials.	Not supported.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Number of undefined materials	Checks for the number of undefined materials.	Lists all undefined materials.	Not supported.	Not supported.
Number of defined properties	Checks for the number of defined properties.	Lists all defined properties.	Not supported.	Not supported.
Number of undefined properties	Checks for the number of undefined properties.	Lists all undefined properties.	Not supported.	Not supported.
Number of Components	Checks for the number of components.	Lists all components.	Not supported.	Not supported.
Number of undefined Components	Checks for the number of undefined components.	Lists all undefined components.	Not supported.	Not supported.
Number of elements	Checks for the number of elements.	Lists all elements.	Not supported.	Not supported.
Number of groups	Checks for the number of groups.	Lists all groups.	Not supported.	Not supported.
Number of nodes	Checks for the number of nodes.	Lists all nodes.	Not supported.	Not supported.
Nodes not connected to elements	Checks if there are any nodes that are not connected to elements.	Lists all of the nodes in the model that are not connected to elements.	Not supported.	Not supported.
Number of defined curves	Checks for the number of defined curves.	Lists all defined curves.	Not supported.	Not supported.
Number of undefined curves	Checks for the number of undefined curves.	Lists all undefined curves.	Not supported.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Number of output blocks	Checks for the number of output blocks.	Lists all output blocks.	Not supported.	Not supported.

Permas

Errors

Name	Description	Result	Auto correction action	Manual correction action
Youngs modulus and poissons ratio is not defined	Checks if Youngs modulus has been defined for materials.	Lists all materials that do not have Youngs modulus defined.	Not supported.	Opens the Material panel.
Poisson Ratio is not defined	Check if Poissons ratio has been defined for materials.	Lists all materials that do not have Poissons ratio defined.	Not supported.	Opens the Material panel.
Density is not defined	Checks if density has been defined for materials.	Lists all materials that do not have density defined.	Not supported.	Opens the Material panel.
Shell properties without thickness	Checks if thickness has been defined for shell properties.	Lists all shell properties that do not have thickness defined.	Not supported.	Opens the Property panel.
Component with missing material	Checks if components are assigned a material.	Lists all components that do not have a material assigned.	Not supported.	Opens the Component panel.

Name	Description	Result	Auto correction action	Manual correction action
Component with missing property	Checks if components are assigned a property.	Lists all components that do not have a material assigned.	Not supported.	Opens the Component panel.
Component with solid and shells	Checks if a component consists of both solid and shell elements.	Lists all failing components.	Not supported.	Opens the Organize panel.
MPC (RBE2) with free dependent node	Checks if dependent nodes of MPC (RBE2) are free.	List all failing MPC (RBE2) elements.	Not supported.	Deletes the link between the independent node and free dependent node.
MPC (RBE3) with free independent node	Checks if independent nodes of MPC (RBE3) are free.	List all failing MPC (RBE3) elements.	Not supported.	Deletes the link between the independent node and free dependent node.

Errors

Name	Description	Result	Auto correction action	Manual correction action
Unused materials	Checks for materials that are not referenced by a component.	Lists all unused materials.	Deletes unused materials.	Not supported.
Unused properties	Check for properties that are not used in the model.	Lists all unused properties.	Deletes unused properties.	Not supported.
Property is missing a card image	Checks for properties that do not have a card image.	List all properties that do not have card images.	Not supported.	Opens the Property Create panel.
Property missing mass value	Checks if mass properties that do not have element mass defined.	Lists all Mass properties that do not have mass assigned.	Not supported.	Opens the Property panel.

Name	Description	Result	Auto correction action	Manual correction action
Damper property is missing Damping and Mass value	Checks if Damper properties are missing the Damping or Mass value.	Lists all Damper properties that do not have damping/mass assigned.	Not supported.	Opens the Property panel.
Beam Property missing HyperBeam Section	Checks if Beam propertiesare missing HyperBeam sections.	Lists all Beam properties that do not have beam section properties assigned.	Not supported.	Opens the Property panel.
Empty Components	Checks if components have one or more elements.	Lists all components that do not have elements.	Deletes empty components.	Not supported.
Unsed Sets	Check if sets are not used in the model.	Lists all unused sets.	Deletes empty sets.	Not supported.
Empty sets	Check if sets are empty.	List empty sets.	Deletes empty sets.	Not supported.
Empty groups	Checks if groups have either a master entity or slave entity that are assigned a contact surface/set.	Lists all groups with missing master and slave entities.	Deletes empty groups.	Not supported.
Empty contactsurfs	Checks if contactsurfs are assigned any elements.	Lists all empty contactsurfs.	Deletes empty contactsurfs.	Not supported.
Unused Sections	Check if there are any unused Beamsections defined in the model.	List all unused Beamsection collectors in the model.	Deletes unused beamsections.	Not supported.
Empty loadcollectors	Checks if there are any loads in the load collectors.	Lists all loadcollectors that do not have loads.	Deletes empty loadcollectors.	Not supported.
Empty Vectorcollectors	Checks if there are any empty vector collectors.	Lists all empty vectors.	Deletes empty vectors.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Empty assemblies	Checks if there are any empty assemblies.	Lists all empty assemblies.	Deletes empty assemblies.	Not supported.

Info

Name	Description	Result	Auto correction action	Manual correction action
Number of Material	Checks for the number of material entities in the model.	Lists all materials in the model.	Not supported.	Not supported.
Number of Properties	Checks for the number of property entities in the model.	Lists all properties in the model.	Not supported.	Not supported.
Number of Comp	Checks for the number of component entities in the model.	Lists all components in the model.	Not supported.	Not supported.
Number of Groups	Checks for the number of group entities in the model.	Lists all groups in the model.	Not supported.	Not supported.
Number of sets	Checks for the number of set entities in the model.	Lists all sets in the model.	Not supported.	Not supported.
Number of Steps	Checks for the number of step entities in the model.	Lists all steps in the model.	Not supported.	Not supported.

Radioss

Errors

Name	Description	Result	Auto correction action	Manual correction action
Id of material out of bounds	Checks if there are any material ids that are either less than the min_limit or more than the max_limit	List all of the materials with ids outside of the limit.	Not supported.	Opens the Renumber panel.

Name	Description	Result	Auto correction action	Manual correction action
	of allowable ids in the ID Manager.			
Nfunc must be greater than 0 for LAW36	Checks if LAW36 has a value defined for N_func.	Lists all materials of LAW36 which have N_func = 0.	Not supported.	Open the Entity Editor of the material entity.
LAW36 has empty function definition	Checks if material LAW36 has curves defined under NFUNC.	Lists all materials of LAW36 which have empty curve definitions.	Not supported.	Open the Entity Editor of the material entity.
Alpha greater than 1.0 for LAW37	Checks if materials of LAW37 have alpha1 values greater than 1.0.	Lists all materials of LAW37 that have alpha1 values that are less than or equal to 1.0.	Not supported.	Open the Entity Editor of the material entity.
Id of property out of bounds	Checks if there are any property ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the properties with ids outside of the limit.	Not supported.	Open the Entity Editor of the material entity.
Id of group out of bounds	Checks if there are any group ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the groups with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Master set not defined	Checks if there are any empty master sets in the interfaces.	List all interfaces that have an empty master.	Not supported.	Opens the Interface panel and updates the page.
Slave set not defined	Checks if there are any empty slave sets in the interface.	List all interfaces that	Not supported.	Opens the Interface panel and updates the page.

Name	Description	Result	Auto correction action	Manual correction action
		have an empty slave.		
Tied contacts: Slave nodes in master surface	Checks if the slave nodes of tied contacts are also present in its master.	Lists the tied contacts whose slave nodes are also present in its master.	Not supported.	Opens the Interface panel.
Id of node out of bounds	Checks if there are any node ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the nodes with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of system out of bounds	Checks if there are any system ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the systems with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of system collector out of bounds	Checks if there are any system collector ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the system collectors with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of curve out of bounds	Checks if there are any curve ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the curves with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Repeated X values in curve	Checks if abcissa values are clashing or repeated in a curve.	Lists all curves that have clashing or repeated absicca values, that is X values.	Not supported.	Opens the Curve editor to edit the curve fields.
Id of element out of bounds	Checks if there are any element ids that are either	List all of the elements with	Not supported.	Opens the Renumber panel.

Name	Description	Result	Auto correction action	Manual correction action
	less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	ids outside of the limit.		
Spring elements with zero length	Checks if spring elements have a length greater than zero.	Lists all spring elements with a null length.	Deletes all spring elements with a null length.	Not supported.
Pulley spring not defined with 3 nodes	Checks if a spring element that has a property of `/PROP/SPR_PUL` assigned is defined using three nodes.	Lists all spring elements that are not defined using exactly three nodes.	Deletes the element.	Not supported.
Joint nodes not on different rigids	Checks if a spring element that has a property of `/PROP/KJOINT2` assigned to its component has node1 and node2 on different rigid elements.	Lists all spring elements whose node1 and node2 are on the same rigid elements.	Deletes the element.	Not supported.
All nodes not defined for universal joint	Checks if a spring element that has a property of `/PROP/KJOINT2` assigned to its component and is of type 5 is defined using 4 nodes.	Lists all spring elements that are not defined using all four nodes.	Deletes the element.	Not supported.
Third node not defined for joint element	Checks if a spring that has a property of `/PROP/KJOINT2` assigned to its component and is of type revolute, cylindrical, translation, or planar has a third node defined.	Lists all spring elements which are not defined using three nodes.	Deletes the element.	Not supported.
Common master of RBody, RLink, CylJoint	Checks if any rigid bodies (rigids, rigid links and cyl. joints) have common master nodes.	List all rigids that share common master nodes.	Splits the common master nodes.	Opens the corresponding Rigids panel and updates the page.
Common slave of RBody, RLink, CylJoint	Checks if any rigid bodies (rigids, rigid links and cyl. joints) have common slave nodes.	Lists all rigids that share common slave nodes.	Splits the common slave nodes.	Opens the corresponding Rigids panel and updates the page.

Name	Description	Result	Auto correction action	Manual correction action
0 slave nodes in Rbody	Checks if there are 0 slave nodes in Rbody.	Lists all Rbodies that have 0 slave nodes.	Not supported.	Opens the corresponding Rigids panel and updates the page.
Rbody master is also slave	Checks if the master node of Rbody is also present in the slave side of the same Rbody.	Lists all Rbodies that have master node present in slave definition.	Not supported.	Opens the corresponding Rigids panel and updates the page.
Ikrem value is incompatible with RWALL	Checks if the flag for rigidwall deactivation, Ikrem, is incompatible with RWALL. It is incompatible if the value of Ikrem is equal to one.	Lists all rigids that have Ikream incompatible to RWALL.	Sets Ikrem to 0.	Opens the corresponding `RBODY` card image.
Rbody master is in box that defines slave	Checks if Rbody master node is in the box defined by slave group of this RBODY.	Lists all bodies that have master node in the box defined by slave group of this RBODY.	Not supported.	Opens the corresponding Rigids panel and updates the page.
Rbody master on element	Checks if the rbody master element is attached to a finite element.	Lists all of the rbodies whose master element is on a finite element.	Not supported.	Opens the Rigid panel.
Cascades of RBodies	Checks if the rbody master is also the slave of another rbody.	Lists all of the rbodies whose master is also a slave.	Not supported.	Opens the Rigid panel.
1D elements with duplicate nodes	Checks if 1D elements consists of two of the same nodes.	Lists all of the 1D elements with duplicate nodes.	Not supported.	Opens the Bars panel.
Duplicate 1D elements	Checks if two or more 1D elements are connected between the same nodes.	Lists all of the duplicate 1D elements.	Deletes the failing elements.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Duplicate 2D elements	Checks if two or more 2D elements are connected between the same nodes.	Lists all of the duplicate 2D elements.	Deletes the failing elements.	Not supported.
Duplicate 3D elements	Checks if two or more 3D elements are connected between the same nodes.	Lists all of the duplicate 3D elements.	Deletes the failing elements.	Not supported.
Degenerated 4 node Shells	Checks if four node shells have two of the same nodes within it.	Lists all of the 4 node shells with duplicate nodes.	Not supported.	Not supported.
Negative or zero volume of solids	Checks if a solid has zero or negative volume; Ideally when using HyperMesh you should not have negative volume.	Lists all of the solid elements with a zero or negative volume.	Deletes the failed solid elements.	Not supported.
Box definition with null length	Checks if there is a length defined for all three directions of a box, that is xmin!=xmax, and so on.	Lists all of the boxes with null lengths.	Not supported.	Opens the Entity Editor for the Block entity.
N1 and N2 are not defined on Box	Checks if N1 and N2 are defined on box/recta.	Lists all boxes of primitive type Rectangle that do not have N1 and N2 defined.	Not supported.	Opens the Entity Editor for the box entity.
Id of component out of bounds	Checks if there are any component ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the components with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Component has more than one element type	Checks if a component has more than one unique element type (shell, solids, beam, discrete, and SPH).	Lists all components that have multiple element types.	Not supported.	Opens the Organize panel.
Component does not have a part card	Checks if any components are assigned the Part card image.	Lists all components that do not	Not supported.	Opens the Entity Editor for the component entity.

Name	Description	Result	Auto correction action	Manual correction action
		have a card image.		
Component missing property	Checks if the part component has a property attached.	Lists all of the components with no property attached.	Not supported.	Opens the Entity Editor for the component entity.
Component missing material	Checks if the part component has a material attached.	Lists all of the components with no material attached.	Not supported.	Opens the Entity Editor for the component entity.
Part, mat and prop incompatible	Checks if parts, properties, and materials are compatible according to the material compatibility table, and few other constraints mentioned in individual material and property definitions.	Lists all components whose elements are not compatible with the material and property attached or the attributed defined on them.	Not supported.	Opens the Entity Editor for the component entity.
Id of Set out of bounds	Checks if there are any set ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the sets with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of load out of bounds	Checks if there are any load ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the loads with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of loadcols out of bounds	Checks if there are any load collector ids that are either less than the min_limit or more than the	List all of the load collectors with ids outside of the limit.	Not supported.	Opens the Renumber panel.

Name	Description	Result	Auto correction action	Manual correction action
	max_limit of allowable ids in the ID Manager.			
Initial velocity of type T+G.	Checks if initial velocity is of type T+G.	Lists all of the invel load collectors of type T+G.	Not supported.	Opens the Entity Editor for the load collector entity.
Skew and frame defined in Impvel or Impdisp	Checks if `/IMPVEL` and `/IMPDISP` have only one system associated with them.	Lists the `/IMPVEL` and `/IMPDISP` with different systems present in skew and frame.	Not supported.	Opens the Entity Editor for the load collector entity.
Isensor for Cload is not defined	Checks if Isensor is defined for Cload.	Lists all Cloads with Isensor not defined.	Not supported.	Opens the Entity Editor for the entity.
Grnod id for Cload is not defined	Checks if Grnod id is defined for Cload.	Lists all Cloads with Grnod ID not defined.	Not supported.	Opens the Entity Editor for the entity.
Skew for Cload is not defined	Checks if skew_csid is defined for Cload.	Lists all Cloads with skew_csid not defined.	Not supported.	Opens the Entity Editor for the entity.
Curve for Cload is not defined	Checks if fun_a1 is defined for Cload.	Lists all Cloads with fun_a1 not defined.	Not supported.	Opens the EEntity Editor for the entity.
Id of beamsection out of bounds	Checks if there are any beam section ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the beam sections with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of beamsection collector out of bounds	Checks if there are any beam section collector ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the beam section collectors with ids outside of the limit.	Not supported.	Opens the Renumber panel.

Name	Description	Result	Auto correction action	Manual correction action
Id of parameters out of bounds	Checks if there are any parameter ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the parameters with ids outside of the limit.	Not supported.	Opens the Renumber panel.
ISkew cannot be a system of type Frame	Checks if the system attached to a sensor is of type Frame.	Lists all of the sensors that contain systems of type Frame.	Not supported.	Opens the Entity Editor for the sensor entity.
Id of output blocks out of bounds	Checks if there are any output block ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the output blocks with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of contact surfaces out of bounds	Checks if there are any contact surface ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the contact surfaces with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Id of control volumes out of bounds	Checks if there are any control volume ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the control volumes with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Monitored volumes with empty surfaces	Checks if surfaces are defined on the airbag.	Lists all airbags that do not have surfaces defined on them.	Not supported.	Opens the Entity Editor for the control volume entity.
Monitored volumes with invalid vent holes data	Checks if area and surfs are defined on each vent of the airbag.	Lists all control volumes with zero area or no surface defined on any vents.	Not supported.	Opens the Entity Editor for the entity.

Name	Description	Result	Auto correction action	Manual correction action
Id of cross sections out of bounds	Checks if there are any cross section ids that are either less than the min_limit or more than the max_limit of allowable ids in the ID Manager.	List all of the cross sections with ids outside of the limit.	Not supported.	Opens the Renumber panel.
Sections: N1, N2, N3 badly defined	Checks if the nodes of a cross section define a plane.	Lists all of the cross sections without a plane.	Not supported.	Opens the Entity Editor for the cross section entity.
Prop17 Incompatible with `/INISHE_ORTHO_` or `/INISH3_ORTHO_`	Checks if the elements in property of type P17_STACK is referenced by `/INISHE/ORTHO_LOC` or `/INISH3/ORTHO_LOC` tables.	Lists all properties linked to elements referenced by `/INISHE/ORTHO_LOC` or `/INISH3/ORTHO_LOC` tables.	Not supported.	Opens the Entity Editor for the property entity.
Curve x axis values not ascending	Check if the value of x axis is not in ascending order	List all the curves whose x axis values are not in ascending order	Not supported	Opens Curve Editor
Parameter name length is greater than 9	Check if the character length of parameter name string is greater than 9	Opens Entity Editor to rename parameter name	Not supported	Opens Entity Editor

Name	Description	Result	Auto correction action	Manual correction action
Output blocks with no variables				

Warnings

Name	Description	Result	Auto correction action	Manual correction action
Unused materials	Checks if there any unused materials in the model.	Lists all unused materials.	Not supported.	Opens the material card image of /MAT/PLAS_TAB.
Strain rate coefficient equal to 0 for LAW2	Checks if the strain rate coefficient is equal to zero for material LAW2.	Lists all of the law2 materials that have a strain rate coefficient equal to zero.	Not supported.	Opens the material card image of either /MAT/PLAS_JOHNS or /MAT/PLAS_ZERIL.
Bad strain rate order in Law36	Checks for bad strain rate order in material Law36. The strain rate should be in increasing order.	Lists all the law36 materials having bad strain rate order.	Not supported.	Opens the material card image of /MAT/PLAS_TAB.
Material with zero density	Checks if material density is equal to zero.	Lists all of the materials that have zero density.	Not supported.	Opens the corresponding material card image.
Material with zero modulus	Checks if material modulus is equal to zero.	Lists all of the materials that have zero modulus.	Not supported.	Opens the corresponding material card image.
Material with Poisson's Ratio less than 0.001	Checks if material Poisson’s Ratio is less than 0.001.	Lists all of the materials that have Poisson’s Ratio less than 0.001.	Not supported.	Opens the corresponding material card image.

Name	Description	Result	Auto correction action	Manual correction action
Law2,4,15,27: C and Eps0 defined but *F* and *F* blank	Checks for law2, 4, 15, 27: C and Epsdot0 are defined, and *F* and *F* are blank.	Lists all materials that failed the check.	Not supported.	Opens the Entity Editor of the material entity.
LAW25 Tsai-Wu: C and Eps0 defined but *F* and *F* blank	Checks for law25, TSAI-WU: C12 and Epsdot0 are defined, and *F* and *F* are blank.	Lists all materials that failed the check.	Not supported.	Opens the Entity Editor of the material entity.
LAW25 Crasurv: any C and E0 defined but *F*, *F* blank	Checks for law25, CRASURV: C (or any of c1t,c1c,c2t,c2c,c12t) and Epsdot0 are defined, and *F* and *F* are blank.	Lists all materials that failed the check.	Not supported.	Opens the Entity Editor of the material entity.
Law36,59,60: Nfunc > 0 but *F* and *F* blank	Checks for law36, 59, 60: Nfunct > 0, and *F* and *F* are blank.	Lists all materials that failed the check.	Not supported.	Opens the Entity Editor of the material entity.
LAW44: C and p defined but *F* and *F* blank	Checks for LAW44: C and p are defined, and *F* and *F* are blank.	Lists all materials that failed the check.	Not supported.	Opens the Entity Editor of the material entity.
Law48: C,K and E0 defined but *F* blank	Checks for law48: C, K and Epsdot0 are defined, and *F* is blank.	Lists all materials that failed the check.	Not supported.	Opens the Entity Editor of the material entity.
Law52: C and P defined but *F* and *F* blank	Checks for law52: C and p are defined, and *F* and *F* are blank.	Lists all materials that failed the check.	Not supported.	Opens the Entity Editor of the material entity.
Law65,77: Nrate or Nload > 1 but *F* and *F* blank	Checks for law65, 77: Nrate/Nload > 1, and *F* and *F* are blank.	Lists all materials that failed the check.	Not supported.	Opens the Entity Editor of the material entity.
Law70: Nload > 1 but *F* and *F* blank	Checks for law 70: Nload > 1, and *F* and *F* are blank.	Lists all materials that failed the check.	Not supported.	Opens the Entity Editor of the material entity.
Law74: Eps rate scale defined but *F* and *F* blank	Checks for law74: Epsdot_scale is defined, and *F* and *F* are blank.	Lists all materials that failed the check.	Not supported.	Opens the Entity Editor of the material entity.

Name	Description	Result	Auto correction action	Manual correction action
Law80: Ceps and Peps defined but *F* and *F* blank	Checks for law80: Ceps and Peps are defined, and *F* and *F* are blank.	Lists all materials that failed the check.	Not supported.	Opens the Entity Editor of the material entity.
Mat and referenced curves not in same include	Checks if a material and its referenced curves are in the same include.	Lists all of the materials that have curves that are not in same include file as the material.	Not supported.	Opens the Entity Editor of the material entity.
Parameter and material title with &	Checks if a material name has "&" when a parameter is defined in the model.	Lists all materials whose name has "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor of the material entity.
Material with invalid name	Checks if a material name contains no more than 100 characters, and does not begin with a "/".	Lists all materials whose names are greater than 100 characters or begin with "/".	Not supported.	Opens the Entity Editor of the material entity.
Unused properties	Checks if there are any unused properties in the model.	Lists all of the unused properties	Deletes unused properties.	Not supported.
Section shell thickness	Checks if section shell thickness is less than or equal to zero.	Lists all of the shell properties that have a section thickness less than or equal to zero.	Not supported.	Opens the property card image of `/PROP/SHELL`.
Beam or truss property with area less than or equal to 0	Checks if beam and truss properties have an area equal to zero.	Lists all of the beam and truss properties that have an area	Not supported.	Opens the corresponding truss or beam

Name	Description	Result	Auto correction action	Manual correction action
		less than or equal to zero.		property card image.
Beam property with inertia less than or equal to 0	Checks if beam properties have an inertia less than or equal to zero.	Lists all of the beam properties that have inertia less than or equal to zero.	Not supported.	Opens the property card image of /PROP/BEAM.
Spring property with inertia less than or equal to 0	Checks if spring properties have an inertia less than or equal to zero.	List all the spring properties that have an inertia less than or equal to zero.	Not supported.	Opens the property card image of /PROP/SPRING.
Spring properties with null mass	Checks if spring properties have null mass.	Lists all of the spring properties that have null mass.	Not supported.	Opens the property card image of /PROP/SPRING.
Invalid Icpre value on type 6	Checks if *I* is defined for *I*, not equal to 14, 17, and 24.	Lists all properties with icpre!=0 and isolid!= 14,17,24.	Not supported.	Opens the Entity Editor of the entity.
Negative Gap value on type 2	Checks if GAP is negative on Type 2 properties.	Lists all properties of type 2 with a negative gap value.	Not supported.	Opens the Entity Editor of the entity.
Parameter and property title with &	Checks if a property name has "&" when a parameter is defined in the model.	Lists all properties whose name has "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor of the entity.
Property with invalid name	Checks if a property name has no more than 100 characters and does not begin with a "/".	Lists all properties whose names are greater than 100 character,	Not supported.	Opens the Entity Editor of the entity.

Name	Description	Result	Auto correction action	Manual correction action
		or begin with "/".		
Prop and referenced entities not in same include	Checks if a property and its curve, sensor, or system are in the same include.	Lists all properties which reference entities that are not in the same include.	Not supported.	Opens the Entity Editor of the entity.
Istf is not equal to 0 or 1 for interface type 11	Checks if attribute Istf is not equal to zero or one for Inter type 11.	Lists all interfaces of type11 that have istf not equal to zero or one.	Not supported.	Opens the interface card image of `/INTER/TYPE11`.
Idel is not equal to 1 for interface type 7	Checks if attribute Idel is not equal to one for interface type 7.	Lists all interfaces of type7 that have Idel not equal to one.	Not supported.	Opens the interface card image of `/INTER/TYPE7`.
Bad Isearch flag interface in interface type 2	Checks if attribute Isearch is defined badly.	Lists all interfaces of type2 that have Isearch not equal to zero or two.	Not supported.	Opens the interface card image of / INTER/TYPE2.
Bad spotflag in interface type2	Checks if Spotflag is badly defined for interface type 2.	Lists all interfaces of type2 that have a spotflag value not equal to zero.	Not supported.	Opens the interface card image of `/INTER/TYPE2`.
Slave nodes in multiple interfaces	Checks if slave nodes are in multiple interfaces for TYPE2.	Lists all interfaces of type2 that have common slave nodes across multiple interfaces.	Not supported.	Opens the Interface panel and updates the page.

Name	Description	Result	Auto correction action	Manual correction action
Un-Tied nodes in interface type 2	Checks if there are any free nodes in tied contacts, that is, contacts of type 2.	Lists all type2 contacts with free nodes.	Not supported.	Not supported.
Groups and referenced entities not in same include	Checks if group, master, slave, curves, and rbody are in the same include.	Lists all groups whose referenced entities are not in same include files as the group.	Not supported.	Opens the Entity Editor of the group entity.
Undefined master or slave in contact	Checks if groups contain undefined master or slave entities.	Lists all groups that reference undefined master or slave entities.	Not supported.	Opens the Entity Editor of the group entity.
Unresolved master or slave in contact	Checks if a contact contains unresolved entities.	Lists all groups with unresolved master or slave entities.	Not supported.	Opens the Entity Editor of the group entity.
Parameter and group title with &	Checks if a group name has "&" when a parameter is defined in the model.	Lists all groups whose names have "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor of the group entity.
Group with invalid name	Checks if a group name has no more than 100 characters and does not begin with a "/".	Lists all groups whose names are greater than 100 character or begin with "/".	Not supported.	Opens the Entity Editor of the group entity.
Free nodes in the model	Checks if there are any free nodes in the model. Nodes present in EREF are also not considered free by the solver. As they represent the initial positions.	Lists all of the free nodes in the model.	Deletes free nodes.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Incompatible kinematic conditions on nodes	Checks if more than one type of imposed conditions is acting on nodes; for example if nodes are connected to more than one of the following - rigid elements, spotwelds, boundary conditions and tied contacts. Based on skew, attached boundary condition directions are checked if they are constrained in different dofs.	Lists all of the nodes that have more than one imposed condition.	Not supported.	Opens the Delete panel.
Systems with axis badly defined	Checks if the system axes are not defined correctly.	Lists all of the Systs with axis that are badly defined.	Not supported.	Opens the Systs panel and updates the page.
Skew and nodes not in same include	Checks if a system and its nodes are in the same include.	Lists all of the systems that have nodes which are not in the same include.	Not supported.	Opens the Entity Editor of the system entity.
Parameter and system title with &	Checks if a system name has "&" when a parameter is defined in the model.	Lists all systems whose names have "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor of the entity.
Systems with invalid name	Checks if a system name has no more than 100 characters and does not begin with a "/".	Lists all systems whose names are greater than 100 character or begin with "/".	Not supported.	Opens the EEntity Editor of the entity.
Curve has no point	Checks if the function curve has any points defined.	Lists all of the function curves that do not	Creates a (0, 0) point.	Opens the Curves panel and updates the page.

Name	Description	Result	Auto correction action	Manual correction action
		have any points defined.		
Parameter and curve title with &	Checks if a curve name has "&" when a parameter is defined in the model.	Lists all curves whose names have "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor of the entity.
Curve with invalid name	Checks if a curve name has no more than 100 characters and does not begin with a "/".	Lists all curves whose names are greater than 100 character or begin with "/".	Not supported.	Opens the Entity Editor of the entity.
1 slave node in Rbody	Checks if there is only one slave node in a Rbody.	Lists all of the Rbodies that have only 1 slave node.	Not supported.	Opens the Rigids panel and updates the page.
Admas and nodes not in same include	Checks if Admas and nodes are in the same include.	Lists all pf the admas that have nodes which are not in same include.	Not supported.	Opens the Entity Editor of the mass entity.
RBody and referenced entities not in same include	Checks if RBody, nodes, skew, sensor, and surf are in the same include.	Lists all of the rbodies that reference entities which are not in same include as the rbody.	Not supported.	Opens the Entity Editor of the Rbodies.
RLink and referenced entities not in same include	Checks if Rlink, nodes, and skew are in the same include.	Lists all pf the rlinks that have nodes or skew which are not in same include.	Not supported.	Opens the Entity Editor of the Rlink.
Cyclic joint and nodes not in same include	Checks if a cyclic joint and its nodes are in the same include.	Lists all of the cyclic joints that have nodes	Not supported.	Opens the EEntity Editor of the cyclic joint.

Name	Description	Result	Auto correction action	Manual correction action
		which are not in same include.		
Beam: 3 nodes aligned	Checks that the 3 nodes of a beam are non collinear.	Lists beams with 3 nodes that are collinear.	Not supported.	Opens the Bars panel.
Small rigid body with non-spherical inertia	Checks if RBODY has non-spherical inertia.	Lists all small Rbodies that have non-spherical inertia.	Not supported.	Opens the Rigids panel and updates the page.
Spring elements length greater than 35	Checks if a spring with a property of `/PROP/SPR_PUL` assigned to its component has a length that is less than 35.	Lists all spring elements with a length greater than 35.	Delete the element.	Not supported.
Rigid body connected to only one part	Checks if rigid body slave nodes are connected to more than one component.	Lists all of the rigid bodies that are connected to only one part.	Not supported.	Opens the Delete panel.
Rigid body has free slave nodes	Checks if all dependent nodes of rigid bodies are connected to some element.	Lists all RBODY that have at least one free dependent node.	Removes the free end nodes from the rigid body.	Not supported.
Free spring elements	Checks if there are free spring elements	Lists all of the free spring elements.	Deletes free spring elements.	Not supported.
Free bar2 elements	Checks if there are free bar2 elements.	Lists all the free bar2 elements.	Deletes free bar2 elements.	Not supported.
Spring elems with null length	Checks if spring elements have a null length. Excludes those attached to properties of type 8.	Lists all spring elements that have a null length, other than those attached to	Delete the spring elements.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
		properties of type 8.		
Bar2 elems with null length	Checks of bar2 elements have a null length.	Lists all bar2 elements that have a null length.	Deletes the bar2 elements.	Not supported.
Mass elements have zero or negative mass	Checks if there are any mass elements that have zero or negative mass.	Lists all mass elements that have zero or negative mass.	Not supported.	Opens the Admas panel and updates the page.
Admas not compatible with 2D analysis	Checks if admas is present in the model, along with 2D analysis.	Lists all admas elements, if the 2D analysis flag is set.	Not supported.	Opens the Entity Editor of the entity.
Free mass elements	Checks if there are free mass elements.	Lists all of the free mass elements.	Deletes free mass elements.	Not supported.
Joint nodes are non-coincident	Checks if a spring element that has a property of `/PROP/KJOINT2` assigned to its component has a node1 and node2 that are coincident.	Lists all spring elements with non-coincident node 1 and node 2.	Deletes the elements.	Not supported.
3-4 Node shells with thickness	Checks if 3 or 4 noded shell elements have a thickness defined on the element itself.	Lists all 3 or 4 noded shell elements with a thickness.	Not supported.	Opens the Entity Editor of the entity.
Xalea of Random noise less than or equal to zero	Checks for random noise. You will have a random noise issue if the attribute XALEA of RANDOM control card is not equal to or less than zero.	Lists all the RANDOM control cards that have an attribute XALEA not equal to or less than zero.	Not supported.	Opens the card image of control card `/RANDOM`.
Unused boxes	Checks if there are unused boxes in the model.	Lists all unused boxes.	Deletes unused boxes.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Parameter and box title with &	Checks if a box name has "&" when a parameter is defined in the model.	Lists all boxes whose names have "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor of the entity.
Boxes with invalid name	Checks if a box name has no more than 100 characters and does not begin with a "/".	Lists all boxes whose names are greater than 100 character or begin with "/".	Not supported.	Opens the Entity Editor of the entity.
Component is referring to undefined entity	Checks if components reference undefined properties or materials.	Lists all components containing undefined materials or properties.	Not supported.	Opens the Entity Editor of the entity.
Component is referring to unresolved entity	Checks if components reference unresolved properties or materials.	Lists all components containing unresolved materials or properties.	Not supported.	Opens the Entity Editor of the entity.
Hourglass possible on component	Checks if hourglass is possible on a component; if the attached property has ishell = 1,2 or 3 or isolid = 1 or 2.	Lists all components with property attributes as mentioned earlier.	Not supported.	Opens the Entity Editor of the entity.
Parameter and component title with &	Checks if a component name has "&" when a parameter is defined in the model.	Lists all components whose names have "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor of the entity.
Comps with invalid name	Checks if a component name has no more than	Lists all components	Not supported.	Opens the Entity Editor of the entity.

Name	Description	Result	Auto correction action	Manual correction action
	100 characters and does not begin with a "/".	whose names are greater than 100 character or begin with "/".		
Part and referenced entities not in same include	Checks if a part and its associated entities are in the same include file.	Lists all parts which reference entities that are not in the same include file.	Not supported.	Opens the Entity Editor of the entity.
Unused sets	Checks if a set is not referenced by another entity.	Lists all sets which are unused.	Deletes unused sets.	Not supported.
Empty sets	Checks if a set has entity ids.	Lists all sets which do not contain any entities.	Deletes empty sets.	Not supported.
Sets with no finite elements or nodes	Checks if the defined set contains valid finite elements/nodes. This check does not report sets already reported under "empty sets".	Lists all sets which have empty components, sets, and so on attached to them.	Not supported.	Opens the Entity Editor of the entity.
Grspri with inconsistent element	Checks if Grspri contains only spring elements.	Lists all Grspri that contain more than spring elements.	Not supported.	Not supported.
Grbeam with inconsistent element	Checks if Grbream contains only beam elements.	Lists all Grbream that contain more than spring elements.	Not supported.	Not supported.
Grshel with inconsistent element	Checks if Grshel contains only 4 noded elements.	Lists all Grshel that contain more than 4 noded shell elements.	Not supported.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Grsh3n with inconsistent element	Checks if Grsh3n contains only 3 noded shell elements.	Lists all Grsh3n that contain more than 3 noded shell elements.	Not supported.	Not supported.
Grtruss with inconsistent element	Checks if Grtruss contains only truss elements.	Lists all Grtruss that contain more than truss elements.	Not supported.	Not supported.
Grquad with inconsistent element	Checks if Grquad contains only quad elements.	Lists all Grquad that contain more than quad elements.	Not supported.	Not supported.
Grbric with inconsistent element	Checks if Grbric contains only brick elements.	Lists all Grbric that contain more than brick elements.	Not supported.	Not supported.
Entity set refers to undefined entity	Checks if sets contain undefined entities.	Lists all sets that reference undefined entities.	Not supported.	Opens the Entity Editor for the entity.
Entity set is referring to unresolved entity	Checks if a set contains unresolved entities.	Lists all sets which refer to unresolved entities.	Removes unresolved entities from the set.	Not supported.
Parameter and set title with &	Checks if a set name has "&" when a parameter is defined in the model.	Lists all sets whose names have "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor for the entity.
Sets with invalid name	Checks if a set name is no more than 100 characters and does not begin with "/".	Lists all sets whose names are greater than 100 characters or begin with "/".	Not supported.	Opens the Entity Editor for the entity.

Name	Description	Result	Auto correction action	Manual correction action
Grbeam and referenced entity not in same include	Checks if a Grbeam and its associated entities are in the same Include file.	Lists all Grbeams which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Grbric and referenced entity not in same include	Checks if a Grbric and its associated entities are in the same Include file.	Lists all Grbrics which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Grnod and referenced entity not in same include	Checks if a Grnod and its associated entities are in the same Include file.	Lists all Grnods which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Grpart and referenced entity not in same include	Checks if a Grpart and its associated entities are in the same Include file.	Lists all Grparts which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Grquad and referenced entity not in same include	Checks if a Grquad and its associated entities are in the same Include file.	Lists all Grquads which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Grsh3n and referenced entity not in same include	Checks if a Grsh3n and its associated entities are in the same Include file.	Lists all Grsh3ns which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Grshel and referenced entity not in same include	Checks if a Grshel and its associated entities are in the same Include file.	Lists all Grshels which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.

Name	Description	Result	Auto correction action	Manual correction action
Grspri and referenced entity not in same include	Checks if a Grspri and its associated entities are in the same Include file.	Lists all Grspris which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Grtruss and referenced entity not in same include	Checks if a Grtruss and its associated entities are in the same Include file.	Lists all Grtrusses which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Surf and referenced entity not in same include	Checks if a Surf and its associated entities are in the same Include file.	Lists all Surfs which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Bcs, grnod, skew not in same include	Checks if Bcs, Grnod, and skew are in same include.	Lists all bcs that have nodes or skew which are not in same include.	Not supported.	Opens the Entity Editor of the load collector entity.
Inivel, grnod and skew not in same include	Checks if Inivel, Grnod and skew are in same include.	Lists all inivel whose nodes or skew are not in same include.	Not supported.	Opens the Entity Editor of the load collector entity.
Impvel, and referenced entities not in same include	Checks if impvel, grnod, sensor, skew and curves are in the same include.	Lists all impvel with nodes, sensor, skew, or curves that are not in same include as itself.	Not supported.	Opens the Entity Editor of the load collector entity.
Impdisp, and referenced entities not in same include	Checks if impdisp, grnod, sensor, skew, and curves are in the same include.	Lists all impdisp with nodes, sensor, skew, or curves that are not in same include as itself.	Not supported.	Opens the Entity Editor of the load collector entity.

Name	Description	Result	Auto correction action	Manual correction action
Pload, referenced entities not in same include	Checks if Pload, Surf, sensor, Skew, and Curves are in the same include.	Lists all pload with surfs, sensor, contact surfs, skew, or curves that are not in same include as itself.	Not supported.	Opens the Entity Editor of the load collector entity.
Grav, and referenced entities not in same include	Checks if Grav, Grnod, sensor, Skew, and curves are in the same include.	Lists all grav with nodes, sensor, skew, or curves that are not in same include as itself.	Not supported.	Opens the Entity Editor of the load collector entity.
Parameter and loadcol title with &	Checks if a load collector name has "&" when a parameter is defined in the model.	Lists all load collectors whose names have "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor for the entity.
Detpoint and referenced entities not in same include	Checks if a Detpoint and its associated entities are in the same Include file.	Lists all Detpoints which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Activ and references entities not in same include	Checks if an Activ and its associated entities are in the same Include file.	Lists all Activs which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Unresolved ids in Activ	Checks if an Activ contains unresolved entities.	Lists all Activs which reference entities that are unresolved.	Not supported.	Opens the Entity Editor for the entity.
Cload and referenced entities not in same include	Checks if a Cload and its associated entities are in the same Include file.	Lists all Cloads which reference entities that are	Not supported.	Opens the Entity Editor for the entity.

Name	Description	Result	Auto correction action	Manual correction action
		not in the same Include file.		
Convec and referenced entities not in same include	Checks if a Convec and its associated entities are in the same Include file.	Lists all Convec which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Impacc and referenced entities not in same include	Checks if an Impacc and its associated entities are in the same Include file.	Lists all Impaccs which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Imptemp and referenced entities not in same include	Checks if an Imptemp and its associated entities are in the same Include file.	Lists all Imptemps which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Pfluid and referenced entities not in same include	Checks if a Pfluid and its associated entities are in the same Include file.	Lists all Pfluids which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Radiation and referenced entities not in same include	Checks if a Radiation and its associated entities are in the same Include file.	Lists all Radiations which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Detline and referenced entities not in same include	Checks if a Detline and its associated entities are in the same Include file.	Lists all Detlines which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Waveshapers and referenced entities not in same include	Checks if a Waveshaper and its associated entities are in the same Include file.	Lists all Waveshapers which reference entities that are	Not supported.	Opens the Entity Editor for the entity.

Name	Description	Result	Auto correction action	Manual correction action
		not in the same Include file.		
Laser and referenced entities not in same include	Checks if a Laser and its associated entities are in the same Include file.	Lists all Lasers which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Inout and referenced entities not in same include	Checks if an Inout and its associated entities are in the same Include file.	Lists all Inouts which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Loadcols with Invalid name	Checks if a load collector name is not more than 100 characters and does not begin with a "/".	Lists all load collectors whose names are greater than 100 characters or begin with "/".	Not supported.	Opens the Entity Editor for the entity.
Unused BeamSections	Checks if any beamsection defined in the model is unused.	Lists all of the unused beamsections in the model.	Deletes the unused beamsections.	Not supported.
Unused parameters	Checks if there are any unused parameters in the model.	Lists all parameters that are unused/not referenced in another entity.	Deletes unused parameters.	Not supported.
Parameter definition in include file	Checks if parameters are present in include files. If yes, then give a warning. PARAMETER/LOCAL is currently not supported, hence the check.	Lists all parameters that are defined in the include file.	Not supported.	Opens the Entity Editor of the parameter entity.
Fcut is greater than or equal to 0 for Accel	Checks if Fcut is greater than zero on an Accel card.	Lists all Accels with Fcut less than or equal to zero.	Not supported.	Opens the Entity Editor for the entity.

Name	Description	Result	Auto correction action	Manual correction action
More than one Accel on the same node	Checks if more than one Accel is acting on the node.	Lists all Accels repeated on the node.	Not supported.	Opens the Entity Editor for the entity.
Accel and referenced entities not in same include	Checks if an Accel and its referenced entities are in the same Include file.	Lists all Accels which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Node id and skew id for accel card are unresolved	Checks if node_id and skew_id are unresolved for the accel card.	Lists all Accels with an unresolved node_id or skew_id.	Not supported.	Opens the Entity Editor for the entity.
Nodes with same skew in different TH	Checks if the same node with the same system is referenced in more than one TH.	Lists all output blocks that contain a node that is repeated with the same system.	Not supported.	Opens the Entity Editor for the entity.
TH with repeated entities	Checks if entities are repeated within each TH.	Lists all output blocks that contain repeated node IDs.	Not supported.	Opens the Entity Editor for the entity.
Parameter and output block title with &	Checks if an output block name has "&" when a parameter is defined in the model.	Lists all output blocks whose names have "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor for the entity.
Output blocks with invalid name	Check if an output block name is no more than 100 characters and does not begin with "/".	Lists all output blocks whose names are greater than 100 characters or begin with "/".	Not supported.	Opens the Entity Editor for the entity.

Name	Description	Result	Auto correction action	Manual correction action
Th and referenced entities not in same include	Checks if Th and its associated entities are in the same Include file.	Lists all Th which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Parameter and contact surf title with &	Checks if a contact surface name has "&" when a parameter is defined in the model.	Lists all contact surfaces whose names have "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor for the entity.
Contact surfs with invalid name	Checks if a contact surface name is not more than 100 characters and does not begin with "/".	Lists all contact surfaces with names greater than 100 characters or begins with "/".	Not supported.	Opens the Entity Editor for the entity.
Monvol, surfs, sensors, curves not in same include	Checks if a control volume and its associated surface, sensor, or curve are in the same Include file.	Lists all control volumes which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Control vols with invalid name	Checks if a control volume name is not more than 100 characters and does not begin with "/".	Lists all control volumes with names greater than 100 characters or begin with "/".	Not supported.	Opens the Entity Editor for the entity.
Parameter and control vol title with &	Checks if a control volume name has "&" when a parameter is defined in the model.	Lists all control volumes whose names have "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor for the entity.

Name	Description	Result	Auto correction action	Manual correction action
Incompatible material	Checks if materials and properties are compatible with xref and refsta.	Lists all xref/ refsta with incompatible materials or properties.	Not supported.	Opens the Entity Editor for the entity.
Eref incompatible material	Checks if materials and properties are compatible with Eref.	Lists all Eref with incompatible materials or properties.	Not supported.	Opens the Entity Editor for the entity.
XRef or Refsta, and nodes not in same include	Checks if Xref/Refsta and its nodes are in the same include.	Lists all impdisp with nodes, skew, or curves that are not in the same include as itself.	Not supported.	Opens the Entity Editor of the control volume entity.
Airbag1 and referenced entities not in same include	Checks if an Airbag1 and its associated entities are in the same Include file.	Lists all Airbag1s which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Area and referenced entities not in same include	Checks if an Area and its associated entities are in the same Include file.	Lists all Areas which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Commu1 and referenced entities not in same include	Checks if a Commu1 and its associated entities are in the same Include file.	Lists all Commu1s which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Fvmbag1 and referenced entities not in same include	Checks if a Fvmbag1 and its associated entities are in the same Include file.	Lists all Fvmbag1s which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.

Name	Description	Result	Auto correction action	Manual correction action
Pres and referenced entities not in same include	Checks if a Pres and its associated entities are in the same Include file.	Lists all Pres which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Perfect gas and referenced entities not in same include	Checks if an Perfect gas and its associated entities are in the same Include file.	Lists all Perfect gases which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Airbag and referenced entities not in same include	Checks if an Airbag and its associated entities are in the same Include file.	Lists all Airbags which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Commu and referenced entities not in same include	Checks if a Commu and its associated entities are in the same Include file.	Lists all Commus which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editorfor the entity.
Fvmbag and referenced entities not in same include	Checks if a Fvmbag and its associated entities are in the same Include file.	Lists all Fvmbags which reference entities that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Sections, N1, N2, N3 not in same include	Checks if a Cross Section and its associated nodes are in the same Include file.	Lists all Cross sections which reference nodes that are not in the same Include file.	Not supported.	Opens the Entity Editor for the entity.
Sections, sets and frames not in same include	Checks if a Cross section and its associated sets and frames are in the same Include file.	Lists all Cross sections which reference sets and frames that are not in the	Not supported.	Opens the Entity Editor for the entity.

Name	Description	Result	Auto correction action	Manual correction action
		same Include file.		
Parameter and cross section title with &	Checks if a cross section name has "&" when a parameter is defined in the model.	Lists all cross sections whose names have "&" when a parameter is defined in the model.	Not supported.	Opens the Entity Editor for the entity.
Cross sections with invalid name	Checks if a cross section name has no more than 100 characters and does begin with "/".	Lists all cross sections whose names are greater than 100 characters or begin with "/".	Not supported.	Opens the Entity Editor for the entity.
Prop17 Incompatible with /`INISHE_ORTHO_` or /`INISH3_ORTHO_`		Lists all properties linked to elements referenced by `/INISHE/ORTHO_LOC` or `/INISH3/ORTHO_LOC` tables.	Not supported.	Opens the Entity Editor for the property entity.

Info

Name	Description	Result	Auto correction action	Manual correction action
Number of defined materials	Checks for the number of defined material entities in the model.	Lists all materials in the model.	Not supported.	Not supported.
Number of undefined materials	Checks for the number of undefined material entities in the model.	Lists all of the undefined materials.	Not supported.	Not supported.

Name	Description	Result	Auto correction action	Manual correction action
Number of defined properties	Checks for the number of defined property entities in the model.	Lists all of the defined properties.	Not supported.	Not supported.
Number of undefined properties	Checks for the number of undefined property entities in the model.	Lists all of the undefined properties.	Not supported.	Not supported.
Number of groups	Checks for the number of group entities in the model.	Lists all group entities.	Not supported.	Not supported.
Nodes not connected to elements	Checks if there are any nodes that are not connected to elements.	Lists all of the nodes in the model that are not connected to elements.	Not supported.	Not supported.
Number of defined curves	Checks for the number of defined curve entities in the model.	Lists all of the defined curves.	Not supported.	Not supported.
Number of undefined curves	Checks for the number of undefined curve entities in the model.	Lists all of the undefined curves.	Not supported.	Not supported.
Number of defined sets	Checks for the number of defined set entities in the model.	Lists all of the defined sets.	Not supported.	Not supported.
Number of undefined sets	Checks for the number undefined set entities in the model.	Lists all of the undefined sets.	Not supported.	Not supported.
Number of output blocks	Checks for the number of output blocks in the model.	Lists all of the output blocks.	Not supported.	Not supported.

Part Browser

Use the Part Browser to create, organize and manage the CAE part structure/hierarchy.

To open the Part Browser, click **View** > **Browsers** > **HyperMesh** > **Part Browser** from the menu bar.

The browser opens in its own tab in the tab sidebar. You can undock the browser so that it appears as a free-floating dialog.

You can undo and redo actions made in the Part Browser using the Undo and Redo commands on the toolbar. The undo/redo history will be reset as soon as you create a part or part assembly. Edits made to parts and part assemblies cannot be undone, whereas edits made to components can be undone.

Part Browser Interface

The Part Browser consists of two panes; the first pane displays the model structure, part assembly, and part attributes and the second pane displays the Entity Editor.

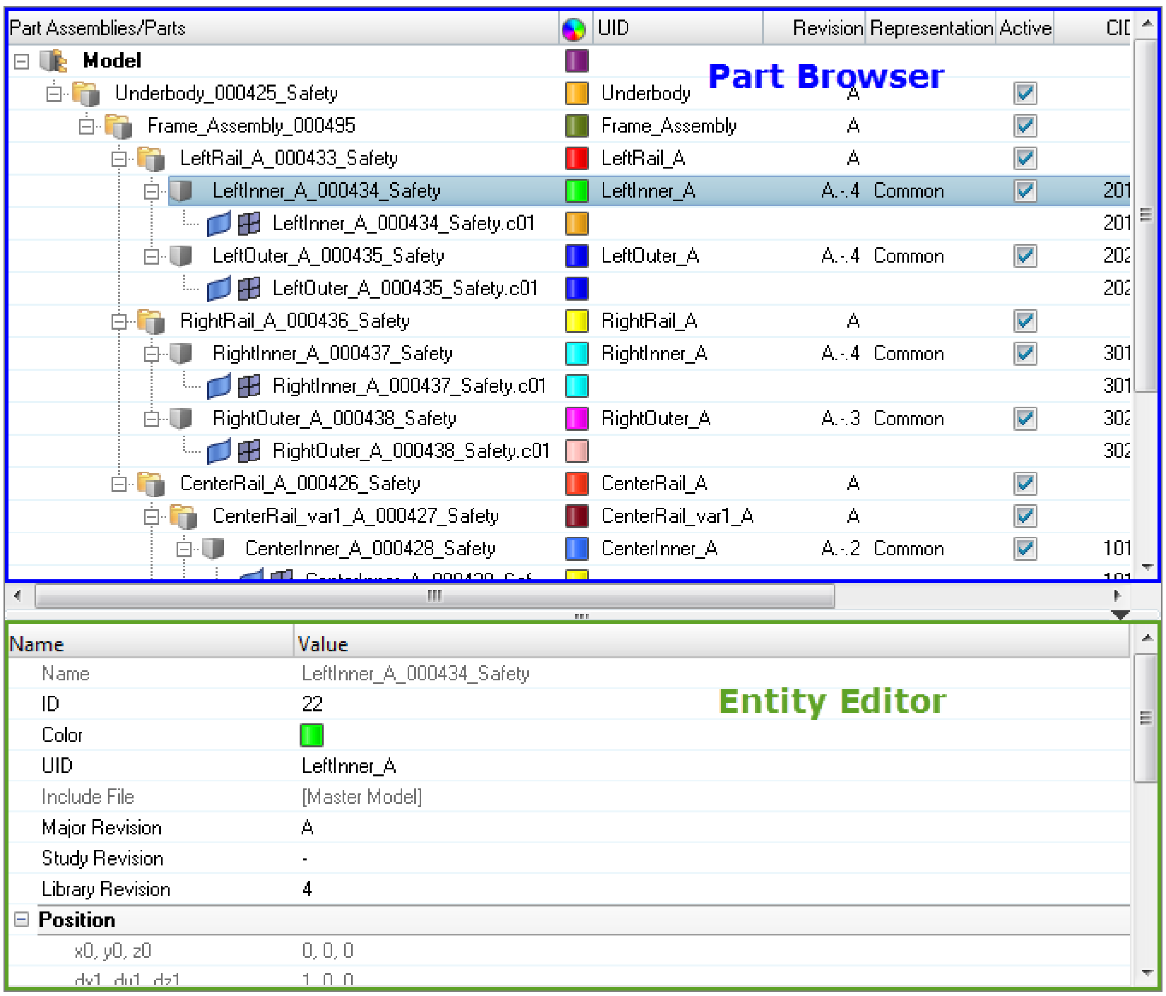


Figure 459:

By default, the following attributes are listed as columns in the first pane.

Add or remove columns by right-clicking on a column and checking/unchecking the appropriate attribute. You can also select **Column Visibility** from the right-click context menu to invoke the **Column Visibility** dialog, from which you can turn the display of columns on/off.

Sort by a particular entity in the browser by clicking on the heading of each column. For fields that are numeric and/or alphanumeric, repeated clicks toggles between ascending and descending order.

Column	**Description**
Entity	List of the Model, Part Assemblies, Parts and Components. By default the browser is displayed in Hierarchical View.
Color	Displays the Component and Part entity colors.
UID	Displays the Part Assembly and Part Unique IDs.
Revision	Displays the Major revision, Study revision, and Library Part revision.
Representation	Displays the in-session/loaded representation.
Active	Displays the Part entity active status. For a full description of usage, refer to Configuration Management.
CID	Displays the entity specific Component ID's. At the Part level it displays IDs for owned components. At Component level it shows the ID of components.
PID	Displays the entity specific Property ID's. At the Part level it displays IDs for referenced properties. At Component level it shows the ID of the referenced property.
MID	Displays the entity specific Material ID's. At the Part level it displays IDs for referenced materials. At Component level it shows the ID of the referenced material.
Material	Displays the material name.
Thickness	Displays the entity specific thickness. At the Part level it displays the thicknesses of the referenced properties. At Component level it shows the thickness of the referenced property.

PDM metadata captured upon the importation of a BOM via the Import tab will also be listed as columns.

Column	Description
PDM PID	PDM Property ID metadata.
PDM MID	PDM Material ID metadata.
PDM Material	PDM Material name metadata.
PDM Thickness	PDM Thickness metadata.
PDM MeshFlag	PDM Mesh metadata (not case sensitive).
PDM Variant Condition	If non-empty, displays the part that is used as a variant in one or multiple part configurations. It is user editable in the Entity Editor.
PDM Variant Scope	Along with the Variant Condition attribute, displays which part configurations the part belongs to as a variant. It is user editable in the Entity Editor.

View Modes

At the top of the Part Browser, you can access different views to be used to create and organize parts, part assemblies, part sets and configurations.

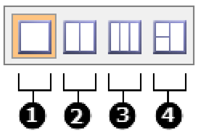


Figure 460:

1. **Part View**. Displays a single view in the Part Browser, from which you can create, organize, and manage parts, part assemblies, and components in the CAE part structure/hierarchy.
2. **Part Set View**. Splits the Part Browser into two views: Part Set and Part. In the Part Set view you can create, organize and, manage part sets.
3. **Configuration View**. Splits the Part Browser into three vertical views: Configuration, Part Set, and Part. In the Configuration view you can create, organize and, manage configurations.
4. **Configuration View, Split Left.** Splits the Part Browser into three views: Configuration, Part Set, and Part. In the Configuration view you can create, organize and, manage configurations.

Part View Modes

In Part view, change the display of entities in the browser using the predefined browser views.

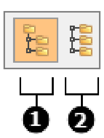


Figure 461:

1. **Hierarchical Part View**. Displays all entities, including components, in a hierarchical view.

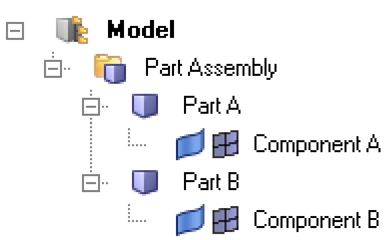
   

   Figure 462:

2. **Flat Part View**. Displays all entities, with the exception of components, in a flat list. Part Assemblies and Parts are logically grouped into their own collectors.

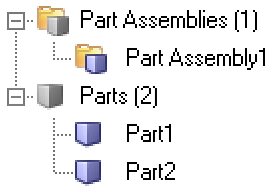


Figure 463:

For example, in Flat View, you can use the Query Builder to isolate part entities. In the example BOM shown above, setting the entity type filter to **Part** will result in the following view.

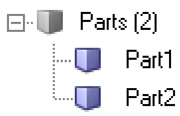


Figure 464:

Query Builder

Use the query builder to find and filter entities by building advanced filters for attributes listed in browser columns.

Recent filters are saved and can be quickly accessed by clicking .

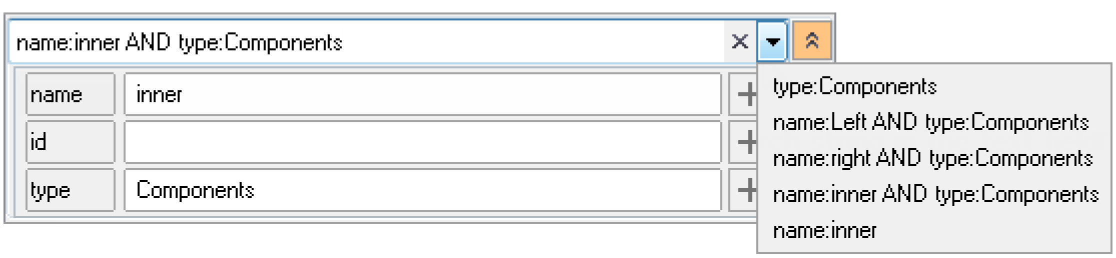


Figure 465:

Interactively Build Queries

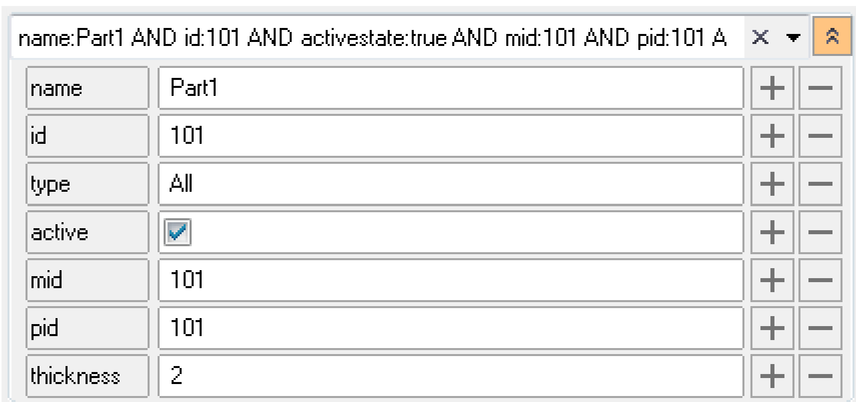


Figure 466:

Tip:
- To select multiple entity types, press and hold Control.
- Add additional attribute fields by clicking +. Remove attribute fields by clicking −.

Manually Build Queries

Manually build queries by typing into the text field.

1. To expose the interactive query builder, click the Expand/Collapse icon .

   By default, the name, id, and type attribute fields are exposed.
2. Filter via name and/or id by typing a string into the text field and pressing Enter.
3. Change the entity type by clicking on the type field and selecting a desired entity type.
4. Apply your selection by clicking .

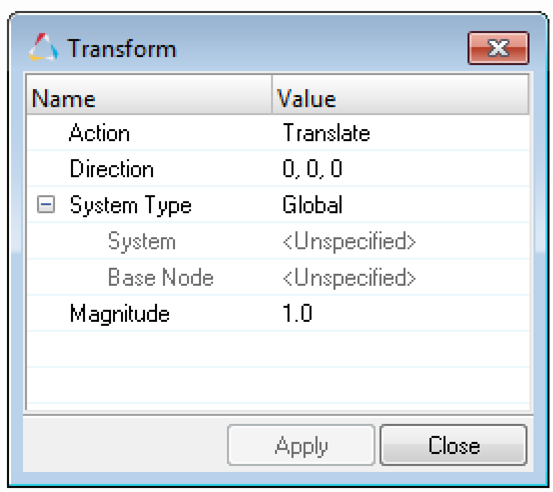
   An example of an interactively defined query is shown in the image below. In this example, four additional attribute fields were added.

Entering this string...	Returns this result...
2	All entities that contain the number 2 in their ID.
"2"	All entities with the ID 2.
shell	All entities that contain shell in their name.
"shell"	All entities with the name shell.
name: shall AND id: 2	All entities that contain shell in their name and 2 in their ID.
id:>10 AND id:<20 AND name:cent	All entities that contain cent in their name and have an ID greater than 10 and less than 20.
1-4	All entities with the IDs 1, 2, 3, and 4.
type:comp AND id:1-4	All components with the IDs 1, 2, 3, and 4.

Context Menu

Use the Part Browser context menu to invoke key functions and visualization options. Options vary based on the entity type selected.

Option	Description
Create	Creates a Part Assembly, Part, Part Instance, or Component. If Create is invoked in the white space of the browser, the entity will be placed in the root level (Model). If Create is invoked on an entity, such as a Part Assembly, the resulting entity will reside in the selected entity. If a part contains components, and it is selected as the current part, then part assemblies and parts cannot be created from the context menu

Option	Description
	when you right-click in the white space of the browser.
Unused & Empty	Delete unused and/or empty part assemblies and parts.
Create Variants	Creates Part Set(s) per PDM Variant Condition attribute found in the global Part Assembly/Part hierarchy. Only available in the Part Set pane of the Part Set and Configuration views. Part Set(s) will be recreated on each invoke of the operation.
Transform	Invokes the Transform tool, from which you can transform the selected Part Assemblies and/or Parts via the following actions; Translate, Rotate, and Reflect. The Transform tool is automatically invoked on the creation of a Part Instance. Transformations are performed on the contents of the Part, but the transformation matrix is updated on the Part(s).  *Figure 468:*
Delete	Deletes the selected entity. Select the **Delete Part Contents** checkbox to delete the selected entity and all of its contents. For example, if Delete is invoked on a Part

Option	Description
	Assembly that contains a Part (also containing a Component), then the Part Assembly, Part, and Component will be deleted from the model.
Rename	Enables you to rename the selected entity in the browser. Alternatively, entities can also be renamed in the Entity Editor, General Information section.
Make Current	Sets the selected entity to current. The current collector status is indicated in bold. Only applicable to parts.
Move under Current	Moves the selected entities to the current entity. Not applicable to PDM parts.
Move Contents to Current	Moves the selected part's component(s) to the current entity. Only available for parts.
Representations	Displays the Representations sub-menu.
Library	Displays the Library sub-menu. **Sync** Syncs the in-session version with the latest, available version in the Part Library. If the version available in the current session is out of sync with the Library reversion, use Sync to update the session version with the Part Library version. **Libraries** Opens the **Libraries** dialog for library management. **Library Viewer** Opens the **Library Viewer** dialog, which can be used to explore the contents of your library.
Save As	Saves the currently selected Parts/ Part Assembly to a HyperMesh binary file. All child Parts and Components and the Part structure are saved. Parts under different sub assemblies can be

Option	Description
	selected and saved under one HyperMesh binary file.
XRef Entities	Invokes the References Browser for the selected entities, which displays any referenced entities in the model.
Review	Invokes Review mode, which displays selected entities irrespective of their display state, masked, active state (Entity State Browser), but not outside of the spherical clipping (if enabled).
Show	Displays the selected entity in the graphics area. A shaded entity icon in the Part Browser indicates that the entity is displayed in the graphics area.
Hide	Hides the selected entity in the graphics area An opaque entity icon in the Part Browser indicates that the entity is not displayed in the graphics area.
Isolate	Isolates the selected entity in the graphics area, and hides all other entities.
Collapse/Collapse All	Collapses the selected entity folder(s). Collapse All is only available when you invoke the context menu by right-clicking in the white space of the browser.
Expand/Expand All	Expands the selected entity folder(s). Expand All is only available when you invoke the context menu by right-clicking in the white space of the browser.

Entity Editor

Edit part and component attributes in the Entity Editor.

Edits can be made for a single part or component or multiple parts and components. If a component is selected, the Entity Editor displays component specific attributes.

The attributes available in the Entity Editor vary based on the entity type selected.

Part/Part Assembly

General Data

Displays attributes that are unique to the part or assembly, including part/assembly name, Unique ID (UID), color, and include file, and Revision data.

Parts and part assemblies require a unique name and a unique identifier (UID), both of which can contain alphanumeric characters. The UID is an optional field that is utilized in part representation management and import model entity management. For example, when merging a component with a part (module), the two entities will be matched using the UID if it exists. If a UID does not exist, merging will be based on the part name.

Representations Data

Displays attributes that are specific to a part representation, such as representation name, file location, property ID, material ID/name, and thickness. These attributes are non-editable at the part level.

Position Data

Displays the 4 x 3 transformation matrix of a part, namely its translation, rotation, and scaling. These attributes are non-editable.

PDM Data

Displays PDM attributes that were parsed as metadata during the BOM importation, such as PDM ID, PDM Revision, PDM Variant Condition and PDM Variant Scope. This information, namely the PDM PID, PDM MID, PDM Material and PDM Thickness, is used to generate the initial component, property, and material cards upon creation of the common representation. These attributes can be used to cross-reference common and discipline specific mesh representation attributes.

Part Set

General Data

Displays attributes that are unique to the part set, including part/assembly name and ID.

Part Assemblies / Parts Data

Displays the number of parts organized into the selected part set.

Click the Part Assemblies/Part field to select parts and part assemblies to organize into the selected part set. Select entities in the **Select Part Assemblies/Parts** dialog, or from the graphics area when the Parts selector is enabled.

Part Sets Data

Displays the number part sets organized into the selected part set.

Click the Part Sets field to select part sets to organize into the selected part set. Select entities in the **Select Part Sets** dialog, or from the graphics area when the Partsets selector is enabled.

Configuration

General Data

Displays attributes that are unique to the configuration, including name, and ID.

Parts Assemblies / Parts Data

Displays the number of parts organized into the selected configuration.

Click the Part Assemblies/Part field to select parts and part assemblies to organize into the selected configuration. Select entities in the **Select Part Assemblies/Parts** dialog, or from the graphics area when the Parts selector is enabled.

Configurations Data

Displays the number of configurations organized into the selected configuration.
Click the Configurations field to select other configurations to organize into the selected configuration. Select entities in the **Select Configurations** dialog, or from the graphics area when the Configurations selector is enabled.

Part Sets Data

Displays the number part sets organized into the selected configuration.
Click the Part Sets field to select part sets to organize into the selected configuration. Select entities in the Select Part Sets dialog, or from the graphics area when the Partsets selector is enabled.

Active

Indicates the active/inactive state of the selected configuration. This attribute is non-editable. To activate a configuration, enable the configuration's associated checkbox in the Active column of the Configuration view.

Reference Browser

The Reference Browser enables you to quickly understand complex relationships between different entities that constitute the model.

To navigate and understand the relationships between the different entities, double-click on an entity to trigger a cross referencing operation.

Cross referencing operations are recorded in the Reference Browser. To navigate between cross referenced states, click the forward and back buttons in the top, left corner of the browser.

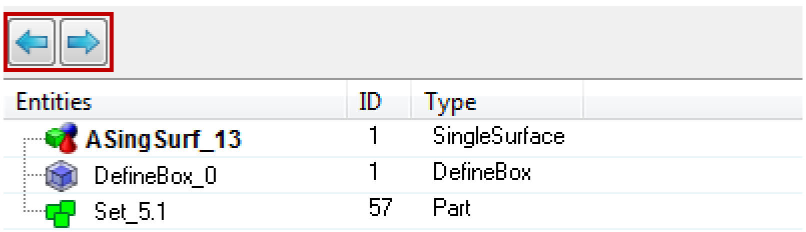


Figure 469:

Forward and back buttons enabled to navigate between cross references states.

Use the Entity Editor to review and edit HyperMesh specific and solver specific data, for one or more of the entities listed in the browser. To invoke the Entity Editor, select the entities you want to review.

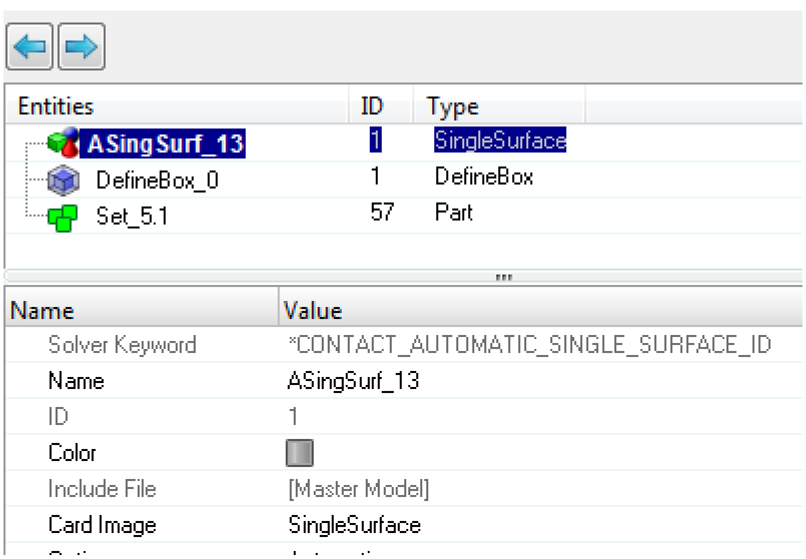


Figure 470:

Entity Editor invoked in the Reference Browser.

Simplified and Detailed View

The Reference Browser consists of two views: Simplified and Detailed. Select which view to display from the right-click context menu.

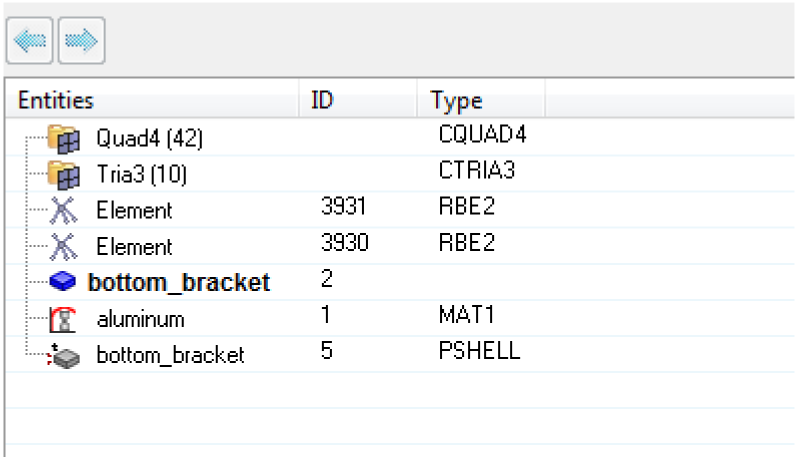


Figure 471: Simplified View

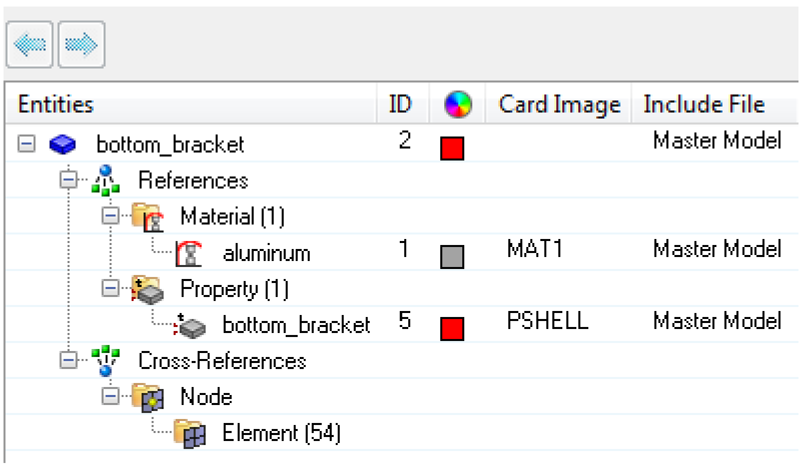


Figure 472: Detailed View

In the Simplified view the name, ID, Type and Include File of the populated entity(s) are displayed. The selected entity(s) for cross referencing is displayed in bold to differentiate. References of the selected entity(s) are listed below the entity(s), and cross references are listed above the entity(s).

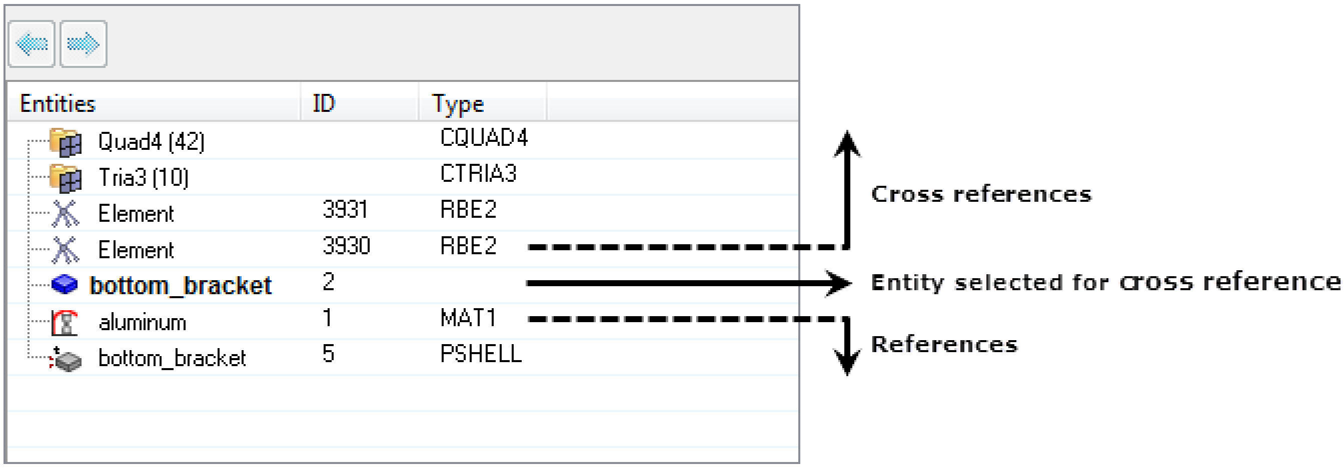


Figure 473: References and Cross References of a Component

Cross referenced entities are shown in a flat list. If there is more than five instance of a given entity or element configuration, then entities will be displayed in a folder. For elements and nodes, the detailed content of the folders is not exposed, as the number of entities cannot be contained in a folder.

Elements are also segregated by type and configuration. Rigid elements, rbe3 and spotweld are listed separately.

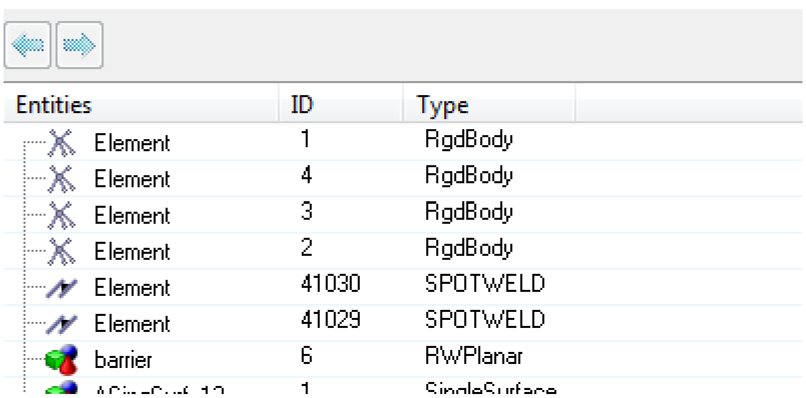


Figure 474: Segregation of Elements by Type and Configuration

In the Detailed view, related entities are broadly classified into Reference and Cross-reference folders, and entities are listed accordingly.

References, represented by , correspond to entities that are referenced by the specified entity. The hierarchy is a view down from the selected entity(s) within the model, listing the entities that are referred to by the selected entity(s). For example, in the case of a component, the References folder lists the elements, material, and property assigned to the component.

Cross-references, represented by , correspond to entities that reference the specified entity. The hierarchy is a view up from the selected entity(s) within the model, listing the entities that refer to the selected entity(s). For example, in the case of a component, the Cross-references folder lists the groups, sets, output blocks, and so on that refer to the selected component. For components, the

Cross-reference folder contains sub folders named Node () and Element (), which lists the entities where the selected component's node or element is referred.

In the Detailed view, the name, ID, color, Card Image and Include File of the populated entity(s) are displayed. Nodes and elements are not listed individually in the Reference or Cross-reference folders, however each entity's count will be displayed next to the folder.

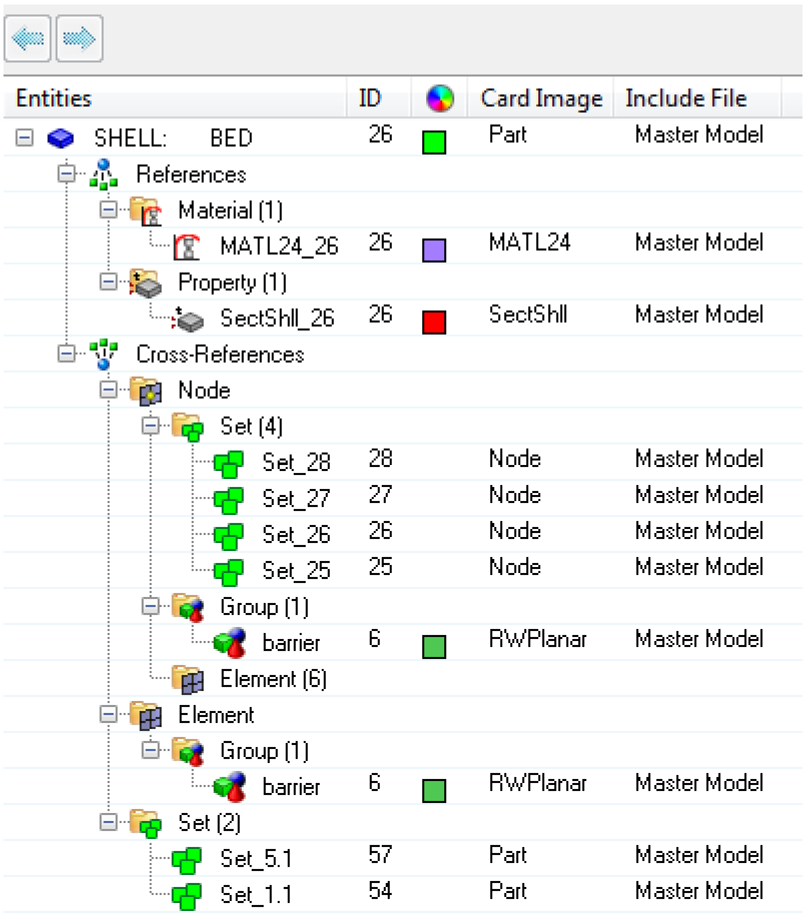


Figure 475: Detailed View in the Reference Browser

Context Menu

The Reference Browser supports its own context menu. To access the context menu, right-click on any entity.

The available menu options vary based on the entity selected.

XRef Entities
: Applies reference on the selected entity and then updates the Reference Browser with new results.

Show
: Displays the selected item in the graphics area if it is currently hidden.

In cases of card images, such as contacts and boundary conditions, that do not have entities of their own but refer to another entity, namely sets, the Reference Browser displays the entities (node, element) that are used in the set that is referred in the selected card image. Also, the Reference Browser displays any handles, geometric representation, associated with entities.

Hide

Turns off the selected item in the graphics area if it is currently visible.
In cases of card images, such as contacts or boundary conditions, that do not have entities of their own, but refers to another entity, namely sets, the entities (node, element) that are used in the set that is referred in the selected card image are hidden. Also, the Reference Browser hides any handles, geometric representation, associated with entities.

Isolate

Displays only the selected item in the graphics area and hides all of the other items.
In cases of card image, such as contacts and boundary conditions, that do not have entities of their own but refer to another entity, namely sets, the Reference Browser isolates the entities (node, element) that are used in the set that is referred in the selected card image. Also, the Reference Browser shows any handles, geometric representation, associated with selected solver entity.

Review

Highlights the selected item in the graphics area and greys out all other items. This option works on all of the individual entities listed in the Reference Browser, except at the folder level.
In cases of card images, such as contacts and boundary conditions, that do not have entities of their own but refer to another entity (namely sets), the Reference Browser highlights the entities that constitute the set. The graphics area remains in that review mode until you select Reset Review.

Card Edit

Opens the Card Editor, from which you can modify any editable input value fields for a selected entity. The modified fields update in the Entity Editor and in all of the browser.

List References

Turns the display of the References folder on and off in the browser. The Reference folder contains all of the entities that are referenced by the selected entity(s). By default the References folder is displayed.

List Cross References

Turns the display of the Cross-references folder on and off in the browser. The Cross-references folder contains entities that reference the selected entity(s). By default, the Cross-references folder is displayed.

Merge Results

Groups the results of the selected entities, which may be of different entity types, and lists them in the browser. By default, the Reference Browser lists each entity separately in the order of its type.

Detailed View

Enables you to switch between Detailed and Simplified view.

Solver Browser

The Solver Browser provides a solver centric view of the model structure in a flat, listed tree structure.

Hierarchical structures are only available for card images that allow variations with themselves. For example, the `MAT` card image has several different material types and each material has its own entity defined in HyperMesh, so a hierarchical structure is used to list them all.

The Solver Browser lists every entity mapped to a solver card image within the session and places those entities into their respective solver card image folders. The total number of entities is displayed in parenthesis next to the entity name. To display every entity type supported by the solver in the browser, go to the **Browser Configuration** dialog, Local Options, and select **Show empty folders**.

In the LS-DYNA user profile, the Solver Browser supports two view modes: Solver and Include Solver view. Switch between the two view modes by clicking their respective icons in the top, left corner of the browser.

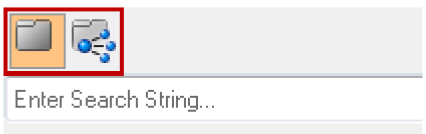


Figure 476:

In the Include Solver view, data is organized into Include files. Data which does not have any references to an Include file is stored in the master model.

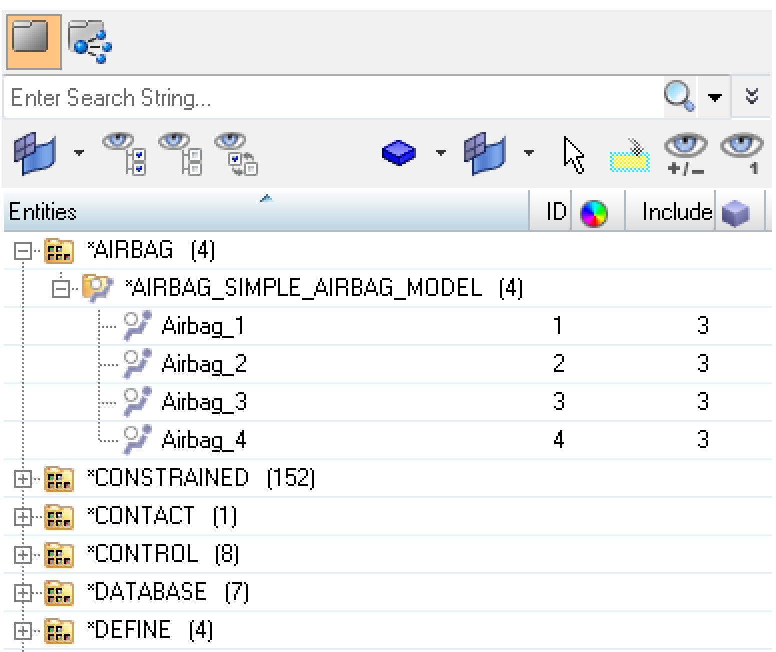


Figure 477: Solver View

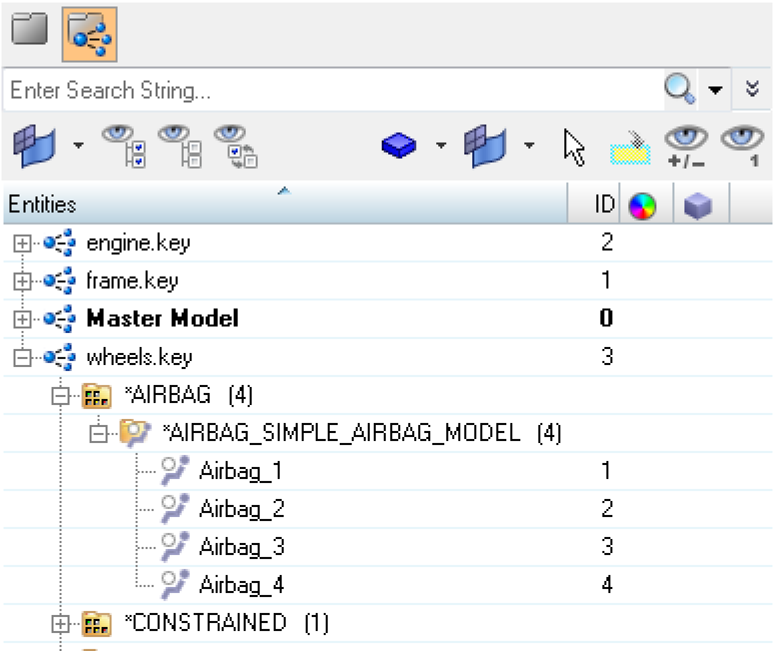


Figure 478: Include Solver View

The Solver Browser includes toolbars, a context menu and controls built into the display. Toolbars provide the ability to show and hide entities (component, material and property) within the model, and add entities to a panel collector. These abilities are collectively tree referred to as display controls and browser modes. The context menu provides basic functions such as card editing, creation, deletion, display control and review.

Browser Attributes (Columns)

By default, the following attributes are listed as columns in the browser.

Add or remove columns by right-clicking on a column and checking/unchecking the appropriate attribute. You can also select **Column Visibility** from the right-click context menu to invoke the **Column Visibility** dialog, from which you can turn the display of columns on/off.

Sort by a particular entity in the browser by clicking on the heading of each column. For fields that are numeric and/or alphanumeric, repeated clicks toggles between ascending and descending order.

Column	Description
Entity	Lists every named entity within the session; however, it does not support non-named entities such as nodes and elements.
ID	Unique entity ID
Color	Entity color
Include	ID of Include file the entity is stored within.

Column	Description
FE-Style	Lists the element style applied to each entity. Click the icon to change the element style.

Context Menu

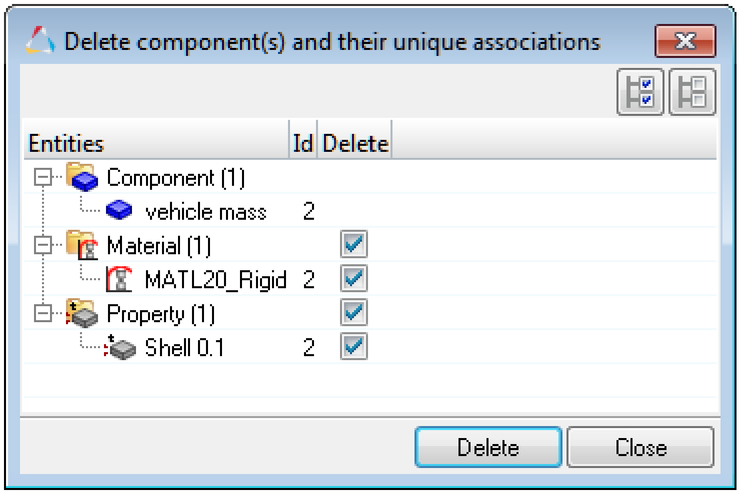
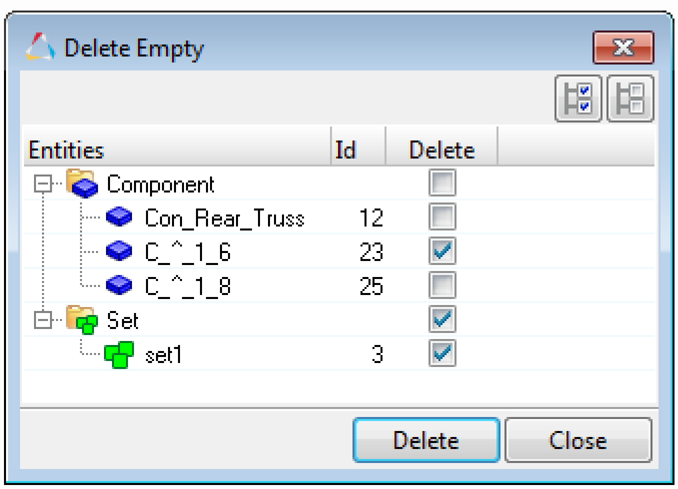
A context menu of actions is available for any selected item in the Solver Browser.

To open the context menu, right-click on either an entity folder or an individual entity.

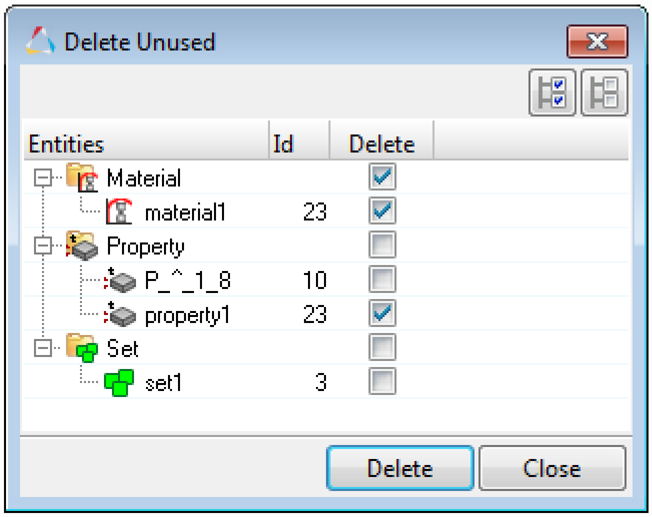
> **Note:** Some of these options vary depending on which user profile you have loaded.

Table 14:

Option	Description
Card Edit	Opens the Card Image panel for the selected card. This operation can be performed on both folder and individual entity levels.
Collapse All	Closes all of the folders in the browser, so that only the top-most level of items displays.
Column Visibility	Turns the display of the Color and FE Style columns in the browser on and off.
Configure Browser	Opens the **Local View Options** dialog, from which you can determine which entities and columns to display in the browser. Global options includes options that are common across all user interfaces; Local options includes solver specific options.
Create	Enables you to create an entity. Once you select a solver keyword, the Entity Editor or panel, applicable to the solver entity, opens.
Delete	Deletes selected item from the session. If the selected item has "children" associated to it, the children are retained and only the entity is deleted. For example, if a contact has a surface and node set associated with it, and the contact is deleted, only the contact card image is deleted. The surface and node set associated with it is retained. This is the same at both folder and individual entity level.
Delete Advanced	Displays a comprehensive preview of the entities uniquely related to the selected component that can also be deleted. Only available for components in the LS-DYNA user profile.

Option	Description
	 *Figure 479:*
Empty	Preview and delete empty collectors. This operation can be performed on several entity types at the same time. To append entity types to the selection, left-click while pressing Control. Only available for the following entity types: Component, Load Collector, System Collector, Set, Group, Assembly, Vector Collector, Output Block, Load Step, Beam Section Collector, Control Volume (OptiStruct) and Multibody.  *Figure 480:*
Expand All	Opens all of the folders in the entire browser, exposing every item nested at every level.

Option	Description
Find Attached	Finds entities attached to the selected card(s) or entity(s). Find attached is implemented only for card images mapped to component collectors, namely PART cards and 1D elements that include beams, mass elements, truss, rigid and joints. Find attached displays elements (0D, 1D) and components that are connected to the selected entity through sharing a common node, connectors, and special connection cards like *CONSTRAINED_EXTRA_NODE, *CONSTRAINED_RIGID_BODY. Only available in the LS-DYNA, Radioss and PAM-CRASH 2G user profiles.
Hide	Turns off the selected item in the graphics area if it is currently visible. In case of card images like contacts, boundary conditions, and so on, that do not have entities of its own but refer to another entity, namely sets, it turns off the entities (node, element) used in the set that is referred in the selected card image. Also, it turns off any handles, geometric representation associated with entities.
Isolate only	Displays selected item in the graphics area and hides all of the other items. In cases of card images, such as contacts and boundary conditions, that do not have entities of its own but refers to another entity, namely sets, the Solver Browser isolates the entities (node, element) used in the set that is referred in the selected card image. Also, the Solver Browser shows any handles, geometric representation, associated with selected solver entity.
Make Current	Makes the selected collector entity the current collector. The current collector status is indicated in bold. New components, loads, beam sections, or multibodies will be created within the respective current collector.
Rename	Enables an editable text field, from which you can rename the selected entity.
Reset Review	Resets the screen from review mode to normal mode.
Review	Invokes Review mode, which displays selected entities irrespective of their display state, masked, active state (Entity State Browser), but not outside of the spherical clipping (if enabled).
Show	Displays the selected item in the graphics screen if it is currently hidden. In the case of card images (contacts, boundary conditions, and so on) that do not have entities of their own but reference other entities (namely sets), the entities (node, element) used in the set that is referenced by the selected card image are displayed. This operation also displays any handles as geometric representation associated with entities.
Unused	Preview and delete unused property collectors, material collectors, curves, and so on.

Option	Description
	This operation can be performed on several entity types at the same time. To append entity types to the selection, left-click while pressing Control. Only available for the following entity types: System, Property, Curve, Material.  *Figure 481:*
XRef Entities	Opens the Reference Browser and displays the relationship of the selected card(s) or entity(s) to other entities in the model in a hierarchical tree structure.

Configure the Solver Browser

Configure the display and behavior of the Solver Browser.

1. Right-click in the browser and select **Configure Browser** from the context menu.
2. Edit local and global options.

   Global options include options that are common across all user interfaces; Local options include solver specific options.

Local Options

Show empty folders
: Display every entity type supported by the solver in the browser.

Maximum number of items
: Change the maximum number of items to display in the browser.

Show/Hide/IsolateOnly Attached

When performing Show/Hide/Isolate Only on loads, load collectors, load steps, groups, and contact surfaces, attached elements will also be included in the operation in the graphics area. This can be observed if the elements are currently masked or the elements belonging to the component's display state is off.

Show/Hide/Isolate Include files with XRef

When performing Show/Hide/Isolate on Include files, the contents in the Include files will also be included in the operation. If you turn this option off, Show/Hide/Isolate will only affect components and other entities that have graphics (vectors, systems, plies, laminates, loads) in the Include file.

Hide Attached

When performing Hide on loads, load collectors, load steps, groups, and contact surfaces, attached elements will also be masked in the graphics area. This could be observed if the elements are currently displayed.

Delete association

Open the **Delete component(s) and unique associations** dialog when Delete Advanced is selected from the context menu. When this box is checked off, the Delete component(s) and unique associations dialog does not open when Delete Advanced is selected from the context menu, and the entities uniquely related to the selected component are automatically deleted. Only available for components in the LS-DYNA user profile.

Global Options

Autofit

Fit the selected entities to the graphics area whether using the context menu or the Selector, Show/Hide, or Isolate functionality to control the display.

Stripe background

Turn the white and gray stripe background in the browser on. When off, the browser background is white.

Utility Menus

The Utility menu contains options that allow you to customize the standard interface to include function buttons, radio options, and text that have HyperMesh-supplied and user-defined macros associated with them.

The menu is located on a tab of the tab area pane(s), and can be shown or hidden from within the View menu.

The Utility menu includes several pages of its own, each dedicated to different tasks. Thus it is actually a group of menus, although only one displays at a time. Each page is associated with a button at the bottom of the Utility menu; clicking one of these buttons opens the page associated with it. Only one button can be depressed at a time, similar to the way that only one radio button can be active at a time - selecting a button de-selects all of the other buttons in the group.

A macro file (`hm.mac`) controls the display and available operations of the Utility menu. Attributes that you can change include:

- The Utility menu page on which the operations appear
- Text to be displayed on each control
- Location and size of the menu
- The help string to be displayed on the menu bar
- The macro to call when each control is used, with optional arguments to pass

Use page number to create multiple pages, so that you can group the macros by type of operation.

Macros may contain any valid command file command, and are enclosed by the `*beginmacro()` and `*endmacro()` commands. Macros may accept variable arguments, passed to them from a control, by using the arguments $1, $2, etc. to specify where the arguments should be substituted. The `*callmacro()` command calls a macro from within another one, which allows you to create groups of standard reusable macros.

When HyperMesh starts, it looks for a macro file named `hm.mac` in the current directory, HOME directory (UNIX only), or the application's base directory. If it finds this macro, HyperMesh runs it automatically to define the attributes and contents of the Utility menu. You may also select and run a macro file after HyperMesh starts from within the Options panel.

The default `hm.mac` file sources the following additional macro files:

`disppage.mac`
: Populates the Display page of the Utility menu.

`geommeshpage.mac`
: Populates the Geom/Mesh page of the Utility menu.

`globalpage.mac`
: Creates the button group that allows you to switch pages.

`qamodelpage.mac`
: Populates the QA/Model page of the Utility menu.

`userpage.mac`
: Populates the User page of the Utility menu.

A `userpage.mac` file may exist in the installation directory for HyperMesh or in the directory from which HyperMesh launches. When HyperMesh starts, it first looks for the `userpage.mac` file in the directory from which it launches and then in the installation directory. UNIX users also have the option of putting the `userpage.mac` file in their home directory. This file defines the attributes and contents of the User page of the Utility menu.

By default, the Utility menu displays when HyperMesh starts, but display of the menu is controlled by a command in the HyperMesh Configuration.

> **Note:** While macros offer a great deal of flexibility, you must remember that once a macro is executed, there is no way to cancel the execution or reject the results, and a macro may not be called recursively.

QA/Model Utility Menu

The QA Utility menu contains many tools to help you quickly review and clean up the quality of a pre-existing mesh.

The element quality criteria used by these tools comes directly from the values entered on the Check Elements panel. Since the criteria on that panel are customizable, the quality criteria used by these macros remains consistent with those used throughout the rest of HyperMesh and can be indirectly adjusted by changing the settings on the Check Elements panel.

There are eight tools to isolate elements that fail certain element check criteria:

Table 15:

Tool	Description
Length	This macro checks all the displayed elements against the minimum length criteria. If any elements fail the criteria, it displays the failed elements and masks the remaining elements. If none of the displayed elements fail the criteria, it displays a message and leaves the model display unchanged.
Jacob (Jacobian)	This macro checks all the displayed elements against the maximum Jacobian value. If any elements fail the criteria, it displays the failed elements and masks the remaining elements. If none of the displayed elements fail the criteria, it displays a message and leaves the display unchanged.
Warp (warpage)	This macro checks all the displayed elements for their warpage. If any elements fail the warpage test, it displays the failed elements and masks the remaining elements. If none of the displayed elements fail the criteria, it displays a message and leaves the display unchanged.
Aspect (aspect ratio)	This macro checks all the displayed elements for their aspect ratio. If any elements fail the criteria, it displays the failed elements and masks the

Tool	Description
	remaining elements. If none of the displayed elements fail the criteria, it displays a message and leaves the display unchanged.
Max ang: Q (quad)	This macro checks all the displayed quad elements against the maximum internal angle. If any elements fail the criteria, it displays the failed elements and masks the remaining elements. If none of the displayed elements fail the criteria, it displays a message and leaves the display unchanged.
Max ang: T (tria)	This macro checks all the displayed tria elements against the maximum internal angle. If any elements fail the criteria, it displays the failed elements and masks the remaining elements. If none of the displayed elements fail the criteria, it displays a message and leaves the display unchanged.
Min ang: Q (quad)	This macro checks all the displayed quad elements against the minimum internal angle. If any elements fail the criteria, it displays the failed elements and masks the remaining elements. If none of the displayed elements fail the criteria, it displays a message and leaves the display unchanged.
Min ang: T (tria)	This macro checks all the displayed tria elements against the minimum internal angle. If any elements fail the criteria, it displays the failed elements and masks the remaining elements. If none of the displayed elements fail the criteria, it displays a message and leaves the display unchanged.

You can use the following macros to quickly modify any elements that fail the element checks.

Table 16:

Tool	Description
Split Warped	Checks all displayed quad elements for warp exceeding the acceptable value. Each element failing this criterion is then split along its diagonal to form two tria elements instead of the original quad.
Find Attached	Finds all elements attached to displayed elements. Attached elements are those that are physically attached to each other via common nodes.
Find Attached (Tied)	Finds all elements tied to displayed elements. Tied elements are those that are tied to each other, but are not attached physically. See Find Attached (Tied).

Tool	Description
Remesh	Remeshes the selected elements plus one, two, or three attached layers of elements (one button for each). The remesh uses the current size, does not break connectivity, and uses the mixed element type.
Smooth	Applies the smoothing algorithm to the selected elements plus one, two, or three attached layers of elements (one button for each).
Quality Report	Brings up a user interface, from which you can set the various quality values and check the quality of all the 2D elements in the model. The results are shown as the number of elements and percentage of elements failing each criterion. You can also export the results to a text file using save as. **Note:** Changing the criteria on this report interface does not change the settings in the Check Elements panel. They only affect the report.
Model Tour	Opens a macro, from which you can review the selected components individually. This macro displays the component name, number of elements in that component and their ID range. It also displays a dialog from which you can review the free edges of the component and any elements attached to the component.
BOM Comparison Tool	Reads a generic Bill Of Materials file and provides an interface to manipulate data in the BOM as well as the corresponding FE model.

The model tools included on this page are:

Table 17:

Tool	Description
Load Size	These numbered buttons represent different display sizes for load indicators: 0 is the smallest, while 3 is the largest. Since these buttons affect all loads, including forces, pressures, constraints, and so on, the numbers do not directly correspond to any specific values or ratios. **Note:** This only affects the graphical display of load indicators. It does not change the load magnitudes.
Find Elems >> Loads	Automatically finds all elements directly attached to any and all load indicators. If masked, these elements are unmasked.

Tool	Description
Find Comps >> Loads	Automatically finds all components directly attached to any and all load indicators. If masked, these comps are unmasked.
Find Loads >> Comps	Automatically finds all loads directly attached to a selected component. If masked, these loads are unmasked.
Find Elems >> Connectors	Automatically finds all elements directly attached to any and all connectors. If masked, these elements are un-masked.

Find Attached (Tied)

Find elements that are connected to displayed elements using the Find Attached (Tied) option.

Tied elements are those that are tied to each other, but are not attached physically. This option differs from the Find Attached option, which returns elements that are physically attached to each other via common nodes.

Find Attached (Tied) is available for the LS-DYNA user profile on the QA/Model Utility menu.

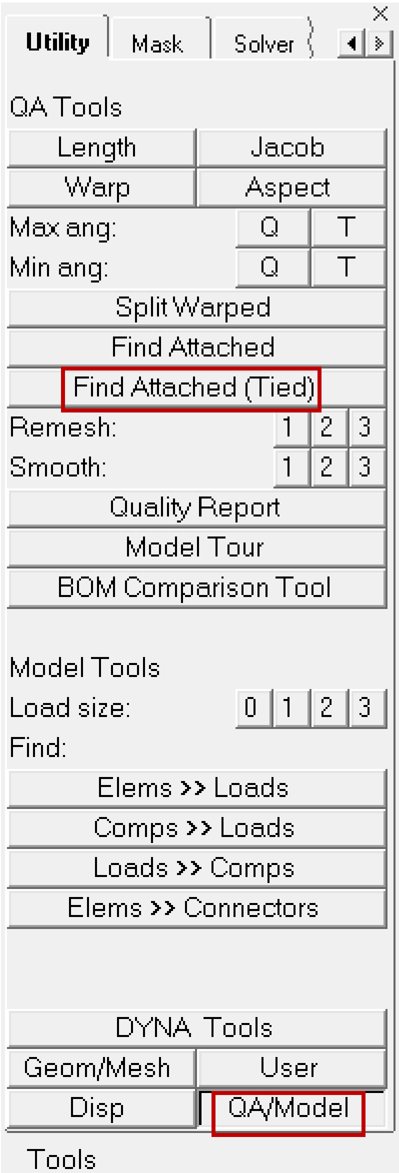


Figure 482:

The relationship between tied elements is defined using the following CONTACT keywords:

Element Type	Contact Keywords
ContactSpotWeld	*CONTACT_SPOTWELD
NodesToSurface	*CONTACT_TIED_NODES_TO_SURFACE *CONTACT_TIEBREAK_NODES_TO_SURFACE *CONTACT_TIED_SHELL_EDGE_TO_SURFACE
SurfaceToSurface	*CONTACT_TIEBREAK_SURFACE_TO_SURFACE

Element Type	Contact Keywords
	*CONTACT_TIED_SURFACE_TO_SURFACE
SlidingOnly	*CONTACT_SLIDING_ONLY
ConstdTieBreak	*CONSTRAINED_TIE-BREAK
XtraNode	*CONSTRAINED_EXTRA_NODES_NODE *CONSTRAINED_EXTRA_NODES_SET
TiedNodes	*CONSTRAINED_TIED_NODES_FAILURE
ConstRigidRbody	*CONSTRAINED_RIGID_BODIES

Features

- Find Attached (Tied) returns any tied elements connected via TIED_CONTACTS as defined using LS-DYNA.
- You can perform a Find Attached (Tied) operation on one or more displayed elements or components.
- Element input and output is based on dimension as follows:
-

Input (displayed)	Output (attached or tied)
1D/3D elements	2D elements
2D elements	1D/3D elements
2D elements	2D elements **Note:** For ContactSpotweld, the 1st and 2nd use case is valid.

Use Cases

Use Case 1:

Two independent beam nodes are connected to a common layer.

Result:

If you start with the green shell and 1D beam in image 1., then the Find Attached (Tied) operation returns the elements in the following sequence:

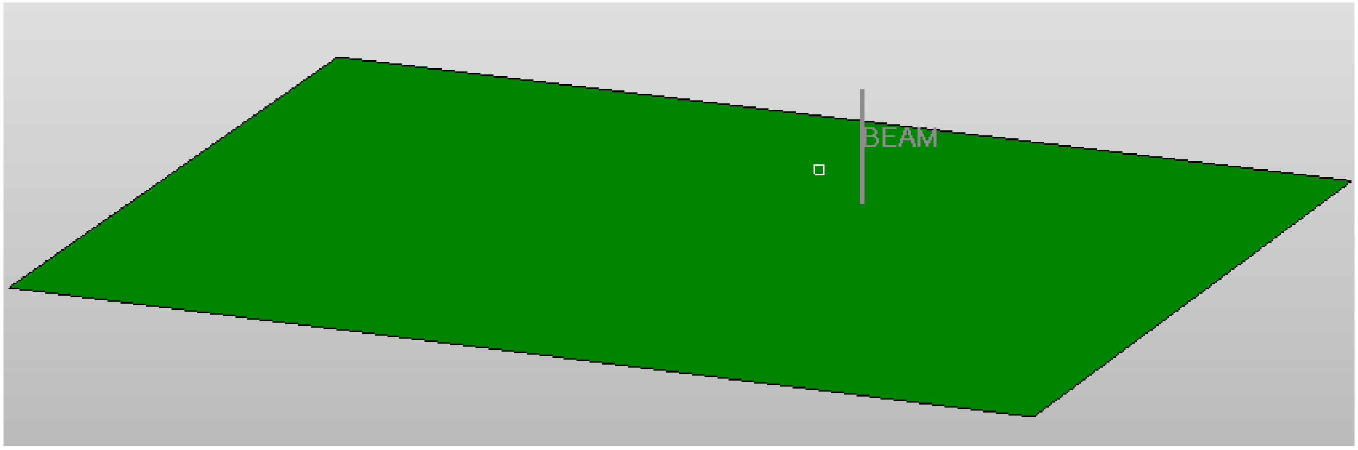


Figure 483:

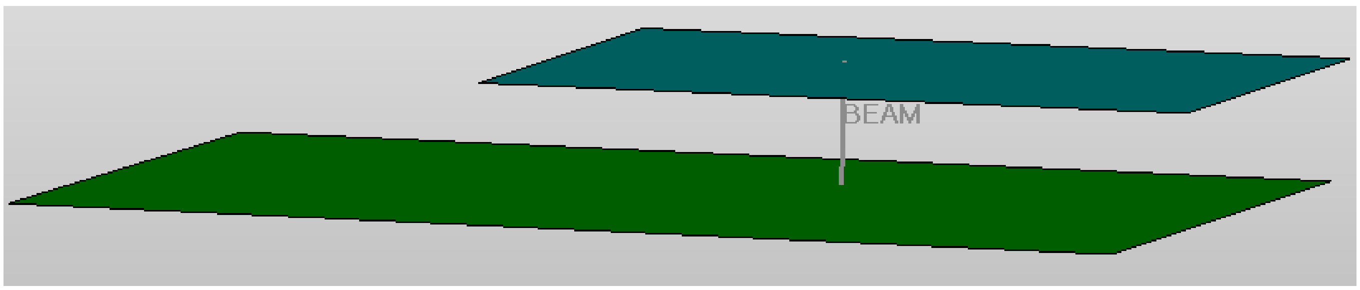


Figure 484:

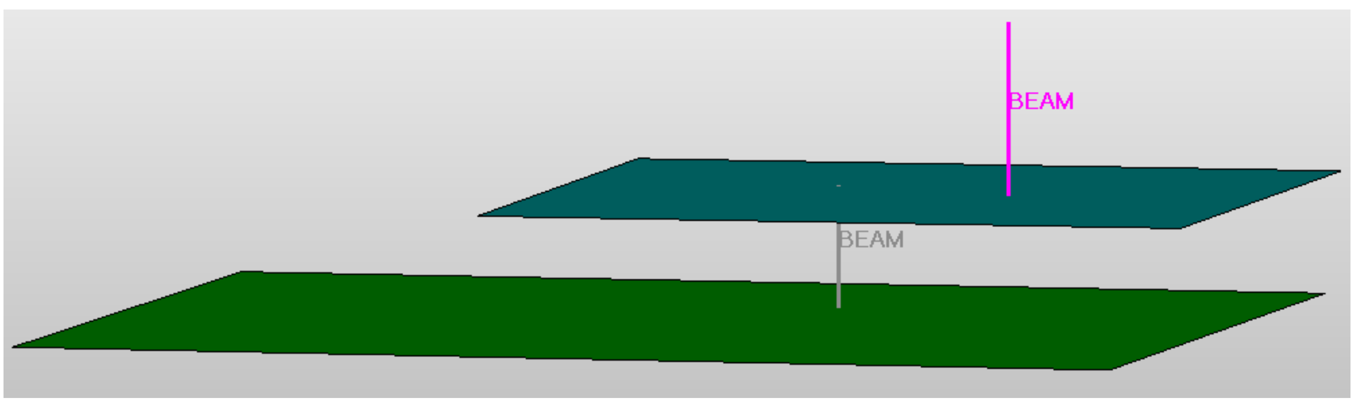


Figure 485:

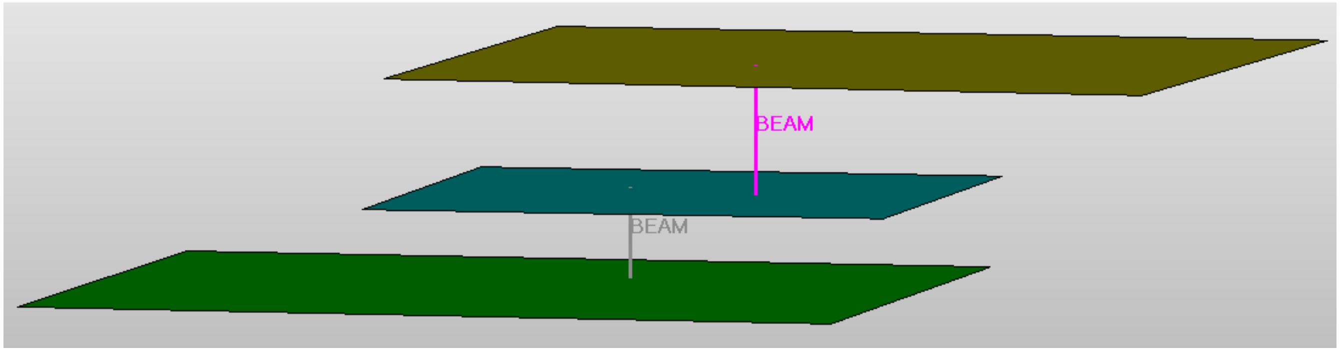


Figure 486:

Use Case 2:

A common node between two 1D elements.

Result:

If you start with the 1D beam in image 1., then the Find Attached (Tied) operation returns the elements in the following sequence:

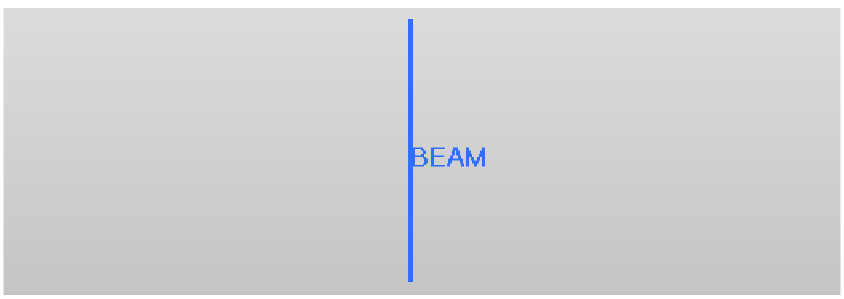


Figure 487:

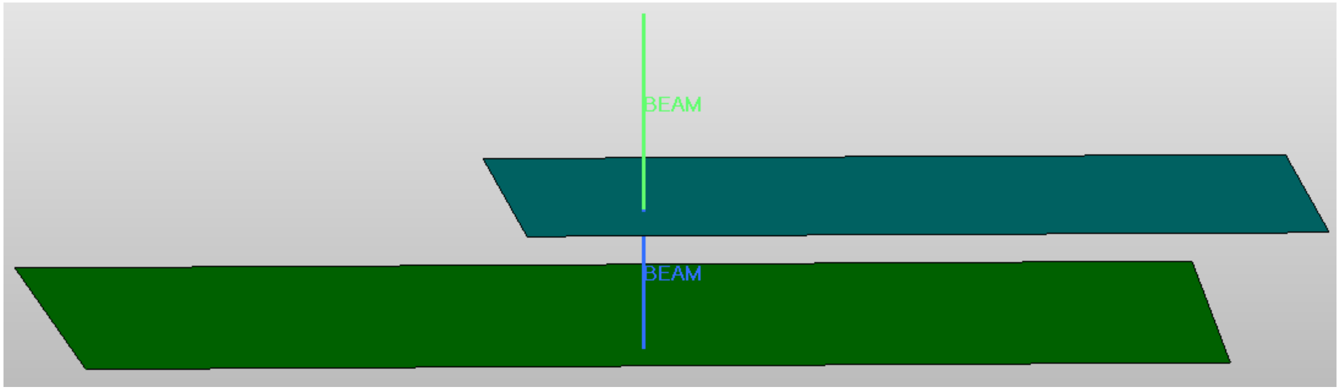


Figure 488:

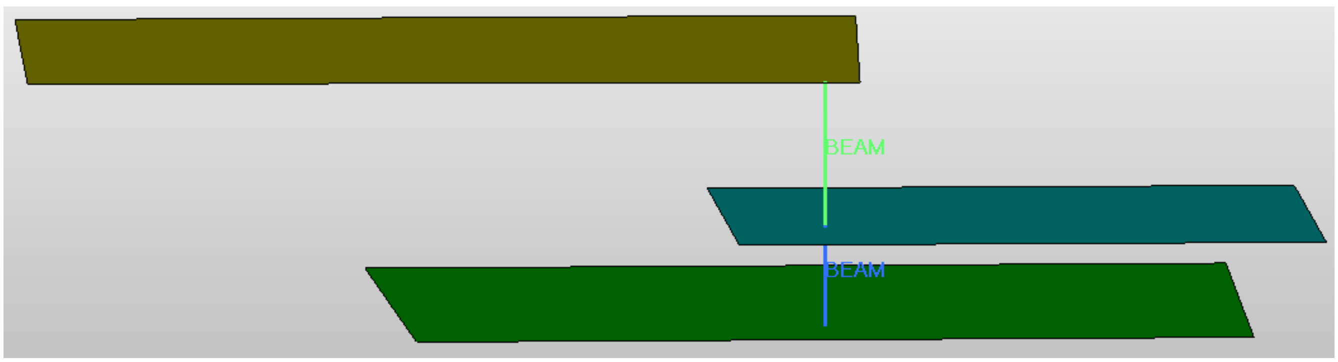


Figure 489:

> **Note:** The Find Attached (Tied) option works with displayed elements only.

BOM Comparison Tool

The BOM Comparison tool, located on the QA/Model Utility menu, reads a generic Bill of Materials (BOM) file and provides an interface to manipulate data in the BOM and its corresponding FE model.

A BOM is often used as the master document for model meshing, assembly, property assignments, model comparison, and updates between design iterations as well as other CAE activities. Since users in different design and analysis groups use BOM information, the formats and content of the BOM can vary. One BOM may contain more data than another BOM for the same program. BOMs usually use Microsoft Excel® format (CSV format) or XML format. The HyperMesh BOM Comparison tool focuses primarily on the Excel format.

The BOM reader includes the following abilities:

- Reads a generic BOM file of CSV format (comma separated values file)
- Provides a GUI to manipulate data in the BOM and the corresponding FE model
- Provides an option to update attributes in the FE model based on the data available in the BOM
- Provides an option to complete the existing BOM based on the data available from the model
- Filters out all vague information present in the BOM and provides a feature to edit the vague information into a valid data and move it back to the BOM
- Provides a functionality to export a new BOM file

Note: The BOM Comparison Tool only applies to the Nastran, LS-DYNA, Radioss and Abaqus user profiles.

BOM Comparison Tool GUI

The BOM Comparison Tool's GUI consists of seven sections, as shown below:

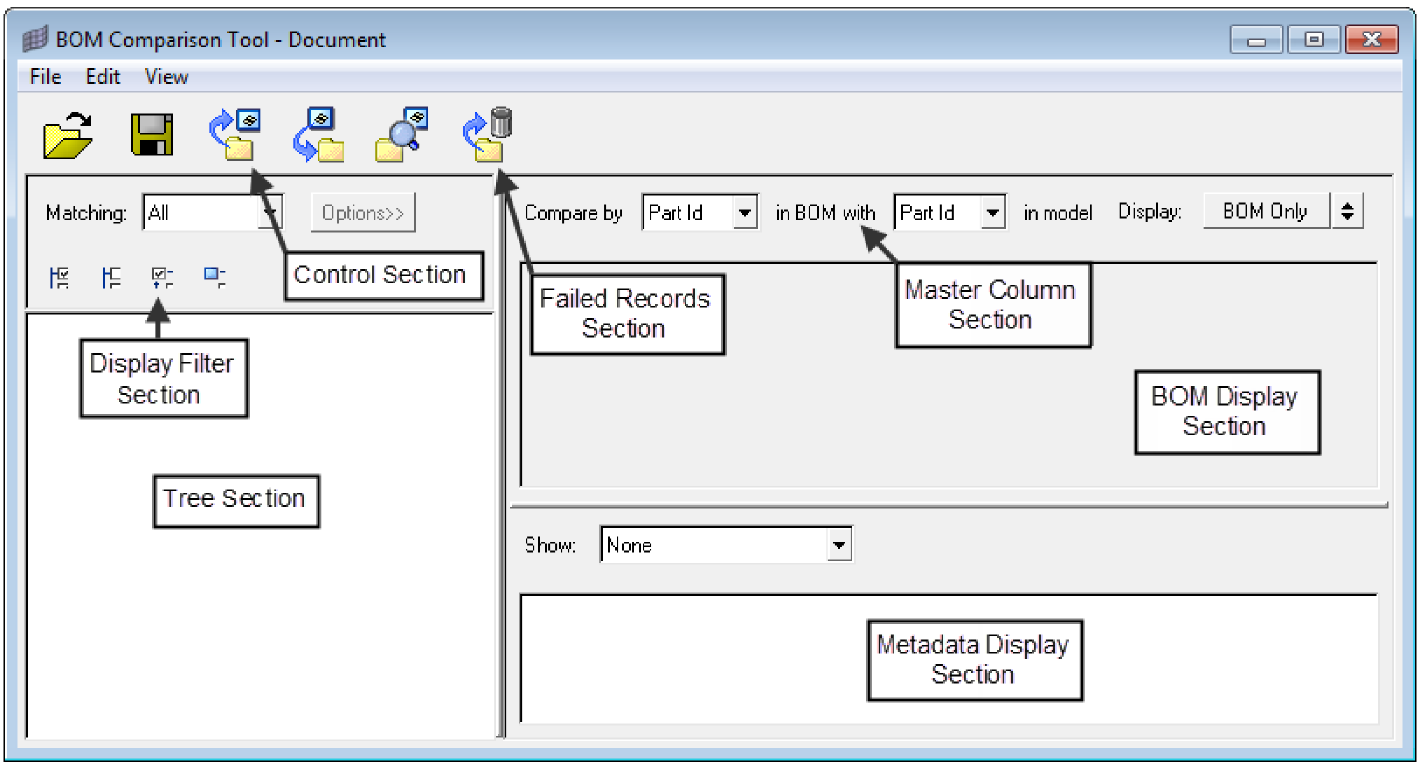


Figure 490:

Table 18:

Function	**Description**
Control section	Contains menu items and buttons to perform various operations. This section controls most tool functions.
Tree section	Contains a tree structure displaying part names and IDs.
Master column	Contains master column selection.
BOM display section	This section contains a table to display BOM info as it is seen in the actual BOM file.
Metadata display section	Contains options for metadata management.
Failed records section	Displays failed records from a loaded BOM file.
Display filter section	Contains filtering options for displaying tree and table info; part of the tree section.

BOM Comparison Tool Control Section

This portion of the interface contains drop-down menus and the toolbar.

File menu

New
Create a new session.

Open
Browse for and load a new BOM file. HyperMesh checks for the standard headers Part Name, Part ID, Material, Material ID, and Gauge.
If all are found, details populate the relevant fields in the BOM comparison tool. If any are missing, you will be prompted to select the heading from the BOM file that corresponds to each standard header.

Show Failed
Display all the invalid records that the tool encounters while reading a BOM file in a table. Only valid records from a BOM file display in the BOM Display Section's table. Invalid records can be edited to form valid data and can be moved to the BOM Display table.

Save and Export
Save and export the current information shown in the BOM Display section as a new BOM `.csv` file in a user selected location.

Exit
Close the BOM Comparison Tool.

Edit menu

Update Model
Update the model attributes to match the BOM.

Complete BOM
Sometimes the BOM doesn't contain all of the data you want. If the corresponding model contains the missing data, you can complete the BOM data by querying the database and extracting the data. Use the Complete BOM operation to either complete an existing BOM, or generate a new BOM by querying the model in the current session.
This option opens a new window listing the items to be added to the BOM file. You can select additional items from a combo box, or type a new header into it and add them, or click an item already in the list and insert the new item just above it. You may also select items in the list and delete them from the file. Once you have added or deleted all necessary entries, click **Continue** to generate the new file.

Check Model
Checks the model against the BOM. This option switches the BOM Display Section to Comparison mode if it is currently in BOM View mode (see below).

View Mode

BOM View

Display section displays BOM info as it appears in the BOM file.

Compare View

Categorizes BOM information into four sections:

- Match: components in BOM whose standard attributes match exactly with those in the model.
- Different: components in BOM whose standard attributes differ from those in the model.
- In_BOM_Only: components found in BOM but not in model.
- In_Model_Only: components found in model but not in BOM.

Same function as **File** > **Open**.

Same function as **File** > **Save and Export**.

Same function as **Edit** > **Update Model**.

Same function as **Edit** > **Complete BOM**.

Same function as **Edit** > **Check Model**.

Same function as **File** > **Show Failed**.

BOM Comparison Tool Tree Section

When a BOM file is loaded into the tool, the tool identifies the part name and part id of all valid records. It then displays the part names, appended with part IDs, in brackets in the form of a tree structure located on the left side of the tool window. Each tree branch is associated with a row in the BOM display table containing all standard information for the part in the tree branch.

This section also includes selection and filtering controls, to affect which parts display in the tree and which parts are selected or deselected. Filter options are given for displaying only the desired part info in the tree and the associated data in the BOM display table.

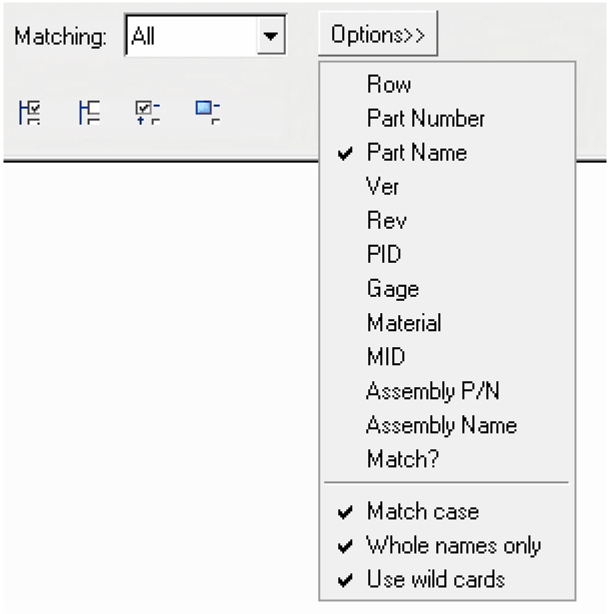


Figure 491:

You can enter a string in the combo box, select the desired header in the options menu, and press RETURN to display the desired information in the tree and BOM display table. The combo box remembers previously entered strings until you quit the tool, and can be used to filter the BOM info anytime in the session.

In addition, there are several filter buttons which are explained below:

Table 19:

Function	**Description**
(Select All)	Displays all the branches in the tree and the associated data in the BOM display table.
(Select None)	Switch off all the branches in the tree and delete all the data in the BOM display table.
(Reverse selection)	Switch on all the "off" branches in the tree and vice versa. Data associated with switched-on branches displays in the BOM display table.

Function	Description
(Show displayed)	Switch on only those branches in the tree, and associated data in the BOM display table, that correspond to the displayed parts in the model.

BOM Comparison Tool Master Column

The central top portion of the tool window contains the master column section. From this section, you can select the desired master column option.

The master column is the column in the BOM file whose attributes are considered as a key in comparison and validation operations. Only columns with three attributes can be used as master columns, such as columns containing part ID, part name and part number. The master column data is used as a key for the following operations:

- Update model attributes as in BOM
- Complete BOM by querying model
- Check model against BOM

Three master column combinations between the BOM and the model are allowed. The tool queries the data in the model based on any one of these column combinations:

Compare Part Id in BOM with Part Id in model
The tool compares the attributes of a part in the BOM with the part in the model using part ID as the key.

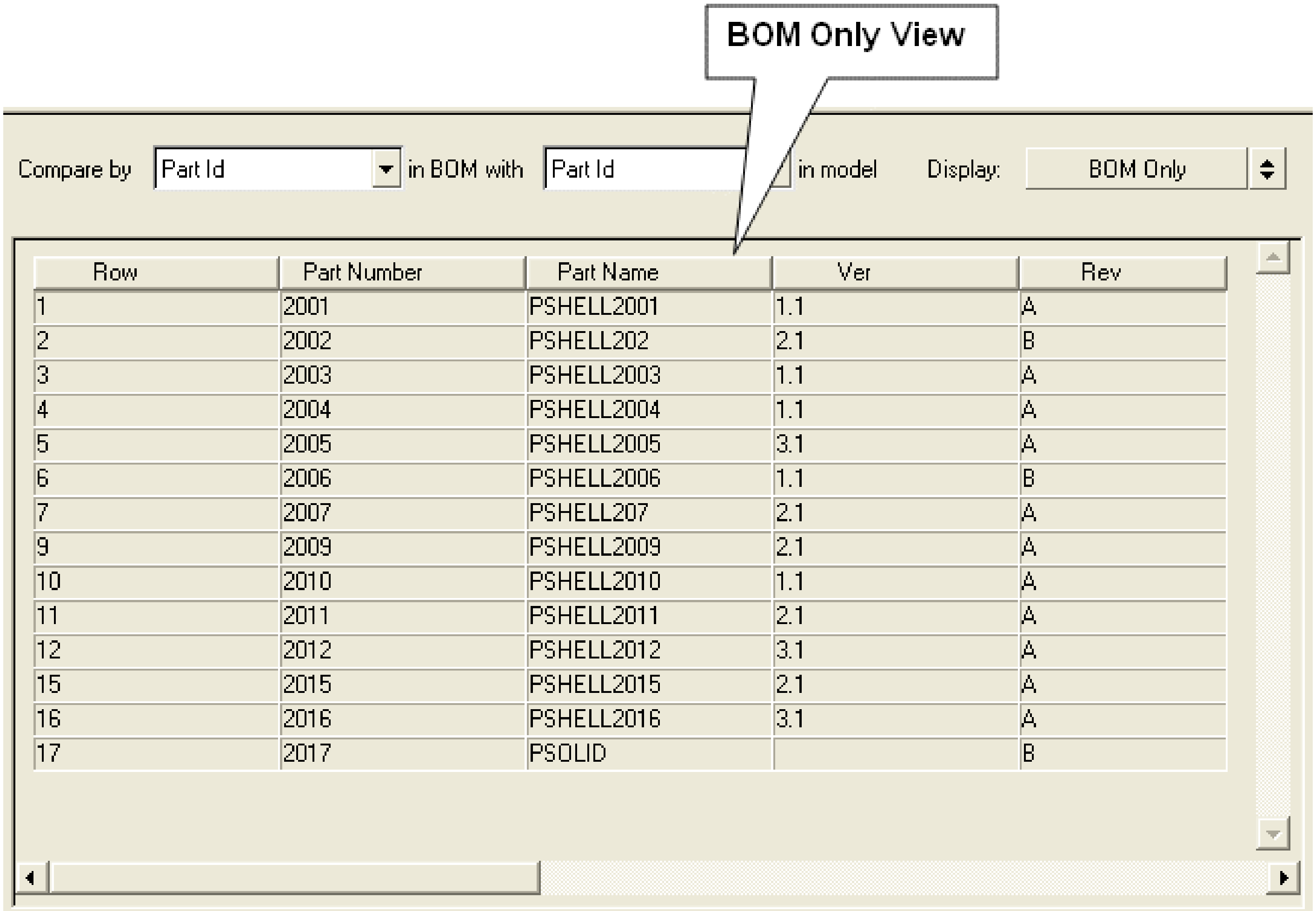
Compare by Part Name in BOM with Part Name in model
The tool compares attributes of a part in the BOM with the part in the model using part name as the key.

Compare by Part Number in BOM with Part Name in model
The tool compares attributes of a part in the BOM with the part in the model using part number as the key.

BOM Comparison Tool BOM Display Section

BOM info displays in a table in the BOM display section, located in the center of the tool window just below the master column section. BOM info can be displayed in two different modes: BOM only and Comparison.

By default information displays in BOM Only view:



Row	Part Number	Part Name	Ver	Rev
1	2001	PSHELL2001	1.1	A
2	2002	PSHELL202	2.1	B
3	2003	PSHELL2003	1.1	A
4	2004	PSHELL2004	1.1	A
5	2005	PSHELL2005	3.1	A
6	2006	PSHELL2006	1.1	B
7	2007	PSHELL207	2.1	A
9	2009	PSHELL2009	2.1	A
10	2010	PSHELL2010	1.1	A
11	2011	PSHELL2011	2.1	A
12	2012	PSHELL2012	3.1	A
15	2015	PSHELL2015	2.1	A
16	2016	PSHELL2016	3.1	A
17	2017	PSOLID		B

Figure 492:

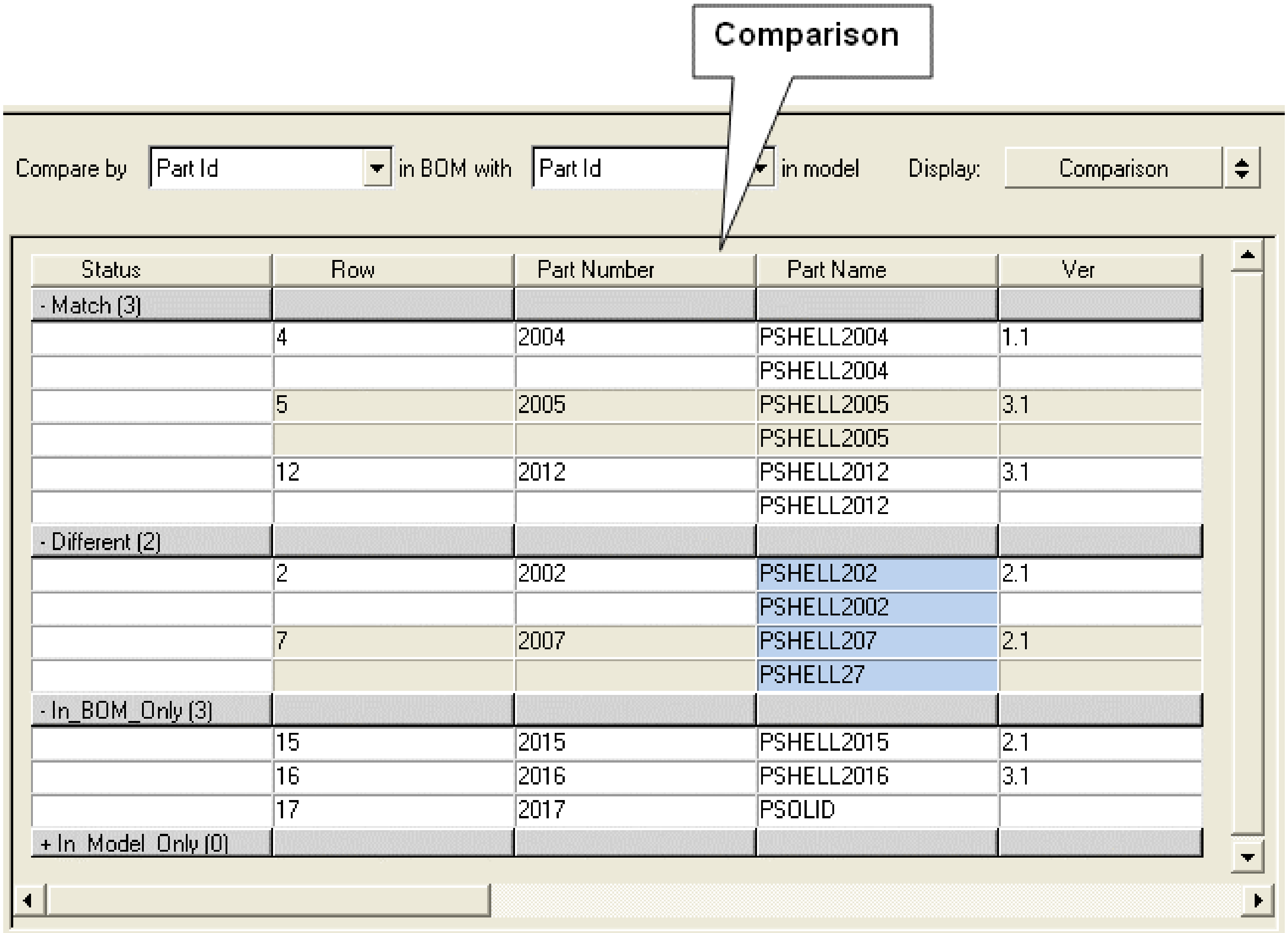
Use the toggle button located in the top-right portion of the GUI to switch to Comparison mode, which categorizes the BOM information into four categories:

Match
BOM components whose standard attributes exactly match those in the model

Different
BOM components whose standard attributes differ from those in the model

In_BOM_Only
Components found in the BOM but not in the model

In_Model_Only
Components found in the model but not in the BOM



Status	Row	Part Number	Part Name	Ver
- Match (3)				
	4	2004	PSHELL2004	1.1
			PSHELL2004	
	5	2005	PSHELL2005	3.1
			PSHELL2005	
	12	2012	PSHELL2012	3.1
			PSHELL2012	
- Different (2)				
	2	2002	PSHELL202	2.1
			PSHELL2002	
	7	2007	PSHELL207	2.1
			PSHELL27	
- In_BOM_Only (3)				
	15	2015	PSHELL2015	2.1
	16	2016	PSHELL2016	3.1
	17	2017	PSOLID	
+ In_Model_Only (0)				

Figure 493: Comparison View

Column 1 shows the category name with the number of parts falling under that category enclosed in brackets; remaining columns display the BOM info. In the Different category, mismatched attributes between BOM and the model are highlighted in light blue.

Right-click menu

Right-clicking on the table opens a menu of functions:

Display selected parts

Displays parts in the model corresponding to the selected rows in the BOM display table.

Display all parts

Display all the parts in the model.

Create metadata

Creates metadata of all the attributes of the parts in the model corresponding to the selected row in the table.

Update metadata
Updates metadata of all the attributes of the parts corresponding to the selected row in the table.

Delete metadata
Deletes metadata of all the attributes of the parts corresponding to the selected row in the table.

Delete
Deletes the selected row in the table.

BOM Comparison Tool Metadata Display Section

You can create, update and delete metadata using some of the menu items on the BOM display table. Metadata information contains all the attributes for a part in the model.

The metadata display section contains four display options in the form of a combo box. After selecting a row in the BOM Display table, use this combo box to select the type of information displayed in the metadata display table:

Table 20:

Item	Description
None	Clear the table if some data already exists.
Metadata related to BOM	Display BOM related metadata for the selected row in the BOM display table.
All metadata	Display all the metadata for the selected row in the BOM display table.
Differences between BOM/metadata	Display two rows of info in the metadata table. First row corresponds to BOM info, second row corresponds to metadata associated with the model.

BOM Comparison Tool Failed Records Section

When a BOM file is loaded, the tool checks for the validity of each standard attribute in a record. A record corresponds to one line of information in the BOM file.

The tool considers the following five terms as standard attributes:

- Part Name
- Part ID
- Material
- Material ID
- Gauge

If at least one attribute is missing or repetitive, the whole record is considered invalid and will be stored out-of-sight. Click the **Show failed** menu item or corresponding button in the control section to see the failed records.

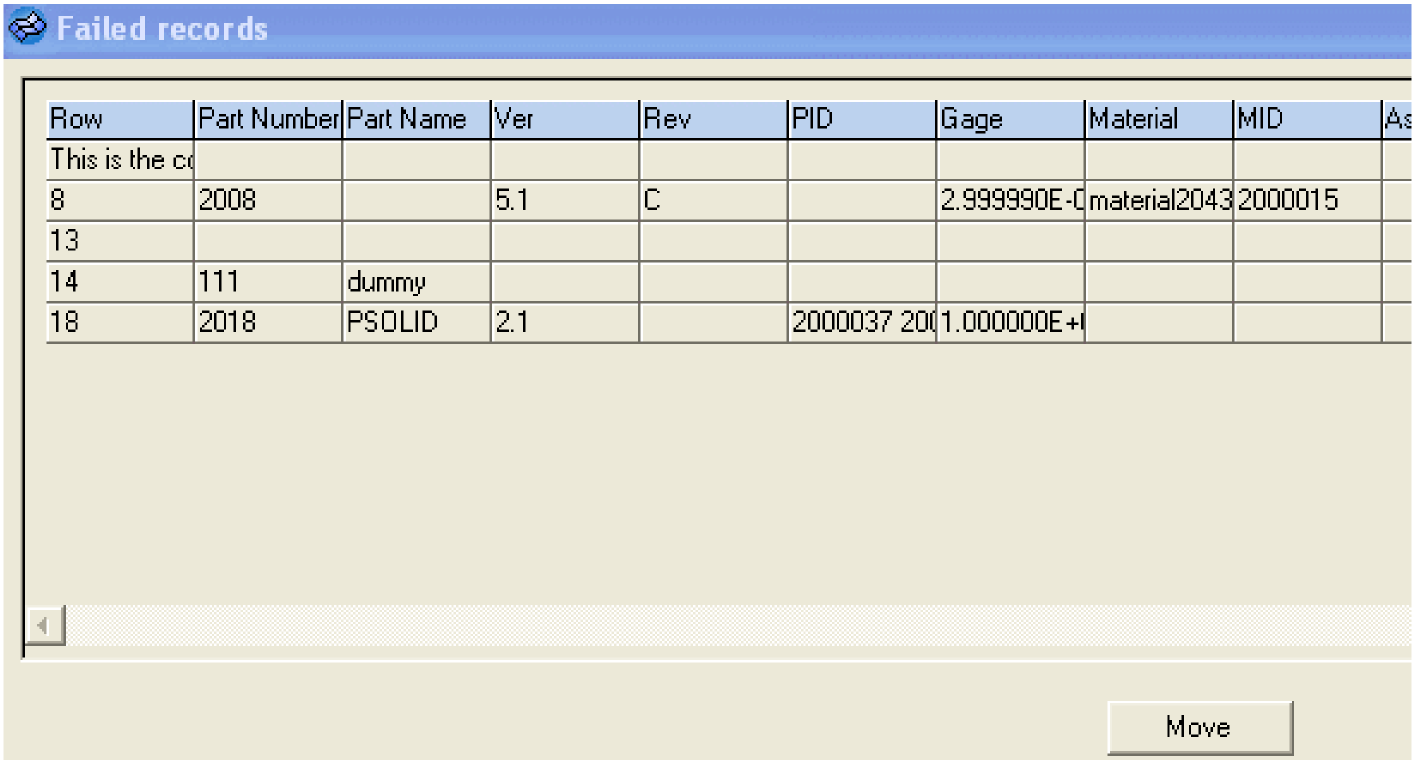


Figure 494: Failed Records

You have the option to edit each of those failed records to make them valid and move them to the BOM display table using the Move button.

Disp Utility Menu

In the Disp Utility menu, clear temporary nodes if needed.

Click **Clear Temp Nodes** to automatically remove any temporary nodes in the model.

Geom/Mesh Utility Menu

This menu contains a set of macros related to working with model geometry, as well as a set for working with FE mesh.

The geometry macros are:

Table 21:

Function	Description
Preserve edges	Prevents specific edges from being suppressed during autocleanup or batchmeshing.
Project points	Projects free points to surface edges. Depending on the tolerance you specify, points may even project to multiple edges. This can be helpful to achieve uniform meshing with regard to weld points.
Isolate Surface	Isolates either an inner or an outer surface layer, based on the user selected surface, from a 3D model. This macro works only on the surfaces attached to the selected surface. The other layers and thickness are then placed in a temp directory and masked.
Washer	Scales a copy of a selected circular line to 1.5 times its original size, and then trims this new line into the surface. This allows a higher quality mesh around circular holes.
Adj Circ Pts	Places four additional fixed points on an inner line, and then projects those points to a concentric line, creating a higher quality mesh.
Auto Connectors	A pop-up menu opens from which you can create connectors and FE realize them from a master connection file.
Quick Tetramesh	Quickly creates an automatic tetrahedral mesh while meeting the requirements for minimum element angle and element size.
Fix 2nd Order Midnodes	Improves element quality by moving the mid-edge nodes of second order elements.

The mesh macros are:

Table 22:

Function	Description
Tetra Mesh Optimization utility	Fixes slivers and wedges (tetra elements that are so thin as to be nearly planar) by moving nodes to make them more three-dimensional and improve their quality criteria.
Add Washer	Creates a layer of washer elements around a circular hole in the mesh.
Trim Hole	Creates a circular hole (of a given radius) in the mesh at the selected node (as the center of the hole). An optional layer of washer elements can be created along with a rigid spider along the hole.
Fill Hole	Fills the selected hole and remeshes the surrounding mesh to maintain connectivity. This macro does not remove any rigid spiders that fill the hole; if necessary, delete the rigid spider before using this macro.
Box Trim	Trims the model along user-defined box and generic box definition. Model cleanup follows the trimming process for LS-DYNA, Nastran and OptiStruct user profiles to remove empty and unused FE entities resulting in an error free solver deck. This feature is useful for reducing the model size to perform local system level analysis.
Bead	Creates a bead of a given height and width along the selected two nodes and connects to the surrounding mesh.

Preserve Edges

Use the Preserve Edges macro to ensure that specific component edges and feature lines do not accidentally get discarded during autocleanup or batch meshing.

Both the Batchmesher and the autocleanup features seek to improve mesh speed and/or quality by suppressing minor features, which are assumed to be insignificant. However, sometimes minor features are still important to your analysis.

1. From the Utility Browser, Geom/Mesh menu, click **Preserve Edges**.
   The **Preserve Edges** dialog opens.
2. Use the options in the macro to define the edges.

Table 23:

Option	Description
Clear at start	When this checkbox is active, any previously stored feature lines will purge each time you click **Select Edges** or **Select Comps**. Thus, picking a new set of lines starts over instead of adding to the selection.
Select Edges	Clicking this button displays a line selector in the panel area. Use the lines selector to choose the edges you wish preserved.
Show Preserved	Click this button to highlight the lines already marked for preservation.
Comps selection boundary	When active, this checkbox prevents the auto-cleanup function from equivalencing the boundaries between adjacent components.
Select Comps	Clicking this button displays a component selector in the panel area. Use the comps selector to choose the components whose boundary edges you wish preserved. **Note:** This will not preserve lines inside the components, only the outer boundary edge.
Clear All Edges	Removes all edges from the preservation list.

Option	Description
Save Preserved	Saves the preservation state, so that autocleanup and Batchmesher will know which lines must be preserved.
Reset Highlight	After clicking **Show Preserved**, use this button to remove the highlight from the preserved lines. The lines remain preserved; only the visual highlighting effect is removed (until you click **Show Preserved** again).
OK	Accepts any changes you have made and closes the pop-up window.
Cancel	Discards any changes and closes the pop-up window.

Project Points

Use this macro to project geometric points, such as weld points, to nearby edges.

1. From the Utility Browser, Geom/Mesh menu, click **Project Points**.
   A surfs selector opens in the panel area.
2. Use the selector to select surfaces whose edges you want to project points to.
3. After selecting surfaces, click **proceed**.
4. A target element size field is displayed.
5. Type a value into this field, using the same units as your model.

   Any points within this distance of the selected surfaces' edges will be projected to those edges.

Auto Connectors Macro

The **Auto Connectors** macro automates the importation and FE realization of connectors from either a Master Connectors File or an older Master Weld File.

Virtually every option available for FE realization in the connectors module is also available in the Auto Connectors macro.

Input requirements for connector entity creation and FE realization are:

- Master connector/weld file
- FE config
- Projection tolerance

> **Note:** In the case of a custom FE config, the custom FE type-to-realize is required. The custom FE type definitions can be found in the appropriate `feconfig.cfg` file. This script automatically reads the default `feconfig.cfg` file and displays a list of all the appropriate user-defined FE types (found in the `feconfig.cfg` file) in the Fe type field.

The property and diameter can be specified, if necessary.

Additional options are:

- Build systems
- Snap to node
- Attach to shells

Master Weld Files

The Master Weld File provides the weld location and parts to be connected.

A format example is shown below.

PointID	1t/2t/3t	X	Y	Z	PartID1	PartID2	PartID3
12::	2::	2.25::	2.25::	1.0::	2::	3::	
23::	3::	3.05::	3.25::	0.25::	2::	3::	5::

Diameter vs. Thickness Files

The Diameter Table associates the thickness of flanges with spot weld diameters.

The equivalent area is taken to determine the side of the hexa. The table includes flange thickness ranges and corresponding spot weld diameters.

The table below is an example of the values assigned in the Diameter Table. If the main flange thickness is between 1.02 to 1.77, the spot weld diameter will be 4.0.

Table 24:

Flange Thickness Range	Spot Weld Diameter
1.02	4.0
1.78	5.0

The Diameter Table as well as the settings that determine the main flange thickness are available in the **Diameter Mapping** dialog. You can import or manually define flange thickness ranges and spot weld diameters in the Diameter Table.

> **Note:** If you import a `.dvst` file from HyperMesh v13.0 or earlier, the data will be converted to the new file format.

You can save the information defined in the **Diameter Mapping** dialog to a `diameter mapping.txt` file, which can be reused in the future.

Example: Assign Flange Thickness Ranges and Spot Weld Diameters and Define Main Flange Thickness Settings

1. Open the Spot panel.
2. Set the diameter/use diameter mapping toggle to **use diameter mapping**.
3. Click ....
   The **Diameter Mapping** dialog opens.
4. Import or manually define flange thickness ranges and spot weld diameters in the Diameter Table.
   - To import predefined values, click **File** > **Open** from the menu bar.
   - To manually define values, click to add a row to the table. Default flange thickness ranges and spot weld diameter values will be automatically interpolated. Edit these values accordingly. To remove a range from the table, click .

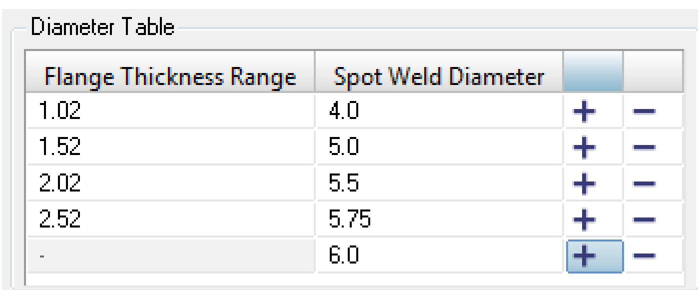


Figure 495:

5. For all flanges, define which flange thicknesses to consider when assigning diameter values.

Figure 496:

   a) From the first pull-down menu, select which flanges to consider.
      - Choose **All** to consider all flanges.
      - Choose **Inner** to consider all of the inside flanges.
      - Choose **Outer** to consider the two outside flanges.

        - Choose **Middle** to consider the middle flanges. If there is an odd number of flanges, the very middle flange is considered. If there is an even amount of flanges, the two middle flanges are considered.

    b) From the second pull-down menu, select whether to consider the 1st, 2nd, 3rd, or 4th flange the meets the criteria in the third pull-down menu.

    c) From the third pull-down menu, select whether to consider the thinnest, thickest, or average (average of all considered flanges, not only the thinnest and thickest) flange.

6. Optional: If you are connecting 2T, 3T, or 4T thicknesses, you can specify specific flange thicknesses to consider when assigning diameter values to each layer.

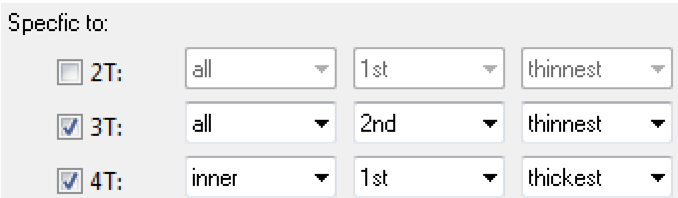


Figure 497:

7. Click **Apply**.
8. To save the diameter mapping table, as well as the settings on how to determine the main flange thickness to a `diameter mapping.txt` file, click **File** > **Save** from the menu bar.

   When the connector is realized, only the flanges that meet the criteria defined in the **Diameter Mapping** dialog will be assigned a diameter value. The flange thickness ranges and spot weld diameter values defined in the Diameter Table will determine the diameter value that is automatically assigned to the spot weld diameter upon realization.

Example

Consider there are five flanges with the following thicknesses:

Flange 1 = 3.0

Flange 2 = 1.5

Flange 3 = 1.0

Flange 4 = 2.0

Flange 5 = 2.5

In the Diameter Table below, four sets of flange thickness ranges and spot weld diameter values have been defined. Below the table, the options have been set to only consider and assign a spot weld diameter to the inner flange with the 2nd thinnest thickness. If there is two layer thicknesses, all of the flanges will be considered, but only the 1st thickest flange will be assigned a spot weld diameter. In this example, there is only one layer.

When the connector is realized, flange 2 will be assigned a spot weld diameter of 4.0.

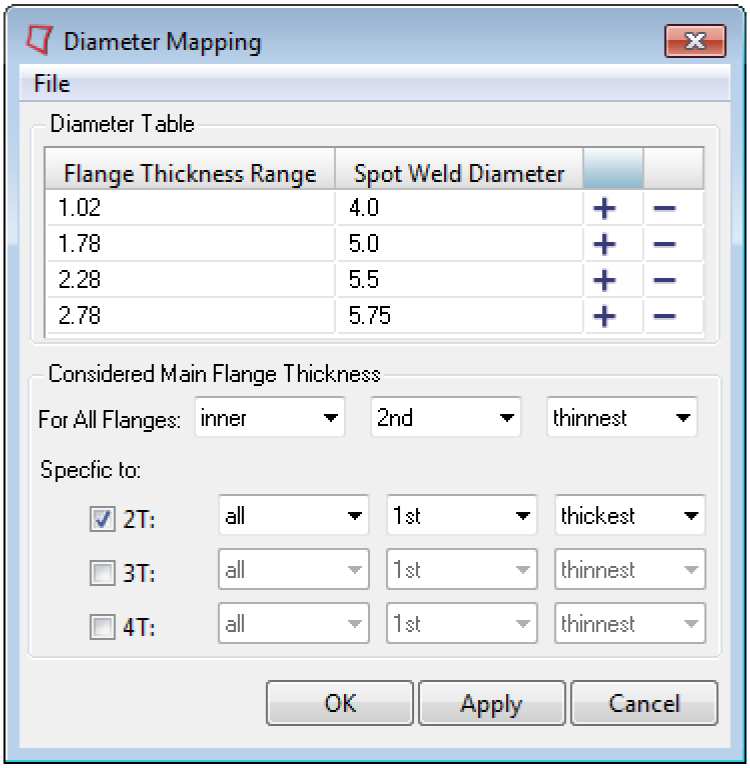


Figure 498:

ACM Welds

An ACM (Area Contact Method) weld is a special representation of a spot weld.

The weld is defined using a solid (HEXA) element whose cross-sectional area is equivalent to the area of the weld nugget. The solid element is created at the exact weld location independent of the shell elements that represent the sheet metal parts. These solid elements are connected to the corresponding components using RBE3 elements. The size of the solid element is determined using the Diameter Table or using a specific value that you specify in the Spot panel. The spot weld diameter corresponding to the thickness ranges of the connecting flanges is obtained from the Diameter Table. The size of the hexa is calculated to match the cross-sectional area of the weld nugget.

The length of the weld element is calculated using one of the following methods:

(T1 + T2)/2
: This creates the hexa elements with a length equal to the average component thickness it is connecting. T1 and T2 are the component thicknesses. The first figure below shows the ACM weld created using this method.

Project to shell
: This creates the hexa elements between the component/element shell surface. The length of the hexa element will be equal to the actual distance between the two connecting components/elements. The second figure below shows the ACM weld created using this method.

The figures below show ACM created using the two currently available methods.

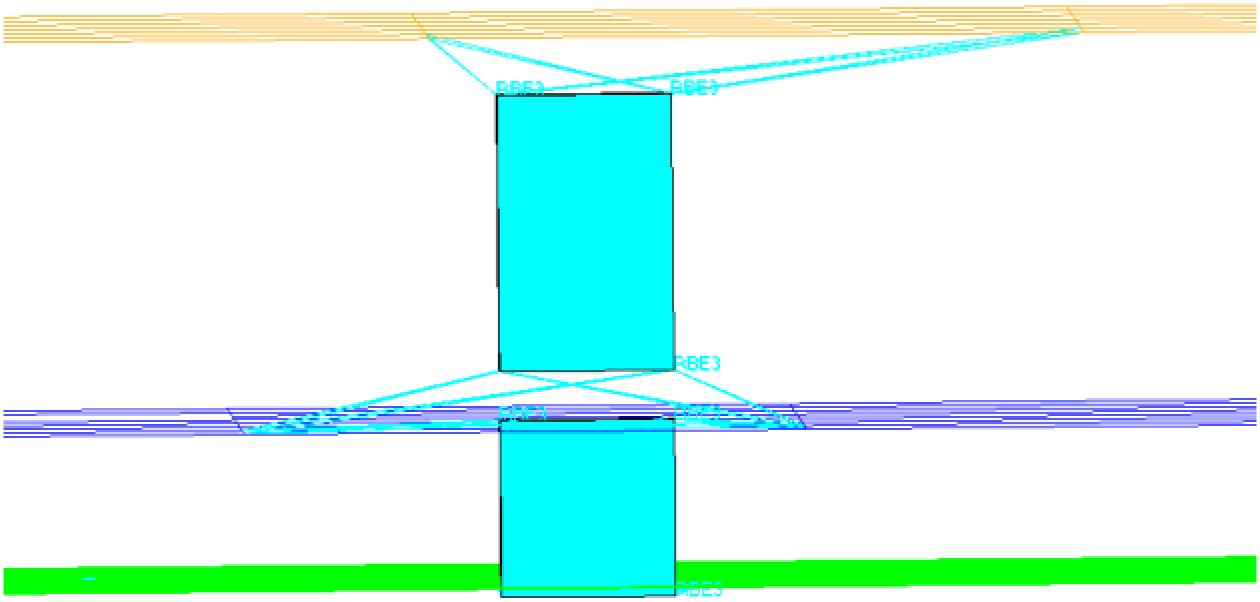


Figure 499: ACM Creation using (T1+T2)/2.0

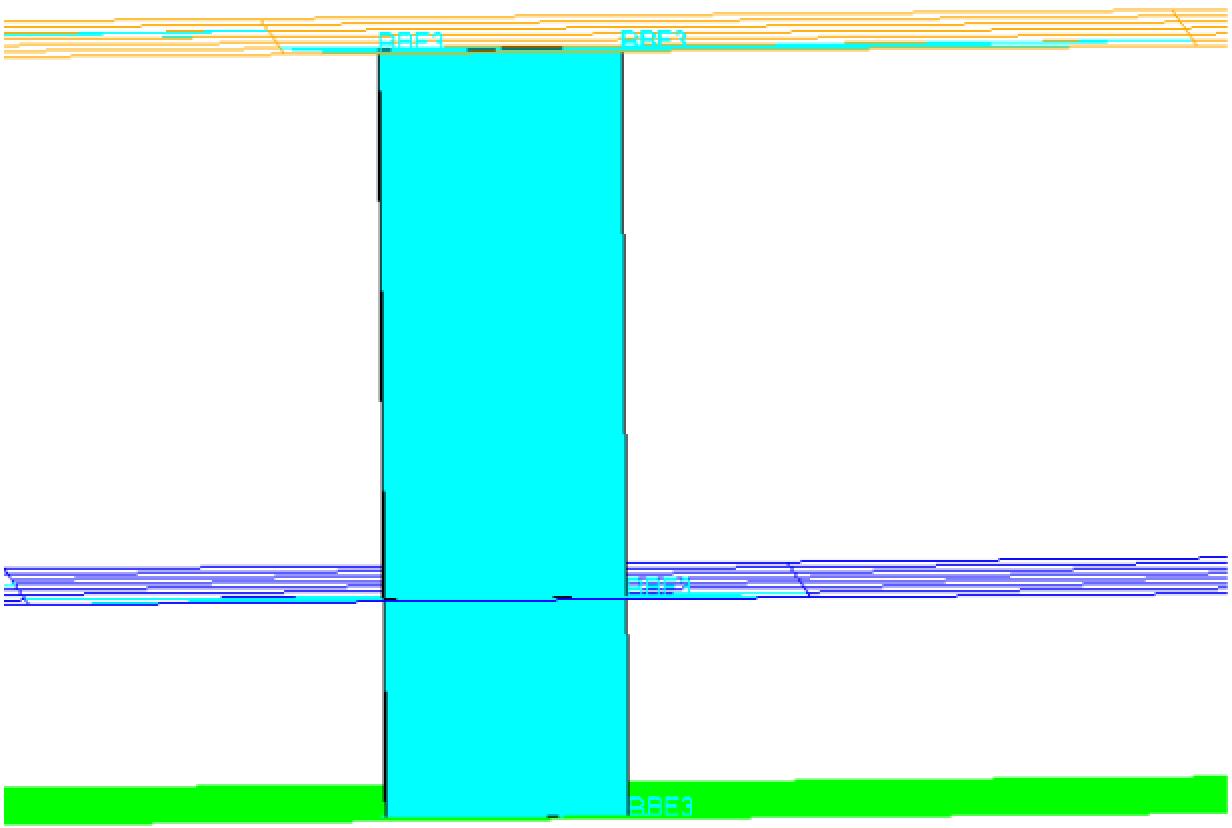


Figure 500: ACM Creation using Project to Shell

The weights of the RBE3 elements are calculated based on the projection of the dependent node on the shell element. The nodes of the shell element closest to the dependent node are assigned a greater weight relative to the node that is farther away.

Realize ACM Spotwelds

ACM welds can be created and managed using connectors. Once a connector is created, they can be realized as ACM spotwelds.

1. Make sure that the connectors are created at each of the weld locations along with connecting parts information.
2. Make sure all the connecting parts have PSHELL cards with correct thicknesses.
3. Select the connectors to be realized as ACMs in the FE Realize panel of the Connectors module.
4. Choose custom element config and select **type = Nastran 70 ACM((T1+T2)/2)** or **type = Nastran 71 ACM (Shell Gap)** per your requirements.

   The appropriate property script is automatically loaded for the selected type.
5. Set the appropriate tolerance (proj tol=) value.
6. Make sure the **attach to shell** and **snap to node** options are turned off in fe options.
7. Select a `DvsT` file, which determines the size of the hexa based on the thicknesses of the components being connected.

   If a `DvsT` file is not selected, hexas are created with weld nugget diameter =1.0.
8. Click **realize**.

CWELD Elements

CWELD elements are created as patch-patch, meshless elements.

The 1D element is not connected to the shell element. For details regarding connected shell elements or nodal information see the element card.

For CWELD elements, the diameter is determined from a `DvsT` file based on the component thickness. In addition to the creation of CWELD elements, a corresponding property card (PWELD) is created with an updated diameter ‘D’ attribute value.

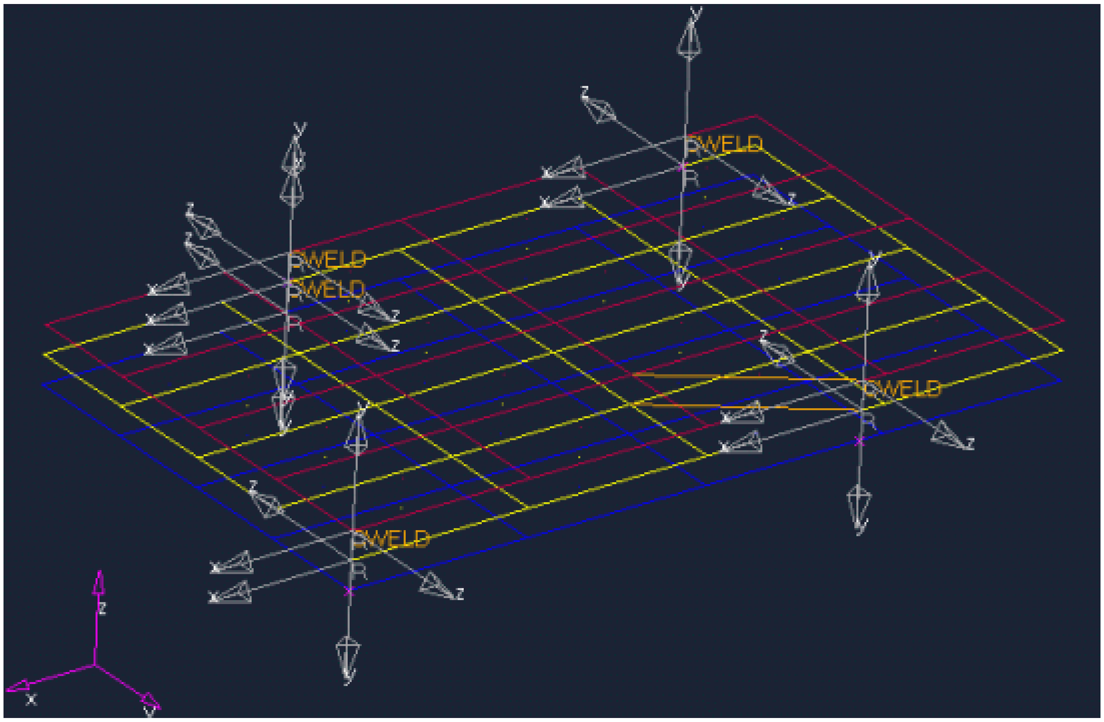


Figure 501:

Quick TetraMesh

The Quick TetraMesh macro quickly creates a tetramesh of an enclosed volume defined by geometry and/or elements.

Its main objective is to quickly and automatically create a tetramesh that meets the minimum interior angle and minimum element size. During the process of quick tetramesh, the mesh may deviate from the underlying geometry in order to maintain good quality elements. To alleviate this, you can select **Sacred elements** so that the tetmeshing function closely follows the original geometry.

1. From the Utility Browser, Geom/Mesh menu, click **Quick TetraMesh**.
   A new tab called Quick Tetra opens in the tab area.

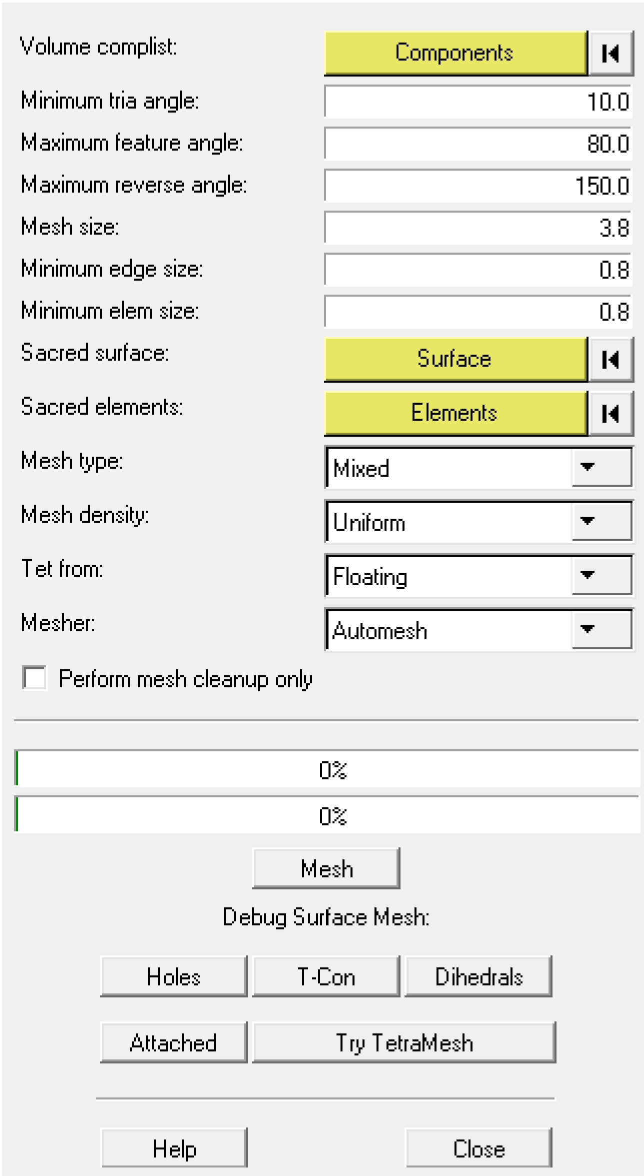


Figure 502:

2. Use the options in the macro to define the tetramesh.

Option	Description
Volume complist	Double-click **components** and use the comps collector that displays in the panel area to select comps representing the geometry of the solid to be tetra meshed. Surfaces and/or elements can be used to define the volume. Click **proceed** to finalize the selection.
Minimum tria angle	The surface trias from which the tetramesh will be extrapolated will be generated with angles that measure at least this many degrees. Use this control to limit how acute the resulting elements will be.

Option	Description
Maximum feature angle	The maximum feature angle protects nodes on corners with a feature angle greater than the value specified, helping to better maintain the geometry. This applies only to cases where you can maintain features while fixing minimum element size. For example, if two nodes of an element share different features (as in thin steps), the features may not be maintained as they do not pass minimum element criteria.
Maximum reverse angle	The maximum feature angle allowed between normals of adjacent elements. If the feature angle exceeds the given value, two adjacent elements are considered reversed and actions are performed to correct the situation.
Mesh size	Average element size of the mesh to be created.
Minimum edge size	No single edge of any generated element will be shorter than this.
Minimum elem size	Minimum allowable area for any element.
Sacred surface	When element nodes are moved to improved element quality, it gives special preference to trying to keep the nodes on a sacred surface. **Note:** This does not work if two adjacent surfaces are both marked as sacred.
Mesh type	The mesh type options are Trias Only and Mixed. With the Mixed mesh type, both trias and quads may be created.
Mesh density	Choose between chordal deviation and uniform. **Chordal deviation** Uses smaller elements along curves, feature lines, and edges to improve accuracy, but requires more computing time. **Uniform** Uses identically-sized elements throughout the mesh, but may produce low-quality elements along such locations.
Tet from	**Floating** The quick tetramesher is free to move nodes in a surface tria mesh to achieve better tetra elements based on them. **Fixed** The mesher must keep the tria mesh unchanged.
Mesher	Choose between **automesh** and **batch**. This determines the meshing engine used: the one used by the Automesh panel, or the one used by the Batchmesher. The Batchmesher generally produces better results, but does

Option	Description
	not currently support sacred surfaces or elements, ignores/replaces existing elements, and always uses uniform density.
Perform mesh cleanup only	When this option is checked, no tetra elements are created and the macro simply goes through the cleanup steps for the shell mesh. Some of the cleanup operations performed are: the suppression of free edges, correction of sliver elements, splitting of elements, and projections onto the original geometry. All the cleanup steps are designed to improve the mesh quality.
Debug Surface Mesh	A series of tools that help you locate problem areas which can cause poor meshing. **Find Holes** Locate holes in your model. **Find T-Con** Locate T-connections in the model. **Dihedrals** Locate features in the model that have feature angles greater than 150 degrees. **Attached** Locate entities attached to the selected components. **Try TetraMesh** After making adjustments, click this to re-run the meshing operation on the same components.

3. Click **Mesh**.
   The quick mesh is performed with the specified settings.

   > **Note:** There is no Undo function. You can, however, attempt to re-mesh using different settings if you do not like the initial results.

The Quick Tetramesh macro meshes the unmeshed surfaces in the model using chordal deviation and fixes all the elements that fail the criteria provided. You can manually mesh some critical geometry and select those elements as sacred elements. These sacred elements need to be trias. As a part of the cleanup, the tool heals small cracks in the model.

Use Quick TetraMesh Effectively

Suggested process for effectively using quick tetramesh.

1. Load the geometry.
2. For critical areas where you want to control the mesh such as bolt holes, manually mesh using chordal deviation.

Select these elements as sacred elements. This helps to obtain the desired mesh in critical areas.

3. Launch the Quick Tetramesh macro and run with the desired mesh size.
4. Identify problem areas, if any.

   For example, any surface edges that were ignored, or if mesh in certain areas is not satisfactory.
5. Use the Delete panel to delete the tetras, then manually mesh problem areas.
6. Re-launch the Quick Tetramesh macro and select sacred elements to protect.

Fix 2nd Order Mininodes

This macro improves element quality by moving the mid-edge nodes of second order elements.

You select the elements on which you want to improve the quality, and specify the quality constraints: Minimum Jacobian (evaluated at the corner nodes or integration points), Minimum Ratio between the minimum and maximum edge length, and Maximum angle.

> **Note:** Moved midnodes are saved to your save list; this persists until you exit the program. In addition, moved midnodes lose any pre-existent association with the underlying geometry.

Typical usage of this utility begins with use of the Check Elems panel to identify poorly-formed elements, and using that panel's **save failed** option. From that point onward, you use the Fix 2nd Order Midnodes utility.

1. From the Utility Browser, Geom/Mesh menu, click **Fix 2nd Order Mininodes**.
   An element selector and proceed button display in the panel area.
2. Click the **elems** selector and select **retrieve** to load the saved failed elements.
3. Click **proceed**.
   The **Fix 2nd Order Midnodes** window opens. This pop-up window exists independently of the rest of the environment, so you can click-and-drag it to any desired location.

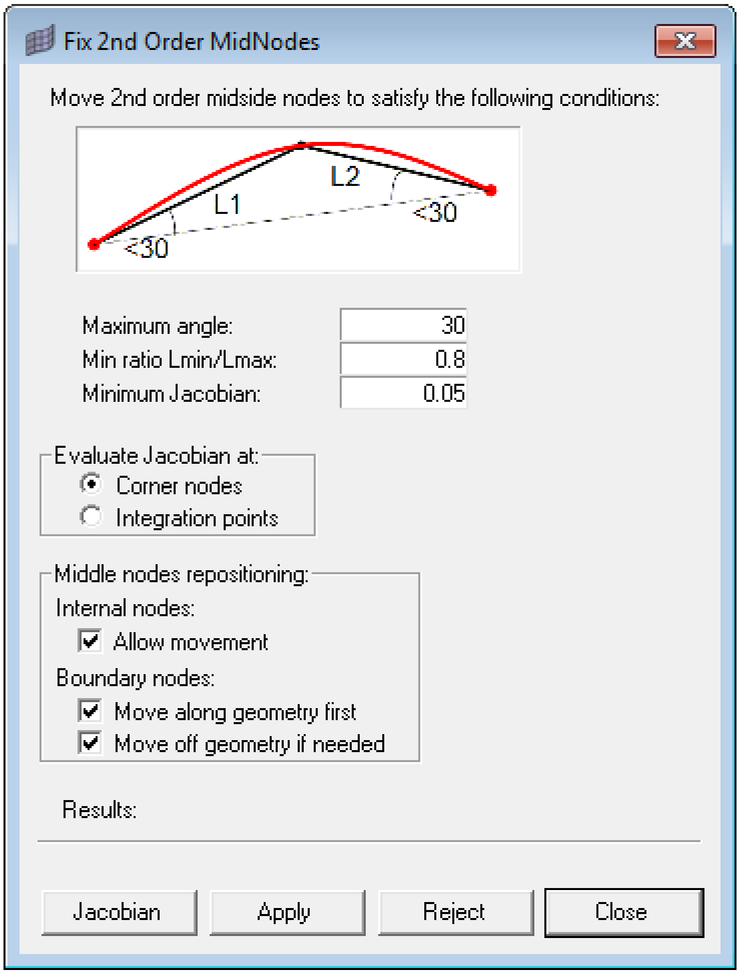


Figure 503:

4. In the **Fix 2nd Order Midnodes** dialog, choose your element quality constraints:

a) Choose a Maximum angle.

The utility will move midnodes such that the angle at the ends of each segment will not deviate from a straight line by more than this amount (thought of another way, the angle between the segments at the midnode will not exceed 180 degrees minus this value). See the figure above for an example using a value of 30 degrees.

b) Specify a limit to the Aspect Ratio (minimum versus maximum length for the segments of the midnode-bearing edges).

A value of 1 represents perfectly equal segment length, while a length of 0 would mean that the shorter segment might not exist, so this value must be greater than 0, but no greater than 1. Remember that this is a minimum ratio, so a value of 0.5 would allow the shorter segment to be half as long as the longer segment, or longer — but not shorter than half the length of the longer segment.

c) Specify a Minimum Jacobian value and use the radio buttons to determine whether HyperMesh should evaluate each element's Jacobian at the **Corner nodes** or the **Integration points**.

d) To tell HyperMesh to keep the boundary nodes on the underlying model geometry but attempt to improve the Jacobian value by moving internal nodes, select the **Allow movement** checkbox.

e) To allow nodes on geometry to move along (but not leave) the geometry features before any other node movement occurs, select the **Move along geometry first** checkbox.

f) To allow HyperMesh to move boundary nodes off of the underlying geometry if a satisfactory Jacobian value cannot be achieved by moving along geometry or moving internal nodes, select the **Move off geometry if needed** checkbox.

> **Note:** This feature is always active when **Allow movement** is unchecked.

5. Click one of the command buttons to perform an action:

- Click **Jacobian** to check the current selected elements' Jacobian values and displays them in the results area..
- Click **Apply** to tell HyperMesh to move the midnodes to try to match the criteria you specified..
- Click **Reject** to undo any changes made when you clicked **Apply**
- Click **Close** to close the **Fix 2nd Order Midnodes** dialog.

When you click **Apply**, a message displays under the Results heading to inform you of exactly what HyperMesh did to the mesh. The images below illustrate the before-and-after state of a specific midnode and the criteria used, as well as the overall results:

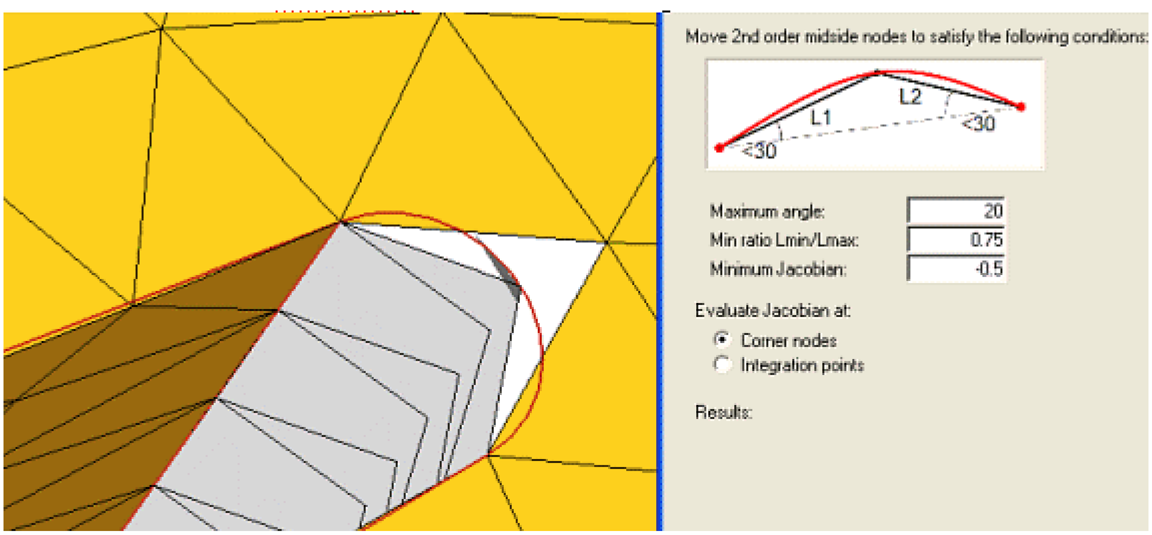


Figure 504: Before Clicking Apply

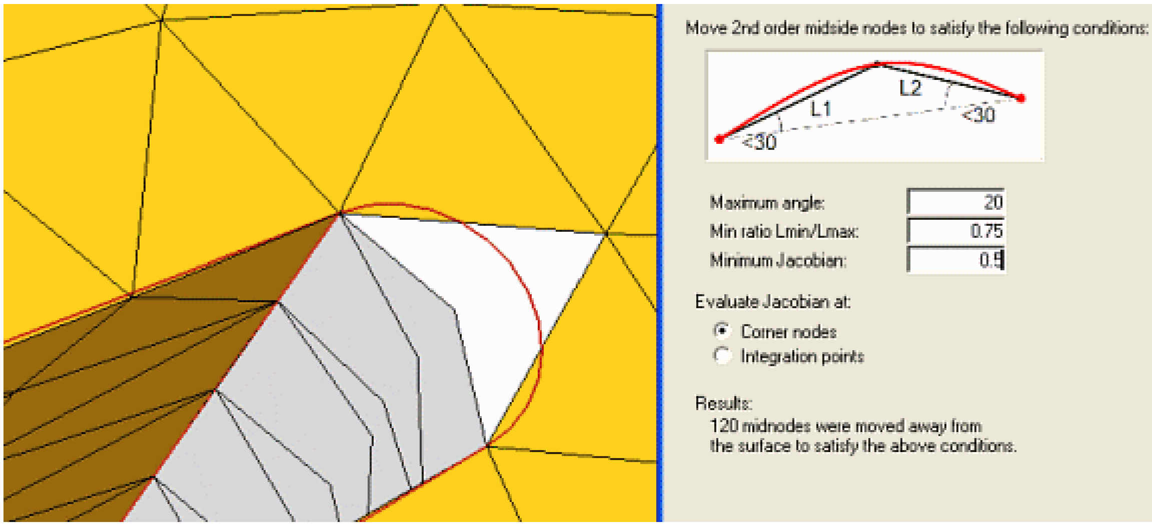


Figure 505: After Clicking Apply

Add Washers

Use the Add Washer utility to create one or more layers of washer elements around a circular hole in an existing mesh.

1. From the Utility Browser, Geom/Mesh menu, click **Add Washer**.
   A temporary panel displays in the panel area.
2. Select a single node from the edge of a hole.

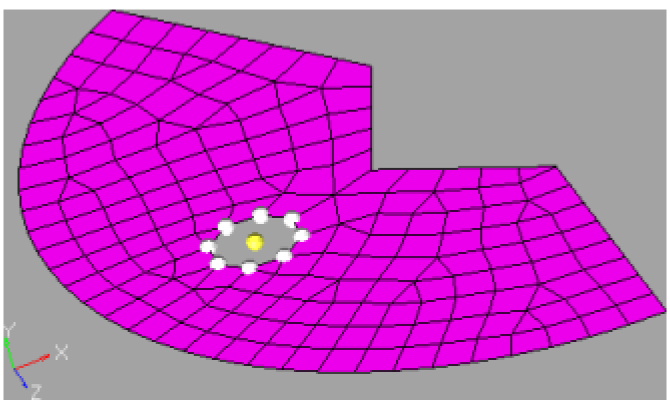


Figure 506:

3. Click **proceed**.
   All of the nodes on the hole are selected and the utility opens.

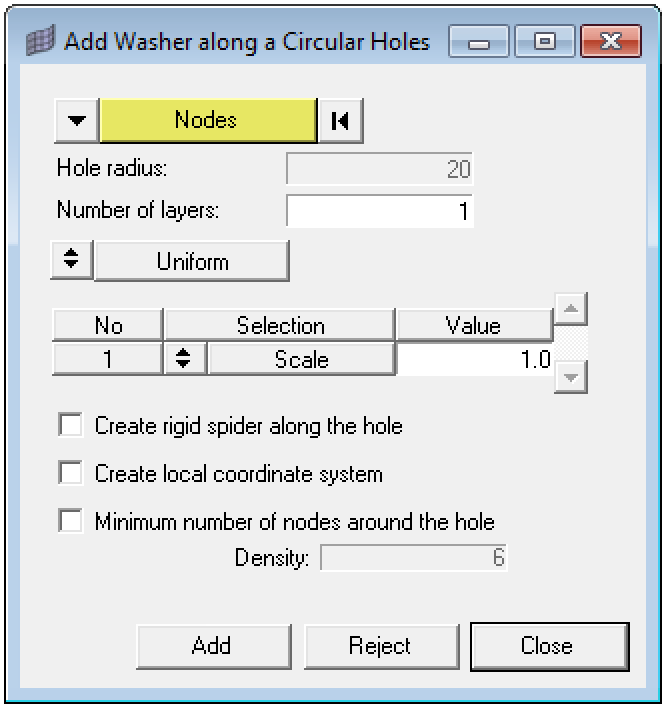


Figure 507:

4. The utility automatically determines the Hole radius. You can specify the Number of layers of concentric washer elements to add around the hole.
    - If you choose to add more than one layer, you can also choose whether or not to have all layers be **Uniform** in width, or to allow them to have **Varying** widths from one another.
    - If you choose **Varying**, each layer displays separately in the table below this option, allowing you to specify a different value for each layer.
5. Specify a **Width** (the size of the elements) or a **Scale** (a factor of the hole's radius – for example, using a scale of 1.0 produces washer elements whose size is the same as the hole's radius) for each layer of elements.

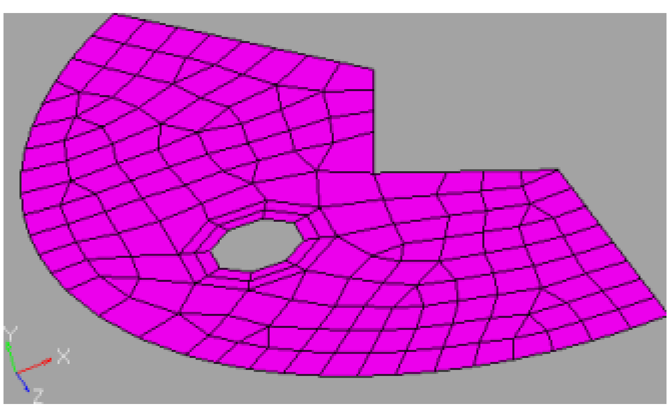


Figure 508:

Mesh size 5, 2 washer layers of width 2.

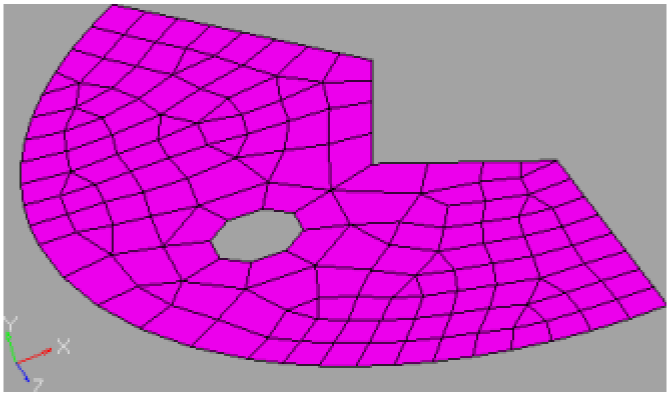
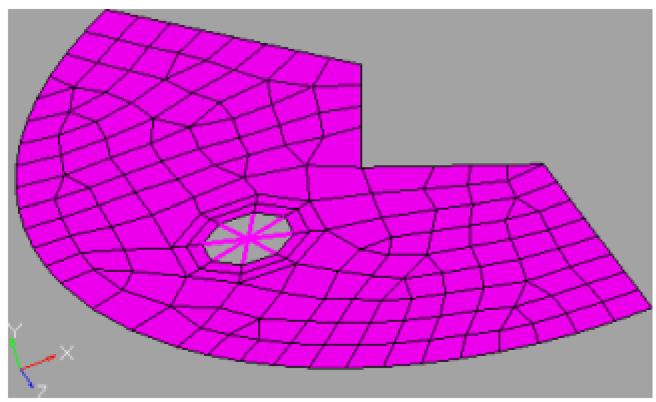
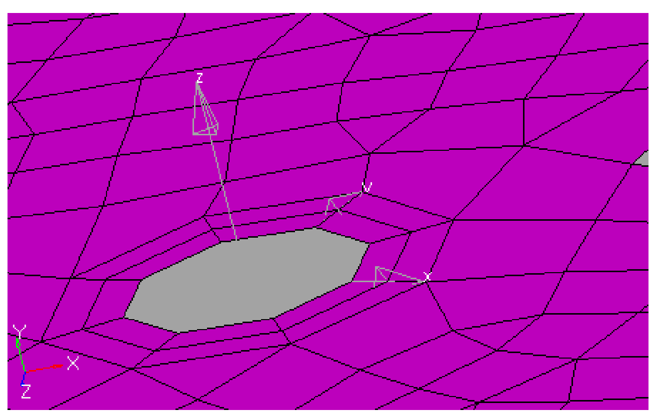


Figure 509:

Mesh size 5, 1 washer layer of scale 1.0.

6. Activate the checkboxes of any desired creation options.

Option	Description
Create rigid spider along hole	 *Figure 510:*
Create local coordinate system	 *Figure 511:*

Minimum number of nodes around hole	Prevents the washer from using fewer nodes than this around the hole, in order to maintain a desired level of granularity. **Note:** A larger number than this may be generated in order to generate a uniform mesh of washer elements, particularly when using smaller numbers for the minimum. When active, this also enables the Density numeric box, which lets you specify the exact minimum number.

7. Click **Add** to create the washer layers. If the results are not acceptable, click **Reject** and alter your settings.

Trim Holes

Create a circular hole of a given radius in the mesh at a node specifying the center of the hole, as well as specify a number of layers of washer elements to include.

1. From the Utility Browser, Geom/Mesh menu, click **Trim Hole**.
   A node selector panel opens.
2. Pick nodes on your model for the centers of each hole that you wish to create, then click **proceed**.
   The **Mesh Trimming with Circular Holes** dialog opens.

3. Edit the options in the dialog to determine the type of hole that is created at each chosen node.

Table 25:

Option	Description
Hole Radius	Each node will receive a hole of this radius, measured from the node.
Number of layers	This is the number of layers of washer mesh elements that you want to surround each hole.
Uniform/Varying	This toggle only applies when the number of layers is more than zero, and specifies whether you want mesh layers to all be the same width, or to vary from one another.
No.	The number of a specific washer layer. If you chose **varying width** for the layers, the table displays one row for each of the number of layers that you specified. Otherwise, only one row displays because all layers will be set to the same values.
Scale/Width	Determines the width of the washer layers. **Scale** You can specify each layer's scale relative to the Hole radius. For example, use "0.5" for a washer layer that's half as wide as the hole radius. **Width** Specify a fixed width for each layer.
Value	The scale factor or width of the layer(s).
Create rigid spider along the hole	This checkbox will create a rigid spider in each of the new holes created, and enables two more options: **individual rigid links** Create rigid elements at each node of the new hole. **single rigid link** Create one rigid element that connects to all of the nodes around the new hole.
Minimum number of nodes around the hole	This determines the mesh density around the new hole(s). Each new hole will be created with at least the number of nodes that you specify in the density field, evenly spaced around its circumference.
Reject	If you do not like the results of the last trim operation, click this button to undo it.

Option	Description
	Note: This only undoes a single click of the Trim button, so it can only undo multiple holes if they were created simultaneously during a single trim operation.

4. Click **Trim** to create the new hole(s).

Fill Holes

The Fill Hole function fills in one or more holes in your geometry with automatically-generated mesh.

> **Note:** This macro does not remove any rigid spiders that currently fill the hole; if necessary, delete the rigid spider before using this macro.

1. From the Utility Browser, Geom/Mesh menu, click **Fill Hole**.
   The **Filling holes with mesh** dialog opens.
2. Choose a method for filling holes.

Option	Description
Manual	Use this option to select the holes that you wish to fill: 1. Click **Select Nodes**. The panel area is once again displayed with a nodes selector active. 2. Select nodes on the edges of the holes that you wish to fill. 3. Click **proceed** in the panel area. The **Filling holes with mesh** dialog returns. The Select Nodes button is now green to indicate that nodes have been chosen. 4. Click **Fill** to fill the selected holes with mesh.
Automatic	Use this option to select holes automatically based on size. Type a value into the entry field labeled Fill circular holes with radius smaller than. The model is automatically scanned for holes smaller than this value, and attempt to fill them with mesh.

3. Click **Fill**.
   If you do not like the results of the last fill operation, click **Reject** to undo it.

   > **Note:** This only undoes a single click of the fill button, so it can only undo multiple fills if they were created simultaneously during a single fill operation.

Box Trim Macro

Use the Box Trim utility to trim the model, or selected subset, along the global axis to fit the selected 3D box.

For example, a full car model can be trimmed along the Y=0 axis to obtain the left or right side of the car. A fixed boundary condition is applied at the trimmed edges. It also removes the FE entities outside of the box, and cleans the model so that it is a runnable deck. The axis directions and terminology are based on modeling standards in the automotive industry.

This option is available in the LS-DYNA, Nastran, Abaqus, OptiStruct and Radioss user profiles.

Different scenarios for using the Box Trim utility include:

- This tool is useful in applications where some types of analyses can be performed on one-half (or quarter) of the model using symmetry boundary conditions.
- Trimming a full car model into a quarter or half model for further analysis.
- Trimming a model to significantly gain computational time.
- Trimming a model to debug an analysis error in a particular location in order to understand the cause of the error.

The Box Trim utility is very useful when you are doing a system level analysis, and when portions of a full vehicle model include few complete subsystems or partial subsystems.

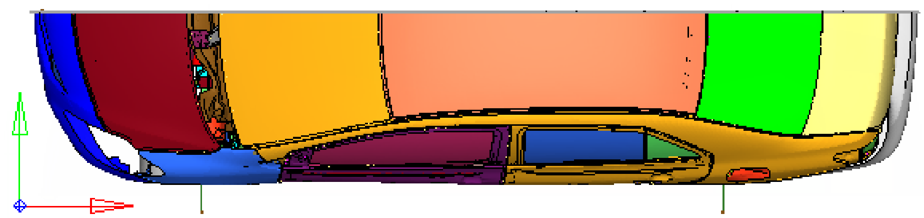
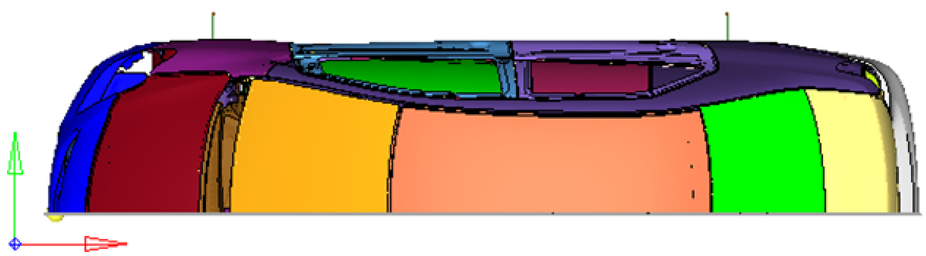
> **Note:** This macro can only be used for 1st order elements.

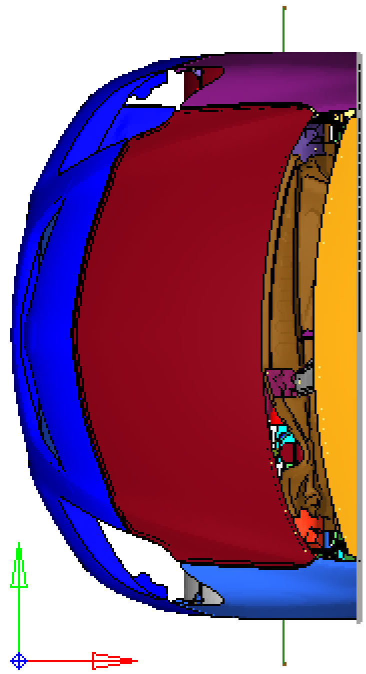
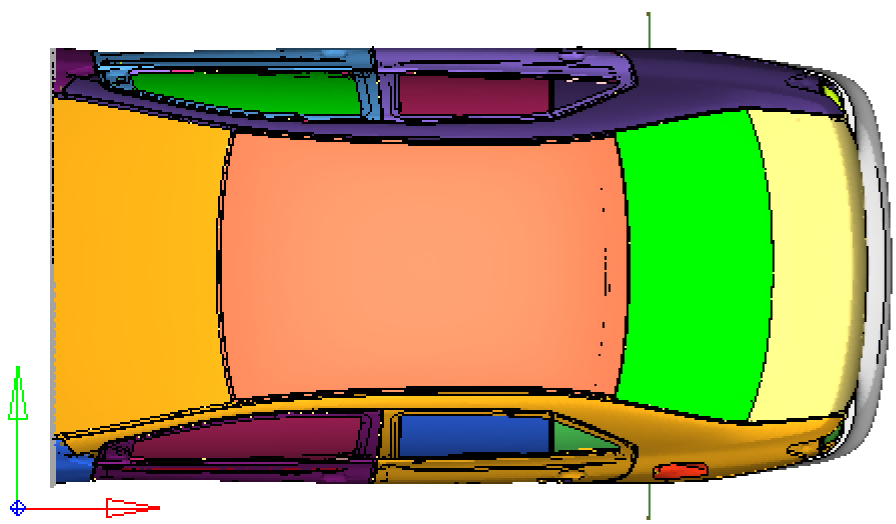
Trim Boxes

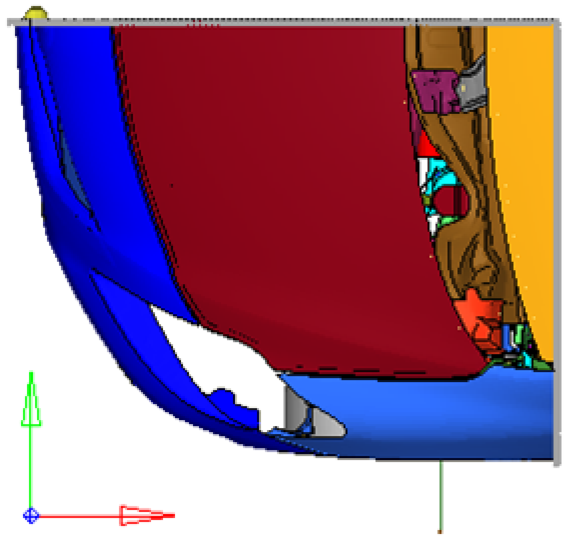
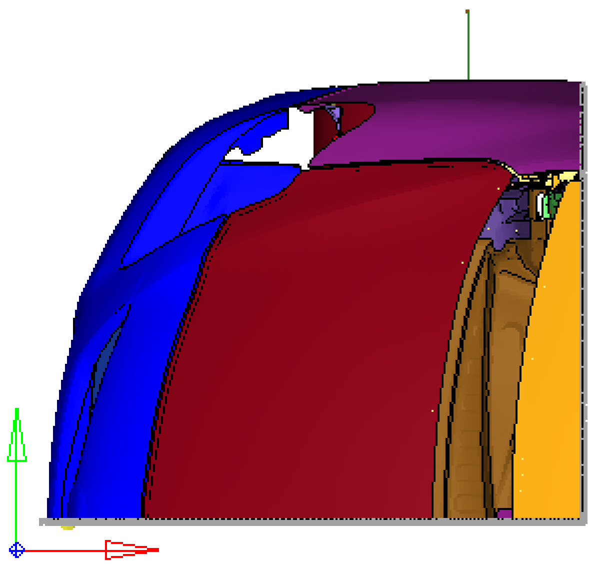
1. From the Utility Browser, Geom/Mesh menu, click **Box Trim**.
   A warning message appears if you in the LS-DYNA or Nastran user profiles. If you are not in either of these profiles, then the **Box Trim** dialog appears.
2. Click **Yes**.
   You are directed to the panel area.
3. Using the elems selector, select the elements that you want to trim.
4. When you are finished, click **proceed**.
   The **Box Trim** dialog opens.

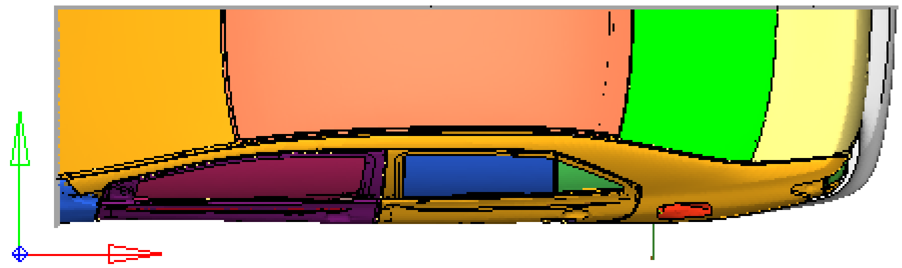
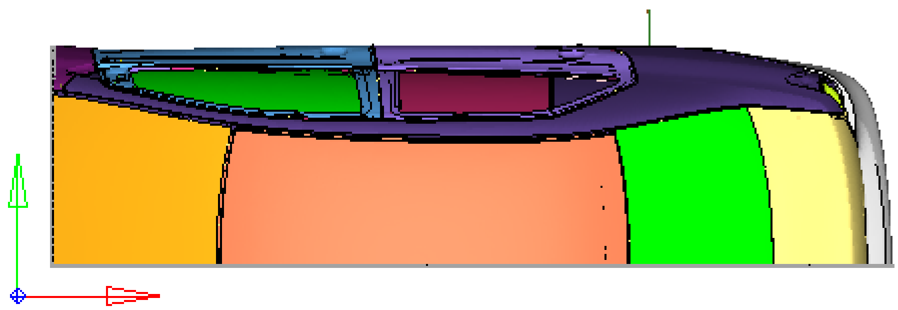
5. Edit the options in the dialog.

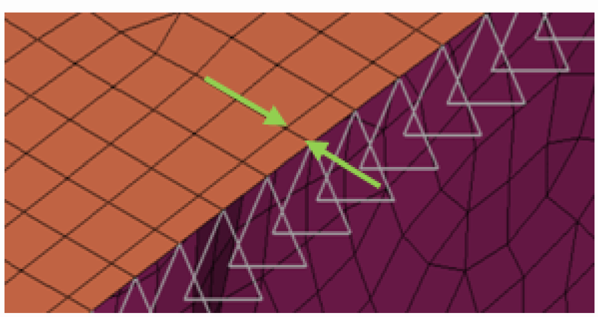
Table 26:

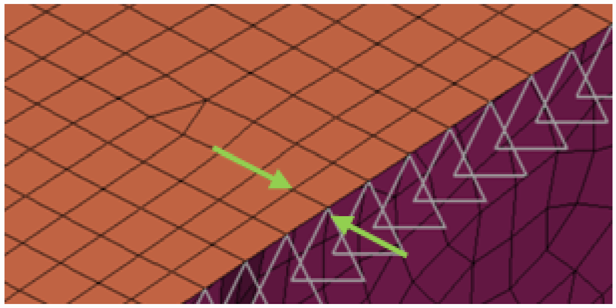
Option	Description
Box trim type	Determine how the selected model will be trimmed by selecting one of the following options from the pull-down menu: **left** Split the model along global Y=ymiddle and save the model between Y=ymin and Y= ymiddle (ymiddle =(ymin+ymax)/2).  *Figure 512:* **right** Split the model along global Y=ymiddle and save the model between Y= ymiddle and Y=ymax.  *Figure 513:* **front** Split the model along global X=value (selected value) and save the model between X=xmin and X=value.

Option	Description
	 *Figure 514:* **rear** Split the model along global X=value (selected value) and save the model between X=value and X=xmax.  *Figure 515:* **frontleft** Split the model along global Y=ymiddle and X=value (selected value) and save the model between Y=ymin and Y=ymiddle, and X=xmin and X=value.

Option	Description
	 *Figure 516:* **frontright** Split the model along global Y=0.0 and X=value (selected value) and save the model between Y=0.0 and Y=ymax, and X=xmin and X=value.  *Figure 517:* **rearleft** Split the model along global Y=0.0 and X=value (selected value), and save the model between Y=ymin and Y=0.0, and X=value and X=xmax.

Option	Description
	 *Figure 518:* **rearright** Split the model along global Y=0.0 and X=value (selected value) and save the model between Y=0.0 and Y=ymax, and X=value and X=xmax.  *Figure 519:* **custom** Define the box by either selecting two corner nodes (Corners) or selecting the center node and dimensions (Distance from center).
X limit	Define the trim location by selecting one of the following options: **By node** Directs you to the Node Selection panel where you can select the node for the box trim location. **By value** Enter the value for the box trim location.
Defined by	Define the trim location by selecting one of the following options: **Pick Corner Node** Directs you to the Node Selection panel where you can select the two corner nodes that define the outer X, Y and Z bounds of the box. **By Center Node** Directs you to the Node Selection panel where you can select the center node, and then enter Delta X, Delta Y and Delta Z values

Option	Description
	which is the distance from the center node to the outer bounds of the box in global X, Y and Z directions. **Note:** This option is only available when you select custom as the Box trim type.
Create constraints	Create Single Point Constraints (SPCs) for all of the nodes along the trimmed edges. When this option is activated, the following options are enabled: **SPC collector** Specify a load collector to store the Single point constraint (SPC) created along the trimmed edge. For OptiStruct, Nastran, and Abaqus, all of the nodes along the trimming plane will be created as a node set. **Box collector** Specify a box collector. A large hexa element that represents the box will be created for visualization in the specified collector. Only available in the Radioss user profile.
Remesh options	Enables access to additional options that help clean the trimmed edge elements in order to achieve a better mesh quality.
Min size	Defines the minimum size of remeshed elements on the trimmed edge. Element size can be defined by selecting one of the following options from the pull-down menu and entering a corresponding value in the elem size field. **relative** Multiplier of the local element size **absolute** Actual element size  *Figure 520:* *Min size > 3 (absolute)*

Option	Description
	 *Figure 521:* *Min size > 5 (absolute)*
Delete comps with	Enables a filter to delete components after the trim. The components that are deleted after the trim can be defined by selecting one of the following options from the pull-down menu and entering a corresponding value. **num elems** If the number of elements remaining after the box trim are lesser than specified value, then entire component will be deleted. **% area** Specify the percentage of the total area remaining after the trim. If the area after the trim is lesser than the specified value, then the component will be deleted.
Feature angle=	This option is similar to the global feature angle specified in the mesh option. This value is used in element cleanup after the trim.

6. Click **Trim**.

> **Tip:** Click **Preview** to view the trim before confirming your selections.

How Entities on a Cutting Plane are Treated

The entities on a cutting plane must be treated as follows:

0D Elements
Delete 0D elements that are defined on the nodes that were deleted.
There are some special cases where the entity set and part are referred in it as a pointer. In these cases alone retain them.

1D Elements

Beams, Bars, Springs
Remove the element.

FE Joints
Remove the element.

RBE3
If the dependent node is deleted, remove the element.
If any independent nodes lie on the plane of the box trim, remove the element.

Rigids
If the independent node is deleted, remove the rigids.
If any dependent nodes lie on the trimming line edge, remove the rigids.
In all other cases, retain the rigid.

2D Elements
Trim the element on the cutting plane.

3D Elements
Remove the element on the cutting plane.

Connectors
Remove the connector and it's corresponding FE realized.

Equations
This is mapped to loads in HyperMesh. Delete the equations if one of the nodes are deleted as part of Box trim.

How the Box Trim Macro Executes a Model Cleanup

The model cleanup includes:

1. Empty deletion
   - Deletes empty components (components with no elements), assemblies, sets, contact surfaces, output blocks, groups, plies, plots, and load steps. The following entities will also be deleted if there is no card image: load collectors, system collectors, vector collectors, and section collectors.
   - Deletes unused materials, properties, sets, beam sections, curves, systems, vectors, laminates, plies, tables, tags, and optimization entities.
2. Location based deletion
   - Deletes connectors with a FE realization that is located outside of the box and at the box boundary.
   - Deletes all of the morphing entities and blocks that are completely outside of the defined box.
3. Solver specific
   - *CONTACT – mapped Group

        - Checks if the master or slave surface is selected in the definition. If either of them are empty, the *CONTACT will be deleted.
    - *CONSTRAINED_EXTRA_NODE - Mapped to ConstrainedExtraNode
        - Checks if the nodes or node sets are defined as the slave entity. If there are not any nodes or node sets associated, the entity will be deleted.
        - Checks the card image to see if the PSID assigned is blank. If it is blank, the keyword itself will be removed.
    - *CONSTRAINED_RIGD_BODIES, *CONSTRAINED_TIE-BREAK, *CONSTRAINED_LAGRANGE_IN_SOLID, *CONSTRAINED_EULER_IN_EULER, and *ALE_options – Mapped to Group
        - Checks if the slave or master definition is defined in the Group. If either the slave or the master definition are empty, then the entity will be deleted.
    - *AIRBAG_options – Mapped to control volume
        - Checks if the entity has a slave definition. If the definition is empty, then the entity will be deleted.

          > **Note:** This rule does not apply to *AIRBAG_PARTICLE, *AIRBAG_INTERACTION and *AIRBAG_ADVANCED_ALE.

    - *AIRBAG_PARTICLE
        - Checks if the card image is referenced by part set ID or part ID, corresponding to the variables SID1 or SID2. If the card image has no reference, then the entity will be deleted.
    - *AIRBAG_INTERACTION
        - Checks if the variable in the card images AB1 and AB2 are assigned a value. If they are not assigned a value, then the entity will be deleted.
    - *DEFINE_ALEBAG_BAG and *DEFINE_ALEBAG_HOLE
        - Checks these keywords to see if SID_PART or SID_SET are empty. If either of them are empty, then the keyword will be deleted.
    - *AIRBAG_ADVANCED_ALE
        - Checks if all of the variables in Bag_ID 1…. and Bag_ID 8 are empty. If they are empty, then the entity will be deleted.

4. LS-DYNA specific cleanup
    - Moves all of the leftover control cards to the box trim include, as they are valid definitions.
    - If the following keywords are defined in the model, entire model, and do not require an explicit definition of entities, then they will need to be gathered and moved to the box_trim_include.
        - *CONTACT (If the master or slave type is assigned a value of 5, corresponding to all.)
        - *LOAD_BLAST
        - *LOAD_BODY
        - *LOAD_BRODE

- *INITIAL_VELOCITY_options expect Initial_Velocity_Rigidbody. In cases of Initial_Velocity_rigidbody, if the PSID is 0, then it will be ignored. This implies that it is not applicable on any entity, therefore it should not be included back.
- *RIGIDWALL_ - If the variable NSID is assigned a value of 0, this indicates all of the nodes in the model.

- Deletes any remaining entities, except for the entities in the box_trim_include.
- Moves the entities from the box_trim_include to the master file.

5. Abaqus, Nastran, and OptiStruct specific enhancements and cleanup

- Creates a node set from the nodes on the trimming plane while trimming.
- Updates the following contact and contact surface entities:
  - Abaqus: *SURFACE ELEMENT, *SURFACE NODE, *CONTACT PAIR, and *TIE
  - Nastran: BSURF, BCBODY, and BCTABLE
  - OptiStruct: SURF, CONTACT, and TIE
- Limitation: Abaqus, Nastran, and OptiStruct support direct property assignment, therefore when trimming such models you will be required to assign a property to the elements remeshed at the trimming plane.

Use the Bead Utility

Use this utility to add a bead between two points in a mesh.

> **Note:** If you need to make a curved bead, or a bead across jointed or highly-curved components, this is best accomplished with the sculpting tools in HyperMorph's Freehand panel.
>
> However, the bead utility presents a quick and easy way to create simple linear beads, such as those used to initiate crumple zones in vehicular crash mitigation.

Beads can be of any height or radius, and can be sharp (curved or angled along the top) or flat (raised from the surface, but flat along the top). However, this distinction will only be apparent if the radius and height are relatively close to the existing element size.

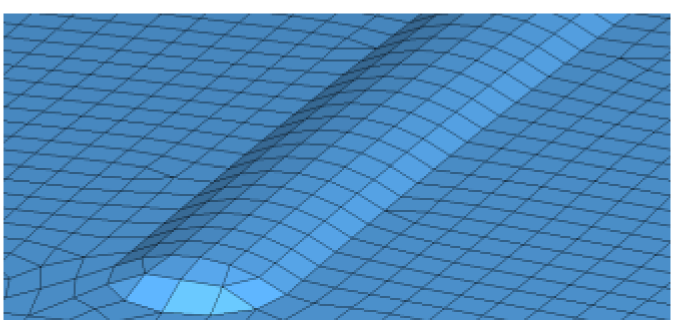


Figure 522:

Radius 20, height 5, either sharp or flat, with mesh size 8

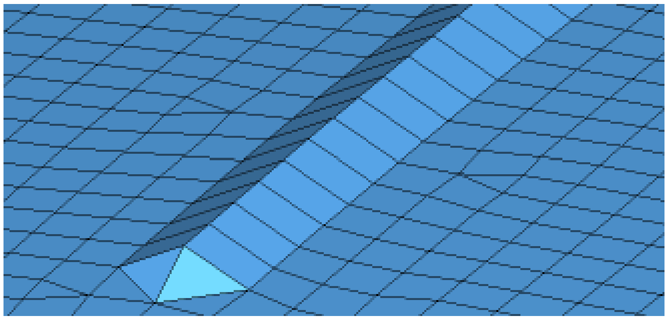


Figure 523:
Radius 10, height 5, sharp, with mesh size 8

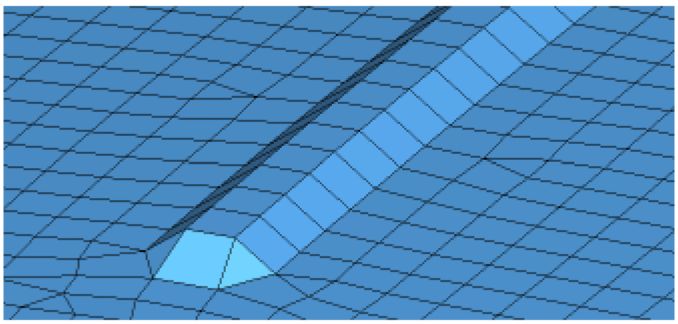


Figure 524:
Radius 10, height 5, flat, with mesh size 8

1. From the Utility Browser, Geom/Mesh menu, click **Bead**.
   A temporary panel is displayed in the panel area.
2. Select two nodes to define the beginning and end of the bead.
   Only two nodes are supported by this tool.
3. Click **proceed**.
   The panel closes and the **Create Bead - Two Nodes** dialog opens.
4. Enter a value for the beam radius.
   This value determines the width of the bead at its base. It is best to base this to some degree on the existing element size.
5. Enter a value for the base height.
   This is how far the bead rises above the mesh on which the end nodes reside.
6. Use the bead shape options to determine whether the bead has a flat top, or a peaked or rounded one.
7. When the characteristics are set, click **Create** to generate the bead. If the results are not satisfactory, click **Reject** and change the characteristics, then create again.

If you need to change the start and end nodes, you will need to **Reject** any bead already created, **Close** the utility, and then re-open it to select new nodes.

Optimize Tetramesh

Use this tool to modify an existing tetramesh, either by moving nodes or remeshing, to meet required parameters.

One function is to remove sliver elements – tetrahedral elements which are so flattened that all of their nodes are very close to planar. If the element's Aspect Ratio (the ratio of its maximum length to its minimum length) is high, the element is a sliver; otherwise, it is a wedge.

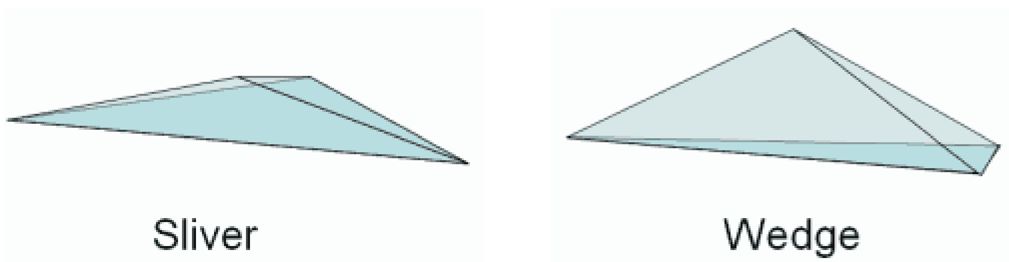


Figure 525:
This sliver is nearly flat in the horizontal plane, while this wedge is nearly flat in the vertical plane.

1. From the Utility Browser, Geom/Mesh menu, click **Tetra Mesh Optimization**.
   A temporary panel is displayed in the panel area.
2. Select a set of elements to fix.
3. Click **proceed**.
   The **Tetra Mesh Optimization** dialog opens, which contains the tools and settings for fixing slivers and wedges. The utility also has the ability to constrain trias, feature lines, nodes or elements within a refinement box.

   There are many criteria that you can consider in fixing such elements, each of which is drawn from the Criteria File Editor.
4. Edit the options in the dialog.

Option	Description
Aspect Ratio	The ratio of the longest edge of an element to its shortest edge.
Tet Collapse	Tetra collapse is calculated by the following procedure. At each of the four nodes of the tetra, the distance from the node to the opposite side of the element is divided by the square root of the area of the opposite side. The minimum value found is normalized by dividing it by 1.24, and then reported. As the tetra collapses, this value approaches 0.0. For a perfect tetra, this value is 1.0.
Vol Skew	Volumetric skew is calculated by the following procedure. A sphere is fit through the four nodes of the tetra. That sphere defines an ideally shaped equilateral

Option	Description
	tetra, whose volume is $8r^3/\left(9\sqrt{3}\right)$. The actual volume of the tetra element is then calculated. The element's volumetric skew is then (Videal - Vactual)/Videal. This measure will, normally, equal the skew measure from Tgrid, and equal 1 minus the equivalent check in Abaqus.
Skew	Skew applies to trias, so in this case it is applied to the faces of a tetrahedron. In trias is calculated by finding the minimum angle between the vector from each node to the opposing mid-side and the vector between the two adjacent mid-sides at each node of the element. Ninety degrees minus the minimum angle found is reported as the skew.
Vol AR	Vol AR for tetrahedral elements is calculated using the following procedure: first it finds the longest edge of the tetrahedron, then it finds the shortest altitude of the tetrahedron. The element's Vol AR, then, is the length of the longest edge divided by the length of the shortest altitude. For other types of 3D element, the ratio of the longest to the shortest edge is reported.
Edit Criteria	Opens the Criteria and Parameter File Editor, from which you can change the element quality requirements.
Triangles	The 2D triangles on the outermost surface of the tetrameshed volume, from which the initial tetramesh is derived. Multiple options for fixing exist:

Fix all

Nodes will be moved to improve tria quality, which will affect any tetras stemming from those trias.

Edge Swap

Pairs of triangles can have their edges swapped to create better tria elements:

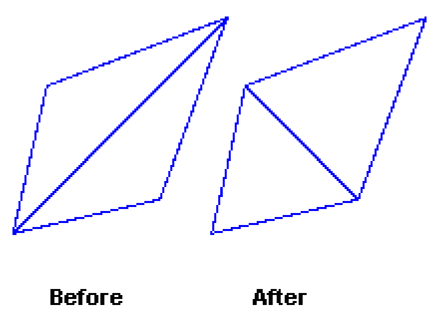


Figure 526:

Remesh

The current mesh will be discarded and a new mesh generated to fit the specified criteria. This usually produces the best results.

Option	Description
Constraints	Constraints prevent the utility from altering nodes in specific groups of elements. Four types of constraint can be specified: **Fixed Trias** Click **elements** and then select trias on the screen that you wish to remain unchanged during optimization. Only applies during Edge Swap or Remesh. **Feature Line** Click the **elements** button and then select 1D feature elements (if present) along one or more feature lines that you wish to preserve during optimization. These elements may be modified, but not in ways that alter the feature line. Only applies during Edge Swap or Remesh. **Anchor Nodes** Click the button and then select nodes on the screen that you wish to remain unchanged; these nodes will not be moved during the optimization process. **Refinement Box** Only elements inside of the refinement box will be optimized; the remainder will not. When you click **Components**, a temporary panel opens and you can select the refinement box component.
Fix shell comp boundaries	When unchecked, nodes on the edges of components will not be moved.
Maintain geometry edges	Nodes on the edges of geometric entities will not be moved.
Save to current comp	The resulting fixed mesh will be added to the current component. If this option is left unchecked, the results are saved to the same component they originated in.
Update input shells	Updates the shell elements attached to modified tetra elements. Only applies during Edge Swap or Remesh.
Optimize tetras by force	Adds a sophisticated node insertion algorithm to the optimization process. This typically yields higher element quality in the resulting mesh, but can also increase computation time.
Max tetra size	Threshold for the maximum tetra element edge length during mesh optimization.
Min tetra size	Threshold for the minimum tetra element edge length during mesh optimization.

Option	Description
Maximum Iteration	The tool will make this number of passes to fix elements; the more iterations, the better the results are likely to be, but each iteration takes some amount of time depending on the mesh complexity. Use smaller numbers to limit the amount of time spent, or larger numbers to achieve better end results.
Feature Angle	If the angle between the normals of two adjacent shell elements exceeds this value, the corresponding edge is treated as a feature edge and is preserved during the optimization process.
Show Failed	Isolates only the failed elements in the graphics area.
Check	Examine the mesh and count the number of bad elements, according to the criteria supplied (Jacobian, Volume Skew, and so on.) The results display in the Status area.

5. Click **Apply**.
   The mesh is scanned and the program will try to fix as many elements as it can in accordance with the specified settings and criteria.

   You can abort the fix attempt early by clicking holding down the right-mouse button.

   > **Note:** There can be a significant delay before HyperMesh finishes its current fix attempts and stops processing.

Abaqus Utility Menu

Macros are included on the Abaqus page of the Utility menu when you load the Abaqus user profile.

Table 27:

Utility	Description
Solid Face Alignment	3 Standard3D, Explicit
Align Faces	Determines the default stack or thickness direction for Abaqus composite solid, gasket and continuum shell elements.
Review	Opens the Element Selector panel, from which you to select solid elements. Selected elements are highlighted. When you click **proceed**, it highlights the face1 of selected solids and draws an arrow along the default stack (or thickness) direction of selected solids.
Reset	Deletes the stack (or thickness) direction arrows.

Utility	Description
Dummy	Applies to templates: Explicit
Positioning Process Manager	Tool that guides you through a workflow of positioning a dummy in a seat.
Tools	Applies to templates: Standard2D, Standard3D, Explicit
Contact Comparison	Activates the Tie-Comparison tool, from which you can compare the Tie-Contact elements created in HyperMesh to the Tie-Contact elements created by the Abaqus Solver.
Step Manager	Activates the Abaqus Step Manager, from which you can define Abaqus history (*STEP) information in HyperMesh.
Contact Manager	Activates the Abaqus Contact Manager, from which you can create, edit and review the following cards in HyperMesh: • *CONTACT • *CONTACT DAMPING • *CONTACT PAIR • *FRICTION • *PRE-TENSION SECTION • *SHELL TO SOLID COUPLING • *SURFACE, TYPE = ELEMENT • *SURFACE, TYPE = NODE • *SURFACE, COMBINE • *SURFACE, CROP • *SURFACE, TYPE = CUTTING SURFACE • *SURFACE, TYPE = CYLINDER, REVOLUTION or SEGMENTS • *SURFACE INTERACTION • *SURFACE BEHAVIOR • *TIE
Find and Replace	Searches the entity names in your model for every occurrence of a specific character, and automatically replaces the character with a new one that you specify.
SPH Mesh	Generates SPH elements for selected components in your model.

Utility	Description
	(Explicit only)

Compare Contacts

Use the Tie-Comparison tool to compare Tie-Contacts created in HyperMesh to Tie-Contacts created by the Abaqus solver.

In HyperMesh, Tie-Contact elements are created as RDE (Redundant Dummy Elements) elements, which correspond to Plotel elements in Nastran. The following types of elements are automatically converted to RDE elements: COUP_DIS, DCOUP3D, and KINCOUP. Once elements are converted, they are placed in the ^Master_Rods component.

1. From the Utility Browser, Abaqus menu, click **Contact Comparison**.
   The **Tie - Comparison Tool** dialog opens.
2. Select the component(s) containing the Tie-Contact elements to compare.
   a) Click **Component**.
   b) In the panel area, use the comps selector to select the components that contain the elements for which the comparison is required.
   c) Click **proceed**.

      The ^Master_Rods component appears under Rigid/RBE3 Elements.
3. Create a component for the Tie-Contact elements created by the Abaqus solver.
   a) Click **Create**.
   b) In the **Select Abaqus dat file containing tie constraints** dialog, open the file that contains the constraints.

      The ^Visualization_Rods component appears under Solver Tie Elements, which contains the RDE elements that were created using the information in the constraint file.
   c) To delete a component that was added to the list of Solver Tie Elements, select the component and click **Delete**.
4. Click **Compare**.
   The elements in the component from the Rigid/RBE3 Elements list is compared to the elements in the component from the Solve Tie Elements list. After the comparison, the corner nodes of RDE elements that have different independent nodes in the components being compared are marked.
5. Review the comparison.
   a) To clear the corner nodes that were marked during the comparison, click **Clear Nodes**.
   b) To retrieve and mark the corner nodes that were a result of the comparison, click **Retrieve Nodes**.
   c) To compare the elements in the component from the Rigid/RBE3 Elements list to the elements in the component from the Solver Tie Elements list, and mask the elements that have the same corner node and independent nodes, click **Mask Unchanged Elems**.

      If you click **Mask Unchanged Elems** again, the masked elements are unmasked.

d) To display the elements that are in the component from the Rigid/RBE3 Elements list but not in the component from the Solver Tie Elements list, and mask all of the other elements, click **Solver Missed Elements**.

If you click **Solver Missed Elements** again, the elements that were masked are unmasked.

Contact Manager

Use the Abaqus Contact Manager to create, edit and review cards.

Supported cards include:

- *CAVITY DEFINITION
- *CONTACT
- *CONTACT DAMPING
- *CONTACT PAIR
- *FRICTION
- *PRE-TENSION SECTION
- *SHELL TO SOLID COUPLING
- *SURFACE, TYPE = ELEMENT
- *SURFACE, TYPE = NODE
- *SURFACE, COMBINE
- *SURFACE, CROP
- *SURFACE, TYPE = CUTTING SURFACE
- *SURFACE, TYPE = CYLINDER, REVOLUTION or SEGMENTS
- *SURFACE INTERACTION
- *SURFACE BEHAVIOR
- *TIE

Start the Contact Manager

1. Load the Abaqus user profile.
2. From the Utility Browser, Abaqus menu, click **Contact Manager**.

   The following rules apply when you are using the Abaqus Contact Manager.

   - When the Contact Manager is minimized or behind the HyperMesh window, restore it by clicking **Contact Manager** in the Abaqus Utility menu.
   - To display the bubble help for a button, place the cursor over the button for a few moments.
   - Double-click on the interface, surface and surface interaction names in the table to open the corresponding edit windows. Right-click on the names to display pull-down menu options.
   - Table columns can be resized by positioning the cursor along a column border, pressing the left or right mouse button, and dragging the border to a new position.

- The Shift and Control keys can be used with a left mouse click to select multiple items in a table.
- Press Control and the left or right arrow key to move the cursor within the active cell. Use the left, right, up and down arrows to change the active cell.
- Right-click on the Review button to clear the review selections.
- If you create, update or delete components, groups, properties, or entity sets from HyperMesh panels while the Contact Manager is open, click **Sync** to update the Contact Manager with the new changes.
- In the Friction and Surface Behavior tables, right click in the tables to display a pull-down menu containing copy, cut and paste options. Comma delimited data can be copied, cut, or pasted in these tables. Relevant hot keys, for example, Control-c, Control-x, and Control-v on PC, will also work.
- In some fields in the Contact Manager, you can access the Entity Browser, which is available via the (…) button. The Entity Browser makes it more convenient to view and sort long lists of components or other entities when selecting them for the field.

Interface Tab

Create, edit, review, and delete interfaces, as welll as edit, review, and delete surfaces and surface interactions that are displayed on the Interface tab.

The Interface tab contains a description of the *CONTACT PAIR, *TIE, *PRE-TENSION SECTION, *CONTACT, and *SHELL TO SOLID COUPLING cards with corresponding surfaces and surface interactions.

The Interface table contains the following columns:

Table 28:

Column	Description
Name	The contact interface names. These names are not exported to the Abaqus input file. They are useful for identifying the various interfaces in HyperMesh.
Interface Type	The interface types. The currently supported types are contact pair, tie, pre-tension section, general contact, and shell to solid coupling.
Slave	The names of the slave surfaces in Abaqus Standard (or the first surface in Abaqus Explicit).
Master	The names of the master surfaces in Abaqus Standard (or the second surface in Abaqus Explicit).
Surface Interaction	The names of the surface interaction properties.

Column	Description
Slave display	The display on/off check boxes and color change buttons for the surfaces shown in the Slave column. The color can be changed by clicking the color button and selecting a color from the menu.
Master display	The display on/off check boxes and color change buttons for the surfaces shown in the Master column. The color can be changed by clicking the color button and selecting a color from the menu.

- The display on/off check boxes and color change buttons are disabled if the corresponding surface is defined with sets and no displays are created for them.
- Double-click on the interface, surface, and surface interaction names in the table to open the corresponding edit windows. Right-click on a name to display menu options.
- Right-click on an interface, surface, and surface interaction name to display menu options. The available options are:
    - Edit
    - Delete
    - Swap Master-Slave
    - Swap CP-Tie
    - Review
    - Review with underlying entity
    - Reset review
    - Review Options (Review by Highlighting, Review by Color Change, Transparency, and Grey Color)
    - Display All
    - Display None
    - Display Reverse
    - Draw Rigid Surfaces

  The Edit, Review, Delete, Display All, Display None and Display Reverse options work like the corresponding buttons (described below). Review with underlying entity highlights the surface along with the attached elements (or nodes). The Reset review button clears the review selections.
- Table columns can be resized by positioning the cursor along a column border, pressing the left or right mouse button, and dragging the border to a new position.
- The Shift or Control key and a left-click can be used to select multiple items in a table.

The Interface tab contains the following buttons:

Table 29:

Button	Action
Auto	Launches the **Auto Contact** dialog, from which you can quickly and easily create interactions between several parts of your model.
New ...	Opens the **Create New Interface** dialog in which you enter the name and type of the new interface. Use the Same as option to create an interface by copying from an existing interface. The Create button in this dialog creates the interface and opens the corresponding Contact Pair, Tie, Pre-Tension Section, Cavity Definition or **Shell to Solid Coupling** dialog.
Edit ...	Opens the corresponding dialog for editing the selected interface, surface, or surface interaction.
Review	Reviews the selected interface, surface, or surface interaction as follows: • For surfaces, the selected surface is highlighted in red in the HyperMesh window. If the surface is defined with sets, the underlying elements are highlighted. A right-click on the **Review** button clears the review selections. • For interface types, corresponding slave and master surfaces are highlighted in red and blue in the HyperMesh window. A right-click on the **Review** button clears the review selections. • For surface interactions, the names of all interfaces using the selected surface interaction in the table are highlighted. There is no graphical review in the HyperMesh window for surface interaction.
Delete	Deletes the selected interfaces, surfaces, or surface interactions. You can delete single or multiple selections from the table.
Rename	Rename the selected interface, surface, or surface interaction.
Sync	Updates the Contact Manager with the current HyperMesh database. If you manually create, update, or delete components, groups, properties, or entity sets

Button	Action
	from HyperMesh panels while the Contact Manager is open, click **Sync** to update the Contact Manager with the new changes.

Contact Pair

In the **Contact Pair** dialog, define the *CONTACT PAIR card. Options vary according to the active template.

The **Contact Pair** dialog contains the following buttons:

Table 30:

Button	Action
OK	Updates the HyperMesh database with the changes and closes the **Contact Pair** dialog.
Apply	Updates the HyperMesh database with the changes without closing the **Contact Pair** dialog.
Cancel	Closes the **Contact Pair** dialog without updates.

Contact Pair: Define Tab

In the Define tab, select the slave surface, master surface, and surface interaction for the *CONTACT PAIR card. You can also review the selected surfaces or create new ones.

The Define tab contains the following options:

Table 31:

Option	Description
Auto-generated surface from component	Select this option for HyperMesh to automatically generate *SURFACE cards from a selected component. When this option is selected, the Surface field becomes a Component field, and you can select a component from the adjacent drop-down list. Click **Slave>>** or **Master>>** to add them to the table of included surfaces as slave or master, respectively.
Surface	The Surface field contains a list of the existing surfaces. Select a slave surface from the list or use the … button to open the Entity Browser to select a surface.

Option	Description
	Click **Slave>>** to add the surface as a slave to the table of selected surfaces. Click **Master>>** to add the surface as a master. Click **Remove>>** to remove any selected surface from the table. You can add multiple sets of surfaces to the table. Click **New** to create a new surface. Once you have specified the surface properties, the surface appears in the drop-down list, where you can select it and add it to the table. The Review button highlights the selected slave surface in red and displays it through solid mesh in performance graphics in the HyperMesh window. If the surface is defined with sets, the underlying elements are highlighted. Right-click on **Review** to clear the review selections.
Interaction	The Interaction field contains a list of the existing surface interaction properties. You can select a surface interaction from the list. You can also use the (...) button to open the Entity browser to select a surface. The New button opens the **Create New Surface Interaction** dialog for creating a new surface interaction. When the new surface interaction has been defined, the **Contact Pair** dialog reflects the newly-created surface interaction as the interaction of the contact pair. **Note:** The surface interaction is optional in explicit template. The Define tab will show a Surface interaction check box if the explicit template is loaded. This option should be checked first if a surface interaction property is intended for the contact pair card.

If you create multiple pairs of contacts, they will appear on the Interface tab in separate entries using the same name.

Contact Pair: Parameter Tab

In the Parameter tab, define optional parameters for the contact pair card. Options vary according to the template loaded.

The supported parameters for the Standard.3d and Standard.2d templates include: Adjust, Extension zone, Smooth, No thickness, Clearance, Hcrit, Tied, Small sliding and Type. When the Type field is set to SURFACE TO SURFACE, the Geometric Correction field becomes activated. See the Abaqus Online Documentation for a detailed description of these parameters.

The supported parameters for the Explicit template include: Mechanical constraint, CPSET, OP, Weight, and Small sliding. See the Abaqus Online Documentation for a detailed description of these parameters.

Clearance Tab

In the Clearance tab, you can define an initial clearance value and a contact direction for a contact pair.

The Contact Manager can only be used to create a clearance definition for contact pairs that have one master-slave pair. If a contact pair has more than one master-slave pairs, then clearance has to be defined for each master-slave pair in the Entity Editor or in the Interfaces panel.

The Clearance tab is enabled when a master and slave surface are selected in the Define tab, and the Small sliding and Clearance check boxes are selected in the Parameter tab.

Create Clearance Definition for Contact Pair
Steps for creating a new clearance definition for a contact pair varies based on the type of clearance you select.
VALUE

1. In the Type field, select **VALUE**.
2. In the Value field, specify an initial clearance/overclosure for the entire set of slave nodes.

   > **Note:** A positive values specifies an initial clearance, and a negative value specifies an initial overclosure.

3. Click **Apply**.

TABULAR WITH INPUT FILE

1. In the Type field, select **TABULAR WITH INPUT FILE**.
2. In the Input File field, specify an input file that contains the clearance data.

   > **Note:** The input file can be a file name, relative path, or complete file path. A maximum of eighty characters are allowed in this field.

3. Click **Apply**.

TABULAR WITH BOLT

When using the TABULAR WITH BOLT clearance type, you can define clearance using nodes, node sets, or both nodes and node sets. In the example below, clearance is being defined using both nodes and node sets.

1. In the Type field, select **TABULAR WITH BOLT**.
2. Specify the following:
   - Half Thread Angle
   - Pitch
   - Major Thread Diameter
   - Minor Thread Diameter

Figure 527:

3. To define clearance with a node, select the **node** checkbox.
4. To add a data line to the table, click **Add**.
5. In the first data line, double-click the **Node Id** field.

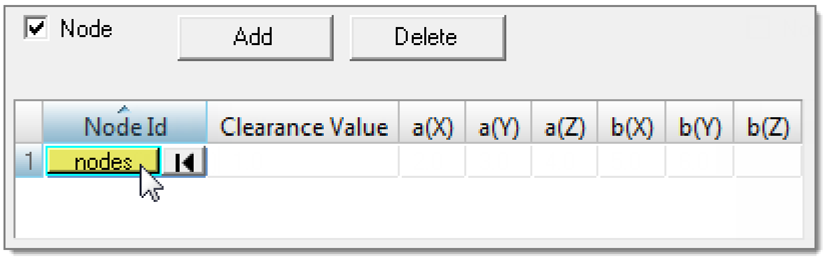


Figure 528:

6. Using the node collector, select a node.
7. To apply your selection and go back to the Contact Manager, click **proceed**.
8. Specify the following:
   - Clearance Value
   - a(X)
   - a(Y)
   - a(Z)
   - b(X)
   - b(Y)
   - b(Z)

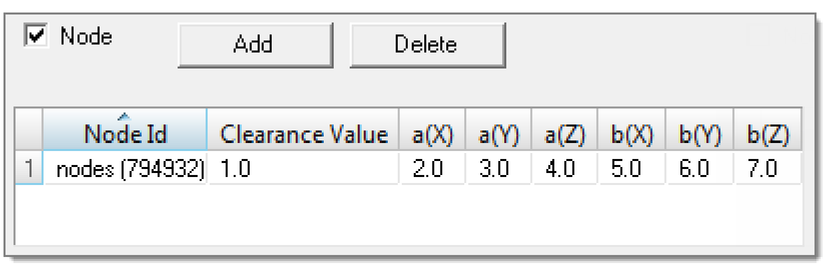


Figure 529:

9. To define clearance with a node set, select the **Node set** checkbox.
10. Optional: To create a new node set or edit an existing node set using the Entity Sets panel, click **Create/Edit**.

11. To add a data line to the table, click **Add**.

12. In the first data line, click the **Node Set** field and then select a node set from the list.

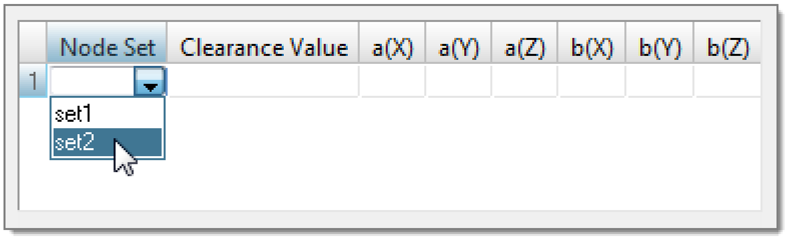


Figure 530:

13. Specify the following:

- Clearance Value
- a(X)
- a(Y)
- a(Z)
- b(X)
- b(Y)
- b(Z)

14. Click **Apply**.

TABULAR WITH BOLT AND INPUT

When using the TABULAR WITH BOLT AND INPUT clearance type, you can define clearance using nodes, node sets, or both nodes and node sets. In the example below, clearance is being defined using both nodes and node sets.

1. In the Type field, select **TABULAR WITH BOLT AND INPUT**.
2. To define clearance with a node, select the **node** checkbox.
3. To add a data line to the table, click **Add**.
4. In the first data line, double-click the **Node Id** field.

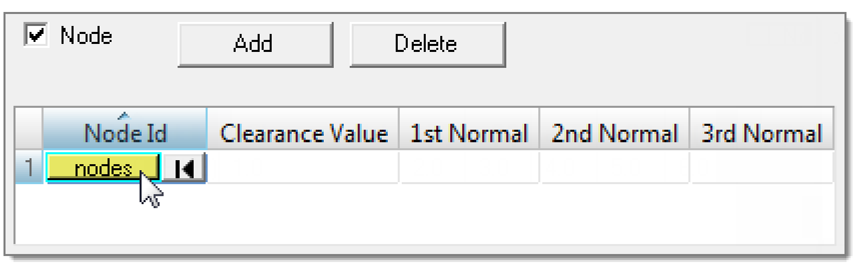


Figure 531:

5. Using the node collector, select a node.
6. To apply your selection and go back to the Contact Manager, click **proceed**.
7. Specify the following:

- Clearance Value
- 1st Normal
- 2nd Normal
- 3rd Normal

8. To define clearance with a node set, select the **Node** set checkbox.
9. Optional: To create a new node set or edit an existing node set using the Entity Sets panel, click **Create/Edit**.
10. To add a data line to the table, click **Add**.
11. In the first data line, click the **Node Set** field and then select a node set from the list.
12. Specify the following:
    - Clearance Value
    - 1st Normal
    - 2nd Normal
    - 3rd Normal
13. Click **Apply**.

Edit an Existing Clearance Definition for a Contact Pair

If you created a clearance definition using the Entity Editor, Interfaces panel, or imported a clearance definition for a given master-slave pair, you can display and edit the clearance details in the Contact Manager.

1. To read and display an existing clearance definition for a master-slave pair, select the **Small sliding** checkbox in the Parameter tab.
2. In the Clearance tab, edit the clearance details.
3. Click **Apply**.

Delete a Clearance Definition for a Contact Pair

To delete a clearance definition that was linked to a contact pair using the Contact Manager:

1. In the Parameter tab, clear the **Small sliding** checkbox.
2. Click **Apply**.

Clearance Tab Options

The following options are available in the Clearance tab:

Table 32:

Option	**Description**
Type	Specify a type of clearance to define: • VALUE • TABULAR WITH INPUT FILE • TABULAR WITH BOLT • TABULAR WITH BOLT AND INPUT

Option	Description
Value	Initial clearance/overclosure for the entire set of slave nodes. A positive value specifies an initial clearance, and a negative value specifies an initial overclosure.
Input File	Input file that contains the clearance data. The input file can be a file name, relative path, or complete file path. A maximum of eighty characters are allowed in this field.
Half Thread Angle	Angle between the threads.
Pitch	Distance between two threads.
Major Thread Diameter	The largest extreme diameter of a thread.
Minor Thread Diameter	The smallest extreme diameter of a thread.
Node	Activates the node table, from which you can specify a node to define clearance with.
Node set	Activates the node set table, from which you can specify a node set to define clearance with.
Add	Adds a data line to the table.
Delete	Deletes a data line from the table.
Create/Edit	Opens the Entity Sets panel, from which you can create a node set.
Node ID	Activates the node collector, which can be used to select a node to define clearance with.
Node Set	Enables a drop-down list from which you can select a node set to define clearance with.
Clearance Value	A positive value indicates an opening between the surfaces, and a negative value indicates overclosure. If this field is left blank, the clearance value will be calculated automatically.
a(X)	X-coordinate of point a along the axis of the bolt/bolt hole.
a(Y)	Y-coordinate of point a along the axis of the bolt/bolt hole.
a(Z)	Z-coordinate of point a along the axis of the bolt/bolt hole.
b(X)	X-coordinate of point b along the axis of the bolt/bolt hole.
b(Y)	Y-coordinate of point b along the axis of the bolt/bolt hole.
b(Z)	Z-coordinate of point b along the axis of the bolt/bolt hole.

Option	Description
1st Normal	First component of the normal.
2nd Normal	Second component of the normal.
3rd Normal	Third component of the normal.

Tie

In the **Tie** dialog, define the *TIE card.

The **Tie** dialog contains the following buttons:

Table 33:

Button	Action
OK	Updates the HyperMesh database with the changes and closes the **Tie** dialog.
Apply	Updates the HyperMesh database with the changes without closing the **Tie** dialog.
Cancel	Closes the **Tie** dialog without updates.

Tie: Define Tab

In the Define tab, select slave surface and master surface for the *TIE card. You can also review the selected surfaces or create new ones.

The Define tab contains the following options:

Table 34:

Option	Description
Auto-generated surface from component	Select this option for HyperMesh to automatically generate *SURFACE cards from a selected component. When this option is selected, the Surface field becomes a Component field, and you can select a component from the adjacent drop-down list. Click **Slave>>** or **Master>>** to add them to the table of included surfaces as slave or master, respectively.
Select slave surface	Select this option for HyperMesh to automatically generate *SURFACE cards from a selected component.

Option	Description
	When this option is selected, the Surface field becomes a Component field, and you can select a component from the adjacent drop-down list. Click **Slave>>** or **Master>>** to add them to the table of included surfaces as slave or master, respectively.
Select master surface	The Surface field contains a list of the existing surfaces. Select a slave surface from the list or use the … button to open the Entity Browser to select a surface. Click **Slave>>** to add the surface as a slave to the table of selected surfaces. Click **Master>>** to add the surface as a master. Click **Remove>>** to remove any selected surface from the table. You can add multiple sets of surfaces to the table. Click **New** to create a new surface. Once you have specified the surface properties, the surface appears in the drop-down list, where you can select it and add it to the table. The Review button highlights the selected slave surface in white and displays it through solid mesh in performance graphics in the HyperMesh window. If the surface is defined with sets, the underlying elements are highlighted. Right-click on **Review** to clear the review selections.

If you create multiple pairs of ties, they will appear on the Interface tab in separate entries using the same name.

Tie: Parameter Tab

In the Parameter tab, define optional parameters for the *TIE card.

The supported parameters are: Position tolerance, Tied nset, Cyclic symmetry (standard only), Constraint ratio, No rotation, Adjust, No Thickness and Type. The Position tolerance and Tied nset are optional mutually exclusive parameters. Select None if you do not want to select either of them. See the Abaqus Online Documentation for detailed descriptions of these parameters.

The Parameter tab contains the following options:

Table 35:

Option	Description
Tied nset	The Tied nset menu contains a list of existing node sets. You can select a node set from the list.
Review Set	The Review Set button reviews the selected node set by highlighting it in the HyperMesh window.

Option	Description
Create/Edit Set	The Create/Edit Set button opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The Tie window is updated with the new set displayed in node set list.

General Contact (Explicit Template Only)

Use the General Contact option to define the *CONTACT, *CONTACT INCLUSIONS, *CONTACT EXCLUSIONS and *CONTACT PROPERTY ASSIGNMENT.

The keywords *SURFACE PROPERTY ASSIGNMENT, *CONTACT FORMULATION, *CONTACT CONTROLS ASSIGNMENT and *SURFACE PROPERTY CONTACT CLEARANCE ASSIGNMENT cards are available for Explicit template only.

To open the Card Editor and define all of the relevant keywords, parameters and data lines, click **Edit** in the Contact Manager. When you are finished, click **return**.

Pre-Tension Section

In the **Pre-Tension Section** dialog, define the *PRE-TENSION SECTION section and related assembly loads and constraints.

The **Pre-Tension Section** dialog contains the following buttons:

Table 36:

Button	Action
OK	Updates the HyperMesh database with the changes and closes the **Pre-tension Section** dialog.
Apply	Updates the HyperMesh database with the changes without closing the **Pre-tension Section** dialog.
Cancel	Closes the **Pre-tension Section** dialog without updates.

Pre-Tension Section: Define Tab

In the Define tab, define a pre-tension section on existing bolts.

These bolts can be modeled by beams, truss or solid elements. This tab offers an automatic and manual way of defining the pretension section. You can also create loads and constraints applied to the pretension node from this tab.

Pre-Tension Definition

When the pre-tension definition is modeled by truss or beam elements, use the Select button to pick the 1D element to which the pre-tension shall be applied. When the Auto generate pre-tension node option is on, a pretension node in the middle of the selected element will be created. The same node will be populated in the By Node ID field and will also receive an HyperMesh tag Pretension_Node_<node id>.

If Auto generate pre-tension node is not active while selecting the 1D element, you can select a pre-existing node or node set, or as an option, create a node set and select this node set to apply pre-tension load and boundary conditions on.

When modeling a bolt by solid elements, you can take advantage of some automation as well. In this case you need to select a surface. If the Auto generate surface and pre-tension node option is enabled (Text dependent on either Element or Surface option), the only thing to specify is a base and a direction node.

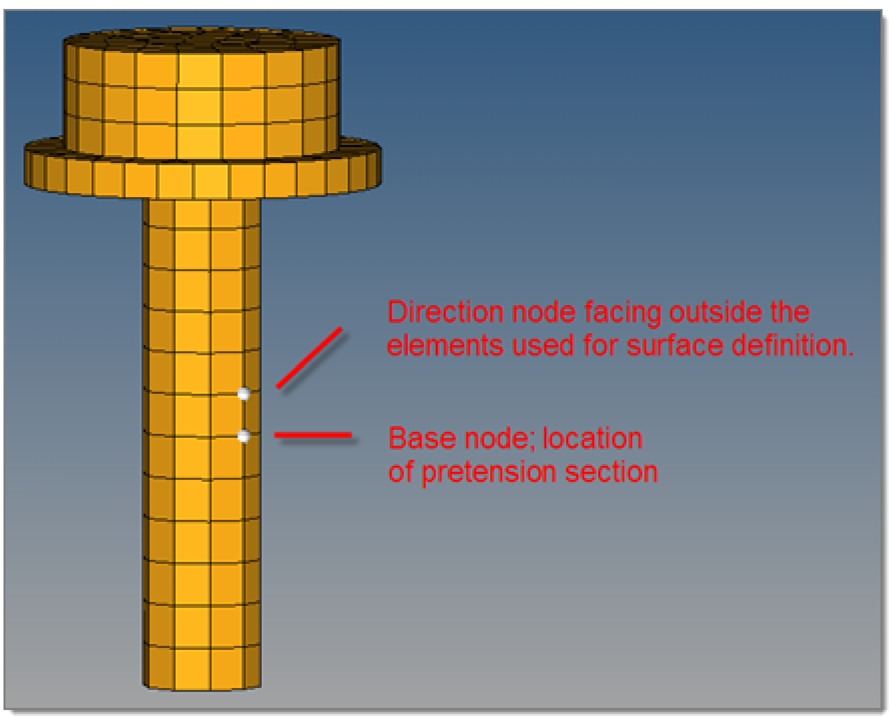


Figure 532:

The tool then creates a surface (*SURFACE) cutting through the bolt at the location of the base node. It also creates a node in the surface plane, outside the bolt and not attached to the model. This node automatically gets an HM tag HMpre-tension_node so that it is easier to find.

If you want to use existing surfaces, pre-tension nodes or node sets, you need to disable the automatic surface and node generation setting and assemble each part of the pre-tension section manually.

Assembly Load Definition

In the lower part of the dialog, you can directly apply a pre-tension load or constraint onto the pre-tension node (the node must exist). For the pre-tension load or constraint, a load collector named HMpre-tension_load or HMpre-tension_constraint is created or reused, if it already exists.

If you enable Fix assembly load, a second constraint is created in a load collector called HMpre-tension fix.

The assembly load/constraint definition part of the dialog can be used for both the manual and automatic process.

Pre-Tension Section: Parameter Tab

In the Parameter tab, define optional data lines for the *PRE-TENSION SECTION card.

Check **Dataline** to activate all three input boxes for the first, second, and third component of the normal.

Click **Define by vector** to define the values in the input boxes by a vector. To create a vector, click **Create/Edit vector**.

> **Note:** The NSET parameter is currently only supported on the card image.

Cavity Definition

In the **Cavity Definition** dialog, combine several element based surfaces to a heat cavity for heat transfer analyses.

Surfaces can either be selected from a drop down list, or from the extended entity selection menu. It is also possible to create a new surface while defining the cavity.

If you want to define thermal properties directly in the cavity definition instead of the surface card itself, surface properties can be selected for each surface, if the option is enabled with the Set property per surface option.

The dialog allows for ambient temperature definition as an option.

To add a surface to the cavity, select it from the Surface drop-down list and click **Add**. Surfaces can be removed from the definition by selecting it in the table and clicking **Remove**.

Auto Contact

In the **Auto Contact** dialog, define the *CONTACT PAIR and *TIE keywords along with the corresponding *SURFACE cards.

In the Abaqus user profile, use Auto Contact to quickly and easily create interactions between several parts of your model. Based on a proximity distance, Auto Contact will search the model and automatically define surfaces from identified components. The interactions and surfaces are placed into a temporary Auto Contact Browser, where you can review the pairs and make adjustments as needed.

Two types of interactions can be created by the auto contact functionality:

***CONTACT PAIR**
: Definition of pairs of surfaces, which can contact or interact during an analysis. When selecting this type of interaction, you must also specify the surface interaction properties.

***TIE**
: Definition of constraints and interactions between pairs of surfaces. No surface interaction definition is required.

Do not split interfaces between two components
: When enabled, Auto Contact will only create one contact pair or tie contact, even if components are within proximity distance in several places.
If disabled, Auto Contact will create separate interfaces (CONTACT PAIR or TIE), if common surfaces between two components are not fully connected.
For example, The yellow and red brackets are a component each and touch each other in two unconnected areas.

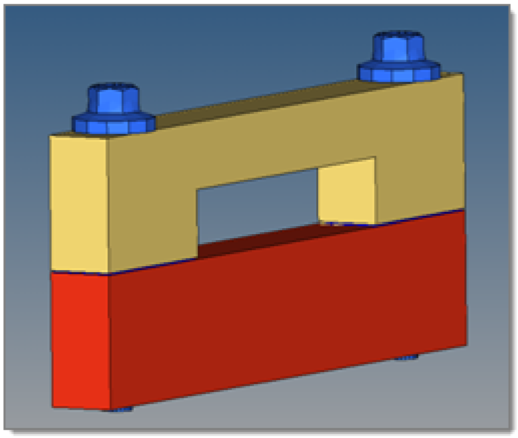


Figure 533:

With the option disabled both contact areas would result into a separate contact pair/tie definition, whereas with the option enabled (only one contact pair definition is created by the algorithm).

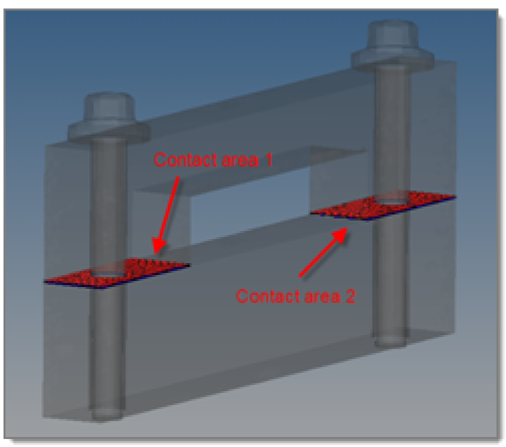


Figure 534:

The **Auto Contact** dialog contains the following buttons:

Button	Action
Find	Searches the model for interacting components.
Cancel	Closes the dialog without updates.
Remove Selection	Removes selected components from the table. You can use the Control and Shift key to select multiple items in the table.
Review Selection	Highlights the selected component in the graphic area. All other components are grayed out. You can use the Control and Shift key to select multiple items in the table.

Button	Action
	Right-click to return the model to normal display.
Help	Opens the Auto Contact online help.

Shell to Solid Coupling

In the **Shell to Solid Coupling** dialog, define the *SHELL TO SOLID COUPLING card.

The **Shell to Solid Coupling** dialog contains the following buttons:

Table 37:

Button	Action
OK	Updates the database with the changes and closes the dialog.
Apply	Updates the database with the changes without closing the dialog.
Cancel	Closes the dialog without updates.

Shell to Solid Coupling: Define Tab
In the Define tab, select the slave surface, master surface, and surface interaction for the *SHELL TO SOLID COUPLING card. You can also review the selected surfaces or create new ones.

The Define tab contains the following options:

Table 38:

Option	Description
Surface	The Surface: field contains a list of the existing surfaces. Select a slave surface from the list or use the … button to open the Entity Browser to select a surface. Click **Slave>>** to add the surface as a slave to the table of selected surfaces. Click **Master>>** to add the surface as a master. Click **<<Remove** to remove any selected surface from the table. You can add multiple sets of surfaces to the table. Click **New** to create a new surface. Once you have specified the surface properties, the surface appears in the drop-down list, where you can select it and add it to the table. The Review button highlights the selected slave surface in white and displays it through solid mesh in performance graphics in the window. If

Option	Description
	the surface is defined with sets, the underlying elements are highlighted. Right-click on **Review** to clear the review selections.

If you create multiple pairs of contacts, they will appear on the Interface tab in separate entries using the same name.

Shell to Solid Coupling: Parameter Tab
In the Parameter tab, define optional parameters for the contact pair card. Options vary according to the template loaded.

The supported parameters are:

For Standard.3d/2d template and Explicit template
INFLUENCE DISTANCE and POSITION TOLERANCE. To add them, enable the checkbox and then add an appropriate value. See the Abaqus Online Documentation for a detailed description of these parameters.

Set Up an Auto Contact Run

1. Load the Abaqus user profile and select either the **2D** or **3D** template.
2. From the Utility Browser, Abaqus menu, click **Contact Manager**.
3. Click **Auto**.
   The **Auto Contact** dialog opens.
4. Select the type of interface to create in the Type of Interface field.
5. Click the yellow components button and select your components.

   The components are automatically placed in the Component table in the **Auto Contact** dialog.
6. Enter a value in the Proximity Distance field.

   The proximity distance is the maximum distance between two selected components. When you create the pair, any surfaces that are farther away than the value entered here will not be created as a contact pair. The default value is zero.

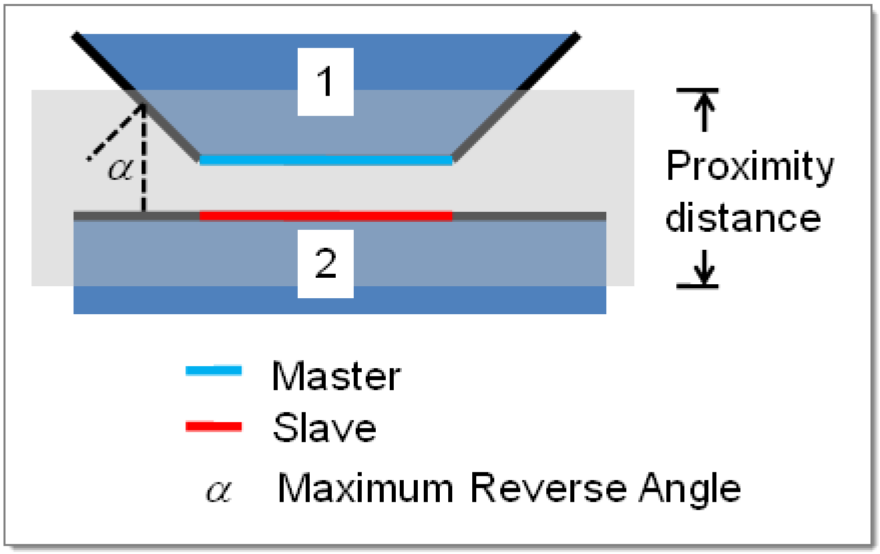
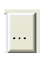


Figure 535:

7. Enter a value in the Maximum reverse angle field.

If the angle between two normals of elements or element faces exceeds this value, the element will not be added to the master or slave surface.

8. If creating a contact pair, select the type of interaction in the Interaction:field. You can use the  icon to open a browser to view the options more easily, or you can click **New** to create a new interaction.

9. Click **Find**.
The status bar activates and the Auto Contact Browser opens.

10. Use the Auto Contact Browser to make any necessary adjustments to the interfaces and surfaces.

11. When finished modifying, click **Create**.
The interfaces and surfaces marked as Accepted are created. The Contact Manager window reopens with the new information listed.

Auto Contact Browser

The Auto Contact Browser contains options for viewing and modifying the contact pairs identified in the auto contact process.

Naming Convention

During creation, a name is assigned to each interface and surface identified.

Interface

Contact Pair example: CP:(comp name)_(comp_name)

Tie example: TIE:(comp name)_(comp_name)

Surface

S:(comp name)

If more than one surface is selected, the naming convention uses S:(comp name):<index>

The Auto Contact browser contains the following columns:

Table 39:

Column	Description
Name	Lists the name of the interfaces, surfaces and surface interactions that were assigned. Underneath the interface name are the temporary surfaces included in that interface. Red indicates a slave surface, and blue indicates a master surface.
Accept	When the Accept box is checked, the Interface will be included in the creation process.
Type	Type of contact pair created.
Color	Color assigned to the interaction and surfaces.
Surface Interaction	Opens the Auto Contact online help.

The Auto Contact browser contains the following icons:

Table 40:

Icon	Description
Options	Opens the **Options** dialog. Enter a new feature angle or customize the transparency for a selected entity. Click **OK** when finished.
Highlight Elements	Highlights the elements stored in selected entities in the graphics window. You can use the Control and Shift key to select multiple items in the table.
Review Elements	Review of elements stored in the selected entities. Elements are highlighted by color, all other components are grayed out. You can use the Control and Shift key to select multiple items in the table. Review and Highlight are mutually exclusive. It is also possible to switch both options off. This is helpful when working with big models.
Fit View to Elements	Automatically zooms in to the elements stored in the currently selected items.
Display All Elements	In combination with the Highlight Elements or the Review Elements option, current contents remain unchanged on the screen.
Display Components with Elements	Highlights or reviews the elements referred by an interaction or surfaces and shows the components they belong to. All other components will be masked.

Icon	Description
Display Only Elements	Only elements are highlighted or reviewed. The rest of the component and other components will be masked.
Select Elements Manually	Opens the element selection panel so that individual elements can be added/removed manually. Click **proceed** when finished.
Add by Adjacent	Adds the elements adjacent to the surface to the selected surface. Right-click to undo one time.
Add by Face	Adds the adjacent face to the selected surface. Right-click to undo one time.
Recheck	Opens the **Auto Contact** dialog to recheck the select interfaces. Recheck will either add more contacts to the existing contacts for modify the existing ones. You can select interfaces from the browser, and the GUI will automatically populate the components that the interaction was based on. This helps modify an existing interface.

Modify Auto Contact Entities

Right-click on an item in the Auto Contact Browser.
A context menu displays which offers options for modifying the surfaces and contact pairs.

The following options are available from the Auto Contact Browser's context menu:

Table 41:

Option	Description
Rename	Rename an existing entry.
Delete	Delete items from the browser.
Swap Master-Slave	Switches the surfaces identified as master and slave. When selected, you will see the surfaces flip from the master/slave positions in the browser. Select multiple entities by using the Control and Shift keys when clicking on entities.
Swap CP-Tire	Changes the type of interface created. Because a surface interaction is required for Contact Pair but not for Tie, any surface interaction identified

Option	Description
	earlier will be lost upon a swamp from CP to TIE. If you switch back from TIE to CP, the surface interaction will not be retained.
Edit Faces	Opens the Elements Selection panel, from which you can select and deselect the elements to include on the face of the surface, and edit the faces of the surfaces.
Add by Adjacent	Adds adjacent elements to the selected surface.
Add by Face	Adds all elements to a selected surface, until the feature angle exceeds the value (the feature angle can by set by clicking the Options icon).
Accept All/None	Automatically accept or reject all items in the Auto Contact Browser.
Reverse	Reverses the current selections in the Accept column.
Expand All/Collapse	Expands or collapses folders in the Auto Contact Browser.

Surface Tab

The Surface tab displays a description of the *SURFACE cards with corresponding types. You can create, edit, review, and delete surfaces from this tab.

The Surface tab contains the following columns:

Table 42:

Column	Description
Name	The name of surfaces in the HyperMesh database.
Type	The types of surfaces. Currently supported types are ELEMENT, NODE, and ANALYTICAL RIGID (SEGMENTS, CYLINDER or REVOLUTION).
Display	The display on/off check boxes and color change buttons for the surface. The color of a surface can be changed by clicking on the color buttons and selecting a color from the menu.

- The display on/off check boxes and color change buttons are disabled if the corresponding surface is defined with sets and a display is not created.
- Double-click on a surface name in the table to open its corresponding editing dialogs.
- Right-click on a surface name to display menu options. The available options are:
    - Edit
    - Delete

   - Review
   - Review with underlying entity
   - Reset review
   - Review Options (Review by Highlighting, Review by Color Change, Transparency, and Grey Color)
   - Display All
   - Display None
   - Display Reverse
   - Draw Rigid Surfaces

  The Edit, Review, Delete, Display All, Display None and Display Reverse options work like the corresponding buttons (described below). Review with underlying entity highlights the surface along with the attached elements (or nodes). The Reset review button clears the review selections.
- Table columns can be resized by positioning the cursor along a column border, pressing the left or right mouse button, and dragging the border to a new position.
- The Shift or Control key and a left-click can be used to select multiple items in a table.

The Surface tab contains the following buttons:

Table 43:

Button	Action
Auto	Launches the **Auto Contact** dialog, from which you can quickly and easily create interactions between several parts of your model.
New	Opens the **Create New Surface** dialog, from which you can enter the name and type of the new surface. Click **Create** to create the surface and open the corresponding Element Based Surface, Node Based Surface, or Analytical Rigid Surface dialog or open the corresponding HyperMesh card image.
Edit	Opens the Element Based Surface, Node Based Surface, or Analytical Rigid Surfac edialog or takes you to the corresponding HyperMesh card image for editing the selected surface.
Review	Reviews the selected surface. Surfaces are highlighted in white and show up through solid mesh in performance graphics the window. If the surface is defined with sets (display option disabled), the underlying elements are highlighted. Right-click **Review** to clear the review selections.
Delete	Deletes selected surfaces. You can delete single or multiple selections from the Surface table.
Rename	Rename the selected surface
Sync	Updates the Contact Manager with the current HyperMesh database. If you manually create, update or delete components, groups, properties, or entity sets

Button	Action
	from HyperMesh panels while the Contact Manager is open, click **Sync** to update the Contact Manager with the new changes.
Close	Closes the Contact Manager.
Display All	Displays all surfaces in the graphic area.
Display None	Hides all surfaces in the graphic area.
Display Reverse	Displays all unchecked surfaces and hides all checked surfaces in the graphic area.
Draw Rigid Surfaces	Shows analytical rigid surfaces in the graphic area.

Element Based Surface

In the **Element Based Surface** dialog, define and edit the *SURFACE, TYPE = ELEMENT card.

The **Element Based Surface** dialog contains the following buttons:

Table 44:

Button	Action
Surface color	Changes the color of the current surface in the display.
Review	Reviews the current surface by highlighting in white and displaying it through solid mesh in performance graphics in the window. If the surface is defined with sets, the underlying elements are highlighted. Right-click **Review** to clear the review selections.
Close	Updates the HyperMesh database with the optional parameters specified and closes the **Element Based Surface** window.

Element Based Surface: Define Tab

In the Define tab, define surfaces for solid, shell, membrane, rigid, gasket, beam, pipe, or truss elements. You can also define the surface by specifying the face identifier for an element set.

Available surface definition options for various element types:

- 3D solid, gasket
- 3D shell, membrane, rigid
- 3D solid coated with shell
- 3D shell - edge based

- 2D solid, axisymmetric, gasket
- Beam, pipe, truss
- Element set

The layout of the Define tab changes, based on your selection (displayed in blue). Some options may be disabled depending on the current template.

3D Solid or Gasket Elements
Use the 3D solid or gasket elements option to define the *SURFACE card by specifying face identifiers for individual solid and gasket elements.

These faces are displayed by special face elements.

In order to create surface, you need to select the underlying solid or gasket elements first.

Click **Elements** to open the Element Selector panel, and then select the underlying 3D solid or gasket elements from the graphic area. Selected elements are highlighted. Click **Reset** to resets the selected elements.

You can then define the face identifiers for the selected solids in two ways: (a) by creating a solid skin and manually picking the faces from the skin, (b) by picking nodes on a specific face and sweeping through a break angle. Under Select faces by, select **Solid skin** for (a) and **Nodes on face** for (b).

(a) Solid skin enables the following:

Table 45:

Button	Action
Faces	Creates a temporary skin of the selected solids. Opens the Element Selector panel, from which you can select face elements from this skin. The selected faces are highlighted. Click **Reset** to reset the selected faces and delete the skin. **Note:** by face on the element selector panel can be used to find all faces within a feature angle of the selected face. The feature angle setting can be accessed by clicking **Preferences** > **Geometry Options**. The skin will initially have the same color as the current surface. You can change the skin color using Solid skin color button.
Add	Adds the selected faces to the current surface and creates special face elements for display. It also checks for duplicate faces and displays a message if any are found. **Note:** The Delete Face tab contains tools to find and delete duplicate faces in the current surface.
Reject	Rejects the recently added faces.

(b) Nodes on face enables the following:

Table 46:

Button	**Action**
Nodes	Opens the Node Selector panel, from which you can select nodes from the graphic area. Three nodes (or two corner nodes) from the same solid element must be picked to define a face of that solid. The selected nodes are highlighted. The corresponding Reset button resets the selected nodes. **Note:** Several three-node or two-corner-node sets can be selected at the same time to define faces in different solids.
Add	Finds all faces from the selected solids that fall within a specified break angle of the face(s) defined by nodes. These faces are then added to the current surface and create special face elements for display. It also checks for duplicate faces and displays a message if any are found. **Note:** The Delete Face tab contains tools to find and delete duplicate faces in the current surface.
Reject	Rejects the recently added faces.

3D Shell, Membrane or Rigid Elements
Use the 3D shell, membrane, and rigid elements option to define the *SURFACE card by specifying face identifiers for individual shell, membrane, and rigid elements.

In the graphics area, these faces are displayed by special face elements. These face elements have their own normals to define the SPOS and SNEG faces. The face with normals along the underlying element normals define the SPOS faces. In contrast, the face with opposing normals define the SNEG face.

The 3D shell, membrane, or rigid elements enable the following:

Table 47:

Button	**Action**
Elements	Opens the Element Selector panel, from which you can select underlying 3D shell, membrane, or rigid elements from the graphic area. The selected elements are highlighted and their normals are displayed. The corresponding Reset button resets the selected elements and hides the normals.
Add	Adds the selected elements to the current surface and creates special face elements for display. It also checks for duplicate faces and displays a message if any are found. By default, SPOS faces are created. In order to create SNEG faces, activate the **Reverse** checkbox and click **Add**.

Button	Action
Reject	Rejects the recently added faces.

3D Solid Coated with Shell

Use the 3D solid coated with shell option to define the *SURFACE card by specifying face identifiers for these 3D solid or gasket elements.

In HyperMesh, surfaces on 3D solid or gasket elements that are coated with shell, membrane, or rigid elements are treated differently from surfaces on regular solids.

The faces are displayed by special contactsurface elements. Unlike, regular solids, there is only one way to define the face identifiers for solids with shell coating: by picking nodes on a specific face and sweeping through a break angle. Therefore, the Nodes on face option is always selected. This option is valid for Standard.3D template or 3D models in Explicit template only.

3D solid coated with shell enables the following:

Table 48:

Button	Action
Elements	Opens the Element Selector panel, from which you can select underlying 3D solid and gasket elements from the graphic area. The selected elements are highlighted. The corresponding Reset button resets the selected elements.
Nodes	Opens the Node Selector panel, from which you can select nodes from the graphic area. Three nodes (or two corner nodes) from the same solid element must be picked to define a face of that solid. The selected nodes are highlighted. The corresponding Reset button resets the selected nodes. **Note:** Several three-node or two-corner-node sets can be selected at the same time to define faces in different elements.
Add	Finds all faces from the selected 3D solids that fall within a specified break angle of the face(s) defined by nodes. These faces are then added to the current surface and special contactsurface elements are created for display. **Note:** You cannot add duplicate contactsurfaces for the same element in HyperMesh. Therefore, the Add button does not check for duplicates and there is no Reject button.

3D Shell - Edge Based

Use the 3D shell – edge based option to define the *SURFACE card by specifying edge identifiers for 3D shell elements.

The edges are displayed by special contactsurface elements. Face identifiers for solids with shell coating are defined by picking nodes on a specific edge and sweeping through a break angle. Therefore, the Nodes on edge option is always selected.

3D shell – edge based enables the following:

Table 49:

Button	Action
Elements	Opens the Element Selector panel, from which you can select underlying 3D shell elements from the graphic area. The selected elements are highlighted. The corresponding Reset button resets the selected elements.
Nodes	Opens the Node Selector panel, from which you can select nodes from the graphic area. Two nodes from the same solid element must be picked to define a edge of that shell. The selected nodes are highlighted. The corresponding Reset button resets the selected nodes. **Note:** Several two-node sets can be selected at the same time to define edges in different elements.
Add	Finds all edges from the selected 3D shells that fall within a specified break angle of the edge(s) defined by nodes. These edges are then added to the current surface and special contactsurface elements are created for display. **Note:** You cannot add duplicate contactsurfaces for the same element in HyperMesh. Therefore, the Add button does not check for duplicates and there is no Reject button.

2D Solid, Axisymmetric or Gasket Elements
The 2D solid, axisymmetric or gasket elements option is valid for Standard.2D template or 2D models in Explicit template only.

Use it to define the *SURFACE card by specifying edge identifiers for individual 2D solid, axisymmetric, and gasket elements. In the graphic area, these edges are displayed by special contactsurface edge elements. Unlike, 3D solids, there is only one way to define the face identifiers for 2D solids: by picking nodes on a specific edge and sweeping through a break angle. Therefore, the Nodes on edge option is always selected.

2D solid, axisymmetric, or gasket elements enable the following:

Table 50:

Button	Action
Elements	Opens the Element Selector panel, from which you can select underlying 2D solid, axisymmetric, and gasket elements from the graphic area. The selected elements are highlighted. The corresponding Reset button resets the selected elements.
Nodes	Opens the Node Selector panel, from which you can select nodes from the graphic area. Two nodes from the same element must be picked to define an edge of that element. The selected nodes are highlighted. The corresponding Reset button resets the selected nodes. **Note:** Several node pairs can be selected at the same time to define edges in different element.
Add	Finds all edges from the selected 2D solids that fall within a specified break angle of the edge(s) defined by nodes. These edges are then added to the current surface and special contactsurface edge elements are created for display. **Note:** You cannot add duplicate contactsurface edges for the same element in HyperMesh. Therefore, the Add button does not check for duplicates and there is no Reject button.

Beam, Pipe or Truss Elements
Use the Beam, pipe or truss elements option to define the *SURFACE card for individual beam, pipe and truss elements.

In the graphic area, these faces are displayed by special contactsurface elements. These contactsurface elements have their own normals to define the SPOS and SNEG faces. The contactsurface with normals along the underlying element normals define the SPOS faces. In contrast, the face with opposing normals defines the SNEG face.

Note: For 3D beam, pipe and truss elements, the SPOS and SNEG faces do not have any meaning. Therefore, these face identifiers will be ignored by the Standard.3d template.

Beam, pipe, or truss elements enables the following:

Table 51:

Button	Action
Elements	Opens the Element Selector panel, from which you can select underlying beam, pipe or truss elements from the graphic area. The selected elements are highlighted. The corresponding Reset button resets the selected elements.
Add	Adds the selected elements to the current surface and creates special contactsurface elements for display. By default, SPOS faces are created. In order to create SNEG faces, activate the Reverse check box and click **Add**. **Note:** You cannot add duplicate contactsurfaces for the same element in HyperMesh. The Add button does not check for duplicates and there is no Reject button.

Element Set
Use the Element set option to define the *SURFACE card for element sets.

Only one elset is allowed in a surface. It does not support a combination of elsets and individual elements in the same *SURFACE data line.

The Element set menu contains a list of the existing elsets. You can also use the … button to open the Entity Browser to select an elset. There are two types of elsets in HyperMesh: Components and Entity sets. The Abaqus elsets that are linked to sectional property cards, such as *SOLID SECTION and *SHELL SECTION, become components in HyperMesh. Others become entity sets. To differentiate between these two types, there is a divider line "- - - - -" in the elset lists that pops up if you click the **Element set** menu. The elsets listed below the divider line are components.

Element set option enables the following:

Table 52:

Button	Action
Review Set	Reviews the selected elsets set by highlighting them in the the graphic area. Right-click on **Review** to clear the review selections.
Create/Edit Sets...	Opens the Entity Sets panel. When you finish creating/editing the set, click **return**. The Element Based Surface tab is updated with the new set appearing in the element set list.
Show Faces	Creates a temporary skin of the selected elset, opens the element selector panel, from which you can select face elements from this skin. When you return from the Element Selector panel, the selected faces will display color coded face identifier

Button	Action
	tags. In the graphic area, these tags are sometimes blocked by the solid mesh. You may need to rotate the model a little to view the tags.
Update	Adds the selected elset into the current surface. By default, HyperMesh does not create a display for surfaces defined with elsets. However, if you check the Display option before clicking **Update**, it creates a special display using contactsurface elements. **Note:** The special display created with contactsurface elements does not have links to the elset in the HyperMesh database. Therefore, if you edit the elset later on, the display will not automatically reflect your changes. In this case, check the Display option and click **Update** again.

After selecting an element set, click the arrow keys to move the set into table on the right. Once an elset has been added to the table, the face column becomes activated and you can manually define the appropriate face identifier for the selected elset. Select **None** if you do not want to define a face identifier for the set. In this case, Abaqus will create a surface with the free faces for the selected element set.

Element Based Surface: Adjust Normal Tab

In the Adjust Normal tab, you can display and reverse normal direction for surfaces defined on 3D shell, membrane, rigid and 2D beam, pipe, and truss elements.

For these elements, the normal directions are used to define the SPOS or SNEG face identifiers. In HyperMesh, surfaces can be displayed in two ways: by special face elements or by contactsurfaces. The surfaces defined on 3D shell, membrane, rigid elements are displayed by face elements while surfaces on 2D beam, pipe, and truss elements are displayed by contactsurfaces. The normal directions of the contactsurfaces are part of their display. However, the normal directions of the faces have to be turned on for displaying. Use the Display normals checkbox to display the normals of the faces. Use the Size: entry box to define the size of the normals before selecting the **Display normals** checkbox.

Note: 3D solid and gasket elements are also displayed by face elements. But, the normal direction do not have any meaning for them.

There are two options for reversing normals: (a) Reverse all normals at a time, (b) Reverse normals by individual faces or contactsurfaces.

(a) Reverse normal all

Table 53:

Button	Action
Reverse	Reverses normals of all faces for 3D shell, membrane, rigid elements and all contactsurfaces for 2D beam, pipe, truss elements in the current surface. **Note:** The normals of the underlying elements are not reversed. It reverses the normals of the faces or contactsurfaces only.

(b) Reverse normals: By element

Table 54:

Button	Action
Element faces	Opens the Element Selector panel. For 3D shell, membrane, rigid elements, pick the faces (not the underlying elements). However, for 2D beam, pipe, truss elements, pick the elements themselves as the contactsurfaces can not be picked from the graphic area. The corresponding Reset button resets the selected elements.
Reverse	Reverses normals of the selected faces for 3D shell, membrane, rigid elements and contactsurfaces of the selected 2D beam, pipe, truss elements. **Note:** The normals of the underlying elements are not reversed. It reverses the normals of the faces or contactsurfaces only.

Element Based Surface: Delete Face Tab
Use the Delete Face tab to delete faces or contactsurface from all elements types.

Surfaces can be displayed in two ways: by special face elements or by contactsurfaces. The surfaces defined on 3D solid, gasket, shell, membrane, rigid elements are displayed by face elements while surfaces on 2D solid, axisymmetric, gasket and all beam, pipe, truss elements are displayed by contactsurfaces.

This tab contains tools to find and delete duplicate faces for 3D solid, gasket, shell, membrane, or rigid elements.

There are three options for deletion.

(a) All

Table 55:

Button	Action
Delete	Deletes all the faces or contactsurfaces in the current surface.

(b) By element

Table 56:

Button	Action
Element Faces	Opens the Element Selector panel. For 3D solid, gasket, shell, membrane, and rigid elements, pick the faces (not the underlying elements). However, for 2D solid, axisymmetric, gasket or all beam, pipe, truss elements, pick the elements themselves as the contactsurfaces can not be picked from the graphic area. The corresponding Reset button resets the selected elements.
Delete	Deletes the selected faces or contactsurfaces from the selected 2D solid, axisymmetric, gasket, beam, pipe, truss elements.

(c) Duplicate faces

Table 57:

Button	Action
Find Duplicate Faces	Finds duplicate faces, if any exists, for the current surface and highlights them in the graphic area. The status bar shows the number of duplicates found. The corresponding Reset button resets the selected elements.
Delete	Deletes the highlighted duplicate faces found. **Note:** Duplicate contactsurfaces cannot be added to the same surface in HyperMesh. Therefore, Duplicate Faces is only valid for 3D solid, gasket, shell, membrane, and rigid elements.

Element Based Surface: Optional Parameters Tab

Use the Optional Parameters tab to define optional parameters for the *SURFACE card.

The supported parameters are:

- Standard.3D/2D template: Trimming of open free surface, Property selection

- Explicit template: Max ratio, Scale thickness, Region type for adoptive meshing, No offset and No thickness.

Click **Update** to activate the Optional Parameters selection in the HyperMesh database.

Node Based Surface

Use the **Node Based Surface** dialog to define and edit the *SURFACE, TYPE = NODE card.

The **Node Based Surface** dialog contains the following buttons:

Table 58:

Button	Action
Surface color	Changes the color of the current surface in the graphic area.
Review	Reviews the current surface by highlighting it in white and displays it through solid mesh in the graphic area. If the surface is defined with sets, the underlying nodes are highlighted. Right-click **Review** to clear the review selections.
Close	Updates the HyperMesh database with the optional parameters specified and closes the Node Based Surface window.

Node Based Surface: Define Tab

Use the Define tab to define surfaces with individual nodes and define the surface by specifying a node set.

The layout of the Define tab changes based on your selections.

Individual Nodes
Use the Individual nodes option to define the *SURFACE, TYPE = NODE card by specifying individual node IDs.

In the graphic area, these nodes are displayed by special single node elements with a SurfaceNodes tag.

The Individual nodes option has the following buttons:

Table 59:

Button	Action
Pick Nodes	Opens the Node Selector panel, from which you can select nodes the graphic area. The selected nodes will be highlighted. The corresponding Reset button resets the selected nodes.

Button	Action
Add	Adds the selected nodes to the current surface and creates special single-node SurfaceNodes elements for display. The Add button does not check for duplicates and there is no Reject button.

Node Set
Use the Node set option to define the *SURFACE, TYPE = NODE card for node sets.

Only one node set is allowed in a surface. A combination of node sets and individual nodes in the same *SURFACE data line is not supported.

The Node set: menu contains a list of the existing node sets.

Selecting **Node set** enables the following:

Table 60:

Button	Action
Review Set	Reviews the selected node sets by highlighting the nodes in the graphic area. Right-click **Review** to clear the review selections.
Create/Edit Set	Opens the Entity Sets panel. When you finish creating/editing the set, click **return**. The **Node Based Surface** dialog is updated with the new set appearing in the node set list.
Area	Select this check box to define the optional cross-sectional area at each node of the node set selected. **Note:** The cross-sectional area data item is only supported for surfaces defined by node sets.
Update	Adds the selected node set into the current surface. HyperMesh does not create a display for surfaces defined with node sets.

Node Based Surface: Delete Surface Node Tab

Use the Delete Surface Node tab to delete the SurfaceNode elements.

Node bases surfaces are displayed by special single node SurfaceNode elements.

There are two options for deletion:

(a) All

Table 61:

Button	Action
Delete	Deletes all SurfaceNodes elements from the current surface.

(b) By node

Table 62:

Button	Action
Surface nodes	Opens the Element Selector panel for you to select SurfaceNodes elements from the graphic area. The corresponding Reset button resets the selected elements.
Delete	Deletes the selected SurfaceNodes elements from the current surface.

Node Based Surface: Optional Parameter Tab

Use the Optional Parameter tab to define optional parameters for the *SURFACE card.

The supported parameters are:

- Standard.3D/2D template: Trim
- Explicit template: Max ratio, Scale thickness, Region type, No offset and No thickness.

Click **Update** to activate the optional parameter selection in the HyperMesh database.

Surface Combine or Crop
Use the Surface Combine or Crop option to define the *SURFACE, COMBINE, or CROP.

The current version of Contact Manager does not have a Tcl/Tk dialog to define the surface combine or crop. Review by highlighting or color change also does not work for these types of surfaces in HyperMesh. Instead, it takes you to the corresponding Card Image panel for edit or review.

Click **edit** to go into the Card Editor to define all relevant keywords, parameters, and data lines. When you are finished, click **return** to go back to the Contact Manager.

Cutting Surface
Use the Cutting Surface option to define the *SURFACE, TYPE = CUTTING SURFACE.

The current version of Contact Manager does not have a .tcl/Tk dialog to define the cutting surface. Review by highlighting or color change also does not work for these types of surfaces in the graphic area. Instead, it takes you to the corresponding Card Image panel for edit or review.

Click **edit** to go into the Card Editor to define all relevant keywords, parameters, and data lines. When you are finished, click **return** to open the Contact Manager.

Analytical Rigid Surface

Use the **Analytical Rigid Surface** dialog to define the *SURFACE, TYPE = CYLINDER, REVOLUTION or SEGMENTS card.

When you click **Create**, the **Analytical Rigid Surface** dialog opens.

Each tab of the **Analytical Rigid Surface** dialog contains the following buttons:

Table 63:

Button	Action
Surface color	Changes the color of the current surface in the display.
Review	Reviews the current surface by highlighting line segments and generated revolute or swept surfaces in white. Right-click on the **Review** button to clear the review selections.
Close	Updates the HyperMesh database with the optional parameters that you specified and closes the Analytical Rigid Surface window.

Analytical Rigid Surface: Define Tab

Specify the main characteristics of the surface on this tab.

Depending on which Abaqus template you have loaded (Standard3D, Standard2D, or Explicit), you can create surfaces defined by segments, cylinders, or revolutions.

- The Standard3D template enables you to define rigid surfaces of type REVOLUTION and CYLINDER.
- The Standard2D template enables you to define surfaces of type SEGMENTS.
- The Explicit template enables you to define all three types: REVOLUTION, CYLINDER, and SEGMENTS.

You can define the line segments in three different ways, by picking nodes, by picking existing lines, or by manually entering the coordinate values. The options on the Define tab vary according to the type selected.

SEGMENTS Option

Select the following options on the Define tab for planar rigid surfaces:

Plane Definition - Plane/axis

Abaqus does not require the plane/axis definition for SEGMENTS type. In HyperMesh, however, the XY plane must be used for a 2D model. Therefore, the XY plane is selected by default. If the model is not in the XY plane, you can choose to select a "User Defined" plane on which the rigid surface should be defined.

If you choose to manually define the plane/axis, you must enter values for three points (the origin, x axis, and y axis) that define the local coordinate system.

You can also use the following buttons to automatically define the plane in the Local System table:

Pick System...
: Click to select an existing system from the model. Once you select a plane from the model and click **proceed**, the values are populated in the table.

Create/Edit System
: Click to create new coordinate systems using the Systems panel.

Line Definition - No. of line segment datalines

In this field you specify the number of datalines needed to define the line segments. The actual number of line segments is one less than this number.
Start by typing the number of datalines needed to define the line segments and click **Set**. The corresponding number of rows appears in the Line Type table below. In this table, specify the coordinates of the ends of each line segment.
The first entry in the table is always the START node. This value specifies the beginning point of the first segment. The subsequent segments' starting point is always the end point of the previous segment, or the START node if the segment is the first in the definition.
For each line type, select a type from the Line Type column: LINE, CIRCL, or PARAB. Each selection activates the appropriate number of columns for the segment definition.
The segments can be circles, parabolas, or lines. Enter data in the columns as described below:

- Lines are defined with the x and y coordinates of the end point in the two active columns.
- For circles, specify the x and y coordinate of the end point in the first two columns and then, define the x and y coordinate of the center in the last two columns.
- For parabolas, specify the x and y coordinate of the mid point in the first two columns and then, define the x and y coordinate of the end point in the last two columns. You can also pick nodes or lines from existing geometry using the following buttons:

Pick Nodes...
: Click to pick nodes from the HyperMesh model for selected line segments. When you click **proceed**, the coordinate values of the selected nodes will appear in selected line segment cells. In addition, temporary line segments (white color) will also be drawn in the graphic area from the picked nodes. Ensure that you select two nodes in the correct order for circles and parabolas.

Pick Lines...
: Use this option to define line segments from existing lines in HyperMesh. These lines must be single curvature, connected and node1 of a line must be same as node2 of the previous line.

: Click to pick a line from the HyperMesh model. When you click **proceed**, the coordinate values and line types for the selected lines will appear in the table.

> **Note:** In HyperMesh, the sequence of node1 and node2 for lines can be visualized from the Line Edit/Extend Line panel.

Click **Update** to update the HyperMesh database with your settings.

CYLINDER Option

Select the following options on the Define tab for cylindrical rigid surfaces:

Plane Definition - Plane/axis

Choose User Defined if you want to create or select the system, or choose XY, YZ, or XZ to define it in the respective plane.

If you choose to manually define the plane/axis, you must enter values for three points (the origin, x axis, and generator axis) that define the local plane on which the line segments will be defined.

You can also use the following buttons to define the plane:

Pick System...

Click to select an existing system from the model. Once you select a plane from the model and click **proceed**, the values are populated in the table.

Create/Edit System

Click to create new coordinate systems using the Systems panel.

Line Definition - No. of line segment datalines

In this field you specify the number of datalines needed to define the line segments. The actual number of line segments is one less than this number.

Start by typing the number of segment datalines to define the surface and click **Set**. The corresponding number of rows appears in the Line Type table below. In this table, specify the coordinates of the ends of each line segment.

The first entry in the table is always the START node. This value specifies the beginning point of the first segment. The subsequent segments' starting point is always the end point of the previous segment, or the START node if the segment is the first in the definition.

For each line type, select a type from the Line Type column: LINE, CIRCL, or PARAB. Each selection activates the appropriate number of columns for the segment definition.

The segments can be circles, parabolas, or lines. Enter data in the columns as described below:

- Lines are defined with the local x- and local y-coordinates of the end point in the two active columns.
- For circles, specify the local-x and local-y coordinate of the end point in the first two columns and then, define the local-x and local-y coordinate of the center in the last two columns.
- For parabolas, specify the local-x and local-y coordinate of the mid point in the first two columns and then, define the local-x and local-y coordinate of the end point in the last two columns.

> **Note:** The local x- and local y-coordinates must be relative to the plane defined in the Plane/axis table.

You can also pick nodes or lines from existing geometry using the following buttons:

Pick Nodes...

Click to pick nodes from the HyperMesh model for selected line segments. When you click **proceed**, the coordinate values of the selected nodes will appear in selected line segment cells. In addition, temporary line segments (white color) will also be drawn in the graphic area from the picked nodes. Ensure that you select two nodes in the correct order for circles and parabolas.

Pick Lines...

Use this option to define line segments from existing lines in HyperMesh. These lines must be single curvature, connected and node1 of a line must be same as node2 of the previous line.

Click to pick a line from the HyperMesh model. When you click **proceed**, the coordinate values and line types for the selected lines will appear in the table.

> **Note:** In HyperMesh, the sequence of node1 and node2 for lines can be visualized from the Line Edit/Extend Line panel.

Line Definition - Sweep Distance

Abaqus does not need the sweep distance. The CYLINDER type surfaces are swept to infinity in Abaqus. However, in HyperMesh, you must define a sweeping distance to draw the three-dimensional surface. Select the **Sweep distance** check box to specify a sweep distance and type a value in the adjacent box. Select the **Both directions** check box to sweep in opposite directions along the generator vector.

Click **Update** to update the HyperMesh database with your settings.

REVOLUTION Option

Select the following options on the Define tab for rigid surfaces of revolution:

Plane Definition - Plane/axis

Choose User Defined if you want to create or select the system, or choose XY, YZ, or XZ to define it in the respective plane.
If you choose to manually define the axis, you must enter values for two points (the origin and the z axis). Abaqus does not need the x axis values because any plane that passes through the z axis will define the same revolute surface. However, HyperMesh requires the definition of the x (or radial) axis to define the plane on which the line segments are drawn.
You can also use the following buttons to automatically define the axis of revolution and x (radial) axis.

> **Note:** The coordinate values to define the x axis change if you pick a node (or line) from HyperMesh as the START node.

Pick System...

Click to select an existing system from the model. Once you select a plane from the model and click **proceed**, the values are populated in the table.

Create/Edit System

Click to create new coordinate systems using the Systems panel.

Line Definition - No. of line segment datalines

In this field you specify the number of datalines needed to define the line segments. The actual number of line segments is one less than this number.
Start by typing the number of line segment datalines to define the surface and click the **Set** button. The corresponding number of rows appears in the Line Type table below. In this table, specify the coordinates of the ends of each line segment.

The first entry in the table is always the START node. This value specifies the beginning point of the first segment. The subsequent segments’ starting point is always the end point of the previous segment, or the START node if the segment is the first in the definition.
For each line type, select a type from the Line Type column: LINE, CIRCL, or PARAB. Each selection activates the appropriate number of columns for the segment definition.
The segments can be circles, parabolas, or lines. Enter data in the columns as described below:

- Lines are defined with the local-x (or r) and local-y (or z) coordinates of the end point in the two active columns.
- For circles, specify the local-x (or r) and local-y (or z) coordinate of the end point in the first two columns and then, define the local-x (or r) and local-y (or z) coordinate of the center in the last two columns.
- For parabolas, specify the local-x (or r) and local-y (or z) coordinate of the mid point in the first two columns and then, define the local-x and local-y (or z) coordinate of the end point in the last two columns.

You can also pick nodes or lines from existing geometry using the following buttons:

Pick Nodes...
Click to pick nodes from the HyperMesh model for selected line segments. When you click **proceed**, the coordinate values of the selected nodes relative to the defined plane will appear in selected line segment cells. In addition, temporary line segments (white color) will also be drawn in the graphic area from the picked nodes. Ensure that you select two nodes in the correct order for circles and parabolas.

Pick Lines...
Click to pick a line from the HyperMesh model. When you click **proceed**, the coordinate values and line types for the selected lines will appear in the table.

> **Note:** In HyperMesh, the sequence of node1 and node2 for lines can be visualized from the Line Edit/Extend Line panel.

Line Definition - Revolution angle
Abaqus does not need the revolution angle. The REVOLUTION type surfaces are revolved around 360 degrees in Abaqus. However, in HyperMesh, you must define a revolution angle to draw the three-dimensional surface. Select the Revolution angle check box to specify the angle of revolution and type a value in the adjacent box.

Click **Update** to update the HyperMesh database with your settings.

Analytical Rigid Surface: Adjust Normal Tab

Select the following options in the Adjust Normal tab:

Table 64:

Option	Description
Display normals	Select the Display normals check box to show the normals on the display. Specify a length for the normals in the Size field. To change the size, toggle the check box to update the display.
Reverse normals	Select one of the radio buttons to reverse the direction of the normals. You can only choose to reverse the normals of all line segments in this type of surface. Click **Reverse** to do this. When reversed, all coordinates and sequence of all the line segments will be updated. The analytical rigid surface should be oriented so that the outward normals point toward any body of contact.

Analytical Rigid Surface: Rigid Body Tab

Associate the surface to an existing *RIGID BODY component or create a new one from this tab.

An analytical rigid surface must have a *RIGID BODY card with a reference node associated with it. When you create a new rigid surface in the Contact Manager, an empty *RIGID BODY component is created automatically.

This tab also includes several options related to preparing the model for visualization in HyperView. HyperView currently does not support geometrical entities like analytical rigid surfaces. If you mesh analytical rigid surfaces with rigid elements that point to the same *RIGID BODY card, these elements would not participate in the analysis; they would move with the reference node as a rigid body. These rigid elements would act like a "display body" in Abaqus, and would be imported in HyperView.

Select the following options on the Rigid Body tab:

Table 65:

Option	Description
Select a *Rigid Body	Use the drop-down list or the Entity Browser to select a rigid body. You can also create a new rigid body by clicking **Create New**.
Reference Node	Type the node ID used as a reference to define the rigid body motion. Click **Pick Node** to pick a node from the model. Alternatively, type a node value in

Option	Description
	the box and click **Review** to view the location of the node in the model. If you type a value that does not exist in the model, nothing is highlighted.
Line mesh density	Specify the density of the line mesh. Uniform uses the value you specify as the mesh density for each line segment. Variable brings you to the Line Mesh panel to create the mesh. Select either Uniform or Variable and click **Mesh** to create the line mesh.
Sweep distance/ Sweep angle	The sweep field differs depending on whether the surface is of type CYLINDER or REVOLUTION. If the surface is cylindrical, the field is Sweep distance. Type a value for the distance of the sweep of the mesh. Select the Both direction check box to extend the sweep in both directions for the specified distance. When the Both direction check box is selected, the number of layers refers to the depth of segments in the sweep (number of layers of elements). If the surface is a revolution, the field is Sweep angle. Type a value for the angle of the revolution. In the No. of layers field, type the value of how many segments the rim of the revolution should be divided into.

Analytical Rigid Surface: Optional Parameters Tab

Select the following options on the Optional Parameters tab:

Table 66:

Option	Description
Trimming of open free surface	Select to specify open free surface trimming. Then click the adjacent button to select whether to trim.
Max ratio	Select to adjust the thicknesses for surface facets in which the thickness to minimum edge or diagonal length ratio exceeds the specified value. Then type an adjustment value in the adjacent field.
Scale thickness	Select to scale all of the surface facets by a single factor. Then type the scaling factor in the adjacent field.
Region type for adaptive meshing	For surfaces defined on the boundary of an adaptive mesh domain, select to create a boundary region for the surfaces. Click the adjacent button to select the type of region.
No offset	Select for the surface to ignore midplane offsets.

Option	Description
No thickness	Select for the surface to ignore thicknesses. Do not select this option if the surface will be double-sided or a self-contact surface.
Fillet radius	Select to define a radius of curvature to smooth discontinuities between segments. Then type a length in the adjacent field.

Surface Interaction Tab

The Surface Interaction tab displays a description of the *SURFACE INTERACTION cards. You can create, edit, review and delete surface interactions from this tab.

The Surface Interaction table contains one column, the Name column. The name of the surface interaction cards in the HyperMesh database are listed in this column.

- Double-click on a surface interaction name in the table to open the corresponding **Surface Interaction** dialog. Right-click on a name to display an option menu.
- Table columns can be resized by positioning the cursor along a column border, pressing the left or right mouse button, and dragging the border to a new position.
- The Shift or Control key and a left-click can be used to select multiple items in a table.

The Surface Interaction tab contains the following buttons:

Table 67:

Button	Action
Auto	Launches the **Auto Contact** dialog, from which you can create interactions between several parts of your model.
New	Opens the **Create New Surface Interaction** dialog in which you enter the name of the new surface interaction. Click **Create** to create the surface interaction and open the corresponding **Surface Interaction** dialog.
Edit	Opens the **Surface Interaction** dialog for editing the selected surface interaction.
Review	Not active on this tab.
Delete	Deletes single or multiple surface interactions from the Surface Interaction table.
Rename	Rename the selected surface interaction.
Sync	Updates the Contact Manager with the current HyperMesh database. If you manually create, update, or delete components, groups, properties, or entity sets from HyperMesh panels while the Contact Manager is open, click **Sync** to update the Contact Manager with the new changes.

Button	Action
Close	Closes the Contact Manager

Surface Interaction

Use the **Surface Interaction** dialog to define the *SURFACE INTERACTION card.

Supported cards:

Mechanical interaction properties

- *SURFACE BEHAVIOR
- *CONTACT DAMPING
- *FRICTION

Thermal interaction properties

- *GAP CONDUCTANCE
- *GAP HEAT GENERATION
- *GAP RADIATION

The **Surface Interaction** dialog contains the following buttons:

Table 68:

Button	Action
OK	Updates the HyperMesh database with the changes and closes the **Surface Interaction** window.
Apply	Updates the HyperMesh database with the changes without closing the Surface Interaction window.
Cancel	Closes the **Surface Interaction** window without any update.

Surface Interaction: Define Tab

Use the Define tab to select the surface interaction properties.

The available options are: Surface behavior, Contact damping, Friction and Gap Conductance. Once you select an interaction property, its corresponding tab will be activated. You can also define optional parameters (Pad thickness) and data lines (out-of-plane thickness for 2D model or cross-sectional area at every nodes for node based surface). See the Abaqus Online Documentation for a detailed description of these parameters.

In addition, the Gap heat generation value can be defined directly in this dialog, it does not require you to open a new tab, as the other options do. If the Gap heat generation is enabled, the following two values need to be defined:

- Dissipated energy fraction - specifies the ratio of how much of the energy in a non thermal contact such as friction is producing heat
- Weighting factor - decides how much heat goes in one or the other surface of the contact

By default 50 percent of one value is applied to the other (so the default value is 0.5).

Surface Interaction: Surface Behavior Tab

Use the Surface Behavior tab to create *SURFACE BEHAVIOR cards with optional parameters and corresponding data lines.

The supported optional parameters are: No separation and Pressure overclosure. Options vary according to the selection made in the Pressure overclosure drop down.

Five types of pressure-overclosures are supported:

Table 69:

Pressure overclosure	Description
Hard	In the 2D/3D template, selecting this option provides four new radio buttons: Augmented lagrange, Direct, Penalty, and None. • If Penalty is selected, select whether the penalty is linear or nonlinear and enter the corresponding information in the fields that appear. • In the Explicit template, there are no data lines needed for this option.
Exponential	There are three fields to define the data line for this option. They are: Clearance at zero contact pressure, Pressure at zero clearance and Direct (for 2D/3D templates only) or maximum stiffness (for explicit only).
Linear	There are two fields to define the data line for this option. They are Direct (for 2D/3D templates only) and Slope of the pressure-overclosure curve.
Tabular	There is a table available for defining the data line values for this option. You need to input the number of data lines required at the Number of data lines entry box. Clicking the corresponding Set button will update the table to have the specified number of rows. For inputting values in the table, click on a cell to make it active and write down the values from keyboard. The table works like a regular spread sheet. You can also read comma delimited data from a text file by clicking **Read from file**. This button opens a file browser window. Select the file and click **Open** to export the comma delimited data. The row number is set to the number of data lines found in the file.

Pressure overclosure	Description
	Note: • Right-click in the table to display a pull-down menu containing copy, cut and paste options. Comma delimited data can be copied/ cut into or pasted from the clipboard using these options. Hot keys, for example, Control-C, Control-X and Control-V on PC, can also be used. • Left-click in a cell to activate the cell. Click in an active cell to move the insertion cursor to the character nearest the mouse. • The Shift and Control keys can be used with a left mouse click to select multiple items in a table. • Press Control and the left or right arrow key to move the cursor within the active cell. Use the left, right, up and down arrows to change the active cell. • Press Backspace to delete the character before the insertion cursor in the active cell. If multiple cells are selected, Backspace deletes all selected cells. • Press Delete to delete the character after the insertion cursor in the active cell. If multiple cells are selected, Delete deletes all selected cells. • Table columns can be resized by positioning the cursor along a column border, pressing the left or right mouse button, and dragging the border to a new position.
Scale Factor	Modify the default contact stiffness by a scale factors. Available only in the Explicit user profile.

Surface Interaction: Contact Damping Tab

Use the Contact Damping tab to create *CONTACT DAMPING cards with corresponding parameters and data lines.

The supported parameters are: Definition and Tangent fraction (explicit only). The two definition types supported are: Damping coefficient and Critical damping fraction (explicit only). The data item entry options change based on the current template. See the Abaqus Online Documentation for a detailed description of these parameters.

Surface Interaction: Friction Tab

Use the Friction tab to create *FRICTION cards with corresponding parameters and data lines.

The supported friction types (mutually exclusive parameters) are: Default (Coulomb), Elastic slip, Slip tolerance, Lagrange multiplier, and Rough. Depending on the template loaded and friction type selected, the window layout changes to show only the relevant options for defining other parameters and data items. Other supported optional parameters are: Exponential decay, Test data, Anisotropic, Taumax, and Dependencies. See the Abaqus documentation for detailed descriptions of these parameters.

> **Note:** The friction type User is not supported in the Contact Manager. However, it is supported in HyperMeshin the *SURFACE INTERACTION card image.

For all friction types (except Rough), there are four options to define the friction coefficient:

Table 70:

Option	Description
Direct	This is the default method for defining the friction coefficient. Selecting this option means that the Exponential decay and Anisotropic parameters will not be written in the input file. The No of Dependencies checkbox and corresponding entry box should be used to define the Dependencies parameter. There is a table available for defining the corresponding data lines. The available data items are: Friction coefficient, Slip rate, Contact pressure, Average temperature at the contact point, and average field variable values. The column numbers in the table will change based on the setting for No of Dependencies. The row numbers can be defined at the No of data lines entry box. Click **Set** to update the table to reflect the specified number of rows. To enter values in the table, click on a cell to make it active and write down the values from keyboard. The table works like a regular spread sheet. You can also read comma delimited data from a text file by clicking **Read From a File**. This button opens a file browser. Select the file and click **Open** to export the comma delimited data. The row number will be set to the number of data lines found in the file.
Anisotropic	Defines the data lines for the Anisotropic parameter. The No of Dependencies check box and corresponding entry box should be used to define the Dependencies parameter. There is a table available for defining the corresponding data lines. The available data items are: Friction coefficient1 (first slip direction), Friction coefficient2

Option	Description
	(second slip direction), Slip rate, Contact pressure, Average temperature at the contact point, and average field variable values. The column numbers in the table will change based on the setting for No of Dependencies. The row numbers can be defined at the No of data lines entry box. Click **Set** to update the table to reflect the specified number of rows. To enter values in the table, click on a cell to make it active and write down the values from keyboard. The table functions like a regular spread sheet. You can also read comma delimited data from a text file by clicking **Read From a File**. This button opens a file browser. Select the file and click **Open** to export the comma delimited data. The row number will be set to the number of data lines found in the file.
Exponential decay	Defines the data lines for the Exponential decay parameter. The available data items are: Static friction coefficient, Kinetic friction coefficient, and decay coefficient
Exponential decay, test data	Defines the data lines for the Exponential decay, test data parameter. The available data items are: Friction coefficient at point 1 (first data line), Friction coefficient at point 2 (second data line), Slip rate at point 2 (second data line) and Kinematic friction coefficient (optional third data line).

When using the Direct and Anisotropic tables:

- Right-click in the table to display a pull-down menu containing copy, cut and paste options. Comma delimited data can be copied/cut into or pasted from the clipboard using these options. Hot keys, for example, Control-C, Control-X and Control-V on PC, can also be used.
- Left-click in a cell to activate the cell. Click in an active cell to move the insertion cursor to the character nearest the mouse.
- The Shift and Control keys can be used with a left mouse click to select multiple items in a table.
- Press Control and the left or right arrow key to move the cursor within the active cell. Use the left, right, up and down arrows to change the active cell.
- Press Backspace to delete the character before the insertion cursor in the active cell. If multiple cells are selected, Backspace deletes all selected cells.
- Press Delete to delete the character after the insertion cursor in the active cell. If multiple cells are selected, Delete deletes all selected cells.
- Table columns can be resized by positioning the cursor along a column border, pressing the left or right mouse button, and dragging the border to a new position.

Surface Interaction: Gap Conductance Tab

Use the Gap Conductance tab to define the heat transfer between two surfaces.

Two definitions are possible, both of which can be combined:

- Clearance dependant relationship
- Pressure dependent relationship

If enabled, the upper part of the dialog writes a *GAP CONDUCTANCE card with the distance versus conductance relationship entered in the table. If you enable the Pressure dependent definition in the lower part of the dialog, the same card with an additional PRESSURE parameter is written out, specifying a pressure versus conductance relationship.

For both tables, additional dependencies can be specified by increasing the number in the No of Dependencies field.

Surface Interaction: Gap Radiation Tab

Use the Gap Radiation tab to define a viewfactor versus distance relationship for heat transfer between surfaces which are very close to each other.

Emissivities for both involved surfaces can be defined. By default, the dialog offers two datalines to define a function of the viewfactor depending on the gap with between two surfaces. The length of the table can be increased by changing the Number of datalines field.

Use the Dummy Positioning Process Manager

The Dummy Positioning Process Manager tool guides you through the workflow of positioning a dummy in a seat.

The recommended steps include an interactive positioning process, during which you specify the H-point position of the dummy, as well as the rotation angles of each joint, such as arm and leg. When finished with this phase, you will be able to store a transformation file which can later be applied on the dummy only in the next phase, the automatic positioning process. In this phase, you can choose to export only the nodes, which creates a copy of the original dummy file with the updated nodes. The rest of the dummy input file remains unchanged with this option.

In the Process Manager tab, you will see the workflow process. As you complete the positioning process, the boxes will become filled with green check marks to indicate the completion of each step.

Use the H-Point subpanel to position the dummy to the H-Point, or rotate the entire dummy about the H-Point. For positioning, you can specify either the coordinates or a node for the new H-Point. For rotating, specify the axis of rotation and the angle. In either case, picking any component in the dummy is sufficient.

Use the incremental subpanel to rotate an assembly about the coordinate system specified in the tree structure. In this panel, you have an option to rotate about the child or the parent system. The min stop and max stop angles for the x, y, z, axis associated with the joint are retrieved from the dummy database. When the minimum and maximum angles are reached, the assembly is not allowed to rotate any further.

> **Note:** Do not use the Rotate panel in the Tools panel to rotate the dummy. Doing so corrupts the tree structure and produces incorrect results when you reposition the dummy.

1. In the Abaqus Explicit template, open the Utility menu and click **PositioningProcess**.

2. Click **Create/Open** to start a new Process Manager instance.
3. In the panel area, select the **Interactive positioning** radio button and click **Apply**.
4. Click the file browser icon to browse for the dummy input file (`*.nip`) to open.
5. In the Positioner filename field, browse for the `*.pos` file to load. When both files are established, click **Load** .

   You can change the default settings for the import options as needed. Default settings include using free format import and preserving include files upon import.

   > **Note:** The **Import Process Message** dialog may appear. If it does, click **Close** and continue the process. If you are intending to exchange only the nodal coordinates of the dummy file at the end of the process, eventually reported unsupported cards do not require action in this case.

6. In the next panel, use the browse button to select the non-dummy files, such as the seat. If you have more than one file to load, click **Add** to display more fields. Click **Load** to load the file(s).

Figure 536:
Use the Load File icon to browse for the file to open.

7. After the files are loaded, click on the **H-point** subpanel and position the dummy in the H-Point using the fields available. Click the **comps** button and select the dummy graphically. Enter the coordinates to position the model and rotate as needed.
8. Once the dummy is positioned in the H-point, click on the **Incremental** subpanel. Make your changes and click **return**.
9. In the next panel, you can visualize the stop angles (lower/upper/current) for the elements used in the dummy. Click the **Show Plot** button to display the plot in the graphics area. Click **Next** to move to the next panel.
10. In the field, enter a name and folder location and click **Create**.
    The transformation file is created. This is needed to automatically position the dummy later when no other file (such as the seat) is loaded.
11. In the Create documentation panel, you can create an HTML report that lists the document angles and positions of the dummy, as well as screen shots of the model. Enter a name and location for the HTML document and click **Create**.
12. In the Export Files steps, select whether to export the model or save it as an HyperMesh file. You can also choose to skip this task and export the dummy later, during the automatic process. Click **Export** to export the model, or click **Next** to skip the step.
13. The Automatic Positioning phase now begins. In this panel, the files entered at the beginning of the InteractivePositioning process are already filled in the fields. Click **Load** to load the files into HyperMesh. Click **OK** to delete the current model.

> **Note:** The **Import Process Message** dialog box may appear. If it does, click **Close** and continue the process. If you are intending to exchange only the nodal coordinates of the dummy file at the end of the process, eventually reported unsupported cards do not require action in this case.

The Transformation filename is automatically loaded in the panel.

14. Click **Execute**.
15. Similarly to the InteractivePositioning phase, you can create an HTML document with the model specifics. Click **Create**, or click **Next** to skip the documentation task.
16. The last task is to export the dummy solver format or create an HyperMesh file. If exporting in solver format, the recommended setting is to select **exchange only nodes**. This will copy the original dummy file to a new one that uses a name you specified, and will exchange the node positions. You'll find a comment in front of and after the node block to highlight the exchanged nodes.

    The comments are:

    ** Begin replaced node block by dummy positioning tool

    ** End replaced node block by dummy positioning tool

    The Process Manager tree should now be completed, with each task showing a green check mark.

Solid Face Alignment Utility

Use this tool to align the face1 of selected solid elements (hexa and penta) to match with a planar face.

The Solid Face Alignment utility uses the face1 to face2 direction to determine the default stack or thickness direction for Abaqus composite solid, gasket and continuum shell elements.

As a result, the default stack (or thickness) directions for all selected elements become normal to a plane. In addition, you can review the face1 and default stack (or thickness) direction of selected solids. The utility has three buttons:

Table 71:

Button	Action
Align Faces	Click Align Faces to open the Element Selector panel, from which you can select solid elements. Selected elements are highlighted. When you click proceed, it creates a temporary skin of the selected solids and allows you to pick face elements from this skin. The selected faces are highlighted. When you click proceed, the face1 of all selected solid elements will match with the selected face element and a review of the stack (or thickness) direction will be shown.
Review	Opens the Element Selector panel, from which you can select solid elements. Selected elements are highlighted. When you click proceed, it highlights

Button	Action
	the face1 of selected solids and draws an arrow along the default stack (or thickness) direction of selected solids.
Reset	Deletes the stack (or thickness) direction arrows.

Abaqus Step Manager Overview

Use the Step Manager to create, edit and review supported keywords.

To access this tool, load the Abaqus user profile and select **Step Manager** from the Utility menu. Use the Step Manager to create, edit, and review the following keywords in HyperMesh begins in the Step tab:

Analysis types:

- *BUCKLE
- *DYNAMIC
- *FREQUENCY
- *HEAT TRANSFER
- *MODAL DYNAMIC
- *STATIC
- *STEADY STATE DYNAMICS

Loads:

- *BOUNDARY
- *CLOAD
- *DLOAD
- *DSLOAD
- *FILM
- *CFILM
- *SFILM
- *TEMPERATURE
- *INERTIA RELIEF
- *INITIAL CONDITIONS {TYPE=TEMPERATURE, VELOCITY}

Output requests:

- *OUTPUT
- *NODE OUTPUT
- *ELEMENT OUTPUT
- *STEP OUTPUT
- *ENERGY OUTPUT
- *NODE FILE

- *EL FILE
- *STEP FILE
- *ENERGY FILE
- *NODE PRINT
- *EL PRINT
- *STEP PRINT
- *ENERGY PRINT

Interface controls:

- *MODEL CHANGE
- *CONTACT INTERFERENCE
- *CHANGE FRICTION
- *CLEARANCE
- *CONTACT CONTROLS

Others:

- *MONITOR
- *FILE FORMAT
- *PRINT
- *RESTART
- *STEP

Step Manager Dialog Environment

Procedures used to navigate you through the **Step Manager** dialog.

- When the **Step Manager** dialog is minimized or it is behind HyperMesh, restore it by clicking **Step Manager** in the Abaqus Utility menu.
- Double-click on a cell entry to open the corresponding editing window.
- The first row contains a list of all initial condition (or model) load collectors. Double click on an **Initial Condition** name to open the corresponding dialog for defining initial condition loads.
- Right-click on a table to display menu options. Examples of options are Display: all, Display: none, Display: reverse, Text review, Review Load collectors, Reset review, Review Options, Reorder, Export: all, and Export: none. The Text review and Review load collectors options work like the Text and Review buttons, respectively (see Abaqus Step Manager: Load Step Dialog). The Reset review option clears the highlighted selections.
- Table columns can be resized by positioning the cursor along a column border, pressing the left or right mouse button, and dragging the border to a new position.
- The Shift or Control key combined with a left-click can be used to select multiple items in a table.
- To display bubble help, place the cursor over a button for a few moments.
- Press Control and the left or right arrow key to move the cursor within the active cell. Use the left, right, up, and down arrows to change the active cell.
- Right-click on the **Review** button to clear the highlighted selections.

- If you create, update or delete steps, load collectors, output blocks, components, groups, properties, or entity sets from panels while the **Step Manager** is open, click **Sync** to update the **Step Manager** with the new changes.
- In some fields in the **Step Manager**, you can access the Entity Browser, which is available via the (…) button. The Entity Browser makes it more convenient to view and sort long lists of components or other entities when selecting them for the field.

Entity Browser

Use the Entity Browser to easily sort and select an entity for a field on a dialog.

The Entity Browser is available through the … button and lists the entities of the relevant type for the selection.

The entities are listed in a tree view. To select an entity, highlight it in the list and click **OK**. Use the buttons on the dialog to sort the list to more easily view the choices when there are many entities in the list. The buttons perform the following actions:

Table 72:

Button	Action
Filter	Type text in the adjacent field and click **Filter** to show only the entities with names that contain the text. You can use the wildcard character (*) to specify that the text can appear anywhere within the entity name. To clear the results and show all entities, right-click **Filter**.
Review	Highlights the selected entity in the model.
Sort	Lists the entities in alphabetical order. Click again to reverse the order.
OK	Selects an entity for the field on the dialog.
Cancel	Closes the Entity Browser without making a selection.

Step Manager Tab Environment

- Editable cells have a white background, unless it is the active cell. You can input values using the keyboard in editable cells. Non-editable cells have a gray background.
- Left-click in an editable (white background) cell to activate it for input. Click in an active cell to move the insertion cursor to the character nearest the mouse.
- Press Control and the left or right arrow key to move the cursor within the active cell. Use the left, right, up, and down arrows to change the active cell.
- Press Backspace to delete the character before the insertion cursor in the active cell.
- Press Delete to delete the character after the insertion cursor in the active cell.

- Table columns can be resized by positioning the cursor along a column border, pressing the left or right mouse button, and dragging the border to a new position.
- In some fields, you can use the (…) button to open the Entity Browser to select the entity for that field.

Abaqus Step Manager: Step Tab

You can create, edit, review, rename, reorder, and delete load steps from this tab as well as set the export and display status of the load steps.

The Abaqus **Step Manager** opens when you load the Abaqus user profile and click **Step Manager** in the Abaqus Utility menu. The **Step Manager** window is organized into two tabs: Step and Load Case.

The Step tab contains the descriptions of existing load steps with the corresponding analysis types, load collectors, output requests, and interface controls.

The Step table contains the following columns:

Table 73:

Column	Description
Export	The export status of the load step. If the export status is on, the load step is written to the input file when exported from HyperMesh. If the export status is off, the STEP is not exported to the input file.
Name	The name of the load step.
Analysis Type	The analysis type of the load step.
Load Collector	The list of load collectors in the load step.
Output Block	The list of output blocks in the load step.
Interface Controls	The list of interface controls in the load step. Interface Controls defines the following Abaqus keywords: Contact Pair, Surface Interaction, Contact, Contact Controls, Clearance, Contact Interference, Model Change, Change Friction, and Controls.
Display	Turns the load step display on/off.

The Step tab contains the following buttons:

Table 74:

Button	Action
New...	Opens the **Create New Step** dialog. Enter the name of the new load step in the Name: text box and click **Create** to create the load step and open the

Button	Action
	corresponding **Load Step editing** dialog. Use the Same as option to create a new load step by copying from an existing load step.
Edit...	Opens the **load step editing** dialog for the selected load step.
Review	Reviews the selected load collectors. All loads in the load collectors are highlighted in the graphic area. The highlighted loads show through the solid mesh in performance graphics. If a load is defined with set, the underlying nodes or elements are highlighted. Right-click **Review** to clear the highlighted selections.
Text	Reviews the selected load step in a text window.
Rename	Opens the Rename panel for renaming load steps, load collectors, output blocks, or various interface controls. When you finish renaming, click **return** to update the **Step Manager** with the new names.
Delete	Deletes the selected load steps.
Sync	Updates the **Step Manager** with the current HyperMesh database. If you manually create, update, or delete load steps, load collectors, output blocks, groups, or entity sets from panels while the **Step Manager** is open, click **Sync** to update the **Step Manager** with the new changes.
Close	Closes the **Step Manager**.
↑	Moves the selected load steps up one row.
↓	Moves the selected load steps down one row.

Abaqus Step Manager: Load Step Dialog

Use the **Load Step** dialog to define the *STEP-*END STEP block as well as the associated Abaqus history keywords.

To open this dialog, double-click on the load step name in the Name: column on the Step tab or select the load step from the Name column and click **Edit**. Options vary according to the active template. There are three vertical sections in this dialog.

Left-Most Section

Contains a tree structure with various Abaqus history options. Selecting an option from the tree changes the dialog layout.

Middle Section

Contains the corresponding collectors in a table and relevant buttons to create, review, organize, reorder, rename, or delete these collectors. All loads must be organized into load collectors and all output requests must be under output blocks. When you select a load type, output request type, or interface control from the tree, the load collector table, output block table, or interfaces table is displayed in the middle section.

Right-Most Section

Contains the various tabs and options to define, edit, review, or delete the currently selected tree item. If the tree item needs to be organized in a collector, such as a load or output request, a collector must be selected from the middle section.

Table 75:

Button	Action
Review	Reviews all load collectors in the current load step. All loads in the load collectors are highlighted in the graphic area. The highlighted loads show through the solid mesh in performance graphics. If a load is defined with set, the underlying nodes or elements are highlighted. Right-click **Review** to clear the highlighted selections.
Text...	Reviews the current load step in a text window.
Synchronize	Updates the **Step Manager** with the current HyperMesh database. If you manually create, update, or delete load steps, load collectors, output blocks, groups, or entity sets from panels while the **Step Manager** is open, click **Synchronize** to update the **Step Manager** with the new changes.
Close	Closes the **Load Step** window and opens the **Step Manager** window.

The following options are available on the **Load Step** dialog tree:

Load Step

Title
Parameter
Analysis procedure
Load collector

- Boundary
- **Concentrated loads**
    - CLOAD
    - CFILM
    - CRADIATE
- **Distributed loads**
    - DLOAD
    - FILM
    - RADIATE
- **Surface Loads**
    - DSLOAD

        - SFILM
        - SRADIATE
- Temperature
- Inertia Relief
- Interface controls
    - Contact Pair
    - Surface Interaction
    - General Contact
    - Contact Controls
    - Clearance
    - Contact Interface
    - Model Change
    - Change Friction
    - Controls
- Output requests
    - ODB file
    - Result file (.fil)
    - Data file (.dat)
- Monitor
- Print
- File Format
- File Output
- Restart write
- Unsupported cards

Load Step: Load Collector Table

You can create, edit, review, rename, reorder, and delete load collectors from this table. You can also organize (copy or move) loads into load collectors and edit the load labels here.

Each load or constraint must belong to a load collector. Therefore, when you select load types from the tree, the load collector table appears in the middle section of the **Load Step** dialog.

The Load collector table contains a list of the load collectors with their corresponding display, color, and history status.

The Load collector table contains the following columns:

Table 76:

Column	Description
Status	The history status of the load collector. If the status is on, the load collector belongs to the current load step. This means, all loads in the load collector will be

Column	Description
	written under the current *STEP block. If the status is off, no loads from the load collector will be written under the current *STEP block.
Name	The name of the load collector. The load collector names are for internal use only. The Abaqus input file does not need them.
Display	The display on/off check boxes and color change buttons for the load collector. The color can be changed by clicking the color button and selecting a color from the menu.

Click on a load collector name to set it as the current load collector. All loads created from this point are placed into the selected load collector.

The Load Type: status bar (below the load collector table) shows all the load types present in the selected load collector. In addition, these load types are highlighted with bold font in the tree.

See Step Manager Dialog Environment for tips on navigating through the dialogs.

The Load collector table contains the following buttons:

Table 77:

Button	Action
New	Opens the **Create Load Collector** dialog in which you enter the name of the new load collector. Use the Same as option to create a load collector by copying attributes (except loads created from panels) from an existing load collector. Click **Create** to create the load collector and add it to the current load step.
Review	Reviews the selected load collector. All loads in the load collector are highlighted in the graphic area. The highlighted loads show through the solid mesh in performance graphics. If a load is defined with set, the underlying nodes or elements are highlighted. Right-click on the **Review** button to clear the highlighted selections.
Organize...	Opens the Organize panel where you can copy/move loads into different load collectors. When you have finished, click **return** to update the **Step Manager** with the new organization.
Label...	Opens the corresponding panel where you can turn on/off load labels or update the label size.

Button	Action
Delete	Opens the Delete panel where you can delete load collectors. When you are finished, click **return** to update the Load collector table.

Load Step: Title

Select **Title** to define a one line title or sub-heading for the load step.

This line appears under the *STEP keyword in the input file. The title is also used in the ODB file to identify the step. Check the **Step Heading** option to input the title and click **Update** to update the HyperMesh database.

Load Step: Parameter

Select **Parameter** to define the parameters of the *STEP card.

The supported parameters are: `Name`, `Amplitude`, `Extrapolation`, `Unsymmetric`, `Increment`, `Nlgeom`, and `Perturbation`.

Click **Update** to activate the parameters defined in the HyperMesh database.

Load Step: Analysis Procedure

Select **Analysis procedure** to define the analysis type of the load step.

The following analysis types with the corresponding parameters and data lines are supported:

Table 78:

Analysis Types	Parameter
*FREQUENCY	PROPERTY EVALUATION, EIGENSOLVER = {SUBSPACE, LANCZOS, AMS}, NORMALIZATION = {MASS, DISPLACEMENT}, RESIDUAL MODES, ACOUSTIC COUPLING, NUMBER INTERVAL, BIAS, USER BOUNDARIES
*BUCKLE	EIGENSOLVER = {SUBSPACE, LANCZOS}
*DYNAMIC (Standard)	HAFTOL, DIRECT, DIRECT=NO STOP, SUBSPACE, ADIABATIC, ALPHA, INITIAL, NOHAF
*DYNAMIC (Explicit)	EXPLICIT, SCALE FACTOR, ADIABATIC FIXED TIME INCREMENTATION, DIRECT USER CONTROL, ELEMENT BY ELEMENT IMPROVED DT METHOD = YES or NO
*HEAT TRANSFER	DELTMX, END = {PERIOD, SS}, STEADY STATE, MXDEM

Analysis Types	Parameter
*MODAL DYNAMIC	CONTINUE = YES or NO
*STATIC	ADIABATIC, FULLY PLASTIC, RIKS, STABALIZE, DIRECT, DIRECT=NO STOP, FACTOR, LONG TERM
*STEADY STATE DYNAMICS	DIRECT, SUBSPACE PROJECTION = {ALL FREQUENCIES, CONSTANT, EIGENFREQUENCY, PROPERTY CHANGE}, FREQUENCY SCALE, INTERVAL, REAL ONLY, DAMPING CHANGE, STIFFNESS CHANGE
*COUPLED TEMPERATURE-DISPLACEMENT	STEADY STATE, MAXIMUM TEMPERATURE CHANGE DELTMX
*DYNAMIC TEMPERATURE-DISPLACEMENT(Explicit)	TIME INCRIMINATION, IMPROVED DT METHOD, SCALE FACTOR

There are two tabs for each analysis type: Parameter and Dataline. The layout of the tabs change, based on the analysis types selected.

Click **Update** to activate the optional parameter or data item selection in the HyperMesh database.

Load Step: Boundary

In the **Boundary** dialog, define and edit the *BOUNDARY card.

To open this dialog, select **Boundary** from the tree and a load collector from the Load collector table.

Load Step: Boundary: Define Tab
In the Define tab, define *BOUNDARY cards on individual nodes or geometry (surfaces, points, lines). You can also define the boundary on node sets.

There are five different types of boundary conditions available:

Table 79:

Boundary Types	Abaqus Keyword
Default (disp)	*BOUNDARY
Velocity	*BOUNDARY, TYPE = VELOCITY
Acceleration	*BOUNDARY, TYPE = ACCELERATION
Temperature	*BOUNDARY on dof 11
Electric potential	*BOUNDARY on dof 9

It is recommended that you use only one boundary type per load collector in HyperMesh. If you need to use multiple boundary types in the same STEP, define each type in a separate load collector and add them to the same load step.

You can define a *BOUNDARY card on nodes/geometry or on node sets. For Define Boundary on:, the following options are available:

- Nodes or geometry
- Node sets

The layout of the Define tab changes, based on your selection.

Define Boundary On: Nodes or Geometry
Use the Define Boundary on: Nodes or geometry option to define various types of boundaries on individual nodes or geometry.

Boundaries created on nodes have a special graphical display in HyperMesh. Loads created on geometric entities like surfaces, lines or points are automatically mapped to FEA mesh on export. You can also map them using the Map Loads on Geometry button.

The Define tab for Define Boundary on: Nodes or geometry has the following buttons:

Table 80:

Button	Action
Define from 'Contraints' panels	Opens the Constraints panel to create/update boundary conditions. To create a boundary on nodes, go to the create subpanel, select the **nodes** button, pick the desired nodes from HyperMesh graphics, check the constrained degrees of freedoms, and click **create**. To create a boundary on geometry, go to the create subpanel, select **surfs**, **points**, or **lines** using the switch, pick the desired geometry from the HyperMesh graphics, check the **constrained degrees of freedom**, and click **create**. **Note:** • Loads created on geometric entities are automatically mapped to FEA mesh on export. You can also map them using the Map Loads on Geometry button. • An existing boundary can be updated from the updatesubpanel. • While you are in the constraints panel, press the **H** key to view panel-specific help. • When you are finished creating or updating boundary conditions, click **return** to update the **Step Manager** with the new loads.
Map Loads on Geometry	Opens the HyperMesh loads on geom panel to map loads on geometry to FEA mesh entities. Click **Map loads** to map all geometric loads in the current load collector to FEA entities.

Button	Action
	Note: • You can also pick other load collectors by clicking the **loadcols** button and map loads in all of them together. • While you are in the loads on geom panel, press the **H** key to view panel-specific help. • When you are finished, click **return** to update the **Step Manager** with the new loads.

Define Boundary On: Node Sets
The Define Boundary on: Node sets option defines various types of boundaries on node sets.

The node set names are used in the *BOUNDARY data lines instead of the individual nodes. Unlike Abaqus surfaces in HyperMesh, you can combine node sets with individual node IDs in the same *BOUNDARY card.

Note: HyperMesh does not graphically display loads created on sets. Therefore, when you review a load collector in the **Step Manager**, only loads created on individual entities are highlighted. For loads defined on sets, the underlying nodes or elements are highlighted.

This dialog contains a Node sets menu with a list of the existing node sets. It also has a table for data line input containing the following columns:

Table 81:

Column	Description
Nset	The name of the node sets. Node sets can only be added or deleted from this column using the → or ← buttons, respectively.
1st dof	The first degree of freedom. You can input any integer or any of the following types in this column: XSYMM, YSYMM, ZSYMM, ENCASTRE, PINNED, XASYMM, YASYMM, ZASYMM, NOWARP, NOOVAL, NODEFORM
Last dof	The last degree of freedom
Magnitude	The magnitude
Load ID	The ID of the load collector

The Define tab for Define Boundary on: Node sets contains the following buttons:

Table 82:

Button	Action
Review Set	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click the **Review** button to clear the review selections.
Create/Edit Set	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
→	Add the selected node set from the pull down menu to the data line table on the right.
←	Delete the selected node set from the data line table.
Review	Reviews the selected node set in the data line table. Right-click **Review** to clear the highlighted selections.
Update	Updates the HyperMesh database with the data lines defined in the table. By default, HyperMesh does not create a display for loads defined with sets.
Display/Review from panel	Opens the appropriate HyperMesh panel. Use the Review button to expand the loads and constraints on the sets for visualization purposes.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

Load Step: Boundary: Delete Tab
In the Delete tab, delete boundaries and other loads from HyperMesh.

There are three options:

Table 83:

Option	Description
All loads in current collector	The Delete button deletes all the loads from the current load collector.
All 'Boundary' in current collector	The Delete button deletes only *BOUNDARY loads from the current load collector.
By selection	The Pick Loads button opens the HyperMesh load selector panel. Pick the loads you want to delete and click **proceed**. The corresponding Reset button resets the selected loads. The Delete button deletes the selected loads.

Load Step: Boundary: Parameter Tab
In the Parameter tab, define optional parameters for the *BOUNDARY card.

The supported parameters are: `Amplitude`, `OP`, `Load Case`, `Fixed`, and `Region Type`.

Click **Update** to activate the optional parameter selection in the HyperMesh database.

Load Step: CLOAD

In the **CLOAD-Force** dialog, define the *CLOAD card for dofs 1 through 3 and the CLOAD-Moment for dofs 4 through 6.

In Abaqus, *CLOAD can have degrees of freedom (dof) 1 through 6. In HyperMesh, concentrated loads with dofs 1 through 3 are called force and those with dofs 4 through 6 are called moments. The force and moments are two separate entities that are defined from separate panels. Their graphical displays are also different. As a result, the **Step Manager** has two distinct tree options for *CLOAD: CLOAD-Force and CLOAD-Moment.

Open Concentrated loads in the tree, select **CLOAD-Force** or **CLOAD-Moment**, and a load collector from the Load collector table to open the corresponding dialog in the right-most section of the Load Step window.

It is recommended that you do not use both CLOAD–Force and CLOAD-Moment in the same load collector in HyperMesh. If you need to use both types of *CLOAD in the same STEP, define each type in a separate load collector and add them to the same load step.

Load Step: CLOAD: Define Tab
In the Define tab, define *CLOAD (force or moment) cards on individual nodes or geometry (points). You can also define the *CLOAD on node sets.

You can define CLOAD-Force/Moment on nodes/geometry or on node sets. For Define CLOAD-Force/Moment on:, the following options are available:

- Nodes or geometry
- Node sets

The layout of the Define tab changes based on your selection.

Define CLOAD-Force/Moment On: Nodes or Geometry
Use the Define CLOAD-Force/Moment on: Nodes or geometry option to define *CLOAD (force or moment) on individual nodes or geometry.

The concentrated loads created on nodes have special graphical display in HyperMesh. Loads created on geometric entities such as surfaces, lines, or points are automatically mapped to FEA mesh on export. They can also be mapped using the Map Loads on Geometry button.

The Define tab for Define CLOAD-Force/Moment on: Nodes or geometry contains the following buttons:

Table 84:

Button	Action
Define from 'Forces'/'Moments' Panel	Opens the HyperMesh Forces or Moments panel to create/update a CLOAD. To create a CLOAD on nodes, go to the create subpanel, select **nodes** using the switch, pick the desired nodes from HyperMesh graphics, select the global/local system, select a vector, input a magnitude, and click **create**. To create a CLOAD on geometry, go to the create subpanel, select **points** using the switch, pick the desired geometry from HyperMesh graphics, select the global/local system, select a vector, input a magnitude, and click **create**. **Note:** • Loads created on geometric entities are automatically mapped to FEA mesh on export. You can also map them using the Map Loads on Geometry button. • You can also update an existing force or moments from the update subpanel. • While you are in the forces or moments panel, press the **H** key to view panel-specific help. • When you finish creating or updating CLOAD, click **return** to update the **Step Manager** with the new loads.
Map Loads on geometry	Opens the HyperMesh Loads on Geom panel to map loads on geometry to FEA mesh entities. Click **Map loads** to map all geometric loads in the current load collector to FEA entities. **Note:** • You can also pick other load collectors by clicking on the **loadcols** button and map loads in all of them together. • While you are in the Loads on Geom panel, press the **H** key to view panel-specific help. • When you are finished, click **return** to update the **Step Manager** with the new loads.

Define CLOAD-Force/Moment On: Node Sets
The Define CLOAD-Force/Moment on: Node sets option defines CLOAD on node sets.

The node set names are used in the *CLOAD data lines instead of the individual nodes. Unlike Abaqus surfaces in HyperMesh, you can combine node sets with individual node IDs in the same *CLOAD card.

Note: Loads created on sets are not graphically displayed in HyperMesh. Therefore, when you review a load collector in the **Step Manager**, only loads created on individual entities are highlighted. For loads defined on sets, the underlying nodes or elements are highlighted.

This Define tab for Define CLOAD-Force/Moment on: Node sets includes a Node sets menu containing a list of existing node sets. A data line input table also appears on the Define tab. The table contains the following columns:

Table 85:

Column	**Description**
Nodeset	The name of the node sets. Node sets are added or removed using → or ←, respectively.
Comp_x	The component in x direction. The x-direction indicates dof 1 for force and dof 4 for moment.
Comp_y	The component in y direction. The y-direction indicates dof 2 for force and dof 5 for moment.
Comp_z	The component in z direction. The x-direction indicates dof 3 for force and dof 6 for moment.
Magnitude	The magnitude. This column is non-editable. The magnitude is calculated based on the Comp_x, Comp_y, and Comp_z defined for each node set when you click **Update**.
Load Id	The ID of the load collector.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

The Define tab for Define CLOAD-Force/Moment on: Node sets contains the following buttons:

Table 86:

Button	**Action**
Review Set	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click **Review** to clear the review selections.
Create/Edit Set..	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
Display/Review from panel	Opens the appropriate HyperMesh panel. Use the **review** button to expand the loads and constraints on the sets for visualization purposes.

Button	Action
→	Add the selected node set from the drop-down menu to the data line table on the right.
←	Delete the selected node set from the data line table.
Define by vector	Opens the HyperMeshvector selector panel. Pick a vector and click **proceed**. This vector is used to define the Comp_x, Comp_y, Comp_z, and Magnitude of the CLOAD for the selected node set.
Create/Edit vector	Opens the Vectors panel in HyperMesh. When you finish creating/editing the vector, click **return**.
Review	Creates special review forces or moments in HyperMesh graphics for the selected node set. These review forces or moments take into consideration the *TRANSFORM cards that may be associated with nodes in the node set. Right-click **Review** to clear the special review loads and highlighting.
Update	Updates the HyperMesh database with the data lines defined in the table. By default, HyperMesh does not display loads defined with sets.

Load Step: CLOAD: Delete Tab
In the Delete tab, delete *CLOAD and other loads.

There are three deletion options:

Table 87:

Option	Description
All loads in current collector	The Delete button deletes all the loads from the current load collector.
All 'CLOAD' in current collector	The Delete button deletes only *CLOAD loads from the current load collector.
By selection	The Pick Loads button opens the HyperMesh load selector panel. Pick the loads you want to delete and click **proceed**. The corresponding Reset button resets the selected loads. The Delete button deletes the selected loads.

Load Step: CLOAD: Parameter Tab
In the Parameter tab, define optional parameters for the *CLOAD card.

The supported parameters are: `Amplitude`, `OP`, `Load Case`, `Cyclic Mode`, `Follower`, and `Region Type`.

Click **Update** to activate the optional parameter selection in the HyperMesh database.

Load Step: CFILM

In the **CFILM** dialog, define the *CFILM card on node sets.

To open this dialog from the **Load Step** dialog, expand **Concentrated loads** from the tree, select **CFILM**, and then select a load collector from the Load Collector table.

Load Step: CFILM: Define Tab
In the Define tab, define a *CFILM card on node sets only.

HyperMesh does not support *CFILM on individual nodes.

Use the Define CFILM-Force/Moment on: Node sets option to define CFILM on node sets. The node set names are used in the *CFILM data lines instead of the individual nodes. Unlike Abaqus surfaces in HyperMesh, you can combine node sets with individual node IDs in the same *CFILM card.

> **Note:** Loads created on sets are not graphically displayed in HyperMesh. Therefore, when you review a load collector in the **Step Manager**, only loads created on individual entities are highlighted. For loads defined on sets, the underlying nodes or elements are highlighted.

This dialog has a node sets menu containing a list of the existing node sets. It also has a data line input table with the following columns:

Table 88:

Column	Description
Nodeset	The name of the node sets. Node sets are added or removed using → or ←, respectively.
Area	The area associated with the nodes.
Sink temp	Reference sink temperature.
Film coef	Reference film coefficient.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

The Define CFILM on: Node sets option has the following buttons:

Table 89:

Button	Action
Review Set	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click **Review** to clear the review selections.

Button	Action
Create/Edit Set..	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the node set list.
→	Add the selected node set from the drop-down menu to the data line table on the right.
←	Delete the selected node set from the data line table.
Review	Reviews the selected node set in the data line table. Right-click **Review** to clear the highlighted selections.
Update	Updates the HyperMesh database with the data lines defined in the table. By default, HyperMesh does not display loads defined with sets.

Load Step: CFILM: Delete Tab
In the Delete tab, delete *CFILM and other loads.

There are three deletion options:

Table 90:

Option	Description
All loads in current collector	The Delete button deletes all the loads from the current load collector.
All 'CFILM' in current collector	The Delete button deletes only *CFILM loads from the current load collector.
By selection	The Pick Loads button opens the HyperMesh load selector panel. Pick the loads you want to delete and click **proceed**. The corresponding Reset button resets the selected loads. The Delete button deletes the selected loads.

Load Step: CFILM: Parameter Tab
In the Parameter tab, define optional parameters for the *CFILM card.

The supported parameters are: `Amplitude`, `Film Amplitude`, `OP`, and `Region Type`.

Click **Update** to activate the optional parameter selection in the HyperMesh database.

Load Step: CRADIATE

In the **CRADIATE** dialog, define the *CRADIATE cards on node sets.

To open this dialog from the **Load Step** dialog, select **CRADIATE** from the tree and a load collector from the Load collector table.

Load Step: CRADIATE: Define Tab
In the Define tab, define a *CRADIATE card on node sets only.

*CRADIATE on individual nodes is not supported.

Use the Define CRADIATE on: Node sets option to define *CRADIATE on node sets.

> **Note:** Loads created on sets are not graphically displayed. Therefore, when you review a load collector in the **Step Manager**, only loads created on individual entities are highlighted. For loads defined on sets, the underlying nodes or elements are highlighted.

This dialog has a node sets drop down menu containing a list of the existing node sets. It also has a data line input table with the following columns:

Table 91:

Column	Description
Nodeset	The name of the node sets. Node sets are added or removed using → or ←, respectively.
Area	The area associated with the nodes.
Sink temp	Ambient reference temperature.
Emissivity	Surface emissivity.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

The Define CRADIATE on: Node sets option has the following buttons:

Table 92:

Button	Action
Review/Reset Set	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click **Review** to clear the review selections.
Create/Edit Set..	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the node set list.

Button	Action
→	Add the selected node set from the drop-down menu to the data line table on the right.
←	Delete the selected node set from the data line table.
Review	Reviews the selected node set in the data line table. Right-click **Review** to clear the highlighted selections.
Update	Updates the HyperMesh database with the data lines defined in the table. By default, HyperMesh does not create a display for loads defined with sets.

Load Step: CRADIATE: Delete Tab
In the Delete tab, delete *CRADIATE and other loads.

There are three deletion options:

Table 93:

Option	Description
All loads in current collector	The Delete button deletes all the loads from the current load collector.
All 'CFILM' in current collector	The Delete button deletes only *CRADIATE loads from the current load collector.
By selection	The Pick Loads button opens the HyperMesh load selector panel. Pick the loads you want to delete and click **proceed**. The corresponding Reset button resets the selected loads. The Delete button deletes the selected loads.

Load Step: CRADIATE: Parameter Tab
In the Parameter tab, define optional parameters for the *CRADIATE card.

The supported parameters include: `Amplitude curve`, `OP`, and `Region Type`.

Click **Update** to activate the optional parameter selection in the HyperMesh database.

Load Step: DLOAD

In the **DLOAD** dialog, define the *DLOAD cards on individual elements or geometry (surfaces). You can also define the DLOAD on element sets.

To open the dialog in the **Load Step** window, expand **Distributed loads** in the tree, select **DLOAD**, and select a load collector from the Load collector table.

Load Step: DLOAD: Define Tab
In the Define tab, define *DLOAD cards on individual elements or geometry (surfaces) as well as on element sets.

There are seven different DLOAD types available: default (Pressure), centrifugal, rotary acceleration, gravity, pressure in pipe/elbow, hydro pressure, and hydro pressure in pipe/elbow.

Only default (Pressure) type DLOAD can be created on an individual element or geometry in HyperMesh. The other types are available only for element sets.

It is recommended that you use only one type of DLOAD in a load collector in HyperMesh. If you need to use multiple types of DLOAD in the same STEP, define each type in a separate load collector and add them to the same load step.

You can define DLOAD on elements, geometry, or element sets. For Define DLOAD on:, the following options are available:

- Elements or geometry
- Element sets

The layout of the Define tab changes, based on your selection.

Define DLOAD On: Elements or Geometry
Use the Define DLOAD on: Elements or Geometry option to define the default (pressure) type of DLOAD on individual elements or geometric surfaces.

Pressure loads created on elements have special graphical display in HyperMesh. Loads created on geometric entities such as surfaces are automatically mapped to FEA mesh on export. You can also map them using the Map Loads on Geometry button.

The Define tab for Define DLOAD on: Elements or Geometry contains the following buttons:

Table 94:

Button	Action
Define from Pressures panel	Opens the Pressures panel to create/update DLOAD. To create a pressure on elements, go to the create subpanel, select the **elems** button, pick the desired elements from the HyperMesh graphics, select nodes using the switch, pick two or three nodes from a face of a selected element, input the magnitude, and click **create**. To create a pressure on geometry, go to the create subpanel, select the **surfs** option from the toggle, pick the desired geometry from the HyperMesh graphics, input the magnitude, and click **create**.

Button	Action
	Note: • Loads created on geometric entities are automatically mapped to FEA mesh on export. You can also map them using the Map Loads on Geometry button. • You can also update an existing DLOAD from the update subpanel. • While you are in the Pressure panel, press the **H** key to view panel-specific help. • When you finish creating or updating boundary conditions, click **return** and the **Step Manager** is updated with the new loads.
Map Loads on Geometry	Opens the Loads on Geom panel to map loads on geometry to FEA mesh entities. Click **Map loads** to map all geometric loads in the current load collector to FEA entities. **Note:** • You can also pick other load collectors by clicking on the **loadcols** and **map loads** in all of them together. • While you are in the Loads on Geom panel, press the **H** key to view panel-specific help. • When you finish, click **return** to update the **Step Manager** with the new loads.

Define DLOAD On: Element Sets
Use the Define DLOAD on: Element sets option to define various DLOAD types on element sets.

The element set names are used in the *DLOAD data lines instead of the individual elements. Unlike Abaqus surfaces in HyperMesh, you can combine element sets with individual element IDs in the same *DLOAD card.

> **Note:** There is no graphical display in HyperMesh for loads created on sets. Therefore, when you review a load collector in the **Step Manager**, only loads created on individual entities are highlighted. For loads defined on sets, the underlying nodes or elements are highlighted.

This dialog contains a element sets menu with a list of the existing element sets. There are two types of elsets in HyperMesh: components and entity sets. The Abaqus elsets that are linked to sectional property cards, such as *SOLID SECTION and *SHELL SECTION, become components in HyperMesh.

Others become entity sets. To differentiate between these two types, there is a divider line "- - - - -" in the elset list that pops up if you click the **element sets** menu. The elsets listed below the divider line are components.

This dialog also contains a table for data line input. The table changes depending on the DLOAD type selected. The table contains the following columns.

For Default (Pressure) type:

Table 95:

Column	Description
Elset	The name of the element sets. Element sets are added and deleted in this column using → or ←, respectively.
Label	The labels of pressure load. The available labels are: P1, P2, P3, P4, P5, P6, and P.
Magnitude	The magnitude of the load.
Load Id	The ID of the load collector.

For Centrifugal type:

Table 96:

Column	Description
Elset	The name of the element sets. Element sets are added and deleted in this column using → or ←, respectively.
Label	The labels of centrifugal loads and Coriolis forces. The available labels are: CENTRIF, CENT, and CORIO.
Magnitude	The magnitude of the load.
Coord1	Coordinate 1 of a point on the axis of rotation.
Coord2	Coordinate 2 of a point on the axis of rotation.
Coord3	Coordinate 3 of a point on the axis of rotation.
DirCos1	1-component of the direction cosine of the axis of rotation.
DirCos2	2-component of the direction cosine of the axis of rotation.
DirCos3	3-component of the direction cosine of the axis of rotation.
Load Id	The ID of the load collector.

For Rotary acceleration type:

Table 97:

Column	Description
Elset	The name of the element sets. Element sets are added and deleted in this column using → or ←, respectively.
Label	The labels of the DLOAD type. The available labels are: ROTA.
Magnitude	The magnitude of the load.
Coord1	Coordinate 1 of a point on the axis of rotary acceleration.
Coord2	Coordinate 2 of a point on the axis of rotary acceleration.
Coord3	Coordinate 3 of a point on the axis of rotary acceleration.
DirCos1	1-component of the direction cosine of the axis of rotary acceleration.
DirCos2	2-component of the direction cosine of the axis of rotary acceleration.
DirCos3	3-component of the direction cosine of the axis of rotary acceleration.
Load Id	The ID of the load collector.

For Gravity type:

Table 98:

Column	Description
Elset	The name of the element sets. Element sets are added and deleted in this column using → or ←, respectively.
Label	The labels of the DLOAD type. The available labels are: GRAV.
Magnitude	The magnitude of the load.
Comp1	1-component of the gravity vector.
Comp2	2-component of the gravity vector.
Comp3	3-component of the gravity vector.
Load Id	The ID of the load collector.

For pressure in pipe/elbow type:

Table 99:

Column	Description
Elset	The name of the element sets. Element sets are added and deleted in this column using → or ←, respectively.
Label	The labels of the pressure in pipe/elbow elements. The available labels are: PE, PI, PENU, and PINU.
Magnitude	The magnitude of the load.
Diameter	The effective inner or outer diameter.
Condition	The end loading condition: CLOSE (default) or OPEN.
Load Id	The ID of the load collector.

For hydro pressure type:

Table 100:

Column	Description
Elset	The name of the element sets. Element sets are added and deleted in this column using → or ←, respectively.
Label	The labels of the hydrostatic pressure. The available labels are: HP.
Magnitude	The magnitude of the load.
Zero press	Z-coordinate of zero pressure level in three-dimensional or axisymmetric cases; Y-coordinate of zero pressure level in two-dimensional cases.
Press point	Z-coordinate of the point at which the pressure is defined in three-dimensional or axisymmetric cases; Y-coordinate of the point at which the pressure is defined in two-dimensional cases.
Load Id	The ID of the load collector.

For hydro pressure in pipe/elbow type:

Table 101:

Column	Description
Elset	The name of the element sets. Element sets are added and deleted in this column using → or ←, respectively.
Label	The labels of the hydrostatic pressure in pipe.elbow elements. The available labels are: HPE, and HPI.
Magnitude	The magnitude of the load.
Zero press	Z-coordinate of zero pressure level in three-dimensional or axisymmetric cases; Y-coordinate of zero pressure level in two-dimensional cases.
Press point	Z-coordinate of the point at which the pressure is defined in three-dimensional or axisymmetric cases; Y-coordinate of the point at which the pressure is defined in two-dimensional cases.
Diameter	The effective inner or outer diameter.
Condition	The end loading condition: CLOSE (default) or OPEN.
Load Id	The ID of the load collector.

The Define tab for Define DLOAD on: Element sets contains the following buttons:

Table 102:

Button	Action
Review Set	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click **Review** to clear the review selections.
Create/Edit Set..	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
Display/Review from panel	Opens the appropriate HyperMesh panel. Use the **review** button to expand the loads and constraints on the sets for visualization purposes.
→	Add the selected node set from the drop-down menu to the data line table on the right.
←	Delete the selected node set from the data line table.
Show Faces	This option is only shown for Default (pressure) type. It is mainly used to review the face identifiers of elements in the selected set. It creates a

Button	Action
	temporary skin of the selected elset, opens the Element Selector panel, from which you can select face elements from this skin. When you return from the element selector panel, the selected faces display color-coded face identifier tags. In performance graphics, these tags are sometimes blocked by the solid mesh. You may need to rotate the model a little to view the tags. Right-click the **Show faces** button to clear the face review.
Define by vector	This option is only shown for Gravity type. It opens the HyperMesh vector selector panel. Pick a vector and click **proceed**. This vector is used to define the Comp1, Comp2, Comp3, and Magnitude of the gravity load for the selected element set.
Create/Edit vector..	This option is only shown for Gravity type. Opens the Vectors panel in HyperMesh. When you finish creating/editing the vector, click **return**.
Review	Creates special review forces or moments in HyperMesh graphics for the selected node set. These review forces or moments take into consideration the *TRANSFORM cards that may be associated with nodes in the node set. Right-click **Review** to clear the special review loads and highlighting.
Update	Updates the HyperMesh database with the data lines defined in the table. By default, HyperMesh does not create a display for loads defined with sets.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

Load Step: DLOAD: Delete Tab
In the Delete tab, delete *DLOAD and other loads.

There are three deletion options:

Table 103:

Option	Description
All loads in current collector	The Delete button deletes all the loads from the current load collector.
All 'Distributed loads' in current collector	The Delete button deletes all distributed (*DLOAD, *FILM) loads from the current load collector.
By selection	The Pick Loads button opens the HyperMesh load selector panel. Pick the loads you want to delete and click **proceed**. The corresponding Reset button resets the selected loads. The Delete button deletes the selected loads.

Load Step: DLOAD: Parameter Tab
In the Parameter tab, define optional parameters for the *DLOAD card.

The supported parameters include: `Amplitude`, `OP`, `Load Case`, `Cyclic Mode`, and `Region Type`.

Click **Update** to activate the optional parameter selection in the HyperMesh database.

Load Step: FILM

In the **FILM** dialog, define the *FILM cards on individual elements or geometry (surfaces). You can also define the FILM on element sets.

To open this dialog in the **Load Step** window, select **FILM** from the tree and a load collector from the Load collector table.

Load Step: FILM: Define Tab
In the Define tab, define *FILM cards on individual elements or geometry (surfaces) as well as on element sets.

For Define FILM on, the following options are available:

- Elements or geometry
- Element sets

The layout of the Define tab changes, based on your selection.

Define FILM On: Elements or Geometry
Use the Define FILM on: Elements or geometry option to define FILM on individual elements or geometric surfaces.

FILM loads created on elements have special graphical display in HyperMesh. Loads created on geometric entities like surfaces are automatically mapped to FEA mesh on export. You can also map them using the Map Loads on Geometry button.

The Define tab for Define FILM on: Elements or geometry has the following buttons:

Table 104:

Button	Action
Define from Pressures panel	Opens the Pressures panel, from which you can create and update FILM. It only allows you to define the reference sink temperature. The film coefficient needs to be defined separately. To create a FILM on elements, go to the create subpanel, select **elems** using the switch, pick the desired elements from the HyperMesh graphics, click **nodes**, pick two or three nodes from a face of a selected element, input the magnitude (sink temperature), and click **create**. To create a pressure on geometry, go to the create subpanel, select **surfs** using the switch, pick the desired geometry from the HyperMesh graphics, input the magnitude (sink temperature), and click **create**.

Button	Action
	Note: • Loads created on geometric entities are automatically mapped to FEA mesh on export. You can also map them using the Map Loads on Geometry button. • You can also update an existing FILM from the update subpanel. • While you are in the Pressure panel, press the **H** key to view panel-specific help. • When you are finished creating or updating boundary conditions, click **return** and the **Step Manager** will be updated with the new loads.
Map Loads on Geometry	Opens the HyperMesh Loads on Geom panel to map loads on geometry to FEA mesh entities. Click the **Map loads** button to map all geometric loads in the current load collector to FEA entities. **Note:** • You can also pick other load collectors by clicking the **loadcols** button and map loads in all of them together. • While you are in the Loads on Geom panel, press the **H** key to view panel-specific help. • When you are done, click **return** and the **Step Manager** will be updated with the new loads.

Define FILM On: Element Sets

Use the Define FILM on: Element sets option to define the FILM load on element sets.

The element set names are used in the *FILM data lines instead of the individual elements. Unlike Abaqus surfaces in HyperMesh, you can combine element sets with individual element IDs in the same *FILM card.

> **Note:** There is no graphical display in HyperMesh for loads created on sets. Therefore, when you review a load collector in the **Step Manager**, only loads created on individual entities are highlighted. For loads defined on sets, the underlying nodes or elements are highlighted.

This dialog contains an element sets menu containing a list of the existing element sets. There are two types of elsets in HyperMesh: components and entity sets. The Abaqus elsets that are linked to sectional property cards, such as *SOLID SECTION and *SHELL SECTION, become components in HyperMesh. Others become entity sets. To differentiate between these two types, there is a divider line

"- - - - -" in the elset list that pops up if you click the **element sets** menu. The elsets listed below the divider line are components.

This dialog also contains a table for data line input, which contains the following columns:

Table 105:

Column	Description
Elset	The name of the element sets. Element sets are added and deleted in this column using → or ←, respectively.
Label	The labels of FILM load. The available labels are: F1, F2, F3, F4, F5, F6, FPOS, and FNEG.
Sink Temp	The reference sink temperature.
Film coeff	The reference film coefficient.
Load Id	The ID of the load collector.

The Define tab for Define FILM on: Element sets contains the following buttons:

Table 106:

Button	Action
Review Set	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click **Review** to clear the review selections.
Create/Edit Set..	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
Display/Review from panel	Opens the appropriate HyperMesh panel. Use the **review** button to expand the loads and constraints on the sets for visualization purposes.
→	Add the selected node set from the drop-down menu to the data line table on the right.
←	Delete the selected node set from the data line table.
Show Faces	Used mainly to review the face identifiers of elements in the selected set. It creates a temporary skin of the selected elset, opens the Element Selector panel, from which you can select face elements from this skin. When you return from the element selector panel, the selected faces will display color-coded face identifier tags. In performance graphics, these tags are sometimes blocked by the solid mesh. You may need to rotate the model a little to view the tags.

Button	Action
	Right-click **Show faces** to clear the face review.
Review	Creates special review FILM loads in HyperMesh graphics for the selected set. Right-click **Review** to clear the special review loads and highlighting.
Update	Updates the HyperMesh database with the data lines defined in the table. By default, HyperMesh does not create a display for loads defined with sets.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

Load Step: FILM: Delete Tab
In the Delete tab, delete *FILM and other loads.

There are three deletion options:

Table 107:

Option	Description
All loads in current collector	The Delete button deletes all the loads from the current load collector.
All 'Distributed loads' in current collector	The Delete button deletes all distributed (*DLOAD, *FILM) loads from the current load collector.
By selection	The Pick Loads button opens the HyperMesh load selector panel. Pick the loads you want to delete and click **proceed**. The corresponding Reset button resets the selected loads. The Delete button deletes the selected loads.

Load Step: FILM: Parameter Tab
In the Parameter tab, define optional parameters for the *FILM card.

The supported parameters include: *`Amplitude`*, *`Film Amplitude`*, *`OP`*, and *`Region Type`*.

Click **Update** to activate the optional parameter selection in the HyperMesh database.

Modify the FILM Coefficient
Options for defining film coefficient on FILM created on individual elements or geometry.

By selection

The Pick Loads button opens the HyperMesh load selector panel. Pick the FILM loads to which you want to assign film coefficient, and click **proceed**.
The corresponding Reset button resets the selected loads.
The Update button assigns the value specified in the Film coefficient: text box to all selected FILM loads.

All FILMS in current collector

The Update button assigns the value specified in the Film coefficient: text box to all the FILM loads in the current load collector.

Load Step: RADIATE

In the **RADIATE** dialog, define the *RADIATE cards on individual elements or geometry (surfaces). You can also define the RADIATE on element sets.

To open this dialog in the **Load Step** window, select **RADIATE** from the tree and a load collector from the Load collector table.

Load Step: RADIATE: Define Tab
In the Define tab, define *RADIATE cards on individual elements or geometry (surfaces) as well as on element sets.

For Define RADIATE on:, the following options are available:

- Elements or geometry
- Element sets

The layout of the Define tab changes, based on your selection.

Define RADIATE On: Elements or Geometry
Use the Define RADIATE on: Elements or geometry option to define RADIATE on individual elements or geometric surfaces.

RADIATE loads created on elements have special graphical display in HyperMesh. Loads created on geometric entities like surfaces are automatically mapped to FEA mesh on export. You can also map them using the Map Loads on Geometry button.

The Define tab for Define RADIATE on: Elements or geometry has the following buttons:

Table 108:

Button	**Action**
Define from Pressures panel	Opens the Pressures panel, from which you can create and update RADIATE. It only allows you to define the reference sink temperature. To create a RADIATE on elements, go to the create subpanel, select **elems** using the switch, pick the desired elements from the HyperMesh graphics, click **nodes**, pick two or three nodes from a face of a selected element, input the magnitude (sink temperature), and click **create**. To create a pressure on geometry, go to the create subpanel, select **surfs** using the switch, pick the desired geometry from the HyperMesh graphics, input the magnitude (sink temperature), and click **create**.

Button	Action
	Note: • Loads created on geometric entities are automatically mapped to FEA mesh on export. You can also map them using the Map Loads on Geometry button. • You can also update an existing RADIATE from the update subpanel. • While you are in the Pressure panel, press the **H** key to view panel-specific help. • When you are finished creating or updating boundary conditions, click **return** and the **Step Manager** will be updated with the new loads.
Map Loads on Geometry	Opens the HyperMesh Loads on Geom panel to map loads on geometry to FEA mesh entities. Click the **Map loads** button to map all geometric loads in the current load collector to FEA entities. **Note:** • You can also pick other load collectors by clicking the **loadcols** button and map loads in all of them together. • While you are in the Loads on Geom panel, press the **H** key to view panel-specific help. • When you are done, click **return** and the **Step Manager** will be updated with the new loads.

Define RADIATE On: Element Sets

Use the Define RADIATE on: Element sets option to define the radiation conditions on element sets. It is possible to apply RADIATE on element sets and components.

> **Note:** There is no graphical display in HyperMesh for loads created on sets. Therefore, when you review a load collector in the Step Manager, only loads created on individual entities are highlighted. For loads defined on sets, the underlying nodes or elements are highlighted.

This dialog contains a drop down menu containing a list of the existing element sets or components. There are two types of element sets in HyperMesh: components and entity sets. The Abaqus element sets that are linked to sectional property cards, such as *SOLID SECTION and *SHELL SECTION, become components in HyperMesh upon import. To differentiate between these two types, there is a divider line "- - - - -" in the element set list that pops up if you open up the extended entity selection ([...] – button). The elsets listed below the divider line are components.

The dialog also contains a table for data line input including the following columns:

Table 109:

Column	Description
Elset	The name of the element sets. Element sets are added and deleted in this column using → or ←, respectively.
Label	The labels of RADIATE load. The available labels are: R1, R2, R3, R4, R5, R6, RPOS, and RNEG.
Ref temp	The reference ambient temperature.
Emissivity	Surface emissivity
Load Id	The ID of the load collector.

The Define tab for Define RADIATE on: Element sets contains the following buttons:

Table 110:

Button	Action
Review Set	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click **Review** to clear the review selections.
Create/Edit Set..	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
Display/Review from panel	Opens the appropriate HyperMesh panel. Use the **review** button to expand the loads and constraints on the sets for visualization purposes.
→	Add the selected node set from the drop-down menu to the data line table on the right.
←	Delete the selected node set from the data line table.
Show Faces	Used mainly to review the face identifiers of elements in the selected set. It creates a temporary skin of the selected elset, opens the Element Selector panel, from which you can select face elements from this skin. When you return from the element selector panel, the selected faces will display color-coded face identifier tags. In performance graphics, these tags are sometimes blocked by the solid mesh. You may need to rotate the model a little to view the tags. Right-click **Show faces** to clear the face review.

Button	Action
Review	Creates special review RADIATE loads in HyperMesh graphics for the selected set. Right-click **Review** to clear the special review loads and highlighting.
Update	Updates the HyperMesh database with the data lines defined in the table. By default, HyperMesh does not create a display for loads defined with sets.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

Load Step: RADIATE: Delete Tab
In the Delete tab, delete *RADIATE and other loads.

There are three deletion options:

Table 111:

Option	Description
All loads in current collector	The Delete button deletes all the loads from the current load collector.
All 'Distributed loads' in current collector	The Delete button deletes all distributed (*DLOAD, *RADIATE) loads from the current load collector.
By selection	The Pick Loads button opens the HyperMesh load selector panel. Pick the loads you want to delete and click **proceed**. The corresponding Reset button resets the selected loads. The Delete button deletes the selected loads.

Load Step: RADIATE: Parameter Tab
In the Parameter tab, define optional parameters for the *RADIATE card.

The supported parameters include: `Amplitude curve`, `OP`, and `Region Type`.

Click **Update** to activate the optional parameter selection in the HyperMesh database.

Modify the Emissivity
Options for defining the emissivity for a RADIATE card created on individual elements or geometry.

By selection

The Pick loads button opens the HyperMesh Load Selector panel. Pick the RADIATE loads to which you want to assign emissivity, and click **proceed**.
The corresponding Reset button resets the selected loads.
The Update button assigns the value specified in the Emissivity: field to all selected RADIATE loads.

All RADIATE in current collector

The Update button assigns the value specified in the Emissivity: text box to all the RADIATE loads in the current load collector.

In a similar way, for shell elements, it is possible to change the element face (RPOS or RNEG) on which the RADIATE load is applied.

Load Step: DSLOAD

In the **DSLOAD** dialog, define the *DSLOAD card on Abaqus surfaces (*SURFACE).

To open the dialog in the **Load Step** window, select **DSLOAD** from the tree and a Load collector from the table.

You can use Abaqus Contact Manager to create Abaqus surfaces.

Load Step: DSLOAD: Define Tab
In the Define tab, define *DSLOAD card on Abaqus surfaces (*SURFACE).

There are two types of DSLOAD available: Pressure and hydro pressure.

It is recommended that you use only one type of DSLOAD per load collector in HyperMesh. If you need to use multiple types of DSLOAD in the same STEP, define each type in a separate load collector and add them to the same load step.

> **Note:** There is no graphical display in HyperMesh for loads created on Abaqus SURFACEs. Therefore, when you review a load collector in the **Step Manager**, only loads created on individual entities are highlighted. For loads defined on SURFACE, the underlying SURFACE elements are highlighted.

This dialog also contains a table for data line input. The table changes depending on the DSLOAD type selected.

The table columns for each DSLOAD type are listed below:

Default (Pressure) type

Table 112:

Column	Description
Surface	The name of the Abaquss surface. Surfaces are added and deleted in this column using → or ←, respectively.
Label	The labels of pressure load. The available labels are P, PNU, and VP.
Magnitude	The magnitude of the load.

Hydro pressure type

Table 113:

Column	Description
Surface	The name of the Abaqus surface. Surfaces are added and deleted in this column using → or ←, respectively.
Label	The labels of the hydrostatic pressure. The available label is HP.
Magnitude	The magnitude of the load.
Zero press	Z-coordinate of zero pressure level in three-dimensional or axisymmetric cases; Y-coordinate of zero pressure level in two-dimensional cases.
Press point	Z-coordinate of the point at which the pressure is defined in three-dimensional or axisymmetric cases; Y-coordinate of the point at which the pressure is defined in two-dimensional cases.

The DSLOAD option has the following buttons:

Table 114:

Button	Action
Review Surfaces	Reviews the selected surface by highlighting them in the HyperMesh graphics. Right-click **Review** to clear the review selections.
Create/Edit Surface..	Opens a message with information about surface definition in HyperMesh.
→	Add the selected surface from the drop-down menu to the data line table on the right.
←	Delete the selected surface from the data line table.
Review	Reviews the selected surface in the data line table. Right-click **Review** to clear the highlighted selections.
Update	Updates the HyperMesh database with the data lines defined in the table. By default, HyperMesh does not create a display for loads defined with sets.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

Load Step: DSLOAD: Delete Tab

In the Delete tab, delete *DSLOAD and other loads.

There are three deletion options:

Table 115:

Option	Description
All loads in current collector	The Delete button deletes all the loads from the current load collector.
All 'DSLOAD' in current collector	The Delete button deletes only *DSLOAD loads from the current load collector.
By selection	The Pick Loads button opens the HyperMesh load selector panel. Pick the loads you want to delete and click **proceed**. The corresponding Reset button resets the selected loads. The Delete button deletes the selected loads.

Load Step: DSLOAD: Parameter Tab
In the Parameter tab, define optional parameters for the *DSLOAD card.

The supported parameters include: `Amplitude`, `OP`, `Load Case`, `Cyclic Mode` and `Region Type`.

Click **Update** to activate the optional parameter selection in the HyperMesh database.

Load Step: SFILM

In the **SFILM** dialog, define the *SFILM card on Abaqus surfaces (*SURFACE).

To open the dialog in the **Load Step** window, select **SFILM** from the tree and a load collector from the Load collector table.

You can use the Abaqus Contact Manager to create Abaqus surfaces.

Load Step: SFILM: Define Tab
In the Define tab, define the *SFILM card on Abaqus surfaces (*SURFACE).

> **Note:** There is no graphical display in HyperMesh for loads created on Abaqus SURFACEs. Therefore, when you review a load collector in the **Step Manager**, only loads created on individual entities are highlighted. For loads defined on SURFACE, the underlying SURFACE elements are highlighted.

This dialog contains a Surface menu containing a list of the existing Abaqus surfaces. It also has a table for data line input. The table contains the following columns:

Table 116:

Column	Description
Surface	The name of the Abaqus surface. Surfaces are added and deleted in this column using → or ←, respectively.
Label	The SFILM labels. The available labels are F and FNU.
Sink temp	Reference sink temperature.
Fillm coef	Reference film coefficient.

The SFILM option has the following buttons:

Table 117:

Button	Action
Review/Reset Surfaces	Reviews the selected surface by highlighting them in the HyperMesh graphics. Right-click **Review** to clear the review selections.
Create/Edit Surface..	Opens a message with information about surface definition in HyperMesh.
→	Add the selected surface from the drop-down menu to the data line table on the right.
←	Delete the selected surface from the data line table.
Review/Reset	Reviews the selected surface in the data line table. Right-click **Review** to clear the highlighted selections.
Update	Updates the HyperMesh database with the data lines defined in the table. By default, HyperMesh does not create a display for loads defined with sets.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

Load Step: SFILM: Delete Tab

In the Delete tab, delete *SFILM and other loads.

There are three deletion options:

Table 118:

Option	Description
All loads in current collector	The Delete button deletes all the loads from the current load collector.
All 'SFILM' in current collector	The Delete button deletes only *SFILM loads from the current load collector.
By selection	The Pick Loads button opens the HyperMesh load selector panel. Pick the loads you want to delete and click **proceed**. The corresponding Reset button resets the selected loads. The Delete button deletes the selected loads.

Load Step: SFILM: Parameter Tab
In the Parameter tab, define optional parameters for the *SFILM card.

The supported parameters include: `Amplitude`, `Film Amplitude`, `OP`, and `Region Type`.

Click `Update` to activate the optional parameter selection in the HyperMesh database.

Load Step: SRADIATE

In the **SRADIATE** dialog, define the *SFILM card on Abaqus surfaces (*SURFACE).

To open the dialog in the **Load Step** window, select **SRADIATE** from the tree and a load collector from the Load collector table.

You can use the Abaqus Contact Manager to create Abaqus surfaces.

Load Step: SRADIATE: Define Tab
In the Define tab, define *SRADIATE card on Abaqus surfaces (*SURFACE).

> **Note:** There is no graphical display in HyperMesh for loads created on Abaqus SURFACEs. Therefore, when you review a load collector in the **Step Manager**, only loads created on individual entities are highlighted. For loads defined on SURFACE, the underlying SURFACE elements are highlighted.

This dialog contains a Surface menu containing a list of the existing Abaqus surfaces. It also has a table for data line input. The table contains the following columns:

Table 119:

Column	Description
Surface	The name of the Abaqus surface. Surfaces are added and deleted in this column using → or ←, respectively.
Label	The *SRADIATE labels.
Ref temp	Ambient reference temperature.
Emissivity	Surface emissivity.

The SRADIATE option has the following buttons:

Table 120:

Button	Action
Review/Reset Surfaces	Reviews the selected surface by highlighting them in the HyperMesh graphics. Right-click **Review** to clear the review selections.
Create/Edit Surface..	Opens a message with information about surface definition in HyperMesh.
→	Add the selected surface from the drop-down menu to the data line table on the right.
←	Delete the selected surface from the data line table.
Review/Reset	Reviews the selected surface in the data line table. Right-click **Review** to clear the highlighted selections.
Update	Updates the HyperMesh database with the data lines defined in the table. By default, HyperMesh does not create a display for loads defined with sets.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

Load Step: SRADIATE: Delete Tab

In the Delete tab, delete *SRADIATE and other loads.

There are three deletion options:

Table 121:

Option	Description
All loads in current collector	The Delete button deletes all the loads from the current load collector.
All 'SRADIATE' in current collector	The Delete button deletes only *SRADIATE loads from the current load collector.
By selection	The Pick Loads button opens the HyperMesh load selector panel. Pick the loads you want to delete and click **proceed**. The corresponding Reset button resets the selected loads. The Delete button deletes the selected loads.

Load Step: SRADIATE: Parameter Tab
In the Parameter tab, define optional parameters for the *SRADIATE card.

The supported parameters include: `Amplitude curve`, `OP`, and `Region Type`.

Click **Update** to activate the optional parameter selection in the HyperMesh database.

Load Step: Temperature

In the **Temperature** dialog, define and edit the *TEMPERATURE card.

To open the dialog in the **Load Step** window, select **Temperature** from the tree and a load collector from the Load collector table.

Load Step: Temperature: Define Tab
In the Define tab, define *TEMPERATURE cards on individual nodes or geometry (surfaces, points, lines). You can also define the temperature on node sets.

It is recommended that you do not use *TEMPERATURE and *BOUNDARY with dof 11 together in a load collector in =HyperMesh. If you need to use multiple types of temperature in the same STEP, define each type in a separate load collector and add them to the same load step.

You can define *TEMPERATURE on nodes, geometry, or node sets. For Define Temperature on:, the following options are available:

- Nodes or geometry
- Node sets

The layout of the Define tab changes, based on your selection.

Define Temperature On: Nodes or Geometry

Use the Define Temperature on: Nodes or geometry option to define temperature on individual nodes or geometry.

Temperatures created on nodes have special graphical display in HyperMesh. Loads created on geometric entities like surfaces, lines or points are automatically mapped to FEA mesh on export. You can also map them using the Map Loads on Geometry button.

The Define tab for Define Temperature on: Nodes or geometry contains the following buttons:

Table 122:

<table>
<tr><th>Button</th><th>Action</th></tr>
<tr><td>Define from 'Forces'/'Moments' Panel</td><td>Opens the Temperature panel to create/update temperature.

To create a temperature on nodes, go to the create subpanel, select nodes using the switch, input a value, and click create.

To create a temperature on geometry, go to the create subpanel, select surfs, points, or lines using the switch, input a value, and click create.

Note:
Loads created on geometric entities are automatically mapped to FEA mesh on export. You can also map them using the Map Loads on Geometry button.
You can also update an existing force or moments from the update subpanel.
While you are in the temperature panel, press the H key to view panel-specific help.
When you finish creating or updating temperature, click return to update the Step Manager with the new loads.</td></tr>
<tr><td>Map Loads on geometry</td><td>Opens the Loads on Geom panel to map loads on geometry to FEA mesh entities.

Click Map loads to map all geometric loads in the current load collector to FEA entities.

Note:
You can also pick other load collectors by clicking on the loadcols button and map loads in all of them together.
While you are in the Loads on Geom panel, press the H key to view panel-specific help.
When you are finished, click return to update the Step Manager with the new loads.</td></tr>
</table>

Define Temperature On: Node Sets
The Define Temperature on: Node sets option defines temperature on node sets.

The node set names are used in the *TEMPERATURE data lines instead of the individual nodes. Unlike Abaqus surfaces in HyperMesh, you can combine node sets with individual node IDs in the same *TEMPERATURE card.

> **Note:** There is no graphical display in HyperMesh for loads created on sets. Therefore, when you review a load collector in the **Step Manager**, only loads created on individual entities are highlighted. For loads defined on sets, the underlying nodes or elements are highlighted.

This dialog contains a node sets menu with a list of the existing node sets. It also contains a table for data line input with the following columns:

Table 123:

Column	Description
Nset	The name of the node sets. Node sets are added or removed using → or ←, respectively.
Temperature	Reference temperature value.
Gradient1	Temperature gradient in the n2-direction for beams or temperature gradient through the thickness for shells.
Gradient2	Temperature gradient in the n1-direction for beams.
Load Id	The ID of the load collector.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

The Define tab for Define CLOAD-Force/Moment on: Node sets contains the following buttons:

Table 124:

Button	Action
Review Set	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click **Review** to clear the review selections.
Create/Edit Set..	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
Display/Review from panel	Opens the appropriate HyperMesh panel. Use the **review** button to expand the loads and constraints on the sets for visualization purposes.

Button	Action
→	Add the selected node set from the drop-down menu to the data line table on the right.
←	Delete the selected node set from the data line table.
Review/Reset	Reviews the selected node set in the data line table. Right-click on **Review** to clear the highlighted selections.
Update	Updates the HyperMesh database with the data lines defined in the table. By default, HyperMesh does not create a display for loads defined with sets.

For tips on entering information and navigating in the Define tab, see Step Manager Tab Environment.

Load Step: Temperature: Delete Tab
In the Delete tab, delete boundaries and other loads.

There are three deletion options:

Table 125:

Option	Description
All loads in current collector	The Delete button deletes all the loads from the current load collector.
All 'Temperature' in current collector	The Delete button deletes only *TEMPETATURE loads from the current load collector.
By selection	The Pick Loads button opens the HyperMesh load selector panel. Pick the loads you want to delete and click **proceed**. The corresponding Reset button resets the selected loads. The Delete button deletes the selected loads.

Load Step: Temperature: Parameter Tab
In the Parameter tab, define optional parameters for the *TEMPERATURE card.

The supported parameters include: `Bstep`, `Binc`, `Estep`, `Einc`, `Input file`, `Result file`, `OP`, and `MidSide`.

Click **Update** to activate the optional parameter selection in the database.

Load Step: Inertia Relief

In the **Inertia relief** dialog, define and edit the *INERTIA RELIEF card.

To open the dialog in the **Load Step** window, select **Inertia relief** from the tree and a load collector from the Load collector table.

Load Step: Inertia Relief: Define Tab
In the Define tab, define * INERTIA RELIEF cards.

Activate the **Inertia relief** checkbox to make the *INERTIA RELIEF load active for the current load collector.

There are two data line options available:

Free direction
: Integer list of degrees of freedom identifying the free directions

Reference points
: Global X, Y, and Z-coordinates of the reference point.

Click **Update** to activate the data line selection in the HyperMesh database.

Load Step: Inertia Relief: Parameter Tab
In the Parameter tab, define optional parameters for the * INERTIA RELIEF card.

The supported parameters include: `Orientation`, `Fixed`, `Remove`, and `None`.

Click **Update** to activate the optional parameter selection in the HyperMesh database.

Load Step: Interface Controls

The Interface Controls option defines Abaqus keywords.

The following Abaqus keywords are supported: *CONTACT, *CONTACT CONTROLS, *CLEARANCE, *CONTACT INTERFERENCE, *MODEL CHANGE, *CHANGE FRICTION, and *CONTROLS. It also allows you to add *CONTACT PAIR and *SURFACE INTERACTION cards created from the Abaqus **Contact Manager** to a load step. When you select an interface controls option from the tree, the corresponding table is displayed in the **Load Step** window.

The Interface controls: table contains a list of interface controls of the type selected in the tree. You can create, edit, rename, reorder, and delete interface controls from this table.

The Interface controls table contains the following columns:

Table 126:

Column	Description
Status	The history status of the interface control. If the status is on, the corresponding interface control parameters and data lines will be exported in the current load

Column	Description
	step. If the status is off, the interface control will not be exported under the current *STEP block.
Name	The name of the interface controls. Some of the interface controls names are for HyperMesh internal use only. Abaqus input files do not require them.

Note:

- Right-click on the table to display menu options. The available options are Rename and Reorder.

The Interface controls table contains the following buttons:

Table 127:

Button	Description
New	Opens the **Create** dialog for the corresponding interface control. The name of the new interface control is entered in this dialog. Use the Same as option to create an interface control by copying attributes from an existing interface control of the same type. Click **Create** to create an interface control and add it to the current load step.
Edit	Opens the Card Image panel for the selected interface control. Click **edit** to open the Card Editor and define all relevant keywords, parameters, and data lines. When you are finished, click **return** twice and the **Step Manager** is updated.

Load Step: Output Request

Use the Output requests option define output options for ODB, result (`.fil`) and data (`.dat`) file formats.

In HyperMesh, Abaqus output requests are organized into HyperMesh collectors called output blocks. When you select an output request file format from the tree, the output block table is displayed in the **Load Step** window.

The Output block: table contains a list of the output blocks with corresponding history status. You can create, edit, review, rename, reorder, and delete output blocks from this table.

The Output block: table contains the following columns:

Table 128:

Column	Description
Status	The history status of the output block. If the status is on, the output block belongs to the current load step. This means, all parameter and data line information in the output block will be written under the current *STEP block. If the status is off, no parameters or data lines from the output block will be written under the current *STEP block.
Name	The name of the output block. These names are for HyperMesh internal use only.

Note:

- Click on an output block name to set it as the current output block in HyperMesh. All changes and additions to the parameters and data line will be for the selected output block.
- The Output type: status bar, below the output block table, shows all the output types present in the selected output block.
- Right-click on the table to display menu options. The available options are Rename and Reorder.

See Step Manager Dialog Environment for tips on navigating through the dialogs.

The Output block table contains the following buttons:

Table 129:

Button	Action
New...	Opens the **Create Output block** dialog in which you enter the name of the new output block. The Create button in this dialog creates the output block and adds it to the current load step.
Review	Reviews the selected output block in a text window. All parameters and data lines associated with the selected output block are listed in the text window.
Delete	Opens the HyperMesh Delete panel to delete output blocks. When you are finished, click **return** and the Output block table will be updated.

Load Step: Output Request: ODB File

In the **ODB file** dialog, define and edit the output requests for the ODB file.

Supported keywords are: *OUTPUT, *NODE OUTPUT, *ELEMENT OUTPUT, *CONTACT OUTPUT, and *ENERGY OUTPUT. To open the dialog in the **Load Step** window, select **ODB file** from the tree and an Output block from the table.

Load Step: Output Request: ODB File: Output
In the Output tab, define *OUTPUT cards with associated parameters.

Supported parameters are *`Field`* and *`History`* (in the Output drop down menu), and *`Name`*, *`OP`*, *`Variable`*, *`Frequency`*, *`Time marks`*, *`Number interval`*, *`Time interval`*, and *`Mode list`* and *`Filter`* under the Optional parameters list.

Click **Update** to activate the parameters defined in the HyperMesh database.

To enable the Node Output, Element Output, Contact Output, Energy Output and Radiation output tabs used to define their respective cards, you must first activate the corresponding checkboxes.

Load Step: Output Request: ODB File: Node Output
In the Node Output tab, define *NODE OUTPUT for the selected Output block.

`Nset` and *`Variable`* parameters are supported. Activate/deactivate the check boxes in the tree on the Node Output tab to add/remove identifier keys in the table. The data lines added in this manner have a gray background indicating they cannot be manually edited. You may also add user-defined identifiers, by typing them directly into the table. User-defined data lines appear with a white background, indicating they are editable.

Right-click on the **Data lines** table to display menu options. The available options are Cut, Copy, Paste, Add row, and Delete row.

The Node Output tab contains the following buttons:

Table 130:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the node set list.
Review	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Output Request: ODB File: Element Output
In the Element Output tab, define *ELEMENT OUTPUT for the selected Output block.

The following parameters are supported: *`Elset`*, *`Position`*, and *`Variable`*. Activate/deactivate the check boxes in the tree on the Element Output tab to add/remove identifier keys in the table. Data lines added in this manner have a gray background indicating they cannot be manually edited. You may also add user-defined identifiers, by typing them directly into the table. The user-defined data lines appear with a white background, indicating they are editable.

Right-click on the **Data lines** table to display menu options. The available options are Cut, Copy, Paste, Add row, and Delete row.

The Element Output tab has the following buttons:

Table 131:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
Review	Reviews the selected element sets by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Output Request: ODB File: Contact Output
In the Contact Output tab, define *CONTACT OUTPUT for the selected Output block.

The following parameters are supported: `Nlset`, `Master`, `Slave`, `General Contact`, and `Variable`. Activate/deactivate the checkboxes in the tree on the Contact Output tab to add/remove identifier keys in the table. The data lines added in this manner have a gray background, indicating they cannot be manually edited. You may also add user-defined identifiers, by typing them directly into the table. The user-defined data lines appear with a white background, indicating they are editable.

Right-click on the **Data lines** table to display menu options. The available options are Cut, Copy, Paste, Add row, and Delete row.

The Contact Output tab has the following buttons:

Table 132:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the node set list.
Review	Reviews the selected node sets, master surface, or slave surface by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.

Button	Action
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Output Request: ODB File: Energy Output
In the Energy Output tab, define *ENERGY OUTPUT for the selected Output block.

The following parameters are supported: `Elset` and `Variable`. Activate/deactivate the checkboxes in the tree on the Energy Output tab to add/remove identifier keys in the table. The data lines added in this manner have a gray background, indicating they cannot be manually edited. You may also add user-defined identifiers by typing them directly into the table. The user-defined data lines appear with a white background, indicating they are editable.

Right-click on the **Data lines** table to display menu options. The available options are Cut, Copy, Paste, Add row, and Delete row.

The Energy Output tab has the following buttons:

Table 133:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
Review	Reviews the selected element sets by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Output Request: Result File (`.fil`)
In the **Result file (.fil)** dialog, define and edit the output requests for the Result file.

Supported keywords are *NODE FILE, *ELEMENT FILE, *CONTACT FILE, and *ENERGY FILE. To open the dialog in the **Load Step** window, select **Result File (.fil)** from the tree and an Output block from the table.

Load Step: Output Request: Result File: Define
In the Define tab, enable the Node File, Element File, Contact File, and Energy File tabs, which are used to define the *Node File, *Element File, *Contact File, and *Energy File cards, respectively.
Load Step: Output Request: Result File: Node File
In the Node File tab, define *Node file for the selected Output block.

The following parameters are supported: `Nset`, `Frequency`, `Last mode`, `Global`, and `Mode`. Activate/deactivate the checkboxes in the tree on the Node File tab to add/remove identifier keys in the table. The data lines added in this manner have a gray background, indicating they cannot be manually edited. You may also add user-defined identifiers by typing them directly into the table. User-defined data lines appear with a white background, indicating they are editable.

Right-click on the **Data lines** table to display menu options. The available options are Cut, Copy, Paste, Add row, and Delete row.

The Node File tab contains the following buttons:

Table 134:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the node set list.
Review	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Output Request: Result File: Element File
In the Element File tab, define *ELEMENT FILE for the selected Output block.

The following parameters are supported: `Elset`, `Directions`, `Mode`, `Frequency`, `Position`, and `Last Mode`. Activate/deactivate the check boxes in the tree on the Element File tab to add/remove identifier keys in the table. The data lines added in this manner have a gray background, indicating they cannot be manually edited. You may also add user-defined identifiers by typing them directly into the table. User-defined data lines appear with a white background, indicating they are editable.

Right-click on the **Data lines** table to display menu options. The available options are Cut, Copy, Paste, Add row, and Delete row.

The Element File tab contains the following buttons:

Table 135:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
Review	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Output Request: Result File: Contact File
In the Contact File tab, define *CONTACT FILE for the selected Output block.

The following parameters are supported: *Nset*, *Master*, *Slave*, and *Frequency*. Activate/deactivate the checkboxes in the tree on the Contact File tab to add/remove identifier keys in the table. The data lines added in this manner have a gray background, indicating they cannot be manually edited. You may also add user-defined identifiers, by typing them directly into the table. The user-defined data lines appear with a white background, indicating they are editable.

Right-click on the **Data lines** table to display menu options. The available options are Cut, Copy, Paste, Add row, and Delete row.

The Contact File tab contains the following buttons:

Table 136:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the node set list.
Review	Reviews the selected node sets, master surface, or slave surface by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Output Request: Result File: Energy File
In the Energy File tab, define *ENERGY FILE for the selected Output block.

The following parameters are supported: *Elset* and *Frequency*.

The Energy File tab contains the following buttons:

Table 137:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
Review	Reviews the selected element sets by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Output Request: Data File (`.dat`)
In the **Data file (.dat)** dialog, define and edit the output requests for the data file.

Supported keywords are *NODE PRINT, *ELEMENT PRINT, *CONTACT PRINT, and *ENERGY PRINT. To open the dialog in the **Load Step** window, select **Data File (.dat)** from the tree and an output block from the Output block table.

Load Step: Output Request: Data File: Define
In the Define tab, enable the Node Print, Element Print, Contact Print, and Energy Print tabs, which are used to define the *Node Print, *Element Print, *Contact Print, and *Energy Print cards.

Load Step: Output Request: Data File: Node Print
In the Node Print tab, define *NODE PRINT for the selected Output block.

The following parameters are supported: *`Nset`*, *`Frequency`*, *`Mode`*, *`Global`*, *`Summary`*, *`Last mode`*, and *`Totals`*. Activate/deactivate the checkboxes in the tree on the Node Print tab to add/remove identifier keys in the table. The data lines added in this manner have a gray background, indicating they cannot be manually edited. You may also add user-defined identifiers by typing them directly into the table. User-defined data lines appear with a white background, indicating they are editable.

Right-click on the **Data lines** table to display menu options. The available options are Cut, Copy, Paste, Add row, and Delete row.

The Node Print tab contains the following buttons:

Table 138:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the node set list.
Review	Reviews the selected node sets by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Output Request: Data File: Element Print
In the Element Print tab, define *Element print for the selected Output block.

The following parameters are supported: *`Elset`*, *`Position`*, *`Totals`*, *`Frequency`*, *`Last mode`*, *`Summary`*, and *`Mode`*. Activate/deactivate the checkboxes in the tree on the Element Print tab to add/remove identifier keys in the table. The data lines added in this manner have a gray background indicating they can not be manually edited. You may also add user-defined identifiers by typing them directly into the table. User-defined data lines appear with a white background, indicating they are editable.

Right-click on the **Data lines** table to display menu options. The available options are Cut, Copy, Paste, Add row, and Delete row.

The Element Print tab contains the following buttons:

Table 139:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
Review	Reviews the selected element sets by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Output Request: Data File: Contact Print
In the Contact Print tab, define *CONTACT PRINT for the selected Output block.

The following parameters are supported: `Nset`, `Master`, `Slave`, `Frequency`, `Totals`, and `Summary`. Activate/deactivate the checkboxes in the tree on the Contact Print tab to add/remove identifier keys in the table. The data lines added in this manner have a gray background, indicating they cannot be manually edited. You may also add user-defined identifiers by typing them directly into the table. User-defined data lines appear with a white background, indicating they are editable.

Right-click on the **Data lines** table to display menu options. The available options are Cut, Copy, Paste, Add row, and Delete row.

The Contact File tab contains the following buttons:

Table 140:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the node set list.
Review	Reviews the selected node sets, master surface, or slave surface by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Output Request: Data File: Energy Print
In the Energy Print tab, define *energy print for the selected Output block.

The following parameters are supported: `Elset` and `Frequency`.

The Energy File tab contains the following buttons:

Table 141:

Button	Action
Create/Edit...	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the element set list.
Review	Reviews the selected element sets by highlighting them in the HyperMesh graphics. Right-click on the **Review** button to clear the review selections.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Monitor

Select **Monitor** to define the *MONITOR card.

The supported attributes are: `Node`, `Node Set`, `DOF`, and `Frequency`.

Click **Update** to activate the monitor attributes defined in the HyperMesh database.

The following buttons are available on the **Monitor** dialog:

Table 142:

Button	Action
Pick node	Opens the node selection panel. When you finish picking a node from the model, click **return**. The **Step Manager** is updated with the selected node number appearing in the Node entry.
Create/Edit	Opens the Entity Sets panel in HyperMesh. When you finish creating/editing the set, click **return**. The **Step Manager** is updated with the new set appearing in the node set list. For the Abaqus solver, the set must contain one node only.
Update	Updates the HyperMesh database with the data lines defined in the table and the parameters.

Load Step: Print

Select **Print** in the tree to define the *PRINT card.

The supported attributes are: `Contact`, `Model change`, `Plasticity`, `Residual`, `Solve`, `Frequency`, `Allke`, `Critical element`, `Dmass`, and `Etotal`.

Click **Update** to activate the monitor attributes defined in the HyperMesh database.

Load Step: File Format

Select **File format** in the tree to define the *FILE FORMAT card.

This option is available when a standard Abaqus template is loaded. The supported attributes are *`File format`* and *`Zero increment`*.

Click **Update** to activate the monitor attributes defined in the HyperMesh database.

Load Step: File Output

Select **File output** in the tree to define the *FILE OUTPUT card.

This option is available when the Abaqus explicit template is loaded. The supported attributes are *`File output`*, *`Number interval`*, and *`Time marks`*.

Click **Update** to activate the monitor attributes defined in the HyperMesh database.

Load Step: Restart Write

Select **Restart write** in the tree to define the *RESTART, WRITE card.

The supported attributes are *`Overlay`*, *`Frequency`*, *`Number interval`* and *`Time marks`*.

Click **Update** to activate the monitor attributes defined in the HyperMesh database.

Load Step: Unsupported Cards

The Unsupported cards option in the tree enables you to review and edit unsupported history data within the **Step Manager**.

Select the checkbox to activate the text area. You can enter the unsupported cards by typing directly or copying and pasting into the text area. Click **Update** to include the cards.

Abaqus Step Manager: Load Case Tab

You can create, edit, review, rename, reorder and delete load cases from this tab as well as set the display status of the load steps.

The Load Case tab contains the descriptions of all existing load cases with the corresponding load collectors.

See Step Manager Dialog Environment for tips on navigating through the dialogs.

The Load Case tab contains the following buttons:

Table 143:

Button	Action
New...	Opens the **Create New Load Case** dialog. Enter the name of the new load case in the Name: text box and click **Create** to create the load step and open the corresponding **Load Case Editing** dialog.
Edit...	Opens the **Load Step Editing** dialog for the selected load step.
Review	Reviews the selected load collectors. All loads in the load collectors are highlighted in the HyperMesh graphics. The highlighted loads show through the solid mesh in performance graphics. If a load is defined with set, the underlying nodes or elements are highlighted. Right-click **Review** to clear the highlighted selections.
Text	Reviews the selected load step in a text window.
Rename	Opens the Rename panel for renaming load steps, load collectors, output blocks, or various interface controls. When you finish renaming, click **return** to update the **Step Manager** with the new names.
Delete	Deletes the selected load steps.
Sync	Updates the **Step Manager** with the current HyperMesh database. If you manually create, update, or delete load steps, load collectors, output blocks, groups, or entity sets from HyperMesh panels while the **Step Manager** is open, click **Sync** to update the **Step Manager** with the new changes.
Close	Closes the **Step Manager**.
↑	Moves the selected load cases up one row.
↓	Moves the selected load cases down one row.

Find and Replace

Use the Find and Replace tool to search the entity names in your model for every occurrence of a specific character, and automatically replace the character with a new one that you specify.

For example, using " . " may cause errors in certain solvers, therefore you may want to replace it with a new character.

1. In the Abaqus Utility menu, click **Find and Replace**.
   The **Find and Replace** dialog opens.
2. In the Replace What field, enter the character you want to find.
3. In the Replace With field, enter the character you would like to replace the existing character with.

4. Set the Entity Type selector to **ALL** or to a specific entity type.

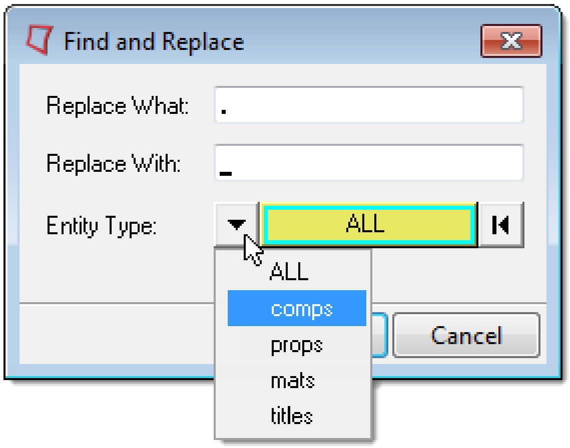


Figure 537:

5. Optional: If you set the Entity Type selector to a specific entity type, you can activate the selector and select specific entities to perform the find and replace on.
6. Click **Replace**.

ANSYS Utility Menu

The macros on the ANSYS Utility menu simplify some common tasks for the ANSYS user profile.

The ANSYS Utility menu is loaded when you open the ANSYS user profile. The following ANSYS macros are available.

ANSYS Component Manager

This macro displays components and their associated attributes in an interactive table.

You can also configure the table; only configured items are displayed in the table.

With this macro, you can also create components, select components, assign materials to components, change component colors, and change component visualization modes. Most actions are available from shortcut (right-click) menus. You can also find options in the drop-down menus.

Before performing actions such as changing the values of component data, you must select **Editable** from the Table menu. Once the components are writable, you can modify the values of existing components. The following sections describe how to use the **Component Manager** in both read-only mode and editable mode.

Use the Component Manager in Read-Only Mode

When you open the **Component Manager**, existing components are listed in a table using a default configuration. This configuration displays the name, ID number, ET reference number, element type, real set number, material set number, section set number, and number of elements and nodes in each component.

A sum of elements is shown at the bottom of the table. If a component is invalid for any reason and cannot be exported to the ANSYS data deck, its row in the table will appear in red.

The display of the data in the **Component Manager** can be customized according to your preferences. You can:

- Change which columns are displayed
- Change the order of the columns
- Sort the components by column data, ascending or descending
- Filter which components are displayed based on column data values (see below)

You can save your settings by creating a configuration file.

From the Table menu, open the Configure submenu and select the **Save CFG-File** option.

This configuration file saves the set of table configuration options so you can use them again. By default, a configuration file (`comptable.cfg`) is saved in the working directory for each component table session and settings from this file are applied each time the table is built.

Use the Component Manager in Editable Mode

When you switch the **Component Manager** from the default read-only mode to editable mode, you can perform all the actions described in the section above, plus edit the attributes of the components listed in the table.

1. Select **Editable** from the Table menu.
2. To change the value of an attribute, select the attribute in the Assign Values drop-down, type the new value in the adjacent field, and click **Set**.

Filter the List of Components

If you have a long list of components and you want to narrow down the list of components that appear in the table, you can use the filtering feature to specify the criteria for matching components.

1. Select **Filter...** from the Table menu.
   The **Filter** dialog opens.
2. Type a match value in the box next to the criteria by which you want to filter.

Create a New Component Card with the Component Manager

1. In the ANSYS **Component Manager**, click the **Action** menu and select **Create New...**.
   The **Create Component** dialog opens.
2. Type a name for the component in the Component name field.
3. Select an element type in the Element reference number field.
4. Select a real set in the Real set number field.
5. Select a material in the Material reference number field.

6. Select a section in the Section reference number field.
7. Click the button in the Color field to select a color for the component that will be used in HyperMesh.
8. Click **Create**.
9. Click **Close** to go back to the **Component Manager**.

Edit a Component Card

1. Click anywhere in the component's row in the **Component Manager** table to select it.
2. If not already in Edit mode, click **Table** and **Editable**.
3. In the Assign Values drop-down, select a field to modify.
4. In the adjacent field, type or select the new value.
5. Click **Set**.
   A new value appears in the table.

ANSYS Material Macro

The **Material** macro creates and defines material cards.

This macro lists the existing materials in the model in a table and displays the material Set Number, Type, and Name for each material. From this macro dialog, you can create a new material, edit an existing material, or delete a material.

Rows can be sorted by set number, material type and name columns.

The following buttons are available on the **Material** macro dialog:

Table 144:

Button	Action
Help	Provides information about the macro.
New...	Opens the **Create Material** dialog, from which you can specify parameters for a new material and create it.
Edit...	Opens the **Edit Material** dialog, from which you can review and change parameters for an existing material that is selected in the table.
Refresh	Refreshes the HyperMesh database with changes you made through the **Material** macro. You must click this button before closing the dialog to successfully export the ANSYS deck.
Delete	Deletes the material that is selected in the table.
Close	Closes the **Material** macro dialog.

Right-clicking on a row displays a context menu with the option to delete unused materials. Selecting this option can help you clean up your model by deleting materials.

You can also edit materials, edit material properties, and delete materials from the menu that appears when you right-click in the table when a row/material is selected.

Create Material Dialog

Create new materials from the **Material** dialog.

Specify the following options and click **Create** to create the material and return to the **Material** macro, or **Create/Edit...** to create the material and open the card image panel in HyperMesh to specify the material's properties.

The following options are available in the **Material** dialog:

Table 145:

Option	Description
Material Set No	Type a number for the material set. If you do not specify a value, a number will be automatically provided that is one higher than the highest current material set number.
Material Name	Type a name to identify the material.
Material Type	Select **MP** or **MPDATA** depending on the material card you want to use.

Edit Material Dialog

Edit existing materials from the **Edit Material** dialog.

Specify the following options and click **Update** to save the changes. Then click **Close** to return to the **Material** macro. You can also click **Edit Material Properties...** to open the card image panel in HyperMesh to modify the material's properties.

The following options are available in the **Material** dialog:

Table 146:

Option	Description
Material Set No	This field is initially populated by the reference number of the material card that you have chosen to edit. You can change the number to any other reference number if the number is not already used by an existing material card.
Material Name	This field is initially populated with the current name of the material. You can modify the name of the material.

Option	Description
Material Type	You can change the card image type.

Note: If you make changes to a material, you must click **Update** to reflect the changes in the card image.

ANSYS Section Macro

The **Section** macro creates and defines section cards for beam and shell sections.

You can create new section, edit existing sections, and use the HyperMesh application to create and edit beam section and associate them to section cards.

The following buttons are available on the **Section** macro dialog:

Table 147:

Button	Action
Help	Provides information about the macro.
New...	Opens the **Create Section** dialog, from which you can specify parameters for a new section and create it.
Edit...	Opens the **Edit Section** dialog, from which you can review and change parameters for an existing material that is selected in the table.
Refresh	Refreshes the HyperMesh database with changes you made through the **Material** macro.
Delete	Deletes the section card(s) that is selected in the table.
Close	Closes the **Section** macro dialog.

Right-clicking on a row displays a context menu with the option to delete unused sections. Selecting this option can help you clean up your model by deleting unused sections.

You can also edit sections, edit section properties, and delete sections from the menu that appears when you right-click in the table when a row/section is selected.

Create Section Dialog

Create a new beam, shell or pretension section from the **Create Section** dialog.

Specify the following options and click **Create** to create the section card and return to the **Section** macro, or **Create/Edit...** to create the section card and open the Card Image panel in HyperMesh to specify section properties.

The following options are available in the **Create Section** dialog:

Table 148:

Option	Description
Section Ref No	Type a reference number for the section. If you do not specify a value, a number will be automatically provided that is one higher than the highest current section ID number.
Section Name	Type a name for the section.
Section Type	Select the type of section to create: Beam, Taper, Shell or Pretension
Sub Type	This selection only applies to beam sections. Choose a subtype from the drop-down list.
Define by HyperBeam	This option is only available if **Beam Section** is selected. Select the check box to define the section with HyperBeam. Then select: • Choose **New Section** to create a section in HyperBeam and associate it to the section card image. • Choose **Existing Section** to select an available HyperBeam section of the chosen sub-type in the Sub Type field. When you click **Create**, a section card is created with the selected HyperBeam section or if you click **Create/Edit...**, the card image is opened in HyperMesh. When you are in the HyperBeam application, it is possible to create multiple HyperBeam sections. However, only the most recently created section is attached the section card.

Edit Section Dialog

Use the **Edit Section** dialog to edit existing beam, shell or pretension sections.

Modify the following options and click **Update** to save the changes. Click **return** to return to the **Section** macro. You can also click **Edit Properties...** to open the Card Image panel in HyperMesh to edit the section's properties.

The following options are available in the **Edit Section** dialog:

Table 149:

Option	Description
Section reference number:	This field is initially populated by the reference number of the section card that you have chosen to edit. You can change the number to any other reference number if the number is not already used by an existing material card.
Section name:	This field is initially populated with the current name of the section. You can modify the name of the section.
Section type:	You can change the type of section to create.
Sub type:	This selection only applies to beam sections. You can select another subtype from the drop-down list.
Define by HyperBeam	This option is only available if **Beam Section** is selected. Select the checkbox to modify the section definition with HyperBeam. Then click **Edit HyperBeam...** to modify the section in HyperBeam. If you edit a section card that was created with HyperBeam sections, then this checkbox is initially selected. However, you can clear the checkbox and make updates via the card image or vice versa.

If you make changes to a section, you must click **Update** to reflect the changes in the card image.

Create a SECDATA Card with the Section Macro

1. From the ANSYS Utility menu, click **Section...**.
   The **Section** dialog appears.
2. To create a new section card, click **New...** at the bottom of the window.
   The **Create Section** dialog appears.
3. Type a reference number for the section in the Section reference number field.

   A number will already be provided by HyperMesh, but you can replace this value.
4. Type the name for the section card in the Section name field.
5. Select the type of section you want to create.
   - Beam
   - Shell
6. If you chose **Beam** in step 5, then select a Sub type from the pull-down menu.

   Shell sections do not have subtypes.
7. At this stage you need to decide if you want to associate a HyperBeam section to the section card image you are creating. If you do not want to use HyperBeam sections, clear the **Define by HyperBeam** checkbox and skip to step 10.

8. To use a HyperBeam section, select **Define by HyperBeam**.
9. If you have already created sections in HyperBeam of the selected subtype, you can associate that section with the new section card image. To create a new section, skip to step 10. Select the existing section option and choose an existing HyperBeam section from the pull-down menu. Skip to step 10.
   a) To create a new section in HyperBeam and associate it to the section card image you are creating, select the **New section** option and click **Create** to create a section without properties defined, or **Create/Edit...** to edit the properties in the section card before saving. The HyperBeam application opens.
   b) Create the section and exit from HyperBeam.
   c) Click **return** in the HyperBeam panel in HyperMesh to go back to the **Create Section** dialog. Skip to step 11.
10. Click **Create** to create a section without properties defined, or **Create/Edit...** to edit the properties in the section card before saving.

    The following images indicate the location of the value fields in the SECDATA card, such as W1, W2, t1 and t2.

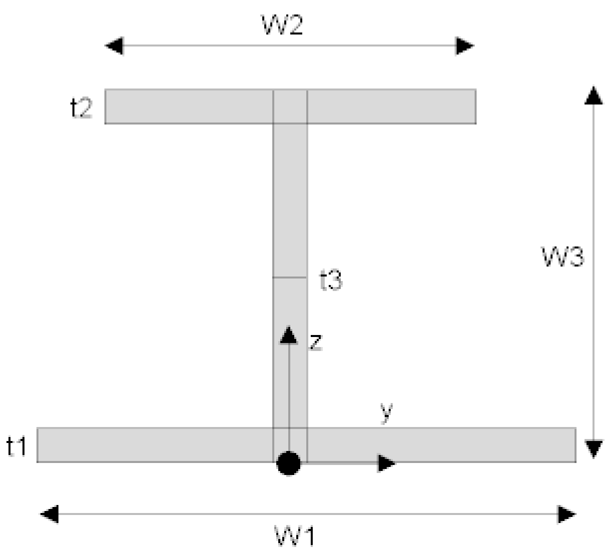


Figure 538: I Beam

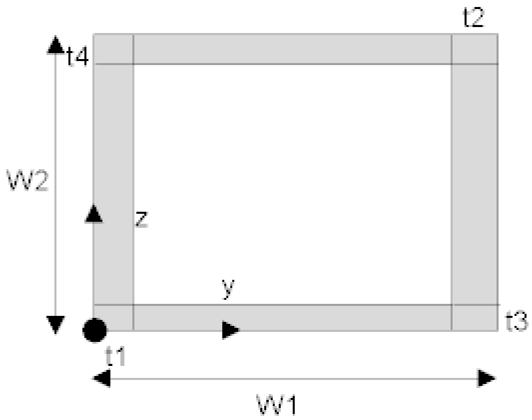


Figure 539: HREC Beam

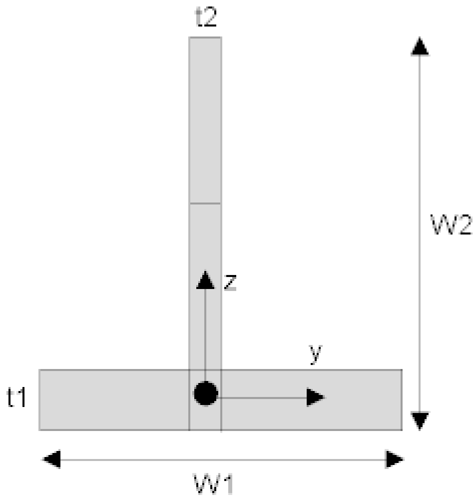


Figure 540: T Beam

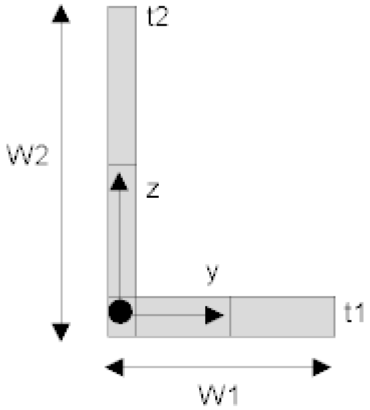


Figure 541: L Beam

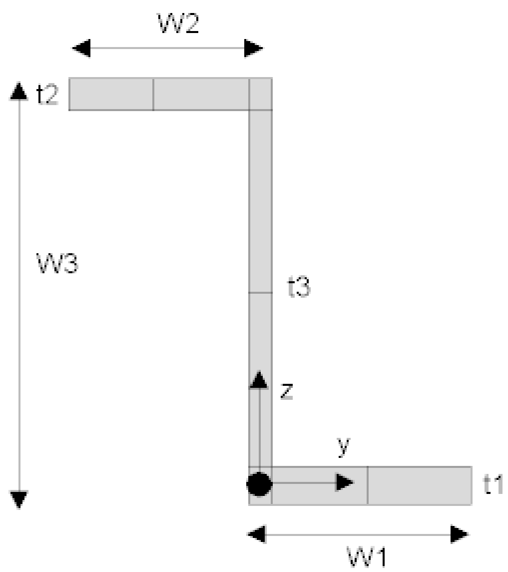


Figure 542: Z Beam

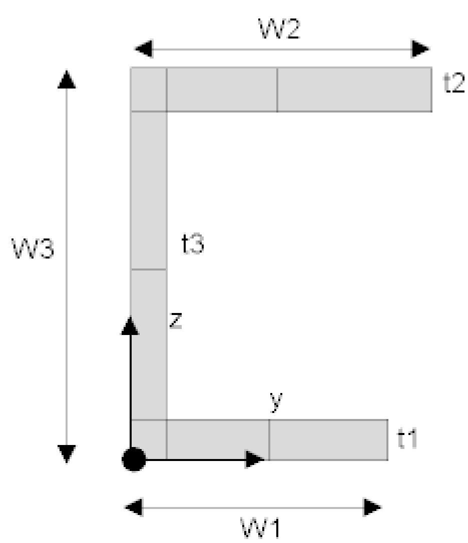


Figure 543: CHAN Beam

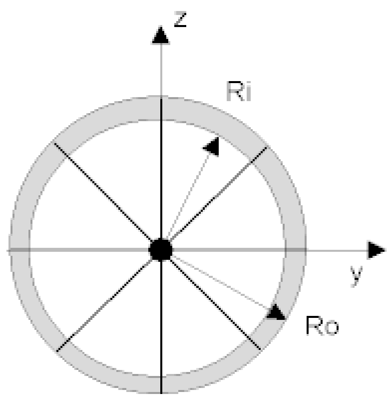


Figure 544: CTUBE Beam

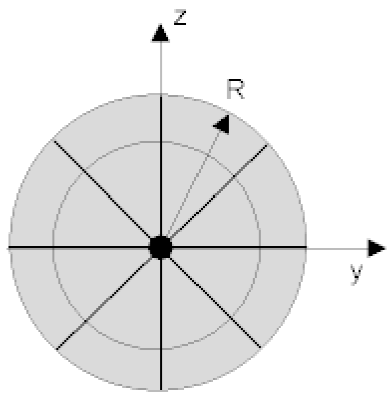


Figure 545: CSOLID Beam

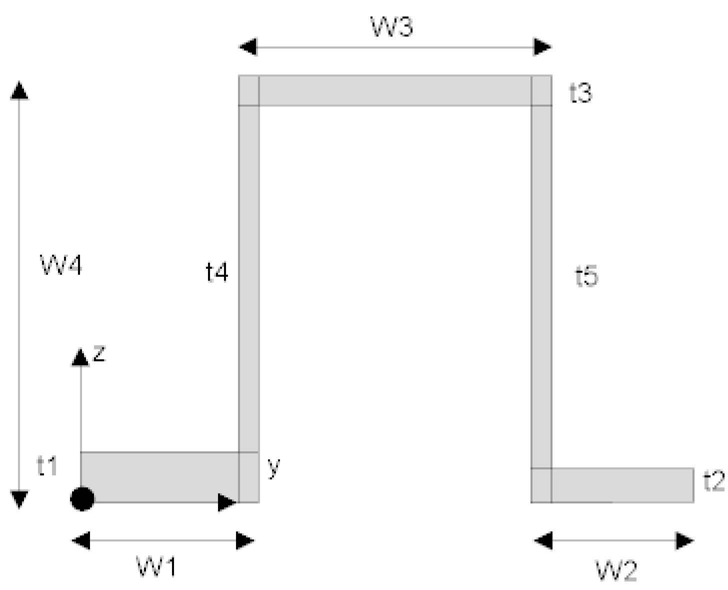


Figure 546: HATS Beam

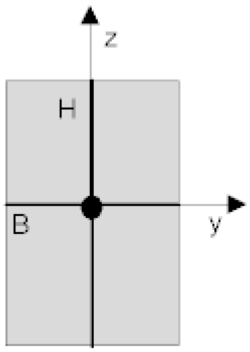


Figure 547: RECT Beam

11. In the **Section** dialog, click **Close** to return to HyperMesh.

ANSYS Real Sets Macro

The **Real Sets** macro creates property card images for all elements that the ANSYS interface supports.

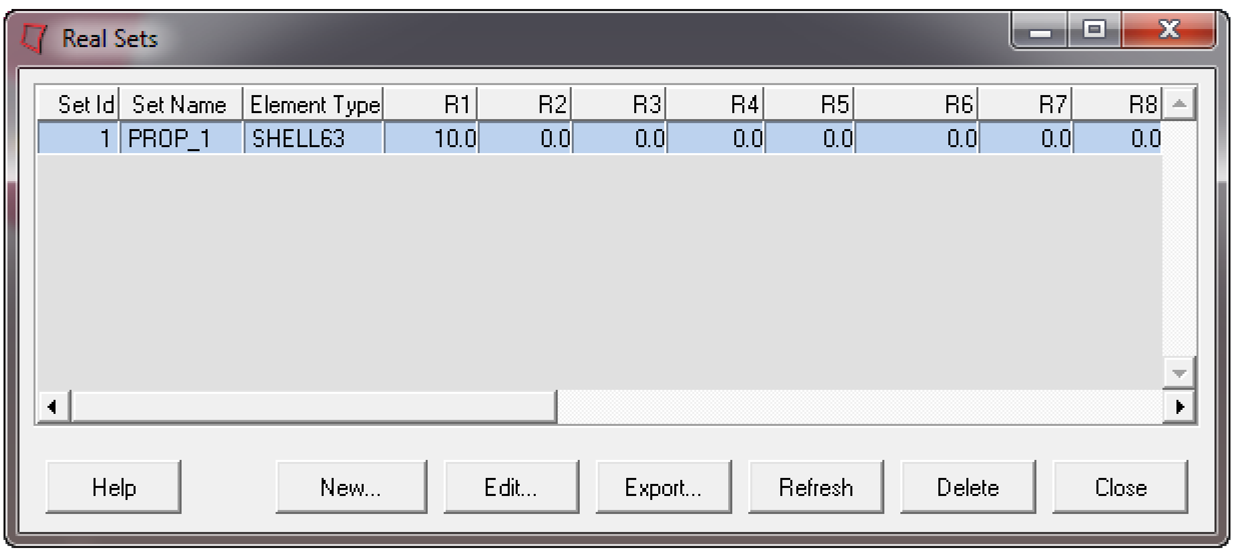


Figure 548:

With this macro, you can view the real constant values of existing real sets, create new real sets, and edit the properties and reference numbers of existing real sets.

The **Real Sets** macro dialog displays the real set, reference numbers, element type of the property set, and the real constant values for that element type.

Rows can be sorted by set name, set ID and the element type column.

The following buttons are available on the **Real Sets** macro dialog:

Table 150:

Button	Action
Help	Provides information about the macro.
New...	Opens the **Create Real Sets** dialog, from which you can specify parameters for a new real set and create it.
Edit...	Opens the **Edit Real** dialog, from which you can review and change parameters for an existing real set that is selected in the table.
Export...	Allows you to export the table in `.csv` format
Refresh	Refreshes the HyperMesh database with changes you made through the **Real Sets** macro.
Delete	Deletes the real set(s) that is selected in the table.
Close	Closes the **Real Sets** macro dialog.

Right-clicking on a row displays a context menu with the option to delete unused real constants. Selecting this option can help you clean up your model by deleting unused real constants.

You can also edit real sets, edit real constants, and delete real sets from the menu that appears when you right-click in the table when a row/real set is selected.

Create Real Sets Dialog

Use the **Create Real Sets** dialog to create new real sets.

Specify the following options and click **Create** to create the real set and return to the **Real** macro, or **Create/Edit...** to create the real set and open the Card Image panel in HyperMesh to specify the real constants for the element type.

The following options are available in the **Create Real Sets** dialog:

Table 151:

Option	Description
Real Set No	Type an identification number for the real set. If you do not specify a value, a number will be automatically provided that is one higher than the highest current real set ID number.

Option	Description
Element Type	Select an element type from the drop-down list for which a real set is to be created. The element types are listed in groups sorted by type, as shown in the image below.

Edit Real Sets Dialog

Use the **Edit Real Sets** dialog to edit existing real sets.

Modify the following options and click **Update** to save the modified real set. Then click **return** to return to the **Real Sets** macro. You can also modify the real constants by clicking the **Edit Real Constants...** button, which opens the Card Image panel.

The following options are available in the **Edit Real Sets** dialog:

Table 152:

Option	Description
Real Set No	This field is initially populated by the reference number of the real set that you have selected to edit. You can change the number to any other reference number that is not already used by an existing real set.
Element Type	This field is initially populated by the element type of the real set that you have selected to edit. You can change the element type by selecting a new element type from the drop-down list. The element types are listed in groups sorted by type, as shown in the image below.

If you make changes to a real set, you must click **Update** to reflect the changes in the card image.

ANSYS ET Type Macro

The **ET Type** macro creates HyperMesh card images for ANSYS element types.

The **ET Type** macro dialog displays the ET type identification number, the element type name, and the key option values for each element type. From this macro dialog, you can create card images for any element type that HyperMesh supports, view existing ET types along with their reference numbers and key options, edit existing element types, or delete element types.

Rows can be sorted by ET Type name, ID and the element type column.

The following buttons are available on the **ET Type** macro dialog:

Table 153:

Button	Action
Help	Provides information about the macro.
New...	Opens the **Create ETType** dialog, from which you can specify an element reference number and key options for a new element type and create it.
Edit...	Opens the **Edit ETType** dialog, from which you can review and change the reference number and key options for an existing element type that is selected in the table.
Export...	Allows you to export the table in `.csv` format
Refresh	Refreshes the HyperMesh database with changes you made through the HyperMesh panels.
Delete	Deletes the element type(s) that is selected in the table.
Close	Closes the **ET Type** macro dialog.

Right-clicking on a row displays a context menu with the option to delete unused ET types. Selecting this option can help you clean up your model by deleting unused ET types.

You can also edit element types, edit key options, and delete element types from the menu that appears when you right-click in the table when a row/element type is selected.

Create ET Type Dialog

Use the **Create ETType** dialog to create new element types.

Specify the following options and click **Create** to create the element type and return to the **ET Type** macro, or **Create/Edit...** to create the element type and open the Card Image panel in HyperMesh to specify the key options for the element type.

The following options are available in the **Element Ref No** dialog:

Table 154:

Option	Description
Real Set No	Type an identification number for the real set. If you do not specify a value, a number will be automatically provided that is one higher than the highest current real set ID number.

Option	Description
Element Type	Select an element type from the drop-down list. The element types are listed in groups sorted by type, as shown in the image below.

Edit ETType Dialog

Use the **Edit ETType** dialog to edit existing element types.

Modify the following options and click **Update** to save the changes to the element type. Then click **return** to return to the **ET Type** macro. You can also click **Edit Key Options** to modify the key options for the element type in the HyperMesh Card Image panel.

The following options are available in the **Edit ETType** dialog:

Table 155:

Option	Description
Real Set No	This field is initially populated with the reference number of the element type you have selected to edit. You can change the reference number to any number that is not already used by an existing element type.
Element Type	This field is initially populated with the type of the element type you have selected to edit. You can select another element type from the drop-down list. The element types are listed in groups sorted by type, as shown in the image below.

If you make changes to an ET Type, you must click **Update** to reflect the changes in the card image.

ANSYS Convert to Special 2nd Order Macro

When you run the Convert to Special 2nd Order macro, a mesh matching is used to remove the mid-side nodes at the shared edges between these first and second order elements.

Models created for the ANSYS solver often contain second-order pyramid and tetra elements in which most sides contain "mid-side nodes". These types of elements exist in a transition layer between the first-order hexa and second-order tetra elements, as shown below.

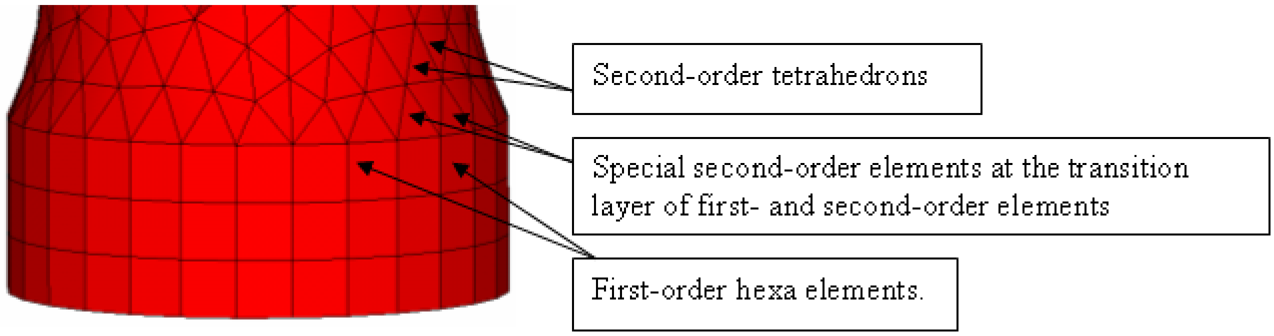


Figure 549:

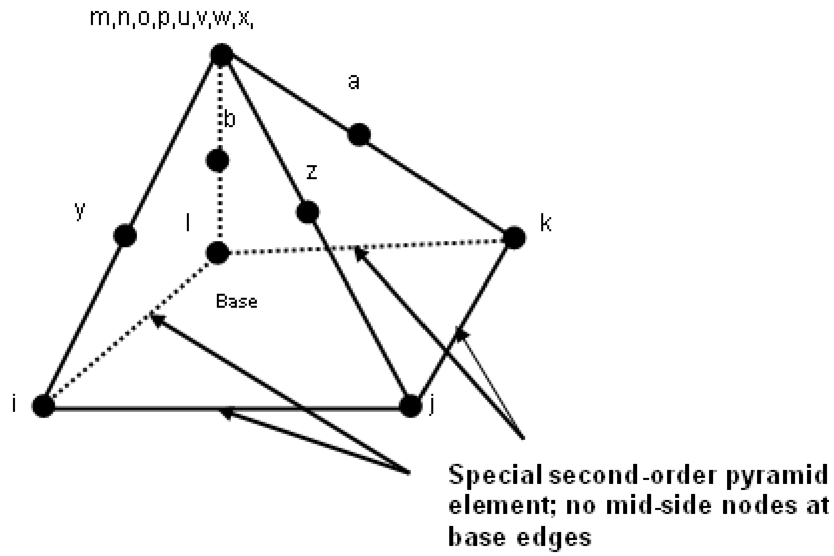


Figure 550:

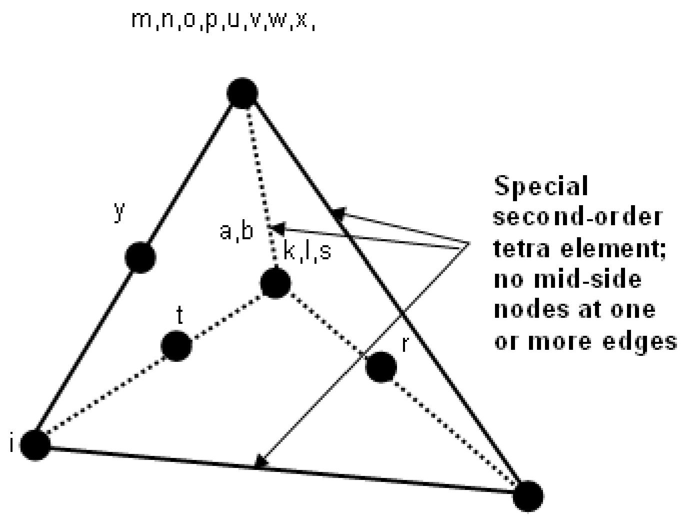


Figure 551:

Beginning in HyperMesh 8.0, these types of elements are supported and preserved in the model. HyperMesh can import:

- Pyramid-shaped SOLID95 and SOLID92 elements with side edges containing mid-side nodes and bottom (base) edges that do not contain mid-side nodes
- Tetrahedron elements with one or more edges that do not contain mid-side nodes
- SOLID95 (special type)
- SOLID187 (special type)

These special elements will be imported as full second-order elements, including mid-side nodes.

Imported full second-order elements are exported as special elements, thereby restoring the original element configuration. Similarly, special second-order elements created in HyperMesh are also exported as special second-order elements.

1. Mesh the part for first-order with hexa or penta elements. Place these elements in a collector with the correct element type (SOLID45).
2. Mesh the mating volume with second-order tetra elements. Place this mesh in a separate component with the correct element type (SOLID95 or SOLID92).
3. Ensure that the two mesh patterns have a common layer with shared edges between.
4. From the ANSYS Tools page of the Utility menu, click the **Convert to Special 2nd Order macro**.
5. Select the first-order component from the drop-down menu that shares a common face with the second-order meshed component.
6. Select the second-order meshed component in the next drop-down menu and click **Apply**.
   The special order elements are generated
7. Export the file. Read it in the solver and check the elements.

   The following images show examples of proper meshing for the above practice.

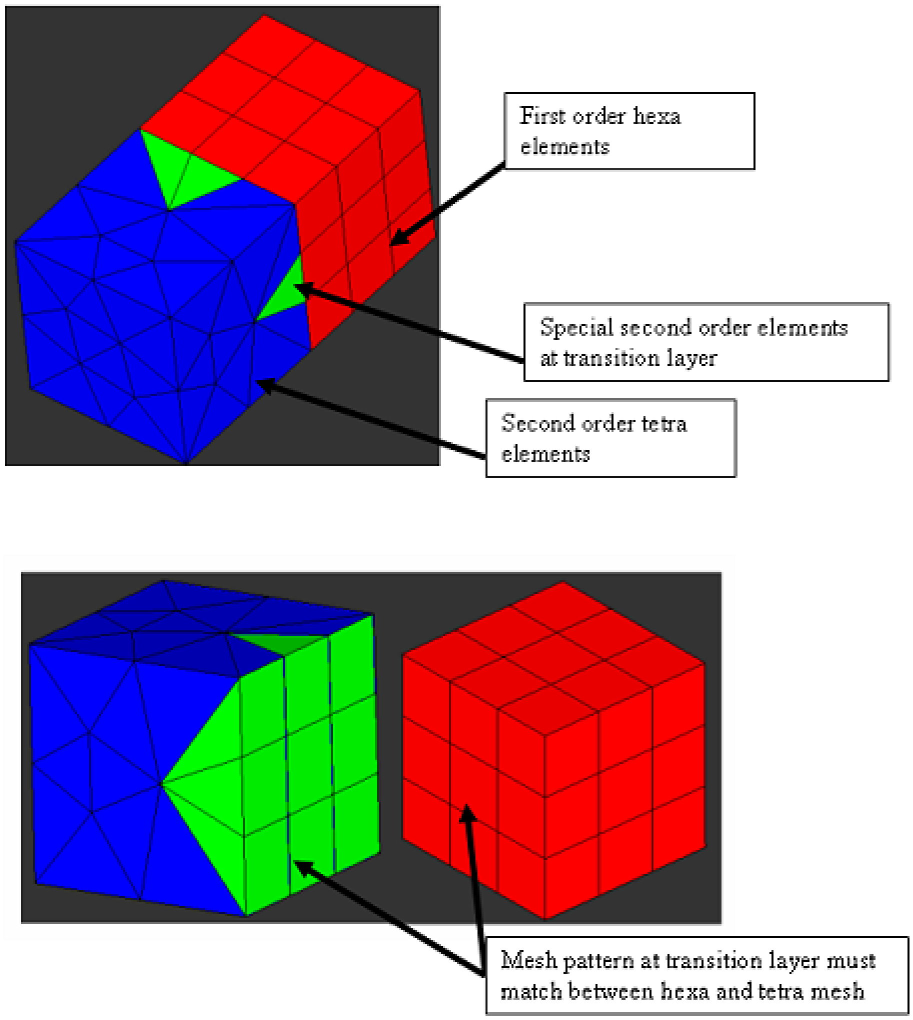


Figure 552:

ANSYS Surface Manager

Manage existing contact pairs and surface elements, and create new contact pairs and ANSYS thermal and structural surface elements using the **Ansys Surface Manager**.

In the Contacts tab you can review a list of existing contact pairs, and in the Surface Elements tab you can review a list of existing surface element components. Contact pairs are listed according to their target and master components, color, assigned ET type, property cards, material, and pilot node name. You can update the data in any of these columns, as well as modify the element key options, properties, and materials of the contact pairs and surface components. You can view the contact normal and reverse it with the buttons provided in the Contacts table. These features are also available for surface elements.

Create and Edit Contact Pairs

Create 2D and 3D contair pairs, and define contacts.

You can create 2D and 3D contact pairs with the Surface to Surface and Point to Surface options. You can also define contacts using the Pilot Node option.

1. From the Tools menu, click **Surface Manager**.
   The **Ansys Surface Manager** opens and the Contact tab is displayed by default.
2. Click **New** to create a new contact pair.
   The **Create New Contact Pair** dialog opens.
3. Under Creation method, select **Flexible** or **Pilot node**.
4. In the Contact type field, select **2D** or **3D**.
5. Under Create from, select whether to create a **surface to surface** contact pair or a **point to surface** contact pair.
6. Under Symmetric Contact, select **Yes** or **No**.

   To create two pairs of symmetry contacts, click **Yes**. In order to create a symmetry pair, the selected elements must meet all of the element configuration requirements.

   - Contact elements in the first pair will be target elements in the second pair.
   - Target elements in first pair will be contact elements in the second pair.
   - Different properties and ET types will be created for the second pair. You can update the contact pair with desired properties and contact options later.
7. Click **Pick Target** to select the target body components.
8. From the panel area, click the **comps** selector.
9. Select the components that belong to the target component elements.
10. Click **select**.
11. Click **proceed**.
12. In the **Target Elements Selection** dialog, click the **Elements** selector.
13. In the panel area, verify that the elems selector is active, then select the target elements of the contact pair.
14. Click **proceed**.
15. In the **Target Elements Selection** dialog, click **Next**.
    The **Target Component Details** dialog opens.

Note: This dialog displays a default name and color for the target component, which can both be modified. You can also modify the ET Type name and the ID of the target component, as well as the values of the key options. Once **Save Keyopts** is clicked, all of your changes will be applied to the target element. The next time you create a new contact pair, the options you selected in the dialog will be saved and you will not need to apply them again.

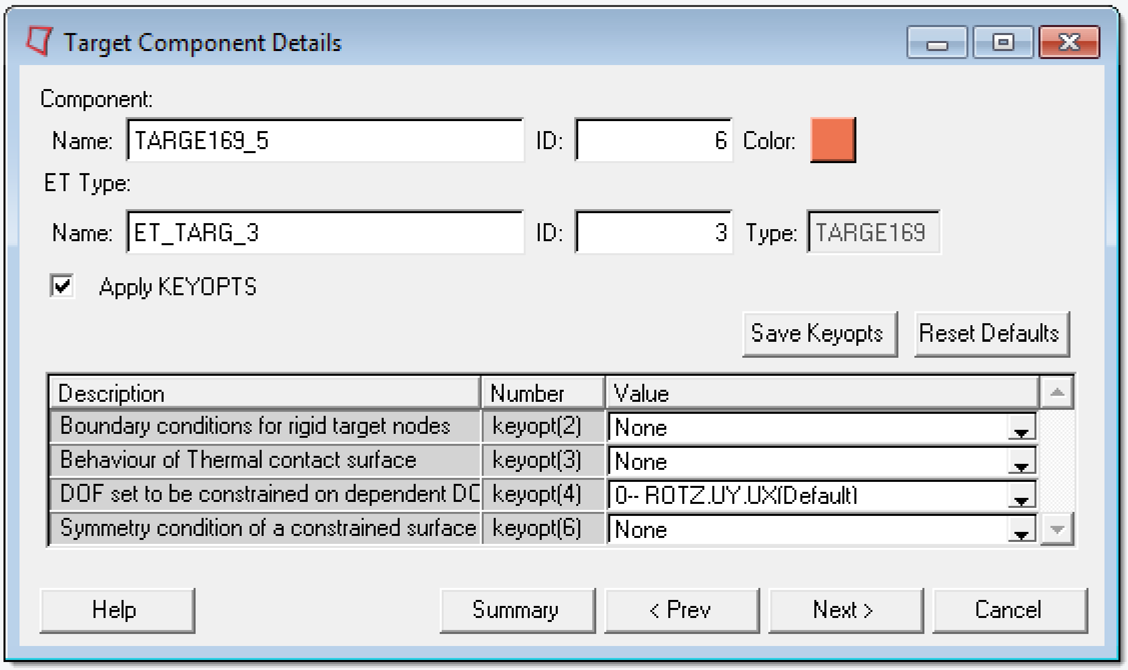


Figure 553:

16. Click **Next**.
17. In the **Contact Components Selection** dialog, click the **Components** selector.
18. In the panel area, click the **comps** selector.
19. Select the components to assign as the target body.
20. Click **select**.
21. Click **proceed**.
22. Click **Next**.
23. In the **Select Contact Elements** dialog, click the **Elements** selector.
24. In the panel area, verify that the elems selector is active, then select the elements to assign as the contact surface elements.
25. Click **proceed**.
26. Click **Next**.
    The **Contact Component Details** dialog opens.

> **Note:** This dialog displays a default name and color for the contact component, which can be modified if desired. The ET type and the ID of the contact component can also be modified.

27. Click **Next**.

28. In the **Contact Property** dialog, enter the values for items shown. You can create a new property, or create a new one based on an existing property, which helps expedite the process. Click **Save real constants** to save the property values you entered. In the future, when the **Contact Property** dialog opens, the values you saved will appear by default.

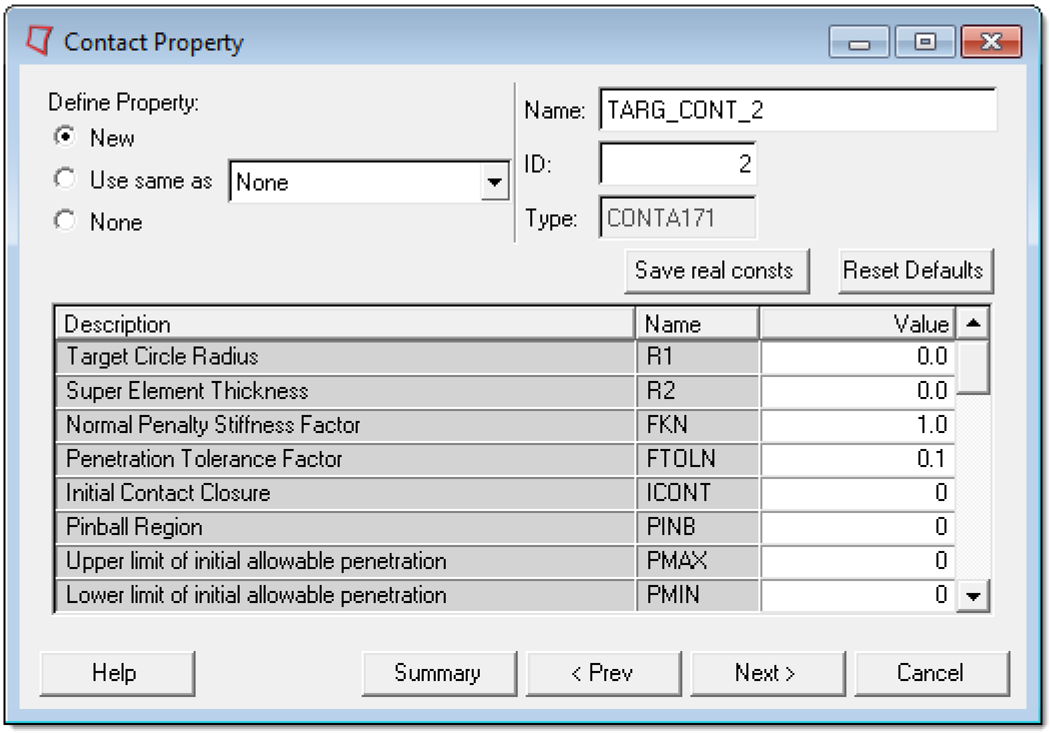


Figure 554:

29. Click **Next**.

30. In the **Contact Material** dialog, click **New** to define a new material or select **None** to skip defining a material.

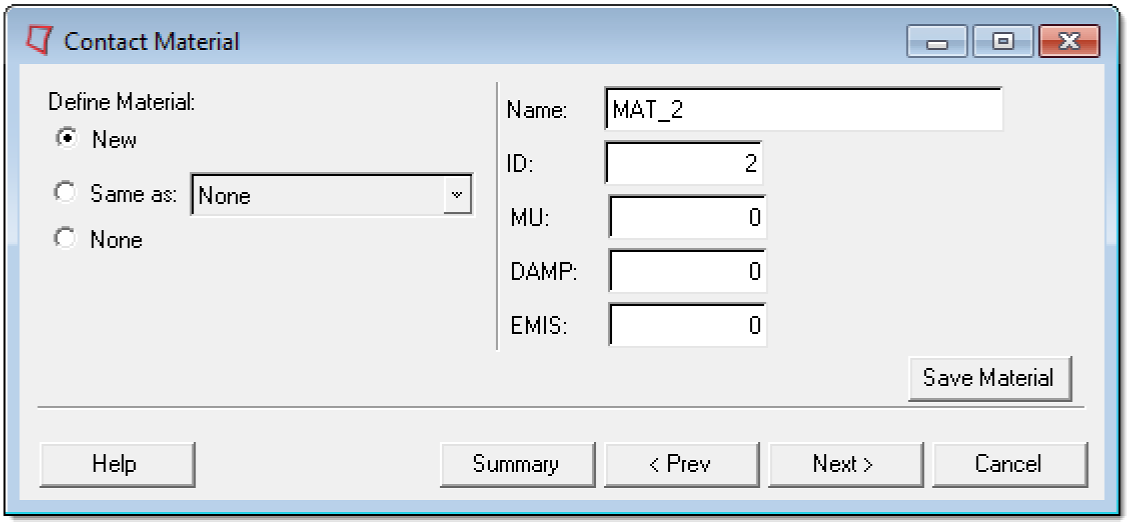


Figure 555:

31. Click **Next**.
    The **Ansys Surface Manager** dialog opens and displays a summary of the target and contact elements. The contact pair you created is displayed.
32. To exit the **Ansys Surfaace Manager**, click **Close**.
33. To repeat this process, click **Restart**.

Create and Edit Surface Elements

Surface elements can be created on the edges of plane elements (2D) or on faces of solid elements (3D).

Using the **Ansys Surface Manager**, you can create the thermal surface elements SURF151 and SURF152, the structural surface elements SURF153 and SURF154, and the radiosity elements SURF251 and SURF252.

1. From the Tools menu, click **Contact Manager**.
2. In the **Ansys Contact Manager**, click the **Surface Elements** tab.
   A list of existing surface element components displays.
3. Click **New** to create new surface elements.
   The **Surface Effect Elements** dialog opens.
4. From the Element Type field, select the type of surface elements to create.
5. Select whether to create **2-D Surface** or **3-D Surface** elements.

> **Note:** 2D Surface creates surface elements on the edges of a plane element, and 3D Surface creates surface elements on the faces of solid elements.

6. To create surface elements with extra nodes, select the **With Extra Nodes** check box.

> **Note:** With Extra Nodes is only available for thermal surface elements.

7. If you selected Radiosity Elements as the Element type, you must specify an Emissivity and Enclosure number value to apply to the edges and faces of base elements.

> **Note:** Radiosity elements require radiation loads to be applied on the faces of the elements over which the skin elements are created. The **Contact Manager** automatically creates these loads with the emissivity and enclosure values that you specify.

8. Click **Pick Components**.
9. In the panel area, click the **comps** selector.
10. Select components on which surface elements will be created.
11. Click **select**.
12. Click **proceed**.
13. In the **Elements Selection** dialog, click the **Elements** selector.
14. In the panel area, verify that the elems selector is active, then select elements on which surface elements will be created.
15. Click **proceed**.
16. In the **Elements Selection** dialog, click **Next**.
17. In the **Nodes Selection** dialog, click the **Nodes** selector.

> **Note:** The **Nodes Selection** dialog only appears if you selected the With Extra Nodes checkbox.

18. In the panel area, verify that the Nodes selector is active, then select extra nodes to create surface elements with.

> **Note:** A maximum of two nodes can be selected.

19. Click **proceed**.
20. In the **Nodes Selection** dialog, click **Next**.
21. In the **Surface Element Component Details** dialog, select the appropriate key options.

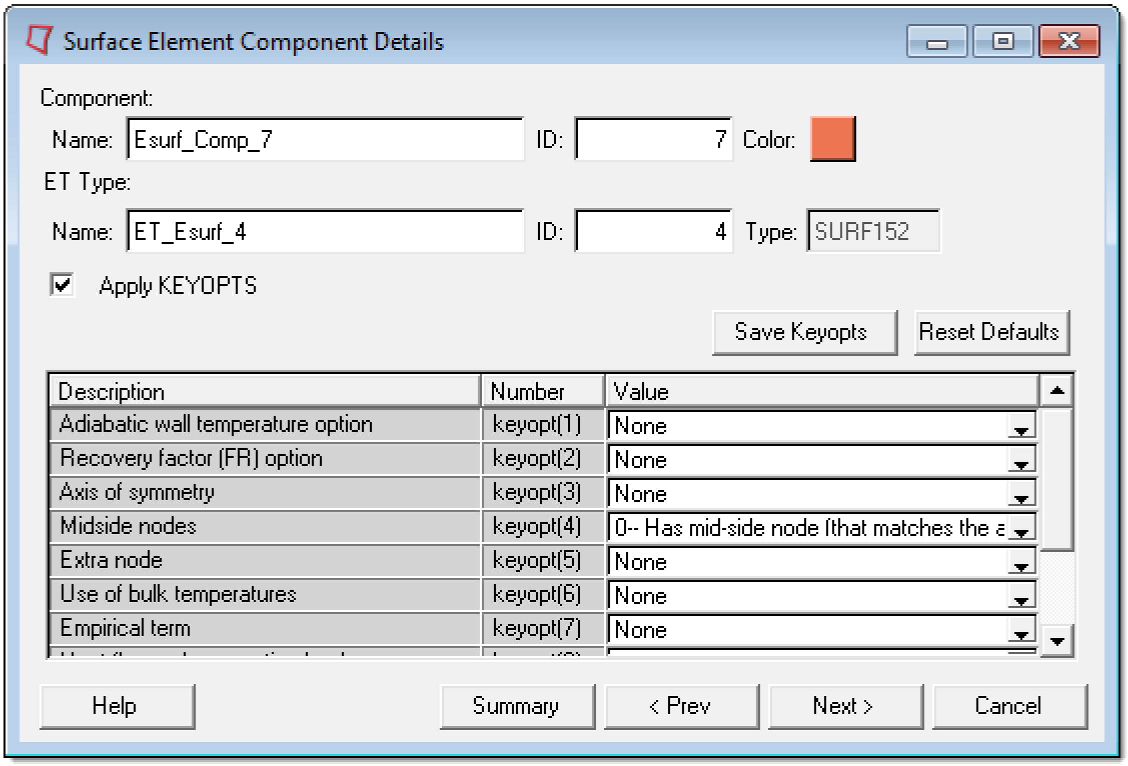


Figure 556:

22. Click **Next**.

23. In the **Contact Property** dialog, define values for the element real sets (properties).

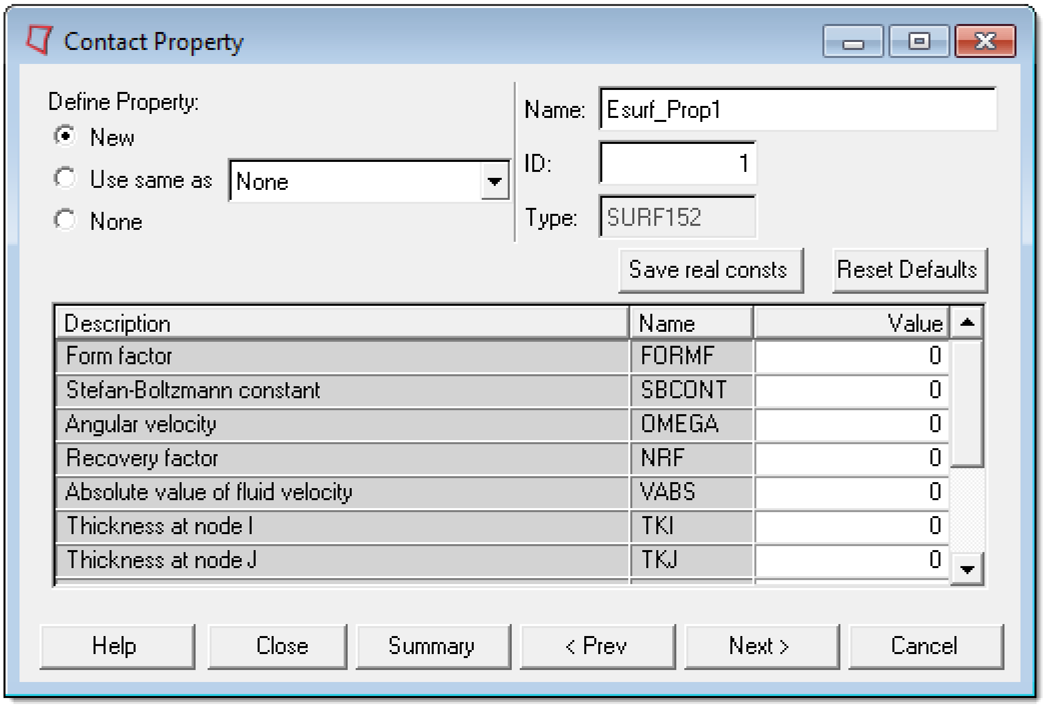


Figure 557:

24. Click **Next**.

25. In the **Surf Material** dialog, click **New** to create and define a new material or select **None** to skip defining a material.

> **Note:** The **Surf Material** dialog only appears if the surface elements being created needs material properties defined.

26. Click **Next**.
The **Ansys Surface Manager** dialog opens and displays the surface elements you created.

27. To exit the **Ansys Surface Manager**, click **Close**.

28. To repeat this process, click **Restart**.

Auto Contact - ANSYS Interface

Use the ANSYS **Auto Contact** to quickly and easily create interactions between several parts of your model.

Based on a proximity distance, **Auto Contact** will search the model and automatically define contact elements from identified components. The interactions and surfaces are placed into a temporary browser, where you can review the pairs and make adjustments as needed.

- Each contact element pair will be created with a contact and target element on each selected element surface.
- Contact element options (ET types) and contact property (REAL sets) are simultaneously created with the contact pair assigning default values. You have to edit these options and properties using

the **Contact Manager**'s edit options if you want to assign different values other than the default. Similar properties (REAL sets) are shared by both contact and target elements.

- Material cards are also generated during the contact pair creation. You have to edit the material card to set the correct material property values.
- Currently, only surface to surface 3D contact elements can be created. Future releases will be enhanced to add other contact element types.

The **Auto Contact** dialog contains the following buttons:

Table 156:

Button	**Action**
Find	Searches the model for interacting components.
Cancel	Closes the **Contact Pair** dialog without updates.
Remove Selection icon	Removes selected components from the table. You can use the Control and Shift key to select multiple items in the table.
Review Selection icon	Highlights the selected component in the graphic area. All other components are grayed out. You can use the Control and Shift key to select multiple items in the table. Right-click to return the model to normal display.
Help icon	Opens the **Auto Contact** online help.

Set Up an Auto Contact Run

1. Load the ANSYS user profile.
2. From the Tools menu, select **Ansys Surface Manager**.
3. Click **Auto**.
   The **Auto Contact** dialog opens.
4. In the Contact Type: field, select the type of contact pair to create.
5. Click the yellow **components** button to select your components.

   The components are automatically placed in the Component table in the **Auto Contact** dialog. The proximity distance is the maximum distance between two selected components. When you create the pair, any surfaces that are farther away than the value entered here will not be created as a contact pair. The default value is zero.

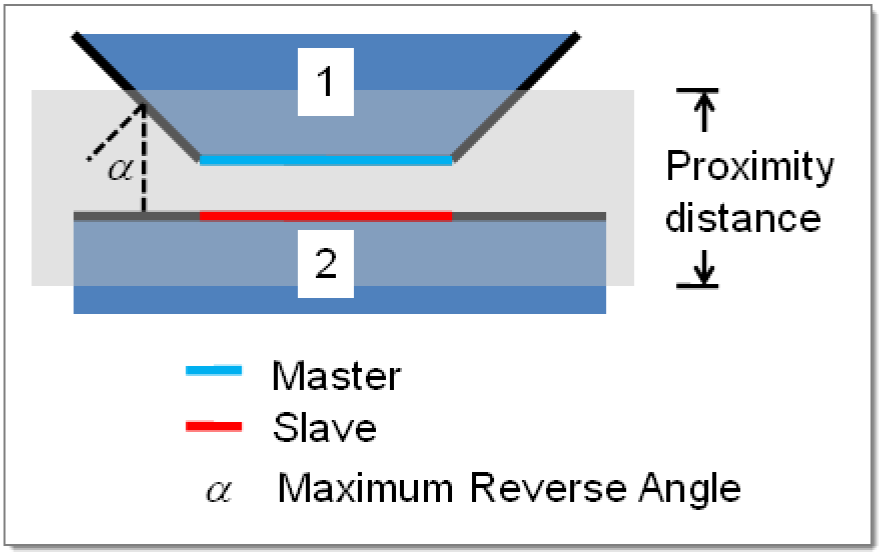


Figure 558:

6. In the Maximum reverse angle field, enter a value.

   If the angle between two normals of elements or element faces exceeds this value, the element will not be added to the master or slave surface.
7. Click **Find**.
   The status bar activates and the **Auto Contact** browser opens.
8. Use the **Auto Contact** browser to make any necessary adjustments to the interface and surfaces. When finished modifying, click **Create**.
   The interfaces and surfaces marked as accepted are created. The Contact Manager window reopens with the new information listed.

Auto Contact Browser

The **Auto Contact** Browser provides options for viewing and modifying the contact pairs identified in the auto contact process.

The **Auto Contact** Browser contains the following columns:

Table 157:

Column	Description
Name	Lists the name of the interfaces, surfaces and surface interactions that were assigned. Underneath the interface name are the temporary surfaces included in that interface. Red indicates a slave surface, and blue indicates a master surface.
Accept	When the Accept box is checked, the Interface will be included in the creation process.

Column	Description
Color	Color assigned to the interaction and surfaces
ET Type	ET Type
Real Set	
Mat	Material assigned

The **Auto Contact** Browser contains the following icons:

Table 158:

Icon	Description
Options	Opens the **Options** dialog. Enter a new feature angle or customize the transparency for a selected entity. Click **OK** when finished.
Highlight Elements	Highlights the elements stored in selected entities in the graphics window. You can use the Control and Shift key to select multiple items in the table.
Review Elements	Review of elements stored in the selected entities. Elements are highlighted by color; all other components are grayed out. You can use the Control and Shift key to select multiple items in the table. Review and Highlight are mutually exclusive. It is also possible to switch both options off. This is helpful when working with big models.
Fit View to Elements	Automatically zooms in to the elements stored in the currently selected items.
Display All Elements	In combination with the **Highlight Elements** or the **Review Elements** option, current contents remain unchanged on the screen.
Display Components with Elements	Highlights or reviews the elements referred by an interaction or surfaces and shows the components they belong to. All other components will be masked.
Display Only Elements	Only elements are highlighted or reviewed. The rest of the component and other components will be masked.
Select Elements Manually	Opens the Element selection panel so that individual elements can be added/ removed manually. Click **proceed** when finished.

Icon	Description
Add by Adjacent	Adds the elements adjacent to the surface to the selected surface. Right-click to undo one time.
Add by Face	Adds the adjacent face to the selected surface. Right-click to undo one time.
Recheck	Opens the **Auto Contact** dialog to recheck the select interfaces. Recheck will either add more contacts to the existing contacts for modify the existing ones. You can select interfaces from the browser, and the GUI will automatically populate the components that the interaction was based on. This helps modify an existing interface.

Modify Auto Contact Entities

Right-clicking on an item in the Auto Contact Browser displays a context menu which offers options for modifying the surfaces and contact pairs.

The following options are available in the **Auto Contact** browser's context menu:

Table 159:

Option	Description
Rename	Rename an existing entry.
Delete	Delete items from the browser.
Swap Master - Slave	Switches the surfaces identified as master and slave. When selected, you will see the surfaces flip from the master/slave positions in the browser. Select multiple entities by using the Control and Shift keys when clicking on entities.
Edit Faces	Opens the elements selection panel where you can select and deselect the elements to include on the face of the surface. You can also manually edit the faces of the surfaces using this panel.
Add by Adjacent	Adds adjacent elements to the selected surface.
Add by Face	Adds all elements to a selected surface, until the feature angle exceeds the value. The feature angle can by set by clicking the **Options** icon.
Accept All/None	Accept or reject all items in the **Auto Contact** browser.

Option	Description
Reverse	Reverses the current selections in the Accept column.
Expand All/Collapse	Expands or collapses folders in the **Auto Contact** Browser.

LS-DYNA Utility Menu

The LS-DYNA Utility menu contains shortcuts and tools that can help simplify LS-DYNA tasks. Set the user profile from the User Profiles option of the Preferences menu.

The LS-DYNA Utility menu on the Utility tab is automatically loaded when you select the LS-DYNA user profile.

The LS-DYNA user profile sets the FE input reader to DYNA KEY and loads the `dyna.key` (ver 971) FE output template and LS-DYNA Utility menu. Also, the graphical user interface becomes LS-DYNA focused, renaming or removing some panels and/or options. The entire ALE Setup is available only when the LS-DYNA user profile is loaded.

Tools Menu

The LS-DYNA Utility menu contains a Tools menu in addition to the standard HyperMesh Utility menu. This menu includes special time-saving setup macros and other features that are specific to an LS-DYNA analysis.

Error Check

The **Error Check** dialog checks your LS-DYNA deck for potential problems with components, properties, materials, rigids, joints, boundary conditions, and other entities and reports them on-screen.

The report identifies the problem entity by ID, describes the error, and then enables you to isolate the entity in the model and quickly make changes.

1. Click **Error check** on the LS-DYNA Utility menu.
   The **LsDyna Error Check** dialog opens.
2. Select the types of errors for which you want to search and click **Check**.

   When the check is complete, the results appear on the Errors tab of the dialog. Each error in the list is a hyperlink that, when clicked, highlights the affected visualizations in the model and opens the relevant card image or panel for correcting the error.
3. Systematically click on each error in the list, correcting them as you go.
4. On the Settings tab, click **Check** again to verify that the errors were corrected.
5. If you want to restore the full view of the model including all components, click **View - Show full model** on the Errors tab of the dialog. To return to the previous view, click **View - Restore View**.
6. Use the Options menu button to update or save settings for the **Error Check** dialog.

You can specify minimum and maximum values for the material check and a maximum value for the distance of a constrained extra node to its part.

7. To save the current settings, choose **Save Settings...** from the Options menu button and specify a file name and location. You can also load previously-saved error check settings.
8. Click **Close** to exit the **Error Check** dialog.

Part Info

The **Part Info** macro summarizes a part's statistics in a dialog.

1. To start the macro, click **Part Info** on the Utility menu.
2. Click **component** on the main menu area to select a component or click a component in the graphics area to select it.
3. Click **proceed**.
   The **Part Information** dialog opens, which lists the part ID, name, thickness, and material type.
4. To view additional statistics about the part, click the **More Details** tab.
5. To display statistics for a different part, select the part in the graphics area or the **components** selector and click **proceed** again.

> **Tip:** Click the middle mouse button instead of the **proceed** button to quickly select components.

Clone Part

The **Clone Part** macro enables you to quickly create a new part from the properties of an existing part.

1. To start the macro, click **Clone Part** on the Utility menu.
2. Select the existing part on which to model the new part by clicking the **...** button.
   A dialog opens listing all of the existing components.
3. Select a component from the list and click **OK**.
4. Type a name for the new part in the New Part field and click the color icon to select a color for the component.
5. Select whether to duplicate the material and section properties or to re-use the original material and section properties.

   Duplicate means that a new material and section is created (the name is suffixed with .n version numbers and new IDs are used) with the same properties, while Reuse refers to the same material and section as the original.
6. Select whether to duplicate the elements.

   Duplicate elements will make a copy of the elements from the selected part to new part in the same location.
7. Click **Create** to either create or create and edit the card.

Create Part

Use the **Create Part** macro to create components on-the-fly.

1. To start the macro, click **Create Part** on the Utility menu.
2. Type a name for the new component in the Part name field and select a color by clicking the adjacent color icon.
3. Select a section in the Section field by choosing **Create New** (create a new section), **Same As** (create a new section based on an existing section), or **Model** (select an existing section) from the selection menu.
4. Select a material for the component in the Material field by the same method as described above for the Section field.
5. Click **Create** to either create or create and edit the card.

Convert to Rigid Macro

Use this macro to convert deformable parts of an LS-DYNA model to rigid.

1. Click **Tools** in the Utility menu.
2. Click **Convert to Rigid**.
3. Select the elements to convert to rigid and click **proceed**.
4. Select an existing rigid component in the model for merging the newly created rigid body and click **Proceed**.
5. Click **return**.

The **Convert to Rigid** macro performs the following steps when the selected elements are converted to rigid.

1. For the selected elements, a check is performed on the comps for rigid (MAT_RIGID or matl20) or deformable materials (all, except matl20). If deformable materials exist, rigid materials (MAT_RIGID) are created with the properties from the original deformable materials. A check is performed for rigid materials that are already defined. If rigid materials are found, the comps and rigid materials are retained.
2. Comps located partially within the window are split into two comps. The new comp has the same property (section ID) but new material (Material ID). For example, if A-pillar is partially within the window, then a new comp A-pillar_rig is created. A-pillar_rig is updated with newly created material.
3. All the spotwelds and rigids located entirely within the window are removed. For example, *CONSTRAINED_NODAL_RIGID_BODY_option, *CONSTRAINED_NODE_SET, *CONSTRAINED_SPOTWELD, and *CONSTRAINED_GENERALIZED_WELD_option.
4. For spotwelds that are connected from the deformable body to the rigid body, an extra node is created and referenced by the master rigid body.
5. A check is performed to detect joints located partially or entirely within the window. Detected joints are deleted.

6. A check is performed to detect springs located partially or entirely within the window. Detected springs are deleted.
7. A check is performed to detect seatbelt elements (seatbelt elements, Retractor, Pretensioner) located partially or entirely within the window. Detected seatbelt elements are deleted.
8. Master and slave comps are defined, for example, CONSTRAINED_RIGID_BODIES. You are prompted to select a comp for master rigid body. A slave set is created with the newly created rigid bodies, except the master rigid body comp.

9. A message is displayed when the conversion is complete.

Convert to Rigid Flow Chart

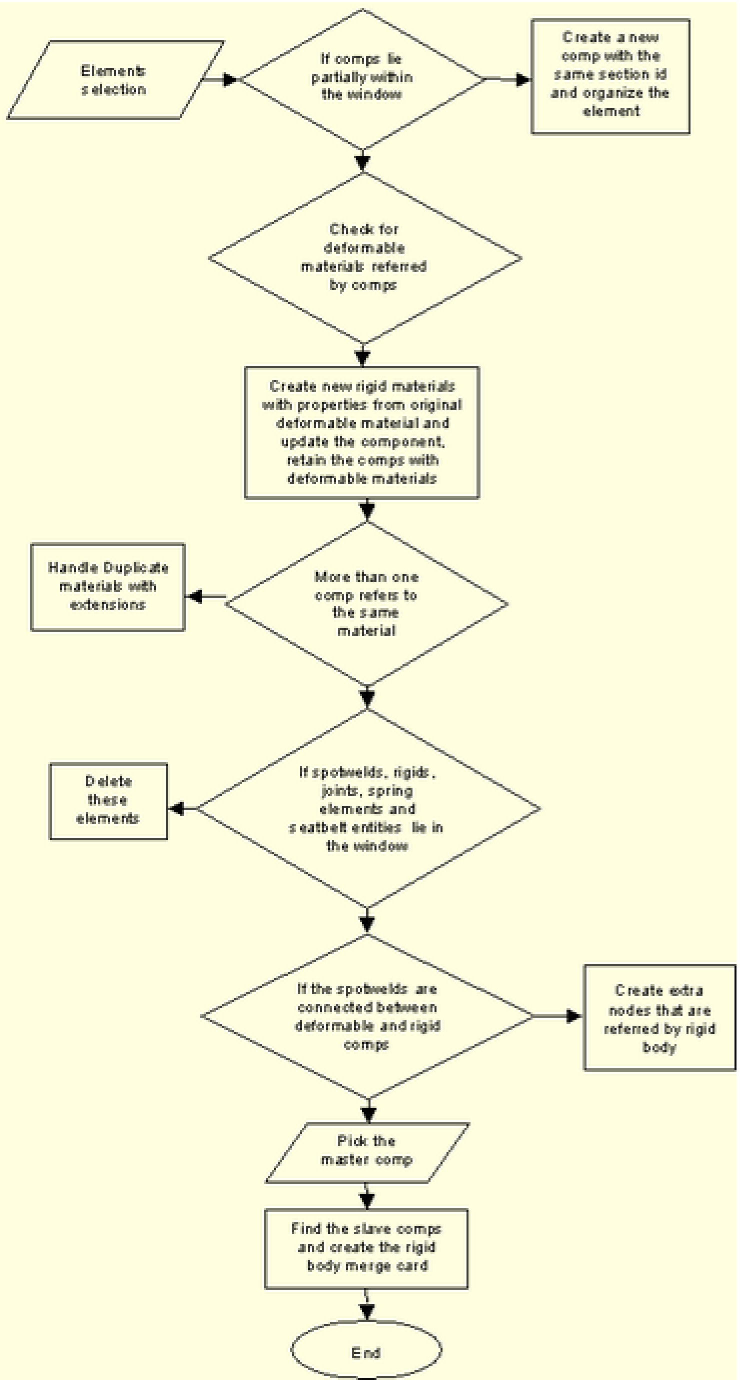


Figure 559:

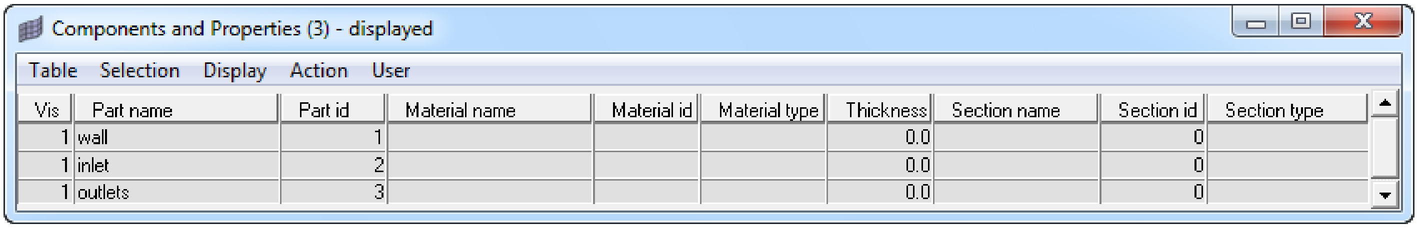


Figure 560:

The table contains a variety of tools that allow you to review, edit, and update the model. The essential features are:

- LS-DYNA components with various associated properties and materials are listed in separate columns.
- You can select the column types from a set of available options.
- There are two modes of operation: review and editable. Use the review mode to review the component information without changing any values. Use the editable mode to change values for the selected components.
- There are enhanced selection, review, display, and filter options for components.
- Components can be sorted according to any available column.
- The current configuration is saved automatically to a file at the end of a session and recalled on reload. You can also save and load a configuration file.
- The table data can be exported in CSV and HTML formats.
- Right-click on the table to display menu options. All pull-down menu options are also available using a right-click.
- Columns can be moved or swapped by holding the left mouse button on a column title and dragging it to the desired location.
- Columns can be resized by positioning the cursor along a column border, pressing the left or right mouse button, and dragging the border to a new position.
- The Shift or Control key combined with a left-click can be used to select multiple rows.

The following tools are available in the LS-DYNA **Component Table**:

Table

Table 160:

Refresh	Regenerates the table with all the parts in the model.
Editable	Sets the table mode to editable mode, allowing you to change values for the selected components.
Filter	Enables the filtering GUI.
Configure	Specify the number and type of columns listed in the table.

Save	Saves the information listed in the table in CSV or HTML format.
Quit	Quit the table function.

Selection

Table 161:

All	Selects all rows or parts.
None	Selects none or deselects parts/rows that were previously selected.
Reverse	Reverses the selection.
Displayed	Selects the rows or the displayed parts.
User	User graphic interaction to select parts.

Display

By default, the table is invoked with only the displayed parts. You can refresh the table to show a new part being displayed or use one of the following display commands.

Table 162:

All	Displays all the components in the model.
None	Turns off every component displayed.
Reverse	Reverses the display of the part.
Show selection	Displays the components of the selected rows.
Show only Selection	Displays only the components of the selected rows.
Hide selection	Hides the components of the selected rows from the display.
By Material	Displays components sorted by material.
By Properties	Displays components sorted by properties.

By Thickness	Displays components sorted by thickness values.

Action

Table 163:

Delete Selection	Deletes selected rows (parts) from the model.

User

Table 164:

Set MatDB Path...	Opens a dialog on which you can set the location of an external database of material definitions.
Refresh Material List	Updates the list of available materials in the **Component Table**.

Editable Mode

When the **Component Table** is in editable mode, you can change values for all selected components at the same time. Select the **Table** > **Editable** option to open the **Component Table** in editable mode. Cells with a white background can be manually edited. When you click on an editable cell, it is selected with a cursor. Once a cell is selected, enter a value and press **Enter**.

If you want to assign the same value to multiple components at once, select the column type and value from the Assign Values menu and click **Set**. All the selected components will be updated with the assigned values.

Filter

The **Component Table** supports advanced filtering based on available columns. The **Table** > **Filter** menu option opens the **Filter** dialog.

You can write any valid string with a wildcard (*) in any of the available column types and click **Apply** to filter the table. For example, if you want to show all components that start with letter 'c' and use material 'steel', you can use the dialog as shown below.

> **Note:** The filter strings are case sensitive.

Show All turns off the filtering and displays all the components. Select the **Table** > **Configure** > **Filter on top** option to keep the **Filter** dialog posted after clicking **Apply** or **Show All**. Otherwise, it closes.

Configure Columns

Column types can be selected from the **Table** > **Configure** > **Columns** menu option. The table displays only the selected columns. The available columns types are:

Table 165:

Vis	Visualization status. 1 = display on, 0 = display off.
Part name	HyperMesh name of the component (maximum 32 characters).
Part id	HyperMesh ID of the component.
Material name	Material name associated with the component.
Material id	Material ID associated with the component.
Material type	Material type associated with the component.
Thickness	Thickness of elements specified in *section_shell.
Section name	*Section name associated with the component.
Section id	*Section ID associated with the component.
Section type	Type of the *Section associated with the component.
Color	Component color.
Int points	Number of integration points specified for the *Section_shell.
HGID	Hourglass ID associated with component.
Elem form	Element formulation for the *section of the component.
Elems	Number of elements in the component.
Nodes	Number of nodes in the component.
Mass	Total mass of the component.
cg_x	Center of gravity for the x coordinate.
cg_y	Center of gravity for the y coordinate.
cg_z	Center of gravity for the z coordinate.

Components All or Displayed mode

The **Component Table** lists components in two modes: All or Displayed. If All is selected from the **Table** > **Configure** > **Components** menu, the table will list all the components in the model.

If Displayed is selected, only the visible components will be shown. Blank components are not shown in the Displayed mode even though their display status is on.

Material Table

Use the LS-DYNA **Material Table** to easily create and edit materials.

To access the Material Table, click **Material Table** on the Utility menu. All the existing materials are retrieved and populated in the table.

From the **Material Table**, you can also merge identical materials, search for duplicate materials, and change the properties of materials.

When you first display the **Material Table**, all materials are listed in the table, showing the material's ID, name, type, description, list of components in which it is used, and the RHO, E, and Nu values.

Materials in the table can be selected by clicking the row, which is then highlighted in blue. Many functions are performed by selecting materials in the table and choosing an option from the context menu or clicking a button below the table. Shift-click and Control+click can be used to select multiple rows. Refer to the links below for details about using the **Material Table**.

Customize Views of the Material Table

The **Material Table** initially lists all existing materials, but you can sort and filter the list to more easily identify materials that you want to work with.

Each of the columns in the table can be used to sort the list. Click the column heading to sort by that characteristic, such as ID number or material type. To view only materials of a particular type, select that type in the Material type drop-down at the top of the window. For example, if you want to identify materials that are not used so you can delete them, you can click the **Comp used** column heading to quickly group together all materials that contain the value "No", which indicates that none of the components use the material.

Note: To view all material properties in the table, select a material type from the drop-down. When all material types are shown in the table, only the RHO (density), E (Young's modulus), and Nu (Poisson's ratio) properties appear. However, when a particular material type is displayed, all the relevant properties for that material type also appear in the table, as shown in the image below.

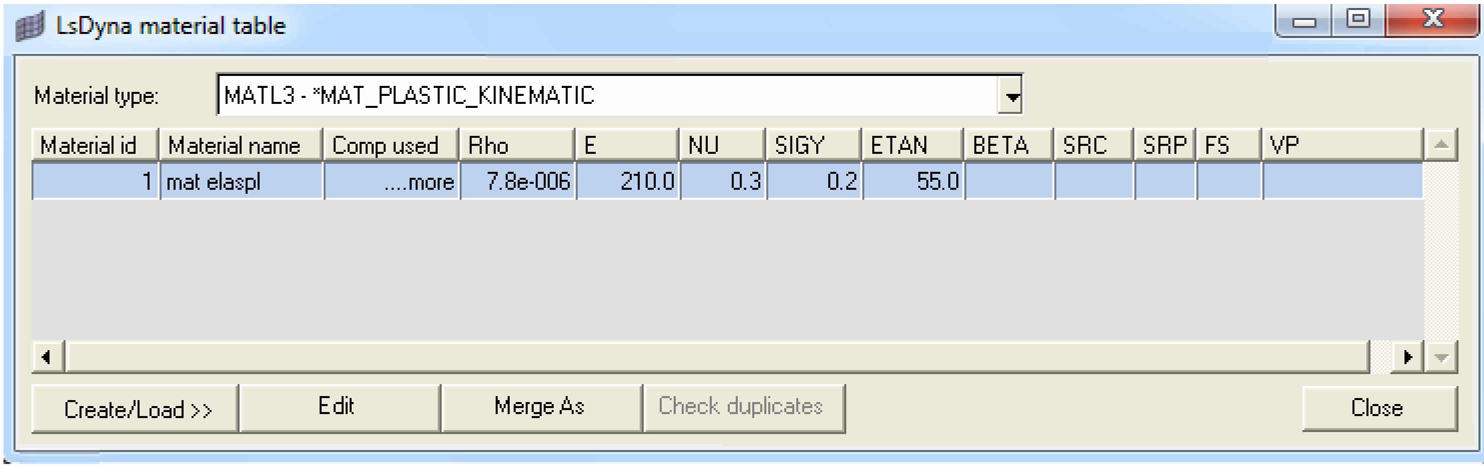


Figure 561:

The **Material Table** also enables you to view the model's components based on the material used. These options are available by selecting **Display** from the menu that appears when you right-click anywhere in the table. Options include:

- Viewing only the selected materials
- Hiding the selected materials
- Viewing all or none of the materials
- Adding the selected materials to the current display
- Reversing the current display option

Once you make your selection, the corresponding components appear or become hidden in the graphics area.

Create, Edit and Load Materials

You can create, edit and load materials all from within the **Material Table**.

Materials can be added or modified with the Create/Load and Edit buttons or by selecting the same options in the menu that appears when you right-click anywhere inside the table. To save time, you can choose the **Same As** selection to begin creating a material with the same properties as the currently-selected material in the table.

When you create a new material, you specify a name and the type of material. The materials are conveniently organized into categories, including groups of recently used materials and only materials that exist in the model. These categories are further listed by the LS-DYNA keyword or type identifier, as shown in the following image.

You can add the material to the table immediately by clicking **Create** or by going to the Card Image panel to specify its properties by clicking **Create/Edit**.

At any time you can select a material in the table and click **Edit** to open the material's card image. In the card image, you can modify values for the keyword's variables. In addition, the material's load curve appears in a pop-up graph, as shown below.

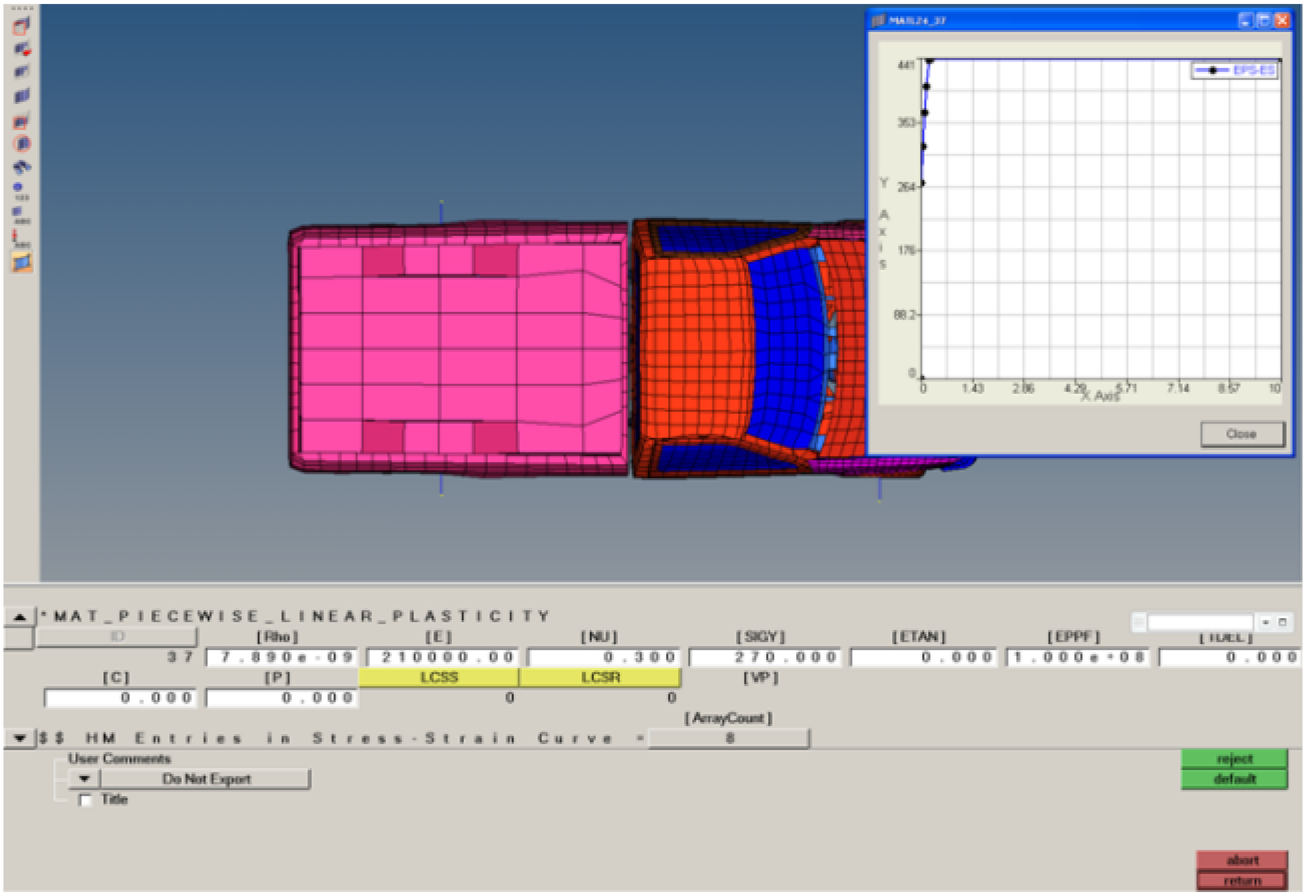


Figure 562:

Manage Materials

In addition to viewing, creating, modifying, and deleting materials, you can also identify duplicate materials, merge like materials into one, and rename materials.

The names of materials and the material IDs can be edited directly in the table. All other values must be edited with the Edit button, which opens the card image.

Materials that have the same properties can be identified using the Check duplicates button. This feature, which is only available when all materials are displayed in the table, finds all materials that have identical properties and returns them in result sets. You can then select each result set to view the matching materials. Optionally, you can merge the duplicate materials into one material using the Merge button, which is the same feature as described in the following paragraph.

When you select multiple materials from the table, you can merge them into one of the selected materials using the Merge As button. Typically this action is performed on materials with like properties to simplify a model, although it can be performed on dis-similar materials with all selected materials taking on the properties of one of the materials. When materials are merged into one, the remaining materials still exist and appear in the table, but do not have any components assigned to them.

Sort Materials

1. Click the column heading of the criteria by which you want to sort.
2. Click the column heading again to list the materials in reverse order.

See Customize Views of the Material Table to learn about other ways to filter the list of materials in the table.

Create New Materials

You can create a new material, or create a new material based on an existing material.

Create a New Material

1. Click **Create/Load** and select **New** from the menu.
   New fields appear at the bottom of the **Material Table**.
2. Type a name for the material in the New Material Name field.
3. Select a material type from the drop-down list.

   The list expands to categories of material types, and also sorts them by keyword or material ID. You can view the complete list of material types under the All category.
4. Click **Create/Edit** to open the material card image to specify the properties, or click **Create** to add the material to the table without immediately specifying any properties.
5. Click **return** to exit the Create/Load mode.

Create a New Material Based on an Existing Material

1. Select a material in the table that you want to use as the basis for a new material.
2. Click **Create/Load** and select **Same as** from the menu.
   New fields appear at the bottom of the **Material Table**. The material you selected appears in the Selected material field.
3. Type a name for the material in the New Material Name field.
4. Click **Create/Edit** to open the material card image to specify the properties, or click **Create** to add the material to the table without immediately specifying any properties.
5. Click **return** to exit the Create/Load mode.

Edit a Material's Properties

1. Select a material in the table that you want to edit.
2. Click **Edit**.

The card image for the material appears and, if applicable, the load curve appears in a pop-up window.

3. Modify values in the card image and click **return** to go back to the **Material Table**.

Merge Materials

1. In the table, select the materials you want to merge.

   Use Shift+click to select multiple, consecutive rows and Control+click to select non-consecutive rows.
2. Click **Merge As**.
   The **Material Table** expands to include new fields for merging materials.
3. Select the material ID to use as the new material in the Retain material(id) field.
4. Click **Merge**.
   The components for each of the selected materials are merged into the material you selected. The remaining materials still exist and are listed in the table, but they are not assigned to any components.

Find Duplicates

1. Ensure that ALL is selected in the Material type field.
2. Click **Check duplicates**.
   The **Material Table** expands to include new fields for handling duplicate materials.
3. Choose a group number from the View materials in duplicate group field.
   The materials in that group appear in the table. The results for the duplicates check are divided into consecutively numbered groups of the same material type.

You can easily merge the duplicate materials using the Merge button. See Merge Materials for steps on using the merge feature.

To view another result group of duplicate materials, select another group number from the View materials in duplicate groups field. That group's list of duplicate materials appears in the table.

View the Load Curve for a Material

1. Select a material in the table for which a load curve ID has been defined.
2. Click **Edit**.
   The load curve appears in a pop-up window.

Export Data from the Material Table

1. Select a material type or **ALL** from the Material type field to export only materials of a particular type or all materials, respectively.
2. Right-click anywhere in the table and select **Save** and then **CSV** for comma- or semicolon-separated values or HTML for an HTML-based table.

The **Select output file** dialog opens.

3. Browse for or type a name in the File name field and click **Save**.
   The file containing material data is saved in the location you specified.

MADYMO Utility Menu

The MADYMO utility contains utilities, tools, macros and shortcuts to display options.

The MADYMO Utility menu (`madymo.mac`) is loaded when you open the MADYMO user profile. The menu and its utilities are fully customizable.

The MADYMO Utility menu contains two pages, Tools and Define Entities. Both menu pages contain the following Display options used to control the display of entities in the graphics window.

Table 166:

Option	Description
Body	Turns on and off all ellipsoids, planes, cylinders, and joints.
Elems	Turns on and off all FE elements.
Triads	Turns on and off all coordinate systems.
Shading	Set to visualization mode for the entire model. Four modes are available: **0** Performance graphics wireframe **1** Shaded **2** Shaded with mesh lines **3** Shaded with feature lines
Only Comps/MBs	Turns off all entities except elements, ellipsoids, planes, cylinders, and joints.
Clear Temp Nodes	Removes all temporary nodes (a.k.a. 3D location markers)
Work in Meters	Reduces display size of coordinate system and boundary conditions for modeling in meters.
Autocolor	Colors all ellipsoids, planes, cylinders, and joints based on their rigid body reference. Also colors FE elements by part card.

The Tools page contains a series of utilities and tools:

Table 167:

Option	Description
Set Light Source...	Opens a window with button-based light source options. Click the icon button that corresponds to your preferred direction of the light source, and click the button that corresponds to your preferred level of specularity. Then click **Close**.
Elems to Ellipsoids	Converts linear elements to ellipsoids. Each element becomes a new body with an ellipsoid. A joint is created between each body. The organize, delete, and mbs joints panels can then be used to move the created ellipsoids to a single rigid body, delete extra bodies and joints, or change the joint type.
Rotate Systems	Used to rotate coordinate systems about their axes.
Resize Ellipsoids	Expands/shrinks ellipsoids about their individual axes.
Mesh Ellipsoids	Used to mesh ellipsoids. Mesh is created to represent the actual geometry of the ellipsoid.
Display Syst IDs	Turns on numerical ID display for specified coordinate systems. Useful for seeing which coordinate system is selected when selecting coincident coordinate systems.
Apply JNTPOS...	Applies contents of JNTPOS file to loaded model. Opens the file browser for selecting the JNTPOS file and applies the Euler parameters for the last time step on all joints contained in the file.
Body Properties...	Opens an editable table of all the rigid bodies in the model listing each body name, center of gravity, mass, and moment of inertia. Also contains non-editable fields for reviewing the body ID and parent body.

Both the Define Entities and Tools pages contain Display options, while the Define Entities page also contains buttons for creating new coordinate systems and location markers in 3D space.

The New Cord Sys options allow you to create the following new coordinate systems:

Parallel Global
Creates new coordinate systems at specified nodes. The created systems are oriented parallel to the global system.

Parallel Local
Creates new coordinate systems at specified nodes. The created systems are oriented parallel to a specified local coordinate system.

From 3 Nodes

Opens the HyperMesh systems panel for created coordinate systems in any orientation by specifying three nodes or temporary nodes.

The New loc marker options allow you to create the following new temporary nodes, which are used as location markers throughout HyperMesh:

At Coord System

Creates a new marker at specified coordinate systems.

At Ellipsoids

Creates new markers at specified ellipsoids' centers and axis.

Cover Ellipsoids

Covers ellipsoids with new markers.

At Body COGs

Creates new markers at specified rigid bodies' center of gravities.

At Other

Opens the HyperMesh create nodes panel for creating markers by entering coordinates (local or global), between existing nodes/temp nodes, on a plane, or on CAD geometry.

Nastran Utility Menu

The Nastran user profile contains two macro menus on the Utility menu: Nastran1 and Nastran.

Nastran1 Page

Macros supported by the **Nastran1** macro menu.

Auto Contact

Create one or more contact interfaces between parts of your model.

Open the **Auto Contact** macro by clicking **Tools** > **Auto Contact** from the Utility menu. To access this tool, you must first load the Nastran user profile.

The Auto Contact tool lets you create one or more contact interfaces between parts of your model. Based on a proximity distance, the Auto Contact tool searches through the parts that you select and automatically creates new contact surfaces and contact interfaces between them. The Auto Contact tool displays and organizes the contact entities inside of a temporary Auto Contact Browser where you can review and adjust the contacts as needed before accepting any changes.

Auto Contact creates Nastran BCBODY-BSURF pairs and puts them into a Nastran BCTABLE card. You can select a variety of contact types from the drop-down list in the **Auto Contact** dialog. The interface type you select is used as the initial configuration for all found interfaces. You can edit the interfaces in the Auto Contact Browser.

The BCBODY-BSURF surface group pairs are created as master and slave entities per each contact interface, with the following characteristics:

- Master and slave surfaces are automatically assigned based on the average element size of each surface:
    - The surface with the smallest, average element size is assigned as a slave, and the other as a master.
    - You can review and swap the master and slave assignments for each contact in the Auto Contact Browser.
- For 3D element faces, the surface normal direction points outward from the solid body.
- For 2D elements, the surface normal direction is assigned according to element normals. You can review and reverse this information in the Auto Contact Browser.

Auto Contact Dialog

The **Auto Contact** dialog contains the following options:

Table 168:

Option	Description
Type of interface	This controls the contact type as defined by the IGLUE option inside of the BCTABLE card. The IGLUE options include: • Touching • No Contact • Regular Glue • Special Glue Please refer to the MSC Nastran documentation for a detailed explanation.
Select components	This directs you to the Component Selection panel, where you can select the components you want the Auto Contact tool to search through.
Create full component BCBODY's	When activated, the Auto Contact tool creates BCBODY-BSURF pairs of the entire outer surface of the given components, instead of just the elements inside the contact area. When turned off (default), the BCBODY-BSURF pairs contain only the elements inside the contact area, as controlled by the Proximity Distance parameter.
Proximity distance	This value specifies that the contact pairs generated between components are closer to each other by less than the specified proximity distance.
Use shell thickness for 2D elements	If activated, the shell thickness values are used for 2D elements instead of the specified Proximity distance. 3D elements keep the specified Proximity distance.
Max reverse angle	This is a maximum value that you specify. If the angle between the normals of two elements, or element faces within the proximity distance, exceeds the specified value, the elements are excluded from the contact. The default value is 15 deg.

Option	Description
Consolidate contact patches between component pairs	When activated, the Auto Contact tool consolidates any separate contact patch areas that participate in contacts between the same two components, so that the final number of BCBODY-BSURF pairs in the model can be reduced. When turned off (default), each contact patch area is treated as a separate BCBODY-BSURF pair.
Find	Executes the Auto Contact search between all of the components in the list.
Remove Selection Icon	Removes the highlighted items from the selected components list. You can use the Control and Shift keys to select multiple items in the list.
Review Selection Icon	Highlights the selected items from the component table in the HyperMesh graphics area, while graying out other components. You can use the Control and Shift key to select multiple items in the list. Right-click to return the model to normal display.
Help Icon	Opens the Auto Contact online help.

Remarks

During the Auto Contact process, temporary components may be created for parts containing 3D elements. These temporary components are named using a preceding ^ symbol, and automatically contain extracted element faces needed for the contact creation process. Auto Contact cleans up and removes these temporary components when you finish or cancel the process. If you decide to export the model before accepting or canceling the process, HyperMesh excludes these temporary components from export.

Auto Contact Browser

The Auto Contact Browser provides options for reviewing and modifying contact interfaces and surfaces that the Auto Contact tool finds.

The Auto Contact Browser automatically loads in the HyperMesh tab area as part of the Auto Contact creation process. All of the entities listed in this browser are temporary, and are not part of the model until you click the **Create** button. Clicking **Cancel** clears all of the temporary entities and closes the Auto Contact Browser, leaving your model unchanged.

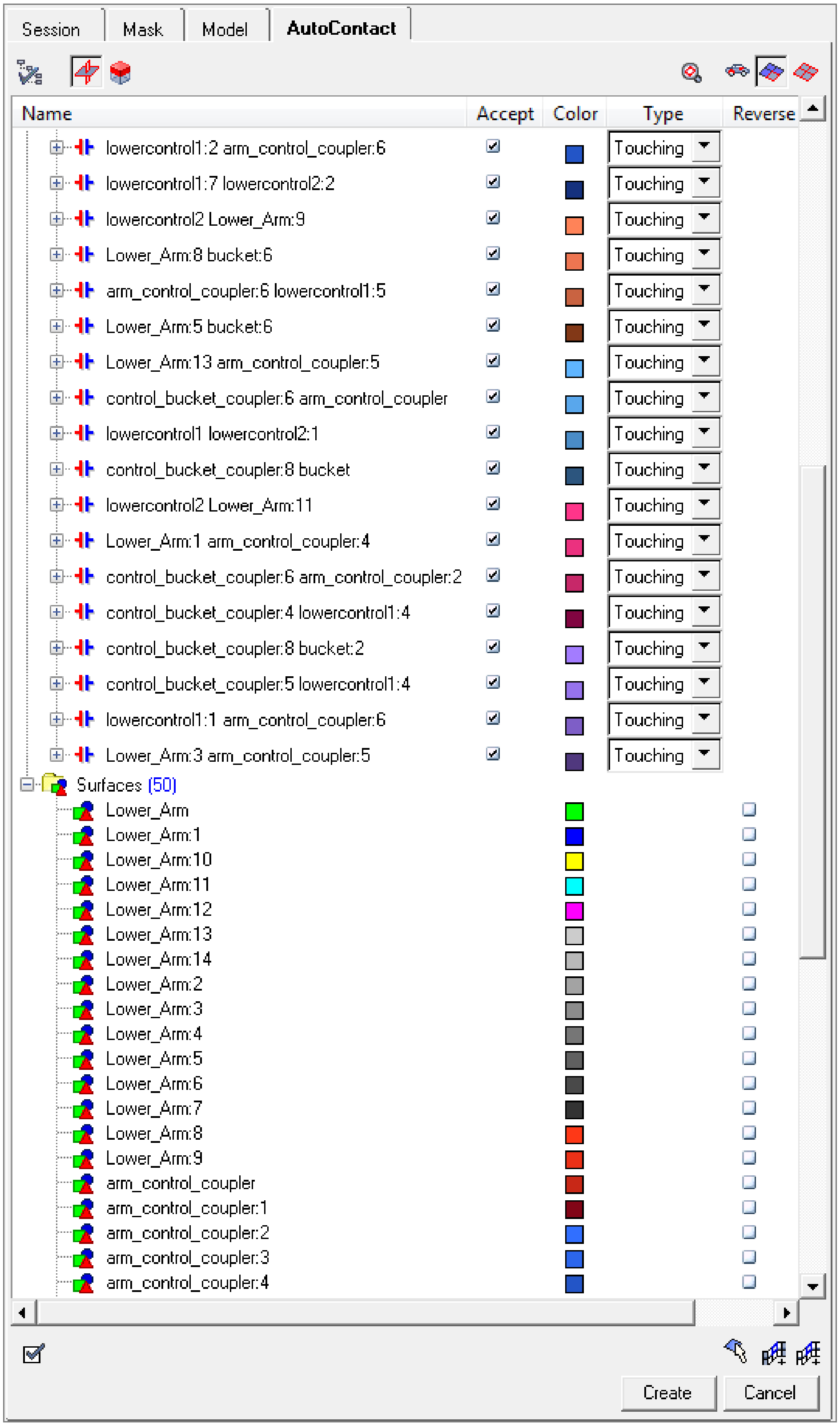

Session
Mask
Model
AutoContact
Name
Accept
Color
Type
Reverse
lowercontrol1:2 arm_control_coupler:6
lowercontrol1:7 lowercontrol2:2
lowercontrol2 Lower_Arm:9
Lower_Arm:8 bucket:6
arm_control_coupler:6 lowercontrol1:5
Lower_Arm:5 bucket:6
Lower_Arm:13 arm_control_coupler:5
control_bucket_coupler:6 arm_control_coupler
lowercontrol1 lowercontrol2:1
control_bucket_coupler:8 bucket
lowercontrol2 Lower_Arm:11
Lower_Arm:1 arm_control_coupler:4
control_bucket_coupler:6 arm_control_coupler:2
control_bucket_coupler:4 lowercontrol1:4
control_bucket_coupler:8 bucket:2
control_bucket_coupler:5 lowercontrol1:4
lowercontrol1:1 arm_control_coupler:6
Lower_Arm:3 arm_control_coupler:5
Touching
Surfaces (50)
Lower_Arm
Lower_Arm:1
Lower_Arm:10
Lower_Arm:11
Lower_Arm:12
Lower_Arm:13
Lower_Arm:14
Lower_Arm:2
Lower_Arm:3
Lower_Arm:4
Lower_Arm:5
Lower_Arm:6
Lower_Arm:7
Lower_Arm:8
Lower_Arm:9
arm_control_coupler
arm_control_coupler:1
arm_control_coupler:2
arm_control_coupler:3
arm_control_coupler:4
Create
Cancel

Figure 563:

Column	Description
	Underneath each contact interface name is the temporary master and slave surfaces associated to that interface. Red indicates a slave surface, and blue indicates a master surface. Renaming entities, as well as Master/Slave surface swapping is possible via the context menu.
Accept	This checkbox defines whether or not a contact interface is included in the creation process once you click **Create**.
Color	Color that is assigned to the contacts and contact surface entities.
Type	Type of CONTACT assigned to each contact interface. This affects the IGLUE option for the given BCBODY-BSURF pair inside the BCTABLE card.
Reverse Normal	This checkbox defines whether or not the normal direction of the given surface should be reversed or not. When OFF (default), surfaces on 3D elements point outwards, and surfaces on 2D elements point in the direction of the element normal.

Context Menu

Right-clicking an item in the Auto Contact Browser displays a context menu that offers options for modifying a selected contact interface or surface.

Table 170:

Option	Description
Rename	Renames the selected entity.
Delete	Deletes the selected items from the browser.
Swap Master-Slave	Allows you to switch the contact surfaces identified as master and slave. When executed, you see the surfaces switch position as master and slave in the browser.
Edit Faces	Opens the Element Selection panel, where you can manually add and remove individual elements from a selected contact surface. Click Proceed when you are finished. **Note:** Only available when selecting contact surfaces. Also accessible via the Select Elements Manually icon .

Option	Description
Add by Adjacent	Automatically adds all of the immediately adjacent elements to a selected contact surface. **Note:** Only available when selecting contact surfaces. Also accessible via the Add by Adjacent icon .
Add by Face	Automatically adds all of the elements by face to a selected contact surface. The feature angle controlling face detection tolerance can be modified inside the **Options** dialog. **Note:** Only available when selecting contact surfaces. Also accessible via the Add by Face icon .
Review Normal	Allowed only for surfaces containing 2D elements. For surfaces containing 3D element faces, normal direction is automatically assigned in an outward direction.
Accept All/None	Automatically checks and clears the Accept checkbox for every item in the Auto Contact Browser.
Reverse	Reverses the Accept checkbox status for every item in the Auto Contact Browser.
Expand/Collapse All	Automatically expands and collapses every folder in the Auto Contact Browser.

Browser Icons and Other Controls

The Auto Contact Browser contains the following icons and controls:

Table 171:

Option	Description
Options	Opens the browser's **Options** dialog. From this dialog you can enter a new feature angle or customize the transparency for a selected entity. Click **OK** when you are finished.
Highlight Elements	When this icon is turned on, the elements that belong to the currently selected contact interfaces or surfaces are highlighted in the graphics area. You can use the Ctrl and Shift keys to select multiple items from the browser.

Option	Description
	Highlight Elements is mutually exclusive and may be switched off. This could be helpful when working with large models.
Review Elements	When this icon is turned on, the elements that belong to the currently selected contact interfaces or surfaces are reviewed in the graphics area. Elements are highlighted by color, while all other components are grayed out. You can use the Control and Shift key to select multiple items in the browser. Review Elements is mutually exclusive and may be switch off. This could be helpful when working with large models.
Fit View to Elements	Automatically zooms in on the elements that belong to the currently selected contact interfaces or surfaces.
Freeze Displayed Components	When this icon is turned off (default), the graphics area will dynamically update depending on the entities that are selected in the browser. When turned on, the current display state will remain unchanged, even when you change your selection in the browser.
Display Components with Elements	When this icon is turned on (default), the elements belonging to the currently selected contact interfaces or surfaces are highlighted and reviewed in the graphics area, while also displaying the components they belong to. All of the other components are masked and will not display in the graphics area.
Display Only Elements	When this icon is turned on, the elements that belong to the currently selected contact interfaces or surfaces are highlighted and reviewed in the graphics area, while masking everything else.
Select Elements Manually	Opens the Element Selection panel, where you can manually add and remove individual elements from a contact surface. Click **Proceed** when you are finished. Also accessible via the context menu (Right-Click > **Edit Faces**).
Add by Adjacent	Automatically adds all of the immediately adjacent elements to a selected contact surface. Right-click the icon to undo one time. Also accessible via the context menu (Right-Click > **Add by Adjacent**).
Add by Face	Automatically adds all of the elements by face to a selected contact surface. Right-click the icon to undo one time. The feature angle controlling face detection tolerance can be modified inside the **Options** dialog. Also accessible via the context menu (Right-Click > **Add by Face**).

Option	Description
Recheck	Opens the **Auto Contact** dialog, so that you can recheck the selected interfaces. When you select interfaces from the browser, the GUI will automatically populate the components that the interaction was based on. This helps modify an existing interface.

Remarks

During the Auto Contact process, temporary components may be created for parts containing 3D elements. These temporary components will be named using a preceding ^ symbol, and will automatically contain extracted element faces needed for the contact creation process. Auto Contact will cleanup and remove these temporary components when you finish or cancel the process. If you decide to export the model before accepting or canceling the process, HyperMesh will always exclude these temporary components from export.

Set Up an Auto Contact Run

1. From the menu bar, select **Preferences** > **User Profiles**.
   The **User Profile** dialog opens.
2. Select **Nastran** as the profile name then click **OK**.
3. From the menu bar, click **Tools** > **Auto-Contact** and do the following in the **Auto Contact** dialog:
   a) Open the Type of interface drop-down list and select the type of interface you want to create.

      The interface you select is used as the initial configuration for all found interfaces. You can edit the interface in the Auto Contact Browser.
   b) Double-click **Component** to open the Component Selection panel.
4. On the Component Selection panel, click **comps**, select your components, and then click **select**.
   The components you select are added to the Components list inside of the **Auto Contact** dialog.
5. Click **Proceed** and enter two values in the **Auto Contact** dialog.
   - For **Proximity distance**, enter a value to specify the maximum proximity distance between the components of contact pairs.
   - For **Max reverse angle**, enter a value to specify the maximum angle between the normals of two elements or element faces within the proximity distance. If this value is exceeded, the element is not included in the contact. The default value is 15 deg.

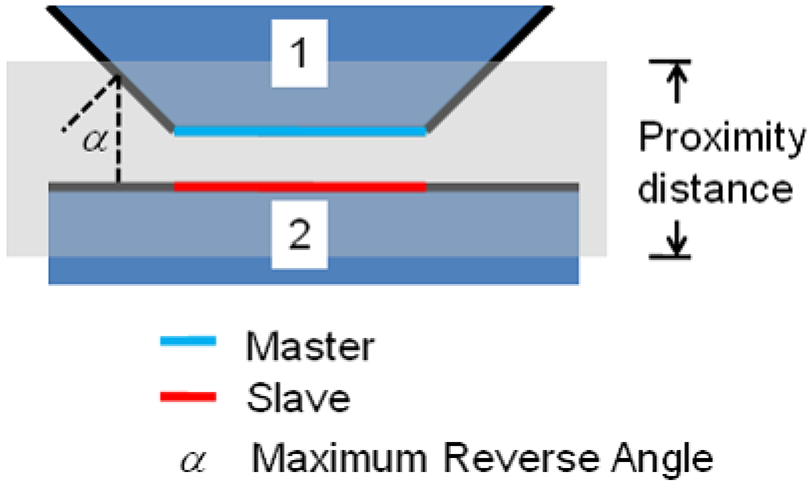


Figure 564: Definition of Proximity Distance and Maximum Reverse Angle

6. Click **Find**.
   The status bar activates. Once it is complete, the Auto Contact Browser automatically opens.
7. Use the Auto Contact Browser to review all of the found interfaces and make any necessary adjustments to your contacts and contact surfaces. When you are finished, click **Create**.

   Contact pairs are created for all items marked as Accepted. Clicking **Cancel** will close the Auto Contact Browser without creating any contacts.

   This creates a BCTABLE card with master-slave entities that use the contact types displayed in the Auto Contact Browser.

BCTABLE Manager

Use the **BCTABLE Manager** to create, edit and delete BCTABLEs from a convenient tabbed interface.

The three tabs, BCTABLE, Contact Elems, and Parameters, each contain tools related to BCTABLEs.

BCTABLE Tab

The BCTABLE tab lists existing BCTABLEs in the database. For each item in the list, you can choose whether to display it with the Display checkbox. To view the content of a particular BCTABLE, select the **Status** checkbox and click the **Contact Elems** tab. The following buttons are also available on this tab:

Table 172:

Button	Action
Sync	Synchronize the settings in the BCTABLE tool and the database.
Delete	Delete the selected BCTABLE.
Create	Create a new BCTABLE.

Button	Action
Close	Exit the **BCTABLE Manager**.

Contact Elems Tab

The Contact Elems tab lists existing contact elements for the selected BCTABLE. The BCTABLE is selected on the BCTABLE tab, as described above. The following buttons are also available on this tab:

Table 173:

Button	Action
Add	Add a row/contact element.
Delete	Delete the selected contact element.
Close	Exit the **BCTABLE Manager**.

Parameters Tab

The Parameters tab lists existing parameter values for each pair of contact elements. The following buttons are also available on this tab:

Table 174:

Button	Action
Reset	Set to default values.
Update	Update the values.
Close	Close the tool.

Click **Slave** and **Master** to see all the BCBODYs.

Create a Spider

Use the **RigidSpider** macro to create a spider (RBE2 elements) around holes.

You can create a spider with or without a washer and with one or multiple RBE2 elements.

1. From the Nastran Utility menu, click **RigidSpider**.
   The **Generate Rigid Spider** dialog opens.
2. Select the components where the spider is needed.

   Click **...** to access a list of all available components.
3. Select the Spider Type.

- Normal
- Washer

If you select **Washer**, an additional field appears on the dialog.

4. In the Rigid Type field, select **Single** or **Multiple**.

   Single indicates that a single RBE2 element will be created between the independent node and all dependent nodes. Multiple indicates that multiple RBE2 elements will be created between the independent node and each dependent node.
5. Click **Generate**.
   The tool redraws the component chosen in step 2 using plotel elements around the holes and component perimeter.
6. Pick a plotel element around the hole where the spider is needed.
7. Click **proceed**.

View Part Information

The **PartInfo** macro summarizes a part's statistics in a dialog.

1. To start the macro, click **PartInfo** on the Utility menu.
2. Click **component** in the main menu area to select a component or click a component in the graphics area to select it.
3. Click **proceed**.
   The **Part Information** dialog opens, which lists the part ID, name, thickness, and material type.
4. To view additional statistics about the part, click the **More Details** tab.
5. To display statistics for a different part, select the part in the graphics area or the components selector and click **proceed** again.

> **Tip:** Click the middle mouse button instead of the proceed button to quickly select components.

Component Table

With this macro, you can create components, select components, assign materials to components, change component colors and change component visualization modes.

This macro opens the Nastran **Component Table**, which displays components and their associated attributes in an interactive table. You can also configure the table; only configured items are displayed in the table.

Most actions are available from shortcut (right-click) menus. You can also find options in the drop-down menus. Before performing actions such as changing the values of component data, you must select **Editable** from the Table menu. Once the components are writable, you can modify the values of existing components. The following sections describe how to use the **Component Table** in both read-only mode and editable mode.

Use the Component Table in Read-Only Mode

When you open the **Component Table**, existing components are listed in a table using a default configuration. This configuration displays the component name, component ID number, properties on component, component color, thickness, property on element, material name, material ID, material type, and the visualization status of each component. The Nodes and Elems display is turned off by default. When activated, the total numbers of elements and nodes are shown at the bottom of the table.

The display of the data in the **Component Table** can be customized according to your preferences. You can:

- Change which columns are displayed
- Change the order of the columns
- Sort the components by column data, ascending or descending
- Filter which components are displayed based on column data values (see below)

You can save your settings by creating a configuration file. From the Table menu, open the Configure submenu and select the **Save CFG-File** option. This configuration file saves the set of table configuration options so you can use them again. By default, a configuration file (`comptable.cfg`) is saved in the working directory for each component table session and settings from this file are applied each time the table is built.

Use the Component Table in Editable Mode

When you switch the **Component Table**from the default read-only mode to editable mode, by selecting **Editable** from the Table menu, you can perform all the actions described in the section above, plus edit the attributes of the components listed in the table. To change the value of an attribute, select the attribute in the Assign Values drop-down, type the new value in the adjacent field, and click **Set**.

Create a New Component

1. From the Table menu of the **Component Table**, select **New**.
   The **Component Create** dialog opens.
2. Type a name in the Component Name field.
3. Select a material type for the component from the Mat Name drop-down field, or click the adjacent **New** button to define a new material and select it.
4. Click **Properties** to assign a property, or click the adjacent **New** button to define a new property and select it.
5. Click **Create**.
   A panel opens on which you must confirm the component creation.
6. Click **return**.

The new component appears in the **Component Table**.

Create a New Material or Edit an Existing Material

1. From the Assign Values drop-down field, select **Mat Name**.
2. Click **New** to create a new material or select a material from the HM-Mats drop-down field and click **Edit** to edit that material.

Assign a Value to Multiple Components

1. Select the components that you want to change.

   You can use Control+click and Shift+click to select non-adjacent and adjacent rows in the table, respectively. Other options are available in the Selection menu.
2. Select the column type you want to change from the Assign Values drop-down field.
3. Type the new value in the adjacent field.
4. Click **Assign**.
   The selected column types are updated to the new value you specified.

Filter the List of Components in the Table

The **Component Table** includes advanced filtering features based on the available columns of data.

If your model contains a large number of components, you can filter the components to quickly see components that are of interest to you.

1. From the Table pull-down menu, select **Filter...**.
   The **Filter** dialog opens.
2. Type a search string, with optional wildcard characters (*), in the fields for the columns you want to search.

   For example, to search for all components that begin with the letter 'c' and have 'steel' as the material type, you would complete the dialog, as shown below:

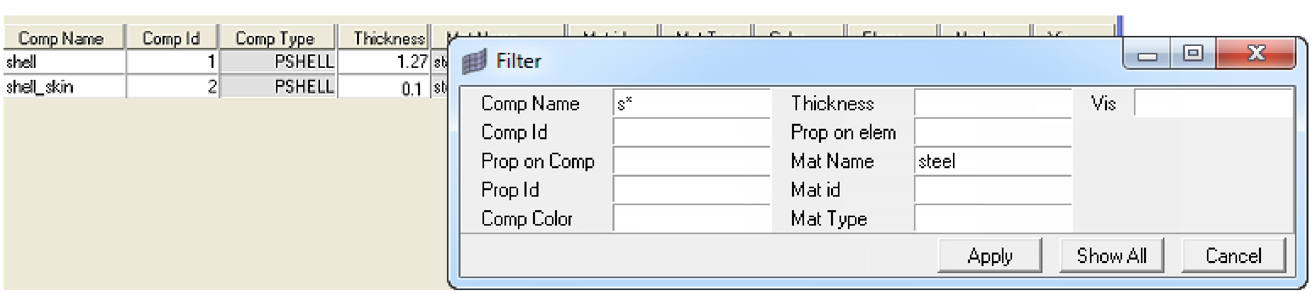


Figure 565:

   Search strings are case-sensitive.
3. Click **Apply** in the **Filter** dialog.

The filter is applied to the **Component Table** and only those components that match the search criteria are shown.

4. To remove filtering and display all the components, click **Show All** on the **Filter** dialog. You can also keep the **Filter** dialog open after applying filter changes by selecting the **Table** menu and selecting **Configure** and **Filter on top**.

Customize the Contents of the Table

To customize what appears in the table, you can specify which columns of data appear in the table.

1. From the Table menu, select **Configure** and **Columns...**.
   The **Configure** dialog opens.
2. The **Configure** dialog contains a list of column types that are available. Select the checkboxes for the data you want to view.
3. Click **OK**.

Export Data in CSV or HTML Format

1. From the Table menu, click **Save** and then either **CSV** or **HTML**, depending on which type of data file you prefer.
2. If you chose **CSV**, you must select **delimiter Comma (,)...** or **delimiter Semi-colon (;)...**, depending on which character you want to be the data delimiter in the output file.
   The **Select Output File** dialog opens, on which you can select an existing file to overwrite or type the name of the file you are creating.
3. Click **Save**.
   The data file is saved in the location you specified.

Property Table

Use this macro to create properties, select properties, assign materials to properties and export the data in CSV or HTML format.

This macro opens the Nastran **Property Table**, which is an interactive table of Nastran one-dimensional properties and their associated materials.

> **Note:** This table contains 1D properties only. Use the **Component Table** to create 2D and 3D properties.

Most actions are available from shortcut (right-click) menus. You can also find options in the drop-down menus. Before performing actions such as changing the values of property data, you must select **Editable** from the Table menu. Once the properties are writable, you can modify the values of existing properties. The following sections describe how to use the property table in both read-only mode and editable mode.

Use the Property Table in Read-Only Mode

When you open the **Property Table**, existing properties are listed in a table using a default configuration. This configuration displays the name, ID, type, material description, material ID, material type, number of elements, and visualization status for each property. The total numbers of elements is shown at the bottom of the table.

The display of the data in the property table can be customized according to your preferences. You can:

- Change which columns are displayed
- Change the order of the columns
- Sort the components by column data, ascending or descending
- Filter which components are displayed based on column data values (see below)

You can save your settings by creating a configuration file. From the Table menu, open the Configure submenu and select the **Save CFG-File** option. This configuration file saves the set of table configuration options so you can use them again. By default, a configuration file is saved in the working directory for each **Property Table** session and settings from this file are applied each time the table is built.

Use the Property Table in Editable Mode

When you switch the **Property Table** from the default read-only mode to editable mode (by selecting **Editable** from the Table menu), you can perform all the actions described in the section above, plus edit the attributes of the components listed in the table. To change the value of an attribute, select the attribute in the Assign Values drop-down, type the new value in the adjacent field, and click **Set**.

Filter the List of Properties in the Table

The **Property Table** includes advanced filtering features based on the available columns of data.

If your model contains a large number of properties, you can filter the list to quickly see properties that are of interest to you.

1. From the Table menu, select **Filter**.
   The **Filter** dialog opens.
2. Type a search string, with optional wildcard characters (*), in the fields for the columns you want to search.

   For example, to search for all properties that begin with the letter ‘P’ and have ‘steel’ as the material type, you would complete the dialog. Search strings are case-sensitive.
3. Click **Apply** in the **Filter** dialog.
   The filter is applied to the **Property Table** and only those properties that match the search criteria are shown.

4. To remove filtering and display all the properties, click **Show All** on the **Filter** dialog. You can also keep the **Filter** dialog open after applying filter changes by selecting the Table menu and selecting **Configure** and **Filter on top**.

Create a New Property

1. From the Table menu of the **Property Table**, select **New**.
   The **Property Create** dialog opens.
2. Type a name in the Property Name field.
3. Select a property type for the property from the Prop Name drop-down field, or click the adjacent **New** button to define a new property and select it.
4. Select the type of property, such as PBAR or PBEAM.
5. Click **Create**.
   A panel opens on which you must confirm the property creation.
6. Click **return**.
   The new property appears in the **Property Table**.

Assign a Value to Multiple Properties

1. Select the properties that you want to change.

   You can use Control+click and Shift+click to select non-adjacent and adjacent rows in the table, respectively. Other options are available in the Selection menu.
2. Select the column type you want to change from the Assign Values drop-down field.
3. Type the new value in the adjacent field.
4. Click **Set**.
   The selected column types are updated to the new value you specified.

Material Table

The Materials Table is used to review and edit MAT1.

Review MAT1

From the Nastran Utility menu, click **Material Table**.
All MAT1 in the model are displayed.

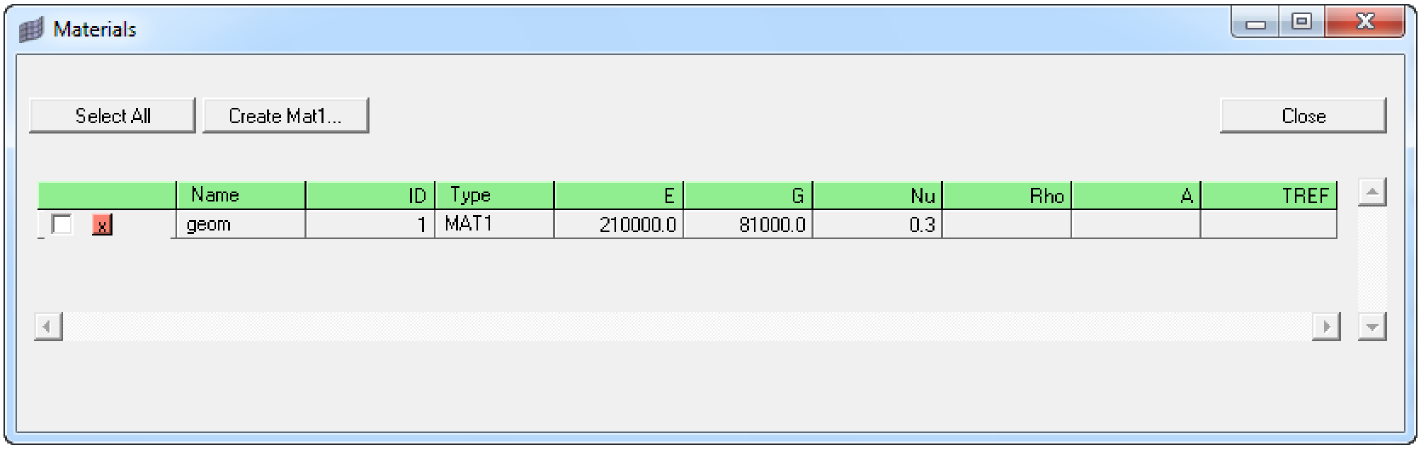


Figure 566:

Create MAT1

1. From the Nastran Utility menu, click **Material Table**.
2. Click **Create MAT1**.
   The following dialog is displayed.

   Figure 567:

3. Fill in the necessary fields.
4. Click **Create**.
   The material is created and added to the **Material Table**.

Delete MAT1

1. From the Nastran Utility menu, click **Material Table**.
2. Select the material to be deleted.
   Check **Select all** to select all materials.
3. Check the small X box next to the checkbox to delete the material.
4. Click **Close**.

Create an RSSCON Element

The **RSSCON Create** macro creates an RSSCON element that connects the shell and solid.

Because RSSCON elements are not directly supported, these elements are stored in unsupported Bulk Data cards.

1. Click the **RSSCON Create** macro.
2. Pick the elements to be connected in the model.
3. Click **proceed**.

> **Note:** When you renumber element or node IDs, RSSCON elements are not also updated because they are supported only as text.

Create an RSPLINE Element

The **RSPLINE Create** macro creates RSPLINE elements as Bulk unsupported cards.

1. Click the **RSPLINE create** macro.
2. Select nodes on the model.
3. Click **proceed**.

   The default value for the third field of this card is 0.1.

Table Create

Use the TABLE Create macro to create a tabular function card or add data to an existing card. You can use the macro to import XY data or enter the data manually.

Create a New Table Card Manually

1. Click **TABLE Create** on the Utility menu.
2. Select **Create/Edit Table**, select the table type, for example, TABLED1, and click **Next**.
   The **Create/Edit Table** dialog opens.
3. Type values in the XY table for the XY pairs you want to include in the table.
4. Select either **Create New Table** or **Edit Existing Table**.
   - If you selected **Create New Table**, type a name for the new load collector in the Name field and select a color for the load collector with the color selector button. A new load collector will be created with the table card image including the data from the XY table on the dialog.
   - If you selected **Edit Existing Table**, choose a load collector from the Select drop-down menu. The data in the XY table will be added to the existing table card that you specified.
5. Click **Apply**.

6. Click **Exit**.

Create or Add Data to a Table Card from a Data File

1. Click **TABLE Create** on the Utility menu.
2. Select **Import Table**, select the table type, for example, TABLED1, and click **Next**.
   The **Import Table** dialog opens.
3. In the File field, specify an XY data file.

   This file must be either `.csv` or `.txt` format.
4. Select either **Create New Table** or **Edit Existing Table**.
   - If you selected **Create New Table**, type a name for the new load collector in the Name field and select a color for the load collector with the color selector button. A new load collector will be created with the table card image including the data from the XY table on the dialog.
   - If you selected **Edit Existing Table**, choose a load collector from the Select drop-down menu. The data in the XY table will be added to the existing table card that you specified.
5. Click **Apply**.
6. Click **Exit**.

Nastran2 Page

Macros supported by the **Nastran2** macro menu.

Convert Degenerate Second Order Shells

Use the **Convert Shells** macro to convert degenerate second order shells into first order shells.

1. Select the file for which the second order shell elements are to be converted.
2. Click **Convert**.

   Messages are displayed in the message box, which state the name and location of the new file as well as the file where all the unconverted second order shells will be placed.

   > **Note:** You can import the Nastran file directly, and all the degenerate second order shells will be written into the `.hmx` file. In doing this, you will miss all the degenerate second order shells in the imported Nastran file.

Display and Expand the SET

Use the **Display SETs** macro to review and expand (before renumber) SETs.

1. Choose the SETs from the Selection column.
2. Click **Display**.

Note:

- "HM SET" means the ID in the SET can be renumbered.
- "TEXT SET" means the ID in the SET cannot be renumbered.
- Empty SET cannot be viewed or renumbered.

Create Tags on Nodes

Use the **Tag on Nodes** macro to create a tag on every node that has a comment in its 10th field.

1. Choose the color of the tags.
2. Click **Create**.

Create SPOINTs

Use the **SPOINT** macro to create SPOINTs.

1. Input nodes in the **Node ID(s)** window using any the following formats:
   - 2
   - 100 THRU 200
   - 13,24,25
   - 13 14 15
   - 13 THRU 25,30 50
2. Click**Add**.
   The new SPOINTs are created and added to the **SPOINTS** window.

Note: SPOINTs are treated as NODEs. Delete them as you would NODEs.

PAM-CRASH 2G Utility Menu

The PAM-CRASH 2G Utility menu (`pamcrash2G.mac`) contains shortcuts and tools that help simplify PAM-CRASH 2G tasks.

Tool Menu

Use the Tool menu options to simplify safety tasks.

Dummy Positioning Tool Start Macro

Use this macro to position dummies.

Once the template is loaded, you will be asked to create a process instance or open an existing process instance. After this step, you could see a task tree defining the process of dummy positioning in HyperMesh. You can then traverse through these tasks and position the dummy. Once the dummy is positioned, this position can be saved as a transformation file and can be later applied to the dummy to bring it into this final position without user interaction. The following tasks are listed in the process tree for dummy positioning.

Table 175:

Task Name	Action	
Configure Process:	Select either Interactive positioning or Automatic positioning. Depending on the selection, the process tree will change.	
Interactive Positioning	LoadDummy	You are asked for the PAM-CRASH 2G dummy and the positioner file. When the PAM-CRASH 2G dummy is loaded in HyperMesh, the pampostohm tool is automatically started and the dummy is prepared for positioning. System collectors, systems, and assemblies are created and nodes are associated with the systems.
	LoadNoDummyFiles	Imports other parts of the model, which may be required in order to position the dummy correctly.
	SelectJoints	Opens the Dummy panel and displays a list of joints in the model, and allows you to select a joint for viewing the load curves associated with that joint. You can select which curves (x,y,z) should be shown, the updatePlot utility shows the current

Task Name	Action	
		position of the joint on the load curve by drawing a vertical line. The deletePlot utility deletes the plots created by this tool. If you exit this task without deleting plots, you would need to do that in the delete panel afterwards. **Note:** Only the plots will be deleted, none of the load curves will be deleted from the database.
	CreateTransformation	Once finished with the positioning of the dummy, you can save this information into a transformation file.
	CreateDocumentation	Update the model documentation as well as create HTML documentation of the process. You can also select an image to be embedded in the HTML file. A browser can also be selected to display the HTML file. An `.h3d` file is also embedded into the HTML documentation.
	ExportFiles	Save the model as an HyperMesh database as well as in PAM-CRASH 2G format. While exporting in PAM-CRASH 2G format, you have the choice of specifying whether you want to delete the additional entities created by the dummy positioning tool.
Automatic Positioning:	LoadOnlyDummy	Same as LoadDummy.
	ExecuteTransformation	Select a transformation file, which will be executed automatically to position the dummy.
	Documentation	Same as CreateDocumentation.
	ExportDummy	Same as ExportFiles.

The same transformation file could be applied to different dummies, provided the tree structure remains same.

Materials Supported for Dummy Positioning in PAM-CRASH 2G

For the computation of the minimum and maximum angle for the rotation in each direction, the PAM-CRASH 2G materials 220 and 221 are implemented.

Stop Angle Implementation

Normally, the stop angles are given by load curves. The second and last curve points are used to determine the stop angle. If a load curve has less than four entries, the first and the last entries are used. You can find the implementation of the stop angle in the HM_JOINT_INFO function in the function template of the PAM-CRASH 2G Interface. If load curves are not defined for a joint, default values for stop angles (-270°C to +270°C) will be displayed in the Dummy Positioning panel.

Update Initial Rotation Angle in the JOINT Card

The initial rotation angles in the JOINT cards are updated automatically.

To update them, use the macro **Update Jt Angles** on the Tool page.

Part Info Macro

The **Part Info** macro summarizes a part's statistics in a dialog.

1. To start the macro, click **Part Info** on the Utility menu.
2. Click **component** on the main menu area to select a component or click a component in the graphics area to select it.
3. Click **proceed**.
   The **Part Information** dialog opens, which lists the part ID, thickness, and material type.
4. To view additional statistics about the part, click **More Detail >>**.
5. To display statistics for a different part, select the part in the graphics area or the components selector and click **proceed** again.

> **Tip:** Click the middle mouse button instead of proceed to quickly select components.

Substructure Tool Macro

The **Substructure Tool** macro provides several features for creating and modifying substructures.

Substructures are user-defined sets of finite elements, elements and nodes, or nodes from the initial model. Information defining the substructures and their boundary node displacements are saved in a special file during the initial run. In subsequent runs, this saved data is read in and the saved displacement time histories are applied as imposed displacements to the boundary nodes. The input data set for a subrun must contain all the information needed to perform the subrun.

Note:

- The SUBDF card is supported as a vector collector in HyperMesh.
- You must have a pre-existing GES created that contains the nodes/elements of the part defining the substructure.

When you open the **Substructure Tool** macro, the existing substructures are listed in a table-based interface, as shown below.

The substructures are sorted by order of creation. The following columns appear in the table:

Table 176:

Column	Description
Keyword Name	The name of the substructure.
IDEF	Indicates the definition type of the substructure: **0** Only via elements **1** Via elements and boundary nodes **2** Only via boundary nodes
DTSUB	Specifies the time intervals for the boundary node displacement time histories.
Elements	Displays the ID of the element-based entity set. You can click **GES...** to use the GES Browser to select an entity set.
Boundary Nodes	Displays the ID of the boundary node-based entity set. You can click **GES...** to use the GES Browser to select an entity set.
Filename SUBURN	Name of the file that contains the definition of the substructure and its boundary node displacements.
Time Factor	Time unit scaling factor.

Column	Description
Length Factor	Length unit scaling factor.

RBODY Manager Macro

The **RBODY Manager** is accessible in the PAM-CRASH 2G Tool menu.

The **RBODY Manager** provides the following features in one convenient tab:

- Display all rigid bodies in the model
- Display all individual rigid bodies
- Create new, and edit existing, simple and complex rigid body formulations
- View and update details of individual rigid bodies, though the card editor and the Rigid panel

The tool is also available in the Radioss user profile and offers similar features.

Existing rigid bodies are explained below. For each rigid body, the display status, ID number, name, master node ID, and type is shown.

Disp
: Indicates whether the rigid body is displayed in the graphics area.

ID
: The ID number of the rigid body.

Title
: The descriptive name of the rigid body.

Master Node
: The ID of the node that serves as the master node of the rigid body.

Type
: S or C. S indicates a simple rigid body, which is a typical spider formulation. C indicates a complex formulation, such as an RBODY that points to a part or a set of sets.

Highlight individual entries or groups of entries to perform an action on the rigid body. Actions are available from the context menu (by right-clicking over the table entries) or the tool bar buttons. These actions are described below:

Table 177:

Icon	Name	Action
	Review Options	Customize the way the selected rigid bodies are displayed. Options include transparency and auto-review selections.
	Review	Highlights the nodes to which the selected RBODY is attached. The master node is shown in blue and the slave nodes are shown in red.

Icon	Name	Action
	Find Attached	Highlights the elements that are attached to the selected rigid body.
	Edit	Modify the definition of the rigid body through the Rigid panel.
	Card Edit	Opens the RBODY card in the card editor.
	Delete	Deletes the selected rigid body.
	Refresh	Update the table of rigid bodies.

New rigid bodies can be created with the **RBODY Manager**. The following fields are available at the bottom of the **RBODY Manager** tab, which enable you to supply all the basic data needed to create a new RBODY. Nodes, parts, materials, properties, and GES can be used to define the slave nodes. Once the RBODY is created, click **refresh** to list it in the table. Then you can select the RBODY to edit the card image, display the RBODY, and so on.

Note:

- When a large number of slave nodes are attached to a master node, the connecting lines are not displayed in the graphical model.
- The table of rigid bodies can be sorted by the ID, title, mater node, and type columns.
- Select **Show Details** from the context menu to display a summary of details about the rigid body including the ID, name, master node ID, and number of slave nodes.
- Select **Editable** from the context menu to make the title column editable. When the Title column is editable, you can modify the names of the rigid bodies.

Apply Initial Metric Macro

The **Apply Initial Metric** macro applies the initial metric to the current model for simulating the inflation of airbags.

Refer to the PAM-CRASH documentation for details about using the initial metric.

Before using this macro, you must specify an `.im` file in the METRIC control card. This file specifies the conditions of the airbag inflation.

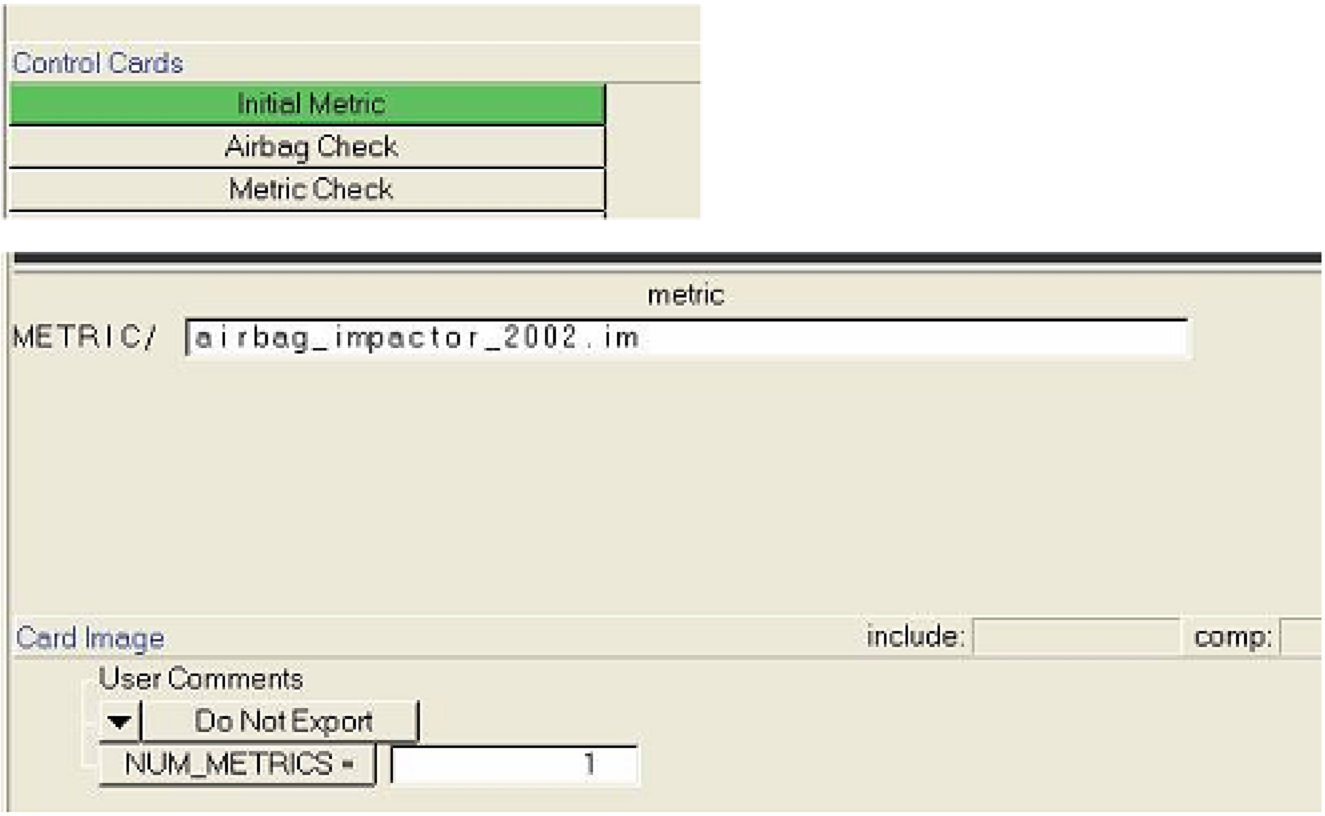


Figure 568:

When you click **Apply Initial Metric**, the macro applies the settings in the `.im` file to the currently-loaded model and displays the inflation motion in the graphics area. When the execution is complete, the macro creates a `.log` file named `initial_metric.nodes` that contains the NODAL information.

Organize Xlinks Macro

The **XLINK Organizer** macro can be used to move and arrange existing LINK type elements (PLINK, ELINK, LLINK, SLINK) to existing components.

The macro contains the following options:

Table 178:

Option	Description
Select element:	Select the element type you want to work with.
Prefix:	Filter the element results by the text you type in the adjacent text box. Click **Set** to run the filter.

The macro contains the following buttons:

Table 179:

Button	Action
	Makes only the selected elements visible in the graphics display.
	Select a component/part to which the selected elements will be added.
	Import a component from a PAM-CRASH 2G input file.

Move an XLINK Element to a Component

1. Select the type of link element from the Select element field.
   The list of elements of that link type are listed in the table along with the parts with which they are associated.
2. Click **component** to select a component to which you want to add the selected elements.
3. Click **Apply**.
   The selected elements are added to the selected component.

MASS Manager Macro

Displays information about masses in the model.

The **MASS Manager** is a tool accessible in the PAM-CRASH 2G Tool menu. The **MASS Manager** provides the following features in one convenient tab:

- Display all masses in the model
- Display individual masses
- Create new, and edit existing simple and advanced mass formulations
- View, find attached, and update details of individual MASS, though the Card Editor and the MASS panel

Input Fields in the Show ID Ranges User Interface

The **Show ID Ranges User Interface** contains the following input fields:

Table 180:

Input	Action
Existing Entity Types in the Model	Select the entity type for which you want to have the ID range information.

Input	Action
	SELECT Displays a list of all entity types present in the model. You can select a single entity or multiple entities. **all** Displays information for all the entities present in the model. Nodes and elements are always selected.
Maximum Number of Ranges	Provides the maximum number of ranges (default is 10) to be displayed in the output file. If an entity has a larger number of ranges, it will be truncated.
Display Id's Type	Display either the used IDs for an entity or the free IDs for an entity. The option to output free IDs is valid only when you choose **Detailed** for the overview type.
Display Entity Type	Select the method in which elements are written to the output file. You can select either HM_entitytype or Element_type. Selecting **Element_type** writes out elements according to the solver definition.
Overview Type	Select an overview type. **Condensed** Only the overall maximum and minimum IDs and the total number of ranges for each entity are displayed. In this case, free IDs will not be displayed even if it is selected. **Detailed** Maximum and minimum IDs for each range number (subject to the maximum number of ranges specified) is displayed along with the corresponding range number. In this case, the overall maximum and minimum IDs and the total number of ranges for that entity will also be displayed at the beginning of the information related to the entity.
Id Info for Entities	Specify whether you want the information for all the entities in the model or only for the entities currently displayed.

Input	Action
Comment String for Solver	Input a string/character that is placed at the beginning of each line in the output file. This enables you to include the information in the solver deck for further use.

Find Menu

The Find menu contains options that help you find and visualize data.

Temporary Nodes

CNODE Clr/All/Dis

Clr
Deletes all temp nodes from the model.

All/Dis
Finds CNODEs in the complete/displayed model and highlights the nodes as temp nodes.

Find Components

By Elems
Finds all components which have elements in the current (masked) display.

Rbody Visualization

All/Dis/Sel
Updates the rigid bodies definition by resolving the references to a GES (set/set of set) by converting them into node lists and displaying the web on the screen.

All
Updates all rigid bodies.

Dis/Sel
Updates displayed or selected rigid bodies, respectively.

Find/Mask
Finds/Masks the respective entities. Review the buttons' tool tips to see the full entity name.

Card Menu

The Card menu contains options that help display the PAM-CRASH 2G cards in an editor.

PAM-CRASH 2G Cards

PARTS
Shows the 1D, 2D, and 3D PART cards of the displayed components in a viewer.

MATER
Shows the 1D, 2D, and 3D MATER cards of the displayed components in a viewer.

NSM
Shows the NSM cards of the displayed groups in a viewer.

CNTAC
Shows the CNTAC cards of the displayed groups in a viewer.

GROUP
Shows the GROUP cards of the displayed sets in a viewer.

HyperMesh Entities

Properties
Shows all properties cards of the displayed properties in a viewer.

Sensors
Shows all SENSOR cards of the displayed sensors in a viewer.

Loads
Shows all loads and load collectors cards of the displayed loads and load collectors in a viewer.

Curves
Shows all FUNCT cards of the model in a viewer.

Airbags
Shows all BAGIN and CHAMBER cards of the displayed control volumes in a viewer.

Sum Menu

The Sum menu contains options that execute the PAM-CRASH 2G summary templates and display the resulting text file in a viewer.

Components All/Dis
Execute the components_txt summary and show the results for the complete/displayed model in a viewer.

Materials All/Dis
Execute the materials_txt summary and show the results for the complete/displayed model in a viewer.

Elements All/Dis
Execute the elements_txt summary and show the results for the complete/displayed model in a viewer.

Center Of Gravity All/Dis
Execute the ctr_of_gravity_txt summary and show the results for the complete/displayed model in a viewer.

Moment Of Inertia All/Dis
Execute the moment_of_inertia_txt summary and show the results for the complete/displayed model in a viewer.

Interfaces All/Dis

Execute the groups_txt summary and show the results for the complete/displayed model in a viewer.

None Struct Masses All/Dis

Execute the nsmas_txt summary and show the results for the complete/displayed model in a viewer.

Property ALL

Execute the property_txt summary and show the results for the complete/displayed model in a viewer.

Sensors ALL

Execute the sensors_txt summary and show the results for the complete/displayed model in a viewer.

M1 Menu

The M1 menu contains options that set the correct element or load type and enter the appropriate HyperMesh panel.

PART/

3D/2D/1D/LINK

Enters the Components panel, selects the comps collector, and sets the correct card image.

MATER/

3D/2D/1D/LINK

Enters the Components panel, selects the mats collector, and sets the correct card image.

PLY_DATA

Enters the Components panel, selects the mats collector, and sets the correct dictionary.

Mass Elements

MASS

Sets the element type mass = to MASS and enters the Mass panel.

NSMASS

Enters the Interfaces panel and sets the card image to nsmas.

Constraints

RBODY

Sets the element type rigid = to RBODY and enters the Rigids panel.

NODCO

Sets the element type rigid = to NODCO and enters the Rigids panel.

RWALL

Enters the Rigid Walls panel and set the card image = to RWALL.

CNTAC

Enters the Interfaces panel and sets the card image = to CNTAC.

TIED

Enters the Interfaces panel and sets the card image = to TIED.

Elements

BAR

Sets the element type 1dele = to BAR and enters the 1d Elems panel.

BEAM

Sets the element type beam = to BEAM and enters the Beams panel.

KJOINT

Sets the element type 1dele = to KJOIN and enters the1d Elems panel.

JOINT

Sets the element type 1dele = to JOINT and enters the 1d Elems panel.

SPRING

Sets the element type spring = to SPRING and enters the Springs panel.

SHELL

Sets the element type tria3 = and quad4 = to SHELL.

MEMBR

Sets the element type tria3 = and quad4 = to SHELL.

TRIA_C

Sets the element type tria3 = to TRIA_C.

SOLID

Sets the element type tetra4, pyramid5, penta6, and hex8 = to SOLID.

BSHELL

Sets the element type hex8 = to BSHEL.

Link Elements

PLINK

Sets the element type mass = to PLINK and enters the Mass panel.

ELINK

Sets the element type 1dele = to ELINK and enters the 1d Elems panel.

LLINK

Sets the element type 1dele = to LLINK and enters the 1d Elems panel.

SLINK

Sets the element type tria3 = and quad4 = to SLINK.

M2 Menu

The M2 menu contains options that set the correct element or load type and enter the appropriate HyperMesh panel.

Auxiliaries

FRICT

Enters the Components panel, selects the props collector, and sets card image = to FRICTION.

RUPMO

Enters the Components panel, selects the props collector, and sets card image = to RUPTURE_MODEL.

SENSO

Enters the Sensors panel.

CURVES

Enters the Edit Curves panel and sets the radio button to modify.

Safety

SLIPR

Sets element type mass = to SLIPRING and enters the Mass panel.

RETRA

Sets element type mass = to RETRACTR and enters the Mass panel.

BAGIN

Enters the Airbag panel and sets the card image to BAGIN.

CHAMB

Enters the Airbag panel and sets the card image to CHAMBER.

GASPC

Enters the Components panel, selects the props collector, and sets card image = to GASPEC.

Plot Output

THNOD

Enters the Output Blocks panel and changes the type to nodes.

THELE

Enters the Output Blocks panel and changes the type to elements.

SENPT

Sets element type mass = to SENPT and enters the Mass panel.

Nodals

FRAME

Enters the Systems panel.

NODE

Enters the Nodes panel.

INVEL

Sets load type velocity = to INVEL and enters the Velocity panel.

VEL3D

Sets load type velocity = to VEL3D and enters the Velocity panel.

RVE3D

Sets load type velocity = to REV3D and enters the Velocity panel.

ACC3D

Sets load type acceleration = to ACC3D and enters the Acceleration panel.

RDV3D

Sets load type velocity = to RDV3D and enters the Velocity panel.

RDA3D

Sets load type acceleration = to RDA3D and enters the Acceleration panel.

RAC3D

Sets load type acceleration = to RAC3D and enters the Acceleration panel.

BOUNC

Sets load type constraint = to BOUNC and enters the Constraints panel.

DIS3D

Sets load type constraint = to DIS3D and enters the Constraints panel.

DIS3DX

Sets load type constraint = to DIS3DX and enters the Constraints panel.

DIS3DM

Sets load type constraint = to DIS3DM and enters the Constraints panel.

RAN3D

Sets load type constraint = to RAN3D and enters the Constraints panel.

RDD3D

Sets load type constraint = to RDD3D and enters the Constraints panel.

CONLO

Sets load type constraint = to CONLO and enters the Constraints panel.

Analysis by Keywords

LOADCOLS

Enters the Collector panel, switches the type to loadcols and sets the card image to INVEL.

Conn Menu

The Conn menu contains options that let you manage connectors in the model.

Connector Organize

ByPlinkPart

Organizes the connectors by the parts of the associated Plink elements.

Renumber

NodeID/Plink ID

Node ID

Renumber nodes for displayed plinks such that plinkId = plinkNodeId.

Plink ID

Renumber displayed plinks such that plinkId = plinkNodeId.

Connector Panel

Feabsorb/Quality/Realize

Enters the Connectors panel, and then fe absorb/quality/fe realize panel, depending on the selected macro.

Find att to P(X)LINKs

Ces

Finds connectors attached to the displayed link entity.

Com

Finds all components attached to the displayed link entity.

Mcom

Finds all master components attached to the displayed link entity.

Scom

Finds all slave components attached to the displayed link entity.

Find att to CE

PL

Finds Plinks attached to the displayed connectors.

Com

Finds all components attached to the displayed connectors.

Find att to Comps

Finds entities attached to the displayed components.

Find/Mask

Finds/Masks the entities depending upon the selected macros.

GES Macro

The **GES** macro displays a GES Browser that lets you manage sets in the model.

Because general entity selection is mapped as a set of sets in the PAM-CRASH 2G interface, the need to manage the sets becomes more crucial for the effective and efficient handling of the model.

Functionalities included in the GES Browser

- Creating a set, group or GES
- Renaming sets
- Modifying sets (drag and drop facility is available)
- Reviewing sets
- Deleting sets
- Adding/deleting entities to sets
- Adding/removing keywords to/from GES (set of set)
- Adding ranges/comments to the sets
- Resolving ranges
- Reviewing as PAM-CRASH 2G card
- Changing the keyword (for example, ELE to DELELE)
- Filter the entities to be displayed in the browser. For example, you can only select sets of sets or component sets for viewing.
- Filtering by ID and name is also possible. For example, if you enter 1-100; 200; 300-400 in the Ids field, it will display all GES/Sets (including child items) with IDs 1 to100, 200 and 300 to 400. Similarly in the Name field, you can enter a keyword such as `face` and the browser displays all items (including child items) whose name contains face.
- Selecting by name
- Finding and deleting empty GES
- Finding unused GES
- Resolving unresolved groups - This function is useful in case of assembling model from different files. It may happen that the group referenced in first file is defined in second file. In this case when first file is loaded, the group is imported as an unresolved group. When the second file is also loaded, this utility can be used to resolve the unresolved group references.
- Creating/modifying interfaces
- Reviewing interfaces as contact surfaces
- Creating/modifying rigid walls
- Creating/modifying section forces
- Creating/modifying loadcols

- Direct access to the card editor for interfaces, loadcols and components

DELNOD Card

The following parameters are available via the DELNOD card:

ELE

GRP

PART

NOD

SEG

EDG

ELE > NOD

PART > NOD

GRP > NOD

DELNOD

DELELE

DELPART

DELPART

DELGRP

DELELE > NOD

DELPART > NOD

DELGRP > NOD

When you add a set to the GES using drag and drop functionality of the browser, a new set is created with the exact copy of the contents of the original set, therefore, the changes made to this new set are local in effect but in case of adding a group to the GES, only a reference is made to the existing GROUP definition. You can modify the GROUP by using the references also but there exist only one copy of the group in the model. Therefore while modifying one of the references, you should always keep in mind that this change will also affect all the other references to this group and the group itself is modified. In case the group does not exist in the model, it is created. If you do not want to create the group, instead use the functionality **Unresolved Groups** > **Edit** and add the group there. This will be exported correctly.

You can also create a config file, which saves all information about various GES Browser options, such as DisplayComments (Yes/No), DisplayRanges (Yes/No) and confirmChanges (Yes/No). Later on, this config file can be used to restore these settings. By default, a config file (`gesbrowser.cfg`) is saved in the working directory for each session of the GES Browser and settings from this file are applied every time the browser is built. These functionalities can be invoked from the buttons LoadCFG/SaveCFG.

Permas Utility Menu

The macros on the Permas Utility menu simplifies some common tasks for the Permas user profile.

The Permas Utility menu is loaded when you open the Permas user profile.

Convert Groups

Use the **Convert Groups** macro to convert element-based surfaces that were created in Abaqus to Permas surfaces.

While in the Abaqus user profile, use the **Contact Manager** to create as many element-based surfaces (*SURFACE, TYPE=ELEMENT) as needed. When finished, switch to the Permas user profile by clicking **Preferences** > **User Profile** and select the **Convert Groups** macro. $SURFACE ELEMENTS are created based on the contact surfaces identified in the model.

Currently, only *SURFACE cards defined on individual element IDs of shells or solids are translated. If a surface is defined on sets, it will not be translated. Also, it is necessary to have face identifiers defined.

Example of Entities Converted

```
*SURFACE, NAME = surf_1, TYPE = ELEMENT
1, SPOS
*SURFACE, NAME = surf_2, TYPE = ELEMENT
2, S1
```

Example of Entities that will not Convert

```
**Element ID, but no face identifier given
*Surface, Name = surf_3, TYPE = ELEMENT
2,
**Surface definition based on element set
*Surface, NAME = surf_4, TYPE = ELEMENT
Element_set1, SPOS
```

Create an NLLOAD Card

To create an NLLOAD card, you must first create and edit a loadstep card.

When a loadstep is created, an NLLOAD card can be created by checking the NLLOAD checkbox in the card image of the loadstep. The NLLOAD card defines the tabular load history for static or transient analysis. To utilize the NLLOAD card, the LOADING option must be selected. LOADING is set as the default. The card image lists all the load collectors currently assigned to the load step. Continue following the steps to set the NLLOAD card time load history:

1. In the **Card Editor**, ensure that the Analysis Procedure toggle is set to **LOADING**.

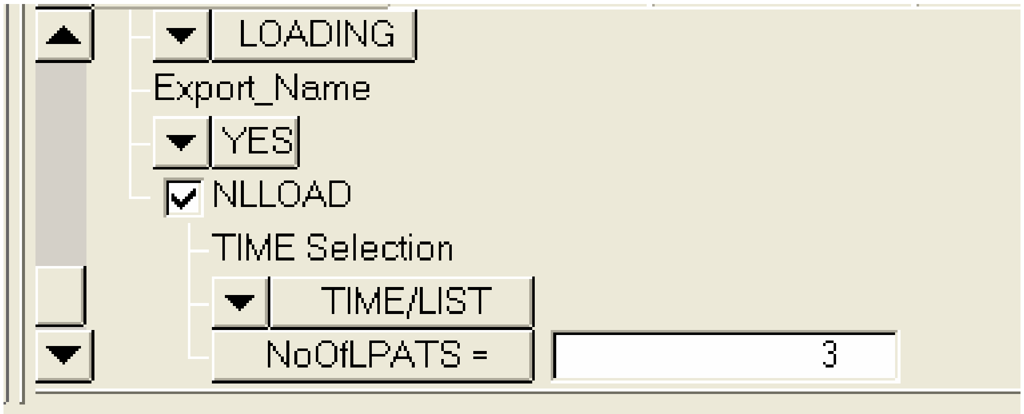


Figure 569:

2. Place a check next to **NLLOAD**.
3. Under TIME Selection, choose either **TIME/LIST** or **TIME/dt**.

   If you select **TIME/LIST**, the load pattern is determined by individual values entered in the TIME fields. It will set the iterations to a series of steps at specific points. If you select **TIME/dt**, you can specify the time steps in the firLOAst dataline with the start value and increment value. All subsequent datalines are automatically populated based on this information. The load history is now set to a series of regular intervals.
4. Enter a value in the NoOfLPATS field.

   This determines the number of load patterns (load collectors) you want to add to the NLLOAD card.
5. Enter the value in the TimeSteps field in the upper part of the card image.

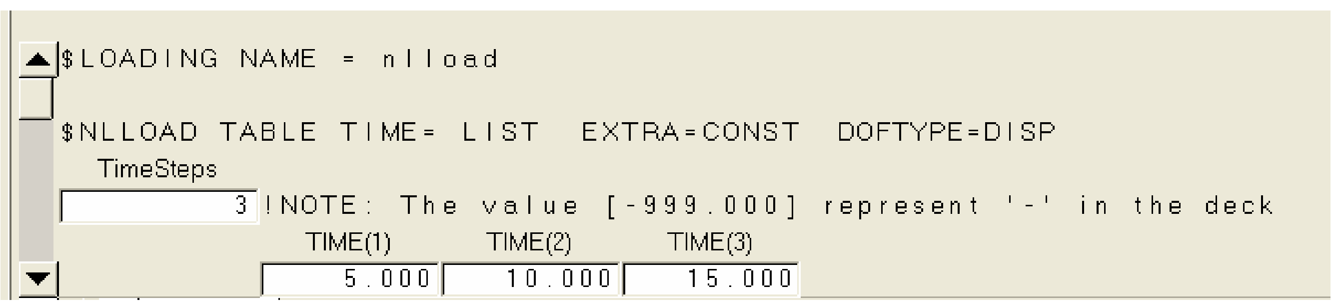


Figure 570:

6. For each TIME/STEP pattern created, enter values in the TIME fields to set the starting value and the increment values.
7. For each TIME/dt pattern created, enter the starting value in the t field and the increment value in the dt field.

   The TIME fields are automatically populated.

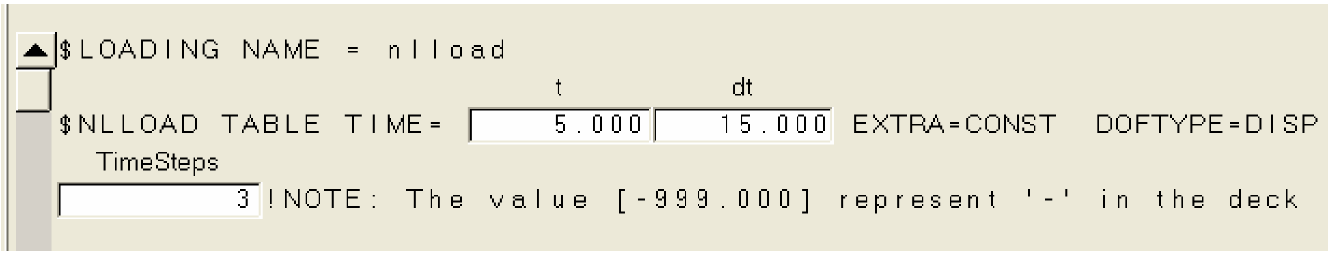


Figure 571:

> **Note:** For a better readablility on export, if the columns exceed the line length (currently set to 80 characters), a new NLLOAD keyword is written. On import NLLOAD will be written in this format but also if lines of each load pattern is continued with ampersands.

8. Click **return** to close the **Card Editor**.
9. Click **return** to close the Load Steps panel.

> **Note:** On export, if the columns exceed the line length (currently set to 80 characters), the lines will be continued by an ampersand (&). This is also the format the reader can understand from the `.dat` file only.

Use the PLOT NLLOAD Macro

You can use the **PLOT NLLOAD** macro on the Utility menu to draw the load history plots.

1. On the Utility menu, click **Plot NLLOAD**.

   The load step just created is displayed and the values entered in the NLLOAD card are shown. You can edit these values in the $NLLOAD table on the right side of the window, although you cannot add new columns or new load collectors at this point.
2. Use the Display checkbox to turn the display of particular load steps on and off.

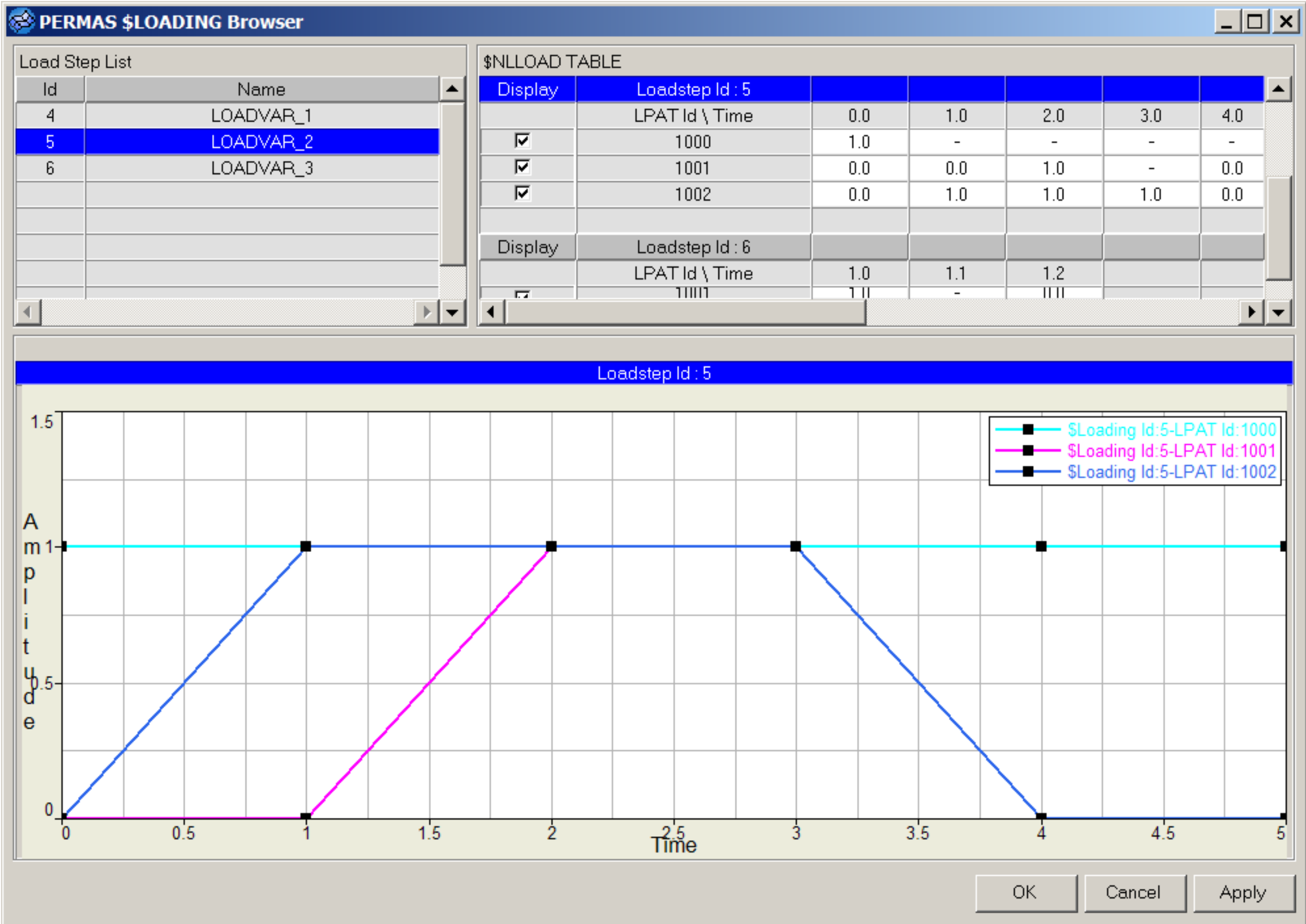


Figure 572:

OptiStruct Utility Menu

The Utility menu for the OptiStruct user profile contains, in addition to the default Utility menus, three pages (Summary, FEA and Opti) of specific utilities for OptiStruct.

The Summary page provides a short summary of the entities making up the model. The FEA page is dedicated to modeling and load setup, while the OPTI page is devoted to optimization. The Utility menu is available on the Utility tab when the OptiStruct user profile is loaded. The Utility tab may be activated/deactivated from the View menu.

Summary Page

The Summary page of the OptiStruct Utility menu lists a short summary using displayed or the entire model for components, loads, elements, center of gravity, moment of inertia, responses and constraints.

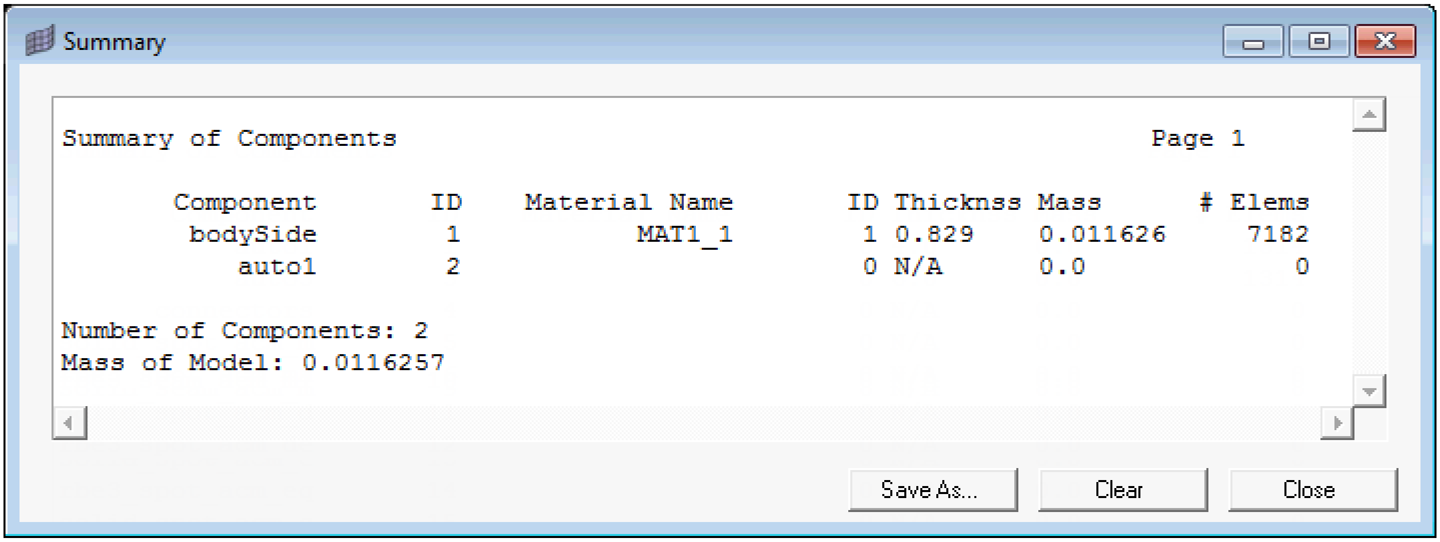


Figure 573: Summary of Components

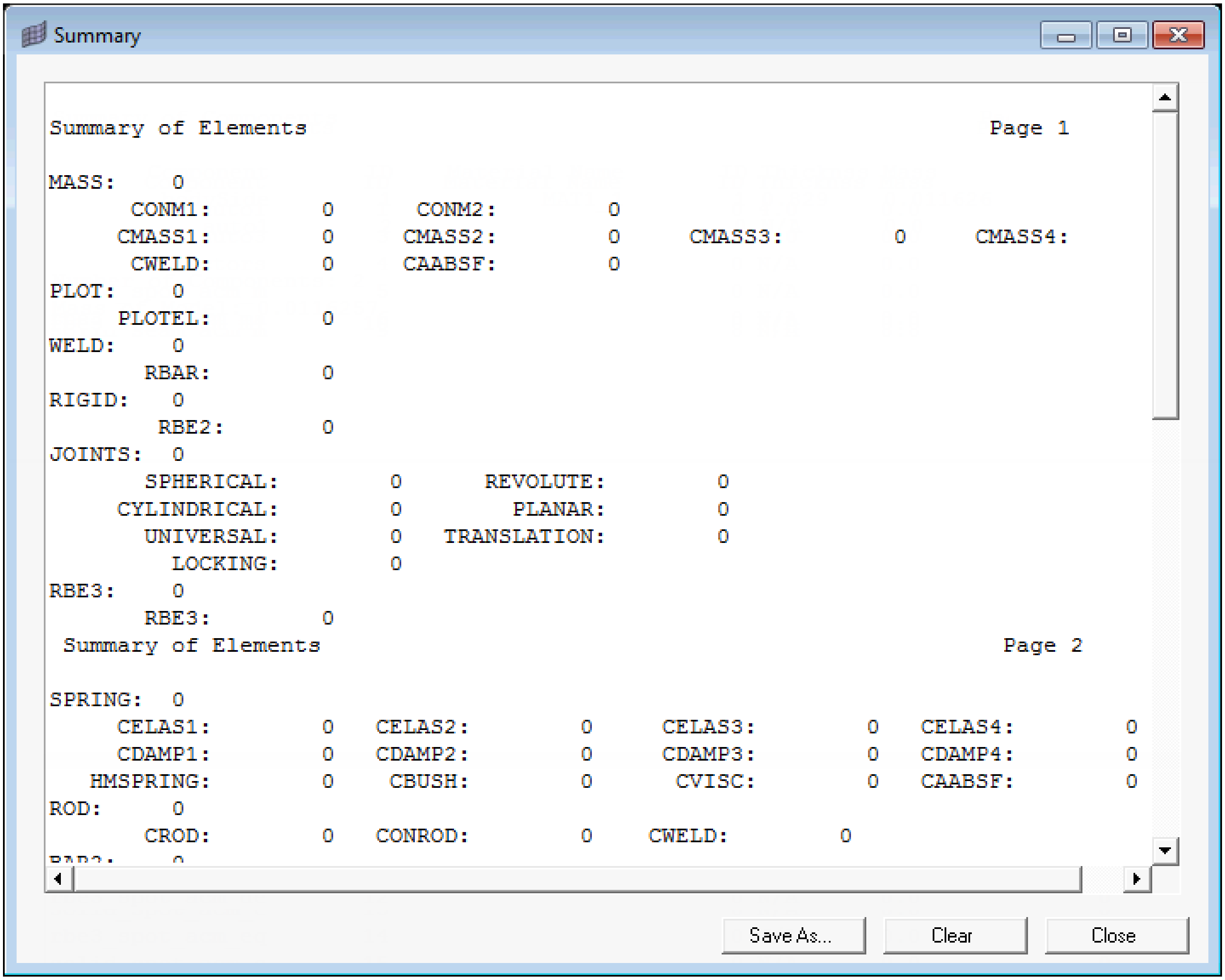


Figure 574: Summary of Elements

FEA Page

Macros available on the FEA page of the OptiStruct Utility menu.

I-DEAS to Radioss

The **I-DEAS to Radioss converter** utility converts an I-DEAS input file to an OptiStruct input file.

1. Select an I-DEAS file as the source file, or check the **Use current HM model** box to use the model loaded in the current session.
2. Select a file name and location to save the OptiStruct file that will be generated, or check the **Apply to current HM session only** box to generate the OptiStruct model in the current session.

3. Click **Convert**.

Export in MDL

Use the **Export in MDL** utility to export body and joint definitions in the current database to a Model Definition Language (`.mdl`) file that can be opened with MotionView.

Export Bodies and Joints Using the Export in MDL Utility

1. Enter a file name in the Save file as field or click  and, on the **Save file** pop-up window, choose where to save the generated `.mdl` file.
2. Click **Accept** to export body and joint definitions to an `.mdl` file.

   The **Treat flexible bodies as rigid bodies** checkbox controls the output of flexible bodies. Flexible bodies may be exported as either rigid or flexible bodies (flexible body export is not available at this time). The **Use prescribed cog, mass and inertias where available** checkbox controls the output of cog, mass and inertia values for each body. If this checkbox is unchecked, HyperMesh determines these properties for each body based on the model data. If this checkbox is checked, the values defined on the parameters subpanel of the bodies panel, should they exist, will be exported instead. The **Select nodes for additional point definitions in MDL** checkbox allows nodes to be selected for which MDL point definitions will be exported to the file.

Material Table

Use the **Material Table** utility to review and edit OptiStruct materials.

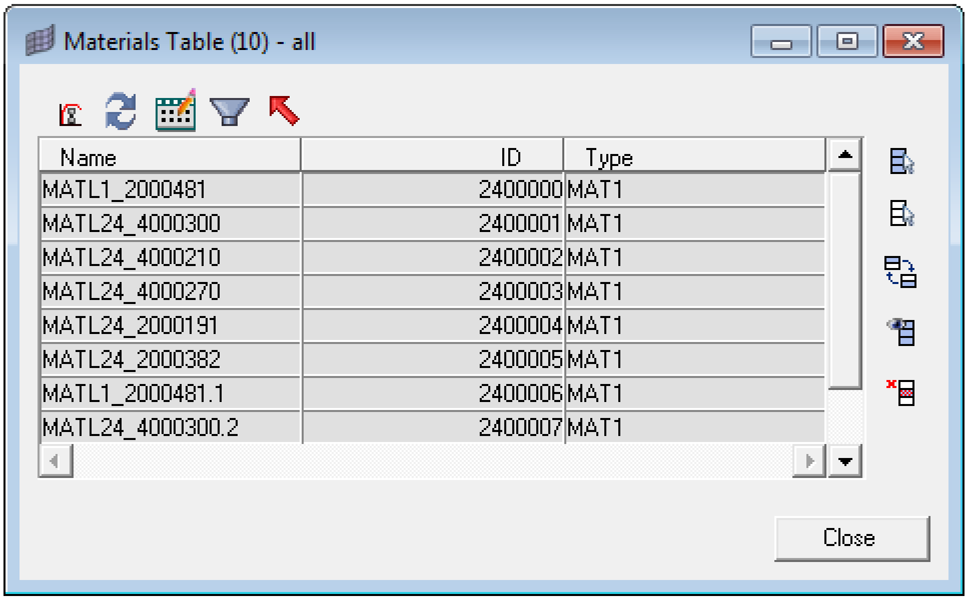


Figure 575:

Table 181:

Icon	Description
	Create new material
	Refresh table
	Toggles between table edit and table display mode
	Filter table
	Export table to CSV format
	Select all
	Select none
	Reverse selection
	Select displayed

Icon	Description
	Delete selected

Component Table

The OptiStruct **Component Table** displays components and their associated attributes in an interactive table.

The information displayed in the table may be configured.

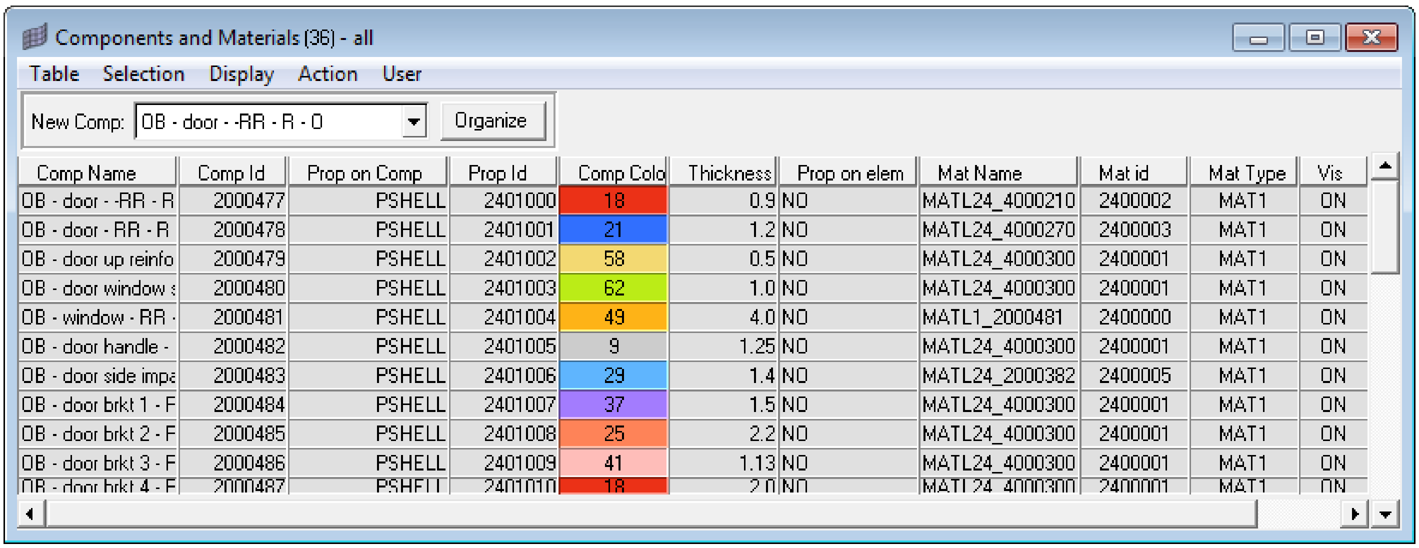


Figure 576:

This utility also allows you to create components, assign materials to components, change component colors, and change component visualization modes. Most actions are available either from shortcut (right-click) menus or from the pull-down menus.

Before performing actions, such as changing the values of component data, you must select **Editable** from the Table menu. Once the table is editable, you can modify the values of existing components. The following sections describe how to use the component table in both read-only mode and editable mode.

Use the Component Table in Read-Only Mode

When you open the **Component Table** displayed components are listed in a table using a default configuration. This configuration displays the name, ID number, type, thickness, material name, material ID, material type, color, and visibility (display) for each displayed component.

The table may be adjusted to display information for all components by selecting **Table** > **Configure** > **Components** > **All**.

The display of the **Component Table** can be customized according to your preferences. You can:

- Change which columns are displayed
- Sort the components by column data, ascending or descending
- Filter which components are displayed based on column data values (see below)

You can save your settings by creating a configuration file. From the Table menu, open the Configure sub-menu and select **Save CFG-File**. This configuration file saves the set of table configuration options so you can use them again. By default, a configuration file (`comptable.cfg`) is saved in the working directory for each component table session and settings from this file are applied each time the table is built.

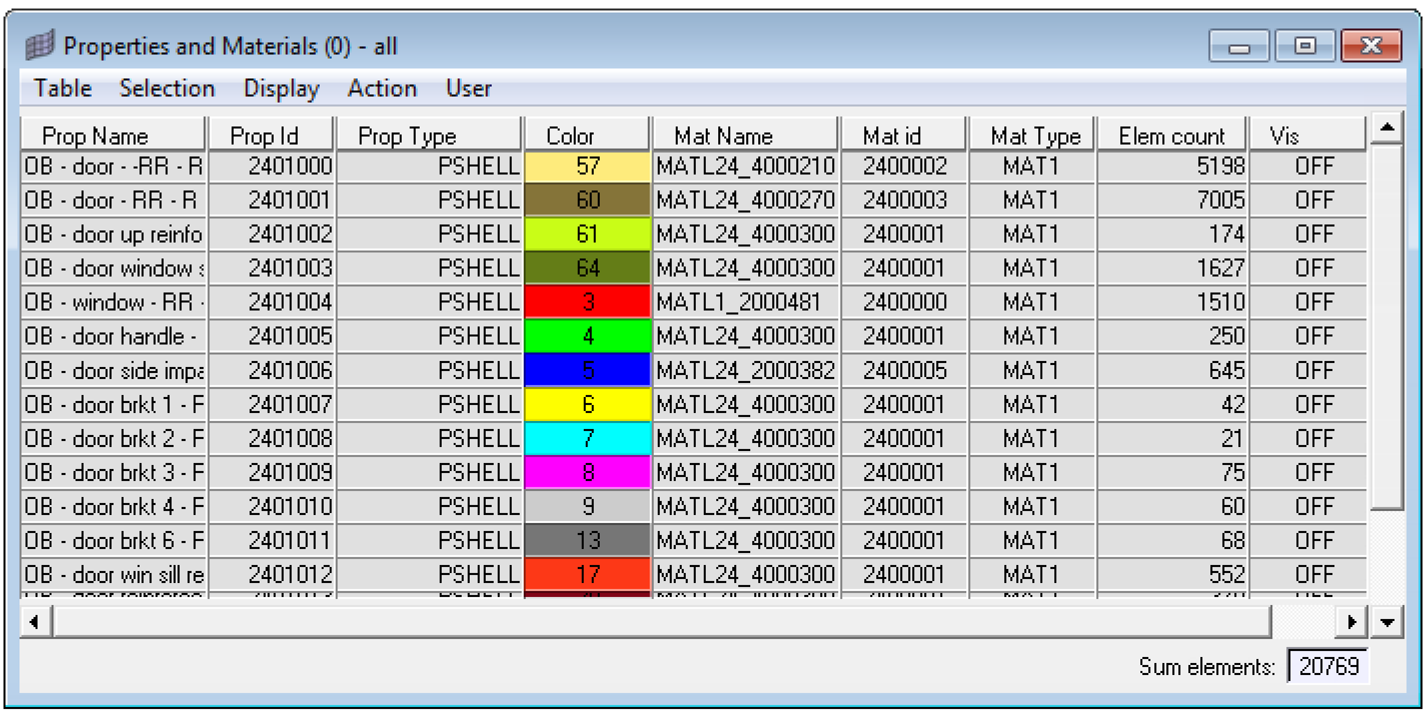
Use the Component Table in Editable Mode

When you switch the **Component Table** from the default read-only mode to editable mode by selecting **Editable** from the Table menu, you can perform all the actions described in the section above, plus edit the attributes of the components listed in the table. To change the value of an attribute, select the attribute in the Assign Values drop-down, type the new value in the adjacent field, and click **Set**.

Property Table

The OptiStruct **Property Table** displays properties and their associated attributes in an interactive table.

The information displayed in the table may be configured.



Prop Name	Prop Id	Prop Type	Color	Mat Name	Mat id	Mat Type	Elem count	Vis
OB - door - -RR - R	2401000	PSHELL	57	MATL24_4000210	2400002	MAT1	5198	OFF
OB - door - RR - R	2401001	PSHELL	60	MATL24_4000270	2400003	MAT1	7005	OFF
OB - door up reinfo	2401002	PSHELL	61	MATL24_4000300	2400001	MAT1	174	OFF
OB - door window s	2401003	PSHELL	64	MATL24_4000300	2400001	MAT1	1627	OFF
OB - window - RR -	2401004	PSHELL	3	MATL1_2000481	2400000	MAT1	1510	OFF
OB - door handle -	2401005	PSHELL	4	MATL24_4000300	2400001	MAT1	250	OFF
OB - door side impa	2401006	PSHELL	5	MATL24_2000382	2400005	MAT1	645	OFF
OB - door brkt 1 - F	2401007	PSHELL	6	MATL24_4000300	2400001	MAT1	42	OFF
OB - door brkt 2 - F	2401008	PSHELL	7	MATL24_4000300	2400001	MAT1	21	OFF
OB - door brkt 3 - F	2401009	PSHELL	8	MATL24_4000300	2400001	MAT1	75	OFF
OB - door brkt 4 - F	2401010	PSHELL	9	MATL24_4000300	2400001	MAT1	60	OFF
OB - door brkt 6 - F	2401011	PSHELL	13	MATL24_4000300	2400001	MAT1	68	OFF
OB - door win sill re	2401012	PSHELL	17	MATL24_4000300	2400001	MAT1	552	OFF

Figure 577:

This utility also allows you to create properties, assign materials to properties, and change property colors. Most actions are available either from shortcut (right-click) menus or from the pull-down menus.

Before performing actions such as changing the values of property data, you must select **Editable** from the Table menu. Once the table is editable, you can modify the values of existing properties. The following sections describe how to use the Property Table in both read-only and editable modes.

Use the Property Table in Read-Only Mode

When you open the **Property Table**, all properties are listed in a table using a default configuration. This configuration displays the name, ID number, type, material name, material ID, material type, color, number of elements and visibility (display) for each property.

The display of the **Property Table** can be customized according to your preferences. You can:

- Change which columns are displayed
- Sort the components by column data, ascending or descending
- Filter which components are displayed based on column data values (see below)

You can save your settings by creating a configuration file. From the Table menu, open the Configure submenu and select the **Save CFG-File** option. This configuration file saves the set of table configuration options so you can use them again. By default, a configuration file (`comptable.cfg`) is saved in the working directory for each component table session and settings from this file are applied each time the table is built.

Use the Property Table in Editable Mode

When you switch the **Property Table** from the default read-only mode to editable mode, by selecting **Editable** from the Table menu, you can perform all the actions described in the section above, plus edit the attributes of the properties listed in the table. To change the value of an attribute, select the attribute in the Assign Values drop-down, type the new value in the adjacent field, and click **Set**.

Auto Contact

Use the Auto Contact tool to quickly and easily create one or more contact interfaces at once between several parts of your model.

This tool's location on the Utility Browser is at **Tools** > **Auto Contact** and **BCs** > **Create** > **Auto Contact**. This tool is only available when the OptiStruct user profile is loaded.

Based on a proximity distance, the Auto Contact tool will search throughout all of the selected components and automatically create new Contact Surfaces and contact interfaces between them. All of the contact entities found by the Auto Contact tool are displayed and organized inside a temporary Auto Contact Browser, where you can review and adjust them as needed before accepting any changes.

Auto Contact creates OptiStruct CONTACT interfaces, with different type options as available from the Type of Interface pull-down menu in the **Auto Contact** dialog. The selected Type of Interface will be used as the initial configuration for all found interfaces. Afterwards, you can individually edit the type of interface in the Auto Contact Browser.

Two contact surfaces are created as Master and Slave entities per contact interface, with the following characteristics:

- Master and Slave surfaces are automatically assigned based on the average element size of each Contact Surface:

    - The Contact Surface with the smallest, average element size will be assigned as a Slave, and the other will be assigned as a Master.
    - The Master and Slave assignments for each contact can be reviewed and swapped in the Auto Contact Browser.
- For 3D element faces, the contact surface direction will point outwards from the solid body.
- For 2D elements, the contact surface direction will be assigned according to element normals, which can be reviewed and reversed in the Auto Contact Browser.

Auto Contact Dialog

The **Auto Contact** dialog window contains the following options:

Table 182:

Option	Description
Type of interface	Creates CONTACT interfaces using three different types of interfaces: **SLIDE** Represents a zero-friction sliding contact. **STICK** Represents an infinite-friction sticking contact. **FREEZE** Represents a tied contact. **FRICTION** Represents a contact with friction using a static friction coefficient (MU1).
Select components	Directs you to the Component Selection panel, where you can select the components to have the Auto Contact tool search through.
Proximity distance	Generates contact pairs between components that are closer to each other by less than the specified proximity distance.
Use shell thickness for 2D elements	If activated, the shell thickness values will be used for 2D elements instead of the specified Proximity distance. 3D elements will still use the specified Proximity distance.
Max reverse angle	Excludes elements in the contact when the angle between the normals of two elements, or element faces within the proximity distance exceeds the specified reverse angle value. Default value is 15 deg.
Consolidate contact patches between component pairs	Select this checkbox to consolidate any separate contact patch areas that participate in contacts between the same two components, so that the final number of contacts in the model can be reduced. When turned

Option	Description
	off (default), HyperMesh treats each contact patch area as a separate individual contact pair.
Find	Executes the Auto Contact search between all of the components in the list.
Cancel	Closes the **Auto Contact** dialog without applying changes.
Remove Selection Icon	Removes the highlighted items from the selected components list. You can use the Control and Shift keys to select multiple items in the list.
Review Selection Icon	Highlights the selected items from the component table in the HyperMesh graphics area, while graying out other components. You can use the Control and Shift key to select multiple items in the list. Right-click to return the model to normal display.
Help Icon	Opens the Auto Contact online help.

Remarks

During the Auto Contact process, temporary components may be created for parts containing 3D elements. These temporary components will be named using a preceding ^ symbol, and will automatically contain extracted element faces needed for the contact creation process. Auto Contact will cleanup and remove these temporary components when you finish or cancel the process. If you decide to export the model before accepting or canceling the process, HyperMesh will always exclude these temporary components from export.

Auto Contact Browser

The Auto Contact Browser provides options for reviewing and modifying contact interfaces and surfaces found by the Auto Contact tool.

This browser will automatically load in the HyperMesh tab area as part of the Auto Contact creation process. All of the entities listed in this browser are temporary, and are not part of the model until you click the **Create** button. Clicking **Cancel** will clear all of the temporary entities and close the Auto Contact Browser, leaving your model unchanged. See Remarks below.

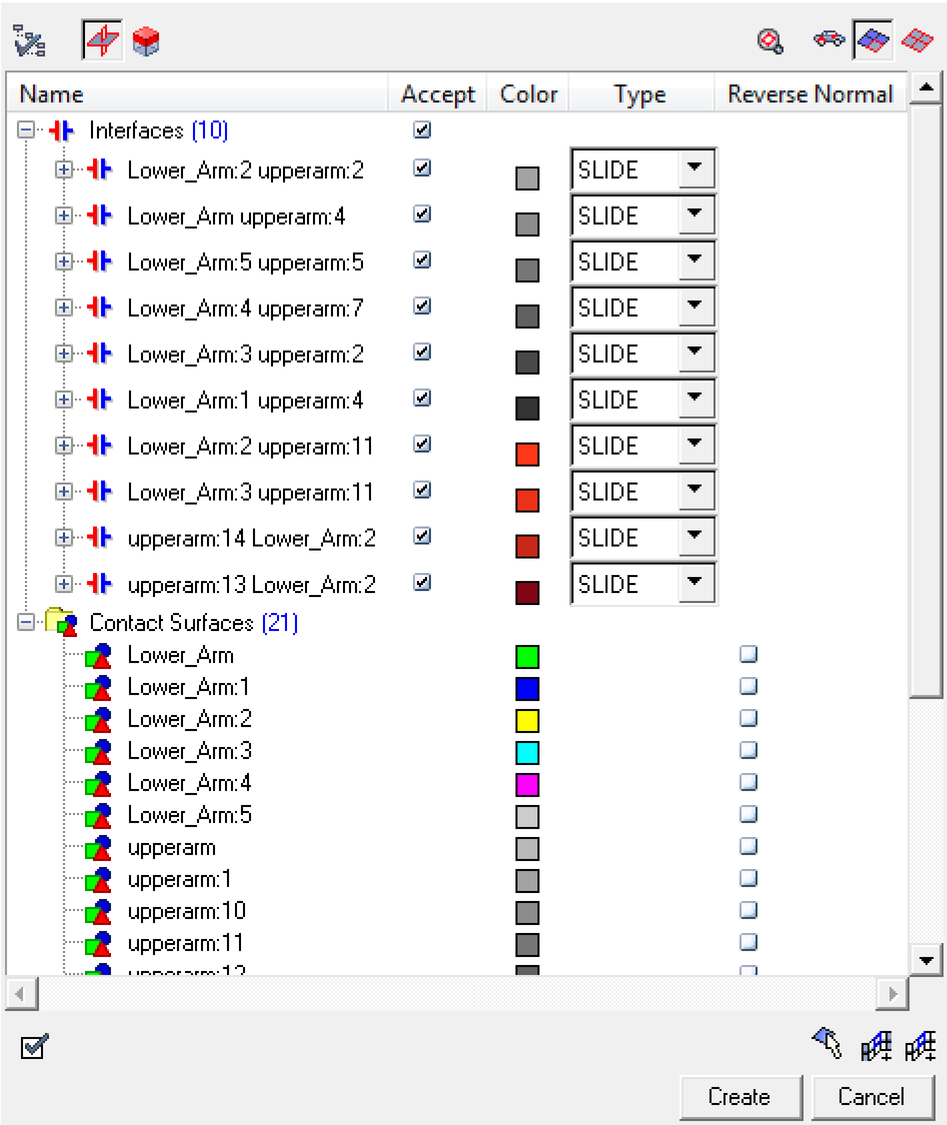


Figure 578:

Auto Contact Browser Columns

The Auto Contact Browser contains the following columns:

Table 183:

Column	Description
Name	Lists the names of all of the contact interfaces and surfaces found by the Auto Contact tool. Underneath each contact interface name is the temporary master and slave surfaces associated to that interface. Red indicates a slave surface, and blue indicates a master surface.

Column	Description
	Renaming entities, as well as Master/Slave surface swapping is possible via the context menu.
Accept	This checkbox defines whether or not a contact interface will be included in the creation process once you click **Create**.
Color	Color that will be assigned to the contacts and contact surface entities.
Type	Type of CONTACT assigned to each contact interface.
Reverse Normal	This checkbox defines whether or not the normal direction of the given surface should be reversed or not. When OFF (default), surfaces on 3D elements will be pointing outwards, and surfaces on 2D elements will point in the direction of the element normal.

Context Menu

Right-clicking on an item in the Auto Contact Browser displays a context menu which offers options for modifying the selected contact interface or surface.

Table 184:

Option	Description
Rename	Renames the selected entity.
Delete	Deletes the selected items from the browser.
Swap Master-Slace	Allows you to switch the contact surfaces identified as master and slave. When executed, you will see the surfaces switch places in the master/slave positions in the browser.
Edit Faces	Opens the Element Selection panel, where you can manually add and remove individual elements from a selected contact surface. Click **Proceed** when you are finished. **Note:** Only available when selecting contact surfaces. Also accessible via the Select Elements Manually icon .
Add by Adjacent	Automatically adds all of the immediately adjacent elements to a selected contact surface. **Note:** Only available when selecting contact surfaces.

Option	Description
	Also accessible via the Add by Adjacent icon .
Add by Face	Automatically adds all of the elements by face to a selected contact surface. The feature angle controlling face detection tolerance can be modified inside the **Options** dialog. **Note:** Only available when selecting contact surfaces. Also accessible via the Add by Face icon .
Review Normal	Allowed only for surfaces containing 2D elements. For surfaces containing 3D element faces, normal direction is automatically assigned in an outward direction.
Accept All/None	Automatically checks and unchecks the Accept checkbox for every item in the Auto Contact Browser.
Reverse	Reverses the Accept checkbox status for every item in the Auto Contact Browser.
Expand/Collapse All	Automatically expands and collapses every folder in the Auto Contact Browser.

Browser Icons and Other Controls

The Auto Contact Browser contains the following icons and controls:

Table 185:

Option	Description
Options	Opens the browser's **Options** dialog. From this dialog you can enter a new feature angle or customize the transparency for a selected entity. Click **OK** when you are finished.
Highlight Elements	When this icon is turned on, the elements that belong to the currently selected contact interfaces or surfaces are highlighted in the graphics area. You can use the Ctrl and Shift keys to select multiple items from the browser. Highlight Elements is mutually exclusive and may be switched off. This could be helpful when working with large models.
Review Elements	When this icon is turned on, the elements that belong to the currently selected contact interfaces or surfaces are reviewed in the graphics area. Elements are highlighted by color, while all other components are grayed

Option	Description
	out. You can use the Control and Shift key to select multiple items in the browser. Review Elements is mutually exclusive and may be switch off. This could be helpful when working with large models.
Fit View to Elements	Automatically zooms in on the elements that belong to the currently selected contact interfaces or surfaces.
Freeze Displayed Components	When this icon is turned off (default), the graphics area will dynamically update depending on the entities that are selected in the browser. When turned on, the current display state will remain unchanged, even when you change your selection in the browser.
Display Components with Elements	When this icon is turned on (default), the elements belonging to the currently selected contact interfaces or surfaces are highlighted and reviewed in the graphics area, while also displaying the components they belong to. All of the other components are masked and will not display in the graphics area.
Display Only Elements	When this icon is turned on, the elements that belong to the currently selected contact interfaces or surfaces are highlighted and reviewed in the graphics area, while masking everything else.
Select Elements Manually	Opens the Element Selection panel, where you can manually add and remove individual elements from a contact surface. Click **Proceed** when you are finished. Also accessible via the context menu (Right-click > **Edit Faces**).
Add by Adjacent	Automatically adds all of the immediately adjacent elements to a selected contact surface. Right-click the icon to undo one time. Also accessible via the context menu (Right-click > **Add by Adjacent**).
Add by Face	Automatically adds all of the elements by face to a selected contact surface. Right-click the icon to undo one time. The feature angle controlling face detection tolerance can be modified inside the **Options** dialog. Also accessible via the context menu (Right-click > **Add by Face**).
Recheck	Opens the **Auto Contact** dialog, so that you can recheck the selected interfaces.

Option	Description
	When you select interfaces from the browser, the GUI will automatically populate the components that the interaction was based on. This helps modify an existing interface.

Remarks

During the Auto Contact process, temporary components may be created for parts containing 3D elements. These temporary components will be named using a preceding ^ symbol, and will automatically contain extracted element faces needed for the contact creation process. Auto Contact will cleanup and remove these temporary components when you finish or cancel the process. If you decide to export the model before accepting or canceling the process, HyperMesh will always exclude these temporary components from export.

Set Up an Auto Contact Run

1. From the menu bar, select **Preferences** > **User Profiles**.
   The **User Profile** dialog opens.
2. Select **OptiStruct** as the profile name and click **OK**.
3. Open the **Auto Contact** dialog in one of the following ways:
   - From the menu bar, select **BCs** > **Create** > **Auto-Contact**.
   - From the menu bar, select **Tools** > **Auto-Contact**.
   - From the Utility Browser, click the **FEA** page and select **Auto-Contact** under the Tools section.
4. Click the **Type of interface** menu and then select the type of interface to create.

   The type of interface selected here will be used as initial configuration for all found interfaces, but can be individually edited afterwards in the Auto Contact Browser.
5. Next to Select components, double-click **Component**.
   The Component Selection panel opens.
6. Click **Comps** then begin to select your components.
7. Click **select**.
   The selected components will be automatically placed in the Components list inside of the **Auto Contact** dialog.
8. Click **Proceed**.
   The **Auto Contact** dialog re-appears.
9. In the Proximity distance field, enter a value for the proximity distance.

   Contact pairs will be generated between components that are closer to each other by less than this proximity distance.
10. In the Max reverse angle field, enter a value for the maximum reverse angle.

    If the angle between the normals of two elements or element faces within the proximity distance exceeds this value, the element will not be included in the contact. Default value is 15 deg.

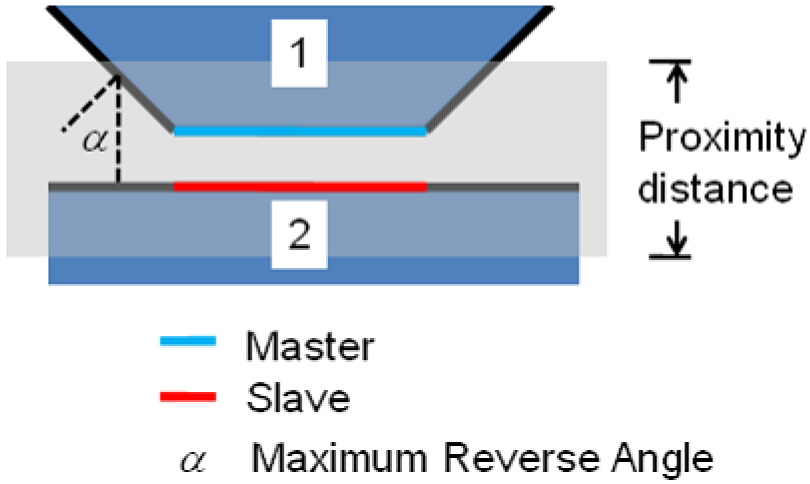


Figure 579: Definition of Proximity Distance and Maximum Reverse Angle

11. Click **Find**.
    The status bar activates. Once it is complete, the Auto Contact Browser automatically opens.
12. Use the Auto Contact Browser to review all of the found interfaces and make any necessary adjustments to contacts and contact surfaces. When you are finished, click **Create**.

    Contact surfaces and contacts are created for all items marked as Accepted. Clicking **Cancel** will close the Auto Contact Browser without creating any contacts.

Pretension Manager

Use the **Pretension Manager** tool to create and edit 1D and 3D pretension bolt loads and bolt sections for the OptiStruct solver.

This tool's location on the Utility Browser is at **Tools** > **Pretension Manager**. This tool is only available when the OptiStruct user profile is loaded.

For details on how the pretensioned bolt analysis works inside of OptiStruct, refer to Pretensioned Bolt Analysis in the OptiStruct online help.

Pretension Manager Dialog

The **Pretension Manager** uses a table format to display all the bolt pretension loads in the model. Each row represents a pretension load in the model, and each column represents the following:

Table 186:

Column	Description	Related OptiStruct Card(s)
Bolt Type	Displays the type of bolt the load is applied to. Can be 1D or 3D.	PRETENS

Column	Description	Related OptiStruct Card(s)
Bolt Id	Displays the identification number of the Bolt Pretension section the load is applied to.	PRETENS
EID/SURFID	Displays the Element (1D) or Surface (3D) identification number where the Bolt Pretension section is located.	PTFORCE, PTADJST
Load Type	Allows you to review and edit the type of pretension load, by selecting one of the following options from the pull-down menu: • Force • Adjustment Use the Control key to select and edit this field in multiple rows at once. Click **OK** or **Apply** to update the load types in the model. Click **Cancel** to exit the **Pretension Manager** without saving any changes.	PTFORCE, PTADJST
	Allows you to review and edit the Load Collector where the pretension load is organized in HyperMesh, by selecting one of the following options from the pull-down menu: **Select Existing** Takes you to the Load Collector Selector panel. **Create New** Automatically creates a new Load Collector with the name prefix PRETENS_* to organize the load into. Use the Control key to select and edit this field in multiple rows at once. Click **OK** or **Apply** to update the load types in the model. Click **Cancel** to exit the **Pretension Manager** without saving any changes.	PTFORCE, PTADJST
Load Id	Displays the HyperMesh internal identification number of the individual loads.	
Load Magnitude	Allows you to review and edit the pretension load magnitude. Use the Control key to select and edit this field in multiple rows at once. Enter a new value and press **Enter** to apply the changes to all of the selected rows.	PTFORCE, PTADJST

<table>
<tr><th>Column</th><th>Description</th><th>Related OptiStruct Card(s)</th></tr>
<tr><td></td><td>Click OK or Apply to update the load types in the model. Click Cancel to exit the Pretension Manager without saving any changes.</td><td></td></tr>
<tr><td>Orientation</td><td>(Optional) Allows you to review and edit the determination type of the normal direction of the bolt pretension section, by selecting one of the following options from the pull-down menu:
Blank
Solver default
AUTO
GRIDS
Takes you to a G1,G2,G3 node selection dialog.
VECTOR
Takes you to the Direction Selector panel.
CID
Takes you to the System Selector panel.
The Orientation type value will be updated in the model immediately after editing the selection.</td><td></td></tr>
<tr><td>Output DOF ID</td><td>(Optional) Allows you to review and edit the SPOINT that contains the pretension deformation and load, by selecting one of the following options from the pull-down menu:
Select Existing
Takes you to the Node Selector panel.
Create New
Automatically creates a new SPOINT and assigns it to the bolt pretension section.
Use the Control key to select and edit this field in multiple rows at once.
The Output DOF ID value will be updated in the model immediately after editing selection.</td><td>PRETENS</td></tr>
</table>

The **Pretension Manager** dialog contains the following buttons:

Table 187:

<table>
<tr><th>Button</th><th>Description</th></tr>
<tr><td>Add 1D Bolts</td><td>Allows you to create one or more 1D Bolt Pretension sections.
To add a 1D Bolt Pretension section:
1. Click Add 1D Bolts.
The Element Selector panel opens.
2. Select one or more beam, bar or rod elements.
3. Click Select.
4. Click Proceed.
The individual 1D Bolt Pretension sections will be created for each selected element.</td></tr>
<tr><td>Add 3D Bolts</td><td>Allows you to create one or more 3D Bolt Pretension Sections. Two options are available when you click Add 3D Bolts:
• Choose Select Existing Surface to open the Contact Surface Selector panel, where you can select existing surfaces. Once you click Select and Proceed, the individual 3D bolt Pretension Sections will be created for each selected contact surface.
• Choose Create New Surface to open the Contact Surface Create (solid faces) panel, where you need to select elements and face nodes to create the surface for the bolt pretension section, as depicted in the images below.
Step 1:
Select elements on one side of the bolt:

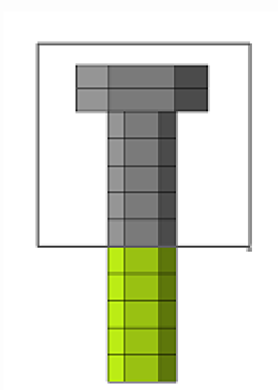



Figure 580:
Step 2:
Select nodes on the section face:</td></tr>
</table>

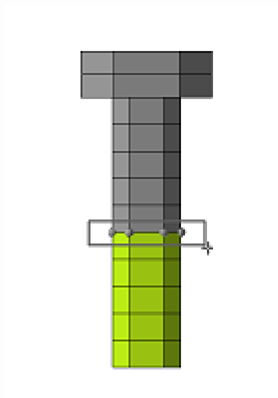
<table>
<tr><th>Button</th><th>Description</th></tr>
<tr><td></td><td>


Figure 581:

When you are using the Create New Surface option, a default name will be automatically assigned in the panel using the prefix PRETENS_* and the appropriate card image. Once you click Create and Return, you will be brought back to the Pretension Manager. The Pretension Manager will now display a 3D Bolt Pretension section for the new contact surface.
</td></tr>
<tr><td>Add Load</td><td>
Allows you to add a new pretension load to an existing Bolt Pretension section.

To add a new pretension load to an existing

Click the checkboxes in the first column for the Bolt Pretension section that you want to add a new load to.
Click Add Load to add the new load to the selected Bolt Pretension section.

The new load will be added directly below the section you wanted to add the new load to, and organized using a hierarchical tree-like display fashion.

Bolt Pretension section:

Example

The model below has four Bolt Pretension sections with IDs 1, 2, 3 and 4, and the pretension Force loads of 9500, 10000, 11500, and 9000 are applied to them, respectively.

<table>
<tr><th></th><th>Bolt Type</th><th>Bolt Id</th><th>EID/SURFID</th><th>Load Type</th><th>Loadcol</th><th>Load Id</th><th>Load Magnitude</th></tr>
<tr><td>☐</td><td>1D</td><td>1</td><td>1000</td><td>Force</td><td>PRETENS_1(1)</td><td>1</td><td>9500.0</td></tr>
<tr><td>☐</td><td>1D</td><td>2</td><td>1001</td><td>Force</td><td>PRETENS_1(1)</td><td>2</td><td>10000.0</td></tr>
<tr><td>☑</td><td>1D</td><td>3</td><td>1002</td><td>Force</td><td>PRETENS_1(1)</td><td>3</td><td>11500.0</td></tr>
<tr><td>☑</td><td>3D</td><td>4</td><td>PT_Surf(1)</td><td>Force</td><td>PRETENS_1(1)</td><td>4</td><td>9000.0</td></tr>
</table>

Figure 582:

An additional pretension Adjustment load of 0.1 needs to be applied to Bolt Sections 3 and 4 only. To do this, you must select the checkboxes
</td></tr>
</table>

<table>
<tr><th>Button</th><th>Description</th></tr>
<tr><td></td><td>
for sections 3 and 4 (as shown above), and then click Add Load. One additional pretension load is added below each of the selected bolt sections.
<table>
<tr><th></th><th>Bolt Type</th><th>Bolt Id</th><th>EID/SURFID</th><th>Load Type</th><th>Loadcol</th><th>Load Id</th><th>Load Magnitude</th></tr>
<tr><td>☐</td><td>1D</td><td>1</td><td>1000</td><td>Force</td><td>PRETENS_1(1)</td><td>1</td><td>9500.0</td></tr>
<tr><td>☐</td><td>1D</td><td>2</td><td>1001</td><td>Force</td><td>PRETENS_1(1)</td><td>2</td><td>10000.0</td></tr>
<tr><td>☐</td><td>1D</td><td>3</td><td>1002</td><td>Force</td><td>PRETENS_1(1)</td><td>3</td><td>11500.0</td></tr>
<tr><td>☐</td><td></td><td></td><td></td><td>Adjustment</td><td>PRETENS_2(2)</td><td>5</td><td>0.1</td></tr>
<tr><td>☐</td><td>3D</td><td>4</td><td>PT_Surf(1)</td><td>Force</td><td>PRETENS_1(1)</td><td>4</td><td>9000.0</td></tr>
<tr><td>☐</td><td></td><td></td><td></td><td>Adjustment</td><td>PRETENS_2(2)</td><td>6</td><td>0.1</td></tr>
</table>
Figure 583:
Looking at the left side of the table, it is easy to see that there are only four Pretension Bolt sections in the model, however, some of them have more than one pretension load applied to them.
In this example, the Force loads on all bolts are organized into Load Collector PRETENS_1, while the Adjustment loads on Bolts 3 and 4 are organized into a different Load Collector PRETENS_2, which allows them to be used in a different Load Step if desired.
</td></tr>
<tr><td>Delete</td><td>
Deletes the selected bolt pretension sections and/or pretension loads.
To delete selected bolt pretensions section:

Click the checkboxes in the first column for the rows you wish to delete.
Click Delete.

When a Bolt Pretension section has more than one load applied to it, deleting the bolt section at the top level will automatically delete all the loads applied to it. On the other hand, deleting single additional loads will only remove the selected loads.
</td></tr>
<tr><td>Review</td><td>
Allows you to graphically review the Element/Surface, Load, and SPOINT assigned to the selected Bolt Pretension section. This button also displays more detailed information about the selected Bolt Pretension section's Orientation.
To review a Bolt Pretension selection:

Click checkboxes in the first column for the Bolt Pretension section you wish you review.
Click Review.
The Review Pretension dialog opens.
Click OK in the Review Pretension dialog to return to the Pretension Manager dialog.

</td></tr>
<tr><td>Select All</td><td>Select all rows in the table.</td></tr>
</table>

<table>
<tr><th>Button</th><th>Description</th></tr>
<tr><td>Select None</td><td>Deselect all rows in the table.</td></tr>
<tr><td>Select Reverse</td><td>Reverses row selection in the table.</td></tr>
<tr><td>View</td><td>Controls the Pretension Manager table's view organization mode. There are two options available from the pull-down menu:

Choose By Bolt (default) to apply multiple loads to the same Pretension Bolt section are displayed directly below it, organized using a hierarchical tree-like display structure.
<table>
<tr><th></th><th>Bolt Type</th><th>Bolt Id</th><th>EID/SURFID</th><th>Load Type</th><th>Loadcol</th><th>Load Id</th><th>Load Magnitude</th></tr>
<tr><td>☐</td><td>1D</td><td>1</td><td>1000</td><td>Force</td><td>PRETENS_1(1)</td><td>1</td><td>9500.0</td></tr>
<tr><td>☐</td><td>1D</td><td>2</td><td>1001</td><td>Force</td><td>PRETENS_1(1)</td><td>2</td><td>10000.0</td></tr>
<tr><td>☐</td><td>1D</td><td>3</td><td>1002</td><td>Force</td><td>PRETENS_1(1)</td><td>3</td><td>11500.0</td></tr>
<tr><td>☐</td><td></td><td></td><td></td><td>Adjustment</td><td>PRETENS_2(2)</td><td>5</td><td>0.1</td></tr>
<tr><td>☐</td><td>3D</td><td>4</td><td>PT_Surf(1)</td><td>Force</td><td>PRETENS_1(1)</td><td>4</td><td>9000.0</td></tr>
<tr><td>☐</td><td></td><td></td><td></td><td>Adjustment</td><td>PRETENS_2(2)</td><td>6</td><td>0.0</td></tr>
</table>
Figure 584:

Choose All to apply multiple loads to the same Pretension Bolt section are displayed directly below it, using a flat display structure with no hierarchical organization. The Bolt Pretension section's Type and ID are displayed for all pretension loads in the model.
<table>
<tr><th></th><th>Bolt Type</th><th>Bolt Id</th><th>EID/SURFID</th><th>Load Type</th><th>Loadcol</th><th>Load Id</th><th>Load Magnitude</th></tr>
<tr><td>☐</td><td>1D</td><td>1</td><td>1000</td><td>Force</td><td>PRETENS_1(1)</td><td>1</td><td>9500.0</td></tr>
<tr><td>☐</td><td>1D</td><td>2</td><td>1001</td><td>Force</td><td>PRETENS_1(1)</td><td>2</td><td>10000.0</td></tr>
<tr><td>☐</td><td>1D</td><td>3</td><td>1002</td><td>Force</td><td>PRETENS_1(1)</td><td>3</td><td>11500.0</td></tr>
<tr><td>☐</td><td>1D</td><td>3</td><td>1002</td><td>Adjustment</td><td>PRETENS_2(2)</td><td>5</td><td>0.0</td></tr>
<tr><td>☐</td><td>3D</td><td>4</td><td>PT_Surf(1)</td><td>Force</td><td>PRETENS_1(1)</td><td>4</td><td>9000.0</td></tr>
<tr><td>☐</td><td>3D</td><td>4</td><td>PT_Surf(1)</td><td>Adjustment</td><td>PRETENS_2(2)</td><td>6</td><td>0.1</td></tr>
</table>
Figure 585:

</td></tr>
</table>

Load Collector Table

Use the **Load Collector Table** utility to review and edit OptiStruct load collectors.

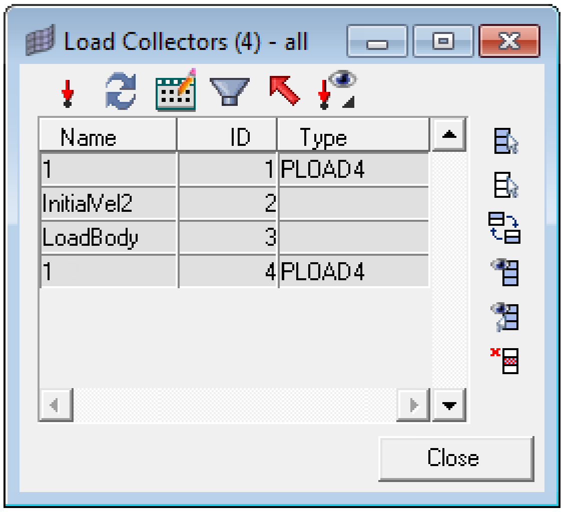


Figure 586:

Table 188:

Icon	Description
	Refresh table
	Toggles between table edit and table display mode
	Filter table
	Export table to CSV format
	Select all
	Select none
	Reverse selection
	Select displayed
	Delete selected

Buckling

The Buckling utility allows a buckling subcase to be defined simultaneously with its referenced static subcase.

1. Enter a name.
2. Provide data for V1, V2 (upper and lower limits for eigenvalue calculation) and/or ND (number of modes to be calculated).
3. Select appropriate load collectors for LOAD and SPC references.

   Only load collectors containing valid loads are displayed in the drop-down menus.
4. Click **Create**.
   The EIGRL load collector is created. The static and buckling load steps are also created.

Opti Page

Macros available on the Opti page of the OptiStruct Utility menu.

Topology

Generate Voxelmesh
Creates voxels (hexa elements) from closed shell meshes.

Hex-core
Creates hybrid mesh for closed volume. The core portion of the volume will be meshed with hex elements, clearance between boundary and hex mesh will be meshed with BL + tetramesh + pyramid coupling.

Design Space
Shortcut to the Topology panel.

Matfrac
Setup a material fraction topology optimization.

Reg. Volfrac
Create regional volfrac responses combining several components, properties or materials.

Topography

Design Space
Shortcut to the Topology panel.

Shape

Create Shapes
Shortcut to the HyperMorph panels.

Shape Variables
Shortcut to the Shape panel.

Size

Size Variables

Shortcut to the Size panel.

PBAR, PROD Opti.

Define size optimization for multiple PBAR or PROD sections.

CBAR, CROD Opti.

Define size optimization for multiple CBAR or CROD elements with circular section.

Optimization Info

Design Variables

Create, review and edit Size (including Gauge) and Shape Design Variables.

Design Constraints

Review and edit optimization constraints.

Solution

OptiStruct

Shortcut to the OptiStruct panel.

OSSmooth

Shortcut to the OSSmooth panel.

OSSmooth Volume

Compute the volume enclosed by iso-density surface (Elements Displayed)

After running OSSmooth this helps you to compute the true volume of the geometry recovered. The iso-surface must be displayed as elements. Use Nastran or STL format when running OSSmooth.

Generate Voxelmesh

Fill an enclose volume with voxels (hexas) of a predefined size.

Available for the OptiStruct user profile on Windows, LINUX, IRIX, HPUX and SUN.

This type of mesh is only useful in topology optimization. It does not give meaningful results in a stress analysis.

To work properly, the volume must be enclosed completely by shell elements (quads and trias) without T-Connections or free edges. The normals of these elements should point inwards. The voxels (hexa elements) are stored in the component called hexas.

1. On the Opti page of the Utility menu, under Topology, click **Voxelmesh**.
   A comps collector displays in the panel area.
2. Select components that contain shell elements enclosing one volume.
   If more than one volume is selected, normals should be adjusted manually.
3. Click **proceed**.
   The **Voxelmesh** dialog is launched.
4. Check the relevant boxes.

Option	Description
Perform element check	Checks for T-connections and free edges. If some are found, the results are stored in collectors of the corresponding names.
Adjust normals	Automatic adjustment of normals (to inward). This works if the selection is one connected volume only. The volume may contain internal voids.
Fill undercuts	Areas that are hidden in each coordinate direction are filled even if they are not touching the enclosed volume. These elements are stored in the component called hexasfill.
One component for each number of inner nodes	The voxels created are stored in nine components (hexas0, hexas1...) depending on the number of nodes that are inside the volume. **Note:** Zero inner nodes may occur if one edge of the volume intersects the center of a hexa-face.
Use local coordinates	Allows selecting a coordinate system along which to align the mesh. If no selection is made, the global (basic, screen) coordinate system is used.

Option	Description
Edge size for hexa elements	Choose from Cubes or Rectangles. For cubes, enter a single value for the edge size; for rectangles enter x, y, and z edge lengths. Keep in mind that a grid of nodes is created for the box wrapping the volume, so the memory usage may be high for unreasonably small values.

5. Click **Start**.
6. If you checked the Use local coordinates box, you will be prompted to select a coordinate system. Select the system and click **proceed**.
   The voxelmesh is generated.

Matfrac

Use the Material Fraction optimization utility to set up an optimization problem to minimize the combined compliance index and constrain the volume fraction to be less than a user-defined value.

1. Click **Matfrac**.
2. Enter a matfrac value.
3. Click **calculate**.
   The objective is set to minimize COMB for the entire model, while constraining the VOLFRAC for the entire model to be less than the MATFRAC value entered.

Reg. Volfrac

The Reg. Volfrac utility creates a regional volume fraction response for a number of components, properties or materials.

1. Click **Reg. Volfrac**.
   You are prompted to select a number of components, properties or materials.
2. Select the entities.

   Volume responses are generated for each entity (vol#). An equation is then created to calculate the total volume fraction over the entire region. A function response is generated using the equation with reference to the volume responses.

PBAR, PROD Opti.

Use this utility to set-up size optimization for multiple PBAR or PROD properties.

1. Click **PBAR, PROD Opti**.
   You are prompted to select a number of properties.
2. Select **properties**.
3. Select either a **PBAR** or a **PROD** card and a section type.

4. Enter initial values and bounds.
5. Click **calculate**.
   Depending on the cross section, a number of design variables, equations, and DVPREL2 cards are generated.

CBAR, CROD Opti.

Use this utility to setup size optimization for multiple CBAR or CROD elements.

1. Click **CROD, CBAR Opti**.
   You are prompted to select 1D elements for topological optimization.
2. Select **elements**.
3. Select **PBAR** or **PROD**.
   A new PBAR or PROD property is created for each 1D element selected. These properties are called DPROP#. A size design variable is created for each 1D element selected.
4. Input the initial value and upper and lower limits for this variable.

   The design variables are called DV#. These design variables represent the cross-sectional area of each 1D element.

   A DVPREL1 card is created linking each of the DV#'s to the cross-sectional area of each DPROP# card. These are called DX#.

   For simplicity, assume the 1D elements have solid, circular cross-sections. Two equations are created to calculate the I and J values if the PBAR property type is chosen:

   a) Equation for Ix and Iy values : Y(X1) = 0.0796*X1**2

   b) Equation for J values: Y(X1) = 0.1592*X1**2

   A DVPREL2 card is created for the I1 value of each DPROP#, referencing equation (a) and the appropriate design variable. These are named DA#.

   A 2nd DVPREL2 card is created for the I2 value of each DPROP#, referencing equation (a) and the appropriate design variable. These are named DB#.

   A 3rd DVPERL2 card is created for the J value of each DPROP#, referencing equation (b) and the appropriate design variable. These are named DC#.

   A 3rd equation is created to calculate the regional volume fraction for all DPROP# properties:

   Y(X1,Xi,....,Xn) = (X1+Xi+....+Xn)/(X1o+Xio+...+Xno)

   where Xi is the value of DVi, and where Xio is initial value of Xi.

   A function response, called BVFRAC, is generated using this equation and referencing all design variables.

Design Variables

Use the Design Variables utility to review and edit OptiStruct size, including gauge, and shape design variables.

All DESVAR design variables are listed in the table.

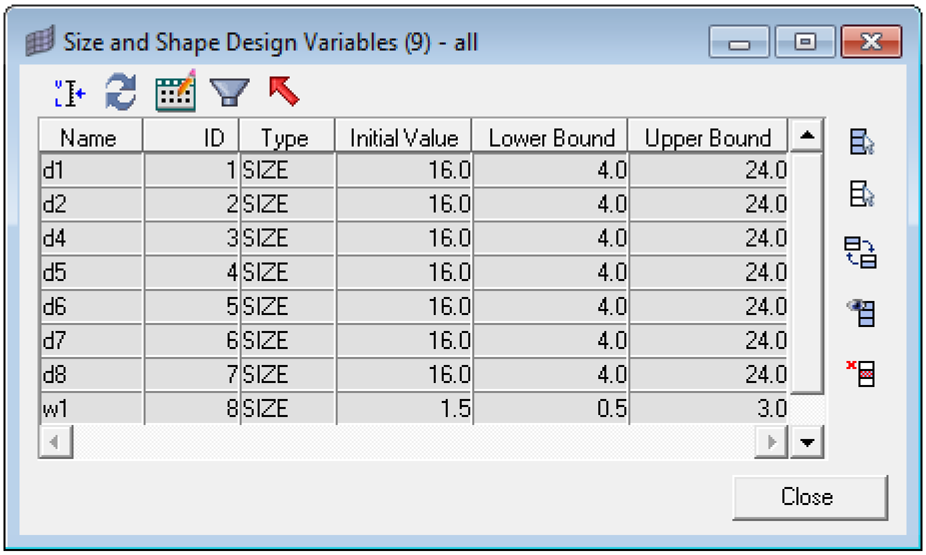
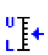


Figure 587:

Table 189:

Icon	Description
	Create new design variables
	Refresh table
	Toggles between table edit and table display mode
	Filter table
	Export table to CSV format
	Select all
	Select none
	Reverse selection

Icon	Description
	Select displayed
	Delete selected

Create a Design Variable

1. Click .
   The **Create Design Variable** dialog opens.
2. Fill in the fields as desired.
3. Click **Create**.

Edit a Design Variable

1. In the table, click the Initial Value, Lower Bound or Upper Bound field of a design variable.
2. Replace the current value with the desired value.
3. Press the Enter key.
   The design variable is updated.

Design Constraints

Use the Design Constraints utility to review OptiStruct optimization constraints.

All optimization constraints are listed in the table.

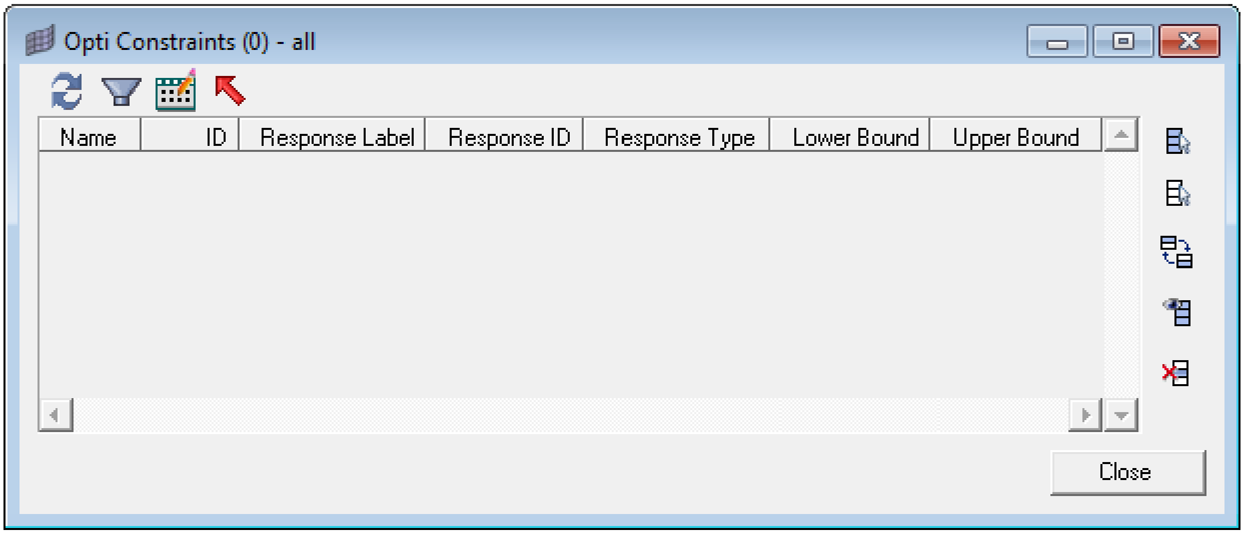
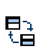


Figure 588:

Table 190:

Icon	Description
	Refresh table
	Toggles between table edit and table display mode
	Filter table
	Export table to CSV format
	Select all
	Select none
	Reverse selection
	Select displayed
	Delete selected

Visualization Controls

Use the Visualization controls to change many of the visual states in HyperMesh and HyperView.

To access the Visualization controls, click on the Display toolbar. There are nine different types of visualization controls:

Connectors

Connector size

Specify a size (in model units) to display connectors in the graphics area.

Color by

Select a Color by scheme (State, Layer, Style, or Component) to change how connectors are color-coded in the graphics area. You cannot change the colors of each individual item within a scheme, but you can change what items display. Select or clear the check boxes next to each item to turn them on and off in the scheme.

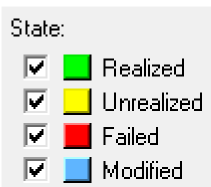


Figure 589:
Colors are locked, but clearing an item's check box hides it in the graphics area.

The Color by schemes and their corresponding color items include:

State
Realized, Unrealized, Failed, and Modified

Layer
< 2t, 2t, 3t, > 3t

Style
Apply mass, area, bolt, seam, spot

Component
HyperMesh colors the Connectors based on the color of the component axis they exist in.

State/Layer/Style

HyperMesh only displays the connectors that match the selected checkboxes in the graphics area. If you clear a checkbox, HyperMesh removes the connectors that match that criteria from the graphics area. You can use any combination of check boxes at a time.

Cylinder Bolts

Change the display of cylinder bolts in the graphics area using the following controls:

Cylinder transparency
Move the slider to change the level of transparency of cylinder bolts in the graphics area.

Display status

Toggle the Display status on and off to show or hide cylinder bolts in the graphics area.

CWELDS

Change the display of CWELD defined connectors in the graphics area by switching the CWELD visualization to on.

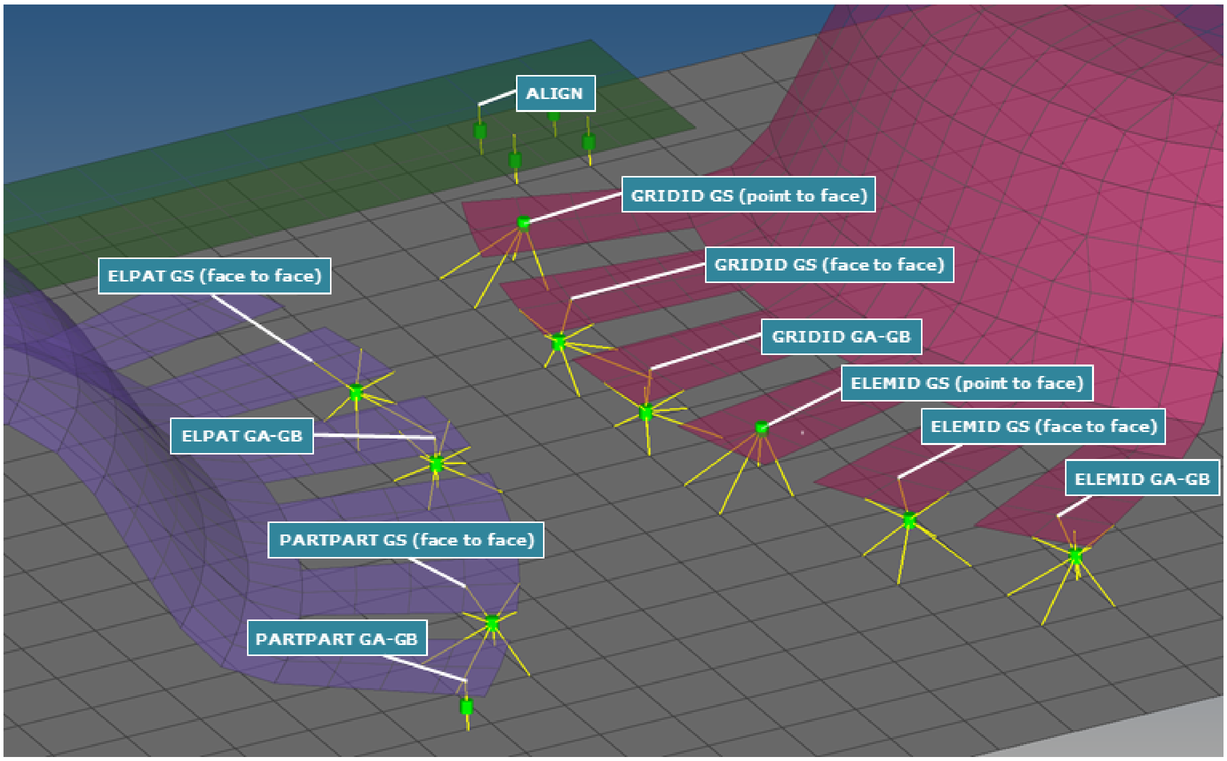


Figure 590:

The graphical representation reflects the CWELD definition, and points directly from the connector to the referenced nodes, nodes of referenced elements, or referenced properties. The graphical representation is always shown in the appropriate CWELD component color.

Certain CWELD manipulations are not automatically synced. Editing the solver card or organizing the element into a new component requires you to reactivate the CWELD visualization.

Constraints

Constraint labels

Select this checkbox to display labels on constraint entities in the graphics area.

Relative constraint size

Specify how large, in model units, constraints are displayed relative to the model size.

Equations

Equation Handles

Select this checkbox to display the handles on equations in the graphics area.

Loads

Template labels (type)

Select this checkbox to display template-specific labels on loads in the graphics area. If you clear this checkbox, HyperMesh will display general HM labels on loads.

Load labels

Select this checkbox to display the labels of load entities (forces, moments, pressures, accelerations, velocities, temperatures, fluxes) in the graphics area.

Change load vector (tip/tail) at application point

Select this checkbox to point the load vector away from the load application point (tail) when the tip of the load vector is attached to the load application point. If you clear this checkbox, HyperMesh points the load vector toward the load application point (tip) when the tail of the load vector is attached to the load application point.

> **Note:** The direction of the vector does not change when you select this option.

Load size

Specify a size (in model units) to display load entities in the graphics area. Use the toggle to select **relative size** or **Uniform** to control the size of load entities. Select Relative size to displays loads relative to the model size (default 100), or select Uniform to apply a constant size.

Morphing

Handle Size

Specify a size (in model units) to display morphing handles in the graphics area.

Symmetry Size

Specify a size (in model units) to display morphing symmetries in the graphics area.

Colors

To change how morphing domains, faces, volumes and symmetries are color-coded in the graphics area, click the color boxes and select a new color from the color selector.

Morph volume edges, Edge points

Specify how many points each morph volume edge will use to represent curves in the graphics area. Use larger numbers for smoother curves and smaller numbers for faster rendering.

Morph volume faves

Use these options to choose how HyperMesh draws the flat faces of morph volumes in the graphics area:

Wireframe

Face has no color at all; only the morph volume edges and handles are drawn.

Transparent
(1, 2, or 3) means that the faces are drawn semi-opaque; higher numbers mean more opacity.

Opaque
Sets the morph volumes' transparency to zero.

Systems

Relative system size
Specify how large, in model units, systems are displayed relative to the model size.

Tags

Push tags to foreground
Select this checkbox to push all of the active tags to the foreground in the graphics area. If the entity the tag is attached to is visible in the graphics area, then HyperMesh will also display the tag. If the entity is not visible in the graphics area, then HyperMesh will hide the tag completely.

Tag text
From the Tag text list, select what type of text to display with the tags in the graphics area. You can choose from the following: only the Label, only the Body, both (Label:Body), or Description.

Tag icon
From the Tag icon list, select how to display the tag icons in the graphics area. You can choose from the following: Text Only, Text and Icon, or Icon Only.

Topology

Show line directions
Select this checkbox to display graphics on line entities in the graphics area to indicate the direction of the line.

Edges/Shaded faces on solid
Control the color-coding of the respective entities in the graphics area, and the display of those entities that match the selected criteria. The changes you make to the color options will only apply when used within the relevant geometry shading and color modes.

Solid transparency
Move the slider to change the level of transparency of the solid entities in the graphics area when they are used within the relevant geometry shading and color modes. The higher the number, the more transparent the solid entities appear.

Mappable solids
Control the color-coding of mappable solids when used with the relevant geometry shading and color modes.

Vectors

Vector Handles
Select this checkbox to display handles on vector entities in the graphics area.

Vector labels
Select this checkbox to display labels on vector entities in the graphics area.

Vector size

Specify a size (in model units) to display vector entities in the graphics area. Use the toggle to select **Vector size %** or **Uniform** to control the size of vector entities. Select Vector size % to use a percentage of the actual length of the vector to determine the size, or select Uniform to apply a constant size.

Geometry

6

Create and edit geometry.

This chapter covers the following:

Geometry Settings

Overview of settings used when creating and editing geometry.

CAD Cleanup Tolerance

The CAD cleanup tolerance is used to determine if two surface edges are the same and if two surface vertices are the same.

CAD cleanup tolerance controls:

- The determination of if two surface edges are close enough to be automatically combined (creating shared edges)
- If a surface is degenerate and should be removed

If you use the automatic setting, the complexity of the surface and edge geometries are taken into account and a tolerance is selected to maximize the number of shared edges. To specify a manual cleanup tolerance value, it must be greater than the default value. The readers only modifies data if the data stays within the original data tolerance.

Increasing the tolerance may cause problems. When this value is modified, any features equal to or less than the tolerance are eliminated. The readers do not include any edge with a length less than the tolerance; if there are edges present that are important to the surface, that surface will be distorted, or will fail to trim properly. Similarly, surfaces smaller than the tolerance may not be imported.

If the file you have read has many very short edges, it may be worthwhile to reread the file using a larger tolerance. The same holds true if surfaces appear to be "inside out" when surface lines are displayed.

The tolerance value should not be set to a value greater than the node tol used for your element mesh, set in the Options panel.

Geometry Cleanup Tolerance

Geometry cleanup tolerance specifies how much the geometry can be modified in the course of cleaning it, either manually or automatically. Cleaning up refers to fixing geometry data by creating proper topology, defeaturing, and eliminating extraneous vertices.

Since the geometry is approximated with a finite element mesh, a cleanup tolerance that is less than the node tolerance used in the mesh generation is required.

The tolerance value should not be set to a value greater than the node tol used for your element mesh, set in the Options panel.

Geometry Feature Angle

Geometry feature angle determines when model geometry should have a new vertex added (creating two surfaces from one) or removed (merging two surfaces into one).

The angle is measured between the positive normal faces of adjacent geometric entities.

Create, Edit, Query Geometry

Create, edit, and query various geometric features, such as nodes, points, line, and surfaces.

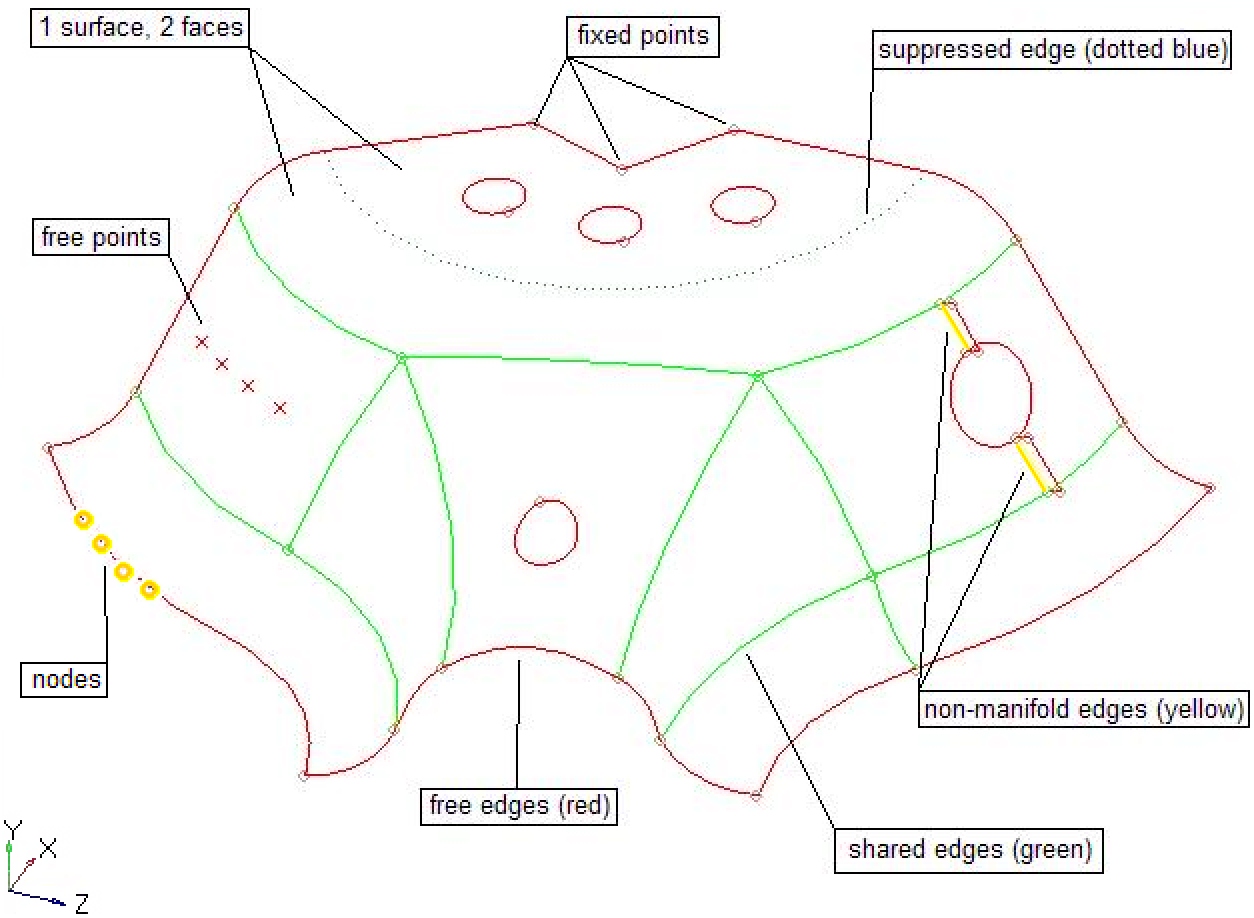


Figure 591:

Nodes

Nodes are the most basic finite element entity. A node represents a physical position on the structure being modeled.

A nodes is used by an element entity to define the location and shape of that element. It is also used as temporary input to create geometry entities.

A node may contain a pointer to other geometric entities and can be associated directly to them.

Nodes are considered to be used if they are referenced in the definition of an element, system, vector, group, load, equation, or are referenced by any card image on any HyperMesh entity. Unused nodes and any loads that are attached to unused nodes are automatically deleted .

Nodes can not be organized into components. Nodes can also be organized into HyperMesh include files, which defines the solver include file they will be exported to.

A node as displayed a small circle or sphere, depending on the mesh graphics mode. Its color is always yellow.

Create Nodes

- xyz - Creates by specifying (x,y,z) coordinates (Nodes panel).
- on geometry - Creates at graphically selected locations on points, lines, surfaces and planes (Nodes panel).
- arc center - Creates nodes at the center of the arc that best approximates the input set of nodes, points or lines (Nodes panel).
- extract parametric - Creates nodes at parametric locations on lines and surfaces (Nodes panel).
- extract on line - Creates evenly spaced or biased nodes on a selection of lines (Nodes panel).
- interpolate nodes - Creates evenly spaced or biased nodes by interpolating between existing nodes in space (Nodes panel).
- interpolate on line - Creates evenly spaced or biased nodes by interpolating between existing nodes on a line (Nodes panel).
- interpolate on surface - Creates evenly spaced or biased nodes by interpolating between existing nodes on a surface (Nodes panel).
- intersect - Creates nodes at the intersection of geometric entities: lines/lines, lines/surfaces, lines/solids, lines/planes, vector/lines, vector/surfaces, vector/solids and vector/plane (Nodes panel).
- temp nodes - Creates nodes by duplicating existing nodes or creating nodes on existing geometry or elements (Temp Nodes panel).
- circle center - Creates nodes at the center of the circle defined by exactly three nodes (Distance panel).
- duplicate - Creates nodes by duplicating existing nodes. This is available in many panels when the "duplicate" advanced entity selector is available on a nodes input collector.
- on screen - Creates nodes by pre-selecting existing geometry or elements and clicking on the locations to create the nodes. This is available in any panel that has a node or node list input collector (Picking nodes on geometry or elements).
- Misc. API commands that do not have an associated panel.

Edit Nodes

- clear - Deletes temp nodes (Temp Nodes panel).
- associate - Associates nodes to fixed points, surface edges and surfaces by moving them onto those entities (Node Edit panel).
- move - Moves nodes along surfaces (Node Edit panel).
- place - Places nodes on a surface at a specified location (Node Edit panel).
- remap - Moves nodes by mapping them from one line or surface edge to another (Node Edit panel).
- align - Aligns/projects nodes to an imaginary line (Node Edit panel).
- find - Create temp nodes by finding FE nodes associated with other FE entities (Find panel).

- translate - Moves nodes along a vector direction (Translate panel).
- rotate - Rotates nodes about a vector axis (Rotate panel).
- scale - Scales the dimensions of nodes either proportionally or uniformly (Scale panel).
- reflect - Reflects nodes about a plane to create a mirror image (Reflect panel).
- project - Projects nodes onto a plane, vector, line/surface edge or surface (Project panel).
- position - Translates and rotate nodes into new positions (Position panel).
- permute - Switches the coordinates of nodes (Permute panel).
- renumber - Renumbers nodes (Renumber panel).
- Misc. API commands that do not have an associated panel.

Query Nodes

- card editor - With an appropriate template loaded, the card editor can be used to review node information (Card Editor panel).
- distance - Finds the distance between two nodes (Distance panel).
- shortest distance - Finds the shortest distance between entities (Shortest Distance dialog).
- angle - Finds the angle between three nodes (Distance panel).
- organize - Moves nodes into different include files (Organize panel).
- numbers - Displays the IDs of nodes (Numbers panel).
- count - Counts the total or displayed nodes (Count panel).
- Misc. API commands that do not have an associated panel.

Supported Solver Cards

Solver cards supported for nodes.

Abaqus

Card	Description
*NFILL	Generates nodes for a region of a mesh by filling in nodes between two bounds. **Note:** Resolved to individual entities on import, and are written back on export the same way.
*NGEN	Generates nodes incrementally.

Card	Description
	Note: Nodes can only be created incrementally between two nodes. Nodes generated along a parabola or curve are not supported (LINE=P or C parameter is not supported). *SYSTEM* parameter is currently unsupported. Resolved to individual entities on import, and are written back on export the same way.
*NODE	Defines nodal coordinates. **Note:** The *SYSTEM* parameter is created automatically during export based upon the type of reference coordinate system that is assigned to the nodes. The card image for a node is displayed in global Cartesian coordinates in HyperMesh.

ANSYS

Card	Description
N	Defines a node.
N	Defines a node.
N	Defines a node.

Card	Description
NBLOCK	Defines a node.

EXODUS

Card	Description
NODE	

LS-DYNA

Card	Description
*NODE	Defines a node and its coordinates in the global coordinate system. **Note:** Card can be previewed, but not edited.
*NODE_RIGID_SURFACE	Defines a rigid node and its coordinates in the global coordinate system. **Note:** Card can be previewed, but not edited.

Nastran

Card	Description
GRID	Defines the location of a geometric grid point, the directions of its displacement, and its permanent single-point constraints. **Note:** Permanent single point constraint field supported for feinput only. On export, equivalent SPC cards are output.
SPOINT	Defines scalar points.

Card	Description
	Note: Supported the same way `GRID` is supported. On import or export, all nodes that are designated to be SPOINT will be converted to nodes at the origin.

OptiStruct

Card	Description
GRID	Defines the location of a geometric grid point of the structural model, the directions of its displacement, and its permanent single-point constraints. **Note:** Bulk Data Entry Exported in large field format by the `optistructlf` template.
SPOINT	Defines sets of single-point constraints, enforced displacements for static analysis, and thermal boundary conditions for heat transfer analysis. **Note:** Bulk Data Entry Ideally a scalar point has no location, but in HyperMesh it is represented as a node.

PAM-CRASH

Card	Description
CNODE/	Defines a common node.
NODE/	Defines a node.

Note: Use the `find_cnodes` summary template to highlight all CNODE nodes as temporary nodes.

Permas

Card	Description
$COOR	Defines nodal points and their coordinates.

Radioss

Card	Description
/CNODE	Defines the coordinate of common node, which could be merged to the nearest selected NODE or CNODE. **Note:** Block Format Keyword
/NODE	Defines the coordinate of node, which is fundamental unit of graphic in structure. **Note:** Block Format Keyword

Samcef

Card	Description
.NOE	Defines the coordinates of the nodes of the structure.

Points

A point is a zero-dimensional geometric entity.

A free point is a zero-dimensional geometry entity in space that is not associated with a surface. It is displayed as a small "x", and its color is determined by the component collector to which it belongs. These types of points are typically used for weld locations and connectors.

A fixed point is a zero-dimensional geometry entity that is associated with a surface. It is displayed as a small "o", and its color is determined by the surface to which it is associated. The automesher places a node at each fixed point on the surface being meshed. A fixed point that is placed at the junction of three or more non-suppressed edges is called a vertex or vertex point. Such vertices cannot be suppressed (removed).

Create Points

Create Free Points

- xyz - Creates free points by specifying (x,y,z) coordinates (Points panel).
- arc center - Creates at the center of the arc that best approximates the input set of nodes, points or lines (Points panel).
- extract parametric - Creates free points at parametric locations on lines and surfaces (Points panel).
- intersect - Creates free points at the intersection of geometric entities: lines/lines, lines/surfaces, lines/solids, lines/planes, vector/lines, vector/surfaces, vector/solids and vector/plane (Points panel).
- suppressed fixed points - Creates free points at suppressed fixed point locations (Point Edit panel).
- circle center - Creates free points at the center of the circle defined by three free or fixed points (Distance panel).
- duplicate - Creates free points by duplicating existing free or fixed points. This is available in many panels when the "duplicate" advanced entity selector is available on a points collector.
- Misc. API commands that do not have an associated panel.

Create Fixed Points

- by cursor - Creates fixed points at cursor locations on surfaces and surface edges (Point Edit panel, Quick Edit panel).
- on edge - Creates fixed points at uniform locations on a surface edge (Point Edit panel, Quick Edit panel).
- on surface - Creates fixed points at existing node/free point locations on/near a surface (Point Edit panel).
- project - Creates fixed points on surface edges by projecting existing free or fixed points (Point Edit panel, Quick Edit panel).
- defeature pinholes - When defeaturing pinholes, fixed points are created at the center of the each removed pinhole (Defeature panel).

- Misc. API commands that do not have an associated panel.

Edit Points

Edit Free Points

- delete - Deletes free points (Delete panel).
- translate - Moves free points along a vector direction (Translate panel).
- rotate - Rotates free points about a vector axis (Rotate panel).
- scale - Scales the dimensions of free points either proportionally or uniformly (Scale panel).
- reflect - Reflects free points about a plane to create a mirror image (Reflect panel).
- project - Projects free points onto a plane, vector, line/surface edge or surface (Project panel).
- position - Translates and rotate free points into new positions (Position panel).
- permute - Switches the coordinates of free points (Permute panel).
- renumber - Renumbers free points (Renumber panel).
- Misc. API commands that do not have an associated panel.

Edit Fixed Points

- suppress/remove - Suppresses non-vertex fixed points (Point Edit panel, Quick Edit panel).
- replace - Combines multiple fixed points by moving them to one fixed point location (Point Edit panel, Quick Edit panel).
- release - Releases fixed point vertices such that any shared edges attached to the point become free edges (Point Edit panel, Quick Edit panel).
- renumber - Renumbers fixed points (Renumber panel).
- Misc. API commands that do not have an associated panel.

Query Points

Query Free Points

- distance - Finds the distance between two free points (Distance panel).
- shortest distance - Finds the shortest distance between entities (**Shortest Distance** dialog).
- angle - Finds the angle between three free points (Distance panel).
- organize - Moves free points into different component collectors (Organize panel).
- numbers - Displays the IDs of free points (Numbers panel).
- count - Counts the total or displayed free points (Count panel).

- Misc. API commands that do not have an associated panel.

Query Fixed Points

- distance - Finds the distance between two fixed points (Distance panel).
- shortest distance - Finds the shortest distance between entities (**Shortest Distance** dialog).
- angle - Finds the angle between three fixed points (Distance panel).
- numbers - Displays the IDs of fixed points (Numbers panel).
- count - Counts the total or displayed fixed points (Count panel).
- Misc. API commands that do not have an associated panel.

Lines

A line represents a curve in space and is not attached to any surface or solid. A line is a one-dimensional geometric entity.

The color of a line is determined by the component collector to which it belongs.

A line can be composed of one or more line types. Each line type in a line is referred to as a segment. The end point of each line segment is connected to the first point of the next segment. A joint is the common point between two line segments. Line segments are maintained as a single line entity, so operations performed on the line affect each segment of the line. In general, HyperMesh automatically uses the appropriate number and type of line segments to represent the geometry.

All lines in HyperMesh are represented mathematically with the following formulations:

- straight
- elliptical
- NURBS

Lines are different from surface edges and are sometimes handled differently for certain operations.

Create Lines

- xyz - Create lines by specifying (x,y,z) coordinates (Lines panel).
- linear nodes - Creates linear lines between nodes (Lines panel).
- standard nodes - Creates standard lines between nodes (Lines panel).
- smooth nodes - Creates smooth lines between nodes (Lines panel).
- controlled nodes - Creates controlled lines between nodes (Lines panel).
- drag along vector - Creates lines by dragging nodes a specified distance along a vector (Lines panel).
- arc center and radius - Creates arcs by specifying the center and radius (Lines panel).
- arc nodes and vector - Creates arcs by specifying two nodes and a vector (Lines panel).
- arc three nodes - Creates arcs by specifying three nodes on the circumference (Lines panel).

- circle center and radius - Creates circles by specifying the center and radius (Lines panel).
- circle nodes and vector - Creates circles by specifying two nodes and a vector (Lines panel).
- circle three nodes - Creates circles by specifying three nodes on the circumference (Lines panel).
- conic - Creates conic lines by specifying the start, end and tangent locations (Lines panel).
- extract edge - Creates lines as copies of surface edges (Lines panel).
- extract parametric - Creates lines at parametric locations on surfaces (Lines panel).
- intersect - Creates lines at the intersection of geometric entities: plane/lines, plane/surfaces, plane/elements, plane/plane and surfaces/surfaces (Lines panel).
- manifold - Creates linear and smooth lines on surfaces using nodes (Lines panel).
- offset - Creates lines by offsetting lines a uniform or variable distance (Lines panel).
- midline - Creates lines by interpolating between existing lines (Lines panel).
- fillet - Creates fillet lines between free lines (Lines panel).
- tangent - Creates tangent lines between a line and a node list or line (Lines panel).
- normal to geometry - Creates lines perpendicular to lines, surfaces and solids from node or point locations (Lines panel).
- normal from geometry - Creates lines perpendicular from node or point locations on lines, surfaces and solids (Lines panel).
- normal 2D on plane - Creates lines that lie on a plane, are perpendicular to a line, and are defined from node or point locations (Lines panel).
- features - Creates lines from element features (Lines panel).
- duplicate - Creates lines by duplicating existing lines. This is available in many panels when the "duplicate" advanced entity selector is available on a lines collector.
- Misc. API commands that do not have an associated panel.
- Additional capabilities are available in solidThinking and solidThinking Inspire.

Edit Lines

- delete - Deletes lines (Delete panel, Lines panel).
- combine - Combines two lines into one (Line Edit panel).
- split at point - Splits lines at graphically selected locations (Line Edit panel).
- split at joint - Splits lines at segment end points (Line Edit panel).
- split at line - Splits lines by using a cut line (Line Edit panel).
- split at plane - Splits lines at plane intersection locations (Line Edit panel).
- smooth - Smooths segmented lines (Line Edit panel).
- extend - Extends lines by either a specified distance, or to meet an existing node, point, line/surface edge or surface (Line Edit panel).
- translate - Moves lines along a vector direction (Translate panel).
- rotate - Rotates lines about a vector axis (Rotate panel).
- scale - Scales the dimensions of lines either proportionally or uniformly (Scale panel).
- reflect - Reflects lines about a plane to create a mirror image (Reflect panel).

- project - Projects lines onto a plane, vector or surface (Project panel).
- position - Translates and rotate lines into new positions (Position panel).
- permute - Switches the coordinates of lines (Permute panel).
- renumber - Renumbers lines (Renumber panel).
- Misc. API commands that do not have an associated panel.
- Additional capabilities are available in solidThinking and solidThinking Inspire.

Query Lines

- shortest distance - Finds the shortest distance between entities (**Shortest Distance** dialog).
- length - Finds the total length of selected lines/surface edges (Lines panel).
- organize - Moves lines into different component collectors (Organize panel).
- numbers - Displays the IDs of lines/surface edges (Numbers panel).
- count - Counts the total or displayed line/surface edges (Count panel).
- Misc. API commands that do not have an associated panel.

Surfaces

A surface represents the geometry associated with a physical part. A surface is a two-dimensional geometric entity that may be used in automatic mesh generation.

The color of a surface is determined by the component collector to which it belongs.

A surface is comprised of one or more faces. Each face contains a mathematical surface and edges to trim the surface, if required. When a surface has several faces, all of the faces are maintained as a single surface entity. Operations performed on the surface affect all the faces that comprise the surface. In general, HyperMesh automatically uses the appropriate number of and type of surface faces to represent the geometry.

The perimeter of a surface is defined by edges. There are four types of surface edges.

Surface edges are different from lines and are sometimes handled differently for certain operations.

The connectivity of surface edges constitutes the geometric topology.

Free Edges

A free edge is an edge that is owned by only one surface.
Free edges are colored red by default.
On a clean model consisting of surfaces, free edges appear only along the outer perimeter of the part and around any interior holes. Free edges that appear between two adjacent surfaces indicate the existence of a gap between the two surfaces. The automesher will leave a gap in the mesh wherever there is a gap between two surfaces.

Shared Edges

A shared edge is an edge that is owned, or shared, by two adjacent surfaces.
Shared edges are colored green by default.

When the edge between two surfaces is a shared edge, there is no gap or overlap between the two surfaces - they are geometrically continuous. The automesher always places seed nodes along the length a shared edge and will produce a continuous mesh without any gaps along that edge. The automesher will not construct any individual elements that cross over a shared edge.

Suppressed Edges

A suppressed edge is shared by two surfaces but it is ignored by the automesher.
Suppressed edges are colored blue by default.
Like a shared edge, a suppressed edge indicates geometric continuity between two surfaces but, unlike a shared edge, the automesher will mesh across a suppressed edge as if it were not even there. The automesher does not place seed nodes along the length of a suppressed edge and, consequently, individual elements will span across it. By suppressing undesirable edges you are effectively combining surfaces into larger logical meshable regions.

Non-Manifold Edges

A non-manifold edge is owned by three or more surfaces.
Non-manifold edges are colored yellow by default.
They typically occur at "T" intersections between surfaces or when two or more duplicate surfaces exist. The automesher always places seed nodes along their length and will produce a continuous mesh without any gaps along that edge. The automesher will not construct any individual elements that cross over a T-joint edge. These edges cannot be suppressed.

Create Surfaces

- square - Creates two-dimensional square surface primitives (Surfaces panel, Planes panel).
- cylinder full - Creates three-dimensional full cylinder surface primitives (Surfaces panel, Cones panel).
- cylinder partial - Creates three-dimensional partial cylinder surface primitives (Surfaces panel, Cones panel).
- cone full - Creates three-dimensional full cone surface primitives (Surfaces panel, Cones panel).
- cone partial - Creates three-dimensional partial cone surface primitives (Surfaces panel, Cones panel).
- sphere center and radius - Creates three-dimensional sphere surface primitives by specifying the center and radius (Surfaces panel, Spheres panel).
- sphere four nodes - Creates three-dimensional sphere surface primitives by specifying four nodes (Surfaces panel, Spheres panel).
- sphere partial - Creates three-dimensional partial sphere surface primitives (Surfaces panel, Spheres panel).
- torus center and radius - Creates three-dimensional torus surface primitives by specifying the center, normal direction, minor radius and major radius (Surfaces panel, Torus panel).
- torus three nodes - Creates three-dimensional torus surface primitives by specifying three nodes (Surfaces panel, Torus panel).
- torus partial - Creates three-dimensional partial torus surface primitives (Surfaces panel, Torus panel).
- spin - Creates surfaces by spinning lines or a node list around an axis Surfaces panel, Spin panel).

- drag along vector- Creates surfaces by dragging lines or a node list along a vector (Surfaces panel, Drag panel).
- drag along line - Creates surfaces by dragging lines or a node list along a line (Surfaces panel, Line Drag panel).
- drag along normal - Creates surfaces by dragging lines along their normal (Surfaces panel).
- ribs - Creates and modifies simple ribs between two surfaces (Ribs panel).
- ruled - Creates surfaces by interpolating linearly between lines or nodes (Surfaces panel, Ruled panel).
- spline/filler - Creates surfaces by filling in gaps, such as a hole in an existing surface (Surfaces panel, Spline panel, Quick Edit panel).
- skin - Creates surfaces by skinning lines (Surfaces panel, Skin panel).
- fillet - Creates constant radius fillet surfaces across surface edges (Surfaces panel).
- from FE - Creates surfaces that closely fit a selection of shell elements (Surfaces panel).
- meshlines - A toolkit for creating lines associated to shell elements for advanced selection or surface creation (Surfaces panel).
- auto midsurface - Creates midsurface geometry automatically from multiple surfaces or solids (Midsurface panel).
- surface pair - Creates midsurface geometry from one surface pair (Midsurface panel).
- duplicate - Creates surfaces by duplicating existing surfaces. This is available in many panels when the "duplicate" advanced entity selector is available on a surfaces collector.
- Misc. API commands that do not have an associated panel.
- Additional capabilities are available in solidThinking and solidThinking Inspire.

Edit Surfaces

- delete - Deletes surfaces (Delete panel, Quick Edit panel).
- trim - Trims surfaces using nodes, lines, surfaces and planes (Surface Edit panel, Quick Edit panel).
- untrim/unsplit - Removes various trim/split lines from surfaces (Surface Edit panel, Edge Edit panel, Quick Edit panel).
- offset - Offsets surfaces along their normal directions while maintaining topological connectivity (Surface Edit panel).
- extend - Extends the edges of surfaces to meet or intersect other surfaces (Surface Edit panel, Midsurface panel).
- shrink - Shrinks surfaces by drawing in all surfaces edges (Surface Edit panel).
- defeature - Removes pinholes, surface fillets, edge fillets and duplicate surfaces (Defeature panel, Edge Edit panel).
- midsurfaces - Modifies and edits extracted midsurfaces (Midsurface panel).
- surface edges - Toggles, suppresses, unsuppresses and equivalences surface edges (Edge Edit panel, Quick Edit panel).
- washer - Trims surfaces using free edge closed loop or shared edge offsets (Quick Edit panel).

- autocleanup - Performs basic automatic geometry cleanup functions in preparation for meshing (Autocleanup panel).
- dimensioning - Modifies dimensions of or between surfaces (Dimensioning panel).
- morphing - Morphs surfaces that have had associated nodes morphed away from them (Morph panel).
- translate - Moves surfaces along a vector direction (Translate panel).
- rotate - Rotates surfaces about a vector axis (Rotate panel).
- scale - Scales the dimensions of surfaces either proportionally or uniformly (Scale panel).
- reflect - Reflects surfaces about a plane to create a mirror image (Reflect panel).
- position - Translates and rotate surfaces into new positions (Position panel).
- permute - Switches the coordinates of surfaces (Permute panel).
- renumber - Renumbers surface edges and surfaces (Renumber panel).
- Misc. API commands that do not have an associated panel.
- Additional capabilities are available in solidThinking and solidThinking Inspire.

Query Surfaces

- shortest distance - Finds the shortest distance between entities (**Shortest Distance** dialog).
- interference check - Finds penetrations/intersections between geometries (**Geometry Interference Check** dialog).
- normal - Reviews the normal direction of surfaces (Normals panel).
- organize - Moves surfaces into different component collectors (Organize panel).
- numbers - Displays the IDs of surface edges and surfaces (Numbers panel).
- count - Counts the total or displayed surfaces (Count panel).
- area - Queries the total area of the selected surfaces (Mass Calc panel).
- dimensioning - Queries dimensions of or between surfaces (Dimensioning panel).
- Misc. API commands that do not have an associated panel.

Solids

Solids are closed volume of surfaces that can take any shape. Solids are three-dimensional entities that can be used in automatic tetra and solid meshing.

The surfaces defining a solid can belong to multiple component collectors. The display of a solid and its bounding surfaces are controlled only by the component collector to which the solid belongs.

Bounding Surface

A bounding surface defines the outer boundary of a single solid.
Bounding surfaces are shaded green by default.
A bounding surface is unique and is not shared with any other solid. A single solid volume is defined entirely by bounding surfaces.

Fin Surface

A fin surface has the same solid on all sides, that is, it acts as a fin inside of a single solid.

Fin surfaces are shaded red by default.

A fin surface can be created when manually merging solids or when creating solids with internal fin surfaces.

Full Partition Surface

A full partition surface defines a shared boundary between one or more solids.

Full partition surfaces are shaded yellow by default.

A full partition surface can be created when splitting a solid or when using Boolean operations to join multiple solids at shared or intersecting locations.

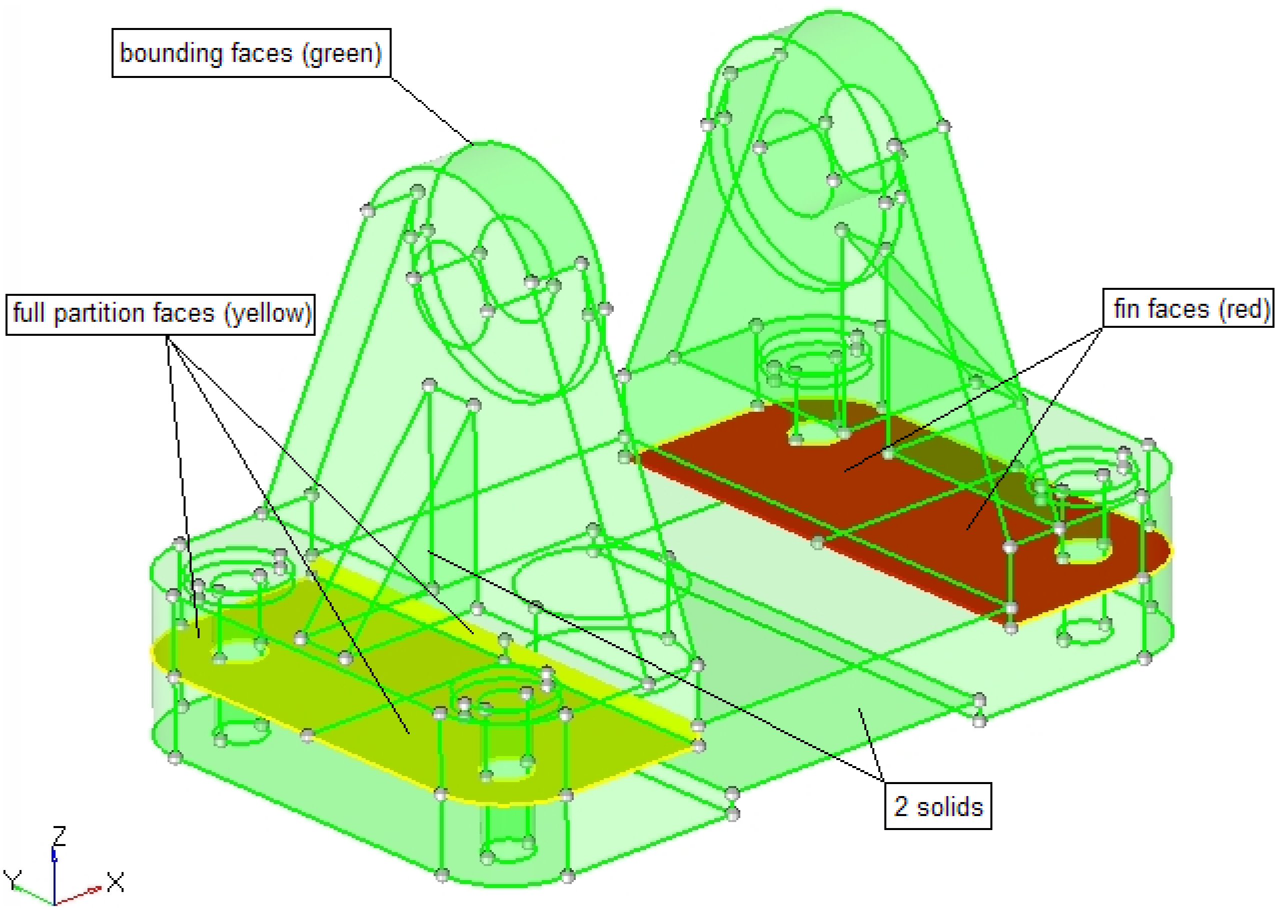


Figure 592:

Create Solids

- block - Creates three-dimensional block-shaped solid primitives (Solids panel).
- cylinder full - Creates three-dimensional full cylinder solid primitives (Solids panel).
- cylinder partial - Creates three-dimensional partial cylinder solid primitives (Solids panel).
- cone full - Creates three-dimensional full cone solid primitives (Solids panel).
- cone partial - Creates three-dimensional partial cone solid primitives (Solids panel).

- sphere center and radius - Creates three-dimensional sphere solid primitives by specifying the center and radius (Solids panel).
- sphere four nodes - Creates three-dimensional sphere solid primitives by specifying four nodes (Solids panel).
- torus center and radius - Creates three-dimensional torus solid primitives by specifying the center, normal direction, minor radius and major radius (Solids panel).
- torus three nodes - Creates three-dimensional torus solid primitives by specifying three nodes (Solids panel).
- torus partial - Creates three-dimensional partial torus solid primitives (Solids panel).
- bounding surfaces - Creates solids by converting closed surface shells which define the solid boundary (Solids panel).
- spin - Creates solids by spinning surfaces around an axis (Solids panel).
- drag along vector - Creates solids by dragging surfaces along a vector (Solids panel).
- drag along line - Creates solids by dragging surfaces along a line (Solids panel).
- drag along normal - Creates solids by dragging surfaces along their normal (Solids panel).
- ribs - Creates and modifies simple ribs between two surfaces (Ribs panel).
- ruled linear - Creates solids by interpolating linearly between surfaces (Solids panel).
- ruled smooth - Creates solids by interpolating smoothly between surfaces (Solids panel).
- duplicate - Creates solids by duplicating existing solids. This is available in many panels when the "duplicate" advanced entity selector is available on a solids collector.
- Misc. API commands that do not have an associated panel.
- Additional capabilities are available in solidThinking and solidThinking Inspire.

Edit Solids

- delete - Deletes solids (Delete panel).
- trim - Trims solids using nodes, lines, surfaces and planes (Solid Edit panel).
- merge - Combines two or more solids into a single solid (Solid Edit panel).
- detach - Detaches solids that have shared fin faces from each other (Solid Edit panel).
- boolean - Performs complex merge and split functions on solids (Solid Edit panel).
- dimensioning - Modifies dimensions of or between surfaces (Dimensioning panel).
- translate - Moves solids along a vector direction (Translate panel).
- rotate - Rotates solids about a vector axis (Rotate panel).
- scale - Scales the dimensions of solids either proportionally or uniformly (Scale panel).
- reflect - Reflects solids about a plane to create a mirror image (Reflect panel).
- position - Translates and rotate solids into new positions (Position panel).
- permute - Switches the coordinates of solids (Permute panel).
- renumber - Renumbers solids (Renumber panel).
- Misc. API commands that do not have an associated panel.

- Additional capabilities are available in solidThinking and solidThinking Inspire. See tutorial HM-2080 for an example.

Query Solids

- shortest distance - Finds the shortest distance between entities (**Shortest Distance** dialog).
- interference check - Finds penetrations/intersections between geometries (Geometry Interference Check dialog).
- normal - Reviews the normal direction of solid surfaces (Normals panel).
- organize - Moves solids into different component collectors (Organize panel).
- numbers - Displays the IDs of solids (Numbers panel).
- count - Counts the total or displayed solids (Count panel).
- area - Queries the total area of the selected solids' surfaces (Mass Calc panel).
- volume - Queries the total volume of the selected solids (Mass Calc panel).
- dimensioning - Queries dimensions of or between surfaces (Dimensioning panel).
- Misc. API commands that do not have an associated panel.

Faces

A face is a single Non-uniform Rational B-Spline (NURBS) and is the smallest area entity. It has a separate underlying mathematical definition, specified when it was created.

All faces are represented mathematically with the following formulations:

- plane
- cylinder/cone
- sphere
- torus
- NURBS

A surface can be made up of a single face type or of multiple face types. Multiple types are used for more complex surfaces that contain sharp corners or highly complex shapes.

Dimensioning

Change the dimensions of existing geometry, thus changing the basic shape of solids and other enclosed volumes.

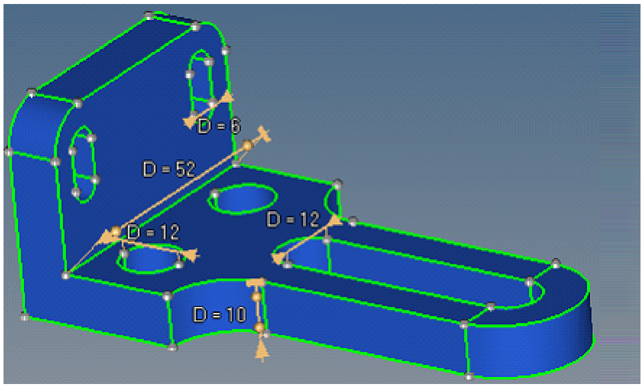


Figure 593: Initial Dimensions

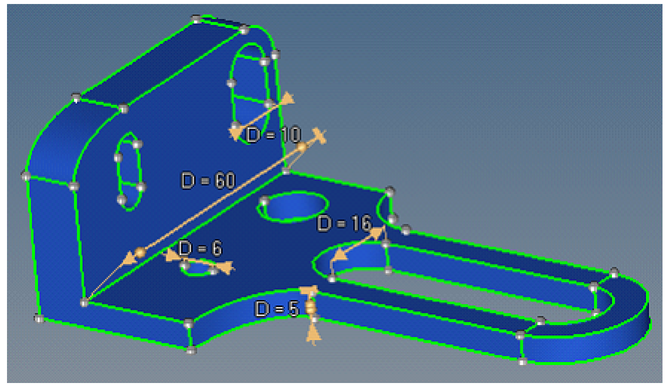


Figure 594: Modified Dimensions

Edit Dimensions

Dimensioning is accomplished with features, parameters and dimension manipulators.

When changing several dimensions, each dimension change is performed separately using the respective manipulator. However, if multiple dimensions are linked to the same parameter or parameter expression, they will be updated simultaneously.

When dimensions are modified, a very limited check for mutual penetrations of the repositioned surfaces is performed. It is the your responsibility to ensure that the new dimensions are appropriate.

The locked end of the dimension manipulator defines the direction in which the affected surfaces move when the dimension is modified. For a dimension to be modified, one or both ends of the dimension manipulator must unlocked.

When dimensions cannot be modified, the locked side is set to Both and you may use the Sides Selection advanced option to specify how the dimension should be changed, when possible.

1. Create dimension feature.

a) In the Model Browser, right-click and select **Create** > **Feature** from the context menu.
The **Feature** dialog opens.

b) In the Name field, enter a name for the dimension.

c) In the Point1 and Point2 fields, use the entity selector to select two fixed points (vertices) between the opposite surfaces where the dimension is defined.
A dimension manipulator is then created between these fixed points (Point1 and Point2).

d) In the Parameterization field, select a parameterization method.

> **Note:**
> A new parameter can be created and assigned to an existing dimension feature at any time via the Create Parameter option in the Entity Editor context menu.

- Choose **Create Parameter** to create and assign a new parameter to a dimension feature.
- Choose **Select Parameter** to assign an existing parameter to the new dimension feature.
- Choose **No Parameter** if you do not want to assign a parameter to the dimension feature.

e) Click **Create**.

2. In the Entity Editor, define dimension feature attributes.

3. Edit dimension.

- In the modeling window, click the dimension's corresponding label and enter a new value.
- In the Model Browser, select the parameter assigned to the dimension feature. In the Entity Editor, enter a new value.

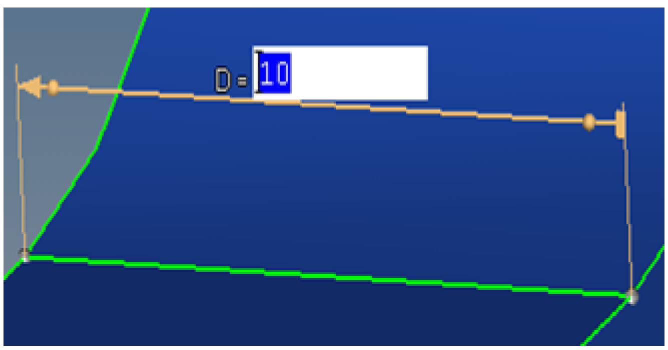


Figure 595:

Dimensioning Concepts

Learn about basic dimensioning concepts, such as continuous surface offset functionality and tolerance and accuracy.

Continuous Surface Offset Functionality

Dimensioning is based on a continuous surface offset functionality. It provides assistance in the selection of the surfaces to offset so that a change to the selected dimension can occur, and calculates the offset values required for each surface to achieve the specified dimension.

The continuous offset modifies both the surfaces you selected for the offset and the adjacent involved surfaces that must also be modified so that the result will remain as continuous as the initial input.

These "selected" and "involved" surfaces are modified with different rules.

- Selected surfaces
    - Offset by a constant value that is normal, or in some cases almost normal, to the surface at each point. For example, a standalone surface is offset by the given constant distance exactly normal to itself.

Figure 596: Normal Offset of a Standalone Surface

    - When the adjacent surfaces form a corner between them, the exact normal offset will result in either disconnected surfaces or in intersecting surfaces, for example if the offset was performed in the opposite direction.

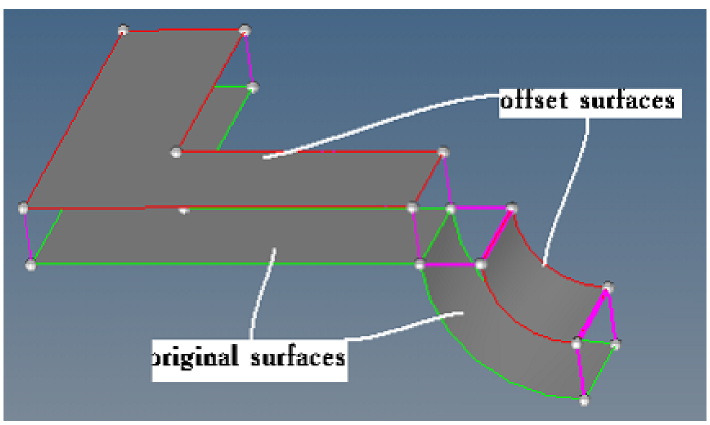


Figure 597: Exact Normal Offset of the Adjacent Surfaces Creates a Rupture

    - A continuous result that is consistent with the given offset distance is obtained by reconciling the offset vectors of the vertices shared by the surfaces being offset.

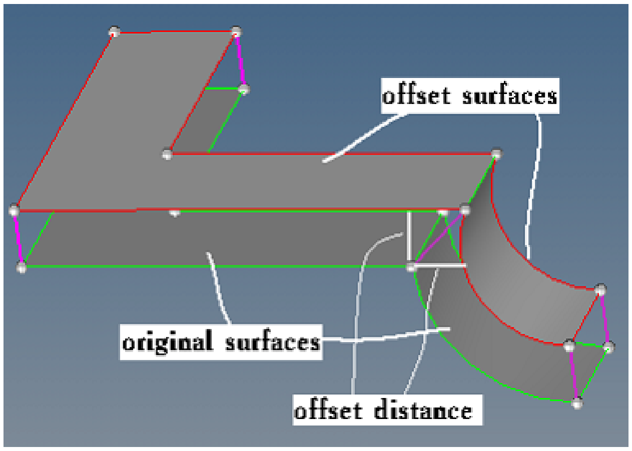


Figure 598: Reconciled Offset Vectors at Shared Edge

- Involved Surfaces
    - The edges of the involved surfaces that are shared with the selected surfaces move with the selected surfaces.
    - The edges of the involved surfaces that do not have a common point with the selected surfaces do not move, for example they are locked.
    - The offset of the edges that connect both the moving and the locked involved surface edges is defined by interpolation. Different interpolation methods are available.

In general cases, the target dimension between the selected vertices is achieved by offsetting the surfaces that contain the vertices in an infinite number of ways. To avoid this, the following rules are implemented.

- If both dimension ends (both vertices) are allowed to move, an attempt is made to move them by the same distance whenever possible.
- If possible, the dimension ends are moved in such a way that the direction of the dimension will not change.

In the following example, the initial positions of the vertices are marked with temp nodes to enable the changes can be easily seen. The locked state of the dimension manipulators is indicated by the lock icons.

> **Note:** These examples are not cumulative, so no two images are directly related. The first image, showing the dimensions of 3, 4, and 5, is the starting point from which all of the other examples derive.

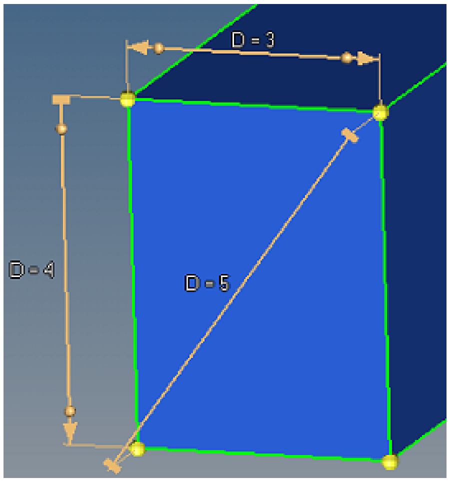


Figure 599: Original Model

3 dimensions selected.

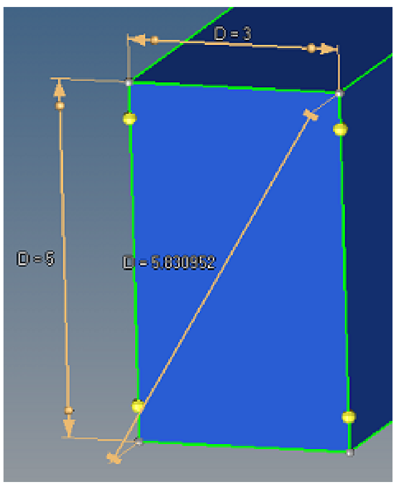


Figure 600: Dim 4 Changed to 5

Top and bottom move.

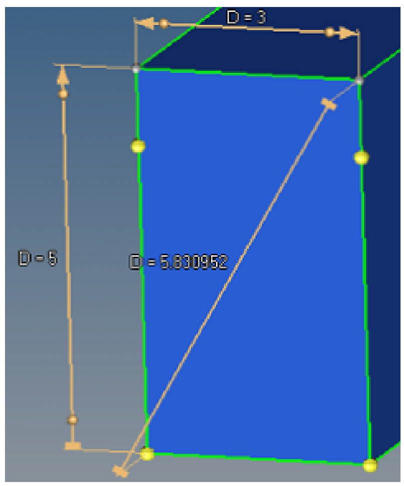


Figure 601: Dim 4 Changed to 5

Top moves, bottom is locked.

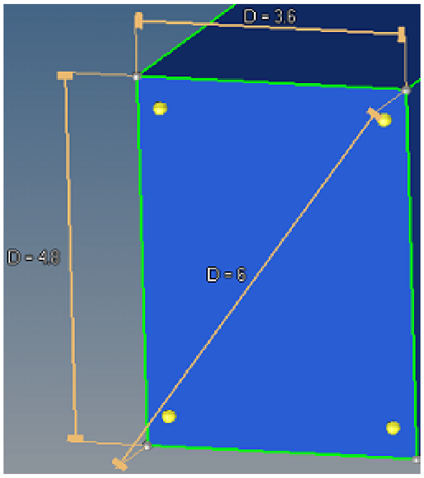


Figure 602: Dim 5 (Diagonal) Changed to 6

All sides move.

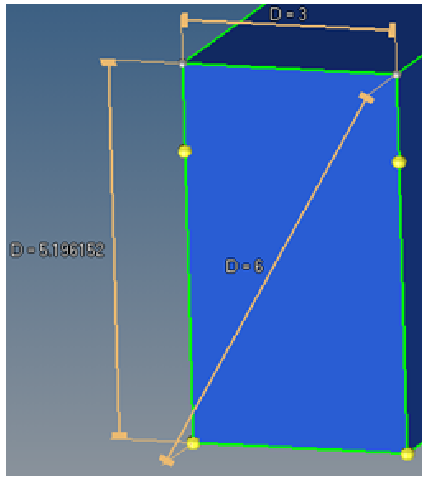


Figure 603: Dim 5 Changed to 6

Only top moves.

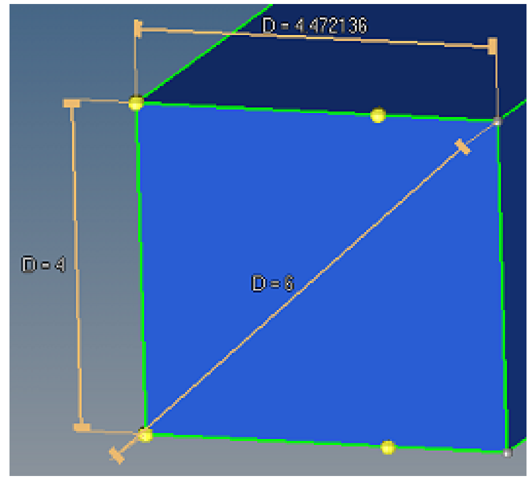


Figure 604: Dim 5 Changed to 6

Only right side moves.

Tolerances and Accuracy

All geometry transformation tools are numerical tools that operate with some accuracy defined by the tolerances, such as the geometry cleanup tolerance set in the Options panel. Curved surfaces and lines have internal structures in 3D that are invisible to you. Significantly reducing the size of such an entity so that these structures fall below the tolerances may result in a structure simplification that you cannot notice at first; the structural data will be lost. When this occurs any subsequent increase in the size will not restore the initial structures. For example, reducing a cylinder diameter 100 times and then increasing the diameter 100 times may not lead to the same cylinder; in some cases, a complex internal representation of the cylinder may lead to a corrupt surface. In general, transformation of a curved entity may result in both the simplification or complication of its internal structure. It is therefore not recommended to perform multiple transformations on curved entities.

Dimension Manipulators

Dimension manipulators are used to alter selected dimensions of solid entities.

A dimension manipulator consists of:

Dimension line

A segment parallel to the line that connects the selected points, but is shifted off the selected points for visibility. The terms manipulator direction and manipulator ends are also used, which are the same as the dimension line direction and the dimension line ends.

Pullout lines

Two parallel segments that connect the ends of the dimension line with the selected points.

Lock icons

Arrow (movable) and block (locked) icons indicate the lock state of a manipulator end.

Lock controls

Sphere handles, located near the lock icons, enable the lock state of a manipulator end to be modified.

Display/input field

Displays the current dimension value, which can be modified or deleted. This value can be modified or deleted. Deleting the value deletes the the manipulator. For dimensions that are parameterized, an "&" symbol will appear before the dimension. Editing a parameterized dimension directly edits the parameter, or parameter expression.

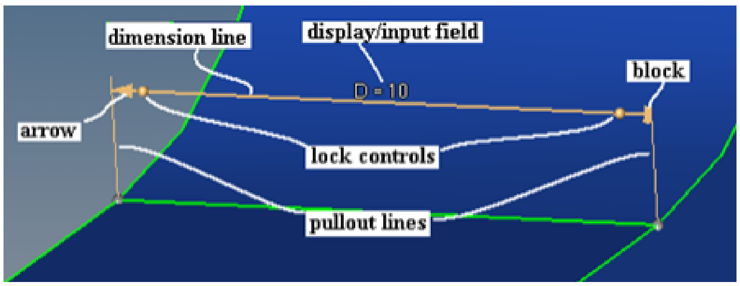
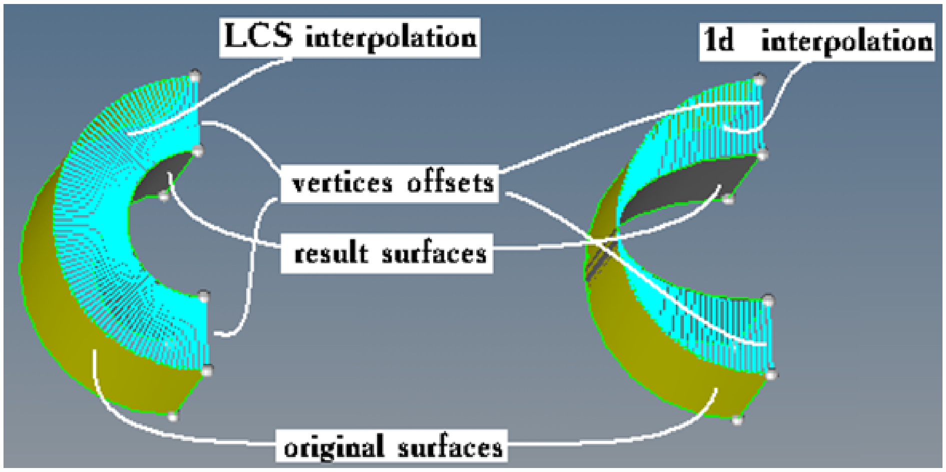


Figure 605: Dimension Manipulator

Dimension Feature Attributes

Attributes associated with dimension features can be modifed in the Entity Editor.

Attribute	Action
Lock Side	Select the locked end of the dimension manipulator, which defines the direction in which the affected surfaces move when the dimension is modified. For a dimension to be modified, one or both ends of the dimension manipulator must unlocked. When dimensions cannot be modified, the locked side is set to Both and you may use the Sides Selection advanced option to specify how the dimension should be changed, when possible.
Surfaces Interpolation System	**Automatic** A heuristic algorithm is used to try and decide which of the two interpolation methods to apply for each individual, applicable involved surface. **Local** A Local Coordinate System (LSC) 2D interpolation method that "slides" along the surfaces to determine the offset vectors, which are

Attribute	Action
	then interpolated and combined into the interpolated offset at each point. Selected surfaces are always interpolated using this method. **Global** A global coordinate system 1D linear interpolation method that stretches/compresses a surface proportionally in a global 1D. Only applicable when all of the offset vectors at the surface's vertices are collinear and proportional to the distance parameter along their common direction.  *Figure 606: Interpolation of the Same Offset Vectors at the Vertices for both Methods*
Minimum Slide Angle	When a selected surface is offset, the involved surfaces must be modified to keep the continuity of the model. Surfaces can be modified by dragging the involved surface behind the selected surface, or by defining it as a "slider" along which the selected surface slides. The Minimum Slide angle determines which method is used. If the slide angle is more than the specified value, then the involved surface will slide; otherwise it will drag. When the involved surface is a slider, the orientation of the surface does not change for planar surfaces. However, for curved involved surfaces, the sliding directions are defined by the tangents to the surface where it is adjacent to the selected surface. Sliding of the selected and involved surfaces along these directions may also result in some change to the shape of the involved surface.

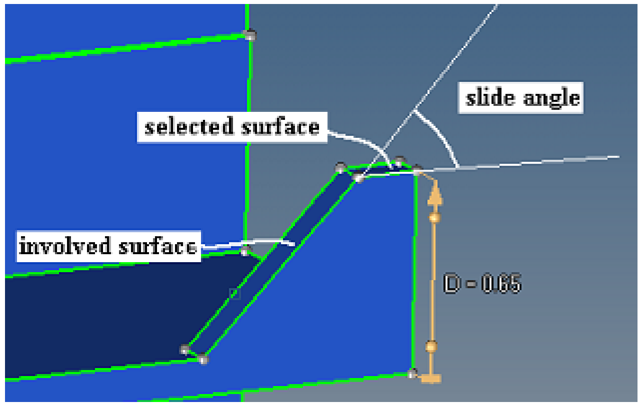
<table>
<tr><th>Attribute</th><th>Action</th></tr>
<tr><td></td><td>


Figure 607: Original Model

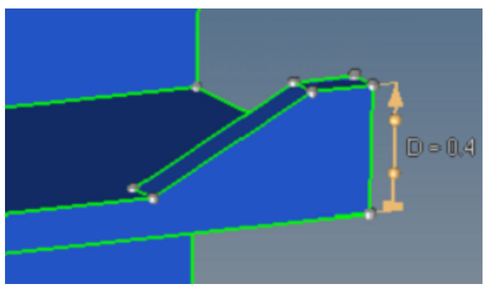


Figure 608: Involved Surface Dragged *Dimension modified to D=0.4.*

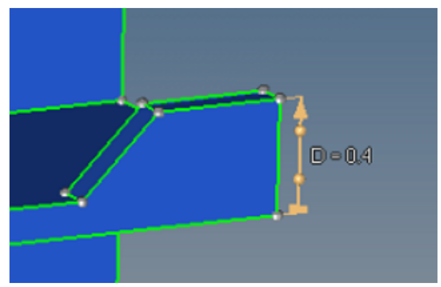


Figure 609: Involved Surface as Slider *Dimension modified to D=0.4.*
</td></tr>
<tr><td>Remove Collapsed Surfaces</td><td>
Remove portions of the offset surfaces that fold into themselves or adjacent surfaces (portions of surfaces that penetrate themselves or adjacent surfaces along the edges they are adjacent over).

For example, suppose that the slide angle is greater than the Minimum Slide Angle and the value in the dimension manipulator is set to 1. If this option is off, the involved surface will slide and ignore the self-penetration, resulting in a corrupt model. If this option is on, the involved surface will slide as far as possible without causing self-penetration. This may not allow the specified dimension to be reached, but will not result in a corrupt model.
</td></tr>
</table>

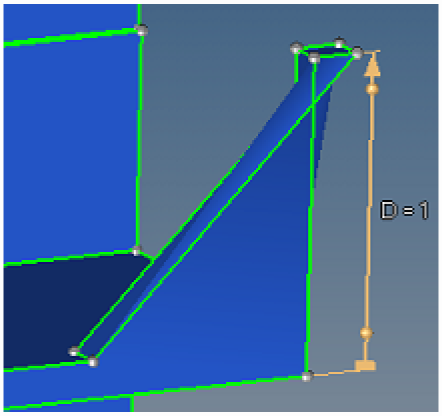
<table>
<tr><th>Attribute</th><th>Action</th></tr>
<tr><td></td><td>


Figure 610: Remove Collapse Surfaces Off

Dimension modified to D=1.

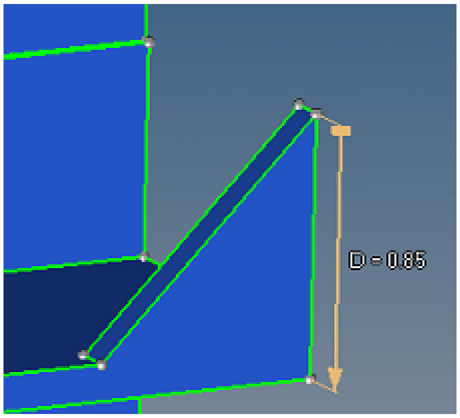


Figure 611: Remove Collapse Surfaces On

Dimension modified to D=1.

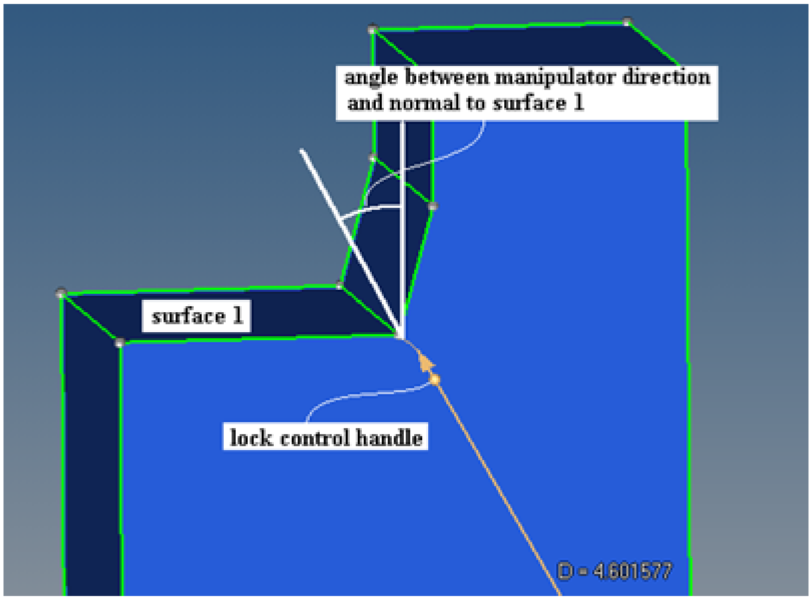
Another useful application is for the removal of holes. If the hole diameter is set to 0 and this option is on, the hole will be removed. If the option is off, a small "straw surface" will still remain.

In general, unless it is known that collapsed surfaces will result, it is better to keep this option off for performance reasons, as this option has no effect on general cases that do not result in penetration.
</td></tr>
<tr><td>Sides Selection</td><td>
Auto

Automatically select the surfaces to offset using the following rules:

1. Surfaces adjacent to the manipulator ends are selected if the angle between the normal to the surface at the dimension end and the dimension direction is less than the Max Pick Tilt.

If surfaces are selected at both ends for the specified Max Pick Tilt value, then the lock control handles will allow for the manual manipulation of the offset scenario.
</td></tr>
</table>

Attribute	Action
	 *Figure 612: Angle between the Normal to the Surface and the Manipulator Direction* **2.** Surfaces adjacent to the selected surfaces are appended, provided that they are planar and the angle along the edge over which they are adjacent to the already selected surface is less than the Max Expand Angle. **3.** The total area of the selected surfaces at each end is calculated. If the area of the selected surfaces at one end is more than the Side Selection Area Ratio and larger than the area of the selected surfaces at the other end, then the surfaces on the larger area side are unselected. In this case, only the surfaces at the smaller area side are used to offset.

<table>
<tr><th>Attribute</th><th>Action</th></tr>
<tr><td></td><td>

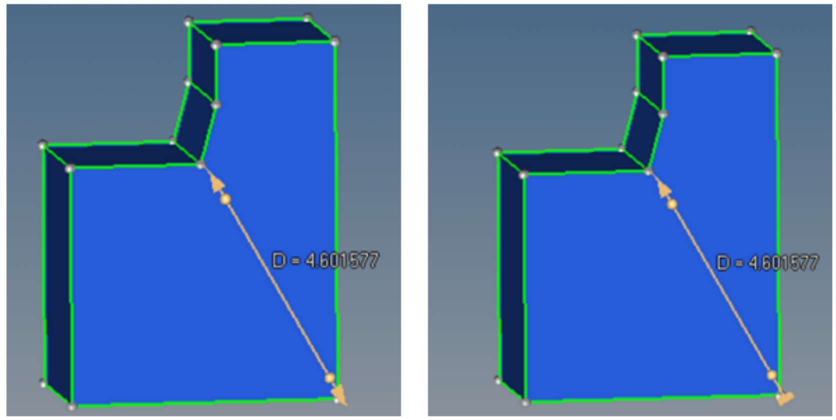


Figure 613: Side Selection Area Ratio

The image on the left has a side selection area ratio = 3, and the image on right has a side selection area ratio = 1.5. The bottom surface area is twice as much as the top surface, therefore the bottom will not move (note the lock indicator).

The ends of the dimension lines that are allowed to move are marked with arrows, while the locked ends are marked with blocks.

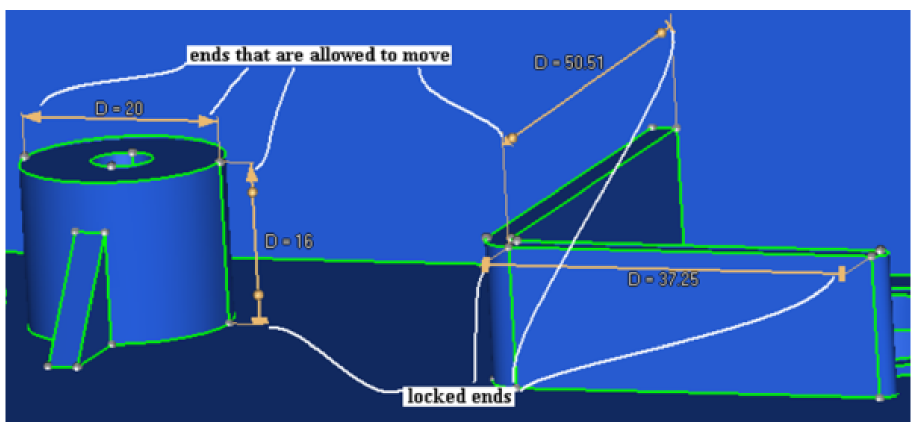
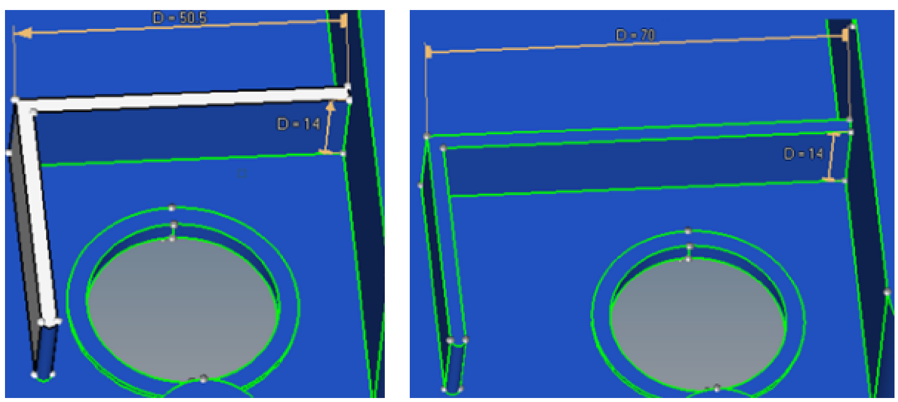


Figure 614: Example of Lock Icons

When both sides have surfaces that satisfy rule 1 above, rule 3 can be manually overridden. In this case the lock controls (spheres near the icons) define the offset scenario. Clicking the lock control handles will toggle the lock state between locked and unlocked for that end. If a lock control state is manually specified, then rule 3 is ignored for

</td></tr>
</table>

Attribute	Action
	that dimension manipulator and the Side Selection Area Ratio option no longer applies. When Sides Selection is set to Manual, the surfaces to offset are selected using the Surfaces to Move selector. The manual surface selection is then governed by the lock state of the dimension manipulator ends. **Note:** Out of the selected surfaces, only those that are linked to at least one of the dimension manipulator ends by a continuous selection are actually used in the offset. With the manual selection, the use of Separator Lines is also available (see the surface edit subpanel for details). **Manual** Manually select the surfaces to offset. With manual side selection, surfaces are selected erroneously, and the results can be unexpected or catastrophic.  *Figure 615: Dim 50.5 Changed to 70 to Move the Wall*

Attribute	Action
	The three highlighted surfaces are selected in order to change the dimension from 50.5 to 70 and move the wall to a new position.

Advanced Considerations

Advanced considerations to keep in mind when changing the dimensions of existing geometry.

In practice, changing of a linear dimension in a model normally implies either stretching/compressing in the direction of the modified dimension or changing of a diameter/radius. With dimensioning funtionality, a combination of both modification types is provided.

In the example below, one of the two D=52 dimensions is changed to D=60. How the offset is performed will give different results, both of which may be valid, depending on which of the two dimension manipulators is changed.

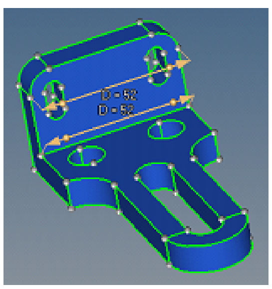


Figure 616: Original model

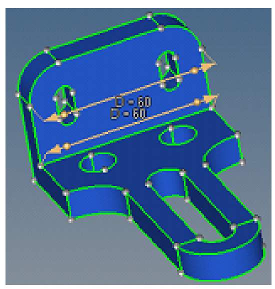


Figure 617: Edge Fillet Surfaces Selected

The fillet radius is scaled.

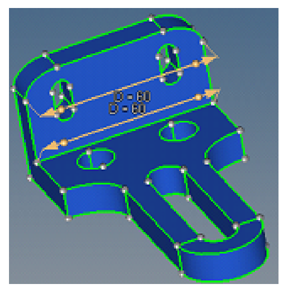


Figure 618: Edge Fillet Surfaces Involved

Linear scaling only, fillet curvature changes.

When the value of the upper dimension manipulator is modified from 52 to 60, the edge fillet surfaces are adjacent to the modified manipulator and are offset as selected surfaces. As such, they are offset with the LSC interpolation, which results in a preservation of their shape along with the change in radii.

When the value of the lower dimension manipulator is modified from 52 to 60, the edge fillet surfaces are not adjacent to the modified dimension manipulator and are curved, so they are offset as involved surfaces. Using automatic interpolation, it is recognized that these two curved surfaces can be simply stretched to provide the model continuity via the global interpolation method.

When using manual surface selection and changing the same lower dimension, a variety of results are obtainable depending on the selected surfaces. Some of the possible results are shown below.

> **Note:** In each row of the three images, the first two show the initial selection from two angles, to reveal all of the selected surfaces, while the third shows the results of the dimension change based on those selected surfaces.

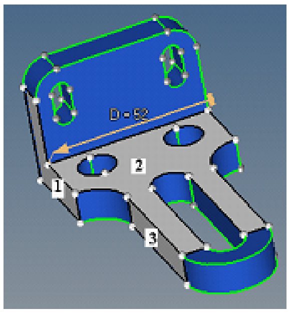


Figure 619: Original model, 3 surfaces selected

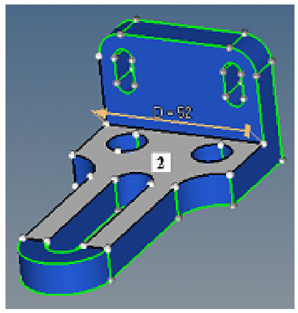


Figure 620: Original model, 3 surfaces selected

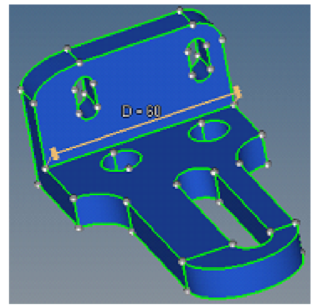


Figure 621: Result of the dimension 52 change to 60

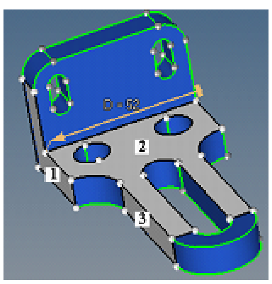


Figure 622: Original model, 4 surfaces selected

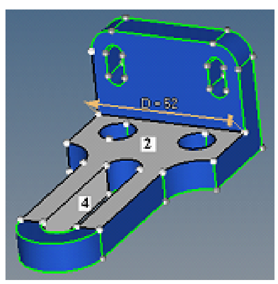


Figure 623: Original model, 4 surfaces selected

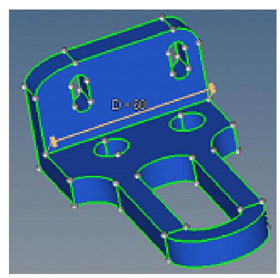


Figure 624: Result of the dimension 52 change to 60

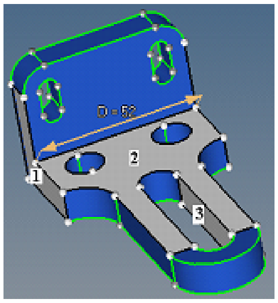


Figure 625: Original model, 5 surfaces selected

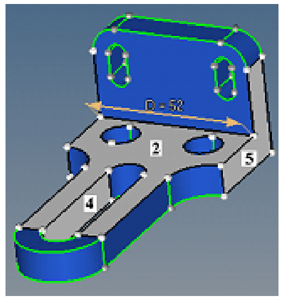


Figure 626: Original model, 5 surfaces selected

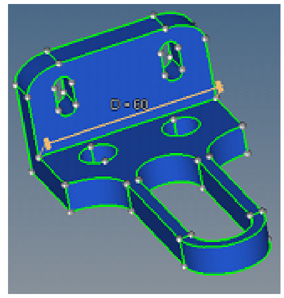


Figure 627: Result of the dimension 52 change to 60

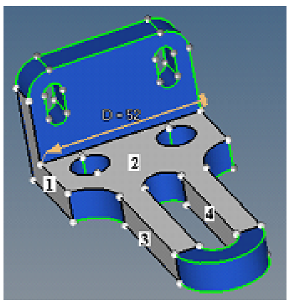


Figure 628: Original model, 6 surfaces selected

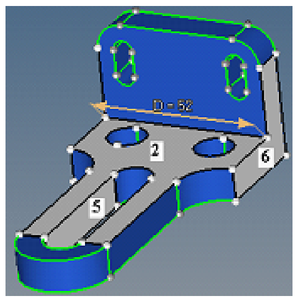


Figure 629: Original model, 6 surfaces selected

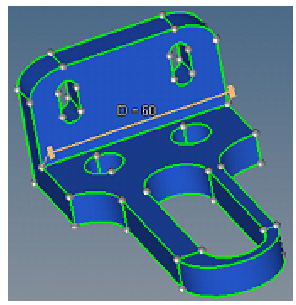


Figure 630: Result of the dimension 52 change to 60

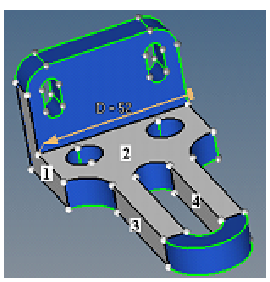


Figure 631: Original model, 7 surfaces selected

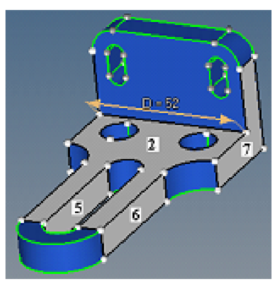


Figure 632: Original model, 7 surfaces selected

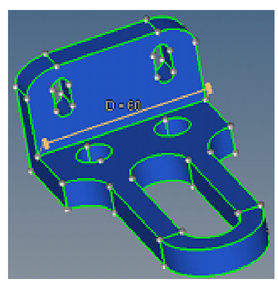


Figure 633: Result of the dimension 52 change to 60

The following steps are used to calculate the offset values of the selected surfaces.

1. The required shift in the dimension manipulator direction is calculated as a difference between the requested distance and the actual distance between the dimension manipulator ends.
2. If both dimension manipulators ends are allowed to move, the required shift is divided by two.

   When Sides Selection is set to **Auto**, an end is allowed to move if it belongs to a surface that is automatically selected to move. When this can be overridden manually by you, the lock controls appear.

   When Sides Selection is set to **Manual**, an end is allowed to move if it belongs to a manually selected surface, and the surface normal at the dimension manipulator end forms an angle with the dimension manipulator direction that is less than arccos(0.05) (87.134016 degrees).

   For example, the right end of the dimension manipulator belongs to only the selected surface 2. The normal to surface 2 at the right end creates a 90-degree angle with the dimension manipulator and thus the end is not allowed to move. The left dimension manipulator end belongs to both selected surfaces 1 and 2. The normal to surface 1 at the left end makes a 0-degree angle with the dimension manipulator direction, and thus the left end is allowed to move.

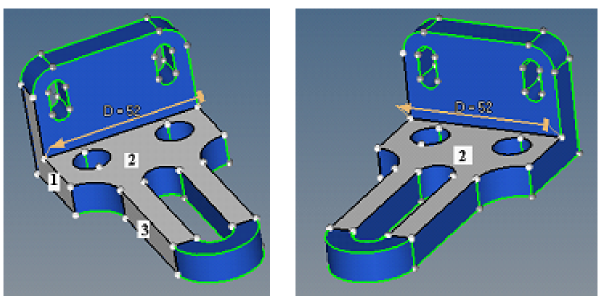
   

   Figure 634:

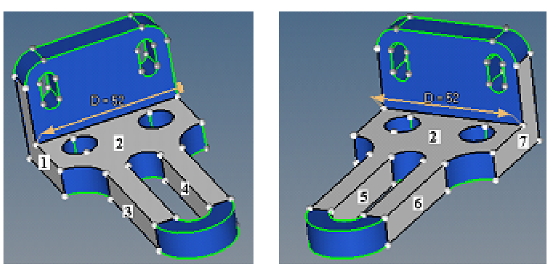
   

   Figure 635:
   In this example, the right end belongs to selected surfaces 2 and 7, with the left end belonging to selected surfaces 1 and 2. Thus, both ends are allowed to move.

3. When only planar surfaces are selected, the absolute value of its normal offset is defined as the absolute value of the required shift multiplied by the cosine between the normal to the surface and the dimension manipulator direction.

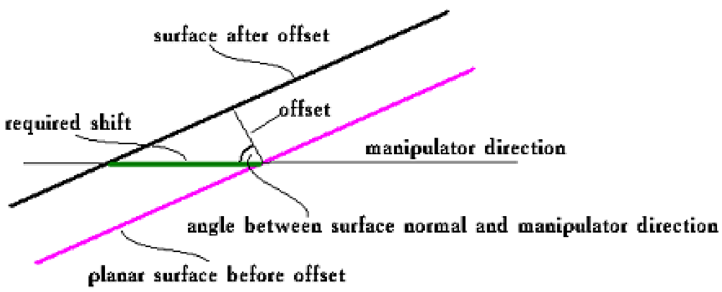


Figure 636:

For a planar surface, this provides that its shift in the dimension manipulator direction is equal to the required shift.

When curved surfaces are included and the Sides Selection is set to Manual, the rules of the offset value calculations are more complex. The problem in this case originates from the fact that a selected curved surface can provide a smooth link between the selected planar surfaces that are tilted by different angles versus the dimension manipulator direction. When smooth, adjacent surfaces are offset, they must be offset by the same value to ensure continuity of the result, because in this case it is not possible to reconcile the different offset values as discussed earlier. This means that the planar surfaces with a different tilt towards the dimension manipulator direction cannot be offset by different distances, as shown above, when the planar surfaces are smoothly linked by a selected surface.

The current algorithm to define the offset value in the general case, for both curved and planar surfaces, is as follows. For a selected surface adjacent to the dimension manipulator end, its offset is calculated as shown in the image above, based on the normal to the surface at the dimension manipulator end. For a selected surface that is not adjacent to the dimension manipulator end, a chain of selected surfaces that links it to the related end is detected, and the offset is calculated along the chain, from the previous surface to the next. The calculation along the chain is based on the following:

- If the surfaces are smoothly adjacent, the offset value is directly passed from one surface to the next.
- If the surfaces are not smoothly adjacent, the offset is calculated in such a way that for a planar surface the result as shown in the image above is obtained.

The problem here is that when several chains of selected surfaces connect a selected surface with the related dimension manipulator end, the offset results for the surface obtained along the different chains can contradict each other. Then the dimensioning result may be corrupt. Therefore, it is important to make appropriate manual surface selections.

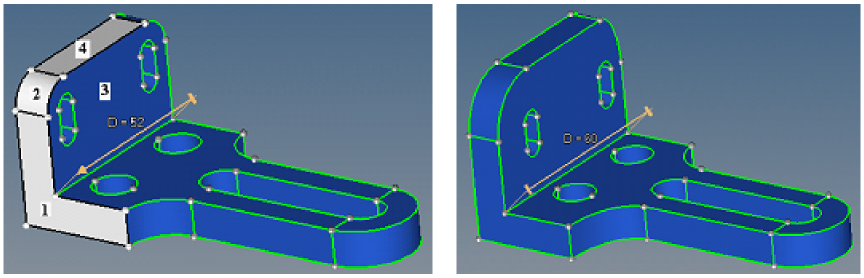


Figure 637:

Surfaces 1, 2, and 4 selected, D=52 changed to D=60. Surfaces 1, 2, and 4 offset by 8.

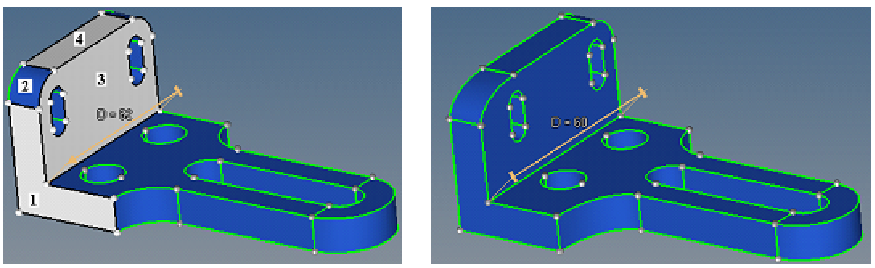


Figure 638:

Surfaces 1, 3, and 4 selected, D=52 changed to D=60. Surface 1 offset by 8, Surfaces 3 and 4 by 0.

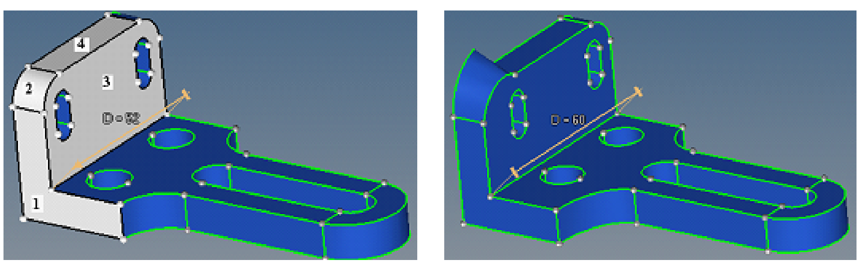


Figure 639:

Surfaces 1, 2, 3, and 4 selected, D=52 changed to D=60. The result is corrupt.

4. For each selected surface the sign of the offset is defined so that it will move in the same direction as the dimension manipulator end to which it is related.

   A surface can be related to one, and only one, of the dimension manipulator ends. For this, first, the dimension manipulator end must be allowed to move. Second, the surface should be linked to the dimension manipulator end over a chain of adjacent selected surfaces. Third, in the case when the surface is linked to both dimension manipulator ends which are allowed to move, the surface will be related to the end that is closer to it.

   As an example, selected surface 2 will have an offset of 0, because cos(90) = 0. The purpose for selection of this surface is just to provide a link from the dimension manipulator ends to the other selected surfaces. Surface 1 is at the moving dimension manipulator end, and surface 3 moves as surface 1.

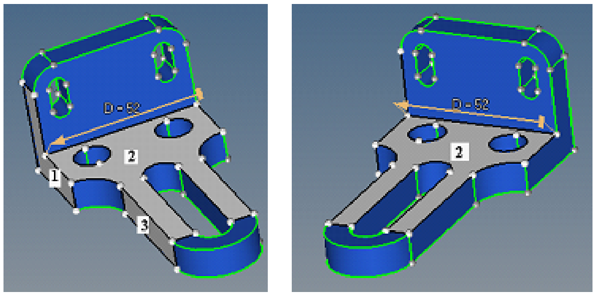


Figure 640:

Following the same rules, surfaces 1 and 7 are at the moving dimension manipulator ends. Surfaces 3 and 5 move as surface 1, and surfaces 4 and 6 move as surface 7.

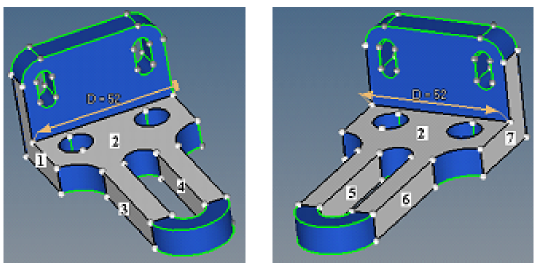


Figure 641:

Extract and Edit Midsurfaces

Extract the midsurface of thin solids, and review and edit midsurface plates.

Extract Midsurfaces

Midsurfaces can be extracted for sheet metal stampings, molded plastic parts with ribs, and other parts that have thickness clearly smaller than width and length.

During midsurface extraction, the original geometry that you select to extract the midsurface from remains unchanged, and the new geometry that represents the midsurface is created. The (variable) thickness of each middle surface is calculated and stored with the surface definition.

1. From the Geom page, select the **midsurface** panel.
2. Select **auto midsurface**.
3. Select the **auto extraction** subpanel.
4. Using the selector, select surfaces or solids to extract the midsurface from.
5. Define all of the midsurface extraction options accordingly.
6. Click **extract**.
   A midsurface representation of a solid part or a finite element shell representation of solid geometry is generated and organized in a new component collector called Middle Surface, or in the current component, depending on your choice.
7. Review and modify the midsurface as needed.

Edit Plates, Base Surfaces, and Collapsed Lines

Plates, base surfaces, and collapsed lines on the midsurface can be modified using the interim edit tools in the midsurface panel.

The modifications you make to the midsurface using the interim edit tools are not automatically applied. You must delete and re-extract the midsurface in the auto extraction subpanel to apply your changes.

Edit Plates

Plates are a group of surfaces in the model in which the middle surface will be inserted.

Each plate has two sides: blue and green. The middle surface is inserted between the two sides of each plate. When you click **show/edit all**, surfaces are organized into components that reflect their plate type. To display the plates as per their component color, change the geometry display mode to Mixed on the Visualization toolbar.

When the automatic detection of plates is not correct, click **show/edit all** to manually edit plates.

Plate information will only be created upon extracting midsurface using the offset+planes+sweeps and offset+planes methods.

Any changes made to the midsurface using the Plate Edit tools will not be visible until you click **update** within the same panel, or after you extract the midsurface using the offset+planes+sweeps or offset +planes methods.

1. From the Geom page, select the **midsurface** panel.
2. Select the **interim edit tools**.
3. Select the **edit plates** subpanel.
4. Click **show/edit all**.
5. Edit plates.

To edit plates by	Do this
Creating new plates out of surfaces	1. Using the single surface selector, select surfaces to create new plates from. 2. On the left side of the panel, select a type of plate to create. 3. Click **new plate**. A new plate is created out of the selected surfaces. 4. Continue modifying the midsurface, or click **update** to re-extract the midsurface.
Creating transition surfaces	Transition surface provide more information to the algorithm regarding inserting surface where two regular plates are intersecting. An algorithm calculates how far intersecting plates can be extended based on the transition surface. 1. Using the single surface selector, select surfaces. 2. Click **transition surface**. The selected surfaces are used as transition surfaces during midsurface extraction. 3. Continue modifying the midsurface, or click **update** to re-extract the midsurface.
Selecting trim surfaces to ignore during midsurface extraction	1. Using the single surface selector, select surfaces that you do not want to be considered trim surfaces. 2. Click **not a trim surface**. The selected surfaces are placed in the component, ^Not a trim surface. 3. Continue modifying the midsurface, or click **update** to re-extract the midsurface. During midsurface extraction, the surfaces in the ^Not a trim surface component will be ignored.

To edit plates by	Do this
Switching the offset side	Each plate has two sides (blue and green) between which the midsurface is inserted. By default, the midsurface is generated by offsetting the green side of the plate. **1.** Using the single surface selector, select surfaces to switch the offset side. **2.** Click **switch sides**. **3.** Continue modifying the midsurface, or click **update** to re-extract the midsurface. During midsurface extraction, the midsurface will be offset from the opposite side that it was originally offset from. If the midsurface was generated by offsetting the green side of the plate, the midsurface will now be generated by offsetting the new green side which was blue before using switch side.
Assigning trimming surfaces	Plate edges act as trimming surfaces for all plate types, and are used to trim inserted midsurfaces. **1.** Using the single surface selector, select surfaces to serve as trimming surfaces. **2.** Click **plate edge**. **3.** Continue modifying the midsurface, or click **update** to re-extract the midsurface. During midsurface extraction, the midsurface is trimmed using the surfaces you selected.
Merging plates	**1.** Using the full plate selector, select plates to merge. **2.** On the left side of the panel, select a new plate type to create. **3.** Click **merge plates**. All of the selected plates are merged into a single plate.

To edit plates by	Do this
	4. Continue modifying the midsurface, or click **update** to re-extract the midsurface.

Edit Base Surfaces

In certain situations, you may need to extract a midsurface from multiple solids as if they were a single solid.

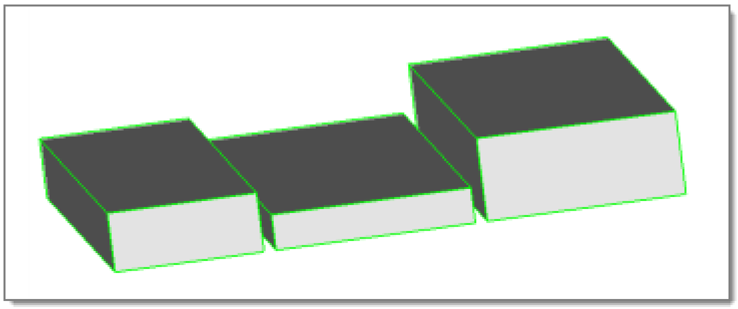


Figure 642:

Normally, this would result in multiple midsurfaces, each one at the middle of the solid it was extracted from, and therefore not aligned with each other.

In the edit base surfaces subpanel you can specify a distance from the base surfaces of multiple solids, that you wish to treat as if they were continuous, to generate separate-but-aligned midsurfaces for during extraction.

You can create aligned midsurfaces for non-aligned solids, by selecting each solid and specifying its offset separately, using a different offset for each. You must know the exact dimensions of each solid to do this accurately.

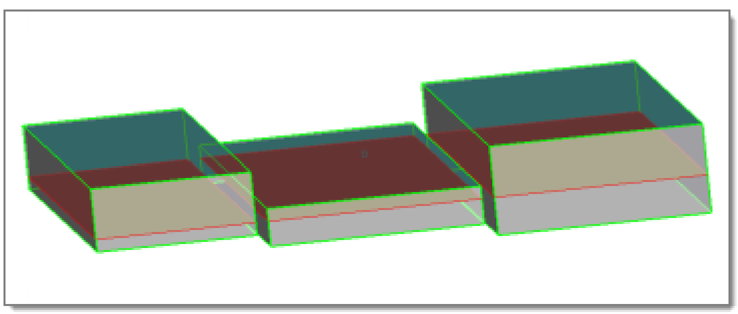


Figure 643:

If use base surfaces is disabled, new midsurfaces will not be generated at the specified distance from the selected base surfaces. The setup of the base surfaces that you modified will be retained for future use as long as you do not delete them.

1. From the Geom page, select the **midsurface** panel.
2. Select the **interim edit tools**.
3. Select the **edit base surfaces** subpanel.
4. Select the **use base surfaces** checkbox.
5. Using the surfs selector, select base surfaces.

   > **Note:** If both sides of a plate are selected as the base surface, the side which is used will be random.

6. In the distance to base field, specify a distance to create new midsurfaces from the selected base surfaces.

   > **Note:** If align steps (extraction options subpanel) and use base surfaces are both used for the same solids, then the base surface distance will overwrite the align steps distance.

7. Click **add base**.
8. Select the **auto extraction** subpanel.
9. Click **extract**.

The original midsurface is deleted and a new midsurface is generated. Separate-but-aligned midsurfaces are generated at the specified distance from the bases of the surfaces that you modified.

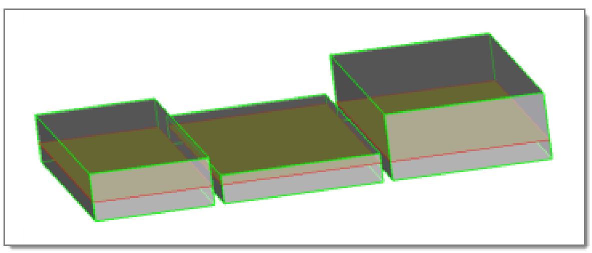


Figure 644:

Edit Collapsed Lines

Enable visualization settings to display point associations on the midsurface after extraction, and manually define lines and line chains to establish the linkage between points that should collapse to the same location.

Automatically extracted midsurfaces may include gaps, which are caused by a failure to correctly interpret relatively complex topology. In many cases, this can occur when a solid part has a fillet on one side but not on the other, or a fillet has a step or "ledge" at one end of its curve. In the image below, a gap in the midsurface is caused by a step in the fillet.

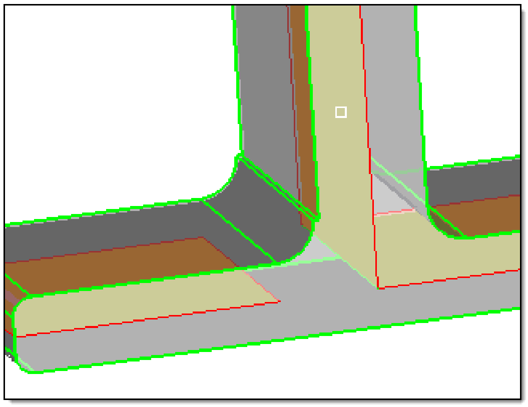


Figure 645:

The auto midsurface extraction tool attempts to determine where each point on the geometry should collapse to meet the generated midsurface. Errors occur when the extraction tool collapses points to the wrong corresponding locations on the midsurface. This often happens when the two end-points on a line collapse to separate points instead of to the same point.

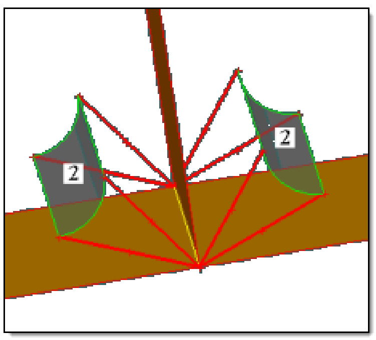


Figure 646:

The curved lines on each fillet are correctly associated with each other.

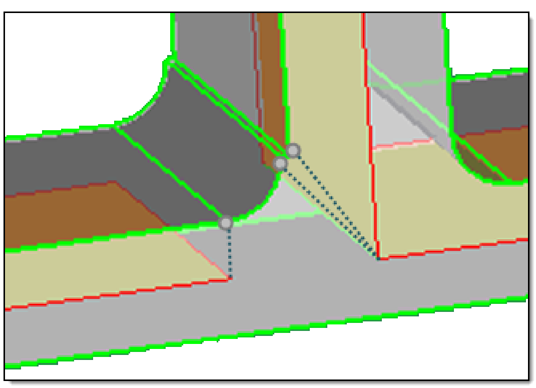


Figure 647:

The step in the fillet caused the extraction tool to associate the two points at the end of the step's edge with each other and collapse (correctly) toward the T-joint of the midsurface, while the remaining points at the other ends of the curved lines were then incorrectly collapsed in a normal direction instead of to the same point.

1. From the Geom page, select the **midsurface** panel.
2. Select the **interim edit tools**.
3. Select the **edit collapsed lines** subpanel.
4. Select the **allow rerun** checkbox.
5. Edit collapsed lines.

To edit collapsed lines by | **Do this**

Displaying point associations on the midsurface

Pre-requisite: Reject the current midsurface.

1. Set the switch to **prepare for rerun**.
2. Select the **auto extraction** subpanel.
3. Click **extract**.

The midsurface is extracted, and the lines that connect associated points are highlighted blue.

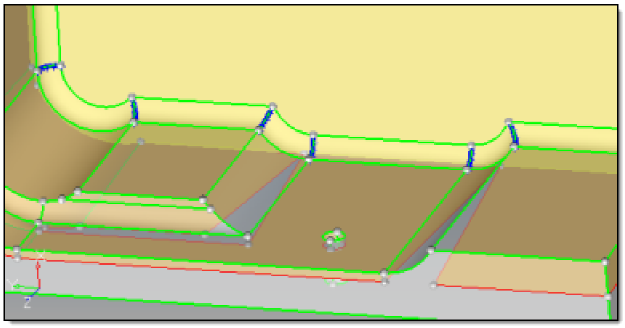
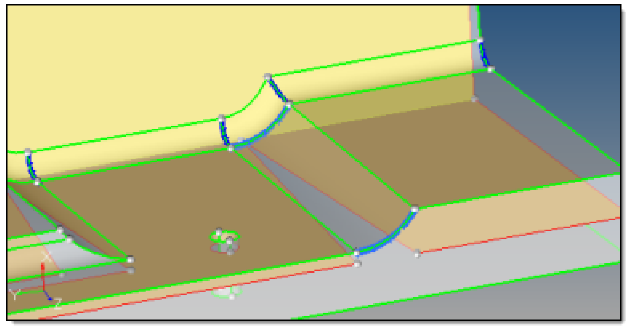
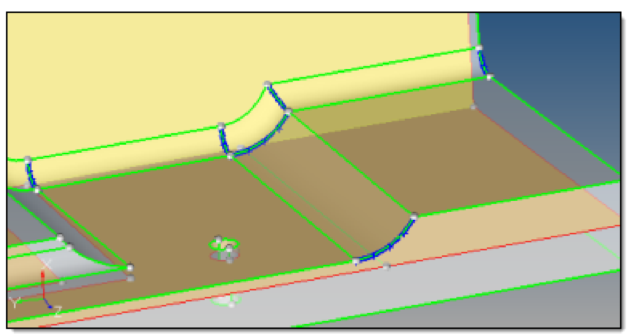


Figure 648:

Collapsing lines

1. Set the switch to **do rerun**.
2. Activate the **lines to collapse** selector.
3. Select each line segment that you wish to collapse to a single point on the midsurface.
4. Click **collapse**.

   The selected line segments collapse and are highlighted light blue, indicating that they are new collapsed lines.

To edit collapsed lines by	Do this
	 *Figure 649:* 5. Select the **auto extraction** subpanel. The collapsed lines that were automatically found during the initial midsurface extraction and the lines that you manually collapsed remain highlighted blue and flagged for collapse. 6. Click **extract**. The original midsurface is deleted and a new midsurface is generated using all of the lines that are flagged for collapse. **Note:** If allow rerun is disabled or the allow rerun switch is set to **no rerun** or **prepare for rerun**, then the lines that you manually collapsed will not be used when a new midsurface is extracted.  *Figure 650:*
Deleting collapsed lines	Pre-requisite: Display point associations on the midsurface. 1. Set the switch to **do rerun**. 2. Activate the **lines to collapse** selector. 3. Select collapsed lines to delete.

To edit collapsed lines by	Do this
	4. Click **delete**.
Trimming surfaces to create additional points and lines	1. Set the switch to **do rerun**. 2. Activate the **trim surfaces with cut line** selector. 3. Select surfaces to trim. 4. Click **drag a cut line**. 5. Click where you wish the line to begin, then click one or more times where you wish the lines to either end or change direction. 6. Press **Esc**. The line is generated and any selected surfaces it crosses are trimmed.

Repair Surfaces, Modify Targets, Imprint Surfaces

After you have extracted the midsurface and do not need to make further changes that require the midsurface to be re-extracted, repair surfaces, modify targets, and imprint surfaces using the final edit tools in the midsurface panel.

Repair Surfaces

Create midsurfaces for failed parts by extracting a midsurface from two faces that represent the two sides.

This function creates one surface that forms the midsurface.

1. From the Geom page, select the **midsurface** panel.
2. Select the **final edit tools**.
3. Select the **surface pair** subpanel.
4. Using the side 1: surfs selector, select a surface that represents one side of the solid.
5. Using the side 2: surfs selector, select a surface that represents the second side of the solid.
6. Select extraction options accordingly to define which surfaces to take into account when constructing midsurface.

   By default, only two surfaces that are selected with the side1 and side2 surf selectors are considered.

   - To take into account offset directions of the adjacent pre-existing midsurfaces, select the **use adjacent midsurfaces** checkbox.

- To allow modifying pre-existing adjacent midsurfaces, select the **combine with adjacent midsurfaces** checkbox.

7. Using the toggle, specify where to store midsurface geometry after extraction.
8. Click **extract**.

Modify Targets

Repair or edit a midsurface by correcting its targets.

Targets are the red and green segments that connect the points on the initial surface with the points where the surface must be offset. You can modify a midsurface that was created earlier, or a surface that is part of the solid.

The appearance of new temporary entities are displayed in different colors.

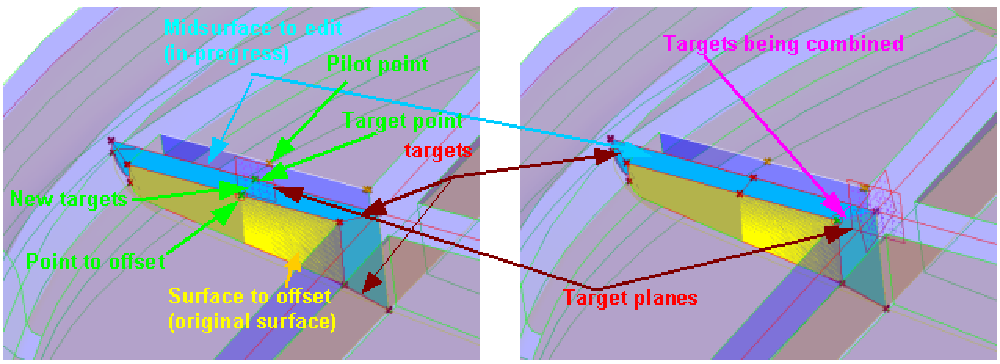


Figure 651:

Surface to offset (yellow)
: Original surface from which the middle surface was created by an offset with a variable direction and distance.

Midsurface to edit (cyan)
: Midsurface (in-progress) that you can modify by reassigning targets.

Targets (red)
: Handles that can be used to change the direction and distance of the offset. Targets are red segments that connect points on the initial surface with the points where the surface must be offset. The offset is interpolated inbetween the assigned targets.

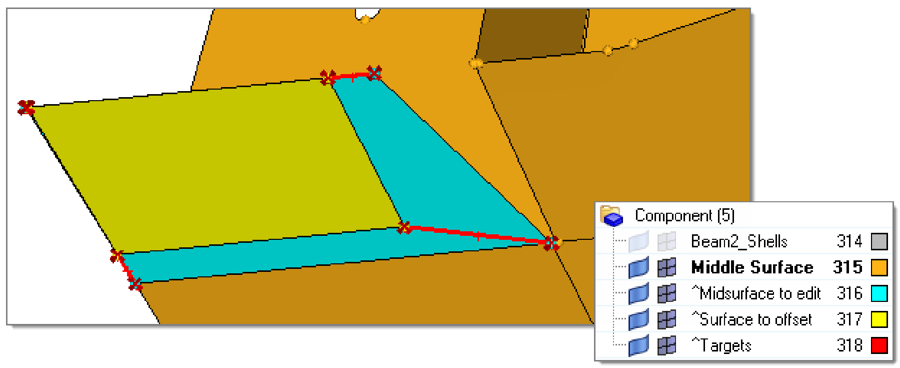
New targets (green)
: Targets that have not been accepted, thus they do not affect the midsurface.

Targets being combined (purple)
: New and existing targets that will be combined once they are accepted.

Target planes
: Planes parallel to the offset surface drawn at target points, which can be displayed for reference.

1. From the Geom page, select the **midsurface** panel.
2. Select the **final edit tools**.

3. Select the **assign target** subpanel.
4. Modify targets.

To modify targets by	Do this
Reassigning target	1. Using the point to point/edge to edge toggle, specify whether to select points or edges. 2. Verify that the surf selector is active, then select a midsurface that was created earlier or a surface that is part of a solid. Temporary components are created for the midsurface to edit, the surface to offset, and targets.  *Figure 652:* 3. Using the point/edge to offset selector, select an existing point/ edge or create a new point on the original surface (yellow) to offset. This point serves as the beginning point of the target. **Note:** To create a new point, hold your mouse on the edge of the surface (yellow) until the cursor turns into a square and the edge is highlighted. Release your mouse and left-click on the highlighted edge where you want to create a new point (target). A red circle is drawn around the point or at the center of the edge to indicate the selection.

To modify targets by	Do this

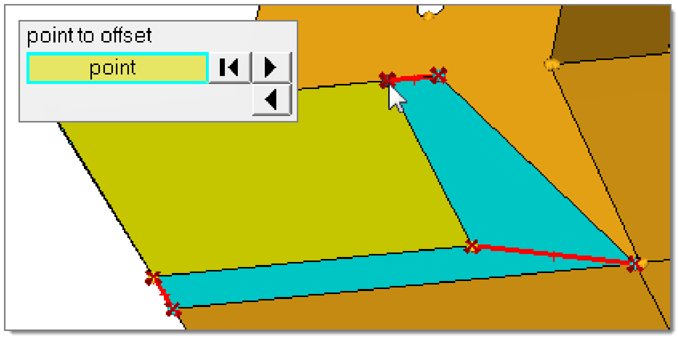


Figure 653:

4. Using the pilot point/edge selector, select an existing point/edge or create a new point to serve as the end point of the target.

> **Note:** Targets are created at the same location as the pilot point selected (as selected) or in the middle of the point/edge to offset and the pilot point/edge (midpoint). Use the target location toggle to specify where targets are created.

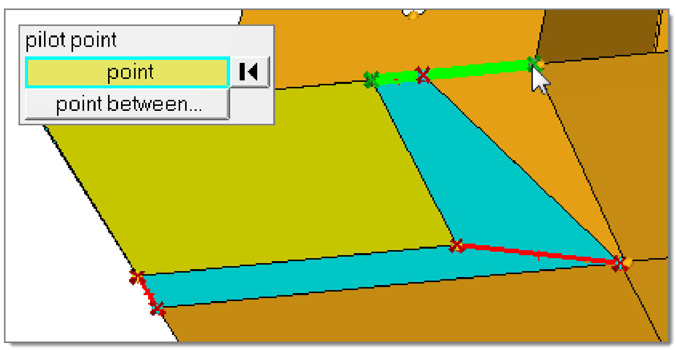
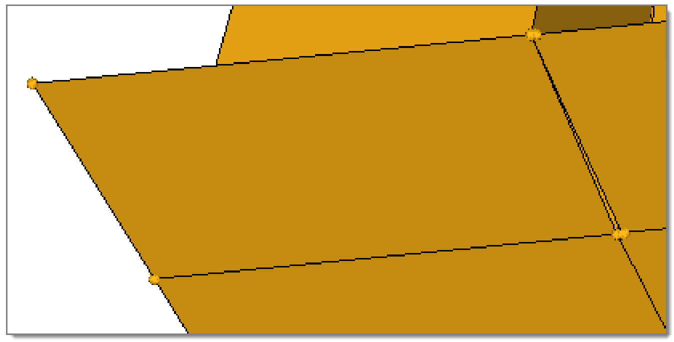


Figure 654:

5. To accept the location of the targets, click **accept targets**.
6. Click **offset**.

   The midsurface is redrawn with updated targets.
7. When you are satisfied with the results of your modifications, click **accept**.

To modify targets by	Do this
	 Figure 655:
Removing targets	1. Using the point to point/edge to edge toggle, select **point to point**. 2. Verify that the surf selector is active, then select a midsurface that was created earlier or a surface that is part of a solid. Temporary components are created for the midsurface to edit, the surface to offset, and targets.

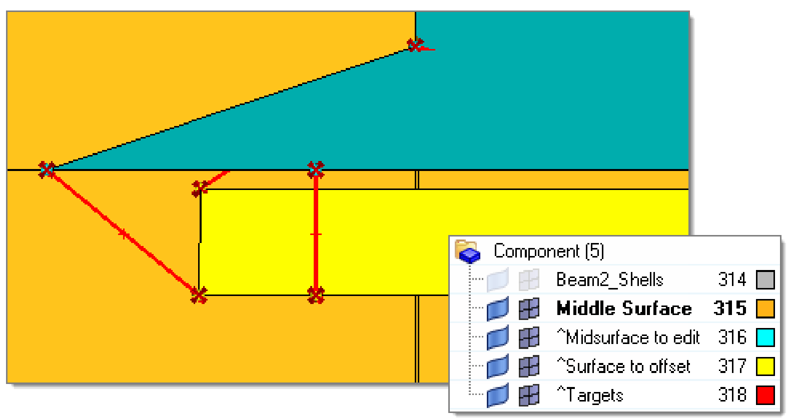
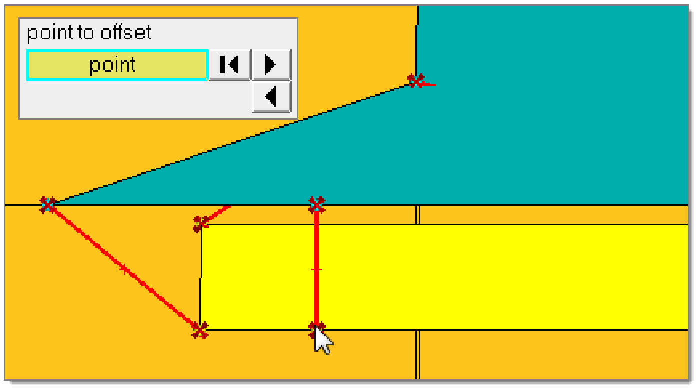
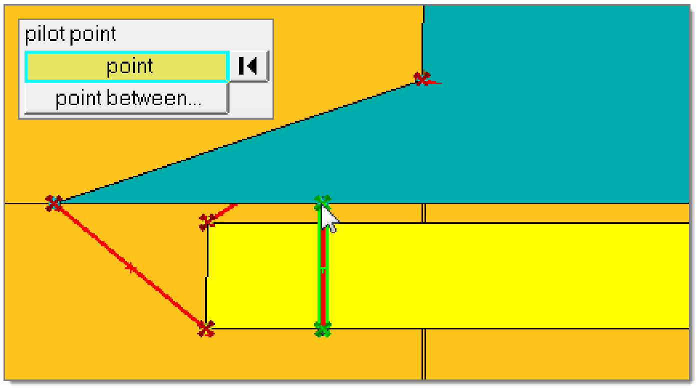


Figure 656:

3. Using the point/edge to offset selector, select a point on the original surface (yellow), which is the beginning point of a target.

To modify targets by	**Do this**
	 *Figure 657:* **4.** Using the pilot point/edge selector, select a point that serves as the end point of the target.  *Figure 658:* **5.** Click **remove target**.

To modify targets by	Do this

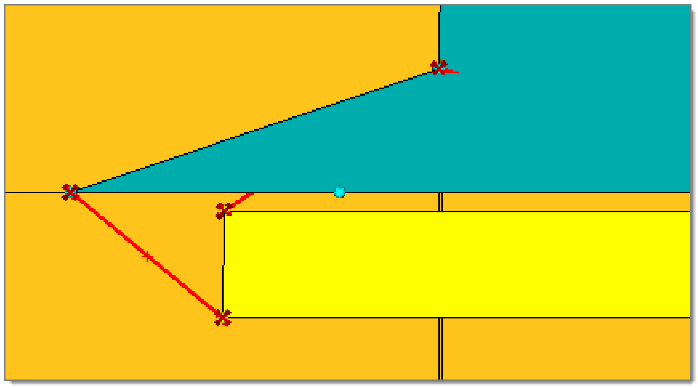


Figure 659:

Imprint Surfaces

Imprint solid geometry pockets, fillets and other feature boundaries on the midsurface.

Imprinting makes midsurface thickness calculations more accurate by creating mesh boundaries that match the feature boundaries on the original solid, and improves the meshing quality by capturing the features of the solid model.

You are not required to select the midsurface as HyperMesh automatically recognizes the midsurface corresponding to the solid/surface component from which it was derived, and imprints the lines on the surfaces in the midsurface component.

1. From the Geom page, select the **midsurface** panel.
2. Select the **final edit tools**.
3. Select the **imprint** subpanel.
4. Imprint surfaces.

To imprint	Do this
Lines	1. Set the lines/surfs to imprint selector to **lines**. 2. Select lines to imprint. 3. Using the line extend switch, specify how to extend lines. 4. To keep line endpoints, select **keep line endpoints**. 5. Using the target surfaces toggle, automatically or manually select target surfaces to imprint. 6. Click **imprint**.

To imprint	Do this
Surfaces	Surfaces (faces of solids) can be used to imprint on the midsurface instead of lines. **1.** Set the lines/surfs to imprint selector to **surfs**. **2.** Select surfaces to imprint. **3.** Using the line extend switch, specify how to extend lines. **4.** To keep line endpoints, select **keep line endpoints**. **5.** Using the surfs imprint toggle, specify which surface edges to imprint. **Note:** If surface edges are close to the Midsurface T edges or boundary, imprint all may produce bad element quality, in which case consider using smart imprint. **6.** Using the target surfaces toggle, automatically or manually select target surfaces to imprint. **7.** Click **imprint**.

Review and Modify Surface Thicknesses

Review the thickness of surfaces (including midsurfaces) and assign new fixed, uniform thicknesses to surfaces.

Surfaces that have thickness data stored are drawn with lines (probes) extending from each vertex of the surface. The length of these probes represent the thickness at those locations. By default, only surfaces created in the midsurface panel have thickness information defined.

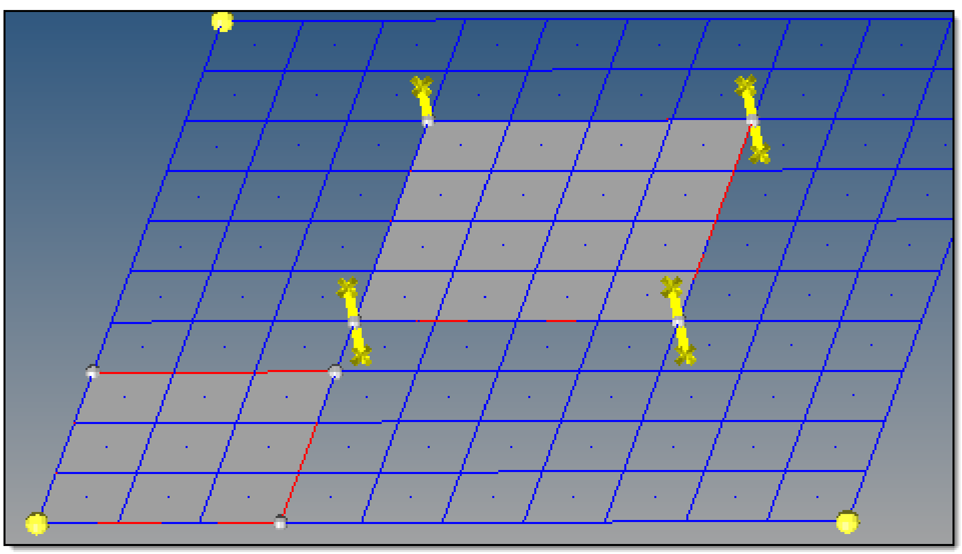


Figure 660:

The center surface (with probes at corners) has thickness data; the lower-left one does not.

1. From the Geom page, select the **midsurface** panel.
2. Select **review thickness**.
3. Edit surface thicknesses.

Option | **Description**

Review the thickness of surfaces

Using the view thickness: surfs selector, select surfaces to review.

The average thickness, maximum thickness, and minimum thickness for the selected surfaces are displayed.

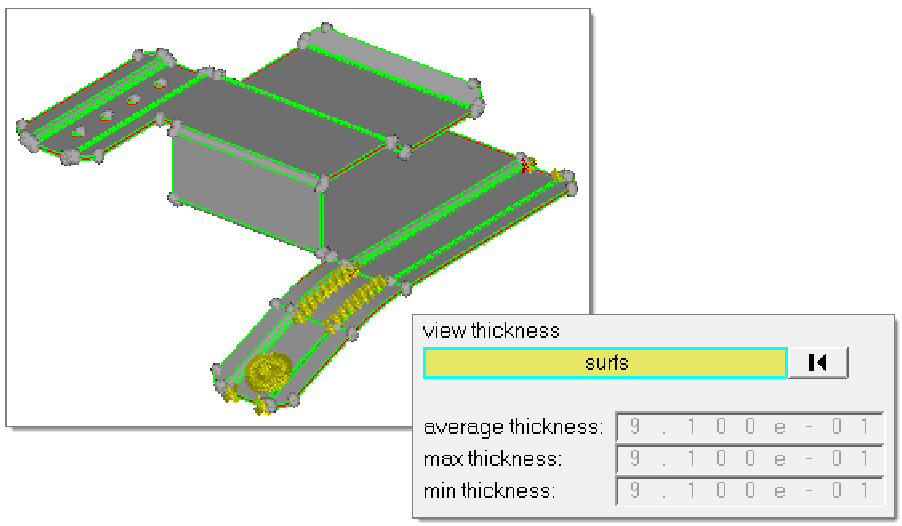


Figure 661:

Assign thicknesses to surfaces

1. Using the assign thickness: surfs selector, select surfaces to assign thicknesses to.
2. In the new thickness field, enter a thickness value.
3. Click **assign**.
4. To reject the thickness, click **reject** before leaving the panel.

Sort Midsurfaces

Manually sort midsurfaces after extraction.

1. From the Geom page, select the **midsurface** panel.
2. Select **sort**.
3. Select a method for sorting midsurfaces.
    - Choose **<~[o]riginal name> comp** to create new components by removing the first letter of the original component name and replacing it with a ~ and sorts the midsurfaces accordingly. For example, my_comp -> ~y_comp.
    - Choose **<Midsurface #nn> comp** to create components with the name Midsurface #, where # increases for each component that exists for the input surfaces/solids.
    - Choose **original comp** to organize the midsurfaces into their parent surface/solid components.
    - Choose **<original name~> comp** to create components using the original component name and adds ~ at the end.
    - Choose **<original nameN> comp** to create new components by incrementing the name of the original.
4. Click **sort**.

Match Topology

Fix gaps between stitched edges and vertices to make the actual geometry of the surfaces consistent with the model topology.

1. From the menu bar, click **Geometry** > **Geom Match Topology**.
   The **Geom Match Topology** dialog opens.
2. Using the **Entities** selector, select geometry to check.
3. In the **Tolerance** field, enter a tolerance to use when checking the topology of the model for gaps between shared, non-manifold, or suppressed edges.

   If a gap is found to be bigger than the specified tolerance, then the surface and edge geometries will be morphed parametrically and in 3D, if necessary, to make the gap smaller than the tolerance. In addition, non-essential degenerate edges are removed.
4. Click **Apply**.

The original geometric entity IDs are also preserved. The same functionality is used for the Optimize for CAD option in the Export - Geometry Browser.

Example: Match Topology

This example demonstrates the difference in results when using the Geom Match Topology tool to update geometry that has previously been repaired with topology-based geometry cleanup operations, for example, toggle/equivalence/replace edges, replace points, and so on.

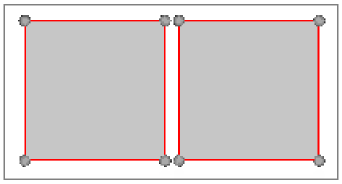


Figure 662: Unrepaired Geometry

The geometry below was repaired using only topology-based geometry cleanup operations. After the geometry was meshed, you can see that some of the elements have become distorted. After untoggling the edges, there is still a large gap in the geometry.

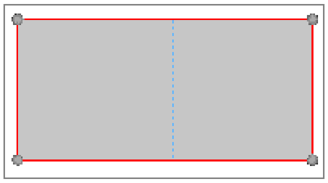


Figure 663: Repaired Geometry not Updated with the Geom Match Topology Tool

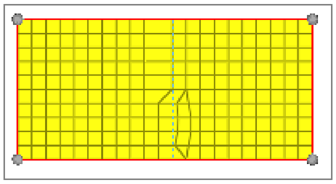


Figure 664: Meshed Geometry

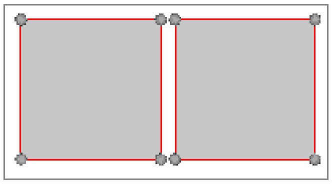


Figure 665: Geometry with Edges Untoggled

The geometry below was repaired using topology-based geometry cleanup operations and then updated with the Geom Match Topology tool. After the geometry was meshed, you can see that there are no distorted elements. After untoggling the edges, there is not a gap in the geometry.

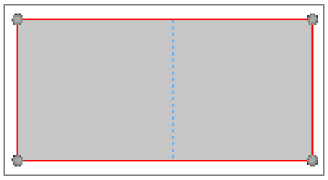


Figure 666: Repaired Geometry Updated with the Geom Match Topology Tool

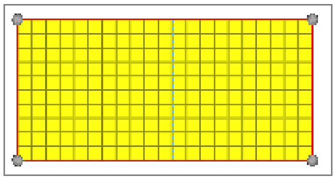


Figure 667: Meshed Geometry

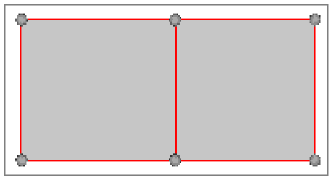


Figure 668: Geometry with Edges Untoggled

Find Intersections and Penetrations

Find intersections and penetrations between geometric surfaces and solids.

1. Open the Geometry Interference Check tool.
   - Click **Geometry** > **Checks** > **Surfaces**.
   - Click **Geometry** > **Checks** > **Solids**.
2. In the Check Type field, select a type.
   - Choose **All** to check all of the input entities against each other. There is only a single input collector. This is useful when it is not known specifically what entities should be checked. Optionally, self-interference can be included.
   - Choose **By pair** to checks entities in the first collector against those in the second. This is generally useful when it is known what specific entities should be checked against which other entities, for example, surfaces in component 1 vs surfaces in component 2. Self-interference cannot be included.
3. Use the entity selector to select geometry to check.
4. In the Interference type field, select a type.
   - Choose **All** to check for both penetrations and intersections.
   - Choose **Intersections only** to check for intersections.
   - Choose **Penetrations only** to check for penetrations.
5. Define thickness options to use when calculating penetrations.

   > **Restriction:** Only available when Interference type is set to **All** or **Penetrations only**.

   Penetration can only occur between two entities when one or both of them have an assigned thickness. Only surfaces not associated with solids utilize a thickness.

   - Choose **Component** to determine the thickness from the property assigned to the component. Each component can have a different thickness.
   - Choose **uniform** to manually enter a thickness. All surfaces will be assigned a constant thickness value.
6. To consider the edge effects of thickness, select the **Include edge effects** checkbox.

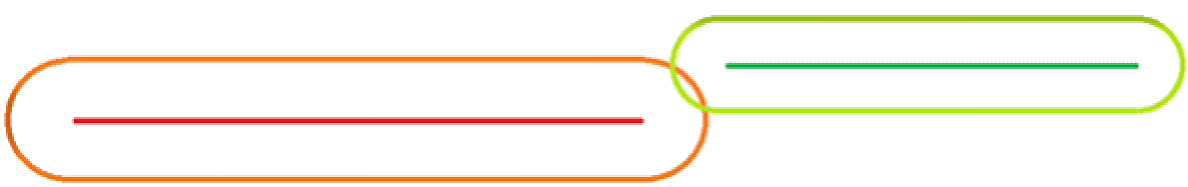
   

   Figure 669: Include Edge Effects On

   By default, the edge effects of thickness are not considered.

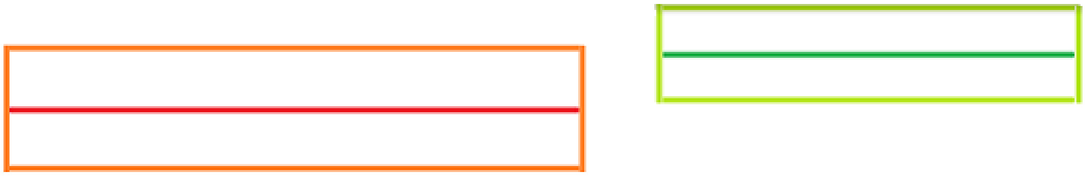


Figure 670: Include Edge Effects Off

7. In the Isolate only failed field, select a method for managing the display of entities that failed the check.
   - Choose **None** to not change the display.
   - Choose **Components** to isolate the components that contain failed surfaces, and hide all other entities.
   - Choose **Surfaces** to isolate failed surfaces (and any associated solids), and hide all other entities.
8. In the Save failed field, select a method for saving entities that failed the check.

   > **Tip:**
   > Use the **retrieve** option in other panels' advanced entity selection menus to retrieve the saved entities.

   - Choose **None** to not save any failed entities to the user mark.
   - Choose **Components** to save the components that contain failed surfaces to the user mark.
   - Choose **Surfaces** to save failed surfaces to the user mark.
   - Choose **Both** to save both components and surfaces to the user mark.
9. Click **Run**.

Setup CAD Models with Metadata

Use metadata generated as part of the CAD import process to setup CAE models.

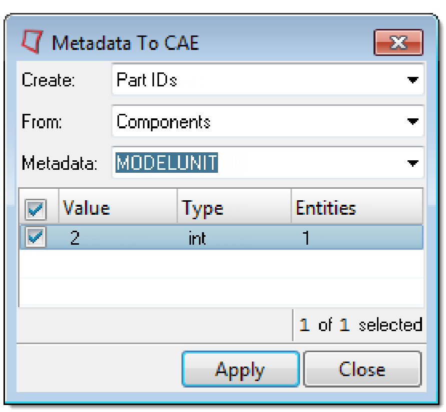
Rename Components from Metadata Attached to Components

1. From the menu bar, click **Geometry** > **Metadata to CAE**.
   The **Metadata to CAE** dialog opens.
2. For Create, select **Part Names**.
3. For From, select **Components**.
4. For Metadata, select a specific metadata name.
   A list of entities is generated.
5. In the table, select entities to operate on.
6. Click **Apply**.

The new names are taken as the values of the specified metadata attached to each selected component. If there are duplicate names, an incremental name is generated.

Renumber Components from Metadata Attached to Components

1. From the menu bar, click **Geometry** > **Metadata to CAE**.
   The **Metadata to CAE** dialog opens.
2. For Create, select **Part IDs**.
3. For From, select **Components**.
4. For Metadata, select a specific metadata name.
   A list of entities is generated.
5. In the table, select entities to operate on.



6. Click **Apply**.

The new IDs are taken as the values of the specified metadata attached to each selected component. If there are duplicate IDs, the entities will not be renumbered.

Create Regions from Geometry with Associated Metadata

1. From the menu bar, click **Geometry** > **Metadata to CAE**.
   The **Metadata to CAE** dialog opens.
2. For Create, select **Regions**.
3. For From, select a type of geometry.
4. For Metadata, select a specific metadata name.
   A list of entities is generated.
5. In the table, select entities to create regions from.
6. Click **Apply**.

Create Spot Connectors from Points with Associated Metadata

1. From the menu bar, click **Geometry** > **Metadata to CAE**.
   The **Metadata to CAE** dialog opens.
2. For Create, select **Connectors**.
3. For From, select **Points**.
4. For Metadata, select a specific metadata name.
   A list of points is generated.
5. In the table, select the points to create spot connectors from.
6. Click **Apply**.

Spot connectors are created from the selected points using components as links based on proximity. By default, HyperMesh uses two links. You can modify and/or realize the connectors using the Connector Browser or connector panels.

Create Seam Connectors from Lines with Associated Metadata

1. From the menu bar, click **Geometry** > **Metadata to CAE**.
   The **Metadata to CAE** dialog opens.
2. For Create, select **Connectors**.
3. For From, select **Lines**.
4. For Metadata, select a specific metadata name.
   A list of lines is generated.

5. In the table, select the lines to create seam connectors from.

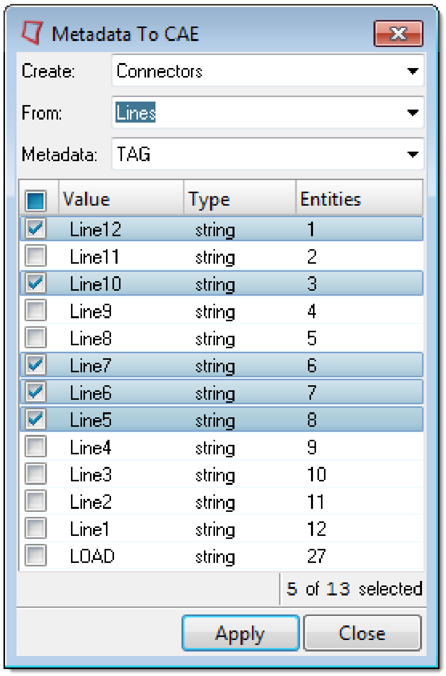


Figure 671:

6. Click **Apply**.

Seam connectors are created from selected lines using components as links based on proximity. By default, HyperMesh uses a spacing of 1.0. You can modify and/or realize the connectors using the Connector Browser or connector panels.

Create Area Connectors from Surfaces with Associated Metadata

1. From the menu bar, click **Geometry** > **Metadata to CAE**.
   The **Metadata to CAE** dialog opens.
2. For Create, select **Connectors**.
3. For From, select **Surfaces**.
4. For Metadata, select a specific metadata name.
   A list of surfaces is generated.
5. In the table, select the surfaces to create area connectors from.

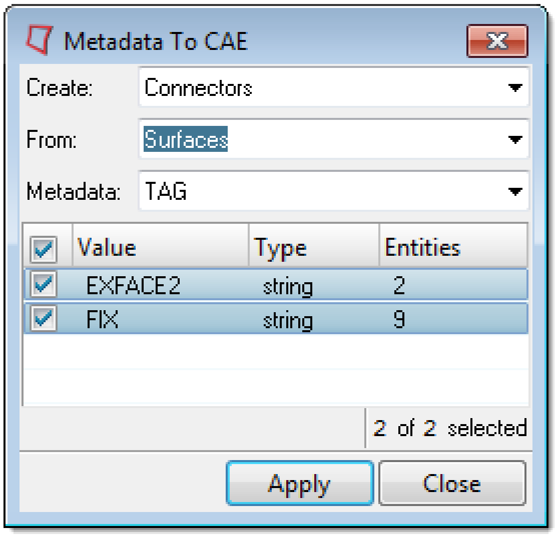


Figure 672:

6. Click **Apply**.

Area connectors are created from selected surfaces using components as links based on proximity. You can modify and/or realize the connectors using the Connector Browser or connector panels. By default, a quad mesh with an element size of 10.0 is used.

Meshing 7

Learn about the different types of mesh you can create in HyperMesh.

This chapter covers the following:

Models are typically meshed automatically, but individual elements can also be modified manually to improve element quality in problem areas.

Meshing is organized by the type of mesh operation involved.

Elements

Elements are FE idealizations for a portion of a physical part.

Every element must be organized into one component, and therefore are mutually exclusive to a component.

Element Configurations

Each element has an associated element configuration. An element configuration tells HyperMesh how to draw, store, and work with the element.

Bar Elements

1D elements created in a space between two or three nodes of a model where beam properties are desired.

The nodes are related to each other based on the properties of the bar or beam element connecting them. Properties associated with bar elements include vector orientation, offset vectors that end at A and B, or at A, B, and C, and pin flags to tell it what degree of freedom should carry through the beam.

Bar elements are displayed as a line between two nodes with BAR2 or BAR3 written at the centroid of the element.

Bar2

Configuration 60 - 1D (1st order) elements with 2 nodes used to model axial, bending, and torsion behavior. Bar2 elements have a property reference, an orientation vector, offset vectors and ends A and B, and pin flags at ends A and B.

Bar3

Configuration 63 - 1D (2nd order) elements with 3 nodes used to model axial, bending, and torsion behavior. Bar3 elements have a property reference, an orientation vector, offset vectors and ends A and B, and pin flags at ends A and B.

Gap Elements

Configuration 70 - 1D elements created in a space between two nodes, or between a node and an element, of a model where contact may occur.

Create a gap element when you want to impose a nonlinear constraint on a model; this constraint will limit the amount of movement possible during analysis.

Gap elements have a property reference and an orientation vector.

Gap elements are displayed as a line between two nodes with `GAP` written at the centroid of the element.

Gap elements can translate to `CGAP` or `CGAPG` elements in OptiStruct, `CGAP` element in Nastran or *GAP option in Abaqus.

Hex Elements

3D hexahedra elements.

Hex8

Configuration 208 - 3D (1st order) hexahedra elements with 8 nodes ordered HyperMesh.

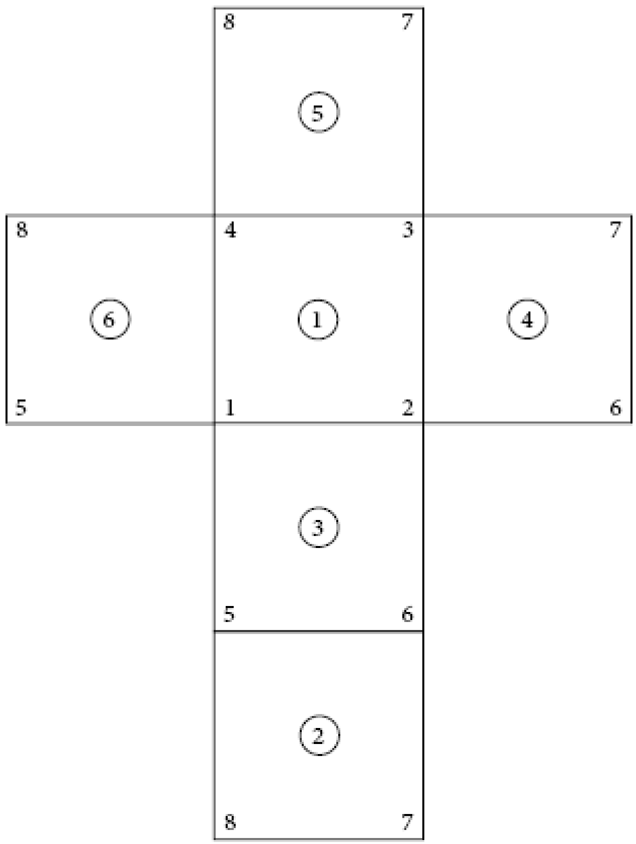


Figure 673:

Hex20

Configuration 220 - 3D (2nd order) hexahedra elements with 20 nodes ordered in HyperMesh.

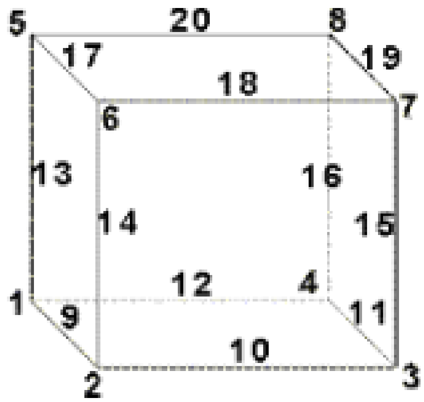


Figure 674: Element Configuration 220, 20-Noded Hexa

Joint Elements

Configuration 22 - 1D elements with 2, 4, or 6 nodes which have a property and orientation systems or nodes.

Joint element is a definition of a connection between two rigid bodies. Joint elements store a property and orientation information.

Joint elements are displayed with lines between the appropriate nodes and the letter J between nodes 1 and 3 of the element.

Only certain types of elements can be used to create joint elements. The type of the element controls the number of nodes used in the element and the permissible orientations of the element.

Table 191: Types of Joint Elements

Type	Type Name	Number of Nodes	Orientation	Solver Interface
1	Spherical joint	2	None Systems Nodes	LS-DYNA PAM-CRASH
2	Revolute joint	4	None Systems	LS-DYNA
3	Cylindrical joint	4	None Systems	LS-DYNA
4	Planar joint	4	None Systems	LS-DYNA

Type	Type Name	Number of Nodes	Orientation	Solver Interface
5	Universal joint	4	None Systems	LS-DYNA
6	Translational joint	6	None Systems	LS-DYNA
7	Locking joint	6	None Systems	LS-DYNA
8	Ball joint	2	None	OptiStruct
9	Fixed joint	2	None	OptiStruct
10	Revolute joint	2	Node Vector Coordinates	OptiStruct
11	Translational1 joint	2	Node Vector Coordinates	OptiStruct
12	Cylindercal 1 joint	2	Node Vector Coordinates	OptiStruct
13	Universal joint	2	Node Vector Coordinates	OptiStruct
14	Constant_velocity joint	2	Node Vector Coordinates	OptiStruct
15	Planar joint	2	Node Vector Coordinates	OptiStruct

Type	Type Name	Number of Nodes	Orientation	Solver Interface
16	Inline joint	2	Node Vector Coordinates	OptiStruct
17	Perpendicular joint	2	Node Vector Coordinates	OptiStruct
18	Parallel axes joint	2	Node Vector Coordinates	OptiStruct
19	Inplane joint	2	Node Vector Coordinates	OptiStruct
20	Orient joint	2	Node Vector Coordinates	OptiStruct
21	Point_to_curve joint	2	Node Vector Coordinates	OptiStruct
22	Curve_to_curve joint	2	Node Vector Coordinates	OptiStruct
23	Point_to_deformable_cu joint	2	Node Vector Coordinates	OptiStruct
24	Point_to_deformable_su joint	2	Node Vector Coordinates	OptiStruct

Type	Type Name	Number of Nodes	Orientation	Solver Interface
25	Translational_2N joint	2	None Systems	PAM-CRASH
26	Revolute_2N joint	2	None Systems	PAM-CRASH
27	Cylindrical_2N joint	2	None Systems	PAM-CRASH
28	Universal_2N joint	2	None Systems	PAM-CRASH
29	Flexion-Torsion joint	2	None Systems	PAM-CRASH
30	Planar_2N joint	2	None Systems	PAM-CRASH
31	General joint	2	None Systems	PAM-CRASH
32	Bracket joint	2	None Systems	PAM-CRASH
33	Free joint	2	None Systems	PAM-CRASH

Mass Elements

Configuration 1 - 0D elements with a single node that allow you to assign concentrated mass to the model in order to represent a physical part that may not be modeled with another FE idealization.

Mass elements are displayed as a dot with the letter M written at the centroid of the element.

Master Elements

Master interface elements.

Master3

Configuration 123 - Master interface elements with 3 nodes. (Must be Type 1).

Master4

Configuration 124 - Master4 elements are master interface elements with 4 nodes. (Must be Type 1).

Penta Elements

3D triangular prism pentahedra elements.

Penta6

Configuration 206 - 3D (1st order) triangular prism pentahedra elements with 6 nodes ordered in HyperMesh.

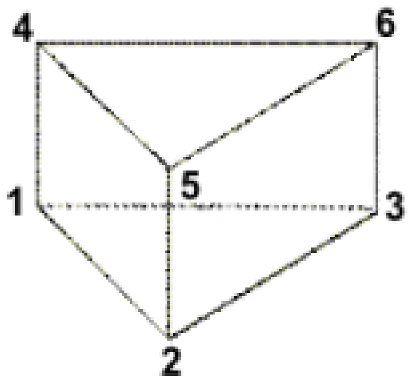


Figure 675: Element Configuration 206, 6-Noded Penta

Penta15

Configuration 215 - 3D (2nd order) triangular prism pentahedra elements with 15 nodes ordered in HyperMesh.

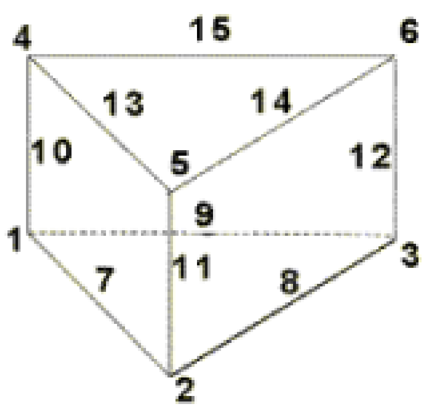


Figure 676: Element Configuration 215, 15-Noded Penta

Plot Elements

Configuration 2 - 1D elements with 2 nodes used for display purposes.

Plot elements are displayed as a line between two nodes.

Pyramid Elements

3D pyramid pentahedra elements.

Pyramid5

Configuration 205 - 3D (1st order) pyramid pentahedra elements with 5 nodes ordered in HyperMesh.

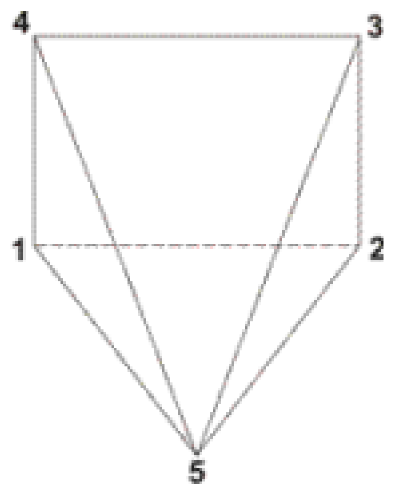


Figure 677: Element Configuration 205, 5-Noded Pyramid

Pyramid13

Configuration 213 - 3D (2nd order) pyramid pentahedra elements with 5 nodes ordered in HyperMesh.

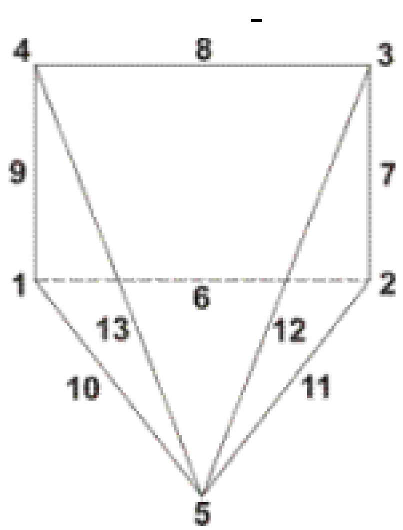


Figure 678: Element Configuration 213, 13-Noded Pyramid

Quad Elements

2D quadrilateral elements.

Types of pyramid elements include:

Quad4

Configuration 104 - 2D (1st order) quadrilateral elements with 4 nodes ordered in HyperMesh.

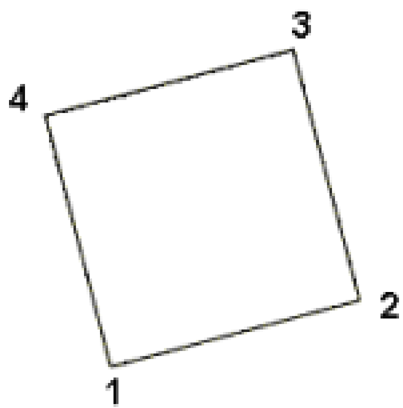


Figure 679: Element Configuration 104, 4-Noded Quad

Quad8

Configuration 108 - 2D (2nd order) quadrilateral elements with 8 nodes ordered in HyperMesh.

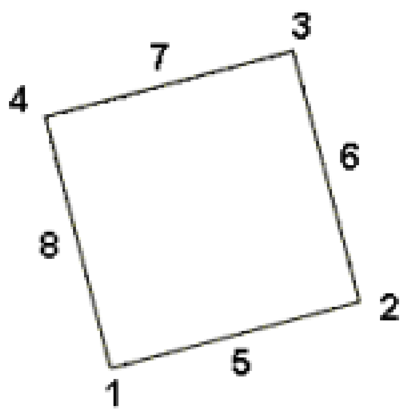


Figure 680: Element Configuration 108, 8-Noded Quad

RBE3 Elements

Configuration 56 - Rigid elements with one dependent node and variable independent nodes typically used to define the motion at the dependent node as a weighted average of the motions at the independent nodes.

Both the dependent node and independent nodes contain a coefficient (weighting factor) and user-defined degrees of freedom. The dependent degrees of freedom and weighting factors can be specified or automatically calculated based on the geometry.

RBE3 elements are displayed as lines between the dependent node and the independent node(s) with RBE3 displayed at the dependent node of the element.

RBE3's are typically used to distribute loads applied on the dependent node amongst the selected independent nodes.

> **Note:** The dependent node cannot be directly constrained, as this would lead to a double-dependency for that node.

Rigid Elements

Configuration 5 - Rigid 1D elements with 2 nodes used to model rigid connections.

Rigid elements are displayed as a line between two nodes with the letter R written at the centroid of the element.

Rigids can translate to RBE2 in Nastran or *MPC in Abaqus.

Rigidlink Elements

Configuration 55 - Rigid elements with one independent node and variable dependent nodes typically used to model rigid bodies.

Rigidlink elements have user-defined degrees of freedom which apply to all dependent nodes.

Rigidlink elements can be created with dependent nodes attached to an element as a SET. If a rigid link with a dependent node set is deleted, the associated node set is also deleted. If the dependent node set is deleted, the connected rigid link element is also deleted. Dependent node sets are automatically created when rigid link elements are created. A node set can be connected as a set of dependent nodes to a rigid link element independent node.

> **Note:** Two-node rigids with a dependent node set attached are always created as rigid link elements

Rigidlink elements are displayed as lines between the independent node and the dependent node(s) with RL displayed at the independent node of the element.

Rod Elements

Configuration 61 - 1D elements with 2 nodes used to model axial behavior only.

The two nodes are related to each other based on the properties of the rod element connecting them. Rod elements have property pointers.

Rod elements are displayed as a line between two nodes with ROD written at the centroid of the element.

Rods can translate to CTUBES in Nastran or a C1D2 element in Abaqus.

Slave Elements

Slave interface elements.

Slave1
: Configuration 135 - Slave interface elements with 1 node. (Must be Type 1).

Slave3
: Configuration 133 - Slave interface elements with 3 node. (Must be Type 1).

Slave4

Configuration 134 - Slave interface elements with 1 node. (Must be Type 1).

Spring Elements

Configuration 21 - 1D elements used to model spring connections.

Spring elements have user-defined degrees of freedom, an orientation vector, and a property reference.

Spring elements are displayed as a line between two nodes with the letter K written at the centroid of the element.

Spring

1D elements with 2 nodes used to model spring connections.

Spring2N

1D elements with 2 nodes used to model spring connections.

Spring3N

1D elements with 3 nodes used to model spring connections.
The third node serves as the direction node.

Spring4N

1D elements with 4 nodes used to model spring connections.
This type of element will mostly be considered as joints, based on the property it is assigned.

Springs can translate to CELAS2 in Nastran or *SPRING in Abaqus.

Tetra Elements

3D tetrahedra elements.

Tetra4

Configuration 204 - 3D (1st order) tetrahedra elements with 4 nodes ordered in HyperMesh.

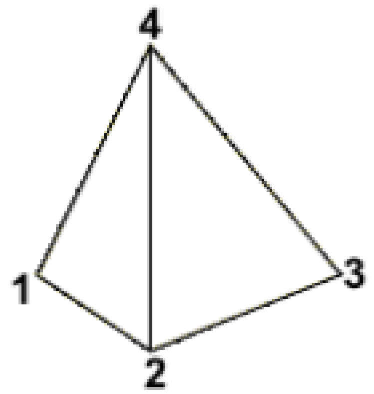


Figure 681: Element Configuration 204, 4-Noded Tetra

Tetra10

Configuration 210 - 3D (2nd order) tetrahedra elements with 10 nodes ordered in HyperMesh.

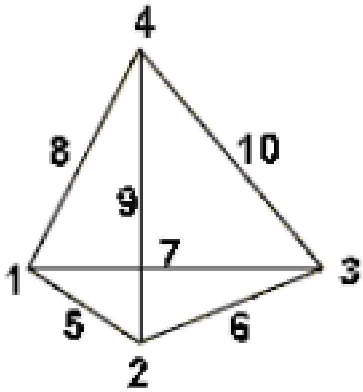


Figure 682: Element Configuration 210, 10-Noded Tetra

Tria Elements

2D triangular elements.

Tria3

Configuration 103 - 2D (1st order) triangular elements with 3 nodes ordered in HyperMesh.

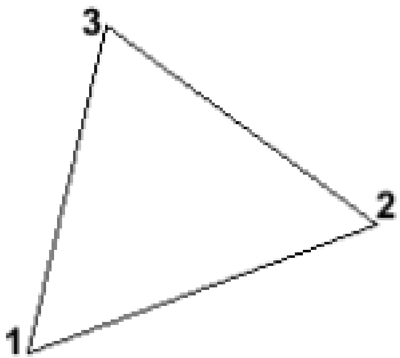


Figure 683: Element Configuration 103, 3-Noded Tria

Tria6

Configuration 106 - 2D (2nd order) triangular elements with 6 nodes ordered in HyperMesh.

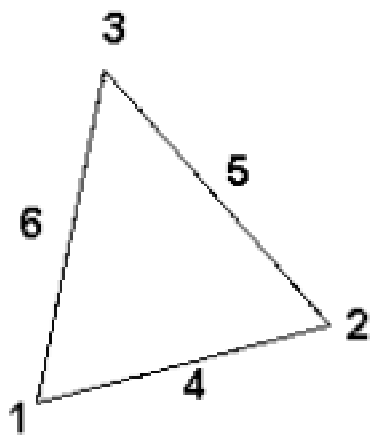


Figure 684: Element Configuration 106, 6-Noded Tria

Weld Elements

Configuration 3 - Rigid 1D elements with 2 nodes used to model welded connections.

Weld elements are displayed as a line between two nodes with the letter W written at the centroid of the element.

Xelems

1D multi-strand elements.

Supported Solver Cards

Solver cards supported for elements.

Abaqus Cards

Card	Supported Element Configurations	Description
*COUPLING	Rigid/RBE3	Define a surface-based coupling constraint where the *SURFACE card points to elements. **Note:** The *COUPLING is also supported as rigid elements (COUP_KIN) and RBE3 (COUP_DIS) when *SURFACE points to nodes.
*ELEMENT	Bar2 Bar3 Gap (Standard 2D/3D)	Defines elements by giving their nodes.

Card	Supported Element Configurations	Description
	Hex20 (Standard 3D) Hex8 (Standard 3D/Explicit) Mass Penta15 (Standard 3D) Penta6 (Standard 3D/Explicit) Pyramid13 (Standard 3D) Pyramid5 (Standard 3D/Explicit) Quad4 Quad8 (Standard 2D/3D) RBE3 Rigid Rod Spring Tetra10 (Standard 3D/Explicit) Tetra4 (Standard 3D/Explicit) Tria3 Tria6 (Standard 2D/3D)	
*ELGEN		Generates elements incrementally. These cards are resolved to individual entities on import and are written back on export the same way. **Note:** Standard 2D only.
*MPC	Rigid	Defines multi-point constraints.
*RELEASE	Bar2	Releases rotational degrees of freedom at one or both ends of a beam element.

Card	Supported Element Configurations	Description
	Bar3	For 2D problems, only dof6 (M1) is active. Add a *RELEASE card to this element by clicking pins a = and pins b = and type in the HyperMesh dof code for the Abaqus release combination code you want. **HyperMesh dof Code** **Abaqus Release Combination Code** **4** T **5** M2 **45** M2-T **6** M1 **46** M1-T **56** M1-M2 **456** ALLM For 2D problems, only dof6 (M1) is active.
AC1D2	Bar2	2-node acoustic link
AC1D3	Bar3	3-node acoustic link
AC2D3	Tria3	3-node linear 2-D acoustic triangle
AC2D4	Quad4	4-node linear 2-D acoustic quadrilateral
AC2D6	Tria6	6-node quadratic 2-D acoustic triangular prism
AC2D8	Quad8	8-node quadratic 2-D acoustic quadrilateral
AC3D10	Tetra10	10-node quadratic acoustic tetrahedron
AC3D15	Pyramid13 Penta15	15-node quadratic acoustic triangular prism

Card	Supported Element Configurations	Description
AC3D20	Pyramid13 Hex20	20-node quadratic acoustic brick
AC3D4	Tetra4	4-node linear acoustic tetrahedron
AC3D6	Penta6	6-node linear acoustic triangular prism
AC3D8	Hex8	8-node linear acoustic brick
AC3D8R	Hex8	
ACAX3	Tria3	3-node linear axisymmetric acoustic triangle
ACAX4	Quad4	4-node linear axisymmetric acoustic quadrilateral
ACAX6	Tria6	6-node quadratic axisymmetric acoustic triangle
ACAX8	Quad8	8-node quadratic axisymmetric acoustic quadrilateral
ACIN3D3	Tria3	3-node linear 3D acoustic infinite element
ACIN3D4	Quad4	4-node linear 3D acoustic infinite element
ACIN3D6	Tria6	6-node quadratic 3D acoustic infinite element
ACIN3D8	Quad8	8-node quadratic 3D acoustic infinite element
B21	Bar2	2-node linear beam in a plane
B21H	Bar2	2-node linear beam in a plane, hybrid formulation
B22	Bar3	3-node quadratic beam in a plane
B22H	Bar3	3-node quadratic beam in a plane, hybrid formulation
B23	Bar2	2-node cubic beam in a plane
B23H	Bar2	2-node cubic beam in a plane, hybrid formulation
B31	Bar2	2-node linear beam
B31H	Bar2	2-node linear beam, hybrid formulation
B31OS	Bar2	2-node linear open-section beam

Card	Supported Element Configurations	Description
B31OSH	Bar2	2-node linear open-section beam, hybrid formulation
B32	Bar3	3-node quadratic beam in space
B32	Bar3	3-node quadratic beam
B32H	Bar3	3-node quadratic beam, hybrid formulation
B32OS	Bar3	3-node quadratic open-section beam
B32OSH	Bar3	3-node quadratic open-section beam, hybrid formulation
B33	Bar2	2-node cubic beam
B33H	Bar2	2-node cubic beam, hybrid formulation
BEAM	Rigid	
C3D10	Tetra10	10-node quadratic tetrahedron
C3D10E	Tetra10	10-node quadratic piezoelectric tetrahedron
C3D10H	Tetra10	10-node quadratic tetrahedron, hybrid, constant pressure
C3D10HS	Tetra10	
C3D10I	Tetra10	10-node general-purpose quadratic tetrahedron, improved surface stress visualization
C3D10M	Tetra10	10-node modified tetrahedron, hourglass control
C3D10MH	Tetra10	10-node modified quadratic tetrahedron, hybrid, linear pressure, hourglass control
C3D10MP	Tetra10	10-node modified displacement and pore pressure tetrahedron, hourglass control
C3D10MPH	Tetra10	10-node modified displacement and pore pressure tetrahedron, hybrid, linear pressure, hourglass control
C3D10MT	Tetra10	10-node thermally coupled modified quadratic tetrahedron, hourglass control
C3D10S		

Card	Supported Element Configurations	Description
C3D15	Pyramid13 Penta15	15-node quadratic triangular prism
C3D15E	Pyramid13 Penta15	15-node quadratic piezoelectric triangular prism
C3D15H	Pyramid13 Penta15	15-node quadratic triangular prism, hybrid, linear pressure
C3D20	Pyramid13 Penta15 Hex20	20-node quadratic brick
C3D20E	Pyramid13 Hex20	20-node quadratic piezoelectric brick
C3D20H	Pyramid13 Hex20	20-node quadratic brick, hybrid, linear pressure
C3D20HT	Pyramid13 Hex20	20-node thermally coupled brick, triquadratic displacement, trilinear temperature, hybrid, linear pressure
C3D20PH	Hex20	20-node brick, triquadratic displacement, trilinear pore pressure, hybrid, linear pressure
C3D20R	Pyramid13 Penta15 Hex20	20-node quadratic brick, reduced integration
C3D20RE	Pyramid13 Hex20	20-node quadratic piezoelectric brick, reduced integration
C3D20RH	Pyramid13 Hex20	20-node quadratic brick, hybrid, linear pressure, reduced integration

Card	Supported Element Configurations	Description
C3D20RHT	Pyramid13 Hex20	20-node thermally coupled brick, triquadratic displacement, trilinear temperature, hybrid, linear pressure, reduced integration
C3D20RP	Hex20	20-node brick, triquadratic displacement, trilinear pore pressure, reduced integration
C3D20RPH	Hex20	20-node brick, triquadratic displacement, trilinear pore pressure, hybrid, linear pressure, reduced integration
C3D20RT	Pyramid13 Hex20	20-node thermally coupled brick, triquadratic displacement, trilinear temperature, reduced integration
C3D20T	Pyramid13 Hex20	20-node thermally coupled brick, triquadratic displacement, trilinear temperature
C3D30P	Hex20	20-node brick, triquadratic displacement, trilinear pore pressure
C3D4	Tetra4	4-node linear tetrahedron
C3D4E	Tetra4	4-node linear piezoelectric tetrahedron
C3D4H	Tetra4	4-node linear tetrahedron, hybrid, linear pressure
C3D4P	Tetra4	
C3D4T	Tetra4	4-node thermally coupled tetrahedron, linear displacement and temperature
C3D6	Pyramid5 Penta6	6-node linear triangular prism
C3D6E	Pyramid5 Penta6	6-node linear piezoelectric triangular prism
C3D6H	Pyramid5 Penta6	6-node linear triangular prism, hybrid, constant pressure
C3D6P	Penta6	

Card	Supported Element Configurations	Description
C3D6T	Penta6	6-node thermally coupled triangular prism, linear displacement and temperature
C3D8	Pyramid5 Penta6 Hex8	8-node linear brick
C3D8E	Pyramid5 Hex8	8-node linear piezoelectric brick
C3D8H	Pyramid5 Hex8	8-node linear brick, hybrid, constant pressure
C3D8HS	Hex8	
C3D8HT	Pyramid5 Hex8	8-node thermally coupled brick, trilinear displacement and temperature, hybrid, constant pressure
C3D8I	Pyramid5 Hex8	8-node linear brick, incompatible modes
C3D8IH	Pyramid5 Hex8	8-node linear brick, hybrid, linear pressure, incompatible modes
C3D8P	Hex8	8-node brick, trilinear displacement, trilinear pore pressure
C3D8PH	Hex8	8-node brick, trilinear displacement, trilinear pore pressure, hybrid, constant pressure
C3D8R	Pyramid5 Penta6 Hex8	8-node linear brick, reduced integration, hourglass control
C3D8RH	Pyramid5 Hex8	8-node linear brick, hybrid, constant pressure, reduced integration, hourglass control
C3D8RHT	Pyramid5	8-node thermally coupled brick, trilinear displacement and temperature, reduced

Card	Supported Element Configurations	Description
	Hex8	integration, hourglass control, hybrid, constant pressure
C3D8RHT	Pyramid5	8-node thermally coupled brick, trilinear displacement and temperature, reduced integration, hourglass control, hybrid, constant pressure
C3D8RP	Hex8	8-node brick, trilinear displacement, trilinear pore pressure, reduced integration
C3D8RPH	Hex8	8-node brick, trilinear displacement, trilinear pore pressure, reduced integration, hybrid, constant pressure
C3D8RT	Pyramid5 Hex8	8-node thermally coupled brick, trilinear displacement and temperature, reduced integration, hourglass control
C3D8S	Hex8	
C3D8T	Pyramid5 Hex8	8-node thermally coupled brick, trilinear displacement and temperature
CAX3	Tria3	3-node linear axisymmetric triangle
CAX3E	Tria3	3-node linear axisymmetric piezoelectric triangle
CAX3H	Tria3	3-node linear axisymmetric triangle, hybrid, constant pressure
CAX3T	Tria3	3-node axisymmetric thermally coupled triangle, linear displacement and temperature
CAX4	Quad4	4-node bilinear axisymmetric quadrilateral
CAX4E	Quad4	4-node bilinear axisymmetric piezoelectric quadrilateral
CAX4H	Quad4	4-node bilinear axisymmetric quadrilateral, hybrid, constant pressure
CAX4HT	Quad4	4-node axisymmetric thermally coupled quadrilateral, bilinear displacement and temperature, hybrid, constant pressure

Card	Supported Element Configurations	Description
CAX4I	Quad4	4-node bilinear axisymmetric quadrilateral, incompatible modes
CAX4IH	Quad4	4-node bilinear axisymmetric quadrilateral, hybrid, linear pressure, incompatible modes
CAX4R	Tria3 Quad4	4-node bilinear axisymmetric quadrilateral, reduced integration, hourglass control
CAX4RH	Quad4	4-node bilinear axisymmetric quadrilateral, hybrid, constant pressure, reduced integration, hourglass control
CAX4T	Quad4	4-node axisymmetric thermally coupled quadrilateral, bilinear displacement and temperature
CAX6	Tria6	6-node quadratic axisymmetric triangle
CAX6H	Tria6	6-node quadratic axisymmetric triangle, hybrid, linear pressure
CAX6M	Tria6	6-node modified axisymmetric triangle, hourglass control
CAX6MH	Tria6	6-node modified quadratic axisymmetric triangle, hybrid, linear pressure, hourglass control
CAX8	Quad8	8-node biquadratic axisymmetric quadrilateral
CAX8H	Quad8	8-node biquadratic axisymmetric quadrilateral, hybrid, linear pressure
CAX8HT	Quad8	8-node axisymmetric thermally coupled quadrilateral, biquadratic displacement, bilinear temperature, hybrid, linear pressure
CAX8R	Quad8	8-node biquadratic axisymmetric quadrilateral, reduced integration
CAX8RH	Quad8	8-node biquadratic axisymmetric quadrilateral, hybrid, linear pressure, reduced integration
CAX8RHT	Quad8	8-node axisymmetric thermally coupled quadrilateral, biquadratic displacement, bilinear

Card	Supported Element Configurations	Description
		temperature, hybrid, linear pressure, reduced integration
CAX8RT	Quad8	8-node axisymmetric thermally coupled quadrilateral, biquadratic displacement, bilinear temperature, reduced integration
CAX8T	Quad8	8-node axisymmetric thermally coupled quadrilateral, biquadratic displacement, bilinear temperature
CAXA41	Quad4	
CAXA4H1	Quad4	
CAXA4R1	Quad4	
CAXA4RH1	Quad4	
CAXA81	Quad8	
CAXA8H1	Quad8	
CAXA8P1	Quad8	
CAXA8R1	Quad8	
CAXA8R4	Quad8	
CAXA8RH1	Quad8	
CAXA8RH4	Quad8	
CAXA8RP1	Quad8	
CCL12	Hex8	12-node cylindrical brick
CGAX3	Tria3	3-node generalized linear axisymmetric triangle, twist
CGAX3H	Tria3	3-node generalized linear axisymmetric triangle, hybrid, constant pressure, twist
CGAX4	Quad4	4-node generalized bilinear axisymmetric quadrilateral, twist
CGAX4H	Quad4	4-node generalized bilinear axisymmetric quadrilateral, hybrid, constant pressure, twist

Card	Supported Element Configurations	Description
CGAX4R	Quad4	4-node generalized bilinear axisymmetric quadrilateral, reduced integration, hourglass control, twist
CGAX4RH	Quad4	4-node generalized bilinear axisymmetric quadrilateral, hybrid, constant pressure, reduced integration, hourglass control, twist
CGAX6	Tria6	6-node generalized quadratic axisymmetric triangle, twist
CGAX6H	Tria6	6-node generalized quadratic axisymmetric triangle, hybrid, linear pressure, twist
CGAX8	Quad8	8-node generalized biquadratic axisymmetric quadrilateral, twist
CGAX8H	Quad8	8-node generalized biquadratic axisymmetric quadrilateral, hybrid, linear pressure, twist
CGAX8R	Quad8	8-node generalized biquadratic axisymmetric quadrilateral, reduced integration, twist
CGAX8RH	Quad8	8-node generalized biquadratic axisymmetric quadrilateral, hybrid, linear pressure, reduced integration, twist
CGAX8T	Quad8	8-node generalized axisymmetric thermally coupled quadrilateral, biquadratic displacement, bilinear temperature, twist
COH2D4	Quad4	4-node two-dimensional cohesive element
COH3D6	Penta6	6-node three-dimensional cohesive element
COH3D8	Hex8	8-node three-dimensional cohesive element
COHAX4	Quad4	4-node axisymmetric cohesive element
CONN2D2	Mass Rod	Connector element in a plane between two nodes or ground and a node
CONN3D2	Mass Rod	Connector element in space between two nodes or ground and a node

Card	Supported Element Configurations	Description
COUP_DIS	rbe3	
COUP_KIN	Rigid	
CPE	Quad4	
CPE3	Tria3	3-node linear plane strain triangle
CPE3E	Tria3	3-node linear plane strain piezoelectric triangle
CPE3H	Tria3	3-node linear plane strain triangle, hybrid, constant pressure
CPE4E	Quad4	4-node bilinear plane strain piezoelectric quadrilateral
CPE4HT	Quad4	4-node plane strain thermally coupled quadrilateral, bilinear displacement and temperature, hybrid, constant pressure
CPE4I	Quad4	4-node bilinear plane strain quadrilateral, incompatible modes
CPE4IH	Quad4	4-node bilinear plane strain quadrilateral, hybrid, linear pressure, incompatible modes
CPE4P	Quad4	4-node plane strain quadrilateral, bilinear displacement, bilinear pore pressure
CPE4PH	Quad4	4-node plane strain quadrilateral, bilinear displacement, bilinear pore pressure, hybrid, constant pressure
CPE4R	Quad4	4-node bilinear plane strain quadrilateral, reduced integration, hourglass control
CPE4RH	Quad4	4-node bilinear plane strain quadrilateral, hybrid, constant pressure, reduced integration, hourglass control
CPE4RP	Quad4	4-node plane strain quadrilateral, bilinear displacement, bilinear pore pressure, reduced integration, hourglass control
CPE4RPH	Quad4	4-node plane strain quadrilateral, bilinear displacement, bilinear pore pressure, hybrid,

Card	Supported Element Configurations	Description
		constant pressure, reduced integration, hourglass control
CPE4T	Quad4	4-node plane strain thermally coupled quadrilateral, bilinear displacement and temperature
CPE6	Tria6	6-node quadratic plane strain triangle
CPE6H	Tria6	6-node quadratic plane strain triangle, hybrid, linear pressure
CPE6M	Tria6	6-node modified quadratic plane strain triangle, hourglass control
CPE6MH	Tria6	6-node modified quadratic plane strain triangle, hybrid, linear pressure, hourglass control
CPE6MP	Tria6	6-node modified displacement and pore pressure plane strain triangle, hourglass control
CPE6MPH	Tria6	6-node modified displacement and pore pressure plane strain triangle, hybrid, linear pressure, hourglass control
CPE8	Quad8	8-node biquadratic plane strain quadrilateral
CPE8H	Quad8	8-node biquadratic plane strain quadrilateral, hybrid, linear pressure
CPE8P	Quad8	8-node plane strain quadrilateral, biquadratic displacement, bilinear pore pressure
CPE8PH	Quad8	8-node plane strain quadrilateral, biquadratic displacement, bilinear pore pressure, hybrid, linear pressure stress
CPE8R	Quad8	8-node biquadratic plane strain quadrilateral, reduced integration
CPE8RH	Quad8	8-node biquadratic plane strain quadrilateral, hybrid, linear pressure, reduced integration
CPE8RP	Quad8	8-node plane strain quadrilateral, biquadratic displacement, bilinear pore pressure, reduced integration

Card	Supported Element Configurations	Description
CPE8RPH	Quad8	8-node biquadratic displacement, bilinear pore pressure, reduced integration, hybrid, linear pressure
CPEG3	Tria3	3-node linear generalized plane strain triangle
CPEG4R	Quad4	4-node bilinear generalized plane strain quadrilateral, reduced integration, hourglass control
CPH4H	Quad4	
CPS3	Tria3	3-node linear plane stress triangle
CPS3E		3-node linear plane stress piezoelectric triangle
CPS4	Quad4	4-node bilinear plane stress quadrilateral
CPS4E	Quad4	4-node bilinear plane stress piezoelectric quadrilateral
CPS4I	Quad4	4-node bilinear plane stress quadrilateral, incompatible modes
CPS4R	Quad4	4-node bilinear plane stress quadrilateral, reduced integration, hourglass control
CPS4T	Quad4	4-node plane stress thermally coupled quadrilateral, bilinear displacement and temperature
CPS6	Tria6	6-node quadratic plane stress triangle
CPS6M	Tria6	6-node modified second-order plane stress triangle, hourglass control
CPS8	Quad8	8-node biquadratic plane stress quadrilateral
CPS8R	Quad8	8-node biquadratic plane stress quadrilateral, reduced integration
DASHPOT1	Mass	Dashpot between a node and ground, acting in a fixed direction
DASHPOT2	Spring	Dashpot between two nodes, acting in a fixed direction

Card	Supported Element Configurations	Description
DASHPOTA	Spring	Axial dashpot between two nodes, whose line of action is the line joining the two nodes
DC1D2	Bar2	2-node heat transfer link
DC1D3	Bar3	3-node heat transfer link
DC2D3	Tria3	3-node linear heat transfer triangle
DC2D3E	Tria3	3-node linear coupled thermal-electrical triangle
DC2D4	Quad4	4-node linear heat transfer quadrilateral
DC2D4E	Quad4	4-node linear coupled thermal-electrical quadrilateral
DC2D6	Tria6	6-node quadratic heat transfer triangle
DC2D6E	Tria6	6-node quadratic coupled thermal-electrical triangle
DC2D8	Quad8	8-node quadratic heat transfer quadrilateral
DC2D8E	Quad8	8-node quadratic coupled thermal-electrical quadrilateral
DC3D10	Tetra10	10-node quadratic heat transfer tetrahedron
DC3D10E	Tetra10	10-node quadratic coupled thermal-electrical tetrahedron
DC3D15	Penta15	15-node quadratic heat transfer triangular prism
DC3D15E	Penta15	15-node quadratic coupled thermal-electrical triangular prism
DC3D20	Hex20	20-node quadratic heat transfer brick
DC3D20E	Hex20	20-node quadratic coupled thermal-electrical brick
DC3D4	Tetra4	4-node linear heat transfer tetrahedron
DC3D4E	Tetra4	4-node linear coupled thermal-electrical tetrahedron
DC3D6	Penta6	6-node linear heat transfer triangular prism

Card	Supported Element Configurations	Description
DC3D6E	Penta6	6-node linear coupled thermal-electrical triangular prism
DC3D8	Hex8	8-node linear heat transfer brick
DC3D8E	Hex8	8-node linear coupled thermal-electrical brick
DCAX3	Tria3	3-node linear axisymmetric heat transfer triangle
DCAX3E	Tria3	3-node linear axisymmetric coupled thermal-electrical triangle
DCAX4	Quad4	4-node linear axisymmetric heat transfer quadrilateral
DCAX4E	Quad4	4-node linear axisymmetric coupled thermal-electrical quadrilateral
DCAX6	Tria6	6-node quadratic axisymmetric heat transfer triangle
DCAX6E	Tria6	6-node quadratic axisymmetric coupled thermal-electrical triangle
DCAX8	Quad8	8-node quadratic axisymmetric heat transfer quadrilateral
DCAX8E	Quad8	8-node quadratic axisymmetric coupled thermal-electrical quadrilateral
DCC3D8	Hex8	8-node convection/diffusion brick
DCC3D8D	Hex8	8-node convection/diffusion brick, dispersion control
DCCAX4	Quad4	4-node axisymmetric convection/diffusion quadrilateral
DCCAX4D	Quad4	4-node axisymmetric convection/diffusion quadrilateral, dispersion control
DCOUP2D	rbe3	Two-dimensional distributing coupling element
DCOUP3D	rbe3	Three-dimensional distributing coupling element
DS3	Tria3	3-node heat transfer triangular shell
DS4	Quad4	4-node heat transfer quadrilateral shell

Card	Supported Element Configurations	Description
DS6	Tria6	6-node heat transfer triangular shell
DS8	Quad8	8-node heat transfer quadrilateral shell
EC3D8R	Hex8	
EC3D8RT	Hex8	
ELBOW31	Bar2	2-node pipe in space with deforming section, linear interpolation along the pipe
ELBOW31B	Bar2	2-node pipe in space with ovalization only, axial gradients of ovalization neglected
ELBOW31C	Bar2	2-node pipe in space with ovalization only, axial gradients of ovalization neglected
ELBOW32	Bar3	3-node pipe in space with deforming section, quadratic interpolation along the pipe
F2D2	Bar2	
F3D3	Tria3	3-node linear 3-D triangular hydrostatic fluid element
F3D4	Quad4	4-node linear 3-D quadrilateral hydrostatic fluid element
FAX2	Bar2	2-node linear axisymmetric hydrostatic fluid element
FC3D4	Tetra4	
FC3D5	Pyramid5	
FC3D6	Penta6	
FC3D8	Hex8	
GAPCYL	Gap	Cylindrical gap between two nodes
GAPSPHER	Gap	Spherical gap between two nodes
GAPUNI	Gap	Unidirectional gap between two nodes
GK2D2	Rod	2-node two-dimensional gasket element

Card	Supported Element Configurations	Description
GK2D2N	Rod	2-node two-dimensional gasket element with thickness-direction behavior only
GK3D12M	Penta6	12-node three-dimensional gasket element
GK3D12MN	Penta6	12-node three-dimensional gasket element with thickness-direction behavior only
GK3D18	Hex8	18-node three-dimensional gasket element
GK3D18N	Hex8	18-node three-dimensional gasket element with thickness-direction behavior only
GK3D2	Bar2	2-node three-dimensional gasket element
GK3D2N	Bar2	2-node three-dimensional gasket element with thickness-direction behavior only
GK3D4L	Quad4	4-node three-dimensional line gasket element
GK3D4LN	Quad4	4-node three-dimensional line gasket element with thickness-direction behavior only
GK3D6	Penta6	6-node three-dimensional gasket element
GK3D6N	Penta6	6-node three-dimensional gasket element with thickness-direction behavior only
GK3D8	Hex8	8-node three-dimensional gasket element
GK3D8N	Hex8	8-node three-dimensional gasket element with thickness-direction behavior only
GKPE4	Quad4	4-node plane strain gasket element
GKPS4	Quad4	4-node plane stress gasket element
GKPS4N	Quad4	4-node two-dimensional gasket element with thickness-direction behavior only
HMCONN	Mass	
ITS		
ITSCYL	Rod	Cylindrical geometry tube support interaction element
ITSUNI	Rod	Unidirectional tube support interaction element

Card	Supported Element Configurations	Description
ITT21	Mass	Tube-tube element for use with first-order, 2-D beam and pipe elements
ITT31	Mass	Tube-tube element for use with first-order, 3-D beam and pipe elements
JOINTC	Spring	Three-dimensional joint interaction element
KINCOUP	Rigid	
LINK	Rigid	
M3D3	Tria3	3-node triangular membrane
M3D4	Quad4	4-node quadrilateral membrane
M3D4R	Quad4	4-node quadrilateral membrane, reduced integration, hourglass control
M3D6	Tria6	6-node triangular membrane
M3D8	Quad8	8-node quadrilateral membrane
M3D8R	Quad8	8-node quadrilateral membrane, reduced integration
MASS	Mass	Point mass
MGAX1	Rod	2-node linear axisymmetric membrane, twist
MGAX2	Bar3	3-node quadratic axisymmetric membrane, twist
PC3D	Mass	
PIN	Rigid	
PIPE21	Bar2	2-node linear pipe in a plane
PIPE21H	Bar2	2-node linear pipe in a plane, hybrid formulation
PIPE22	Bar3	3-node quadratic pipe in a plane
PIPE22H	Bar3	3-node quadratic pipe in a plane, hybrid formulation
PIPE31	Bar2	2-node linear pipe
PIPE31H	Bar2	2-node linear pipe in space, hybrid formulation

Card	Supported Element Configurations	Description
PIPE32	Bar3	3-node quadratic pipe in space
PIPE32H	Bar3	3-node quadratic pipe in space, hybrid formulation
R2D2	Rigid	
R3D3	Tria3	3-node 3-D rigid triangular facet
R3D4	Quad4	4-node 3-D bilinear rigid quadrilateral
RAX2	Rigid	
RB2D2	Rigid	
RB3D2	Rigid	2-node 3-D rigid beam
RDE	Rigid rbe3	2-node linear axisymmetric rigid link (for use in axisymmetric planar geometries)
ROTARYI	Mass	Rotary inertia at a point
S3	Tria3	3-node triangular general-purpose shell, finite membrane strains
S3R	Tria3	3-node triangular general-purpose shell, finite membrane strains
S3RS	Tria3	
S3RT	Tria3	3-node thermally coupled triangular general-purpose shell, finite membrane strains
S4	Quad4	4-node general-purpose shell, finite membrane strains
S4R	Quad4	4-node general-purpose shell, reduced integration, hourglass control, finite membrane strains
S4R5	Quad4	4-node thin shell, reduced integration, hourglass control, using five degrees of freedom per node
S4RS	Quad4	
S4RSW	Quad4	

Card	Supported Element Configurations	Description
S4RT	Quad4	4-node thermally coupled general-purpose shell, reduced integration, hourglass control, finite membrane strains
S4T	Quad4	4-node thermally coupled general-purpose shell, finite membrane strains
S8R	Quad8	8-node doubly curved thick shell, reduced integration
S8R5	Quad8	8-node doubly curved thin shell, reduced integration, using five degrees of freedom per node
S8RT	Quad8	8-node thermally coupled quadrilateral general thick shell, biquadratic displacement, bilinear temperature in the shell surface
SAX1	Bar2	2-node linear axisymmetric thin or thick shell
SAX2	Bar3	3-node quadratic axisymmetric thin or thick shell
SC6R	Penta6	6-node triangular in-plane continuum shell wedge, general-purpose continuum shell, finite membrane strains
SC6RT	Penta6	6-node linear displacement and temperature, triangular in-plane continuum shell wedge, general-purpose continuum shell, finite membrane strains
SC8R	Hex8	8-node quadrilateral in-plane general-purpose continuum shell, reduced integration with hourglass control, finite membrane strains
SC8RT	Hex8	8-node linear displacement and temperature, quadrilateral in-plane general-purpose continuum shell, reduced integration with hourglass control, finite membrane strains
SFM3D3	Tria3	3-node triangular surface element
SFM3D4	Quad4	4-node quadrilateral surface element
SFM3D4R	Quad4	4-node quadrilateral surface element, reduced integration

Card	Supported Element Configurations	Description
SFM3D6	Tria6	6-node triangular surface element
SFM3D8	Quad8	8-node quadrilateral surface element
SFM3D8R	Quad8	8-node quadrilateral surface element, reduced integration
SFMAX1	Rod	2-node linear axisymmetric surface element
SFMAX2	Bar3	3-node quadratic axisymmetric surface element
SFMGAX1	Rod	2-node linear axisymmetric surface element, twist
SFMGAX2	Bar3	3-node quadratic axisymmetric surface element, twist
SPRING1	Mass	Spring between a node and ground, acting in a fixed direction
SPRING2	Spring	Spring between two nodes, acting in a fixed direction
SPRINGA	Spring	Axial spring between two nodes, whose line of action is the line joining the two nodes
STRI3	Tria3	3-node triangular facet thin shell
STRI65	Tria6	6-node triangular thin shell, using five degrees of freedom per node
T2D2	Rod	2-node linear 2-D truss
T2D2E	Rod	2-node 2-D piezoelectric truss
T2D2H	Rod	2-node linear 2-D truss, hybrid
T2D2T	Rod	2-node 2-D thermally coupled truss
T2D3	Rod	3-node quadratic 2-D truss
T3D2	Rod	2-node linear 3D truss
T3D2E	Rod	2-node 3D piezoelectric truss
T3D2H	Rod	2-node linear 3D truss, hybrid
T3D2T	Rod	2-node 3D thermally coupled truss

Card	Supported Element Configurations	Description
TIE	Rigid	

ANSYS Cards

Card	Supported Element Configurations	Description
BEAM3	Bar2	2D Elastic Beam Config 60, Type 2
BEAM4	Bar2	3D Elastic Beam Config 60, Type 1
BEAM23	Bar2	2D Plastic Beam Config 60, Type 9
BEAM24	Bar2	3D Thin-walled Beam Config 60, Type 6
BEAM44	Bar2	3D Elastic Tapered Unsymmetric Beam Config 60, Type 7
BEAM54	Bar2	2D Elastic Tapered Unsymmetric Beam Config 60, Type 10
BEAM188	Bar2	3D Linear Finite Strain Beam Config 60, Type 8
BEAM189	Bar3	3D Quadratic Finite Strain Beam Config 63, Type 1
CERIG	Rigid	Ldof, Ldof2, Ldof3, Ldof4, Ldof5 Config 5, Type 1, 2
CIRCU124	Rod	General circuit element applicable to circuit simulation.
COMBIN14	Spring	Spring-Damper

Card	Supported Element Configurations	Description
		Config 21, Type 1
COMBIN39	Spring	Nonlinear Spring Config 21, Type 2
COMBIN40	Spring	Combination Config 21, Type 3
CONTA171	Plot	2D 2-Node Surface-to-Surface Contact Config 2, Type 3
CONTA172	Bar3	2D 3-Node Surface-to-Surface Contact Config 63, Type 12
CONTA173	Tria3 Quad4	3D 4-Node Surface-to-Surface Contact Config 103, Type 13 Config 104, Type 13
CONTA174	Tria6 Quad 8	3D 8-Node Surface-to-Surface Contact Config 106, Type 10 Config 108, Type 10
CONTA175	Mass	2D/3D Node-to-Surface Contact Config 1, Type 14
CONTA177	Bar2 Bar3	3D Line-to-Surface Contact
CONTA178	Gap	3D Node-to-Node Contact Config 70, Type 3
CONTAC12	Gap	2D Point-to-Point Contact Config 70, Type 2
CONTAC48	Tria3	
CONTAC49	Tria3 Quad4	

Card	Supported Element Configurations	Description
CONTAC52	Gap	3D Point-to-Point Contact Config 70, Type 1
CP	Rigid	Defines (or modifies) a set of coupled degrees of freedom. Config 55, Type 1, 2
CP_ELEC	Rigid	
CP_STRUC	Rigid	
CP_THERM	Rigid	
ELBOW290	Bar3	
FLUID29	Tria3 Quad4	2D Axisymmetric Harmonic Acoustic Fluid
FLUID30	Tetra4 Penta6 Hex8	3D Acoustic Fluid
FLUID80	Hex8	3D Contained Fluid Config 208, Type 9
FLUID116	Rod	Coupled Thermal-Fluid Pipe Config 61, Type 12
FLUID220	Hex20 Penta15 Pyramid13 Tetra10	
FLUID221	Tetra10	
HF118	Tria6 Quad8	2D High-Frequency Quadrilateral Solid Config 106, Type 25 Config 108, Type 25

Card	Supported Element Configurations	Description
HF119	Tetra10	3D High-Frequency Tetrahedral Solid Config 210, Type 11
HF120	Pyramid13 Penta15 Hex20	3D High-Frequency Brick Solid Config 213, Type 3 Config 215, Type 3 Config 220, Type 3
HYPER58	Tetra4 Penta6 Hex8	3D 8-Node Mixed u-P Hyperelastic Solid Config 204, Type 11 Config 206, Type 11 Config 208, Type 11
INFIN9	Rod	
INFIN110	Quad4 Quad8	
INTER192	Quad4 Tria3	
INTER193	Quad8	
INTER194	Hex20 Penta15	
INTER195	Hex8 Penta6	
INTER205	Hex8 Penta6	
LINK1	Rod	2D Spar (or Truss) Config 61, Type 5
LINK8	Rod	3D Spar (or Truss) Config 61, Type 1

Card	Supported Element Configurations	Description
LINK10	Rod	Tension-only or Compression-only Spar Config 61, Type 2
LINK11	Bar2	Linear actuator Config 61, Type 25
LINK31	Rod	Radiation Link Config 61, Type 6
LINK32	Rod	2D Conduction Bar Config 61, Type 7
LINK33	Rod	3D Conduction Bar Config 61, Type 8
LINK34	Rod	Convection Link Config 61, Type 9
LINK68	Rod	Coupled Thermal-Electric Line Config 61, Type 14
LINK180	Rod	3D Finite Strain Spar (or Truss) Config 61, Type 11
MASS21	Mass	Structural Mass Config 1, Type 1
MASS71	Mass	Thermal Mass Config 1, Type 2
MESH200	Bar3 Rod Tria3 Tetra4 Tetra10 Quad4 Hex8	Meshing Facet Config 63, Type 26 Config 61, Type 26 Config 103, Type 26 Config 204, Type 26 Config 210, Type 26 Config 104, Type 26 Config 208, Type 26

Card	Supported Element Configurations	Description
	Tria6 Quad8 Hex20	Config 106, Type 26 Config 108, Type 26 Config 220, Type 26
MPC184	Rod	Multipoint Constraint Elements: Rigid Link, Rigid Beam, Slider, Spherical, Revolute, Universal Config 61, Type 13
PIPE16	Bar2	Elastic Straight Pipe Config 60, Type 3
PIPE18	Bar2	Elastic Curved Pipe (Elbow) Config 60, Type 4
PIPE20	Rod	Plastic Straight Pipe Config 61, Type 4
PIPE60	Bar2	Plastic Curved Pipe (Elbow) Config 60, Type 5
PIPE288	Bar2	
PIPE289	Bar3	
PLANE2	Tria6	2D 6-Node Triangular Structural Solid Config 106, Type 21
PLANE13	Quad4 Tria3	2D Coupled-Field Solid Config 103, Type 3 Config 104, Type 3
PLANE25	Tria3 Quad4	Axisymmetric-Harmonic 4-Node Structural Solid Config 103, Type 6 Config 104, Type 6
PLANE35	Tria6	2D 6-Node Triangular Thermal Solid Config 106, Type 7
PLANE42	Tria3	2D Structural Solid

Card	Supported Element Configurations	Description
	Quad4	Config 103, Type 4 Config 104, Type 4
PLANE53	Tria6 Quad8	2D 8-Node Magnetic Solid Config 106, Type 9 Config 108, Type 9
PLANE55	Tria3 Quad4	2D Thermal Solid Config 103, Type 5 Config 104, Type 5
PLANE67	Tria3 Quad4	2D Coupled Thermal-Electric Solid Config 103, Type 21 Config 104, Type 21
PLANE75	Tria3 Quad4	Axisymmetric-Harmonic 4-Node Thermal Solid Config 103, Type 9 Config 104, Type 9
PLANE77	Tria6 Quad8	2D 8-Node Thermal Solid Config 106, Type 2 Config 108, Type 2
PLANE78	Tria6 Quad8	Axisymmetric-Harmonic 8-Node Thermal Solid Config 106, Type 8 Config 108, Type 8
PLANE82	Tria6 Quad8	2D 8-Node Structural Solid Config 106, Type 1 Config 108, Type 1
PLANE83	Tria6 Quad8	Axisymmetric-Harmonic 8-Node Structural Solid Config 106, Type 3 Config 108, Type 3
PLANE121	Tria6 Quad8	2D 8-Node Electrostatic Solid Config 106, Type 22

Card	Supported Element Configurations	Description
		Config 108, Type 22
PLANE145	Tria6 Quad8	2D Quadrilateral Structural Solid p-Element Config 106, Type 23 Config 108, Type 23
PLANE146	Tria6	Config 106, Type 31
PLANE162	Tria3 Quad4	Explicit 2D Structural Solid Config 103, Type 22 Config 104, Type 22
PLANE182	Tria3 Quad4	2D 4-Node Structural Solid Config 103, Type 23 Config 104, Type 23
PLANE183	Tria6 Quad8	2D 8-Node Structural Solid Config 106, Type 19 Config 108, Type 19
PLANE223	Tria6 Quad8	2D 8-Node Coupled-Field Solid
PRETS179	Bar2	Define a 2D or 3D pretension section within a meshed structure. Config 60, Type 17
RBE3	RBE3	Distributes the force/moment applied at the master node to a set of slave nodes, taking into account the geometry of the slave nodes as well as weighting factors.
SHELL28	Quad4	Shear/Twist Panel Config 104, Type 12
SHELL41	Tria3 Quad4	Membrane Shell Config 103, Type 19 Config 104, Type 19

Card	Supported Element Configurations	Description
SHELL43	Tria3 Quad4	4-Node Plastic Large Strain Shell Config 103, Type 2 Config 104, Type 2
SHELL51	Bar2	Axisymmetric Structural Shell Config 60, Type 14
SHELL57	Tria3 Quad4	Thermal Shell Config 103, Type 7 Config 104, Type 7
SHELL61	Bar2	Axisymmetric-Harmonic Structural Shell Config 60, Type 15
SHELL63	Tria3 Quad4	Elastic Shell Config 103, Type 1 Config 104, Type 1
SHELL91	Tria6 Quad8	Nonlinear Layered Structural Shell Config 106, Type 6 Config 108, Type 6
SHELL93	Tria6 Quad8	8-Node Structural Shell Config 106, Type 4 Config 108, Type 4
SHELL99	Tria6 Quad8	Linear Layered Structural Shell Config 106, Type 5 Config 108, Type 5
SHELL131	Tria3 Quad4	4-Node Layered Thermal Shell Config 103, Type 25 Config 104, Type 25
SHELL132	Tria6 Quad8	8-Node Layered Thermal Shell Config 106, Type 24 Config 108, Type 24

Card	Supported Element Configurations	Description
SHELL143	Tria3 Quad4	4-Node Plastic Small Strain Shell Config 103, Type 10 Config 104, Type 10
SHELL150	Tria6 Quad8	8-Node Structural Shell p-Element Config 106, Type 20 Config 108, Type 20
SHELL157	Tria3 Quad4	Thermal-Electric Shell Config 103, Type 20 Config 104, Type 20
SHELL163	Tria3 Quad4	Explicit Thin Structural Shell Config 103, Type 17 Config 104, Type 17
SHELL181	Tria3 Quad4	4-Node Finite Strain Shell Config 103, Type 11 Config 104, Type 11
SHELL208	Bar2	2-Node Finite Strain Axisymmetric Shell
SHELL209	Bar2	3-Node Finite Strain Axisymmetric Shell
SHELL281	Tria6 Quad8	8-Node Finite Strain Shell
SOLID5	Penta6 Hex8	3D Coupled-Field Solid Config 206, Type 2 Config 208, Type 2
SOLID45	Tetra4 Penta6 Hex8	3D Structural Solid Config 204, Type 1 Config 206, Type 1 Config 208, Type 1
SOLID46	Tetra4 Penta6	3D 8-Node Layered Structural Solid Config 204, Type 6

Card	Supported Element Configurations	Description
	Hex8	Config 206, Type 6 Config 208, Type 6
SOLID62	Tetra4 Pyramid5 Penta6 Hex8	3D Magneto-Structural Solid Config 204, Type 15 Config 205, Type 15 Config 206, Type 15 Config 208, Type 15
SOLID64	Tetra4 Penta6 Hex8	3D Anisotropic Structural Solid Config 204, Type 7 Config 206, Type 7 Config 208, Type 7
SOLID69	Tetra4 Penta6 Hex8	3D Coupled Thermal-Electric Solid Config 204, Type 4 Config 206, Type 4 Config 208, Type 4
SOLID70	Tetra4 Penta6 Hex8	3D Thermal Solid Config 204, Type 3 Config 206, Type 3 Config 208, Type 3
SOLID72	Tetra4	
SOLID73	Tetra4 Penta6 Hex8	
SOLID87	Tetra10	3D 10-Node Tetrahedral Thermal Solid Config 210, Type 5
SOLID90	Tetra10 Pyramid13 Penta15 Hex20	3D 20-Node Thermal Solid Config 210, Type 2 Config 213, Type 2

Card	Supported Element Configurations	Description
SOLID92	Tetra10	3D 10-Node Tetrahedral Structural Solid Config 210, Type 3
SOLID95	Tetra10 Pyramid13 Penta15 Hex20	3D 20-Node Structural Solid Config 210, Type 1 Config 213, Type 1 Config 215, Type 1 Config 220, Type 1
SOLID96	Tetra4 Pyramid5 Penta6 Hex8	3D Magnetic Scalar Solid Config 204, Type 5 Config 205, Type 5 Config 206, Type 5 Config 208, Type 5
SOLID97	Tetra4 Pyramid5 Penta6 Hex8	3D Magnetic Solid Config 204, Type 8 Config 205, Type 8 Config 206, Type 8 Config 208, Type 8
SOLID98	Tetra10	Tetrahedral Coupled-Field Solid Config 210, Type 4
SOLID117	Tetra10 Pyramid13 Penta15 Hex20	3D 20-Node Magnetic Solid Config 210, Type 8 Config 213, Type 8 Config 215, Type 8 Config 220, Type 8
SOLID147	Penta15 Hex20	3D Brick Structural Solid p-Element
SOLID148	Penta15 Tetra10 Hex20	3D Tetrahedral Structural Solid p-Element Config 215, Type 9 Config 210, Type 9

Card	Supported Element Configurations	Description
		Config 220, Type 9
SOLID164	Tetra4 Pyramid5 Penta5 Hex8	Explicit 3D Structural Solid Config 204, Type 14 Config 205, Type 14 Config 206, Type 14 Config 208, Type 14
SOLID168	Tetra10	Explicit 3D 10-Node Tetrahedral Structural Solid
SOLID185	Tetra4 Penta6 Hex8	3D 8-Node Structural Solid Config 204, Type 13 Config 206, Type 13 Config 208, Type 13
SOLID186	Tetra10 Pyramid13 Penta15 Hex20	3D 20-Node Structural Solid Config 210, Type 7 Config 213, Type 7 Config 215, Type 7 Config 220, Type 7
SOLID187	Tetra10	3D 10-Node Tetrahedral Structural Solid Config 210, Type 6
SOLID191	Tetra10 Penta15 Hex20	3D 20-Node Layered Structural Solid Config 210, Type 10 Config 215, Type 10 Config 220, Type 10
SOLID226	Tetra10 Pyramid13 Penta15 Hex20	3D 20-Node Coupled-Field Solid
SOLID227	Tetra10	3D 10-Node Coupled-Field Solid
SOLID278	Hex8 Penta6	

Card	Supported Element Configurations	Description
	Tetra4	
SOLID279	Hex20 Penta15 Pyramid13 Tetra10	
SOLID285	Tetra4	
SOLSH190	Penta6 Hex8	3D 8-Node Layered Solid Shell Config 206 Config 208, Type 17
SURF151	Bar2	2D Thermal Surface Effect Config 60, Type 12
SURF152	Quad4 Quad8 Tria6	3D Thermal Surface Effect Config 104, Type 14 Config 108, Type 14
SURF153	Bar2 Bar3	2D Structural Surface Effect Config 60, Type 16 Config 63, Type 16
SURF154	Quad4 Quad8 Tria6	3D Structural Surface Effect Config 104, Type 18 Config 108, Type 18
SURF156	Bar3	3D Structural Surface Line Load Effect
SURF251	Rod	2D Radiosity Surface Config 61, Type 25
SURF252	Tria3 Quad4	3D Radiosity Surface Config 103, Type 36 Config 104, Type 36
TARGE169	Mass	2D Target Segment

Card	Supported Element Configurations	Description
	Bar2 Bar3	Config 1, Type 13 Config 60, Type 9 Config 63, Type 16
TARGE170	Mass Tria3 Quad4 Tria6 Quad8	3D Target Segment Config 103, Type 16 Config 104, Type 16 Config 106, Type 16 Config 108, Type 16
VISCO88	Tria6 Quad8	2D 8-Node Viscoelastic Solid
VISCO107	Tetra4 Penta6 Hex8	3D 8-Node Viscoplastic Solid Config 204, Type 16 Config 206, Type 16 Config 208, Type 16

EXODUS Cards

Card	Supported Element Configurations	Description
BEAM	bar2	
BEAM2	bar2	
BEAM3	bar3	
CIRCLE	mass	
HEX20	hex20	
HEX8	hex8	
PYRAMID13	pyramid13	
PYRAMID5	pyramid5	
QUAD4	quad4	

Card	Supported Element Configurations	Description
QUAD8	quad8	
RBAR	weld	
RJOINT	rigid	
RROD	rigid	
SHELL3	tria3	
SHELL4	quad4	
SHELL6	tria6	
SHELL8	quad8	
SPHERE	mass	
SPRING	spring	
TETRA10	tetra10	
TETRA4	tetra4	
TRI3	tria3	
TRI6	tria6	
TRIANGLE3	tria3	
TRIANGLE6	tria6	
TRUSS	rod	
WEDGE15	penta15	
WEDGE6	penta6	

LS-DYNA Cards

Card	Supported Element Configurations	Description
*CONSTRAINED_GENERA	Rigid	Define butt welds. Spot(default)/type 1, Fillet/type 2, and Butt/type 3 failure modes are supported. Failure information

Card	Supported Element Configurations	Description
		is based on weld type selected. Coordinate System ID can be selected. No Failure/Type 0 Card 36 entities are defined as *CONSTRAINED_NODAL_RIGID_BODIES in Keyword. They are a separate element type.
*CONSTRAINED_GENERA	Rigid	Define combined welds.
*CONSTRAINED_GENERA	Rigid	Define cross fillet welds.
*CONSTRAINED_GENERA	Rigid	Define fillet welds.
*CONSTRAINED_GENERA	Rigid	Define spot welds.
*CONSTRAINED_INTERPO	RBE3	Define an interpolation constrain.
*CONSTRAINED_JOINT_C	Joint	
*CONSTRAINED_JOINT_C	Joint	Define a joint between two rigid bodies.
*CONSTRAINED_JOINT_C	Joint	
*CONSTRAINED_JOINT_C	Joint	
*CONSTRAINED_JOINT_C	Joint	
*CONSTRAINED_JOINT_C	Joint	
*CONSTRAINED_JOINT_L	Joint	
*CONSTRAINED_JOINT_L	Joint	
*CONSTRAINED_JOINT_L	Joint	
*CONSTRAINED_JOINT_L	Joint	
*CONSTRAINED_JOINT_P	Joint	
*CONSTRAINED_JOINT_P	Joint	
*CONSTRAINED_JOINT_P	Joint	
*CONSTRAINED_JOINT_P	Joint	
*CONSTRAINED_JOINT_P	Joint	
*CONSTRAINED_JOINT_R	Joint	

Card	Supported Element Configurations	Description
*CONSTRAINED_JOINT_R	Joint	
*CONSTRAINED_JOINT_R	Joint	
*CONSTRAINED_JOINT_R	Joint	
*CONSTRAINED_JOINT_R	Joint	
*CONSTRAINED_JOINT_R	Joint	
*CONSTRAINED_JOINT_S		
*CONSTRAINED_JOINT_S	Joint	
*CONSTRAINED_JOINT_S	Joint	
*CONSTRAINED_JOINT_S	Joint	
*CONSTRAINED_JOINT_S	Joint	
*CONSTRAINED_JOINT_S	Joint	Define optional rotational and translational joint stiffness for joints defined by *CONSTRAINED_JOINT_OPTION.
*CONSTRAINED_JOINT_T	Joint	
*CONSTRAINED_JOINT_T	Joint	
*CONSTRAINED_JOINT_T	Joint	
*CONSTRAINED_JOINT_T	Joint	
*CONSTRAINED_JOINT_T	Joint	
*CONSTRAINED_JOINT_U	Joint	
*CONSTRAINED_JOINT_U	Joint	
*CONSTRAINED_JOINT_U	Joint	
*CONSTRAINED_JOINT_U	Joint	
*CONSTRAINED_NODAL_	Rigid	Define a nodal rigid body.
*CONSTRAINED_NODAL_ (2-Noded)	Rigid	
*CONSTRAINED_NODAL_	Rigid	Used when inertial properties are defined rather than computed.

Card	Supported Element Configurations	Description
*CONSTRAINED_NODAL_ (2-Noded)	Rigid	
*CONSTRAINED_NODAL_ _SPC	Rigid	
*CONSTRAINED_NODAL_ _SPC (2-Noded)	Rigid	
*CONSTRAINED_NODAL_	Rigid	
*CONSTRAINED_NODAL_ (2-Noded)	Rigid	
*CONSTRAINED_NODE_S	Rigid	Define nodal constraint sets for translational motion in global coordinates.
*CONSTRAINED_NODE_S (2-Noded)	Rigid	
*CONSTRAINED_NODE_S	Rigid	
*CONSTRAINED_RIVET	Weld	Define massless rivets between non-contiguous nodal pairs.
*CONSTRAINED_SHELL_T	Rigid	Define a tie between a shell edge and solid elements.
*CONSTRAINED_SPOTWE	Weld	Define massless spot welds between non-contiguous nodal pairs. Normal and shear failure values can be edited.
*CONSTRAINED_SPOTWE	Weld	
*ELEMENT_BEAM	Bar	Define two node elements including 3D beams, trusses, 2D axisymmetric shells and 2D plane strain beam elements. Thickness option can be added. This allows you to edit the parameters based on the element formulation in the property to which the beam points.
*ELEMENT_BEAM_OFFSET	Bar	
*ELEMENT_BEAM_OFFSET	Bar	

Card	Supported Element Configurations	Description
*ELEMENT_BEAM_OFFSET	Bar	
*ELEMENT_BEAM_ORIENT	Bar	
*ELEMENT_BEAM_PID	Bar	
*ELEMENT_BEAM_PID_OI	Bar	
*ELEMENT_BEAM_PID_SC	Bar	
*ELEMENT_BEAM_SCALAI	Bar	
*ELEMENT_BEAM_SCALAI	Bar	
*ELEMENT_BEAM_SECTIC	Bar	
*ELEMENT_BEAM_SECTIC	Bar	
*ELEMENT_BEAM_SECTIC	Bar	
*ELEMENT_BEAM_THICKI	Bar	
*ELEMENT_BEAM_THICKI	Bar	
*ELEMENT_BEAM_THICKI	Bar	
*ELEMENT_BEAM_THICKI	Bar	
*ELEMENT_DISCRETE	Spring	Define a discrete (spring or damper) element between two nodes or a node and ground. Scale factor, printing flags, and offset values can be edited.
*ELEMENT_INERTIA	Mass	Define a lumped inertia element assigned to a nodal point.
*ELEMENT_INERTIA_OFFS	Mass	
*ELEMENT_MASS	Mass	Define a lumped mass element assigned to a nodal point or equally distributed to the nodes of a node set.
*ELEMENT_MASS_NODE_	Mass	Mass elements defined on node set
*ELEMENT_MASS_PART	Mass	Define additional non-structural mass to be distributed by an area weighted distribution to all nodes of a given part ID.

Card	Supported Element Configurations	Description
*ELEMENT_MASS_PART_S	Mass	Mass elements defined on part set.
*ELEMENT_PLOTEL	Plot	Define a null beam element for visualization.
*ELEMENT_SEATBELT	Rod	Define a seat belt element.
*ELEMENT_SEATBELT_SE	Sensors	Define seat belt sensor.
*ELEMENT_SHELL	Tria3 Quad4	Define three, four, six and eight node elements including 3D shells, membranes, 2D plane stress, plane strain, and axisymmetric solids. Thickness and beta options can be added singularly or together. This allows you to edit the thickness and material angles to override the SECTION card.
*ELEMENT_SHELL_BETA	Tria3 Quad4	
*ELEMENT_SHELL_BETA_	Tria3 Quad4	
*ELEMENT_SHELL_COMP	Tria3 Quad4	
*ELEMENT_SHELL_DOF	Tria3 Quad4	
*ELEMENT_SHELL_MCID	Tria3 Quad4	
*ELEMENT_SHELL_MCID_	Tria3 Quad4	
*ELEMENT_SHELL_OFFSE	Tria3 Quad4	
*ELEMENT_SHELL_THICK	Tria3	

Card	Supported Element Configurations	Description
	Quad4	
*ELEMENT_SHELL_THICK	Tria3 Quad4	
*ELEMENT_SHELL_THICK	Tria3 Quad4	
*ELEMENT_SHELL_THICK	Tria3 Quad4	
*ELEMENT_SHELL_THICK	Tria3 Quad4	
*ELEMENT_SHELL_THICK	Tria3 Quad4	
*ELEMENT_SOLID	Tetra4 Penta6 Hex8 Tetra10	Define three-dimensional solid elements including 4 noded tetrahedrons and 8-noded hexahedrons.
*ELEMENT_SOLID_ORTH(	Tetra4 Penta6 Hex8 Tetra10	Define a local coordinate system for orthotropic and anisotropic materials.
*ELEMENT_SOLID_TET4T	Tetra4	Converts 4 node tetrahedron solids to 10 node quadratic tetrahedron solids.
*ELEMENT_SPH	Mass	Define a lumped mass element assigned to a nodal point
*ELEMENT_TSHELL	Penta6 Hex8	Define an eight node thick shell element which is available with either fully reduced or selectively reduced integration rules.
*INITIAL_MOMENTUM	Tetra4 Penta6	Defines initial momentum in the solid element at the start of analysis. This momentum could be

Card	Supported Element Configurations	Description
	Hex8 Tetra10	from previous analysis/step carried forward to next analysis/step. This is supported as an attribute to an element to maintain its associativity with element inside HM.
*INITIAL_STRAIN_SHELL	Tria3 Quad4	Defines stress in the shell element at the start of analysis. This stress could be from previous analysis/step carried forward to next analysis/step. This is supported as an attribute to an element to maintain its associativity with element inside HM.
*INITIAL_STRAIN_SOLID	Tetra4 Penta6 Hex8 Tetra10	Defines stress in the solid element at the start of analysis. This stress could be from previous analysis/step carried forward to next analysis/step. This is supported as an attribute to an element to maintain its associativity with element inside HM.
*INITIAL_STRESS_BEAM	Bar	Defines stress in the beam element at the start of analysis. This stress could be from previous analysis/step carried forward to next analysis/step. This is supported as an attribute to an element to maintain its associativity with element inside HM.
*INITIAL_STRESS_SHELL	Tria3 Quad4	Defines stress in the shell element at the start of analysis. This stress could be from previous analysis/step carried forward to next analysis/step. This is supported as an attribute to an element to maintain its associativity with element inside HM.
*INITIAL_STRESS_SOLID	Tetra4 Penta6 Hex8 Tetra10	Defines stress in the solid element at the start of analysis. This stress could be from previous analysis/step carried forward to next analysis/step.

Card	Supported Element Configurations	Description
		This is supported as an attribute to an element to maintain its associativity with element inside HM.

Nastran Cards

Card	Supported Element Configurations	Description
CAABSF	Mass Rod Tria3 Quad4	Defines a frequency-dependent acoustic absorber element in coupled fluid-structural analysis.
CACINF3	Tria3	Defines an acoustic conjugate infinite element with triangular base.
CACINF4	Quad4	Defines an acoustic conjugate infinite element with quadrilateral base.
CAERO1	Quad4	Defines an aerodynamic macro element (panel) in terms of two leading edge locations and side chords. This is used for Doublet-Lattice theory for subsonic aerodynamics and the ZONA51 theory for supersonic aerodynamics.
CAERO2	Rod	Defines aerodynamic slender body and interference elements for Doublet-Lattice aerodynamics.
CBAR	Bar	Defines a simple beam element.
CBEAM	Bar	Defines a beam element.
CBEND	Bar	Defines a curved beam, curved pipe, or elbow element.
CBUSH	Mass Spring	Defines a generalized spring-and-damper structural element that may be nonlinear or frequency dependent. Both elements with grounded terminals are supported.

Card	Supported Element Configurations	Description
CBUSH1D	Mass Spring	Defines the connectivity of a one-dimensional spring and viscous damper element. Both elements with grounded terminals are supported.
CDAMP1	Mass Spring	Defines a scalar damper element. Elements CDAMP1 and CDAMP2 with grounded terminals are not supported.
CDAMP2	Mass Spring	Defines a scalar damper element without reference to a material or property entry. Elements CDAMP1 and CDAMP2 with grounded terminals are not supported.
CDAMP3	Mass Spring	Defines a scalar damper element that is connected only to scalar points. Elements CDAMP1 and CDAMP2 with grounded terminals are not supported.
CDAMP4	Mass Spring	Defines a scalar damper element that connected only to scalar points and without reference to a material or property entry. Elements CDAMP1 and CDAMP2 with grounded terminals are not supported.
CELAS1	Mass Spring	Defines a scalar spring element. Elements CDAMP1 and CDAMP2 with grounded terminals are not supported.
CELAS2	Mass Spring	Defines a scalar spring element without reference to a property entry. Elements CDAMP1 and CDAMP2 with grounded terminals are not supported.
CELAS3	Mass Spring	Defines a scalar spring element that connects only to scalar points. Elements CDAMP1 and CDAMP2 with grounded terminals are not supported.

Card	Supported Element Configurations	Description
CELAS4	Mass Spring	Defines a scalar spring element that is connected only to scalar points, without reference to a property entry. Elements CDAMP1 and CDAMP2 with grounded terminals are not supported.
CFAST	Mass Rod	Defines a fastener with material orientation connecting two surface patches.
CGAP	Gap	Defines a gap or friction element.
CHACAB	Hex8	Defines the acoustic absorber element in coupled fluid-structural analysis.
CHBDYE	Slave3 Slave4	Defines a boundary condition surface element with reference to a heat conduction element. This element is supported as GROUP.
CHBDYG	Slave3 Slave4	Defines a boundary condition surface element without reference to a property entry. This element is supported as GROUP. Creation from scratch is not yet supported.
CHBDYP	Mass Rod	Defines a boundary condition surface element with reference to a PHBDY entry. CONVM is supported as a continuation card inside CHBDYP.
CHEXA (20-noded)	Hex20	Defines a second order solid element, composed of 6 quadrilateral faces. In Nastran, you can define a second order element with missing mid-side nodes. Input data decks containing such elements are read by the translator as a first-order element. A message is written to the nastran.msg file indicating the corresponding element ID.
CHEXA (8-noded)	Hex8	Defines a first order solid element, composed of 6 quadrilateral faces.
CMASS1	Mass	Defines a scalar mass element.

Card	Supported Element Configurations	Description
	Spring	
CMASS2	Mass Spring	Defines a scalar mass element without reference to a property entry.
CMASS3	Mass Spring	Defines a scalar mass element that is connected only to scalar points.
CMASS4	Mass Spring	Defines a scalar mass element that is connected only to scalar points, without reference to a property entry.
CONM1	Mass	Defines a 6 x 6 symmetric mass matrix at a geometric grid point.
CONM2	Mass	Defines a concentrated mass at a grid point.
CONROD	Rod	Defines a rod element without reference to a property entry.
CPENTA (6-noded)	Penta6	Defines the connections of a five-sided solid element with six to fifteen grid points.
CPENTA (15-noded)	Penta15	Defines the connections of a five-sided solid element with six to fifteen grid points. In Nastran, you can define a second order element with missing mid-side nodes. Input data decks containing such elements are read by the translator as a first-order element. A message is written to the `nastran.msg` file indicating the corresponding element ID.
CQUAD4	Quad4	Defines an isoparametric membrane-bending or plane strain quadrilateral plate element.
CQUAD8	Quad8	Defines a curved quadrilateral shell or plane strain element with eight grid points. In Nastran, you can define a second order element with missing mid-side nodes. Input data decks containing such elements are read by the translator as a first-order element. A message is written to the `nastran.msg` file indicating the corresponding element ID.

Card	Supported Element Configurations	Description
CQUADR	Quad4	Defines an isoparametric membrane and bending quadrilateral plate element. However, this element does not include membrane-bending coupling. This element is less sensitive to initial distortion and extreme values of Poisson's ratio than the CQUAD4 element. It is a companion to the CTRIAR element.
CROD	Rod	Defines a tension-compression-torsion element.
CSHEAR	Quad4	Defines a shear panel element.
CSEAM	Rod	
CTETRA (4-noded)	Tetra4	Defines the connections of the four-sided solid element with four grid points.
CTETRA (10-noded)	Tetra10	Defines the connections of the four-sided solid element with ten grid points. In Nastran, you can define a second order element with missing mid-side nodes. Input data decks containing such elements are read by the translator as a first-order element. A message is written to the `nastran.msg` file indicating the corresponding element ID.
CTRIA3	Tria3	Defines an isoparametric membrane-bending or plane strain triangular plate element.
CTRIA6	Tria6	Defines a curved triangular shell element or plane strain with six grid points. In Nastran, you can define a second order element with missing mid-side nodes. Input data decks containing such elements are read by the translator as a first-order element. A message is written to the `nastran.msg` file indicating the corresponding element ID.
CTRIAR	Tria3	Defines an isoparametric membrane-bending triangular plate element. However, this element does not include membrane-bending coupling. It is a companion to the CQUADR element.
CTRIAX	Tria3	Defines an axisymmetric triangular element with up to 6 grid points for use in fully nonlinear (i.e.,

Card	Supported Element Configurations	Description
	Tria6	large strain and large rotations) hyperelastic analysis
CTRIAX6	Tria3 Tria6	Defines an isoparametric and axisymmetric triangular cross section ring element with midside grid points.
CTUBE	Rod	Defines a tension-compression-torsion tube element.
CQUADX	Quad4 Quad8	Defines an axisymmetric quadrilateral element with up to nine grid points for use in fully nonlinear (i.e. large strain and large rotations) analysis or a linear harmonic or rotor dynamic analysis. The element has between four and eight grid points. Ninth grid selected in card edit of the element.
CVISC	Spring	Defines a viscous damper element. Elements CDAMP1 and CDAMP2 with grounded terminals are not supported.
CWELD	Mass Rod	Defines a weld or fastener connecting two surface patches or points. Node-Node, Node-Patch, or Patch-Patch weld elements can be read. CWELD element is stored as an element of the rod configuration. CWELD elements using the ELEMID option not created in HyperMesh will be displayed as zero length. Currently, the Spotweld panel can only create Node-Node and Patch-Patch CWELD elements. HyperMesh always calculates the location of GA and GB by projecting GS in the normal direction of surface patch A and surface patch B, respectively.
GENEL	RBE3	Defines a general element.
HM_SPRING	Spring	Defines a spring element, which is converted to Nastran entities on export, in a manner similar to that explained in Using HM_ELAS.
MBOLT	Mass	Defines a bolt for use in SOL 600 in countries outside the USA.

Card	Supported Element Configurations	Description
MBOLTUS	Mass	Defines a bolt for use only in SOL 600 and only in the USA.
PLOTEL	Plot	Defines a one-dimensional dummy element for use in plotting.
RBAR	Weld	Defines a rigid bar with six degrees-of-freedom at each end. RBAR CNA field defaults to 123456. To edit the CNA, CNB, CMA, or CMB fields, you must view the card image for the RBAR element.
RBE2	Rigid Rigidlink	Defines a rigid body with independent degrees-of-freedom that are specified at a single grid point and with dependent degrees-of-freedom that are specified at an arbitrary number of grid points. An RBE2 element with one dependent node is identified as a rigid element, while an element with multiple dependent nodes is identified as a rigid link element.
RBE3	RBE3	Defines the motion at a reference grid point as the weighted average of the motions at a set of other grid points. Individual weight factors can be created on the independent nodes of RBE3 using the update functionality in the RBE3 panel. See the on-line help for the RBE3 panel for more information.
RJOINT		Defines a rigid joint element connecting two coinciding grid points. RBE2

Card	Supported Element Configurations	Description
RROD		Defines a rigid pin-ended element connection.

OptiStruct Cards

Card	Supported Element Configurations	Description
BMFACE	Tria3 Quad4	Defines quad or tria faces that are in turn used to define a barrier to limit the total deformation for free-shape design regions.
CAABSF	Mass Rod Tria3 Quad4	Defines the frequency-dependent fluid acoustic absorber element in coupled fluid-structural analysis.
CBAR	Bar2	Defines a simple beam element (BAR) of the structural model.
CBEAM	Bar2	Defines a beam element (BEAM) of the structural model.
CBUSH	Spring	Defines a generalized spring-damper structural element.
CBUSH1D	Mass Spring	Defines a one-dimensional spring-damper structural element.
CDAMP1	Mass Spring	Defines a scalar damper element. Represented as a spring element type or as a mass element type (grounded CDAMP1).
CDAMP2	Mass Spring	Defines a scalar damper element without reference to a property entry. Represented as a spring element type or as a mass element type (grounded CDAMP2).
CDAMP3	Mass Spring	Defines a scalar damper element that is connected only to scalar points.

Card	Supported Element Configurations	Description
		Represented as a spring element type or as a mass element type (when a coordinate is constrained).
CDAMP4	Mass Spring	Defines a scalar damper element that is connected only to scalar points and is without reference to a material or property entry. Represented as a spring element type or as a mass element type (when a coordinate is constrained).
CELAS1	Mass Spring	Defines a scalar spring element of the structural model. Represented as a spring element type or as a mass element type (grounded CELAS1).
CELAS2	Mass Spring	Defines a scalar spring element of the structural model without reference to a property entry. Represented as a spring element type or as a mass element type (grounded CELAS2). Exported in large field format by optistructlf template.
CELAS3	Mass Spring	Defines a scalar spring element that connects only to scalar points. Represented as a spring element type or as a mass element type (when a coordinate is constrained).
CELAS4	Mass Spring	Defines a scalar spring element that is connected only to scalar points without reference to a property entry. Represented as a spring element type or as a mass element type (when a coordinate is constrained).
CFAST	Mass Rod	Define a fastener with material orientation connecting two shell surfaces. Represented as a mass or rod element type, depending on fastener configuration.

Card	Supported Element Configurations	Description
CGAP	Gap	Defines a gap or friction element. The type of gap elements (either CGAP or CGAPG) is automatically determined based on whether the element is node-to-node or node-to-elem.
CGAPG	Gap Mass	Defines a node-to-obstacle gap element. The obstacle may be an element face or a patch of nodes. The type of gap elements (either CGAP or CGAPG) is automatically determined based on whether the element is node-to-node or node-to-elem.
CGASK6	Penta6	Defining the connections of the GASK6 solid gasket element.
CGASK8	Hex8	Defining the connections of the GASK8 solid gasket element.
CGASK12	Penta15	Defining the connections of the GASK12 solid gasket element.
CGASK16	Hex20	Defining the connections of the GASK16 solid gasket element.
CHACAB	Hex8	Defines the acoustic absorber element in coupled fluid-structural analysis.
CHBDYE	Slave1	Defines a surface element for application of thermal boundary condition. Defined using the Interfacespanel with the CONDUCTION or CONVECTION type.
CHEXA (8-noded)	Hex8	Defines a first order solid element, composed of 6 quadrilateral faces.
CHEXA (20-noded)	Hex20	Defines a second order solid element, composed of 6 quadrilateral faces. A second order element with missing mid-side nodes can be defined in OptiStruct. Input data decks containing such elements are read by the translator as a first-order element. A message is written to the `OptiStruct.msg` file indicating the corresponding element ID.

Card	Supported Element Configurations	Description
CMASS1	Spring Mass	Defines a scalar mass element. Represented as a spring element type or as a mass element type (grounded CMASS1).
CMASS2	Spring Mass	Defines a scalar mass element without reference to a property entry. Represented as a spring element type or as a mass element type (grounded CMASS2).
CMASS3	Spring Mass	Defines a scalar mass element that is connected only to scalar points. Represented as a spring element type or as a mass element type (when a coordinate is constrained).
CMASS4	Spring Mass	Defines a scalar mass element that is connected only to scalar points, without reference to a property entry. Represented as a spring element type or as a mass element type (when a coordinate is constrained).
CMBEAM	Bar2	Defines a beam element for multibody dynamics solution sequence without reference to a property entry.
CMBUSH	Spring	Defines a bushing element without reference to a property entry.
CMBUSHC	Spring	
CMBUSHE	Spring	
CMBUSHT	Spring	
CMSPDP	Spring	Defines a spring damper element without reference to a property entry for multibody solution sequence.
CMSPDPC	Spring	Defines a spring damper element without reference to a property entry for multibody solution sequence.

Card	Supported Element Configurations	Description
CMSPDPE	Spring	Defines a spring damper element without reference to a property entry for multibody solution sequence.
CMSPDPT	Spring	Defines a spring damper element without reference to a property entry for multibody solution sequence.
CONM1	Mass	Defines a 6x6 mass matrix at a geometric grid point.
CONM2	Mass	Defines a concentrated mass at a grid point of the structural model. Exported in large field format by optistructlf template.
CONROD	Rod	Defines a rod element without reference to a property entry.
CONV	Slave1	Defines a free convection boundary condition for heat transfer analysis through connection to a surface element (`CHBDYE` card). Represented as a continuation to CHBDYE slave element card.
CPENTA (6-noded)	Penta6	Defines a first order solid element, composed of 3 quadrilateral and 2 triangular faces.
CPENTA (15-noded)	Penta15	Defines a second order solid element, composed of 3 quadrilateral and 2 triangular faces. A second order element with missing mid-side nodes can be defined in OptiStruct. Input data decks containing such elements are read by the translator as a first-order element. A message is written to the `OptiStruct.msg` file indicating the corresponding element ID.
CPYRA (5-noded)	Pyramid5	Defines a first order solid element, composed of 1 quadrilateral and 4 triangular faces.
CPYRA (13-noded)	Pyramid13	Defines a second order solid element, composed of 1 quadrilateral and 4 triangular faces. A second order element with missing mid-side nodes can be defined in OptiStruct. Input data

Card	Supported Element Configurations	Description
		decks containing such elements are read by the translator as a first-order element. A message is written to the `OptiStruct.msg` file indicating the corresponding element ID.
CQUADR	Quad4	Equivalent to `CQUAD4`. Unlike other Nastran codes, a 6 degree-of-freedom per node formulation is used for all shell elements.
CQUAD4	Quad4	Defines a quadrilateral plate element (`QUAD4`) of the structural model. This element uses a 6 degree-of-freedom per node formulation.
CQUAD8	Quad8	Defines a curved quadrilateral shell element with eight grid points.
CROD	Rod	Defines a tension-compression-torsion element (ROD) of the structural model.
CSHEAR	Quad4	Defines a shear panel element.
CTAXI	Tria3	Defines an axisymmetric triangular cross-section ring element for use in linear analysis.
CTETRA (4-noded)	Tetra4	Defines a first order solid element, composed of 4 triangular faces.
CTETRA (10-noded)	Tetra10	Defines a second order solid element, composed of 4 triangular faces. A second order element with missing mid-side nodes can be defined in OptiStruct. Input data decks containing such elements are read by the translator as a first-order element. A message is written to the `OptiStruct.msg` file indicating the corresponding element ID.
CTRIAR	Tria3	`CTRIAR` entry is equivalent to `CTRIA3`. Unlike other Nastran codes, a 6 degrees-of-freedom per node formulation is used for all shell elements.
CTRIAX6	Tria6	Defines an axisymmetric triangular cross-section ring element for use in Linear Analysis.

Card	Supported Element Configurations	Description
CTRIA3	Tria3	Defines a triangular plate element (TRIA3) of the structural model. This element uses a 6 degree-of-freedom per node formulation.
CTRIA6	Tria6	Defines a curved triangular shell element with six grid points.
CTUBE	Rod	Defines a tension-compression-torsion element (TUBE) of the structural model.
CVISC	Spring	Defines a viscous damper element. Represented as a spring element type.
CWELD	Mass Rod	Defines a weld or fastener connecting two surface patches or points. Represented as a rod element type.
DHEXA8	Hex8	
DQUAD4	Quad4	
DPENTA6		
DTETRAA4	Tetra4	
DTRIA3	Tria3	
HMSPRING	Spring	Defines a spring element, which is converted to OptiStruct entities on export, in a manner similar to that explained in Using HM_ELAS.
JOINT	Joint	Defines a joint.
PLOTEL	Plot	Defines a one-dimensional dummy element for use in plotting.
PLOTEL3	Tria Quad4	Defines a three-noded, two-dimensional dummy element for use in plotting.
PLOTEL4	Tria Quad4	Defines a four-noded, two-dimensional dummy element for use in plotting.
QBDY1	Flux	Defines a uniform heat flux for `CHBDYE` elements.

Card	Supported Element Configurations	Description
RBAR	Weld	Defines a rigid bar with six degrees-of-freedom at each end.
RBE2	Rigid RigidLink	Defines a rigid body whose independent degrees-of-freedom are specified at a single grid point and whose dependent degrees-of-freedom are specified at an arbitrary number of grid points. An RBE2 element with one dependent node is represented as a rigid element type, while an element with multiple dependent nodes is represented as a rigid link element type.
RBE3	Rbe3	Defines the motion at a "reference" grid point as the weighted average of the motions at a set of other grid points.
RROD	Rod	Defines a pin-ended rod that is rigid in extension.

PAM-CRASH Cards

The component of the element refers to a material, which contains the material definition for PAM-CRASH.

FE input reader will not create connectors for Plinks, instead you must use the FE absorb functionality to create connectors from PLINKs.

Card	Supported Element Configurations	Description
ASSOCIATE	Rigids	Defines entities to be converted from deformable to rigid.
BAR /	Rod	Bar element.
BASE_BODY	Rigid	Rigid bodies on which the boundary conditions are defined, used in Multibody systems.
BEAM /	Bar2	Beam element. If the orientation vector is defined via vectors, the string VECTOR is displayed in the N3 field, and a zero is written in the exported deck. If the y-direction node is directly specified, its ID is displayed in the N3 field.

Card	Supported Element Configurations	Description
BSHEL /	Hex8	8-noded brick shell element.
EDG	Rod	
ELINK /	Rod	Link element. The element must be edited in the card previewer to define the connections.
JOINT /	Rod	Joint element. The element must be edited in the card previewer to define element orientation.
KJOIN /	Rod	Kinematic joint element. The element must be edited in the card previewer to define element orientation.
LLINK /	Rod	Link element. The element must be edited in the card previewer to define the connections.
MASS	Mass	Added mass.
MEMBR /	Tria3 Quad4	Membrane element.
MTJOIN /	FE Joint	Multiple node to one node Kinematic joint element.
MTOCO /	Rigid	Rigid element.
NODCO /	Rigid	Nodal constraint definition. This configuration allows you to create nodal constraints via entity sets. The element must be edited in the card previewer. Degrees of freedom are ignored.
OTMCO /	RBE3	One node to multiple node constraints.
PLINK /	Mass	Plink element. The element must be edited in the card previewer. Mass value is ignored. Use the following templates to handle the welds:

Card	Supported Element Configurations	Description
		find_welds find_master_comps_welds find_slave_comps_welds find_comps_welds
PLINK_VI	Rod	These elements are created during connector realization to show the actual connections. They are not exported.
RETRA /	Mass	The element must be edited in the card previewer. Mass value is ignored.
RBODY /	Weld	Rigid body with 2 nodes. When created, the default value for rigid body type is 1.
RBODY /	Rigid	This configuration allows you to create rigid bodies via entity sets. The element must be edited in the card previewer. Degrees of freedom are ignored.
SEG	Tria3 Quad4	The elements are used in entity selection. With this keyword, only nodes of these elements are output along with SEG keyword.
SENPT /	Mass	
SHELL /	Mass	Shell element. Mass value is ignored. Mass defined with a keyword other than NOD is not supported. The reader creates one mass element for each NOD definition in the MASS card, therefore the exported deck will contain the same number of MASS/cards as many NOD definitions.
SHELL /	Tria3	The default behavior for tria3 elements is Coo triangles. To output standard triangles (N3 = N4).
SHELL /	Quad4	
SLINK /	Tria3 Quad4	The element must be edited in the card previewer to define the connections.

Card	Supported Element Configurations	Description
SLIPR /	Mass	The element must be edited in the card previewer. Mass value is ignored.
SOLID /	Tetra4 Pyramid5 Penta6 Hex8	8-noded brick element.
SPH /	Mass	
SPRING /	Spring	Spring element. The element must be edited in the card previewer to define element orientation.
SPRGBM /	Spring	Spring beam element. The element must be edited in the card previewer to define element orientation.
TETRA /	Tetra10	10-noded tetra element. Local frame definition.
TETR4 /	Tetra4	4-noded tetra element.
TSHEL /	Quad4	4-noded thick shell element. The element must be edited in the card previewer to define the connections.

Permas Cards

Card	Supported Element Configurations	Description
BEAM2	Bar2	2 noded straight general beam.
BECOC	Bar2	2 noded straight thin-walled tube.
BECOS	Bar2	2 noded straight solid beam.
CONAX2	Bar2	

Card	Supported Element Configurations	Description
CONAX3	Bar3	
CONA3	Tria3	3 noded triangular surface convection and radiation element.
CONA4	Quad4	4 noded quadrilateral surface convection and radiation element.
CONA6	Tria6	6 noded triangular surface convection and radiation element.
CONA8	Quad8	8 noded quadrilateral surface convection and radiation element.
CONS3	Tria3	3 noded triangular shell surface convection and radiation element.
CONS4	Quad4	4 noded quadrilateral shell surface convection and radiation element.
CONS6	Tria6	6 noded triangular shell surface convection and radiation element.
CONS8	Quad8	8 noded quadrilateral shell surface convection and radiation element.
DAMP1	Spring	Translational viscous damper.
DAMP3	Spring	Viscous damper for three degrees of freedom.
DAMP6	Spring	Viscous damper for six degrees of freedom.
FLA2	Rod	2 noded straight flange (rod).
FLA3	Bar3	3 noded straight flange (rod).
FLHEX8	Hex8	8 noded fluid hexahedron.
FLHEX20	Hex20	20 noded fluid hexahedron.
FLPENT6	Penta6	6 noded fluid pentahedron.
FLPENT15	Penta15	15 noded fluid pentahedron.
FLPYR5	Pyramid5	5 noded fluid pyramid.
FLTET4	Tetra4	4 noded fluid tetrahedron.

Card	Supported Element Configurations	Description
FLTET10	Tetra10	10 noded fluid tetrahedron.
FSINTA3	Tria3	3 noded triangular fluid structure interface element.
FSINTA4	Quad4	4 noded quadrilateral fluid structure interface element.
FSINTA6	Tria6	6 noded triangular fluid structure interface element.
FSINTA8	Quad8	8 noded quadrilateral fluid structure interface element.
GKHEX8	Hex8	8 noded solid hexahedron.
GKHEX20	Hex20	20 noded solid hexahedron.
GKPNT6	Penta6	6 noded solid pentahedron.
GKPNT15	Penta15	15 noded solid pentahedron.
HEXE8	Hex8	8 noded solid hexahedron.
HEXE20	Hex20	20 noded solid hexahedron.
LOADA3	Tria3	3 node triangular load carrying membrane element.
LOADA4	Quad4	4 node quadrilateral load carrying membrane element.
LOADA6	Tria6	6 node triangular load carrying membrane element.
LOADA8	Quad8	8 node quadrilateral load carrying membrane element.
MASS3	Mass	Point mass.
MASS6	Mass	Rigid mass.
$MPC ASSIGN	Rigid	Dependent nodes are forced to have the same displacement as an independent node. User can specify degrees of freedom for dependent and independent nodes.
$MPC JOIN	Rigid	Pairwise identical displacements.

Card	Supported Element Configurations	Description
$MPC RIGID	Rigid	Rigid regions.
$MPC SAME	Rigid	Identical corresponding displacements.
$MPC WLSCON	RBE3	Weighted averaged connection.
NLDAMP	Spring	Nonlinear translational viscous damper.
NLDAMPR	Spring	Nonlinear rotational viscous damper.
NLSTIFF	Spring	Nonlinear translational spring.
NLSTIFFR	Spring	Nonlinear rotational spring.
PENTA6	Penta6	6 noded solid pentahedron.
PENTA15	Penta15	15 noded solid pentahedron.
PLOTA3	Tria3	3 noded triangular plot element.
PLOTA4	Quad4	4 noded quadrilateral plot element.
PLOTA6	Tria6	6 noded triangular plot element.
PLOTA8	Quad8	8 noded quadrilateral plot element.
PLOTL2	Plot	2 noded straight line plot element.
PLOTL3	Bar3	3 noded straight line plot element.
PYRA5	Pyramid5	5 noded solid pyramid.
QUAD4	Quad4	4 noded quadrilateral shell element.
QUAM4	Quad4	4 noded quadrilateral plane membrane element.
QUAM8	Quad8	8 noded quadrilateral plane membrane element
QUAMS4	Quad4	4 noded quadrilateral solid shell element
QUAMS8	Quad8	8 noded quadrilateral solid shell element
QUAX4	Quad4	4 noded quadrilateral axisymmetric solid element
QUAX8	Quad8	8 noded quadrilateral axisymmetric solid element
SHEAR4	Quad4	4 noded quadrilateral plane shear panel element.
SHELL3	Tria3	3 noded triangular shell element for laminates.

Card	Supported Element Configurations	Description
SHELL4	Quad4	4 noded quadrilateral shell element for laminates.
SPRINGX1	Spring	
SPRINGX2	Spring	
SPRINGX3	Spring	
SPRING1	Spring	Translational spring.
SPRING3	Spring	Spring with three translational stiffnesses
SPRING6	Spring	Spring with three translational and three rotational stiffnesses.
TET4	Tetra4	4 noded solid tetrahedron.
TET10	Tetra10	10 noded solid tetrahedron.
TRIA3	Tria3	3 noded triangular shell element.
TRIA3K	Tria3	3 noded triangular thin shell element.
TRIM3	Tria3	3 noded triangular plane membrane element.
TRIM6	Tria6	6 noded triangular plane membrane element.
TRIMS3	Tria3	3 noded triangular solid shell element
TRIMS6	Tria6	6 noded triangular solid shell element.
X1DAMP3	Mass	Scalar viscous damper at one node with three degrees of freedom.
X1DAMP6	Mass	Scalar viscous damper at one node with six degrees of freedom.
X1GEN6	Mass	General element at one node with six degrees of freedom.
X1MASS3	Mass	Scalar mass at two nodes with three degrees of freedom.
X1MASS6	Mass	Scalar mass at one node with six degrees of freedom.
X1STIFF3	Mass	Scalar spring at one node with three degrees of freedom.

Card	Supported Element Configurations	Description
X1STIFF6	Mass	Scalar spring at one node with six degrees of freedom.
X2DAMP3	Spring	Scalar viscous damper at two nodes with three degrees of freedom.
X2DAMP6	Spring	Scalar viscous damper at two nodes with six degrees of freedom.
X2GEN6	Mass	General element at two nodes with six degrees of freedom.
X2STIFF3	Spring	Scalar spring at two nodes with three degrees of freedom.
X2STIFF6	Spring	Scalar spring at two nodes with six degrees of freedom.

Radioss Cards

Note:

- Shell thickness is included with the connectivity data. The default value is contained in the property set.
- The time history is provided for the elements.
- 3D elements are supported.
- Degenerated 3D solid elements (from Hexa), such as Tetra and Penta, are included with the present 3D elements.

Card	Supported Element Configurations	Description
/ADMAS	Mass	Assign additional non-structural mass to nodes or a group of nodes. Optionally the total additional non-structural mass of a part or a group of parts can be defined (applied to shells and solids only) or a surface mass can be assigned to a surface and Radioss would then compute the added node based mass value using area (volume) - weighted distribution.
/BEAM	Bar	Describes the beam elements. Two properties (`/PROP/TYPE3 (BEAM)` and `/PROP/TYPE18`

Card	Supported Element Configurations	Description
		`(INT_BEAM)`) are available for this beam element. The properties describing a beam element are all defined in a local beam coordinate system.
/BRIC20	Hex20	Describes 3D solid elements (20 Node Brick Elements). This quadratic element should be used with the property `/PROP/SOLID`.
/BRICK	Hex8 Penta6 Pyramid5 Tetra4	Defines a hexahedral solid element and thick shell element with 8 nodes.
/CYL_JOINT	Rigid	Defines cylindrical joints.
/QUAD	Quad4	Describes the 2D solid elements. QUAD elements must be defined in the global YZ plane.
/RBE2	Rigid	Defines a rigid body whose independent degrees of freedom are specified at a single master node and whose dependent degrees of freedom are specified at an arbitrary number of slave nodes.
/RBE3	RBE3	Defines the motion of a reference (slave) node as the weighted average of the motions of sets of master nodes.
/RBODY	Rigid	Defines rigid bodies.
/RLINK	Rigid	Defines a rigid link. The rigid link imposes the same velocity on all the slave nodes in one or more directions.
/SHELL	Quad4	Describes input for 4-node shell elements.
/SH3N	Tria3	Describes the triangular 3-node shell elements.
/SPHCEL	Mass	Describes the SPH cells.
/SPRING2N	Spring2N	Describes the 2-noded spring elements.
/SPRING3N	Spring3N	Describes the 3-noded spring elements.
/SRPING4N	Spring4N	Describes the 4-noded spring elements.

Card	Supported Element Configurations	Description
/TETRA4	Tetra4	Describes a tetrahedral solid element with 4 nodes.
/TETRA10	Tetra10	Describes a tetrahedral solid element with 10 nodes.
/TRUSS	Rod	Describes one dimension truss elements, which could be used with property `/PROP/TYPE2 (TRUSS)`. Truss could only carry axial load (like for bar).
/XELEM	Xelem	Describes the multi-strand elements.

Samcef Cards

Card	Supported Element Configurations	Description
AXISYM	Bar2 Tria3 Tria6 Quad4 Quad8	
BUSH	Spring	
COMP AXISYM	Tria3 Tria6 Quad4 Quad8	
COMP DEFO PLAN	Tria3 Tria6 Quad4 Quad8	
COMP MEMB BIDIM	Tria3 Tria6 Quad4	

Card	Supported Element Configurations	Description
	Quad8	
COMP PLAN GENE	Tria3 Tria6 Quad4 Quad8	
COMP VOLU	Hex8 Hex20 Penta6 Penta15 Pyramid5 Pyramid13 Quad4 Quad8 Tria3 Tria6	
COMP VOLU COQUE	Hex8 Hex20 Penta6 Penta15	
COQU DEFO PLAN	Bar2 Rod Tria3 Tria6 Quad4 Quad8	
DEFO GENE	Tria3 Tria6 Quad4 Quad8	
DEFO PLAN	Bar2	

Card	Supported Element Configurations	Description
	Rod Tria3 Tria6 Quad4 Quad8	
FLUX THERMIQUE	Rod Tria3 Quad4 Tria6 Quad8	
FOUR MULT HARM	Bar2 Tria3 Tria6 Quad4 Quad8	
FOURIER	Bar2 Tria3 Tria6 Quad4 Quad8	
HETEROSIS	Quad8 Tria6	
HYBR VOLU	Bar2 Hex8 Hex20 Penta6 Penta15 Pyramid5 Pyramid13 Rod Tetra4 Tetra10 Tria3 Tria6 Quad4	

Card	Supported Element Configurations	Description
	Quad8	
HYBRID VOLU COQUE	Hex8 Hex20 Penta6 Penta15	
MEMB AXISYM	Bar2	
MEMB BIDIM	Bar2 Rod Tria3 Tria6 Quad4 Quad8	
MEMB FLEXION	Tria3 Tria6	
MEMB FOURIER	Bar2	
MEMB MULT HARM	Bar2	
MINDLIN	Bar2 Rod Tria3 Quad4 Tetra4 Pyramid5 Penta6 Hex8 Tria6 Quad8 Tetra10 Pyramid13 Penta15 Hex20	

Card	Supported Element Configurations	Description
RBE3	RBE3	
SAND VOLU	Quad4 Quad8 Tria3 Tria6	
SOLID SHELL	Hex8 Hex20 Penta6 Penta15	
THER AXISYM	Bar2 Tria3 Tria6 Quad4 Quad8	
THER COQU	Rod Tria3 Quad4 Tria6 Quad8	
THERMIQU	Rod Tria3 Quad4 Tetra4 Pyramid5 Penta6 Penta15 Hex8 Tria6 Quad8 Tetra10 Pyramid13	

Card	Supported Element Configurations	Description
	Penta15 Hex20	
TUYAU	Rod	
TUYAU THERMIQUE	Rod	
VOLU COQUE	Hex8 Hex20 Penta6 Penta15	
VOLUMIC	Bar2 Hex8 Hex20 Rod Tetra4 Tetra10 Tria3 Quad4 Tria6 Quad8 Penta6 Penta15 Pyramid5 Pyramid13	

Criteria and Parameter Settings

Configure criteria and parameter settings.

See Also
Edit Criteria and Parameter Files

Criteria Settings

Setup the quality index (QI) mesh criteria in the **Criteria and Parameters Files Editor**, Criteria tab.

This criteria is used in QI and batch meshing, and in QI-based element checks. Criteria definitions can be saved to a file, and loaded for subsequent editing.

The editor is laid out in a table format, with each check displayed in the first column, and the controls and values associated with each check in subsequent columns to the right.

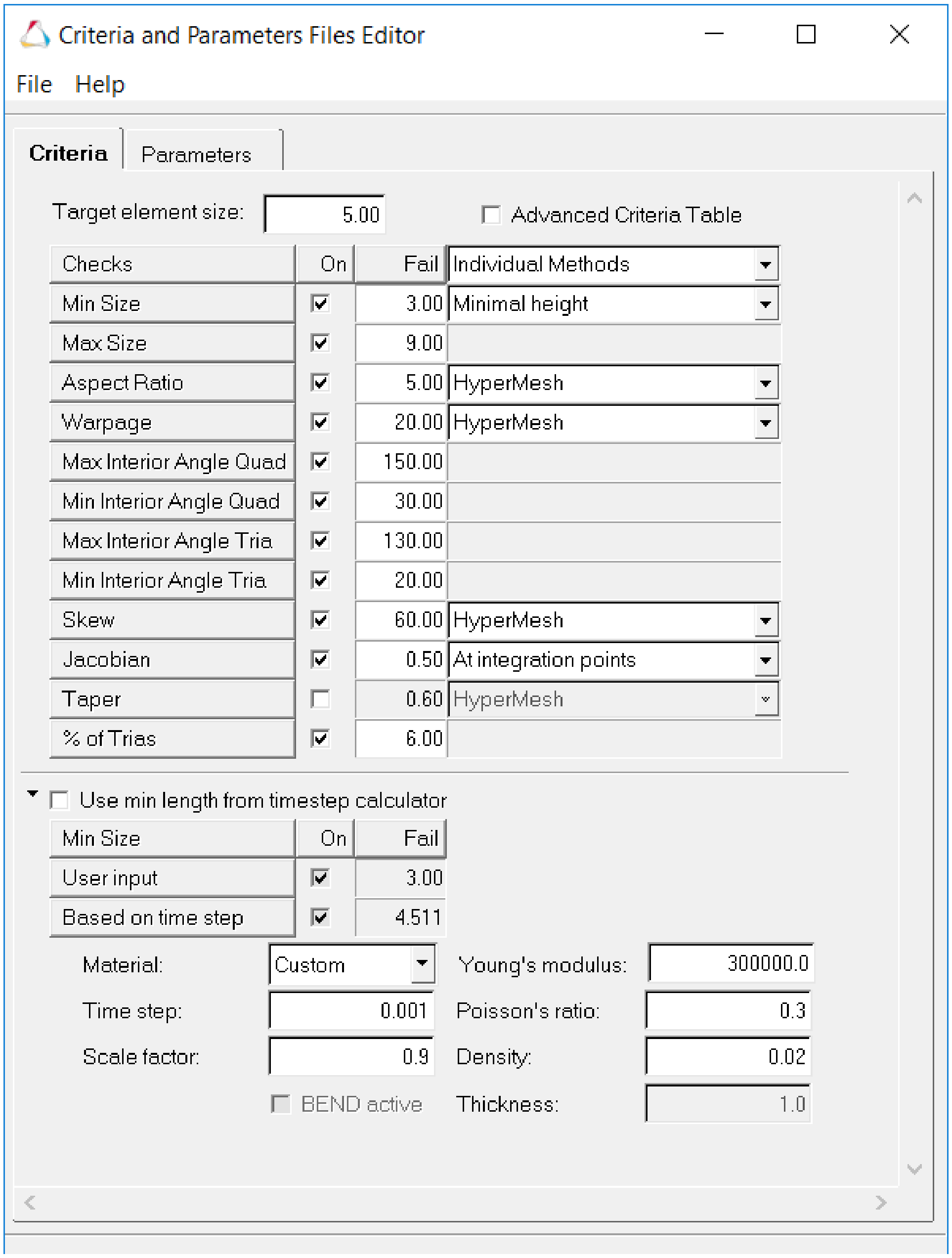


Figure 685:

Target element size

Desired element size for the mesh.

> **CAUTION:** Modifying this value may generate errors in the Min Size and Max Size, which will show up as red text.

Turn checks on/off

In the On column, use the checkbox to turn each individual check on/off.

Fail value

In the Fail column, edit the fail value for each specific check.

Good, warn, worst, and weight value

The Good, Warn, and Worse fields are interpolated or extrapolated automatically.

Select the **Advanced Criteria Table** checkbox to access the Weight, Good, Warn and Worst columns and edit each check's corresponding value.

Such changes modify how the quality index value is calculated, but do not significantly impact the final mesh. Weight factors are also adjustable, but they also do not make a large difference in the final mesh, unless they are set to an order of magnitude higher than the others. Default values for the weight factors are recommended.

Calculation methods

In the last column, select a calculation method to apply for certain checks. Available methods may vary for each check.

To use more than one solver's method, select the **Individual Methods** option from the first list box in the last column. The different solver calculation methods are described in How Element Quality is Calculated. The Chordal Deviation check is currently ignored when performing batch meshing or QI meshing. It is only used to calculate the QI value from the Quality Index panel.

Minimum length calculator

Below the table is a minimum length calculator, based on time step, which can be used to calculate the suggested minimum length, based on a material, time step, and a scale factor.

A pre-defined material can be selected, or a custom material can be defined.

Apply the calculated minimum length by selecting the **Use min length from timestep calculator** checkbox.

Parameter Settings

Setup the geometry cleanup and defeaturing parameters in the **Criteria and Parameters Files Editor** dialog, Parameter tab.

These parameters are used to define things such as washer layers around holes, defeaturing pinholes and solid holes, rows of elements along fillets, and many other options.

The Parameters tab is divided into multiple sections. Each section can be toggled to show or hide its options via the small triangular arrow ( ) to the right of it. Each section represents a specific type of operation, which can be enabled or disabled at several levels.

Element/Import

Figure 686:

Target element size

Desired element size for meshing and optimization.

> **Note:** The element size defined here should match the ideal value for min length and max length as defined in the criteria file. If this does not match, Batchmesher may not be able to produce meshes that adhere to the target quality requirements.

Import model with tolerance

Tolerance value to be used while importing the CAD model.

Select **auto** (recommended) to automatically calculate the tolerance based on the type and dimensions of the model.

Extract Midsurface

The Extract midsurface parameters define the tasks that are performed by Batchmesher when extracting the midsurface.

Select the **Extract midsurface** checkbox to extract the midsurface before meshing using the selected extraction method. Only the midsurface geometry is meshed and the original geometry is deleted.

> **Note:** For parameter files saved before 14.0, the midsurface extraction method used is "offset", which matches with what was used in earlier releases before the option to select the method was available.

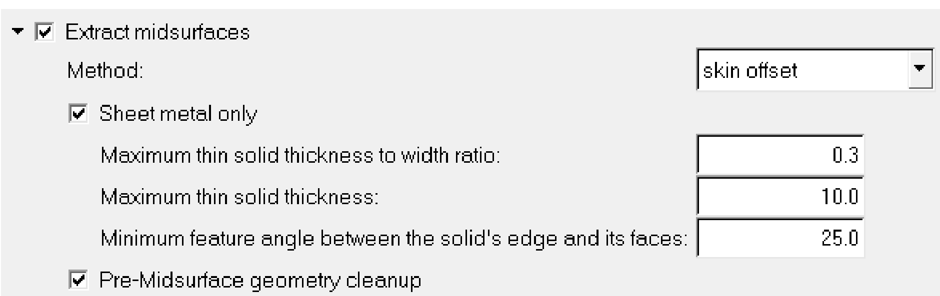


Figure 687:

Method

Select method to use when extracting the midsurface before meshing.

Sheet metal only

Only consider geometry for midsurface extraction that meets the user defined settings for the options specific to sheet metal.

> **Note:** If this option is disabled, it will result in a time savings, but all parts will be attempted to have a midsurface extracted.

Maximum thin solid thickness to width ratio

Maximum ratio between the approximate thickness of the thin solid part (shortest dimension) and its approximate width (2nd shortest dimension). This parameter is used to limit the midsurface extraction to parts for which the thickness is clearly smaller than the length and width.

Maximum thin solid thickness

Ignore thin solids with a thickness less than the specified value during midsurface extraction.

Minimum feature angle between the solid's edge and its faces

Minimum angle used to distinguish top and bottom faces of a thin solid from its sides. Angles less than the specified value will be treated as if they were flat for purposes of midsurface extraction.

Pre-Midsurface Geometry cleanup

Perform geometry cleanup steps on the model before midsurface extraction.

Direct Midmesh

The Direct Midmesh parameters defines settings used to create direct midmesh.

Select the **Allow direct midmesh** checkbox to create direct midmesh for the parts where midsurface extraction is difficult or not possible for example plastics, castings and machined parts. Additional options are also enabled once this checkbox is enabled.

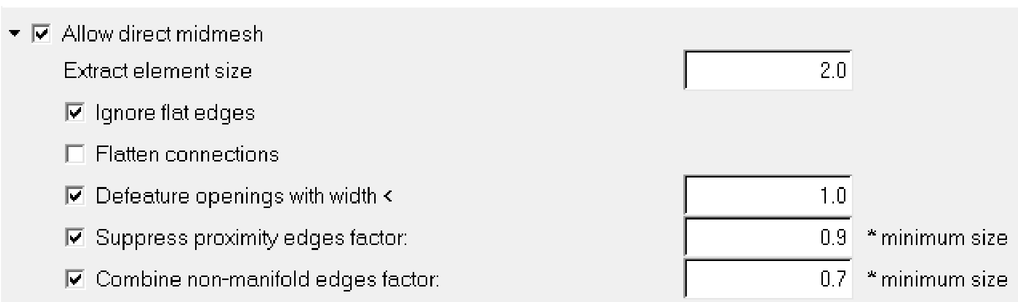


Figure 688:

Ignore flat edges

Do not imprint flat edges from the input geometry onto the midmesh.

Flatten connections

Align/flatten the midmesh at ribs/connections.

Defeature openings with width <

Remove small holes and openings less than the specified width.

Suppress proximity edges factor

Remove 1D topology edges within the given factor of the minimum size from the criteria file.

Combine non-manifold edges factor

Join non-manifold edges within the given factor of the minimum size from the criteria file.

If a midsurface is extracted:	Traditional midsurface geometry cleanup and meshing are performed.		
If a midsurface is not extracted:	Extract midsurfaces enabled	Allow direct midmesh enabled	If the part is determined to be midmeshable, certain input geometry cleanup steps and direct midmesh generation are performed. If the part is determined to not be midmeshable, traditional input geometry cleanup and meshing are performed.
		Allow direct midmesh disabled	Traditional input geometry cleanup and meshing are performed.
	Extract midsurfaces disabled	Allow direct midmesh enabled	If the part is determined to be midmeshable, certain input geometry cleanup steps and direct midmesh generation are performed. If the part is determined to not be midmeshable, traditional input geometry cleanup and meshing are performed.
		Allow direct midmesh disabled	Traditional input geometry cleanup and meshing are performed.

Geometry Cleanup

Geometry cleanup parameters define a variety of geometry feature recognition and preparation tasks performed by Batchmesher.

Geometry cleanup behaviors and options provide excellent feature capture with more user control resulting in more predictability, consistency and ease of use.

Batchmesher recognizes classified features like beads, dimples, flat bottom bosses/depressions, flanges, fillets, holes (2d and 3d), and so on. These features are treated following user defined criterias, allowing the preservation of the main feature edges providing excellent feature capture.

The main tools for geometry cleanup include:

- Flat feature suppression level, a curvature based feature suppression.
- Suppress edges by proximity, allows to handle feature edges in close proximity, generally based on minimum element size.

Controlling the above parameters can result in good feature capture with minimum quality index failures. However, features are given more importance which might increase the failed element count based on geometry and the cleanup parameter values. It is important to define all of the settings appropriately.

Select the **Geometry cleanup** checkbox to enable additional cleanup parameters that can be turned on and off independently.

Surface Hole Recognition

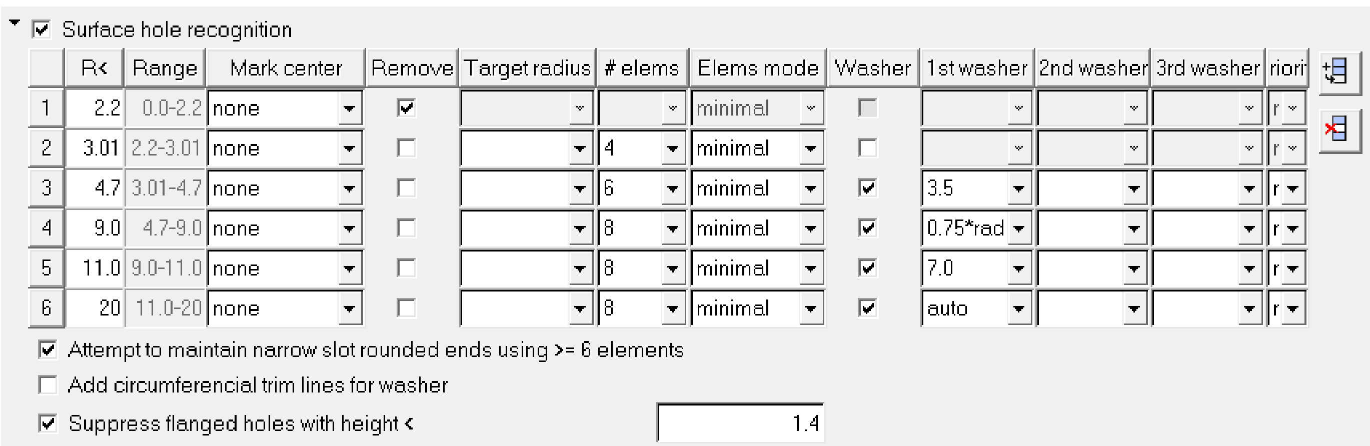


Figure 689:

When **Surface hole recognition** is activated, surface holes of different sizes are recognized and treated appropriately. A table becomes enabled to define the radii ranges and additional options.

Surface hole recognition table data

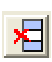
Define radii ranges and additional options in the table. Click  to add a row to the table, or click  to remove a row from the table.

Table 192: Surface Hole Recognition Table Data

Column	Action
R<	Maximum radius of the current hole range. The minimum value is taken as 0.0 for the first row, or as the maximum value from the previous row. For slotted holes, the radius is measured at the tip of the hole.
Range	Radius range for the current row. This value is read-only.
Mark center	Create a node and tag at the center of the hole, or do nothing.
Remove	Remove (defeature) the hole. For slotted holes, the hole is removed only if the tip radius is less than the specified radius threshold, and

Column	Action
	the length of the hole is less than 1.4 times the target element size. If Remove is disabled, additional options are available.
Target radius	Adjust holes in the range to have the specified target radius. The radius can be specified as an exact value, for example 5.0, or as an expression based on the original radius, for example radius*1.1, radius-0.5, radius+0.5.
# elems	Enter the minimum/exact number of elements to create around the holes, or set to **auto** to automatically select the number of elements so that the min and max element size requirements are satisfied, with the best possible representation of the hole shape. **Tip:** Auto is not recommended for holes with washer layers.
Elems mode	Choose whether # elems setting defines the minimum or exact number of elements.
Washer	Create washer layers around holes. If specified, one or two layers can be created.
1st washer/2nd washer/3rd washer	Sets the width of the first, second, third washer as a constant value (select the blank entry in the drop down and enter a value), a scale of the hole radius, for example 0.6*radius, a subtraction formula, for example 14.0-radius, or an automatic determination based on element quality.
Priority	Set the priority of one radii range over the others. For example, to ensure all bolt holes (radii 10-15) have correct washers but other holes are not critical, holes with radii 10-15 will receive higher priority than others. This ensures that if two holes close to each other in the model have overlapping/conflicting washers, the hole with higher priority gets the washer while the other does not, or the hole with the lower priority may get a modified washer instead. In addition, when a hole is set to high priority, washer elements are not modified to correct for failed element quality. If a hole is set to normal priority, washer nodes are allowed to move to correct the quality.

Attempt to maintain narrow slot rounded ends using >=6 elements

Attempt to generate a mesh using the pattern indicated in Figure 690.

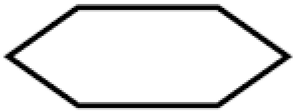


Figure 690:

Clear this checkbox to attempt to generate a mesh using the pattern indicated in Figure 691, which has six elements, two on each long side and one on each end.

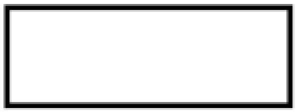


Figure 691:

Add circumferencial trim lines for washer

Keep geometry trim lines for washers.

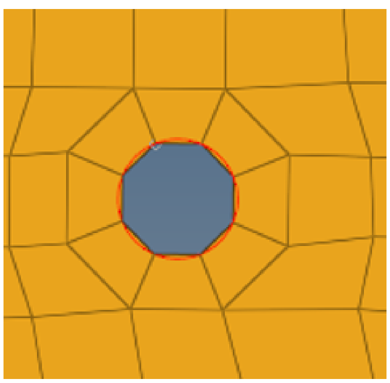


Figure 692: Add circumferencial trim lines for washer - Off

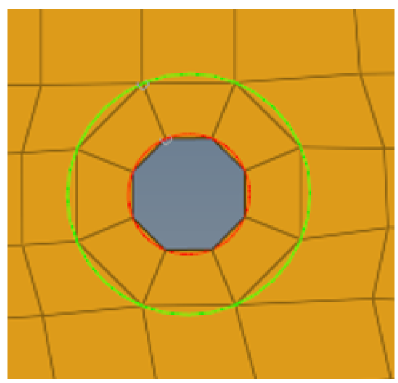


Figure 693: Add circumferencial trim lines for washer - On

Suppress flanged holes with height <

Recognize holes with small downward flanges and eliminate those flanges with a height less than the specified value. Flanges with a height less than the minimal element size are extended to the minimal element size if not removed.

Use File for Hole Recognition

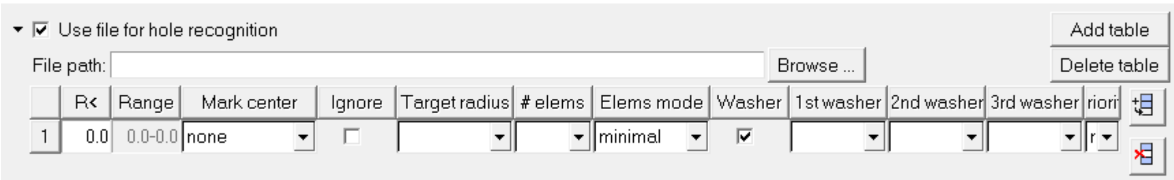


Figure 694:

Select **Use file for hole recognition** to provide a file containing X, Y, Z center locations of all of the holes to consider. This is useful for special treatment of specific holes, usually bolt holes.

Batchmesher compares the defined locations to the holes in the model, and prioritizes the holes that match. All of the options for Surface hole recognition are available for these holes. If one or more holes files are defined, Batchmesher looks for the found holes in each file, in the order the files are defined. If found, it applies the washer table linked to the first found file to the corresponding holes. If a hole is not found in any file, the settings from the default general surface holes table are used.

Multiple files can be specified, each with their own definitions. The order of the files determines the order of precedence in the case where there are overlapping or conflicting definitions.

Click **Add table** to add a new table for creating a new hole file. Click **Delete table** to delete the specified hole file table.

The holes file must contain one line for each hole, with the values either space, tab or comma separated. Each line contains a line number followed by the X, Y, Z locations of each hole center.

```
1 1420 -839 65
2 1724 -846 212
3 1683 -845 265
4 1660 -841 308
```

Figure 695: Spaces/Tabs with Line Numbers

```
1,1420,-839,65
2,1724,-846,212
3,1683,-845,265
4,1660,-841,308
```

Figure 696: Commas with Line Numbers

Solid Hole Recognition

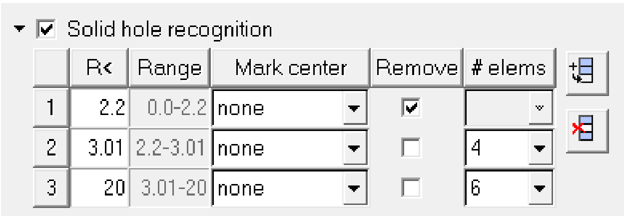


Figure 697:

Select **Solid hole recognition** to recognize and treat solid holes (cylindrical surfaces in volumes) of different sizes. A table becomes enabled to define the radii ranges and additional options.

Click to add a row to the table, or click to remove a row from the table.

Table 193: Solid Hole Recognition Table Data

Column	Action
R<	Maximum radius of the current hole range. The minimum value is taken as 0.0 for the first row, or as the maximum value from the previous row.
Range	Radius range for the current row. This value is read-only.
Mark center	Create a node and tag at the center of the hole, or to do nothing.
Remove	Removes (defeature) the hole. If Remove is disabled, you must specify the minimum/exact # elems to create around the holes.

Surface Fillet Recognition

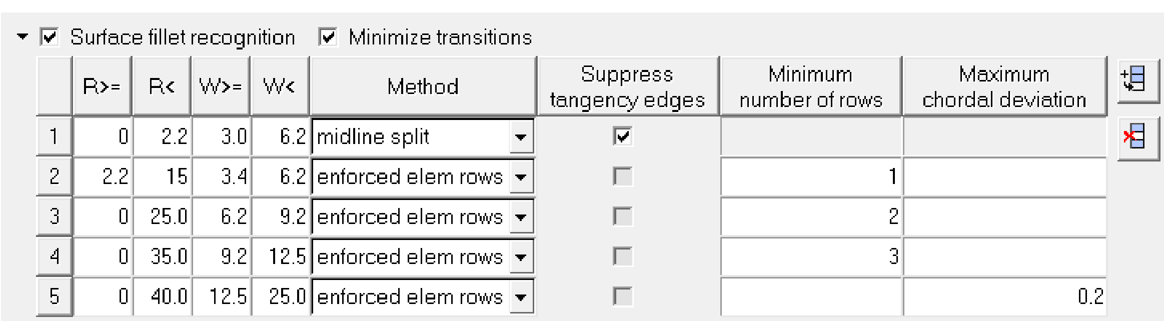


Figure 698:

Select **Surface fillet recognition** to recognize surface fillets in order to perform one or more of the following options:

- Prevent the main (long) edges of the fillets from being suppressed, and also prevent the nodes of those edges from moving while fixing element quality.
- Remove/defeature fillets. Gaps may result if complicated fillets cannot be removed.
- Split the fillets along the mid-line and suppress the edges.
- Specify the number of elements across the width of the fillets for given fillet radii.
- Specify the chordal deviation to be achieved while meshing.

A table becomes enabled to define a desired number of element rows for specific ranges of average fillet radii, width, or both. The width value is defined is the arc length of the fillet.

In Figure 699, uniform fillet strips with an average radius between 3 and 5 and an average width between 2.0 to 9.0 will be meshed with one row of elements; uniform fillet strips with an average radius between 5 and 20 and an average width between 9.0 to 16.0 will be meshed with two rows of elements; and uniform fillets strips with an average radius between 20 and 30 and an avarage width between 16.0 to 24.0 will be meshed with three rows of elements. This rule does not apply to fillets with an average element width below or above the defined ranges of non-uniform fillet strips (when minimal and maximal width of fillets exceed 30%).

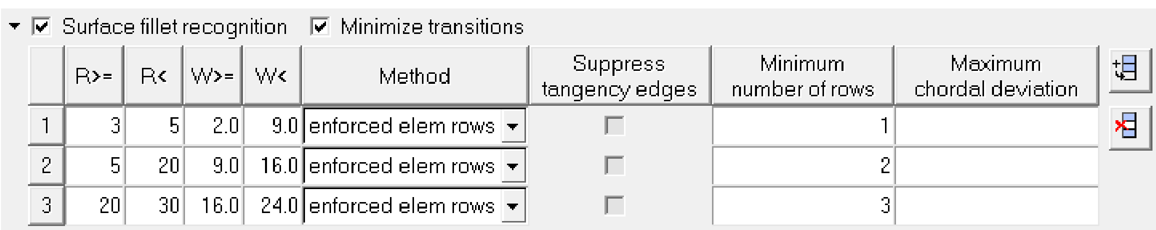


Figure 699:

If the width or number of rows columns in the surface fillet recognition table are empty, the next default value will be applied. In this example, that means uniform fillet strips with an average fillet width between the element sizes of 0 to 2.0 will be meshed with one row of elements.

A fillet can be meshed with enforced rows of elements, or split at its midline and meshed accordingly based on element quality.

The mesh settings can be defined as an exact number of rows when **Minimize transitions** is disabled. This allows the Suppress tangency edges option to also become available. When enabled, fillets are treated by making a midline and suppressing the fillet itself. This combination may be selected to defeature very narrow fillets. Midline spliting without suppressing tangency edges can be used for wide fillets to ensure that the fillet mesh will be symmetrical. Enabling Minimize transitions helps to reduce trias. The mesh settings are then provided either as a minimum number of elements and/or determined based on a maximum chordal deviation criterion. Batchmesher calculates the required number of elements as the maximum of the user-specified number of rows and the number of elements required to meet the maximal chordal deviation.

> **Note:** The minimal element size and aspect ratio criteria requirements are always honored. This means that the element quality restrictions have the highest priority when calculating the element density for a fillet range.

Flange Recognition

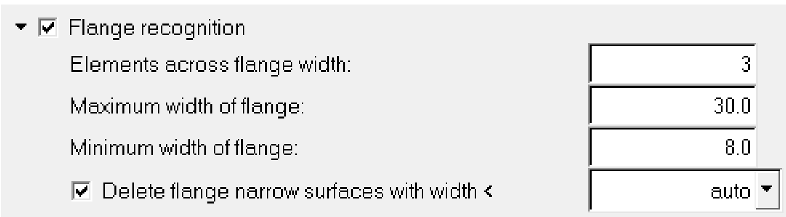


Figure 700:

When **Flange recognition** is activated, geometry that represents flanges on sheet metal parts is recognized and the below options become enabled. Flanges may be modified to suppress construction lines, subdivide them into rectangular areas, or otherwise prepare them for proper meshing. As this functionality is not supported for solid geometries, it should be disabled for such models to improve performance.

Elements across flange width
: Minimum number of elements to be created across the flange width.

Maximum width of flange
: Maximum flange width to consider for flange recognition.

Minimum width of flange
: Minimum flange width to consider for flange recognition.

Delete flange narrow surfaces with width <
: Controls the removal of narrow flange surfaces to avoid creation of sliver elements and disruptions in the mesh flow.

  Auto
  : Delete narrow flange surfaces when the maximal narrow surface width is the minimum of 0.2*element_size and min_element_size.

  <value>
  : Delete narrow flange surfaces when the maximal narrow surface width is the minimum of the specified value.

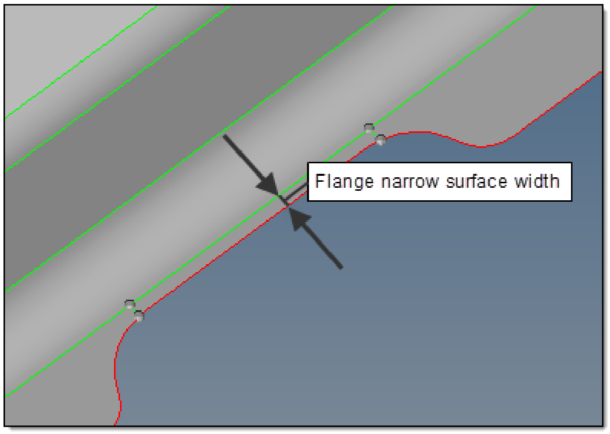


Figure 701: Flange Narrow Surface Width

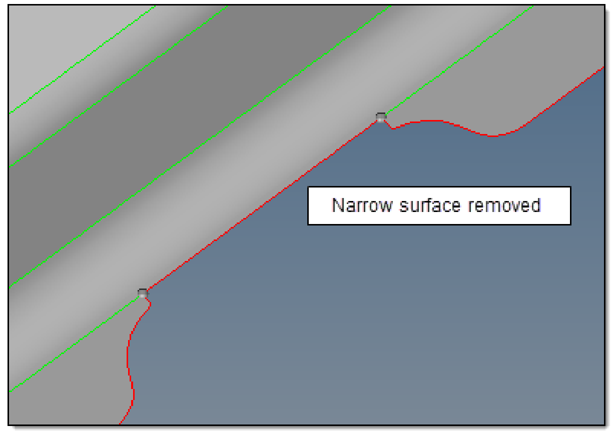


Figure 702: Narrow Surface Removed

Bead Recognition

Figure 703:

When **Bead recognition** is activated, geometry that represents beads on sheet metal parts is recognized and the below options become enabled.

Suppress beads with height <

Enable bead recognition and suppress any beads with a height less than the specified value. This helps eliminate small elements and aids in creating a good mesh flow.

Preserve rounded bead midline

Enforce node placement along the midline of a rounded bead.

Logo Recognition

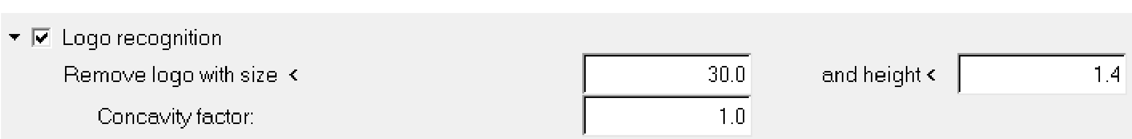


Figure 704:

Use the Logo Recognition parameters to remove small geometric features that represent logos in the model design.

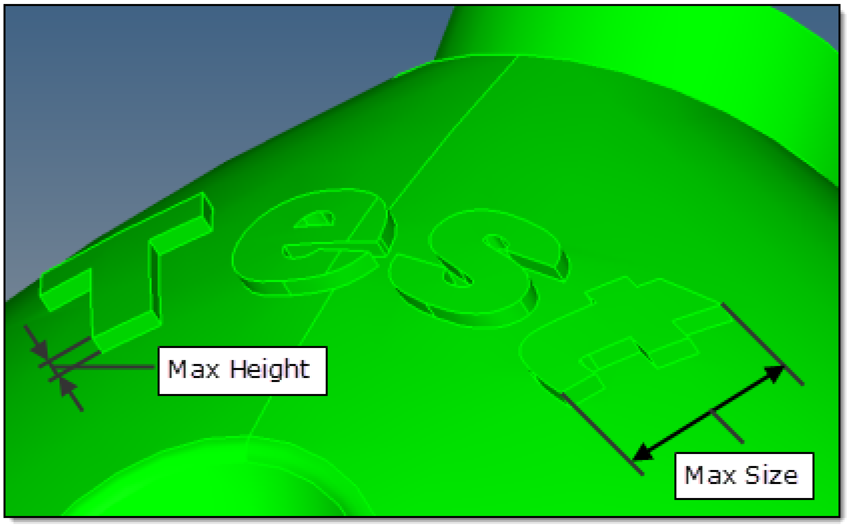


Figure 705: Logo Recognition Parameters

Remove logo with size <

Maximal size of a letter in the logo, as measured along/parallel to the "shiny" surface.

and height <

Maximal height/depth of a letter in the logo, as measured normal to the "shiny" surface.

Concavity factor

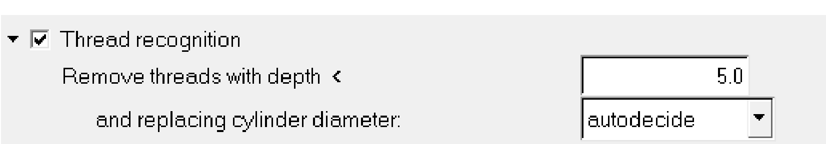
Creates a filter that provides more flexible control of automatic logo recognition. As this is a heuristic tool, it may remove real features, such as flat bottom round dimples, that were not intended for removal. The Concavity factor is a quantitative measure of a letters shape complexity, formally defined as:

$$concavity\_factor = \frac{contour\_accumulated\_turn\_angle}{360} - 1$$

The contour_accumulated_turn_angle is the sum of angles between a letters contour straight parts. Curved parts of a contour letter are approximated by a segmented line composed of short straight segments. For completely concave contour, such as circles, quads, and hexagons, concavity factor contour_accumulated_turn_angle = 360 degrees and concavity factor = 0.

> **Tip:** Extend the recognition and removal of a logo by reducing the **Concavity factor**.

Thread Recognition



When **Thread Recognition** is activated, geometry that represents threads is recognized and the below options become enabled.

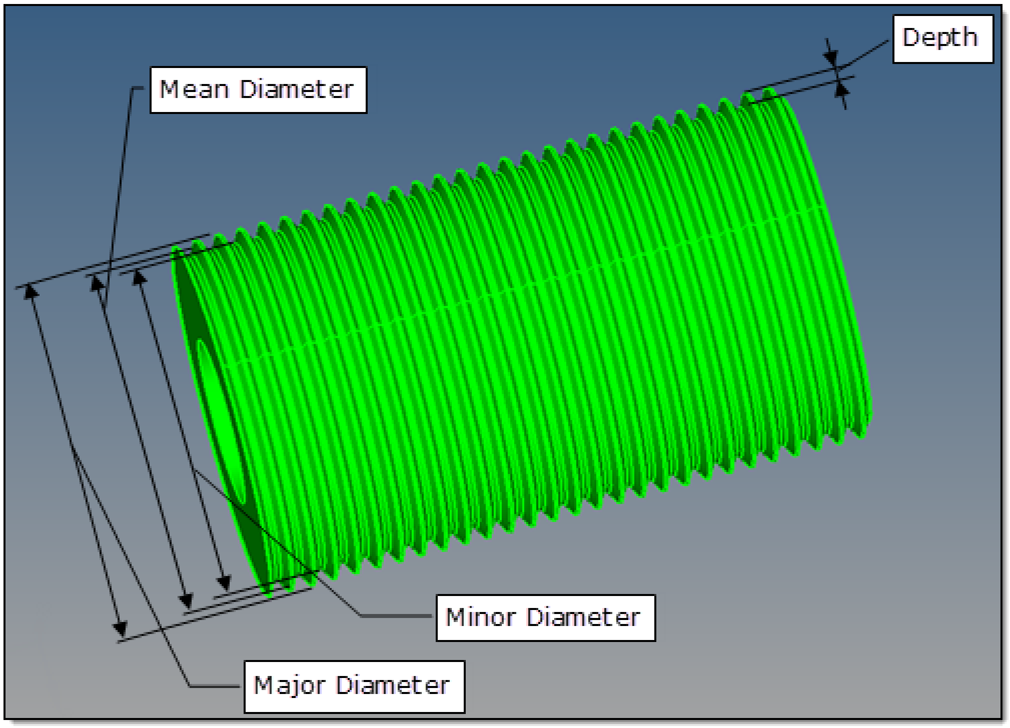


Figure 706: Thread Recognition Parameters

Remove threads with depth <

Remove cylindrical or conical threads with a depth less than the specified value, and replaces them with a smooth cylinder or cone surface.

and replacing cylinder diameter

Method used to define the diameter of the replacing cylinder or cone.

autodecide

Automatically determine diameter based on the diameter of a blank before thread cutting begins.

For inner (hole) threads, it corresponds to the thread minor diameter. For outer (bold) threads, it corresponds to the thread major diameter.

major

Use diameter of the thread major.

mean

Use diameter of the thread mean.

minor

Use diameter of the thread minor.

Other Options

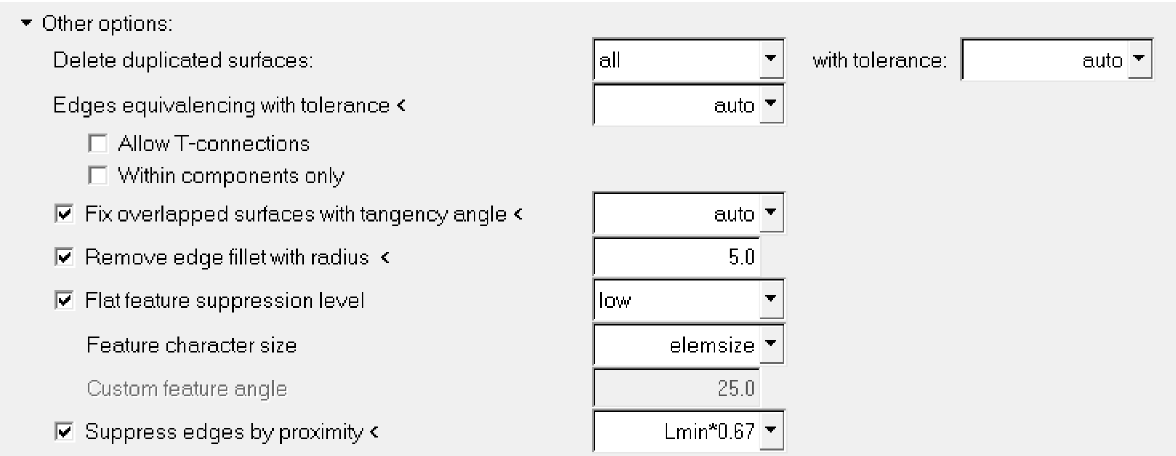


Figure 707:

When **Other Options** is activated, the following options become enabled:

Delete duplicated surfaces / with tolerance

Define which duplicate surfaces to delete before meshing.

For **Delete duplicated surfaces**, choose a method to find duplicate surfaces

All

Consider all of the surfaces in all of the components against each other.

Within components only

Consider all surfaces within components only. Duplicate surfaces between components are not found.

None

Do not remove duplicate surfaces.

For **With tolerance**, define the tolerance used when finding duplicates.

Auto

Automatically calculate the tolerance from the model size and other relevant geometric parameters.

<value>

Enter a tolerance. This is more useful when the auto tolerance is not sufficient to find all of the duplicates.

Edges equivalencing with tolerance <

Tolerance to use for equivalencing (stitching) edges, in conjunction with the options below.

auto

Calculate the tolerance internally.

<value>

Enter a tolerance. This is more useful when the auto tolerance is not sufficient to make all of the necessary connections.

Allow T-connections

Allow T-connections (non-manifold edges) to be created during the stitching process.

Within components only

Allow stitching only within components. Stitching between edges of different components is not allowed.

Fix overlapped surfaces with tangency angle <

Fix overlapping surfaces.

Auto

Calculate the tangency angle internally.

<value>

Enter a maximal tangency angle to fix overlapped surfaces.

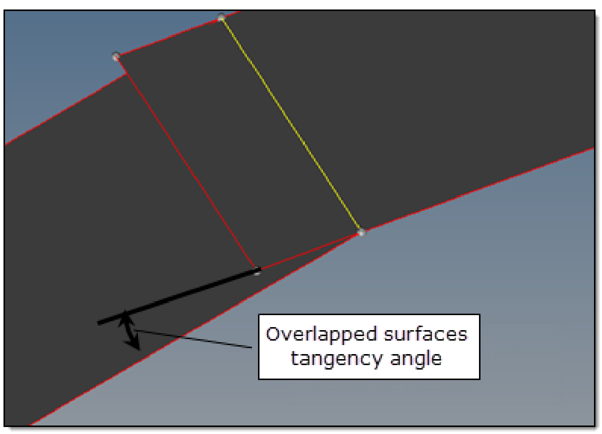


Figure 708: Overlapped Surfaces Tangency Angle

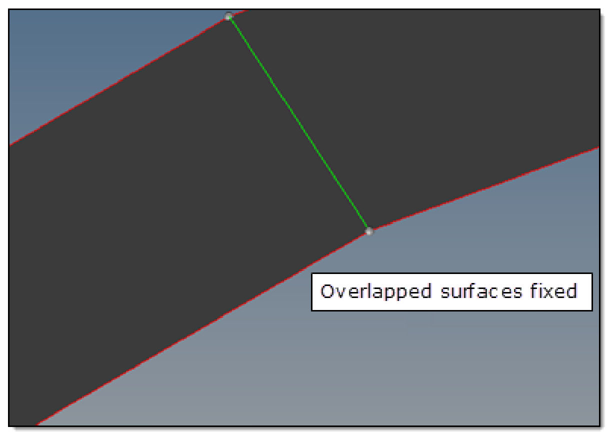


Figure 709: Overlapped Surfaces Fixed

> **Note:** This option may remove the surfaces that should not be deleted. For example, it may happen to surfaces with T-connections. Setting the angle to < 45 may help reduce such side effects.

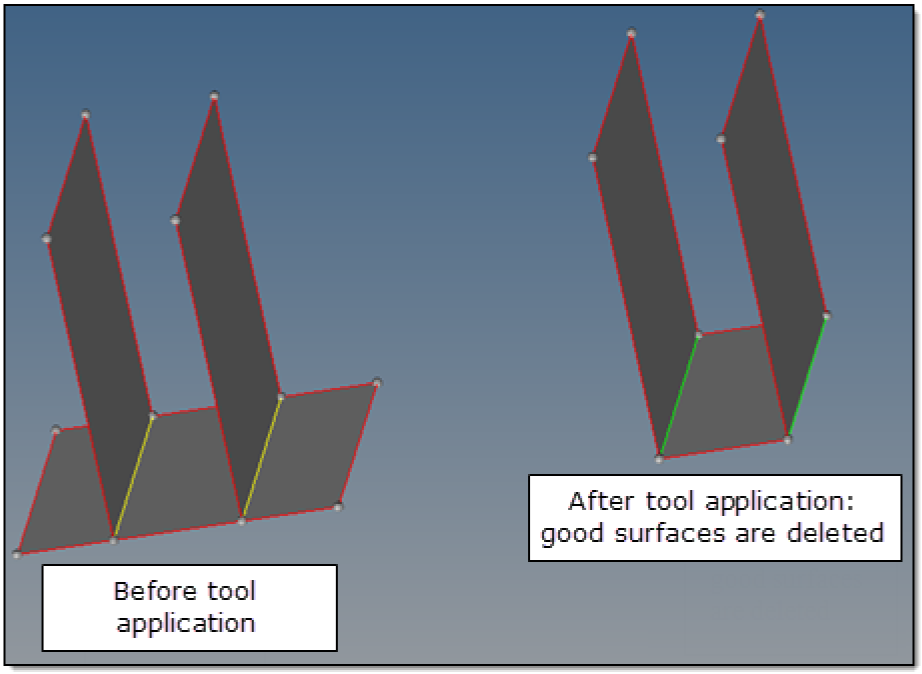


Figure 710: Possible Side Effects of Fixing Overlapped Surfaces

Remove edge fillet with radius <

Square off any fillets/rounded edges located on free edges and having radii below the specified value. This helps to create a good mesh pattern in such areas. For concave fillets, this means material is removed. For convex fillets, this means material is added.

Flat feature suppression level

Suppresses feature edges based on curvature break angle. For the ease of use, you can select a curvature break angle range, which varies from **very low** to **very high**.
The curvature break angle is calculated based on **Feature character size**.
Choose different levels of suppression from **very low** to **very high** for more flexibility and control over capturing feature edges. **very low** suppression level corresponds to keeping maximum feature edges, while **very high** suppression level subjects the geometry to more feature edge suppression.

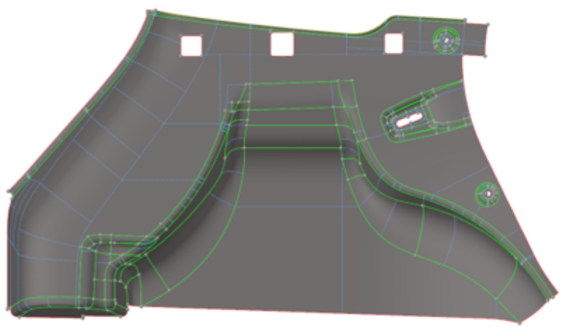


Figure 711: Flat feature suppression level: Very Low

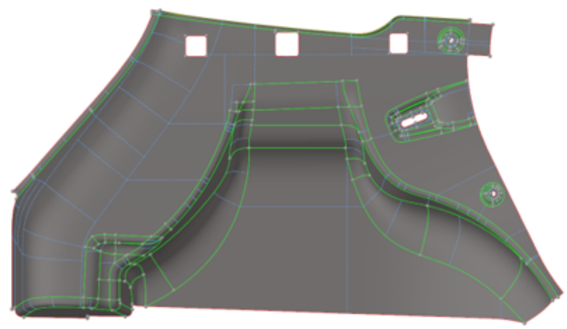


Figure 712: Flat feature suppression level: Very High

If you are not satisfied with the result, choose **user defined** to enter a custom angle in the **Custom feature angle** field.

> **Note:** The **user defined** and **recognized features** options are excluded from this suppression to enable you to capture and protect important features.

Feature character size

Calculate the curvature break angle, which is defined by an element size or calculated automatically based on characteristic dimensions of the part.

Custom feature angle

Custom feature angle.

Suppress edges by proximity <

Suppress full or partial feature edges within the defined proximity value.

This option allows geometry cleanup to consider a minimum element size defined in the criteria file, which helps to avoid minimum size quality failures. You can choose to enter an absolute value for proximity, or you can choose to use the minimum element size or its factor.

When two or more feature edges come in proximity the following guidelines or rules are used in general to determine which feature edge gets suppressed to get more consistent and predictable results:

- Full or partial feature edges within proximity are suppressed.

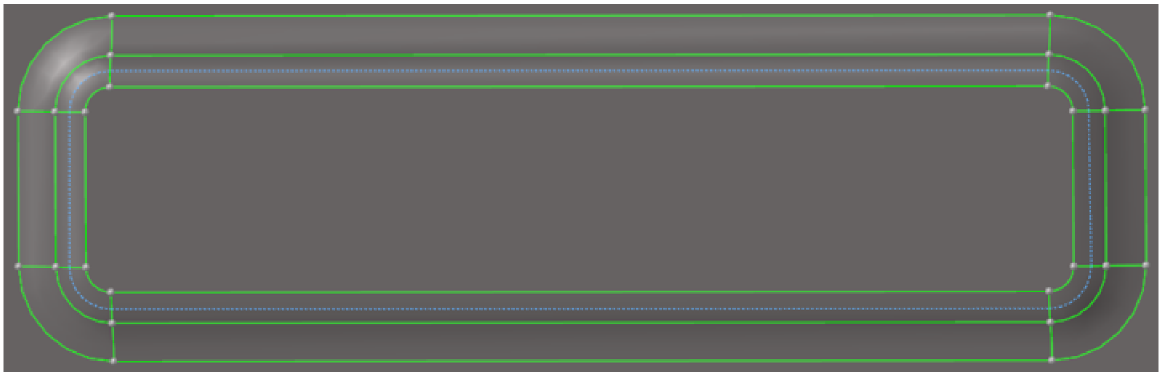


Figure 713:

- Feature edges that have higher curvature values are retained.

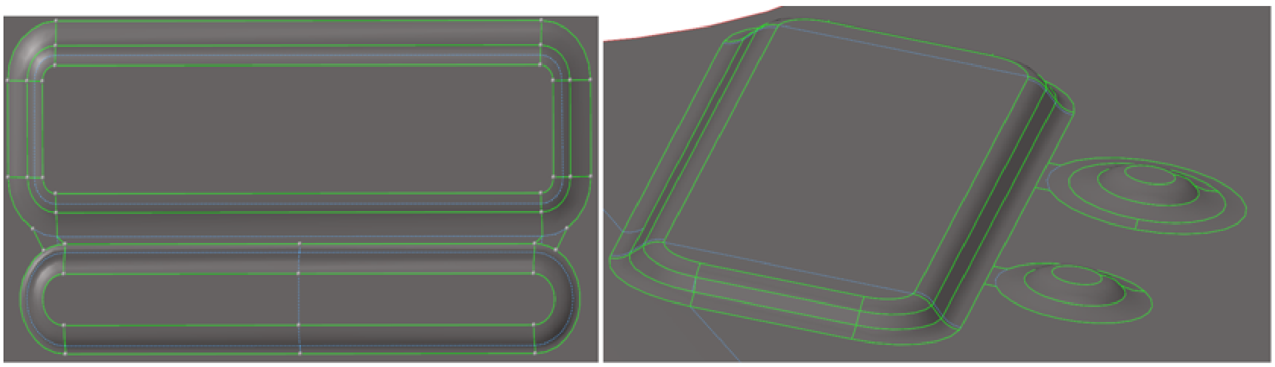


Figure 714:

- Boundary (free) edges are given priority.

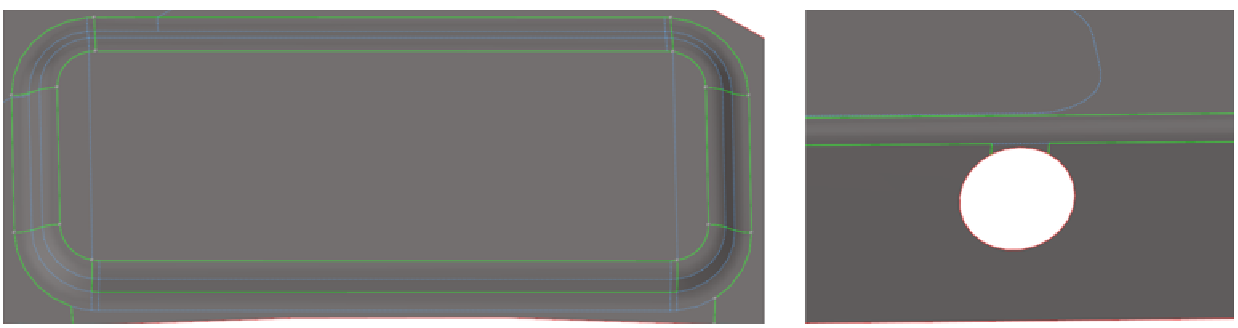


Figure 715:

- Base and top feature edges are given a priority while doing proximity cleanup for features like bead, bosses, and so on.

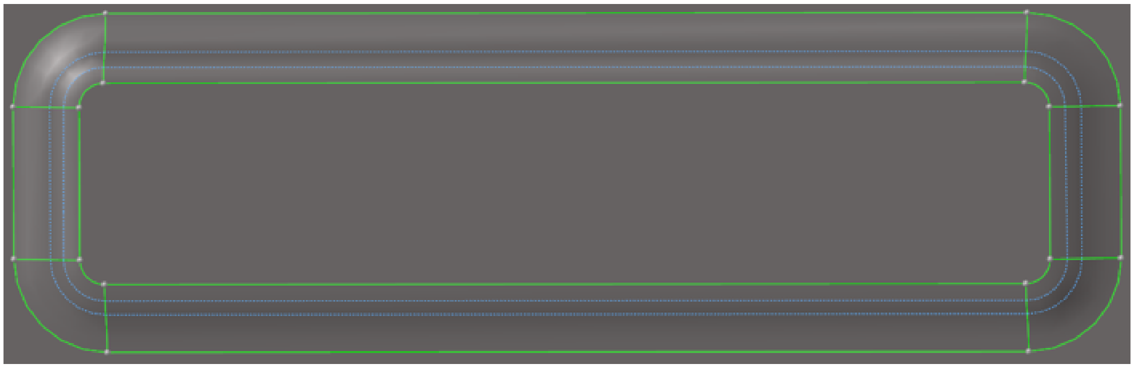


Figure 716:

> **Note:** The proximity value is generally kept less than the minimum element size considering node movement tolerance.

Preserve Boundaries between Components

Figure 717:

Preserve boundaries between components

Do not suppress or remove components' boundary edges during geometry cleanup, and do not move elements nodes across the components' boundaries. In some cases, maintaining boundaries for adjacent components that do not have any structural meaning would significantly worsen the element quality results.

Mesh Options

The Mesh Options parameters are used by Batchmesher to generate a mesh on the cleaned-up geometry.

This is one of the main functions of Batchmesher and is turned on by default. You can choose to turn off this parameter if you only want to perform geometry feature recognition and cleanup without meshing.

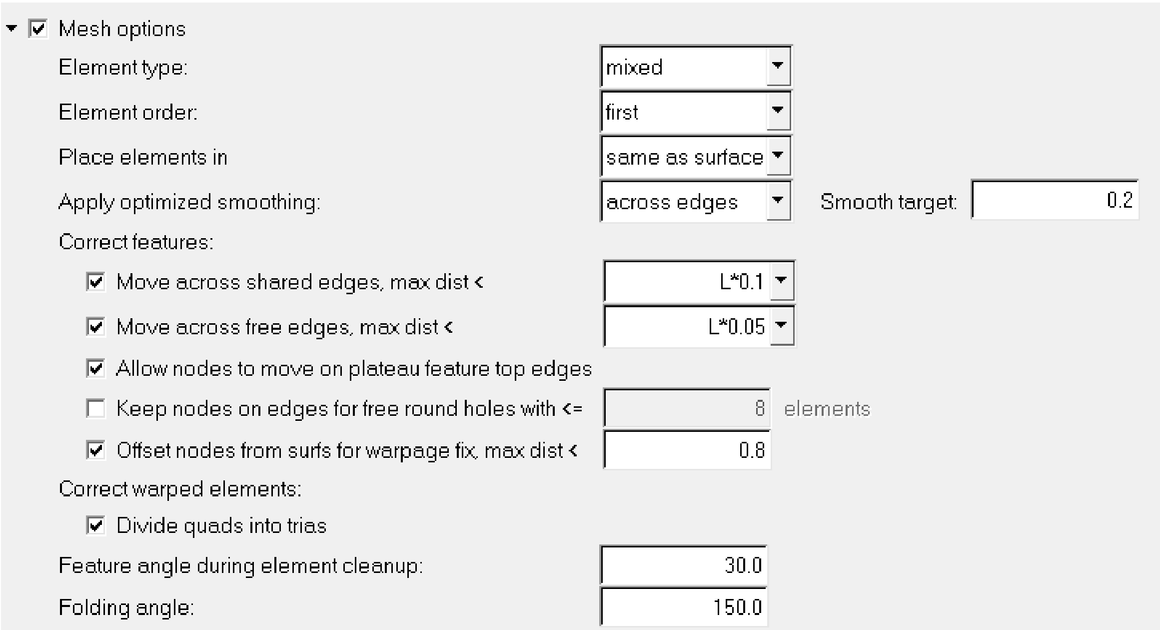


Figure 718:

Batchmesher has a powerful mesh flow algorithm which considers the shape of the geometry and aligns the mesh to create orthogonal meshes automatically. It also helps to reduce number of trias and places them strategically to avoid bad mesh patterns. Batchmesher is able to control the average element size in order to generate a more uniform mesh.

When **Mesh options** is activated, the following options become enabled.

These parameters control the behavior of the post-mesh element cleanup operations. They are intended to fix elements failing the quality criteria, to reduce number of tria elements for mixed/quad meshes, to correct bad mesh patterns, and to fix mesh flow for fillets. All of the element cleanup operations are compliant with the quality criteria, in that they should improve or at least not worsen the mesh quality.

All element cleanup behaviors are based either on nodal movement (smoothing), changing element connectivity (collapsing, splitting, and so on) or local remeshing.

Element type

Type of elements to create.

Element order

Create first or second order elements.

Place elements in

Organize new elements in either the current component or the original surfaces' component(s).

Apply optimized smoothing

After the surfaces are appropriately meshed, the nodes are optimized towards a target smoothing value to improve the element quality while maintaining geometry features.

none

Do not perform smoothing.

within surfaces

Smooth the nodes within surfaces. Nodes on surface edges are not moved.

along edges

Smooth nodes both within a surface and along edges. Nodes on edges are allowed to move only along the edge to improve element quality.

across edges

Smooth nodes both within a surface and across edges. Nodes on edges are allowed to move both along and across the edge to the neighboring surface to improve element quality.

Smooth target

A composite Quality Index rating, ranging from 0 (perfect elements) to 1.0 (failed elements). The default of 0.2 is ideal for most cases, producing elements of good quality without taking too long to optimize, but can be altered if necessary.

Move across shared edges, max dist <

Move nodes across or away from the geometry's shared edges by less than the specified distance.

Move across free edges, max dist <

Move nodes across or away from the geometry's free edges by less than the specified distance.

Allow nodes to move on plateau feature top edges

Do not allow nodes to move off the top/bottom edges of recognized embosses, particularly those containing central bolt holes to fix failed elements.

Figure 719:

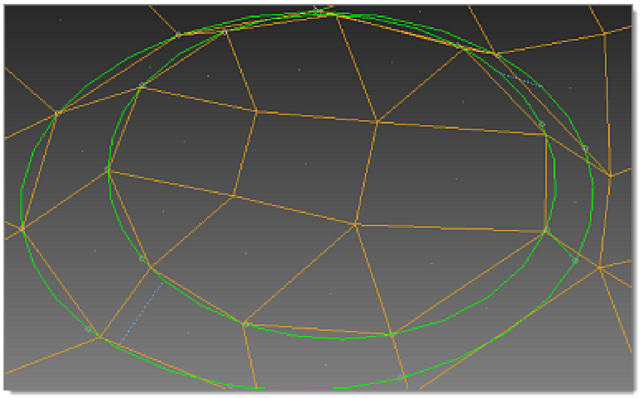


Figure 720: Allow Nodes to Move on Plateau Feature Top Edges = On

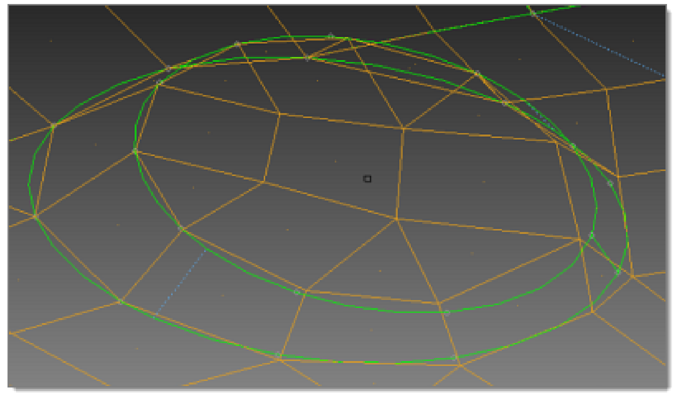


Figure 721: Allow Nodes to Move on Plateau Feature Top Edges = Off

Keep nodes on edges for free round holes with <=

Do not allow any nodes to move off the edges of free holes (without washers) with less than a specified number of elements. This is useful if distortion of the holes is not allowed.

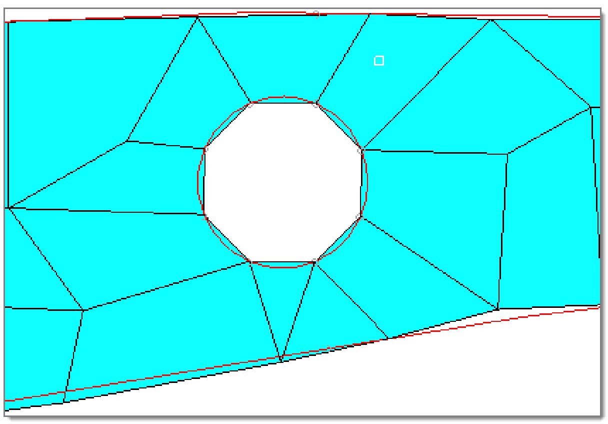


Figure 722: Keep Nodes on Edges for Free Round Holes with <= On

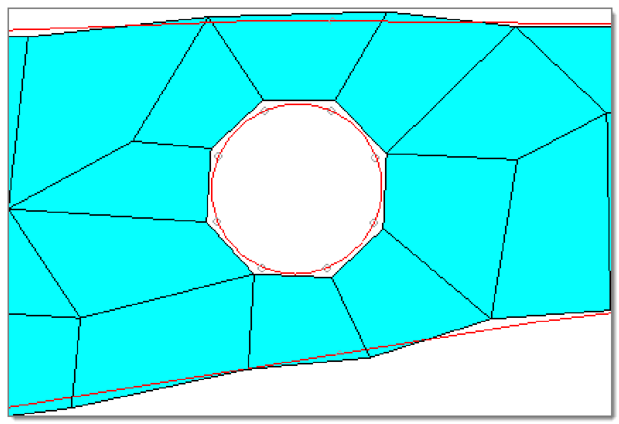


Figure 723: Keep Nodes on Edges for Free Round Holes with <= Off

Divide quads into trias

Split quads into trias to meet the element criteria defined in the criteria file.

Feature angle during element cleanup

Element feature angle to maintain while performing element cleanup.

Folding angle

Elements whose angle exceeds this value are considered folded over, and BatchMesher attempts to clean them up.

Special Component Selection

The Special component selection parameters define a method for selecting special components.

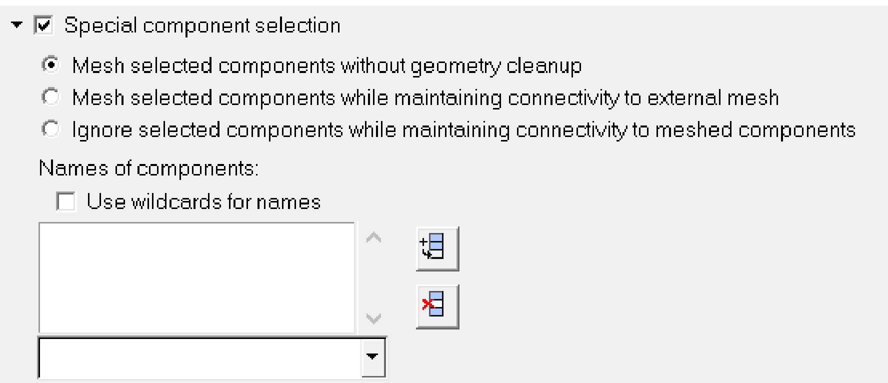


Figure 724:

When **Special component selection** is activated, the following options become enabled:

Mesh selected components without geometry cleanup

Mesh the listed components but will not perform any geometry cleanup on them before meshing. Any remaining components that are not listed will be batch meshed using the normal process, including geometry cleanup.

This is useful for models in which some components do not require geometry cleanup but the rest might. Models in which no components require cleanup can be batch meshed with the Geometry Cleanup checkbox turned off.

Mesh selected components while maintaining connectivity to external mesh

Mesh the listed components while maintaining connectivity to any existing mesh.

This is useful when components are to be meshed with multiple element sizes but transitions at the common edges of the different sizes are required. Each component should be meshed individually with its own parameter and criteria files with this option enabled.

Ignore selected components while maintaining connectivity to meshed components

Ignore the listed components while maintaining connectivity to any existing mesh. The mesh and geometry of the ignored components are not touched during batch meshing. The mesh created on other adjacent components is connected to any existing mesh on the ignored components.

This is useful for batch meshing of different components with different criteria/parameters files, or when pre-meshing components interactively or with some other procedure, followed by batch meshing of other components.

Click [icon] to add the name of a component specified in the drop down to the table. To provide a new name, select the empty entry in the drop down and type a new name. Click [icon] to remove the selected row from the table.

As an example, a model may have two components named front_10 and rear_20, which share common surface edges. The component front_10 is to be meshed with element size 10 and rear_20 with element size 20. This can be accomplished as follows:

1. Create two sets of parameter/criteria files.
   - The first should have a target element size of 10 and the appropriate parameters. In this parameter file, turn on the **Special component selection** option, **Mesh selected components while maintaining connectivity to external mesh** sub-option, and add front_10 to the component list.
   - The second file should have a target element size of 20 and the appropriate parameters. In this parameter file, turn on the **Special component selection** option, **Mesh selected components while maintaining connectivity to external mesh** sub-option, and add rear_20 in the component list.
2. Create a mesh type and assign the first set of criteria and parameter files.
3. Create a second mesh type with the same name as the first, and assign the second set of criteria and parameter files.
4. Choose the geometry file to be batch meshed, assigning it the mesh type from above, and submit the job.

This will mesh front_10 first with the first mesh type, and then take the results of this and mesh rear_20 with the second mesh type, while maintaining connectivity with the mesh created on front_10.

Guidelines and Recommended Practices

Criteria File

- When defining the Target element size, it is recommend to use either the Minimal height or the Minimal normalized height methods. These methods do not affect the quality of quads, and for the same mesh only the tria quality will be reduced with the minimum height method. Moreover, the use of the Shortest edge method may result in the creation of rhombus-like quads, which are only useful for special kinds of meshes.
- After the Target element size, the most important aspects are the Min Size and Max Size. These have the highest impact on the final mesh output.
  - The Min Size should not be too big with respect to the Target element size. The recommended size is 33% of the Target element size. On the upper end, a value of 40%-50% of the Target element size is also acceptable. A larger than recommended Min Size may trigger intensive cleanup that disrupts the mesh flow.
  - The Max Size should not be too small. A value of 175% of the Target element size is reasonable.

    - The range between Min Size and Max Size should not be too tight, otherwise there is limited ability to improve the mesh result.
- A Warpage value between 20°-25° is recommended. A value of 15° may be too tight, over constraining the mesh generation. With a value of 15° or lower, violating quads may have some nodes be moved away from the geometry, or may be split into two trias, depending on the parameter file settings.
- Skew, Min Angle, and Max Angle are dependent on each other. The Jacobian value also has an effect on the element shape. An element with a small Jacobian value, for example, will have poor angles. Calculating the Jacobian at the corner points is a more strict setting.
- Taper is used for orthogonality. In some way, it controls the same outcome that angles and Jacobian also account for. Taper is most useful for solvers that have a direct requirement for it.
- The reduction of tria elements is performed automatically by the Batchmesher, even when the criterion % of Trias is not enabled. Therefore, you will not see a large difference when changing this value with respect to the default Batchmesher result.
- Careful consideration should always be given when changing criteria values. As many criteria are interdependent, over-constrained values can lead to poor mesh flow and mesh adherence to the geometry is likely to be disturbed.

Parameters File

Midsurface Extraction

- The skin offset method is recommended whenever constant thickness parts without ribs/t-connections require mid-surface generation. This method is the fastest and most efficient for such models.

Geometry Cleanup - Surface Hole Recognition

- For improved mesh flow, an even number of elements should be specified. Usually, this value is set to comply with the min and max element size requirements. While not recommended, it is permissible to set this so that the element size violates the min or max element size restrictions. The use of the auto setting is not recommended for holes with washers.
    - The following formula can be used to calculate the resulting element size S given a hole radius R and number of elements around the hole N:

      $$S = 2R\sin\left(\frac{180}{N}\right)$$

      For example, a hole with a 2.5 mm radius and 6 elements around the hole will result in a mesh size of 2.5 mm.
    - To not have elements smaller than the min element size around holes, use a radius threshold for removal > = 0.708*min element size.
- Incompatible settings for holes with washers can lead to the creation of failed elements and/or outer washer ring elements with lengths significantly different from the target mesh size. This can lead to bad mesh flow and bad transitions to the regular mesh.
    - It is not recommended to set the number of elements for a hole with washers to less than 6.
    - Using minimal mode for # elems and auto mode for washers is recommended whenever possible, as it gives the most flexibility. With these settings, the outer washer element

size is close to the target element size, and the washer elements comply with the quality criteria. However, washers will not be built if it is not possible to create elements of permissible quality, or the outer washer ring element size appears significantly larger than the target element size. Another side effect is that as a last resort, a washer may be built with an odd number of elements in order to build a good quality washer.

- When specifying an exact number of elements for holes and/or a fixed number of elements, or radius fraction, for washers, consider the implication to the element quality, aspect ratio, min/max size, Jacobian, and the outer washer ring element size, using the formula as above.
- It is possible to create washers with failed elements and/or with an outer washer element size different from the target element size. This is done by defining washers as a fixed number or as a radius fraction. Use exact mode for # elems if it is expected that the outer ring element size is more than 140% or less than 60% of the target element size. In addition, when a hole is set to high priority, washer elements are not modified to correct for failed element quality. If a hole is set to normal priority, washer nodes are allowed to move to correct the quality.

- Use the Attempt to maintain narrow slots rounded ends using >= 6 elements option only if absolutely necessary. Meshing of small areas produced by this option will often have a negative impact on the mesh flow.

Geometry Cleanup - Solid Hole Recognition

- This should be disabled for models having only shell geometry, as the recognition of the solid holes is potentially time consuming.
- Define even numbers of elements around holes, trying to comply with the min element size criterion and trying to have the element size as close to the target element size as possible.

Geometry Cleanup - Surface Fillet Recognition

- The following formulas define how the fillet geometry parameters and the mesh density are related:

$$W = R\alpha;\ W_{ch} = 2R\sin\frac{\alpha}{2};\ S = 2R\sin\frac{\alpha}{2N};\ D = R\left(1 - \cos\frac{\alpha}{2N}\right);$$

W
: Fillet arc width

R
: Fillet radius

α
: Fillet arc angle (radians)

W_{ch}
: Fillet chordal width

S
: Element size across the fillet

N
: Element density across the fillet

D

Chordal deviation

- Enabling the Minimize transitions option is highly recommended. However, in very rare occasions it may create rows of strongly skewed elements. This may happen for fillets having multiple irregular fixed points on the fillet sides, or multiple intersections of those edges. For such cases, it may be disabled to improve the result.

Geometry Cleanup - Flange Recognition

- It is recommended to use a minimum of two elements across the flange width. The actual number of elements is determined internally by accounting for restrictions on the min and max element sizes, as well as the aspect ratio.
- A reasonable upper limit for the maximum flange width is around (N + 1.5) * element size, where N is the number of elements across the flange. Similarly, a reasonable lower limit is two times the min element size.

Geometry Cleanup - Bead Options

- The **Suppress beads with height <** value is dependent on specific modeling requirements. Typically, this is set around 20% of the target element size, but not larger than the criteria minimal element size.

Geometry Cleanup - Logo Options

- For logo removal, a Concavity factor value of 2.0 is recommended for logos with complex lettering. For logos with simple shapes, though, a value of 1.0 or even 0.0 may be used. In this case, to reduce the danger of removing valid features, assign height < a value no greater than the Suppress beads with height < value.

Geometry Cleanup - Other Options

- The Remove edge fillet with radius < recommended upper limit is between the min element size and the target element size.
- Flat feature suppression level is recommended to be set at “low” with “element size” as Feature character size.
- Suppress edges by proximity value is generally kept less than minimum element size considering node movement tolerance.

Meshing Cleanup

- For most applications, it is recommended to use the mixed type for Create mesh with element type.

  > **Note:** When the quad type is set, a specific algorithm is used which may have negative effects on the mesh flow and may create undesirable mesh patterns, for example, three quads sharing a node.

  - Enabling both the align and size mesh flow options is highly recommended, as they have minimal negative side effects.
- All the feature edges kept after geometry cleanup are respected during meshing and post meshing cleanup. Nodes are moved only for the failed elements considering allowable limit defined in parameters controlling the node movement (across shared and free edges).

- The method chosen for Apply optimized smoothing effects the capabilities of all post-mesh element cleanup operations. The across edges mode is set by default and is strongly recommended, as it allows for the most flexibility in correcting the mesh. Other options are likely to result in a high rate of failed elements because:
    - Geometry cleanup and defeaturing cannot address all problems.
    - The across edges mode attempts cleanup steps in the following order:
        - Move nodes only along surface edges.
        - Move nodes only across non-feature edges.
- Remaining failed elements are fixed using small constrained node movements for feature edges, including strong feature edges. The only exception is that user preserved edges are excluded from such movement. The Correct features parameters provide control over these movements.
    - For Move nodes across shared edges, max dist <, the suggested value is 10% of the target element size.
    - For Move nodes across free edges, max dist <, the suggested value is 5% of the target element size.
- For correcting warpage of quad elements, it is recommended to enable the Offset nodes from surfs, max dist <, and Divide quads into trias options.
    - The recommended value for the offset distance is 10% of the target element size.
    - When both options are enabled, warped elements are first fixed by nodal movements normal to the geometry, followed by splitting any remaining failed quads.
- The recommended value for Feature angle during elements cleanup is 20-30. A lower value may help with better feature representation, but a value lower than 15 may significantly increase the rate of failed elements.

SPH Meshing

Smooth Particle Hydrodynamics (SPH), Finite Point Method (FPM) is a technique used to analyze bodies that do not have high cohesive forces among themselves and undergo large deformation, such as liquids and gases.

Typical applications that use SPH FPM include: airbag modeling in crash, fuel tank slosh, bird strikes, and explosion analysis.

In SPH FPM, a given volume of the body of interest is discretized into particles, called SPH elements. SPH elements (called particles) are like nodes which do not have any geometric connectivity among themselves. Each SPH element has an effective mass. The mass sum of all particles in the filled volume of the body should be equal to the mass of the filled volume.

SPH elements are currently supported as 0D MASS elements.

0D Elements

Supported 0D elements.

0D elements are essentially mesh nodes with an additional value attached to them.

SPH Elements

Node-like particles which have no geometric connectivity among themselves. Each SPH element has an effective mass.

Mass Elements

Configuration 1 - 0D elements with a single node that allow you to assign concentrated mass to the model in order to represent a physical part that may not be modeled with another FE idealization. Masses have the ability to store one node, a value of mass, and a property reference.

Mass elements are displayed as a dot with the letter M written at the centroid of the element.

SPH Element Mapping

The generated SPH elements are mapped specific cards.

Radioss

- SPHCEL

- A new property card PROP/SPH is also created.

LS-DYNA

- *ELEMENT_SPH

PAM-CRASH 2G

- SPHEL

Abaqus

- *SECTION CONTROLS

Create SPH Mesh (LS-DYNA, Radioss, PAM-CRASH 2G)

Create a SPH mesh using existing elements, components, surfaces, or solids in your model.

Before you begin, it is recommended that you create a new component collector before starting the SPH mesh generation so that the generated mesh is stored in a separate collector.

> **Restriction:** The SPH panel is only available in the LS-DYNA, Radioss, PAM-CRASH 2G solver interfaces.

1. From the 1D page, click **sph**.
2. Use the entity selector to select the input which defines the volume to be filled with SPH elements.

   Elements, components, surfaces and solids are supported as input to the SPH mesher. Selected elements can be shell or solid elements; however, the selected elements need to form a closed volume.

   Selected components can contain either a FE mesh or geometry; however, the FE mesh or geometry must also form a closed volume. More than one component can be selected to form a closed volume for SPH meshing. If the selected components are such that one is completely contained within another, SPH elements are created within the volume between the two selected components.

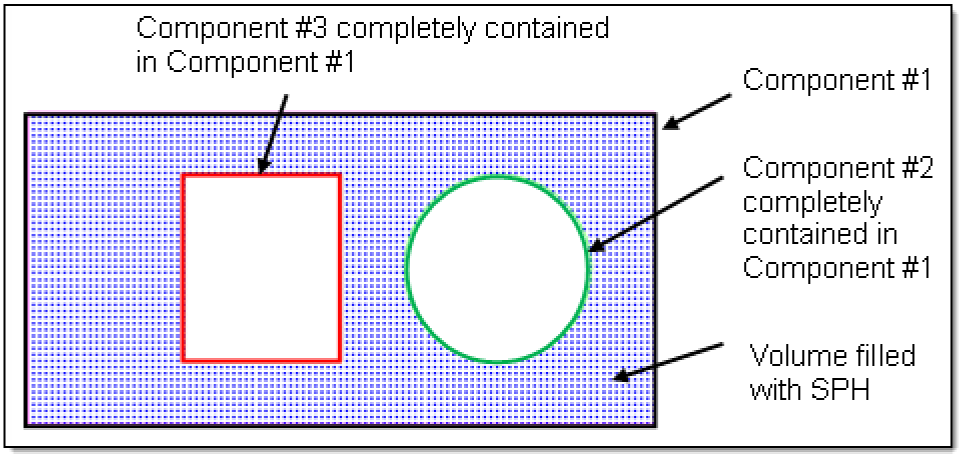


Figure 725: SPH Elements Created within the Volume between Two Selected Components.

3. To specify which point the generation of cubes should be started, select the **use reference** checkbox.

   SPH elements are generated at the corners/face centers of the cubes which fall within the user defined criteria

   - Choose **global origin** to use the default coordinates (0,0,0) for the reference point.
   - Choose **local origin** to manually enter node coordinates for the reference point.
4. Under mesh orientation, define the orientation of SPH elements.
   - Choose **global system** to use the default global system to align generated SPH elements.
   - Choose **local system** to manually select reference systems local to the model orientation. Generated SPH elements are aligned using the user defined local system.
5. Under pitch, select a mesh type and enter a pitch value.

   Pitch is the distances between each SPH particle. Smaller numbers will result in more elements within the same space, but this will not affect the mass or density of the substance (gas, fluid, and so on) that the particles represent.

   - Choose **simple cubic** to arrange SPH particles in groups of 8, each particle being a corner of a cube.
   - Choose **face centered cubic** to arrange the particles in groups of 14, forming the corners and the center of each face of a cube.

     > **Note:** This is similar to a hexagonal close packed (HCP) structure and is recommended for use in Radioss models.

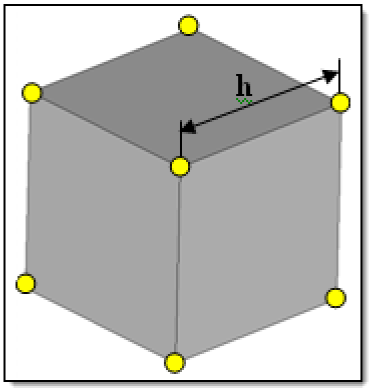


Figure 726: Pitch (h) for Simple Cubic

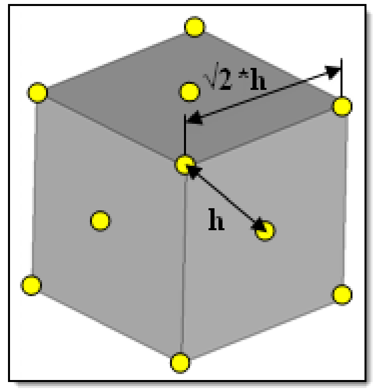


Figure 727: Pitch (h) for Face Centered Cubic

6. Define the quantity of fluid.

   Each SPH element must have an effective mass. The effective mass of each SPH element is determined by entering either a material density of the material which fills the volume; or the total mass of the material in the filled volume, called the filled volume mass.

   - Choose **material density** to enter its density.

     The SPH mesher determines the effective mass for each SPH element from material density, volume to be filled with SPH elements, and number of SPH elements generated.

     $$Mass_{SPHelement} = \frac{Volume\ Filled\ with\ SPH_{element}}{Number\ of\ SPH_{element}\ generated}\rho_{material}$$

   - Choose **filled volume mass** to enter its total mass.

     The SPH mesher calculates effective mass for each SPH element from filled volume mass, and number of SPH elements generated.

     $$Mass_{SPHelement} = \frac{Filled\ \text{Volume Mass}}{Number\ of\ SPH_{element}\ generated}$$

7. Under volume definition, specify which elements to generate SPH elements for.
   - Choose **all** to generate SPH elements in all of the volumes in the model.
   - Choose **enclosed** to generate SPH elements in the volumes enclosed by the defined nodes, and ignores the remaining volumes.
   - Choose **Nth Largest** to enter which volumes to generate SPH elements in by defining the wrap size index in terms of volume size.

     To specify the largest volume, enter 1 in the index field; to specify the second largest volume, enter 2 in the index field.
   - Choose **exclude enclosed** to ignore the volume(s) enclosed by the defined nodes and generate SPH elements in the remaining volumes.
8. Optional: Model a fluid or gas that does not completely fill the selected volume.

a) Select the **partial fill** checkbox.
b) Enter either a percentage or depth of the volume to fill.

   Calculation of the volume is based on the lowest point of the model, parallel to the user defined plane.
c) Use the plane and vector selector to specify the direction of fill, which is generally the opposite of the direction of gravity when the filled volume is installed in the real world.
d) If the particle mass is filled along the correct axis, but in the wrong direction (for example from the top of a fuel tank downward), select the **reverse direction** checkbox to resolve this issue.

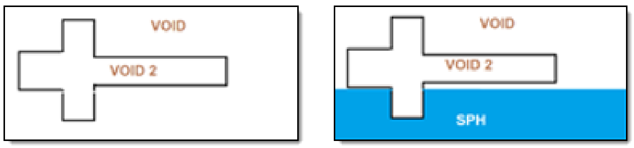


Figure 728: Partial Fill

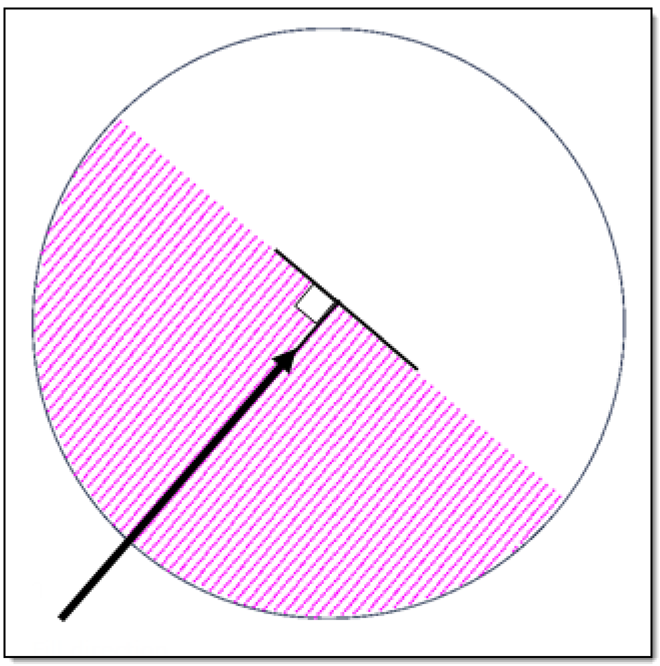
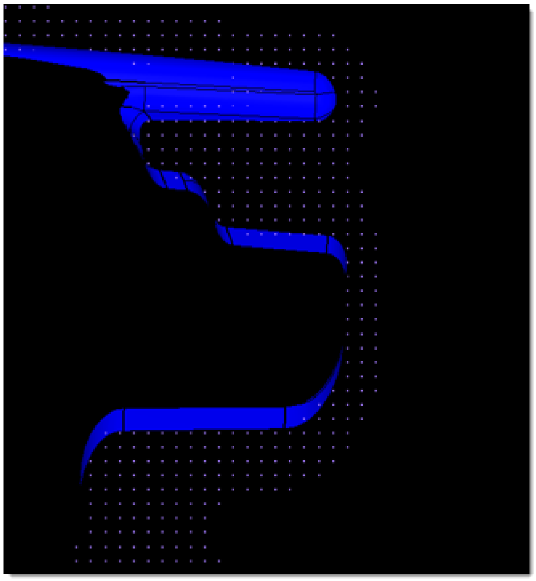


Figure 729: Fill Direction

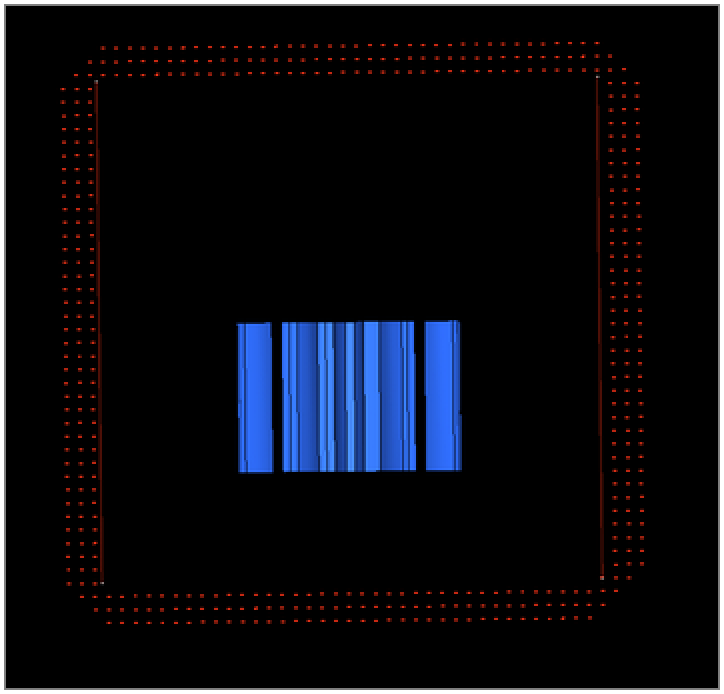
9. Optional: Create SPH particles up to a distance that you specify.

The thickness of SPH elements is created from input. The distance between the SPH particles is driven by the pitch.

a) Select the **wall offset** checkbox.
b) Enter a distance to offset.



c) To create SPH particles outside of the defines volume, select the **external to volume** checkbox.



10. To create SPH particles from a specified distance, enable the **wall clearance** checkbox.

This option is useful when you are trying to avoid contact of SPH elements with walls at the beginning of the solver run (1st iteration) and want the solver to run smoothly.

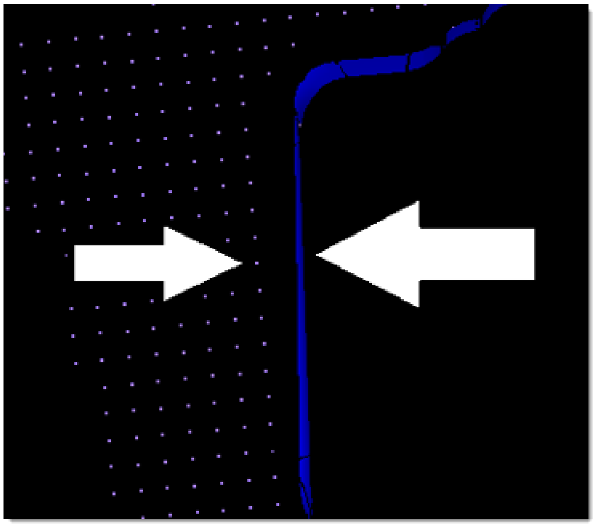


Figure 730:

11. Click **create**.

> **Tip:**
>
> Generated SPH elements (particles) are elements of mass configuration. In the modeling window, SPH elements have a spherical representation and possibly an element handle with an additional label of the element's configuration. To improve visualization of the generated SPH elements (particles), turn off the display of element handles by clicking on the Display toolbar.

Create SPH Mesh (Abaqus)

Create a SPH mesh using existing components in your model.

> **Restriction:** The SPH tool is only available in the Abaqus solver interfaces.

1. From the menu bar, click **Tools** > **SPH**.
   The **SPH Meshing Tool** opens.
2. Select the component(s) to generate SPH elements for.
   a) Click the **components** selector.
   b) In the panel area, click **comps**.
   c) Select the component(s).
   d) Click **select**.
   e) Click **proceed**.

   The components you selected are displayed in the Component Name field.
3. Click the Meshing Method field, and select a meshing method.

- Choose **Simple cubic** to arrange the SPH particles into groups of 8, with each particle being a corner of a cube.
- Choose **Face centered cubic** (FCC) to arrange the SPH particles into groups of 14, forming the corners and the center of each face of a cube.

  This method is similar to a hexagonal close packed (HCP) structure.
- Choose **Node to SPH** to manually select a node and convert it into a SPH element.

4. In the Pitch field, enter the distance you would like placed between each SPH element in the component.

   > **Tip:**
   >
   > Smaller pitch values will result in the creation of more elements within the same space, whereas, larger pitch values will result in the creation of less elements.

5. In the Material Density field, a density that will be used to generate the material and property.
6. To create a *solid section and *material keyword for each component that you mesh, select the **Create Property and Material** checkbox.
7. To create a *surface (node based) for each component that you mesh, select the **Create surface** checkbox.

   HyperMesh uses this surface later for contact definition.
8. In the Selection column, select the checkbox of the component(s) you wish to mesh.

   > **Notice:**
   >
   > In the Status field, notice the red status boxes. A red status box indicates that the component has not been meshed.

9. Click **Mesh**.

SPH elements are generated for the selected component(s).

> **Notice:** In the Status field, the components that were meshed now display a green status box. A green status box indicates that the component has been meshed. If the Status field contains a yellow status box, this indicates that the component you meshed already contains SPH elements.

If you would like to remove components from the SPH Meshing tool, select their corresponding checkbox in the Selection field, and then click .

Line Meshing

1D mesh that allows accurate testing of connectors, such as bolts, and similar rod-like or bar-like objects that can be modeled as a simple line for FEA purposes.

1D Elements

Supported 1D elements.

Bar Elements

1D elements created in a space between two or three nodes of a model where beam properties are desired.

The nodes are related to each other based on the properties of the bar or beam element connecting them. Properties associated with bar elements include vector orientation, offset vectors that end at A and B, or at A, B, and C, and pin flags to tell it what degree of freedom should carry through the beam.

Bar elements are displayed as a line between two nodes with BAR2 or BAR3 written at the centroid of the element.

Bar2

Configuration 60 - 1D (1st order) elements with 2 nodes used to model axial, bending, and torsion behavior. Bar2 elements have a property reference, an orientation vector, offset vectors and ends A and B, and pin flags at ends A and B.

Bar3

Configuration 63 - 1D (2nd order) elements with 3 nodes used to model axial, bending, and torsion behavior. Bar3 elements have a property reference, an orientation vector, offset vectors and ends A and B, and pin flags at ends A and B.

Gap Elements

Configuration 70 - 1D elements created in a space between two nodes, or between a node and an element, of a model where contact may occur.

Create a gap element when you want to impose a nonlinear constraint on a model; this constraint will limit the amount of movement possible during analysis.

Gap elements have a property reference and an orientation vector.

Gap elements are displayed as a line between two nodes with GAP written at the centroid of the element.

Gap elements can translate to CGAP or CGAPG elements in OptiStruct, CGAP element in Nastran or *GAP option in Abaqus.

Joint Elements

Configuration 22 - 1D elements with 2, 4, or 6 nodes which have a property and orientation systems or nodes.

Joint element is a definition of a connection between two rigid bodies. Joint elements store a property and orientation information.

Joint elements are displayed with lines between the appropriate nodes and the letter J between nodes 1 and 3 of the element.

Only certain types of elements can be used to create joint elements. The type of the element controls the number of nodes used in the element and the permissible orientations of the element.

Table 194: Types of Joint Elements

Type	Type Name	Number of Nodes	Orientation	Solver Interface
1	Spherical joint	2	None Systems Nodes	LS-DYNA PAM-CRASH
2	Revolute joint	4	None Systems	LS-DYNA
3	Cylindrical joint	4	None Systems	LS-DYNA
4	Planar joint	4	None Systems	LS-DYNA
5	Universal joint	4	None Systems	LS-DYNA
6	Translational joint	6	None Systems	LS-DYNA
7	Locking joint	6	None Systems	LS-DYNA
8	Ball joint	2	None	OptiStruct
9	Fixed joint	2	None	OptiStruct
10	Revolute joint	2	Node Vector Coordinates	OptiStruct

Type	Type Name	Number of Nodes	Orientation	Solver Interface
11	Translational1 joint	2	Node Vector Coordinates	OptiStruct
12	Cylindercal 1 joint	2	Node Vector Coordinates	OptiStruct
13	Universal joint	2	Node Vector Coordinates	OptiStruct
14	Constant_velocity joint	2	Node Vector Coordinates	OptiStruct
15	Planar joint	2	Node Vector Coordinates	OptiStruct
16	Inline joint	2	Node Vector Coordinates	OptiStruct
17	Perpendicular joint	2	Node Vector Coordinates	OptiStruct
18	Parallel axes joint	2	Node Vector Coordinates	OptiStruct
19	Inplane joint	2	Node Vector Coordinates	OptiStruct

Type	Type Name	Number of Nodes	Orientation	Solver Interface
20	Orient joint	2	Node Vector Coordinates	OptiStruct
21	Point_to_curve joint	2	Node Vector Coordinates	OptiStruct
22	Curve_to_curve joint	2	Node Vector Coordinates	OptiStruct
23	Point_to_deformable joint	2	Node Vector Coordinates	OptiStruct
24	Point_to_deformable joint	2	Node Vector Coordinates	OptiStruct
25	Translational_2N joint	2	None Systems	PAM-CRASH
26	Revolute_2N joint	2	None Systems	PAM-CRASH
27	Cylindrical_2N joint	2	None Systems	PAM-CRASH
28	Universal_2N joint	2	None Systems	PAM-CRASH
29	Flexion-Torsion joint	2	None Systems	PAM-CRASH
30	Planar_2N joint	2	None	PAM-CRASH

Type	Type Name	Number of Nodes	Orientation	Solver Interface
			Systems	
31	General joint	2	None Systems	PAM-CRASH
32	Bracket joint	2	None Systems	PAM-CRASH
33	Free joint	2	None Systems	PAM-CRASH

Plot Elements

Configuration 2 - 1D elements with 2 nodes used for display purposes.

Plot elements are displayed as a line between two nodes.

RBE3 Elements

Configuration 56 - Rigid elements with one dependent node and variable independent nodes typically used to define the motion at the dependent node as a weighted average of the motions at the independent nodes.

Both the dependent node and independent nodes contain a coefficient (weighting factor) and user-defined degrees of freedom. The dependent degrees of freedom and weighting factors can be specified or automatically calculated based on the geometry.

RBE3 elements are displayed as lines between the dependent node and the independent node(s) with RBE3 displayed at the dependent node of the element.

RBE3's are typically used to distribute loads applied on the dependent node amongst the selected independent nodes.

> **Note:** The dependent node cannot be directly constrained, as this would lead to a double-dependency for that node.

Rigidlink Elements

Configuration 55 - Rigid elements with one independent node and variable dependent nodes typically used to model rigid bodies.

Rigidlink elements have user-defined degrees of freedom which apply to all dependent nodes.

Rigidlink elements can be created with dependent nodes attached to an element as a SET. If a rigid link with a dependent node set is deleted, the associated node set is also deleted. If the dependent node set is deleted, the connected rigid link element is also deleted. Dependent node sets are automatically created when rigid link elements are created. A node set can be connected as a set of dependent nodes to a rigid link element independent node.

> **Note:** Two-node rigids with a dependent node set attached are always created as rigid link elements

Rigidlink elements are displayed as lines between the independent node and the dependent node(s) with RL displayed at the independent node of the element.

Rigid Elements

Configuration 5 - Rigid 1D elements with 2 nodes used to model rigid connections.

Rigid elements are displayed as a line between two nodes with the letter R written at the centroid of the element.

Rigids can translate to RBE2 in Nastran or *MPC in Abaqus.

Rod Elements

Configuration 61 - 1D elements with 2 nodes used to model axial behavior only.

The two nodes are related to each other based on the properties of the rod element connecting them. Rod elements have property pointers.

Rod elements are displayed as a line between two nodes with ROD written at the centroid of the element.

Rods can translate to CTUBES in Nastran or a C1D2 element in Abaqus.

Spring Elements

Configuration 21 - 1D elements used to model spring connections.

Spring elements have user-defined degrees of freedom, an orientation vector, and a property reference.

Spring elements are displayed as a line between two nodes with the letter K written at the centroid of the element.

Spring
: 1D elements with 2 nodes used to model spring connections.

Spring2N
: 1D elements with 2 nodes used to model spring connections.

Spring3N
: 1D elements with 3 nodes used to model spring connections.
The third node serves as the direction node.

Spring4N
: 1D elements with 4 nodes used to model spring connections.
This type of element will mostly be considered as joints, based on the property it is assigned.

Springs can translate to CELAS2 in Nastran or *SPRING in Abaqus.

Weld Elements

Configuration 3 - Rigid 1D elements with 2 nodes used to model welded connections.

Weld elements are displayed as a line between two nodes with the letter W written at the centroid of the element.

Surface Meshing

A surface mesh or "shell mesh" represents model parts that are relatively two-dimensional, such as sheet metal or a hollow plastic cowl or case.

Surface meshes placed on the outer faces of solid objects are used as a baseline mapping point when creating more complex 3D meshes (the quality of a 3D mesh largely depends on the quality of the 2D mesh from which it is generated).

2D Elements

Supported 2D elements.

Tria Elements

2D triangular elements.

Tria3

Configuration 103 - 2D (1st order) triangular elements with 3 nodes ordered in HyperMesh.

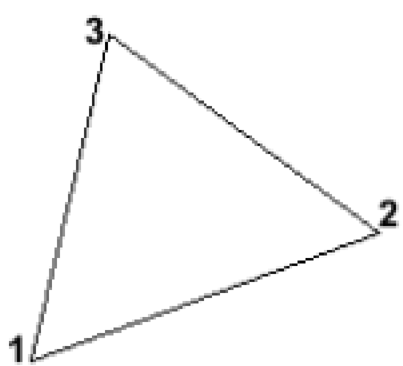


Figure 731: Element Configuration 103, 3-Noded Tria

Tria6

Configuration 106 - 2D (2nd order) triangular elements with 6 nodes ordered in HyperMesh.

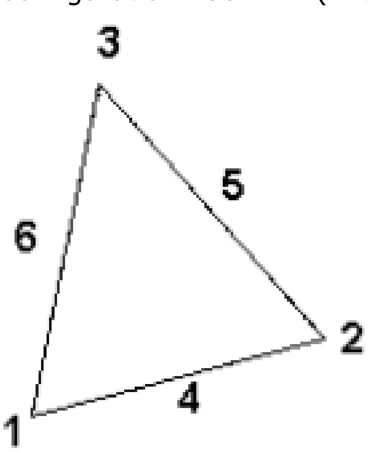


Figure 732: Element Configuration 106, 6-Noded Tria

Quad Elements

2D quadrilateral elements.

Types of pyramid elements include:

Quad4

Configuration 104 - 2D (1st order) quadrilateral elements with 4 nodes ordered in HyperMesh.

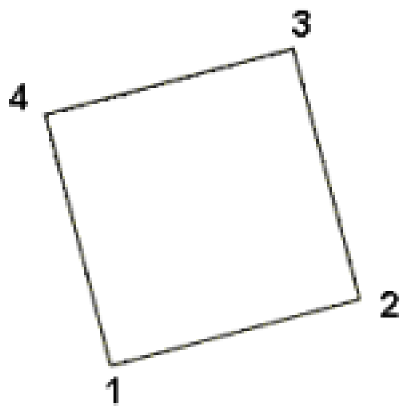


Figure 733: Element Configuration 104, 4-Noded Quad

Quad8

Configuration 108 - 2D (2nd order) quadrilateral elements with 8 nodes ordered in HyperMesh.

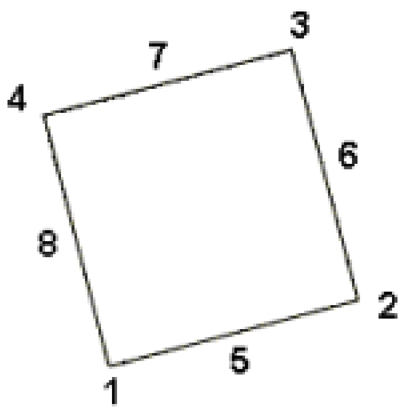


Figure 734: Element Configuration 108, 8-Noded Quad

Automatic Meshing

Automatic meshing generates a mesh of plate elements according to your specifications using surface geometry or existing shell elements.

Mesh Modes

Mesh modes are used during automatic meshing to determine the type of meshing to perform.

Size and Bias

Uses elements of a uniform size that you specify to create mesh. When using existing finite elements as the basis for mesh generation, feature recognition settings allow the mesher to break up the areas defined by the selected elements into logical groupings with mesh controls set for each group boundary.

QI Optimize

An iterative automatic mesh generation method driven by element quality criteria. During the mesh generation process, the quality index of the mesh is determined by evaluating each element against a set of element quality tests. If all required element quality criteria are passed, then that element has a perfect quality index of zero. As the element quality deteriorates, the quality index value increases; so a lower quality index score indicates an element more closely meets the ideal quality requirements.

The compound quality index is the sum of the quality index values for each of the elements that are included in the current meshing area. The quality index value itself has no direct physical meaning; it is a way to compare one generated mesh pattern against another pattern generated for that same area. The quality index based mesh optimization routine attempts to modify the mesh pattern and apply node smoothing routines to obtain a lower overall quality index value.

Edge Deviation

Determines how far the mesh elements can deviate from the actual edges of the surfaces meshed, or when in the case of re-meshing elements, deviation from inferred edges based on features. Edge deviation normally occurs on curved edges, because individual elements have straight edges and therefore can only approximate a curve.

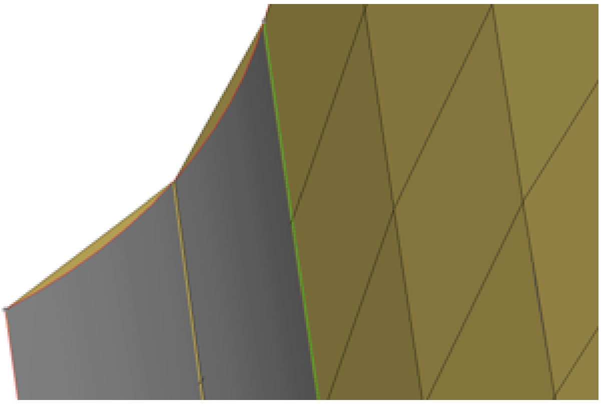


Figure 735: When meshing curved surfaces, the planar elements (tan color) can deviate from the curved grey geometry.

Edge deviation applies to both surface geometry and when re-meshing elements. Automeshing with edge deviation automatically selects the best element size to approximate a curve, within the limits that you specify. The maximum deviation and maximum feature angle parameters are the primary controls for this effect.

This method can produce a mesh in which the element size varies, even within the same surface. Areas of high curvature will tend to have smaller elements than areas of low or no curvature. The element size boundaries controls this effect.

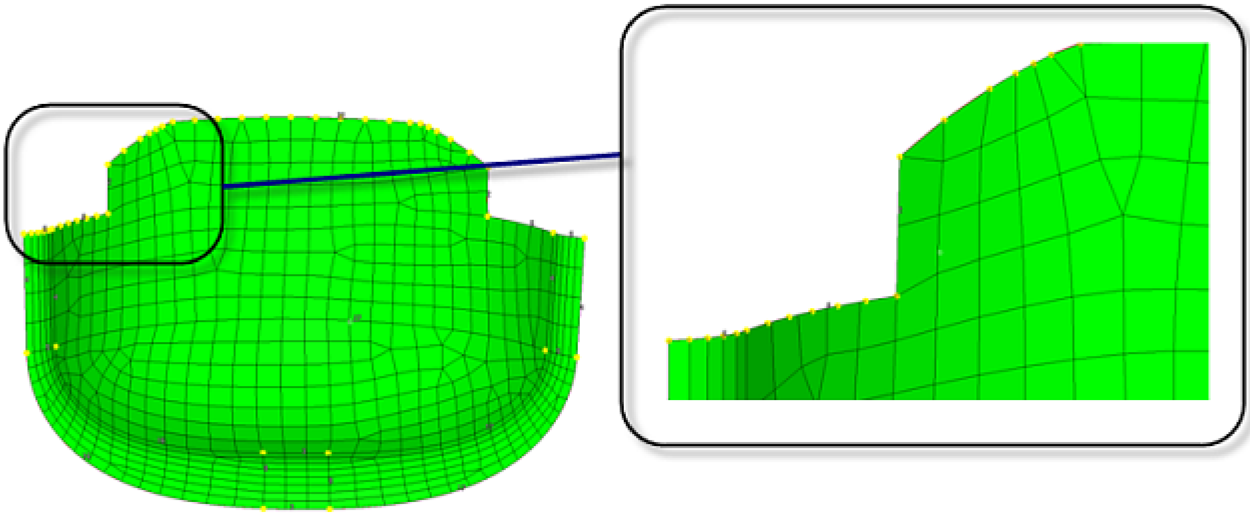


Figure 736:

Edge deviation control when meshing creates smaller elements and spaces nodes closer together to limit how much the elements can deviate from the surface edges.

Surface Deviation

Creates mesh within the limits of element deviation from a surface.

Similarly to edge deviation, meshing is driven by distances between flat elements and model geometry. When flat elements are used to approximate a curved surface, there is always a discrepancy between each element and the actual curve of the surface, because the element uses a straight line between two nodes.

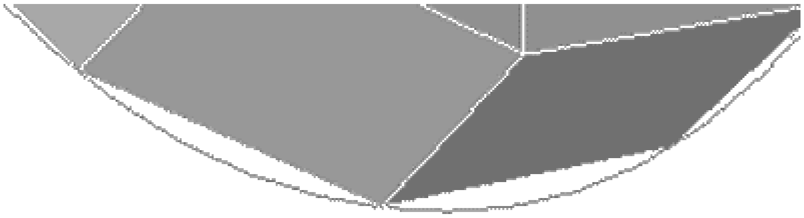


Figure 737:

A gap is visible between the curved edge of the surface and the element edges.

Surface deviation meshing chooses the mesh density based on the severity of this deviation. Where the threshold deviation would be exceeded, smaller elements are used to reduce the deviation.

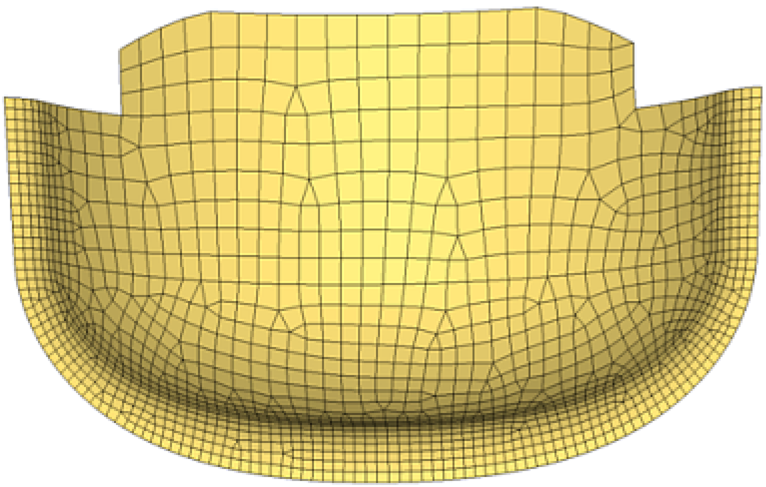


Figure 738:

With surface deviation meshing, smaller elements are used to accurately represent curved surfaces. Larger elements are used where the geometry shows less curvature.

Use refinement to set a specific desired mesh size for a point, line or surface face.

Rigid Body Mesh

Creates mesh to represent the topology of a rigid object.

Rigid bodies are surfaces that are expected to be treated as undeformable in the solution. One example of a rigid body is in metal-forming. When modeling the results of a die pressing down on a metal sheet, it is important to model the shape of the die because that determines the shape of the metal sheet after being pressed. However, during a forming analysis the stresses and deformations of the die itself are not of interest, only those of the formed metal sheet. Other applications for rigid bodies include the impactors used in vehicle crash simulation.

A mesh that accurately represents the rigid geometry is important for such simulations to allow the solver collision detection routines to work effectively and accurately. Since stress and deformation of the rigid body are not calculated by the solver, the rigid body mesh focuses on accurately modeling the shape of the body rather than on producing a high-quality mesh. To this end, Rigid Body meshing uses the same faceting and shading routines that are used for drawing the model graphics. The resulting mesh may have high aspect ratio or extremely tapered elements that would not be suitable for solution, but can accurately represent the geometry.

The images below illustrate the differences between surface deviation meshing and rigid body meshing. Both meshes were generated using the same parameters in terms of min/max element size, maximum deviation and feature angle, and mesh type.

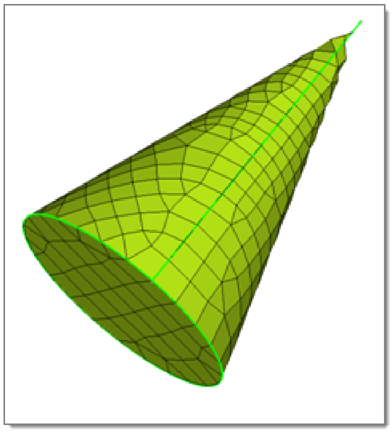


Figure 739:

When creating a surface deviation based mesh on this cone, many small elements are required to capture the geometry, even so, the elements exhibit a lot of warpage and those at the tip are distorted and do not accurately represent the geometry.

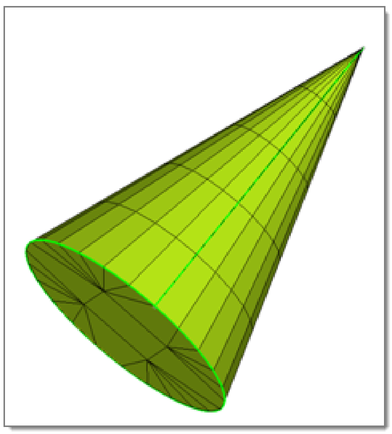


Figure 740:

With the rigid body mesh, the shape of the object can be accurately modeled using fewer larger elements since the element shape is not a concern.

Mesh Generation Algorithms

The mesh generation algorithms are divided into two types: those that require the presence of a surface to provide a context of operation, or those working entirely from node and/or line data.

Autodecide

If you are meshing a surface, the default mesh generation algorithm is Autodecide. In this case, the geometry of each face and the element densities specified for each edge is analyzed, and the algorithm that will give the best results is selected. For most configurations, the Free algorithm is chosen.

Free

The Free meshing algorithm is a general-purpose formula that works for most meshing conditions. The surface can have interior holes or edges and any number of sides. If **quads** or **trias** is the selected element type, an advancing front algorithm is used. If **mixed** is the element type, a sub-mapping algorithm is used.

The advancing front algorithm uses the following process:

- Traverses the perimeter of the region, placing elements along the edges as it proceeds. Each site where an element could be placed is measured and one of several possible elements is chosen. Eventually the entire region is filled with elements.
- Examines the groups of elements to see if a local change in the connectivity might improve element quality.
- Applies repeatedly the selected smoothing algorithm until no node is moved farther than the specified smoothing tolerance.

If **quads** is the selected element type for the current face, HyperMesh attempts to produce an all-quads mesh, but there are some situations in which one or more trias are included:

- If the total number of elements specified for the perimeter of the face is odd, at least one tria always needed.
- If there is a tight corner on the boundary that would require a poor quality quad, a single tria is used.
- Sometimes two or more trias are needed because of the particular order in which the elements were generated; if that is the case, you can usually eliminate them by changing some of the meshing parameters and then remeshing the region.

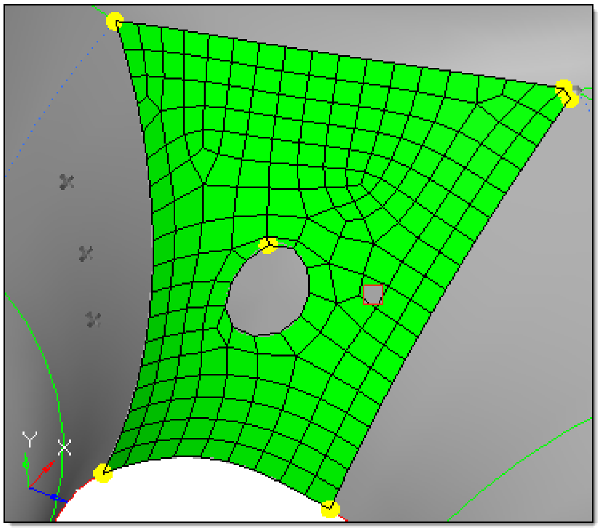


Figure 741:

If **quads only** is the selected element type, a mesh is created consisting entirely of quads; no trias will be used.

> **Note:** This method is more likely to fail to mesh than the quads option, and may produce some poor elements.

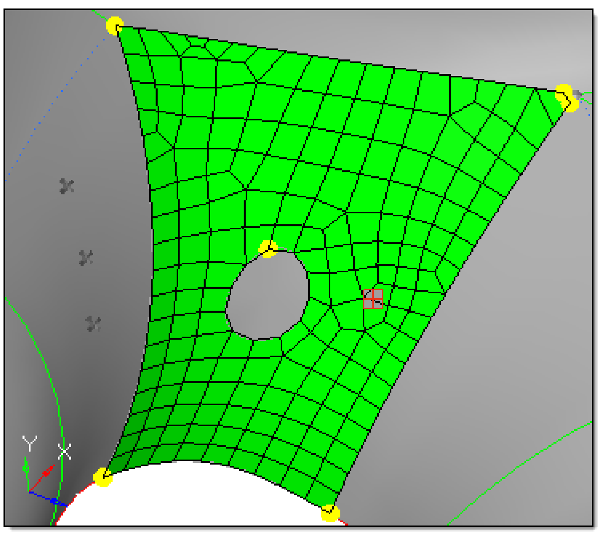


Figure 742:

If **trias** is the selected element type, a streamlined version of this algorithm that is optimized for the different shape and connectivity requirements of tria elements is used. These two examples show the difference between conventional and right-trias.

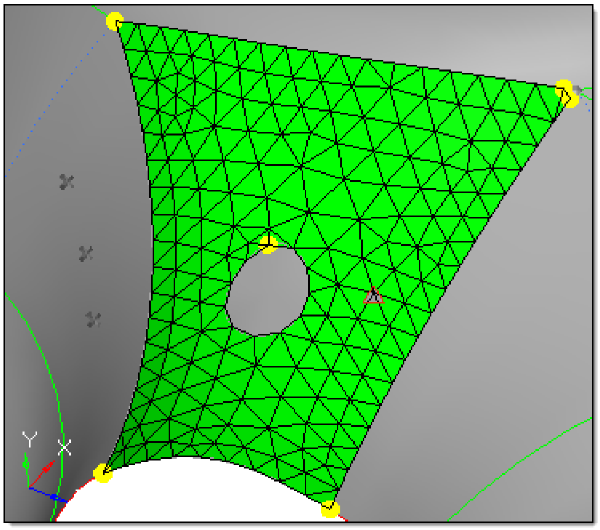


Figure 743:

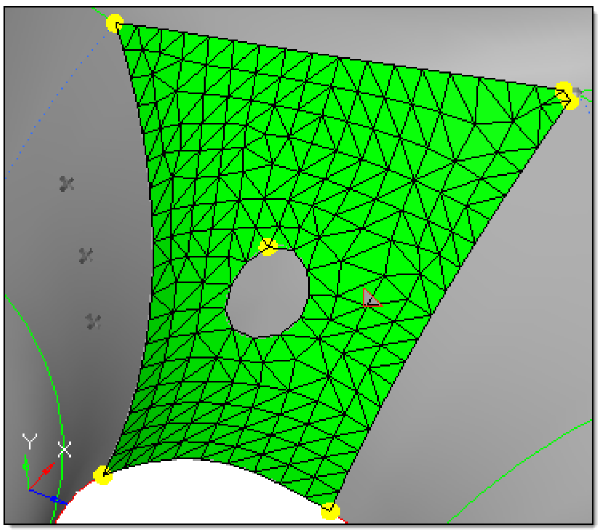


Figure 744:

Map as Triangle, Rectangle, Pentagon, or Circle

If the region is free from internal holes and the boundary is clearly triangular, rectangular, or pentagonal in shape, the best choice of algorithm is usually to map a standard mesh onto the region using transfinite interpolation. Such an operation is exceedingly fast, and where applicable, gives quality results rapidly. A standard template based on the element densities around the perimeter of the region is chosen. Ignoring rotations, more than 18 different configurations requiring distinct templates are recognized. To make tria elements, first a quads mesh is created and then each element is divided along its shortest diagonal.

In general this decision can be left to autodecide, but there may be some cases in which a manual decision is necessary to produce the best results.

> **Note:** If no mesh can be generated using the specified mapping method, one will be generated using autodecide as a fall-back measure.

On the Automesh secondary panel, a white icon denotes the mapping algorithm used for each meshed surface.

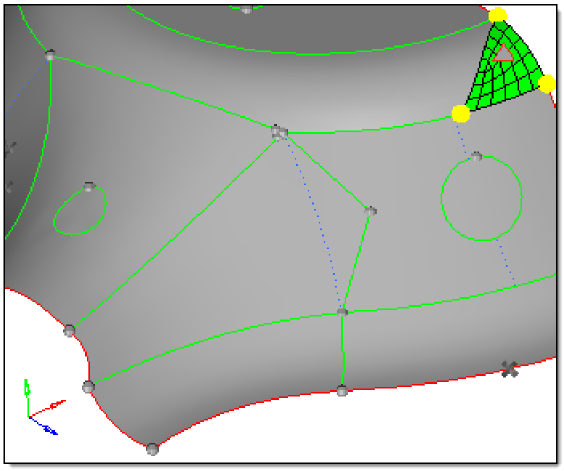


Figure 745: Map as Triangle, Mixed Mesh Type

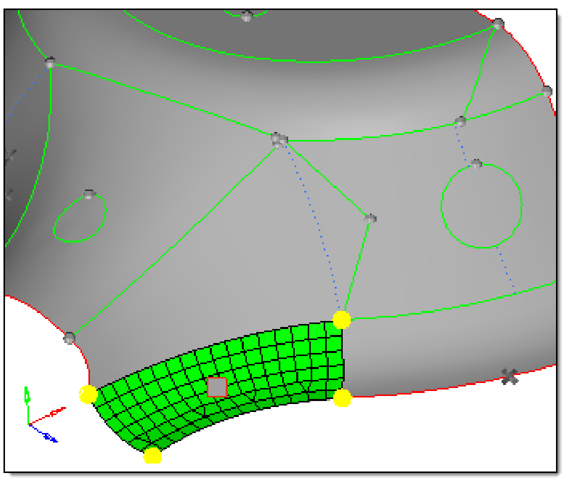


Figure 746: Map as Rectangle, Mixed Mesh Type

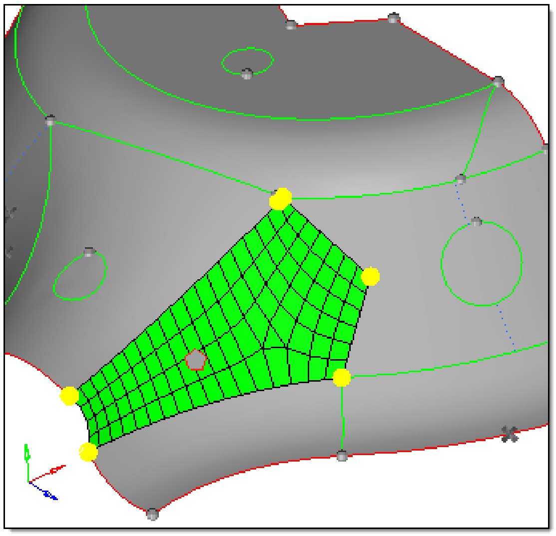


Figure 747: Map as Pentagon, Mixed Mesh Type

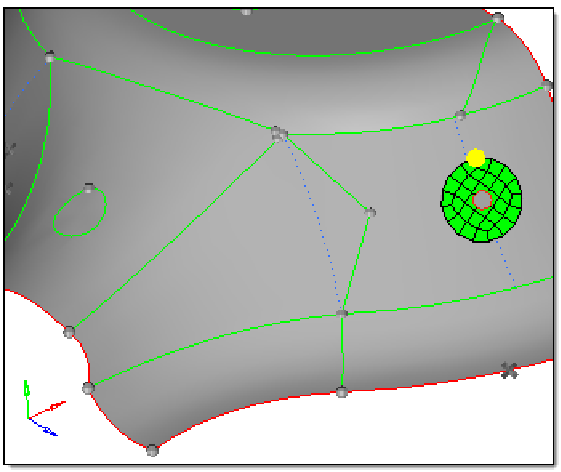


Figure 748: Map as Circle, Mixed Mesh Type

Map without Surface

If you are creating a mesh entirely from line and/or node data, with no surface, the mesh generation algorithm is decided by the tool that was used to describe the desired operation. If you use the Drag panel, the algorithm is to drag. If you use the Spin panel, the algorithm is to spin, and if you use the Spheres panel, the algorithm is to map a sphere-covering mesh. You can still use the density and biasing manipulation tools but some edges will be linked together, so that the configuration always satisfies the balancing requirements of the intended mapping.

Mesh Types

The mesh type determines the type of elements used to create mesh during automatic meshing.

Advanced

Enables you to select a **Mapped Type** for elements on surfaces that can be mapped to simple geometric shapes and a **Free Type** for elements that cannot easily be mapped to simple shapes to use when creating mesh.

Mixed

Uses quad elements to create mesh, but inserts tria elements when making density transitions to improve mesh quality.

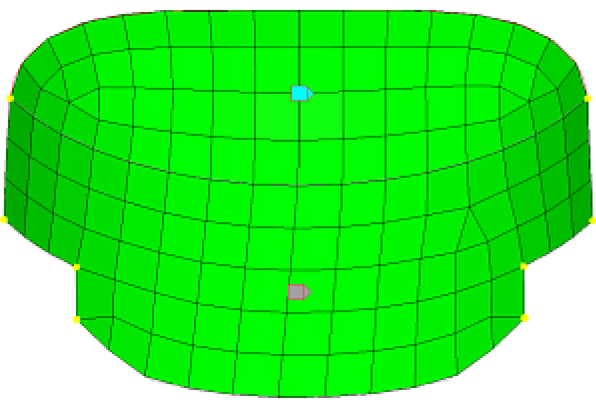


Figure 749: Mixed

Quads

Uses quad elements to create mesh, however if the sum of the element densities around the perimeter of the face or surface is odd then at least one tria element must be created. Adjusting the element densities while meshing interactively can usually eliminate all tria elements.

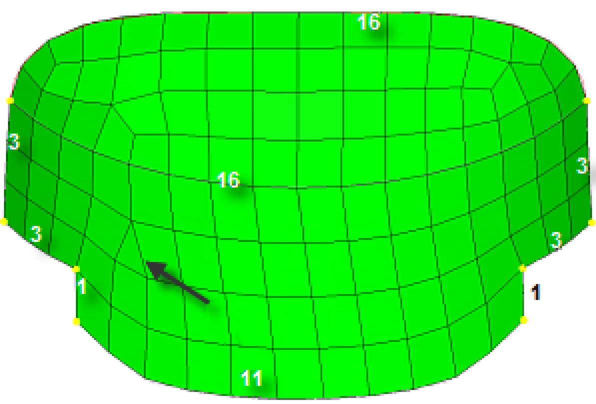


Figure 750:

The sum of element densities on the perimeter of the lower surface is odd, resulting in a tria as indicated.

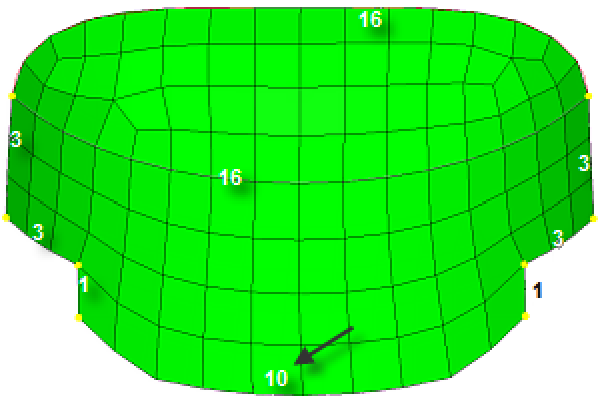


Figure 751:
Adjusting the bottom edge density from 11 to 10 makes the sum even and generates all-quads.

Quads Only

Uses a subdividing routine that tends to generate more orthogonal quad elements to create mesh. Tria elements may still used depending on the density settings as with the quads type.

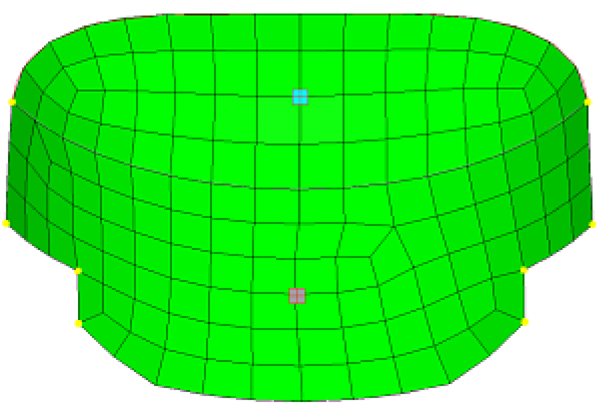


Figure 752: Quads Only

R-Trias

Uses right-angle triangular elements to create mesh.

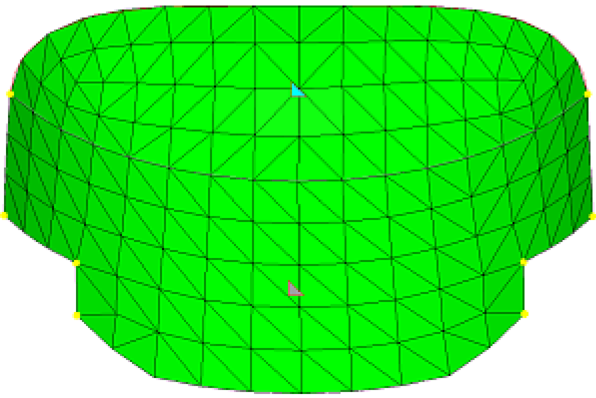


Figure 753: R-Trias

Trias

Uses all tria elements to create mesh.

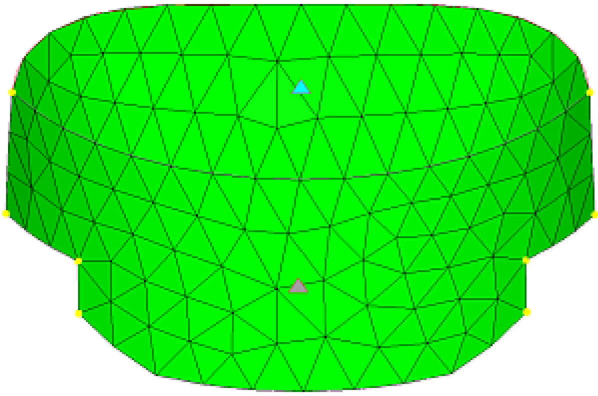


Figure 754: Trias

Element Biasing

The automeshing process enables you to bias the placement of nodes so that their intervals are not uniform in size.

You can designate that the smaller intervals go near the start of the edge, near the end of the edge, near both ends with larger intervals in the middle, or near the middle of the edge.

Within the automesher, you may want to use biasing to improve element quality when transitioning from smaller to larger element sizes. When you use the Drag and Solid Offset panels, you can use biasing to cluster several layers of elements near the surface of a solid. In Linear Solids, the mesh at one end could be scaled several times larger than at the other end. Element biasing allows you to moderate the changes in aspect ratio from the start to the end.

Linear Biasing

In linear biasing, the biasing intensity corresponds to the positive slope of a straight line over the interval [0,1] of the Real Line. This interval is uniformly divided into as many subintervals as specified by the element density and they are mapped along the edge so that the length of the image interval is proportional to the height of the line over the midpoint of the source interval. Each image interval corresponds to the side of an element.

Specifically, let n be the element density and let $s \in \left\{0, \frac{1}{n}, \frac{2}{n}, \frac{3}{n}, \ldots, \frac{n}{n}\right\}$.

We want a node placement function x(s) taking values in [0,1] with x(0) = 0 and x(1) = 1. If m is the slope of the line, and b is its y-intercept, then:

$$x(s) = x(0) + C\int_0^s (m\varepsilon + b)d\varepsilon = x(0) + C\left(\frac{ms^2}{2} + b \cdot s\right).$$

Using x(0) = 0, and x(1) = 1, we find: $C = \left(\frac{2}{m+2b}\right)$ so,

$$x(s) = s \cdot \frac{ms + 2b}{m + 2b}.$$

For this, m is the absolute value of the biasing intensity. If the biasing intensity is negative, the nodes are placed according to 1 - x(s). Thus, a positive biasing intensity puts small elements at the start of the interval.

We can use b to scale the behavior of the function so that convenient values are in the range [0,20]. The value used is b = 1.5.

Exponential Biasing

In exponential biasing, the sizes of the intervals grow geometrically, progressing along the edge, with each successive interval being a constant factor larger than the previous. That factor is 1.0 plus 1/10 of the absolute value of the biasing intensity. This formula was chosen so that an intensity of zero will still represent no biasing, and convenient values will fall in the range [0,20]. Negative biasing intensities just reverse the edge, placing the smaller elements at the end instead of the beginning.

Specifically, let n be the element density and let $s \in \left\{0, \frac{1}{n}, \frac{2}{n}, \frac{3}{n}, \ldots, \frac{n}{n}\right\}$.

We want a node placement function x(s) taking values in [0,1] with x(0) = 0 and x(1) =1.

Let $C = 1.0 + 0.1 \cdot \mathrm{abs}(\mathrm{bias\_intensity})$ be the geometric growth factor.

We need a function $\varphi(\varepsilon)$ so that: $x(s) = \frac{1}{\int_0^1 \varphi(\varepsilon)d\varepsilon} \cdot \int_0^s \varphi(\varepsilon)d\varepsilon$

Let $\varphi(\varepsilon) = C^{n\varepsilon}$ then:

$\varphi(0) = 1, \varphi\left(\frac{1}{n}\right) = C, \varphi\left(\frac{2}{n}\right) = C^2, \ldots \varphi\left(\frac{n}{n}\right) = C^n$ which gives the proper interval lengths,

then x(s) scales them to the range of [0,1]. Thus, $x(s) = \frac{C^{ns} - 1}{C^n - 1}$.

Bellcurve Biasing

In bellcurve biasing, nodes are distributed long the edge in a pattern that is symmetric across the midpoint of the edge. If the biasing intensity is positive, the smaller intervals are placed at the beginning and end of the edge, and if it is negative, they are placed at the middle of the edge.

Specifically, let n be the element density and $s \in \left\{0, \frac{1}{n}, \frac{2}{n}, \frac{3}{n}, \ldots, \frac{n}{n}\right\}$.

We need $\varphi(\varepsilon)$ so that $x(s) = \frac{1}{\int_0^1 \varphi(\varepsilon)d\varepsilon} \cdot \int_0^s \varphi(\varepsilon)d\varepsilon$
takes values in [0,1] with x(0) = 0, x(1) = 1, and has the behavior noted above. If we use:

$\varphi(\varepsilon) = e^{-r\left(\varepsilon-\frac{1}{2}\right)^2}$ for positive biasing intensity r, then x(s) becomes:

$x(s) = \frac{\operatorname{erf}\left(\frac{\sqrt{r}}{2}\right) - \operatorname{erf}\left(\sqrt{r}\cdot\left(\frac{1}{2}-s\right)\right)}{\;}$ where erf() is the statistical error function,

$\operatorname{erf}(t) = \int_0^t e^{-r\left(\varepsilon-\frac{1}{2}\right)^2} d\varepsilon$

Linked vs. Locked Edges

Most of the surface-less mesh generation algorithms require that some edges have exactly the same element density and biasing values as other edges. Those edges are automatically linked together so that they stay balanced.

Any change to one of the edges is immediately applied to all others that are linked to it.

Some of the surface creation panels allow you to use a node list to define one or more sides of a surface. In these circumstances, those nodes are used directly to make elements within the Automesh Secondary panel. The resulting edge is locked and you cannot change the element density or biasing. If you try to adjust the element density numbers corresponding to these locked edges, it has no effect. The error message, "The value of this number cannot be changed" is displayed.

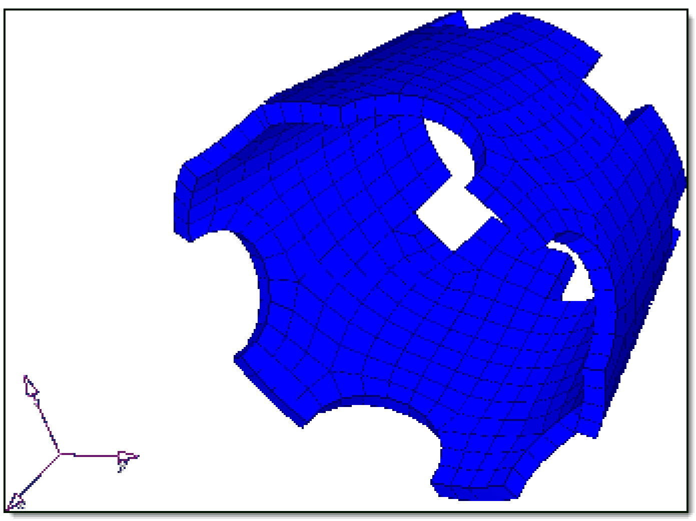


Figure 755:
Use the Automesh secondary panel to prepare input for solid offset.

Smoothing Algorithms

Supported smoothing algorithms.

Autodecide
: By default, the perimeter of the region is traversed looking for variations in element edge length and choose between size-correcting and shape-correcting smoothing algorithms.

Size Corrected

Evens out the sizes of the elements at the cost of some element quality, usually in the form of worsened aspect ratios from the stretching of elements. A modified Laplacian over-relaxation that can correctly handle mixtures of quads and trias is used. If the element spacing around the perimeter is roughly uniform, this choice usually gives the best results.

Shape Corrected

Corrects the elements' shapes, allowing variation in element size. HyperMesh uses a modified isoparametric-centroidal over-relaxation that can correctly handle mixtures of quads and trias. If there is a transition from small elements to large elements in the region, this choice usually gives the best results.

Create Midsurface Mesh

Create a midsurface mesh with a thickness from 3D geometry.

> **Restriction:** Only available in the OptiStruct, Radioss, Abaqus, LS-DYNA, ANSYS, and Nastran solver interfaces.

1. From the menu bar, click **Mesh** > **Create** > **MidSurf Mesh**.
   The **Midsurface Mesh** tool opens.
2. Select a solid geometry to create the mid-mesh for.
   a) Under Input selection, click **Geometry**.

   Figure 756:

   b) In the panel area, use the selector to select solid geometry.
   c) Click **proceed**.
3. Select **Pre-Midsurface Cleanup** to define pre-midsurface cleanup options.
   - To equivalence edges on the input surface, select the **Edges equivalencing** checkbox. Edges can be equivalenced using an automatic or manual tolerance.
   - To fix gaps between stitched edges and vertices to make the actual geometry of the surfaces consistent with the model topology, select the **Match topology** checkbox. Topology can be matched using an automatic or manual tolerance.
   - To remove logos that appear on geometry, select the **Delete logo** checkbox. This is based on settings from the parameter file.

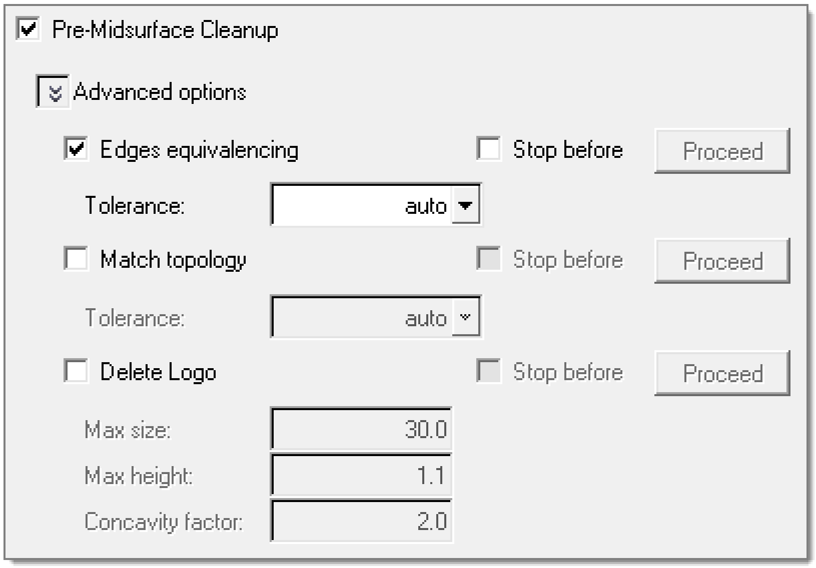


Figure 757:

4. Setup midsurface extraction.

This step is mandatory unless midsurfaces have previously been generated manually.

a) If midsurfaces were previously generated and organized into the Middle Surface component, select **Skip if Midsurface component exist** to skip the midsurface extraction step and preserve the current mesh.

For example, if you manually extracted the Middle Surface and manually meshed some of its part (disconnected from others), then select Skip if Midsurface component exist to preserve the current Midsurface mesh and only mesh the unmeshed surfaces. If you clear this checkbox, the entire Middle Surface will be remeshed, and you will lose your current mesh.

b) To preserve the mesh of the disconnected parts in the Middle Surface component that contain a mesh and only mesh the disconnected parts in the Middle Surface component that are not meshed, select the **Keep existing mesh** checkbox.

If you clear this checkbox, all of the disconnected parts in the Middle Surface component will be remeshed. This option is enabled when you select Skip if Midsurface component exist.

a) To define advanced auto-midsurfacing options, click **Options**.

b) For Methods, select a method for extracting the midsurface.

- Choose **offset + planes + sweeps** to use a midsurfacing algorithm to identify the places where a piece of plane or a piece of a sweep surface can be used as a middle surface. A middle surface is constructed at the remaining places in the model, for example the places where planar or sweep surface pieces cannot be used as a middle surface, by the same algorithm as in offset via the offset of the model's sides.
- Choose **offset + planes** to use a midsurfacing algorithm to identify the places in the model where a piece of plane can be used as a middle surface. At the remaining places in the model, for example the places where planar pieces cannot be used as a middle

surface, the same algorithm as in offset is used to construct the middle surface via the offset of the model's sides.

- Choose **offset** to create pieces of the middle surface by offsetting the model's side surfaces towards the middle. This is the traditional approach for midsurfacing in HyperMesh.
- Choose **skin offset** to generate a midsurface by duplicating and offsetting the inner skin surfaces (those with the smallest area) and assigning them a constant thickness. The parts you provide to the algorithm must be relevant. T-connections, constant thickness, or internal ribs/features/creases are not relevant.

5. Define geometry cleanup and meshing criteria.

a) Select the type of geometry cleanup and meshing to perform, and define the parameters and criteria accordingly.

- Choose **Auto** to automatically cleans up geometry, and enables you to define an element size and a mesh type.
- Choose **Custom** to define topology cleanup parameters and element quality criteria. The parameters and criteria options are the same as in the Autocleanup panel, and is based on settings from the parameter file.

b) To sort the midsurface to the original component or to <original name~> component, select **Sort Midsurfmesh component to**.

6. Optional: Assign a thickness to the midsurface mesh.

a) Select **Thickness assignment**.

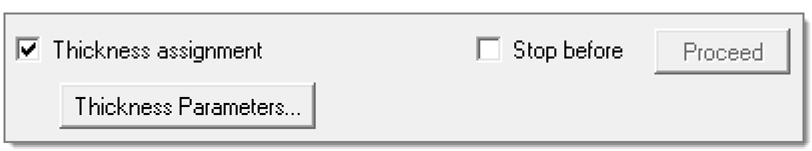


Figure 758:

b) Click **Thickness Parameters**.

c) In the **Map Mid-Mesh Thickness** dialog, calculate the thickness of a mid-mesh from the solid geometry.

7. To pause the generation of the midsurface mesh at any step, select **Step before** next to the desired step to stop.

By default, once you click **Start**, all of the above steps are executed.

Figure 759:

8. Click **Start**.

While the process is running the Proceed buttons change colors, indicating the progress of the process.

Blue
Complete step

Yellow
Currently running step

Green
Next step to be performed (when the Stop before checkbox is selected)

The midsurface mesh is generated.

Generate Midmesh

Automatically generate a mesh at the midplane location, directly from the input geometry (components, elements, solids or surfaces), without first creating a midsurface.

The midmesh functionality in HyperMesh saves significant time over the traditional midsurface-based approach.

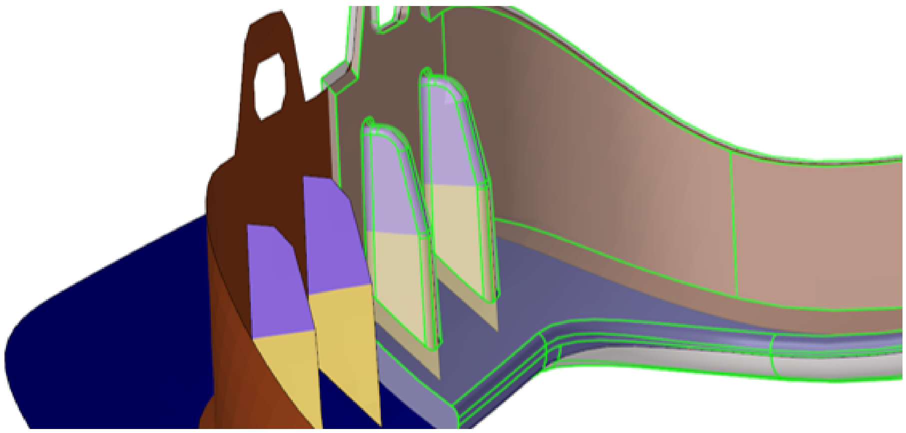


Figure 760: Midmesh Result Example

Direct midmesh is supported for a large majority of parts including cast, machined, injection molded and extruded as shown in Figure 761.

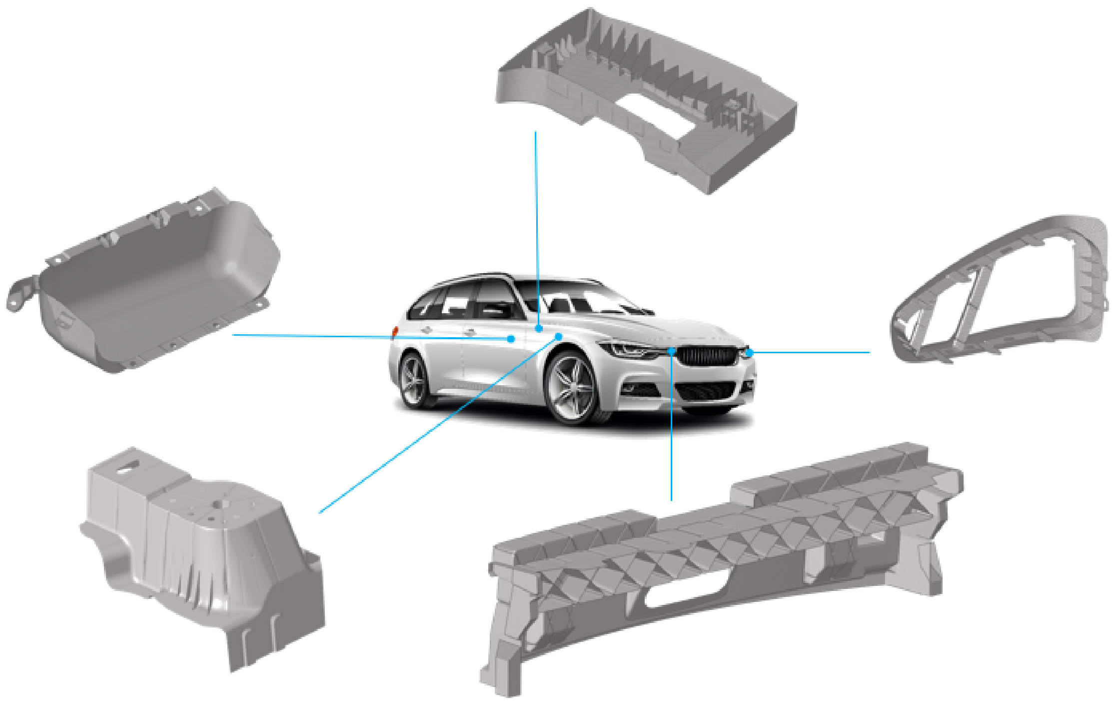


Figure 761: Direct Midmesh Supported Parts

The resulting output consists of 2D shell elements created with the user-provided target size, as well as 1D elements defining the topology of the mesh (vertices/edges/faces). Midmesh generation is also multithreaded to take advantage of multi-core environments.

Midmesh Generation Workflow

There are several steps involved in generating a good quality midmesh. Following the workflow shown in Figure 762 helps guarantee the best result with minimal manual effort.

Nominal Run

Extract the base midmesh.

Cleanup Resulting Topology

Use the semi-automated midmesh edit edge and edit face tools to correct the 1D topology and fix any bad/missing faces. The goal is to prepare the model for final remeshing.

Rebuild Mesh

Remesh to the final size and quality using the rebuild mesh functionality, and correct any remaining mesh quality issues.

Apply Thickness

Map the thickness from the original solid to the midmesh via the Map Thickness tool.

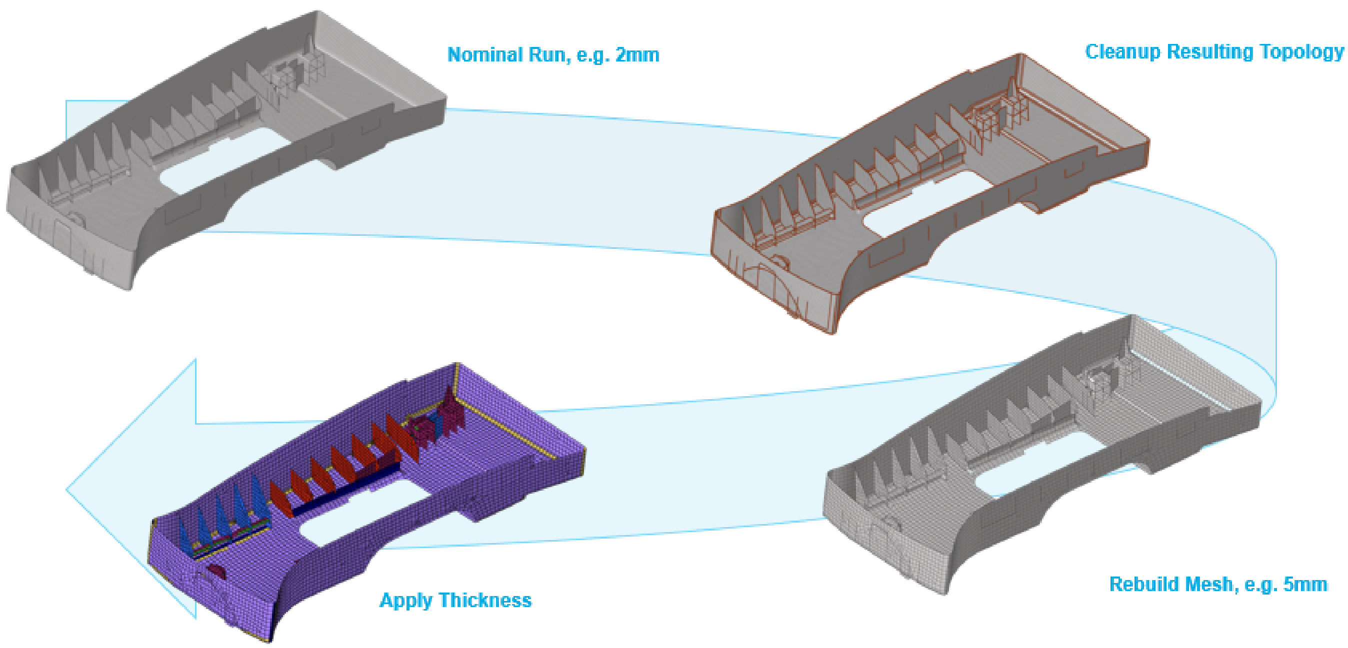


Figure 762: Midmesh Workflow

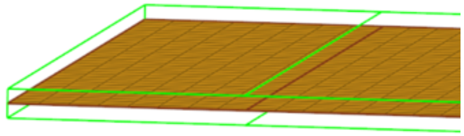
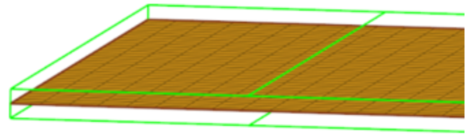
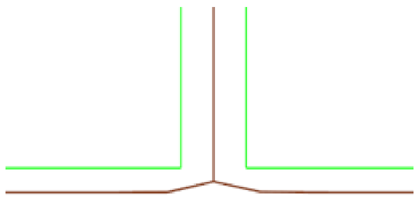
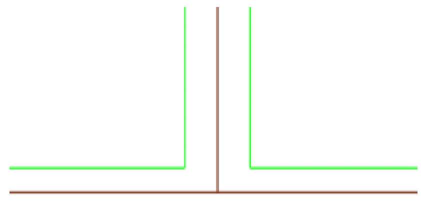
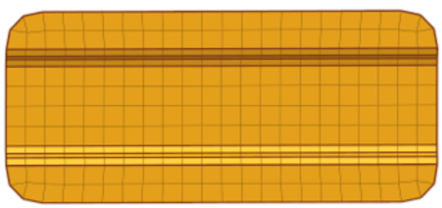
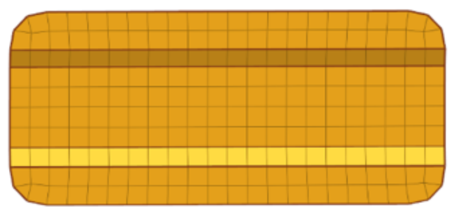
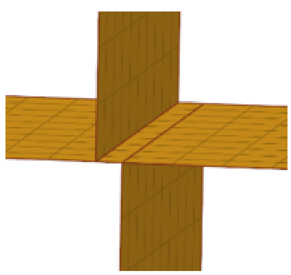
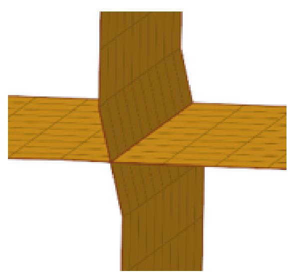
Create Midmesh

Midmesh generation is possible on dirty geometry, but a cleaner output can be obtained by removing duplicate or overlappinng surfaces, stitching free edges, removing logos and other small features that are not of interest, and merging any solids that should be topologically connected.

It is recommended to use a smaller extraction size (for example, 2 or 4mm depending on model scale) in order to get a good sampling of the input geometry. The final rebuild mesh step takes care of remeshing to the desired size and quality. Using an extraction size smaller than the representative feature size will not necessarily give better results and will take significantly more run time.

1. From the 2D page, click the **Midmesh** panel.
2. Select the **create** subpanel.
3. Define options accordingly to control the resulting midmesh output.

Option	Action
ignore flat edges	Do not imprint flat edges from the input geometry to the midmesh.

Option	Action
	 *Figure 763: Option Disabled*  *Figure 764: Option Enabled*
flatten connections	Align/flatten the midmesh at ribs/connections.  *Figure 765: Option Disabled*  *Figure 766: Option Enabled*
suppress proximity edges factor	Remove 1D topology edges within the given factor of the minimum size from the criteria file.  *Figure 767: Option Disabled*  *Figure 768: Option Enabled*
combine non-manifold edges factor	Join non-manifold edges within the given factor of the minimum size from the criteria file.  *Figure 769: Option Disabled*  *Figure 770: Option Enabled*

Option	Action
defeature openings with width <	Remove small holes and openings less than the specified width.

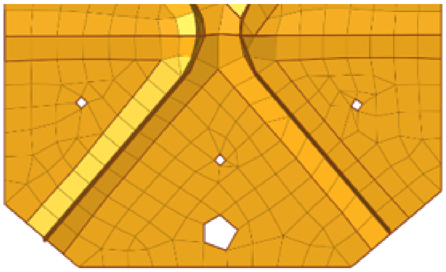


Figure 771: Option Disabled

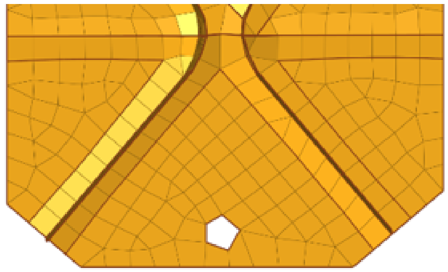


Figure 772: Option Enabled

4. Optional: Edit the **min size** and **target element size** settings in the criteria file to control the resulting midmesh output by clicking **edit criteria**.

Option	Action
Target element size	Target element size for the finalized mesh. This can be different from extraction size. It is used for internal calculations for the output mesh to be ready for rebuilding with the same criteria.
Min size	Minimum size allowed in the finalized mesh. This in combination with the 'suppress proximity edges factor' and 'combine non-manifold edges factor' can ensure that the output mesh is ready for rebuild with the same criteria.

5. Click **create**.

Edit Midmesh

Once the midmesh is generated, there may be problem areas that need to be corrected. The 1D topology is important for the rebuild mesh operation, and care must be taken to prepare it accordingly. In addition, making sure the faces do not have intersected, overlapped, or missing elements, and that they have proper alignment, is also essential. Specialized midmesh edit tools streamline the process of repairing the 1D topology edges, and correcting issues with the midmesh faces.

- Select the **edit edge** subpanel access tools which can be used to repair 1D topology edges.

Option	**Action**
create mid-edge	Create a new mid-edge, using the input geometry as a guide.

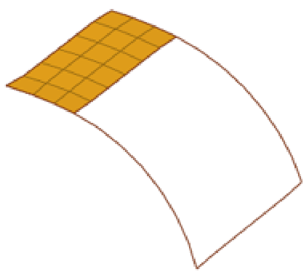


Figure 773: Before

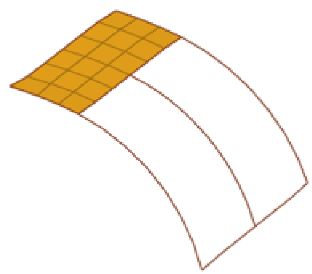


Figure 774: After

split by two nodes	Create a new edge between two nodes.

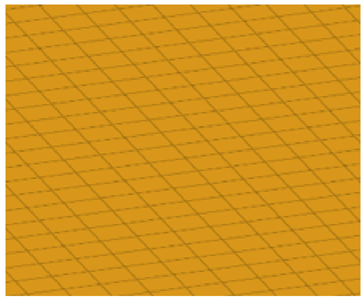


Figure 775: Before

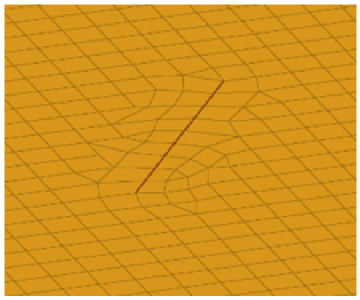


Figure 776: After

split by node-edge Create a new edge between a node and an edge, using a shortest, tangential or mixed path.

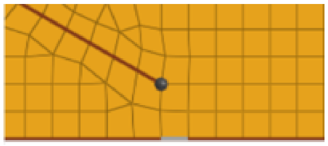


Figure 777: Before

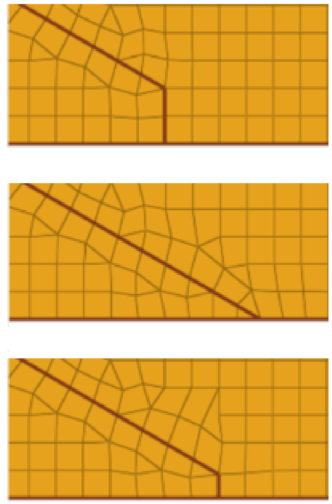


Figure 778: After

delete edge Delete an edge.

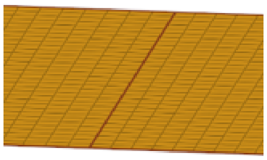


Figure 779: Before

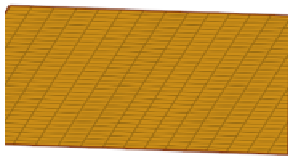


Figure 780: After

t-edge align Align/flatten a t-connection edge to a surface.

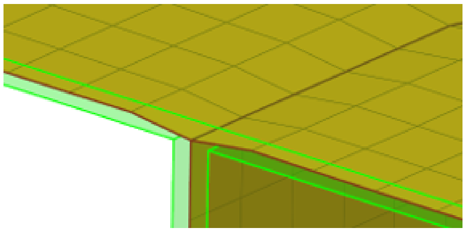


Figure 781: Before

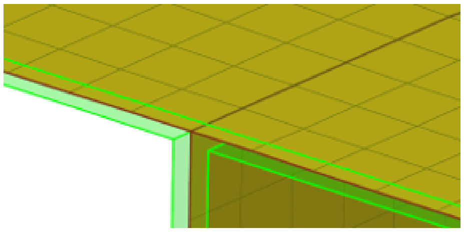


Figure 782: After

by geom edge align Align mesh edges to input geometry lines, and smooth the mesh, or imprint new geometry edges onto the midmesh.

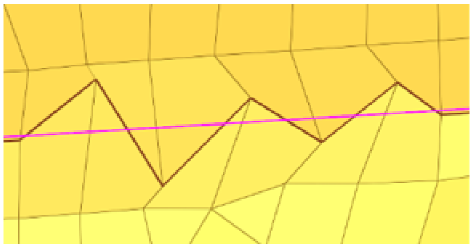


Figure 783: Before Align

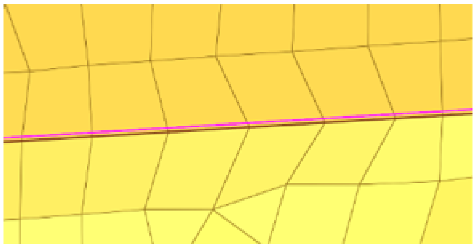


Figure 784: After Align

- Select the **edit face** subpanel to access tools which can be used to correct issues with midmesh faces.

Option	Action
fill face	Create a mesh within a closed 1D topology loop, attempting to keep tangency.

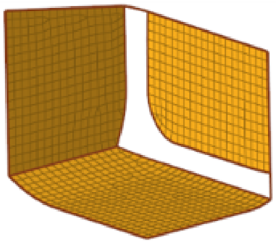


Figure 785: Before

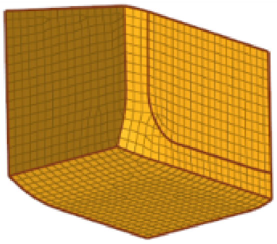


Figure 786: After

repair face	Attempt to fix topological problems (holes/gaps/cracks, intersections, slivers, overlaps) in the mesh and remesh the face.

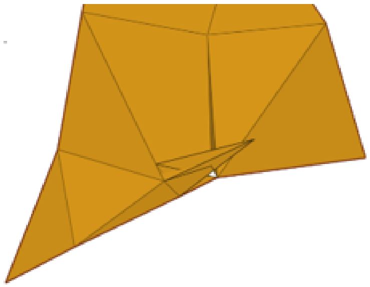


Figure 787: Before

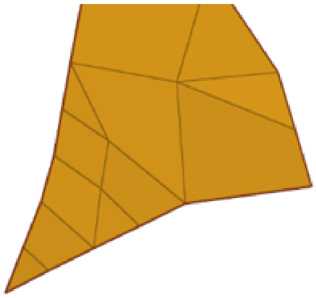


Figure 788: After

detect intersections/ gaps

Detect intersecting element clusters and holes/gaps/cracks, and create element sets for further handling.

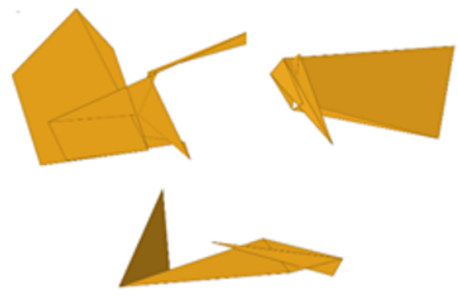


Figure 789: Before

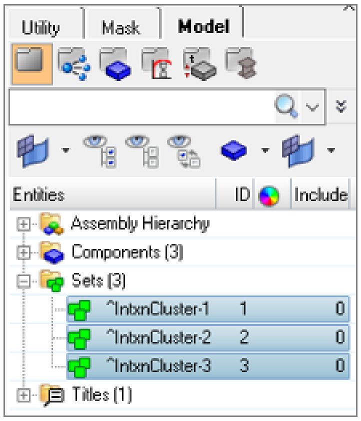


Figure 790: After

align face

Align a selection of elements to an input geometry face, with optional offset.

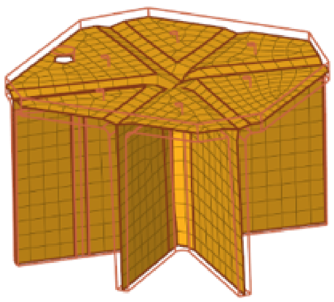


Figure 791: Before

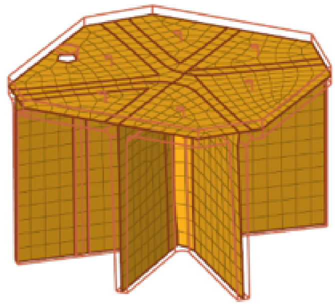


Figure 792: After

Batchmesher Midmesh Generation Rules

Midmesh generation is also supported from within Batchmesher.

This functionality allows for generating midmesh representations via the standalone Batchmesher, via the Part Browser, and interactively via the Automesh panel and Mesh Controls Browser.

If a midsurface is extracted:	Traditional midsurface geometry cleanup and meshing are performed.		
If a midsurface is not extracted:	Extract midsurfaces enabled	Allow direct midmesh enabled	If the part is determined to be midmeshable, certain input geometry cleanup steps and direct midmesh generation are performed. If the part is determined to not be midmeshable, traditional input geometry cleanup and meshing are performed.
		Allow direct midmesh disabled	Traditional input geometry cleanup and meshing are performed.
	Extract midsurfaces disabled	Allow direct midmesh enabled	If the part is determined to be midmeshable, certain input geometry cleanup steps and direct midmesh generation are performed. If the part is determined to not be midmeshable, traditional input geometry cleanup and meshing are performed.
		Allow direct midmesh disabled	Traditional input geometry cleanup and meshing are performed.

Figure 793:

Assign Thickness to Midsurface

Assign a thickness to shell elements that represent the middle surface of a solid part.

Geometric surfaces that represent the mid-plane of a part have thickness information stored in their definition if they are extracted using the midsurface function. Thickness data can be a single value for the entire part or a varying function.

The Midsurface Thickness Map script maps thickness data from surfaces to the associated nodes/ elements via properties. You can also review the contour plot of thickness data with this script.

1. From the menu bar, click **Mesh** > **Assign** > **Midsurface Thickness**.
   The Midsurface Thickness Map browser opens.
2. For Assign thickness to, select which entities to assign thicknesses.
   - Choose **Nodes, properties on elements** to assigns\ the thickness values directly to the node cards, with a single property assigned to the selected elements. The nodes are found from all of the selected 2D elements.
   - Choose **Nodes, properties on components** to assign the thickness values directly to the node cards, with a single property assigned to the selected components. The nodes are found from all of the 2D elements in the selected components.
   - Choose **Elements** to assign the thickness and Z-offset values (where supported) directly to the element cards. For each solver interface, the values will be updated on the element card for that solver.
   - Choose **Properties or elements** to group elements that fall within user-specified thickness intervals into common ranges, then create and assign to each of those elements in each range a property with the range thickness value assigned to the property card image. Most solvers only have Z-offset defined on the element card, so this value will always be populated on the element cards for any solver that supports Z-offset. In order to execute this mode, a base property named t0 must be defined. The t0 property definition will be used for all created properties based on the option specified in the Organization Method section. This option performs the following generic steps:
     1. Create properties with name "t[thickness value]" by copying the properties of the base property t0 and assigning the appropriate thickness based on the value of the Organization Method.
     2. Assign to the elements that have thickness values within the specified ranges, based on the value of the Organization Method, the relevant property.
   - Choose **Properties/Sections on components** to group elements that fall within user-specified thickness intervals into common components, then create and assign to each of those components a property/section with the thickness value assigned to the property/ section card image. Most solvers only have Z-offset defined on the element card, so this value will always be populated on the element cards for any solver that supports z-offset. In order to execute this mode, a base property/section must be assigned to a component named t0. The t0 component property/section will be used as the baseline for all newly created properties/sections based on the option specified in the Organization Method section. This option performs the following generic steps:

1. Create components and properties with name "t[thickness value]" by copying the properties of the base property t0 and assigning the appropriate thickness based on the value of the Organization Method.
2. Assign a property/section to its corresponding component.
3. Remove any property assignments to the elements.
4. Organize the elements that have thickness values within the specified ranges into the new components based on the value of the Organization Method.

- Choose **Organize only** to create new components and sort the selected elements into them according to the Organization Method selected.

3. To take z-offsets into account and assign z-offset values to the element cards for supported solvers, select the **Use Z-offset values** checkbox.

   HyperMesh uses z-offsets when midsurfacing parts that have variable thickness; the z-offset, which is saved as part of the midsurface data, tells a solver how much of a positive-normal offset exists between the actual part surface and the midsurface.

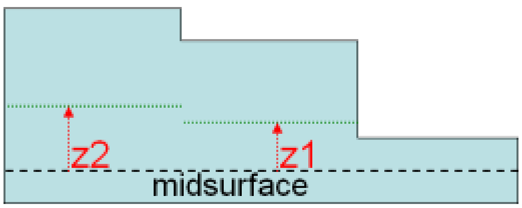


Figure 794:

4. For Thickness calculation method, select a method.
   - Choose **Nodal values** to calculate multiple thickness values for each element by finding the thickness at each of the element nodes. It is possible for a node to have multiple thickness values at a single location (shared surface edge where the surfaces have different thickness). The thickness calculated using that node for an element is dependent on which surface that element is associated to.
   - Choose **Average** to calculate a single thickness value for each element by averaging the thickness at each of the element nodes.
   - Choose **Centroid** to calculate a single thickness value for each element by calculating the thickness at the centroid of the element.
   - Choose **Max** to calculate a single thickness value for each element by calculating the thickness at each node of the element and taking the max value.
   - Choose **Min** to calculate a single thickness value is for each element by calculating the thickness at each node of the element and taking the min value.
5. For Organization method, select an organization method to specify the thickness range intervals used to generate properties based on their thickness values.
   Based on the Assign thickness to option, the properties and components are generated for certain thickness ranges. Any element with a thickness value within that range is assigned that property or organized into that component.

4. For Organization method, select an organization method to specify the thickness range intervals used to generate properties based on their thickness values.
Based on the Assign thickness to option, the properties and components are generated for certain thickness ranges. Any element with a thickness value within that range is assigned that property or organized into that component.

- Choose **Gauge file** to specify the thickness range intervals in a Gauge File.
- Choose **Range Interval** to specify a thickness tolerance.
  Thickness range intervals are automatically generated based on the thickness tolerance using the following formula. The thickness assigned to each created component is n*tolerance.
  - Lower limit = (tolerance / 2) + (tolerance* i)
  - Upper limit = (tolerance / 2) + (tolerance* (i + 1))
  - Assigned value = tolerance*(i+1)
  - Where i = 0……n, n is determined by the maximum thickness in model divided by the user specified tolerance and then rounding to up to the next integer.

6. Click **Apply**.

The thickness is assigned from the surface definition to the selected elements, based on the options specified.

Create a contour plot of the thicknesses on the selected elements/nodes based on the options specified by clicking **Contour**. This option does not assign the thickness to the nodes or elements; it is a review/display function only. It is very useful for visualizing and verifying the results of the utility before applying the Midsurface Thickness Map mapping operation.

If you choose Properties on elements, Properties on components, or Organize only under the Assign thickness to option, HyperMesh honors the Organization method settings during the contour process and the contour value is assigned based on that organization. This allows the contour to match with the applied results.

If you choose Elements or Properties on components for the Assign thickness to option, and choose to use Nodal values for the Thickness calculation method, the values may not exactly match the nodal values that are actually applied. There can be multiple thicknesses associated to a node if it shares an edge with multiple surfaces. Since HyperMesh can only provide one value for the contour, it always chooses the first value which might not match exactly with all of the applied values in these situations.

Example: Gauge File Format

Formatting for a gauge file.

```
Number of Gauges
[Number of Gauge Data Lines]
Gauges
Begin                End                Assigned Value
[min Thk]         [max Thk]         [Assigned Thk]
…
```

Figure 795: Gauge File Format

If the Assigned Value is not specified, then the average of the upper and lower limits will be used as Assigned Value.

```
Number of Gauges
4

Gauges
Begin                End                Assigned Value
0.0                0.05               0.05
0.05                0.1               0.1
0.1                0.15               0.15
0.15                0.2               0.2
```

Figure 796: Gauge File Example

Midsurface Thickness Assignment Behavior

Common and solver interface specific behavior for assigning thicknesses to the midsurface.

General

Organize only

- Creates new components based on the Organization method.
- Organizes elements into these components.

Contour

- Creates a contour of the thickness values of nodes and elements, based on the specified options.
- If you chose Properties on elements, Properties/Sections on components, or Organize only under the Assign thickness to option, HyperMesh honors the Organization method settings during the contour process and the contour value is assigned based on that organization. This allows the contour to match with the applied results.
- If you chose Elements or Properties/Sections on components for the Assign thickness to option, and choose to use Nodal values for the Thickness calculation method, the values may not exactly match the nodal values that are actually applied. There can be multiple thicknesses associated to a node if it shares an edge with multiple surfaces. Since HyperMesh can only provide one value for the contour, it always chooses the first value which might not match exactly with the applied values in these situations.

Abaqus

Nodes, properties on elements

- Creates a single property named tNodalThickness.
- Creates a single node set named tNodalThickness with Nodal_Thickness card image.
- Assigns the property to the selected 2D elements.
- Adds the relevant nodes to the node set.
- Assigns a single thickness to each node using attribute ThicknessValue.

Nodes, properties on components

- Creates a single property named tNodalThickness.
- Creates a single node set named tNodalThickness with Nodal_Thickness card image.
- Assigns the property to the selected components.
- Adds the relevant nodes to the node set.
- Assigns a single thickness to each node using attribute ThicknessValue.

- Clears any element property references for any 2D elements in the selected components.

ANSYS

Properties/Sections on components – nodal values

- Creates new components based on ordered nodal thickness values.
- Creates new properties and sections based on ordered nodal thickness values. Properties are created for those shell elements where thickness is assigned from properties. Shell sections are created for those shell elements where thickness is assigned from shell sections.
- Assigns the properties or shell sections to the corresponding components.
- Assigns multiple thicknesses to each property, using real values. In case of shell sections, only one thickness will be assigned.
- Organizes elements into the corresponding components.

Properties/Sections on components – all others

- Creates new components based on the Organization method.
- Creates new properties or shell sections based on the Organization method. Properties are created for those shell elements where thickness is assigned from properties. Shell sections are created for those shell elements where thickness is assigned from shell sections.
- Assigns the properties or shell sections to the corresponding components.
- Assigns a single thickness to each property or shell section based on the Thickness calculation method, using real values.
- Organizes elements into the corresponding components.

LS-DYNA

Elements

- Assigns multiple thicknesses to each element, based on the nodal thickness values for that element.
    - For tria3 elements, uses attributes: Elem_Option, LSD_ELEM_T1, LSD_ELEM_T2, LSD_ELEM_T3
    - For quad4 elements, uses attributes: Elem_Option, LSD_ELEM_T1, LSD_ELEM_T2, LSD_ELEM_T3, LSD_ELEM_T4

Properties/Sections on components

- Creates new components based on the Organization method.
- Creates new properties/sections based on the Organization method.
- Assigns the properties/sections to the corresponding components.
- Assigns a single thickness to each properties/sections based on the Thickness calculation method, using property attribute LSD_T1.
- Organizes elements into the corresponding components.

Marc

Properties on elements

- Creates new properties based on the Organization method.

- Assigns a single thickness to each property based on the Thickness calculation method, using component attribute TK.
- Assigns the properties to the corresponding elements.

Moldflow

Properties on components

- Creates new components based on the Organization method.
- Assigns a single thickness to each property based on the Thickness calculation method, using component attribute T.
- Organizes elements into the corresponding components.

Nastran

Elements

- Assigns multiple thicknesses to each element, based on the nodal thickness values for that element.
  - For tria elements, uses attributes: CTRIA3_T1T2T3, CTRIA3_T1, CTRIA3_T2, CTRIA3_T3, CTRIA3_ZOFFS [if requested], ZOFFS [if requested]
  - For quad elements, uses attributes: CQUAD4_T1T2T3T4, CQUAD4_T1, CQUAD4_T2, CQUAD4_T3, CQUAD4_T4, CQUAD4_ZOFFS [if requested], ZOFFS [if requested]
- Turns off the TFLAG option when necessary.

Properties/Sections on components

- Creates new components based on the Organization method.
- Creates new properties/sections based on the Organization method.
- Assigns the properties/sections to the corresponding components.
- Assigns a single thickness to each properties/sections based on the Thickness calculation method, using property attribute PSHELL_T.
  - For tria elements, uses z-offset attributes [if requested]: CTRIA3_ZOFFS, ZOFFS
  - For quad elements, uses z-offset attributes [if requested]: CQUAD4_ZOFFS, ZOFFS
- Organizes elements into the corresponding components.
- Clears any element property references for the selected elements.

Properties on elements

- Creates new properties based on the Organization method.
- Assigns a single thickness to each property based on the Thickness calculation method, using property attribute PSHELL_T.
  - For tria3 elements, uses z-offset attributes [if requested]: CTRIA3_ZOFFS, ZOFFS
  - For quad elements, uses z-offset attributes [if requested]: CQUAD4_ZOFFS, ZOFFS

- Assigns the properties to the corresponding elements.

OptiStruct

Elements

- Assigns multiple thicknesses to each element, based on the nodal thickness values for that element.
    - For tria elements, uses attributes: CTRIA3_T1T2T3, CTRIA3_T1, CTRIA3_T2, CTRIA3_T3, CTRIA3_ZOFFS [if requested], ZOFFS [if requested]
    - For quad elements, uses attributes: CQUAD4_T1T2T3T4, CQUAD4_T1, CQUAD4_T2, CQUAD4_T3, CQUAD4_T4, CQUAD4_ZOFFS [if requested], ZOFFS [if requested]

Properties/Sections on components

- Creates new components based on the Organization method.
- Creates new properties/sections based on the Organization method.
- Assigns the properties/sections to the corresponding components.
- Assigns a single thickness to each properties/sections based on the Thickness calculation method, using property attribute PSHELL_T.
    - For tria elements, uses z-offset attributes [if requested]: CTRIA3_ZOFFS, ZOFFS
    - For quad elements, uses z-offset attributes [if requested]: CQUAD4_ZOFFS, ZOFFS
- Organizes elements into the corresponding components.
- Clears any element property references for the selected elements.

Properties on elements

- Creates new properties based on the Organization method.
- Assigns a single thickness to each property based on the Thickness calculation method, using property attribute PSHELL_T.
    - For tria elements, uses z-offset attributes [if requested]: CTRIA3_ZOFFS, ZOFFS
    - For quad elements, uses z-offset attributes [if requested]: CQUAD4_ZOFFS, ZOFFS
- Assigns the properties to the corresponding elements.

PAM-CRASH 2G

Elements

- Assigns a single thickness at the element level based on the Thickness calculation method.
    - Uses element attribute ELEM_THK

Thickness on components

- Creates new components based on the Organization method.
- Assigns a single thickness to each component based on the Thickness calculation method, using component attribute MAT_THK.

- Organizes elements into the corresponding components.

Permas

Properties/Sections on components – Nodal values

- Creates new components based on ordered nodal thickness values.
- Creates new properties/sections based on ordered nodal thickness values.
- Assigns the properties/sections to the corresponding components.
- Assigns multiple thicknesses to each properties/sections, using property attributes: ThicknessSelEnumField, Thick_value_shell1, Thick_value_shell2, Thick_value_shell3, Thick_value_shell4
- Organizes elements into the corresponding components.

Properties/Sections on components – all others

- Creates new components based on the Organization method.
- Creates new properties/sections based on the Organization method.
- Assigns the properties/sections to the corresponding components.
- Assigns a single thickness to each properties/sections based on the Thickness calculation method, using property attributes: ThicknessSelEnumField, Thick_value_shell1
- Organizes elements into the corresponding components.

Radioss

Elements

- Assigns a single thickness at the element level based on the Thickness calculation method.
    - Uses element attribute THICK.

Properties/Sections on components

- Creates new components based on the Organization method.
- Creates new properties/sections based on the Organization method.
- Assigns the properties/sections to the corresponding components.
- Assigns a single thickness to each properties/sections based on the Thickness calculation method, using property attribute THICK.
- Organizes elements into the corresponding components.

Shrink Wrap Meshing

Shrink wrap meshing is a method to create a simplified mesh of a complex model when high-precision models are not necessary, as is the case for powertrain components during crash analysis.

The model's size, mass, and general shape remains, but the surface features and details are simplified, which can result in faster analysis computation. You can determine the level of detail retained by determining the mesh size to use, among other options.

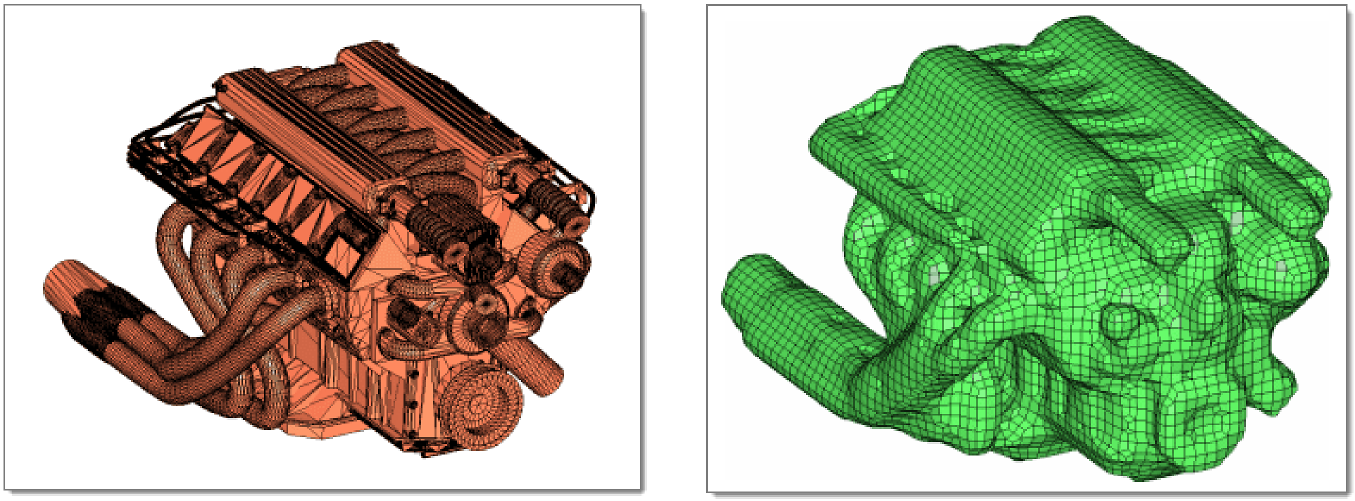


Figure 797:

You can shrink wrap elements, components, surfaces, solids, node clouds, or point clouds.

- Shrink wrap allows for wrapping of multiple components if they are selected.
- Selection provides the option to wrap all nodes, elements, components, surfaces, points, or solids, or only a certain portion of the model if desired. The input to the shrink wrap, that is, the model parts that you wish to wrap, can consist of 2D or 3D elements along with surfaces or solids.

One use case for shrink wrapping is when you need to convert an `.stl` representation of a model into a tria/quad mesh; using shrink wrap provides a quick and efficient way of achieving this. Similarly, crash analysis does not require a highly-detailed powertrain model; in such cases you can use the shrink wrap mesh to quickly generate a simplified approximation of a detailed powertrain model. Crash analysts can then use that coarser shrink wrap representation within the crash simulation model.

Other reasons to use the shrink wrap include being able to stitch over very bad geometry to generate an enclosed volume mesh for tetra-meshing. The shrink wrap tool can work from elements, whether 2D or 3D, or geometry. Thus, in the case of an "unclean" geometry model with many released (free) edges, you can either generate any arbitrary mesh on the unclean geometry using the automesh functionality beforehand and then creating shrink wrap or you can simply select the surface or solid without meshing the geometry first; either of these steps will yield good output mesh. The key in such cases is to ensure that the element size used for the shrink wrap is large enough to stitch over the unclean surface edge splits so that an enclosed volume can be created.

Shrink wrap mesh can be generated as a surface mesh, using a loose or tight wrapping, or as a full-volume hex mesh by use of the Shrink Wrap panel. The distinction between surface or volume mesh is an option labeled generate solid mesh.

Loose Shrink Wrap Mesh

Loose wrap wraps the selected elements or components, surfaces, or solids with the target element size specified, and outputs an outer-volume mesh which approximately adheres to the original FE topology.

A smaller element size generates a shrink wrap mesh that more closely approximates the original FE representation and adheres to more features; a larger element size produces a more basic mesh which ignores more features. The loose wrap does not project the nodes of the shrink wrap mesh to the original mesh, and typically the shrink wrap mesh will have an offset from the original mesh, again, the offset is dependent on the target element size used.

Once the shrink wrap meshing process has completed the new elements will be created in the current component. For every new run of the shrink wrap mesh, both loose and tight, it may be necessary to create a new component collector if you wish the elements to be placed in another collector other than the current component collector.

Comparison of Varying Mesh Sizes (Shell Output)

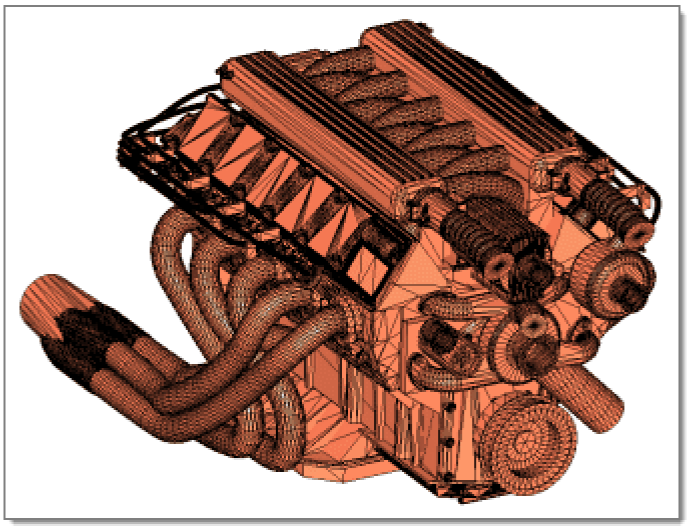


Figure 798: Original .stl Model

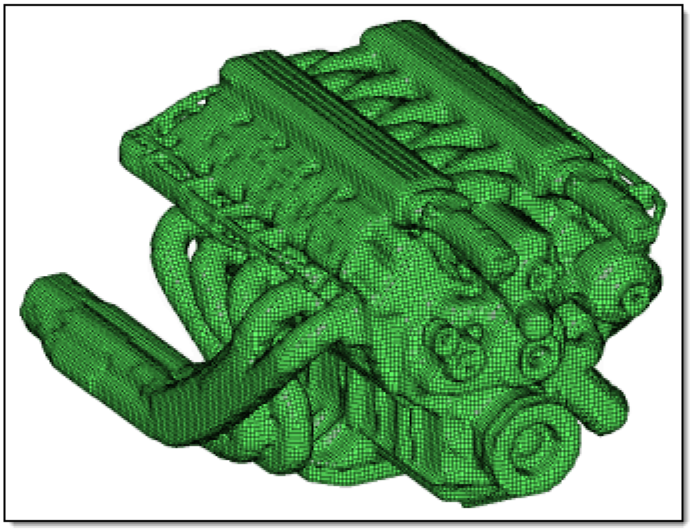


Figure 799: 2mm Mesh

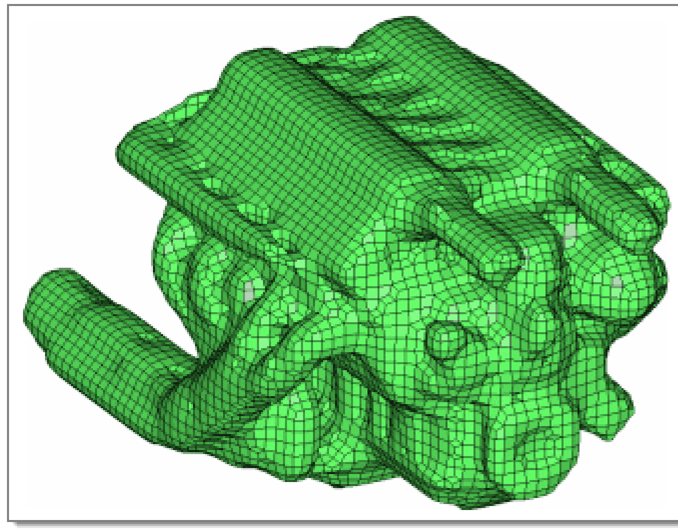


Figure 800: 5mm Mesh

Comparison of Altering the Jacobian Value for Solid Mesh Generation

Within both tight and loose wrap algorithm's there is an option to generate solid mesh. This will generate an all Hexa mesh on completion of the shrink wrap. When the **generate solid mesh** checkbox is selected it will expose a minimum jacobian input; this option essentially hexa meshes the part with this element quality critieria defined. It controls the hexa quality, which is directly linked to the adherence to the topological features of the original component. The jacobian value must be between 0 and 1. The nearer the value is to 1, the cruder the output will appear, the mesh will be more heavily voxelised. When the value is closer to 0, you are allowing the shrink wrap solid mesh algorithm to smooth and adhere to more features, while still maintaining the solid mesh minimum jacobian element quality. By default the minimum jacobian value is 0.3.

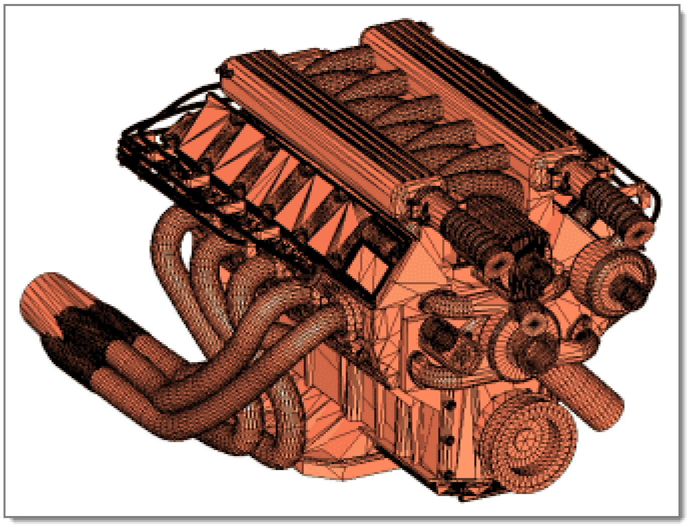


Figure 801: Original .stl Model

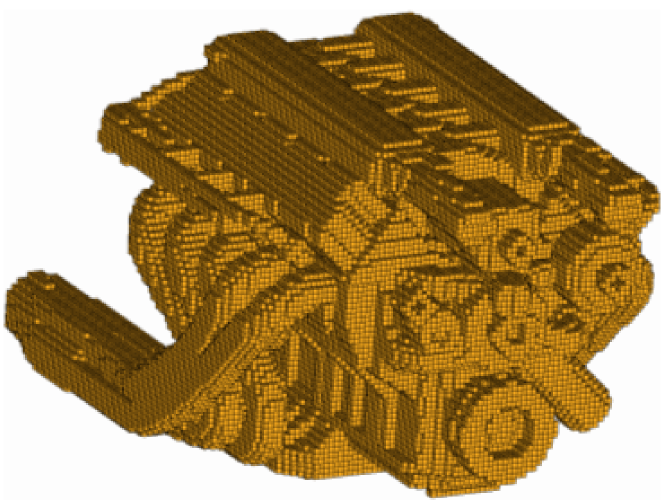


Figure 802: 2mm Solid Mesh, Jacobian=1.0

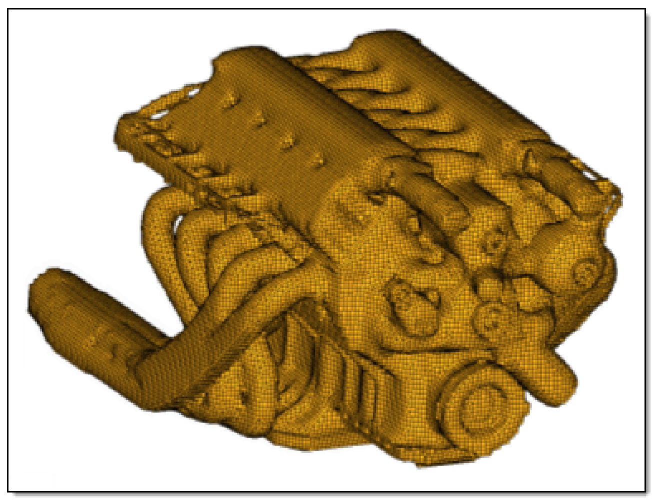
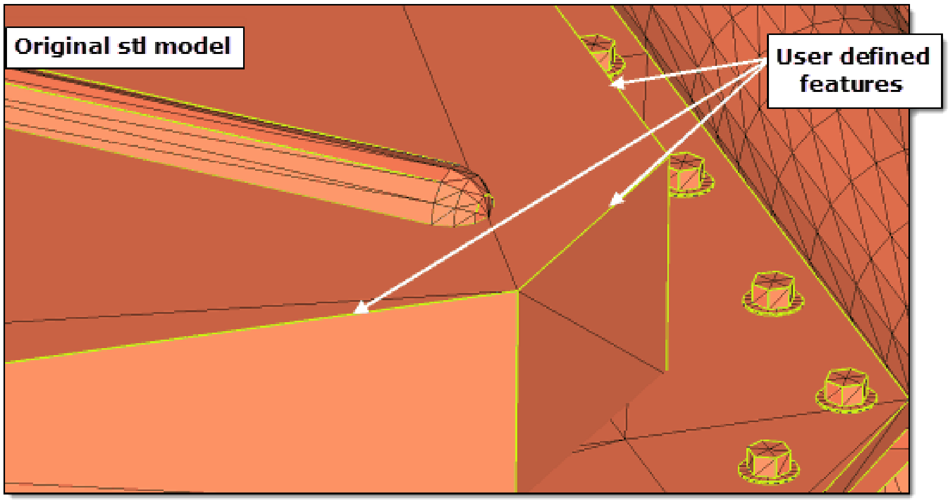
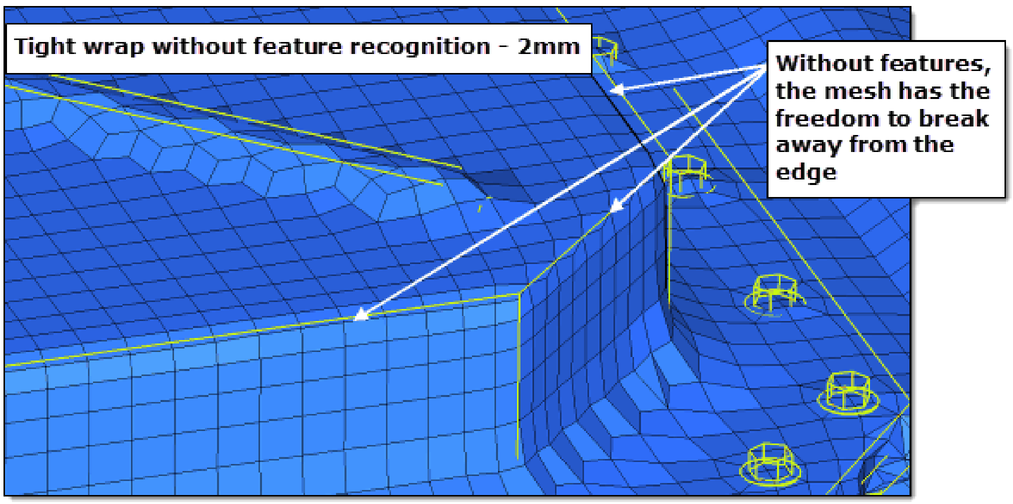


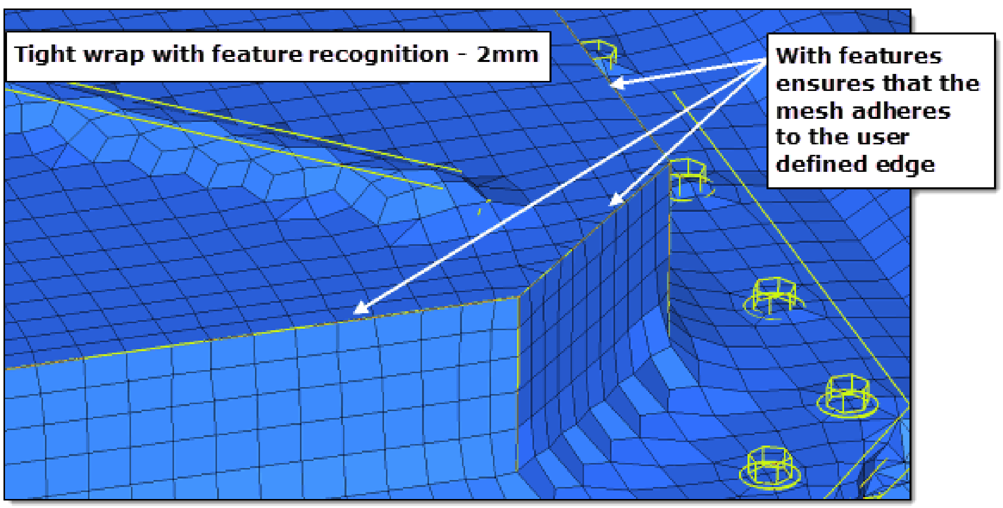
Figure 803: 2mm Solid Mesh, Jacobian=0.3

Shrink Wrapping with Feature Recognition

An additional option can be used to manually define features which will be adhered to during the meshing process. Typically, when using shrink wrap the mesh attempts to follow features, but has some freedom to break away from original edges of the part. However, when the features are manually selected within the panel, the resultant shrink wrap mesh will follow the chosen features. This can be important when defining a face of a component that may be in contact with other parts, or there may just be a feature that needs to be recognized and adhered to and cannot be approximated for whatever reason.


Original stl model
User defined features


Tight wrap without feature recognition - 2mm
Without features, the mesh has the freedom to break away from the edge


Tight wrap with feature recognition - 2mm
With features ensures that the mesh adheres to the user defined edge

you can use this option. By default the mesh orientation always adheres to the global system, however, you can generate a local coordinate system and override the default behavior.

In the example below, you can see the original mesh, the default shrink wrap mesh using the global system, and the new re-orientated mesh using the local coordinate system.

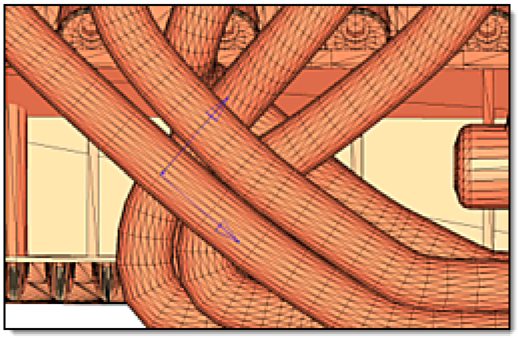


Figure 805: Original .stl Model

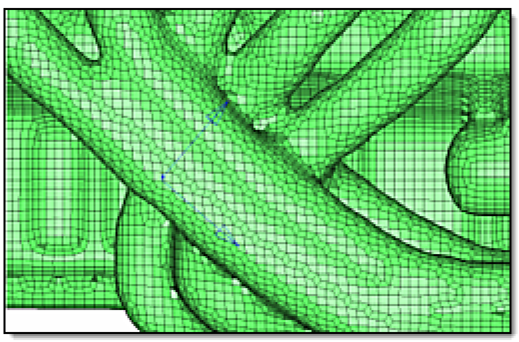


Figure 806: Shrink Wrap Output using Global System

Default shrink wrap mesh using the global system.

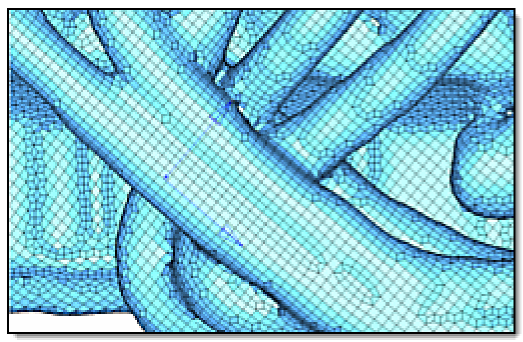


Figure 807: Shrink Wrap Output using Local System

Rows of elements in the reoriented mesh run along the tubes rather than at angles across them.

Tight Shrink Wrap Mesh

Tight wrap creates a wrapped surface mesh which adheres as closely as possible to the original FE topology representation, automatically detecting and following the surface features of the model.

The accuracy of the output is dictated by the element size: the larger the element size the less detail, the smaller the element size the more detail. This algorithm works differently than the loose wrap in that it projects the nodes of the shrink wrap to the original mesh, hence it is able to more accurately capture features.

Comparison of Tight and Loose Meshing

Notice the differences between tight and loose meshing, especially in the pulleys on the front of the engine and the resulting width of the individual cylinder exhaust pipes.

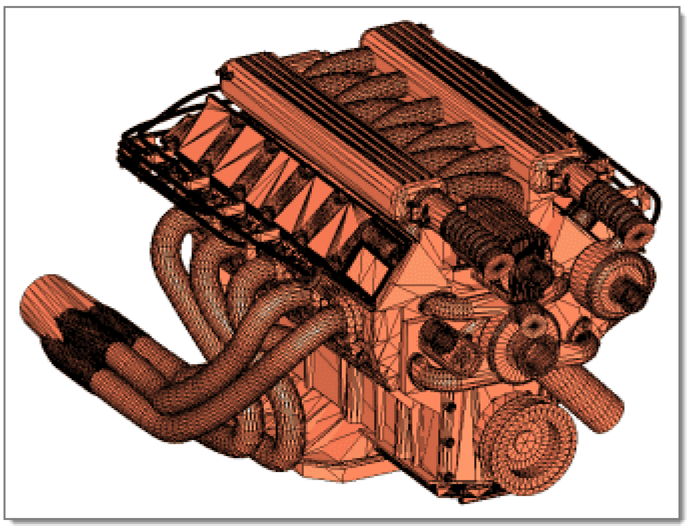


Figure 808: Original Model

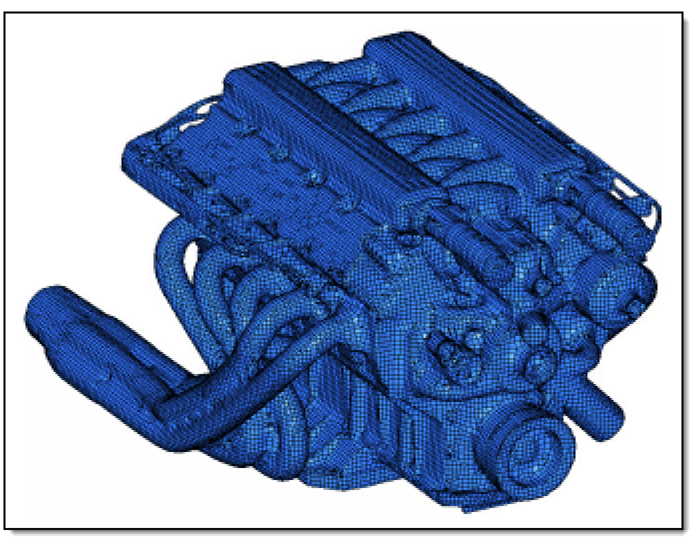


Figure 809: 2mm Tight Wrap

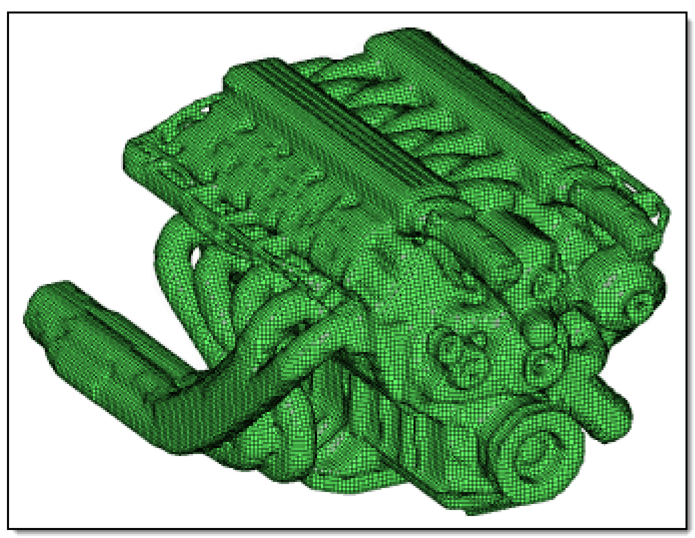


Figure 810: 2mm Loose Wrap

Comparison of Altering the Jacobian Value for Solid Mesh Generation

Within both tight and loose wrap algorithms there is an option to generate solid mesh. This will generate an all-hexa mesh on completion of the shrink wrap. When the **generate solid mesh** checkbox is active it exposes a minimum jacobian input; this option essentially hexa meshes the part with this element quality critieria defined. It controls the hexa quality which is directly linked to the adherence to the topological features of the original component. The jacobian value must be between 0 and 1. The nearer the value is to 1 the cruder the output will appear (the mesh will be more heavily voxelised). When the value is closer to 0, you allow the shrink wrap solid mesh algorithm to smooth and adhere to more features while maintaining the solid mesh minimum jacobian element quality. By default the minimum jacobian value is 0.3.

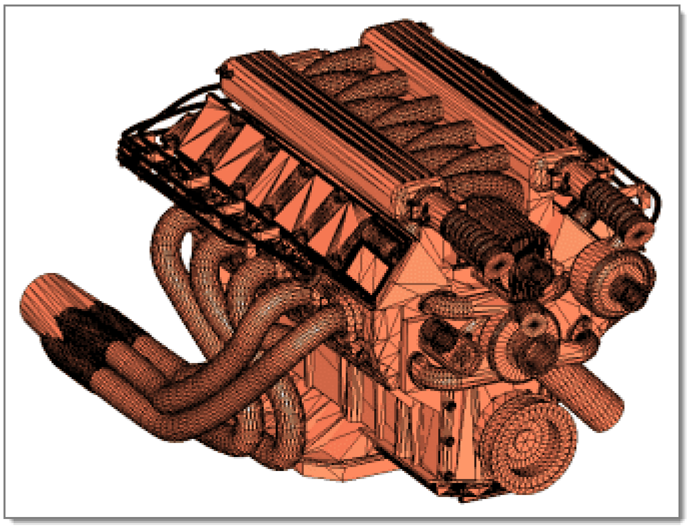


Figure 811: Original Model

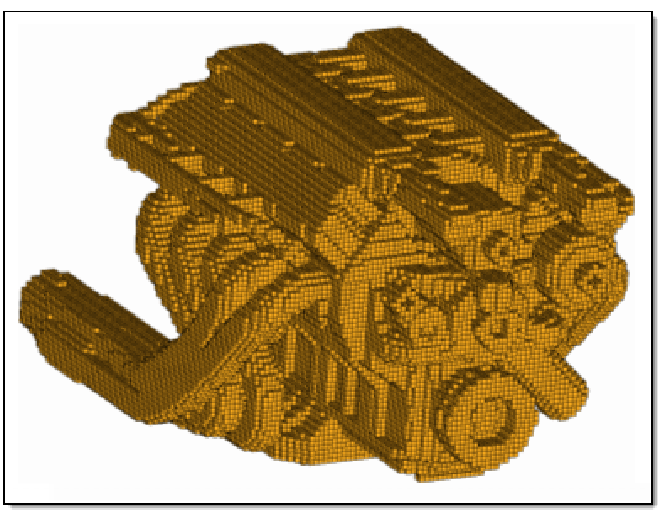


Figure 812: 2mm Solid Mesh, Jacobian=1.0

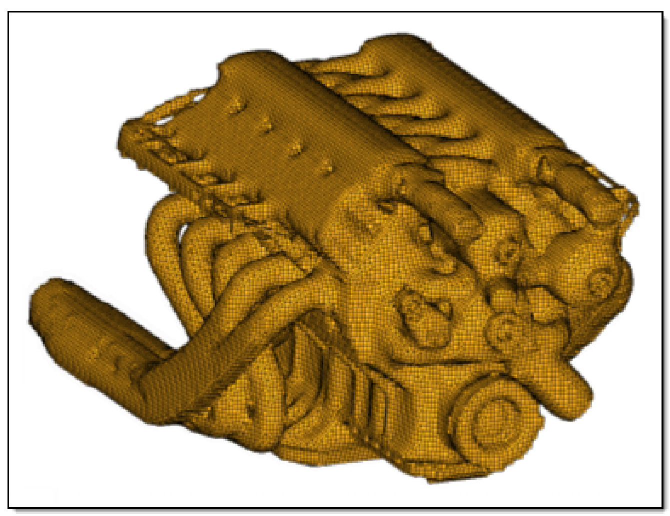
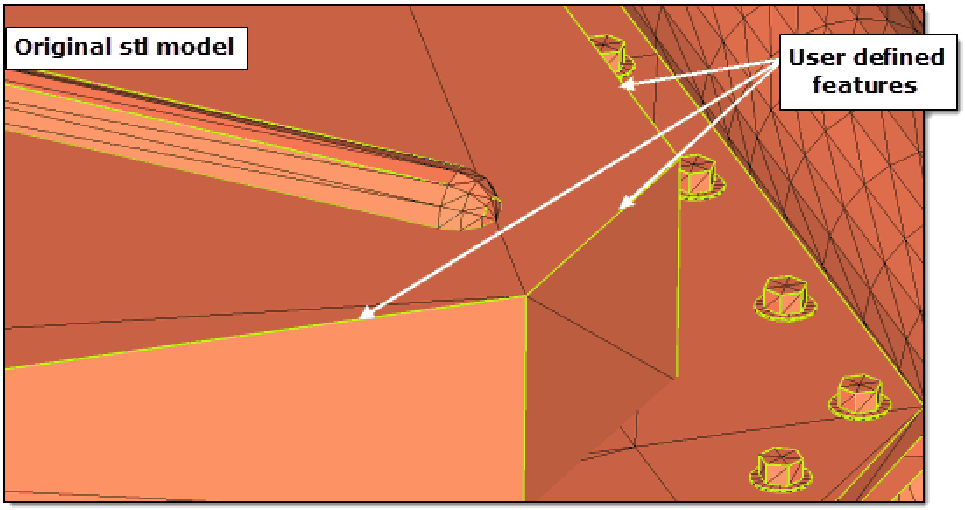
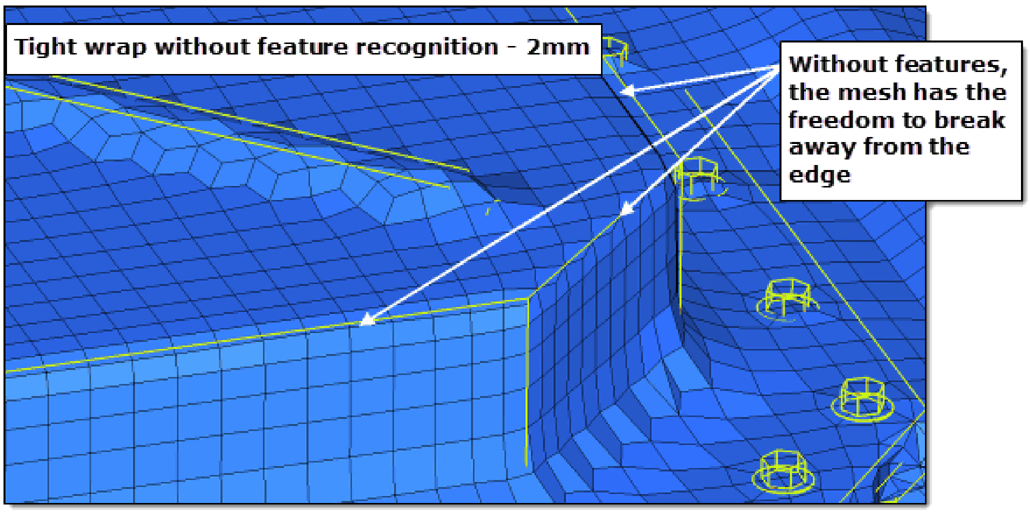


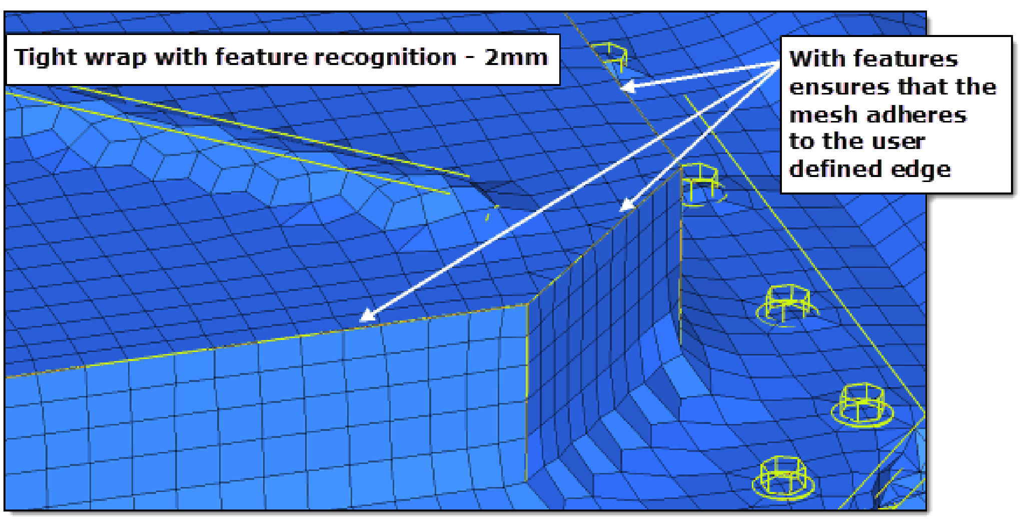
Figure 813: 2mm Solid Mesh, Jacobian=0.3

Shrink Wrapping with Feature Recognition

An additional option can be used to manually define features which will be adhered to during the meshing process. Typically, when using the shrink wrap the mesh attempts to follow features, but has some freedom to break away from original edges of the part. However, when the features are manually selected within the panel the resultant shrink wrap mesh will follow the chosen features. This can be important when defining a face of a component that may be in contact with other parts, or there may just be a feature that needs to be recognized and adhered to and cannot be approximated for whatever reason.


Original stl model
User defined features


Tight wrap without feature recognition - 2mm
Without features, the mesh has the freedom to break away from the edge


Tight wrap with feature recognition - 2mm
With features ensures that the mesh adheres to the user defined edge

you can use this option. By default the mesh orientation always adheres to the global system, however, you can generate a local coordinate system and override the default behavior.

In the example below, you can see the original mesh, the default shrink wrap mesh using the global system, and the new re-orientated mesh using the local coordinate system.

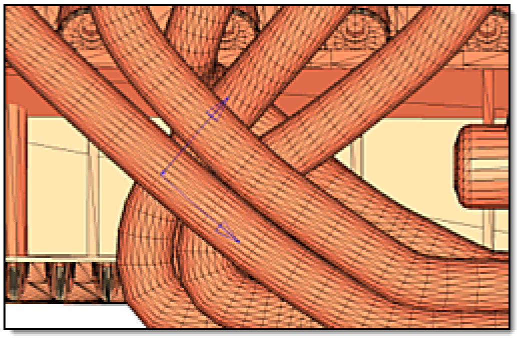


Figure 815: Original .stl Model

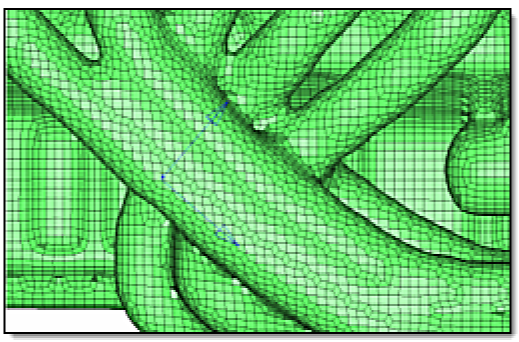


Figure 816: Shrink Wrap Output using Global System

Default shrink wrap mesh using the global system.

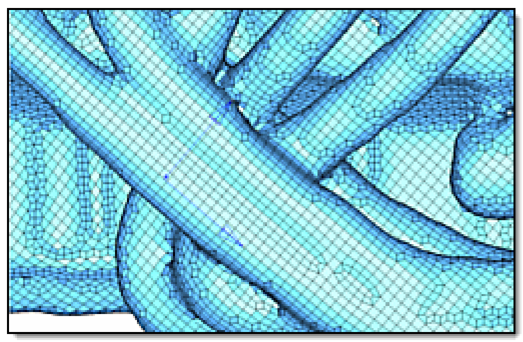


Figure 817: Shrink Wrap Output using Local System

Rows of elements in the reoriented mesh run along the tubes rather than at angles across them.

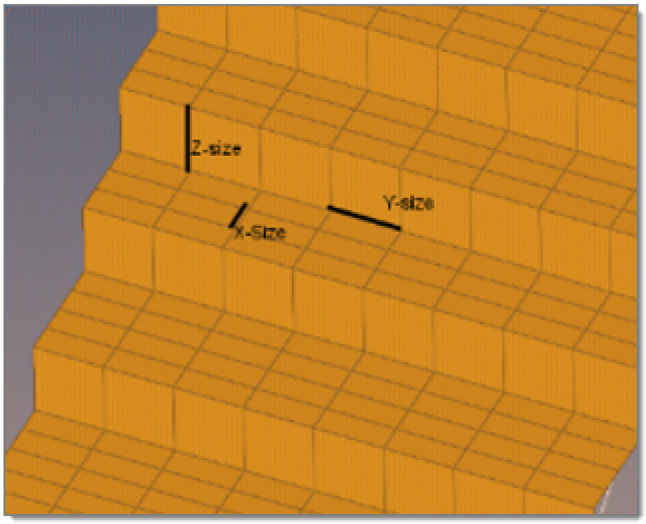
EM Lattice Meshing

EM Lattice meshing is a method to create an axis-parallel mesh for 2D and 1D geometry input.

Create EM Lattice Mesh

1. From the menu bar, click **Mesh** > **Create** > **EM Lattice Mesh**.
   The **EM Lattice Mesh** tool opens.
2. Use the Lines and/or Surfaces selector to select 1D and/or 2D entities.

3. In the Origin fields, enter the origin that defines the left most corner of the domain bounding box within which input entities have to be considered.
4. In the Voxel size fields, enter the size discretization for Lattice along the x, y, and z axes, respectively.



5. In the Num voxels fields, enter the number of voxels that define the right most corner of the domain bounding box.

   The values you enter define the length, breadth, and height of the bounding box in terms of the number of voxels along the axes. The rightmost extreme of the domain bounding box is calculated as:

   $voxel_{sz_x} * mesh_{ext_x}, voxel_{sz_y} * mesh_{ext_y}, voxel_{sz_z} * mesh_{ext_z}$
6. Click **Mesh**.

The Lattice mesh is created within the specified bounding box.

2D BL Meshing

2D BL meshing is a method to create a 2D mesh with or without boundary layers on planar sections defined by sets/groups of edges defining closed loops.

A region is considered closed if it is entirely bounded by edge elements (edge elements should be of type PLOTEL). Element configurations generated by 2D BL meshing are linear quadrilateral (quad4) and triangular (tria3).

Create 2D BL Mesh

Mesh 2D planar areas with boundary layers.

Restriction: Only available in Engineering Solutions when the CFD user profile is loaded.

1. Open the Utility Browser by clicking **View** > **Utility menu** from the menu bar.
2. Under Generate Mesh, click **Generate 2D BL Mesh**.
   The **2D Boundary Layer Mesh** dialog opens.
3. Select the **2D Native BL (planar)** tab.
4. In the Default Value fields, enter the default values that apply to most components.
   a) For 1st Layer Thickness, enter the thickness for the first layer of elements.
   b) For Growth Rate, enter the boundary layer thickness growth rate from layer to layer.
   c) For Bound Type, select a boundary layer type.
      Choose **Wall** to generate boundary layers along the component edges. No boundary layers are generated when Bound Type is set to Farfield, In/Outlet, and Symmetry.

   > **Note:**
   > Edge elements in collectors having Bound Type defined as Farfield, In/Outlet, and Symmetry will be used to define the geometry, but they will not dictate element size/density.

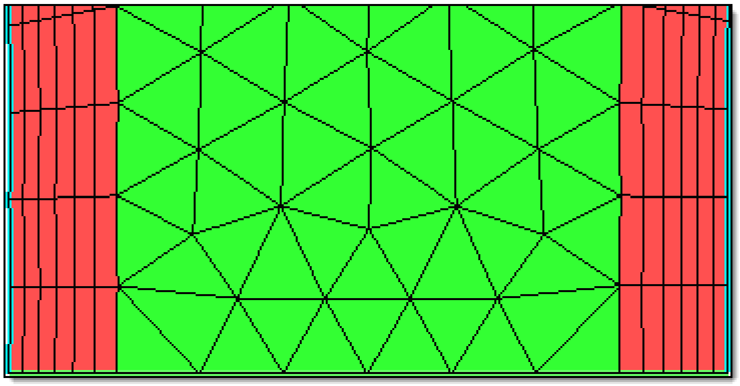


Figure 818:

5. Click Add collector to select or add components containing edge elements (elem type PLOTEL) that define the boundaries of the 2D section.

   Default values (1st Layer Thickness, Growth Rate and Bound Type) are assigned to the selected components.

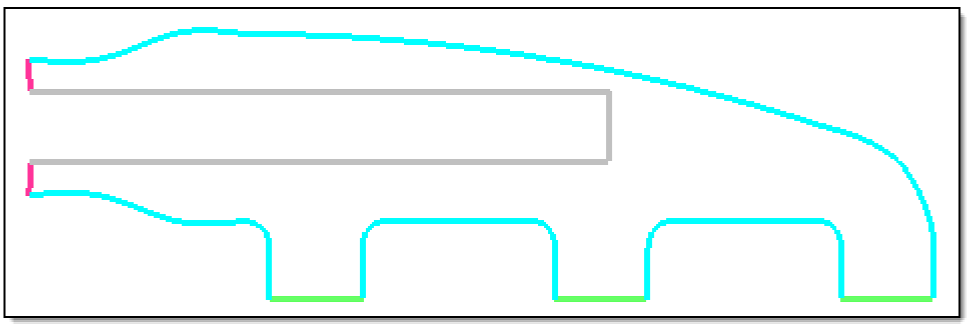


Figure 819:

6. To maintain the node seeding on the edge without BL as much as possible, select the **Retain node seeding on edge w/o BL** checkbox.

   The nodes on the non-BL boundary, which are located inside the BL, will not be maintained. Only the nodes which are located inside of the BL will be maintained.
7. In the Number of boundary layers field, enter the number of boundary layers to generate.
8. To control the aspect ratio of boundary layer elements by refining the edges to generate boundary layer elements that satisfy the Max perimeter element aspect ratio value, select the **Allow boundary node insertion** checkbox.

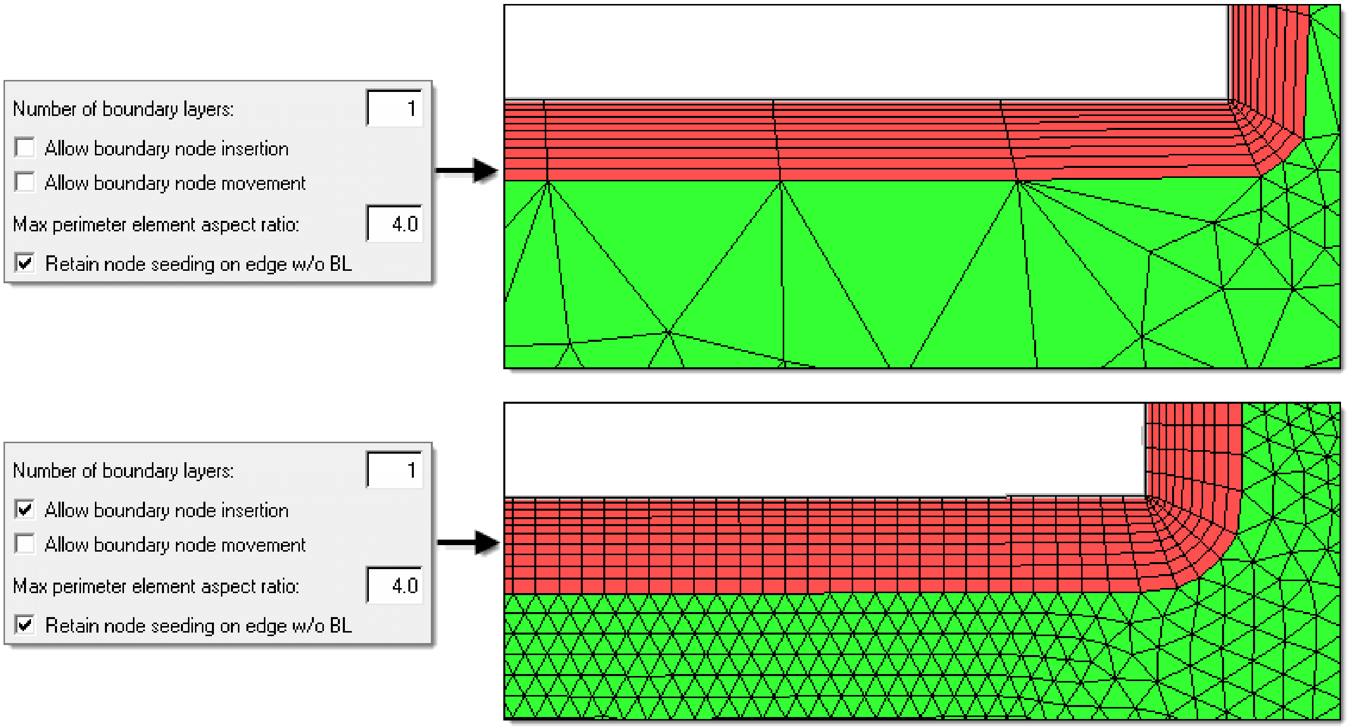


Figure 820:

9. To control the aspect ratio of boundary layer elements by boundary node movement so that generate boundary layer elements will satisfy the Max perimeter element aspect ratio value, select the **Allow boundary node movement** checbox.

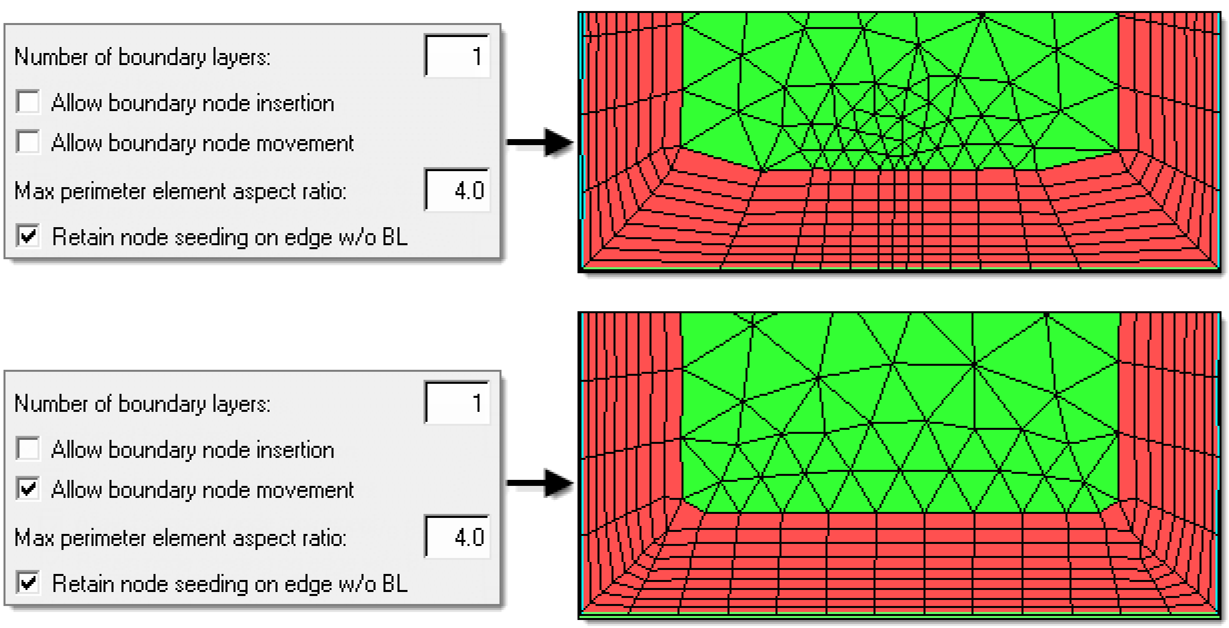
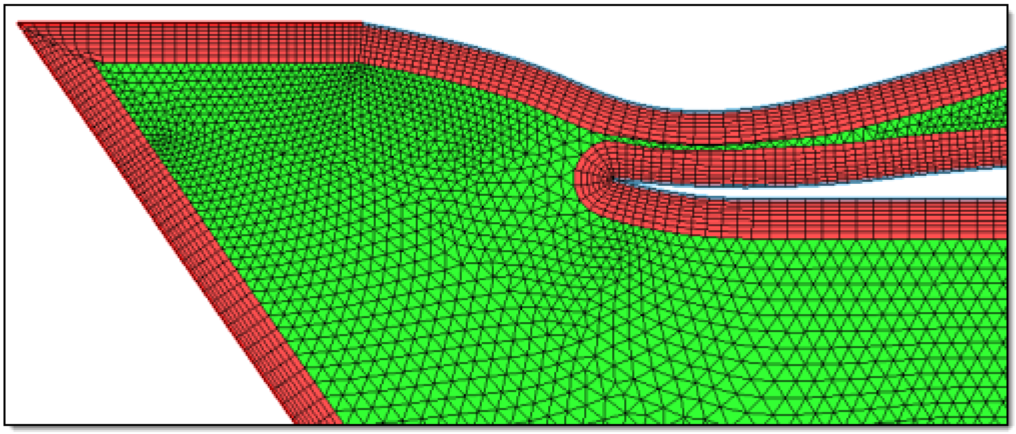
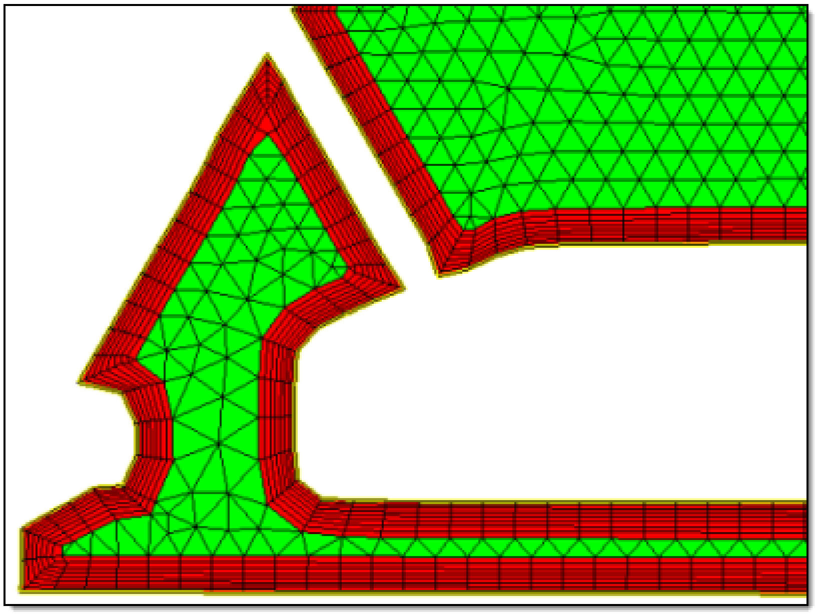


Figure 821:

10. Click **Generate 2D BL mesh**.

The 2D BL mesh is generated.





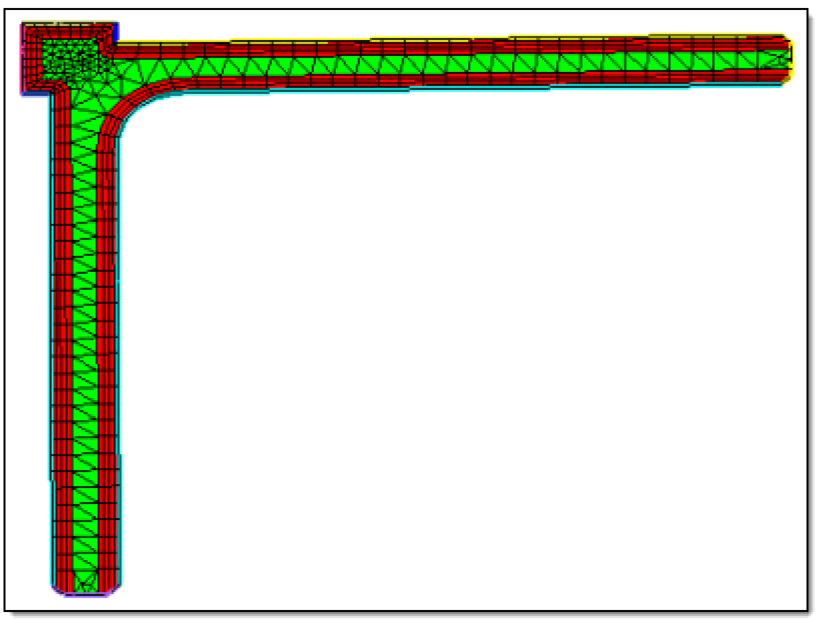


Figure 822:

Periodic Meshing

Periodic meshing is a method to create a grid containing rotational and/or translational symmetric boundaries, for example rotating machinery.

Create Periodic Mesh

Before you begin, generate a mesh on the source geometry using any of the HyperMesh meshing algorithms, for example 2D mesh with BL.

1. From the menu bar, click **Mesh** > **Create** > **2D Elements** > **Periodic Mesh**.
   The **Periodic Mesh** dialog opens.
2. Select surfaces.
   a) Use the Source: Surfaces selector to select source geometry.
   b) Use the Target: Surfaces selector to select target geometry.
3. Use the Symmetry options to describe the mesh transformation between source and target geometry.

   > **Note:** The Translation and Rotation options can be used simultaneously for one transformation.

   - To define the translation vector, select the **Translation** checkbox.
   - To define the rotation by an arc, defined by three nodes, or by an axis and a rotation angle, select the **Rotation** checkbox.
4. Check mesh and geometry.
   - To whether source and target geometry are rotational and/or translational symmetric or not, click **Check/Modify Geometry**.
   - To check for source and target mesh to determine if symmetry exists, click **Check Mesh**.
5. Click **Map Mesh**.

The mesh from the source geometryis mapped onto the target geometry.

After a successful mapping step, information about the transformation is plotted in the log region of the dialog.

Volume Meshing

Volume mesh or "solid meshing" uses three-dimensional elements to represent fully 3D objects, such as solid parts or sheets of material that have enough thickness and surface variety that solid meshing makes more sense than 2D shell meshing.

3D Elements

Supported 3D elements.

Tetra Elements

3D tetrahedra elements.

Tetra4

Configuration 204 - 3D (1st order) tetrahedra elements with 4 nodes ordered in HyperMesh.

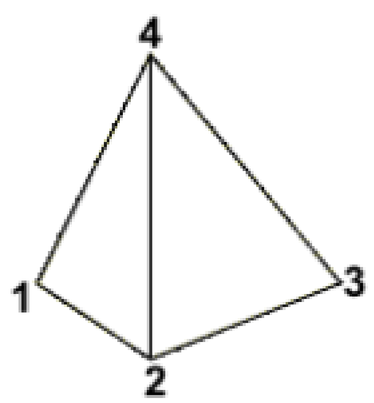


Figure 823: Element Configuration 204, 4-Noded Tetra

Tetra10

Configuration 210 - 3D (2nd order) tetrahedra elements with 10 nodes ordered in HyperMesh.

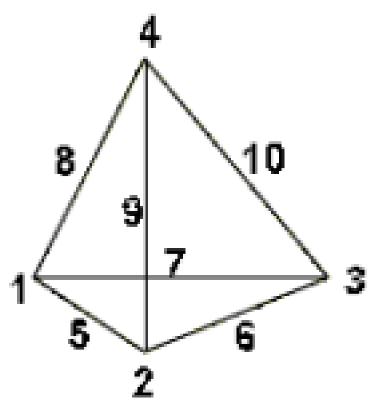


Figure 824: Element Configuration 210, 10-Noded Tetra

Penta Elements

3D triangular prism pentahedra elements.

Penta6

Configuration 206 - 3D (1st order) triangular prism pentahedra elements with 6 nodes ordered in HyperMesh.

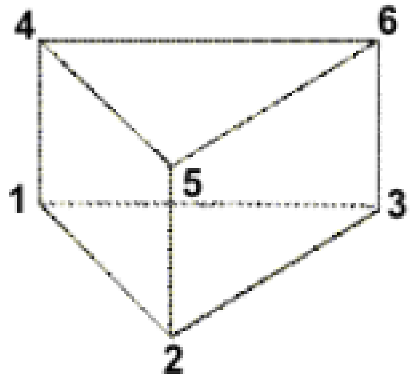


Figure 825: Element Configuration 206, 6-Noded Penta

Penta15

Configuration 215 - 3D (2nd order) triangular prism pentahedra elements with 15 nodes ordered in HyperMesh.

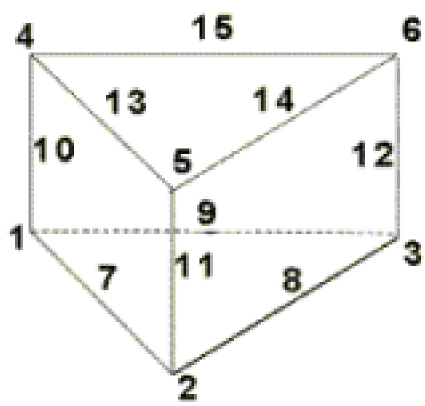


Figure 826: Element Configuration 215, 15-Noded Penta

Hex Elements

3D hexahedra elements.

Hex8

Configuration 208 - 3D (1st order) hexahedra elements with 8 nodes ordered HyperMesh.

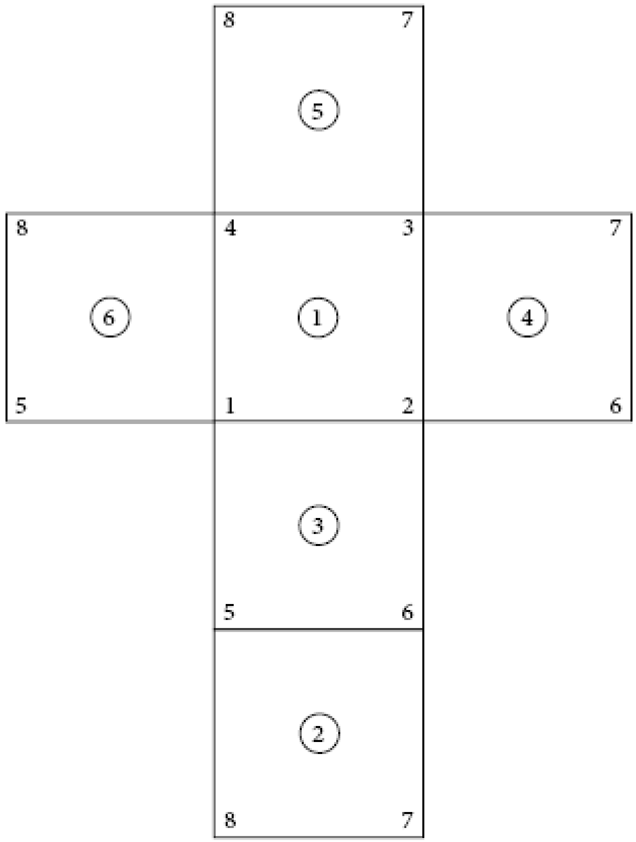


Figure 827:

Hex20

Configuration 220 - 3D (2nd order) hexahedra elements with 20 nodes ordered in HyperMesh.

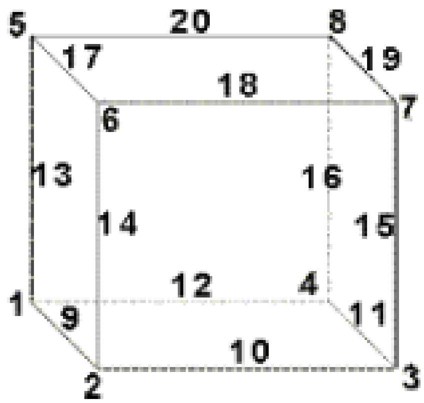


Figure 828: Element Configuration 220, 20-Noded Hexa

Solid Map Meshing

Solid Map meshing is a method that creates a mesh of solid elements in a solid geometric volume.

Solid Map Meshing Multiple Solids

You can select multiple solids for solid map meshing provided that each individual solid is in fact mappable. However, the meshing engine cannot always mesh every selected solid in a single operation, even when all the selected solids display as mappable.

This may happen if the mappable constraints from different solids within the selection contradict each other. For example, one type of mappable constraint is that certain surfaces (along faces) of a mappable solid must be of the map-mesh type. When such constraints are in conflict, faces that caused meshing to fail are marked with a red square icon.

In Figure 829, both solids can be map-meshed individually, but solid 1 (the triangular one) must have all of the marked faces (5, 6, and 9) map-meshed. This, however, causes a conflict for solid 2, which can only be map-meshed by using the shared surface (6) as a destination. This conflicts since this shared surface must match the meshes on surfaces 4 and 8 in order to mesh solid 2.

In such a case, you can mesh the remaining solids by deselecting the ones that are marked with the red icon but retaining the others in your selection. This either allows you to mesh the unmarked solids in a single action, or helps you further diagnose the problem. The remaining solids, unfortunately, will require individual meshing or further partitioning before they can be solid-map meshed as a group.

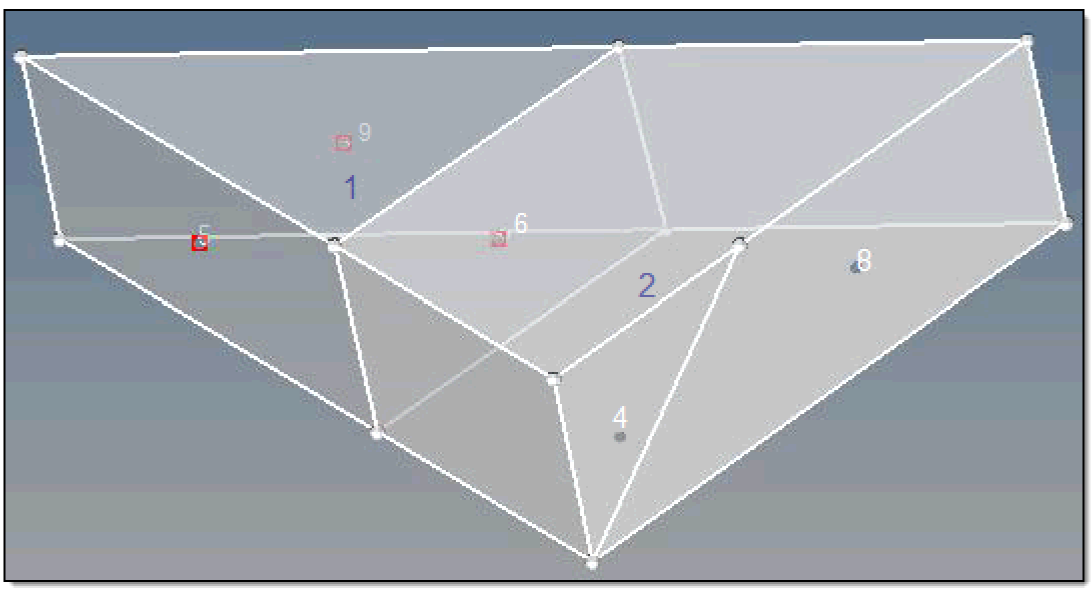


Figure 829:

Partition Solids for Mappability

Before you can successfully solid mesh a model, ensure that the solids have been partitioned so that they are either one directional or three directional mappable.

Solid (3D) meshing can be done automatically, just like 2D shell meshing, but often requires that complex parts be partitioned into groups of smaller, simpler, connected solids instead of one large complex solid. In solid meshing, the ability to be meshed is referred to as mappability.

Mappability is directional and can be likened to putting a surface mesh on one face of the solid, then extending that mesh along a vector through the solid volume. So, for example, a perfect cylinder is mappable in one direction, the axis between its top and bottom faces, while a perfect cube is mappable in three, the axes between each pair of its identical faces. However, a combustion engine's cylinder head consisting of two cylinders of different radius joined together into a single solid entity would need to be partitioned to divide the two cylinders. Once partitioned, each cylinder would become mappable in one direction.

> **Note:** Even when all partitioned sections of the solid are mappable, this does not necessarily mean that they can all be meshed at once. In some cases they may need to be meshed a few at a time, or even individually in extreme cases. Mappability only ensures that the partitioned section can be meshed.

Use the Mappable visualization mode to review solid partitioning for mappability. The “Mappable” mode color codes the solids within the model according to whether the solids are solid meshable. The ignored map, not mappable, 1 directional map, and 3 directional map all relate to the mappable state of the solids.

> **Tip:** Change the color coding for the mappable state from the Options panel, Colors subpanel.

When reading in a new model with solids, the model will be colored blue after you activate the Mappable visualization mode, which indicates that the mappability is currently being ignored. It is then necessary to partition the model so that the state of the solids changes to 1 directional or 3 directional.

> **Note:** If the model does not include any solids, for example, only surfaces are present, you can use the Solids panel on the Geom page to create solids from the surfaces.

If some partitioning has already occurred from a previous session when the `.hm` file has been read in with the Mappable visualization mode already active, it will still be displayed as "ignored" map. To invoke the mappable algorithm calculation, change to another visualization mode, such as By Topo and then change back to Mappable again. This recalculates the state of all solids within the model.

1. From the Visualization toolbar, set the geometry visualization mode to **Mappable**.
   The solid is color codes according to its mappability state.

   Blue

   Solid has not been edited, and therefore cannot evaluated for mappability.

   Orange

   Solid has been edited, but remains completely unmappable (further partitioning may enable mapping).

   Yellow

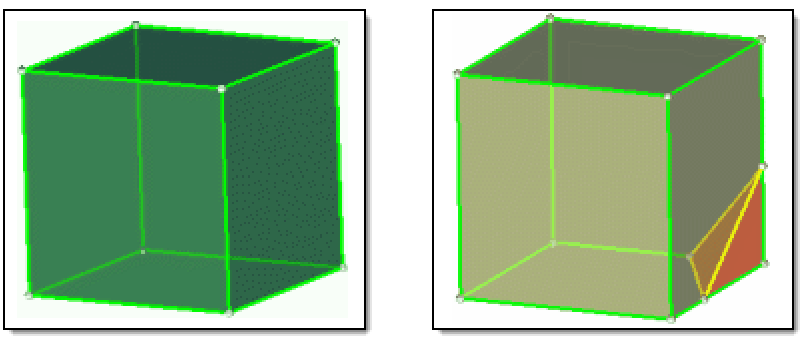
   Solid is mappable in 1 direction.

   Green

   Solid is mappable in three directions. This is very rare.

   Figure 830:

   The first cube is mappable in 3 directions, but if a corner is split off it becomes mappable in only 1 direction, and the corner is not mappable without further partitioning.



2. Partition solids for mappability.

Any solid edit operation will update the display of the solid entities automatically.

Partitioning Solids for Mappability

Figure 831 shows that one trim of the model by a single surface (the top surface of the rectangular shaft, in this case) has created two additional solids within the model. One solid remains in the ignored map state (blue), one is now not mappable (orange) and one is one directional mappable (yellow - transparent).

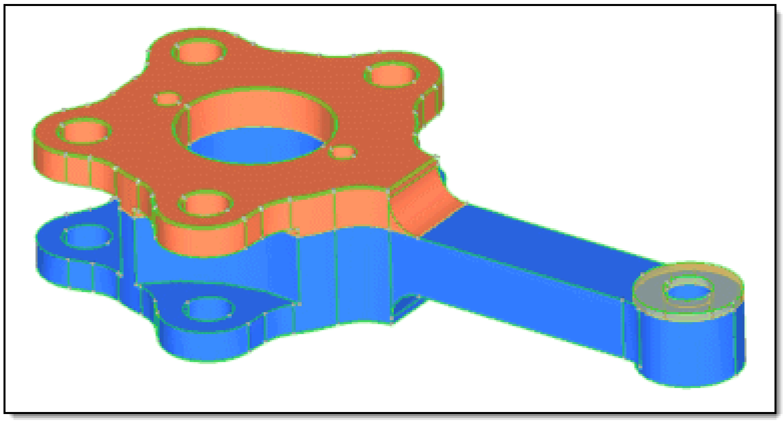


Figure 831:

After additional partitioning the model using the tools found in the Solid Edit panel, the model has transformed from having an ignored map and non-mappable states to having only one directional and three directional mappable states. Figure 832 shows one three directional mappable solid, as indicated by the green transparent solid at the base of the shaft where it joins the part's main body.

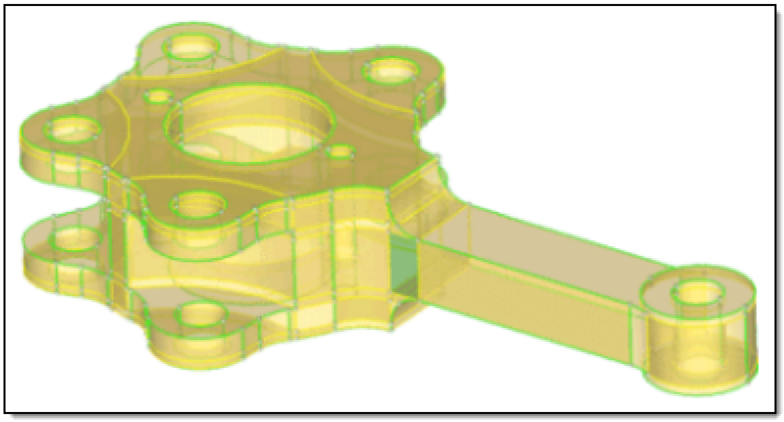


Figure 832:

Once partitioning is successful, meshing can commence. Accessing the Multi-Solids subpanel and selecting all of the solids, plus the required meshing options, yields a complete 3D mesh for the entire complex part, as shown in Figure 833.

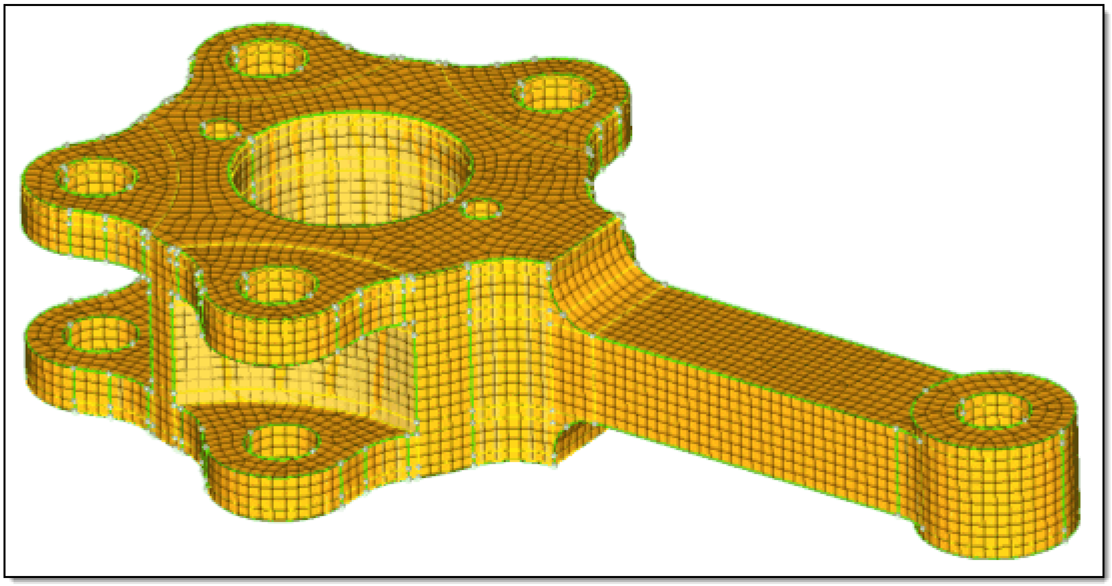


Figure 833:

Selecting and masking a section of the elements confirms that the mesh is a complete 3D mesh, as opposed to just a surface mesh, as shown in Figure 834.

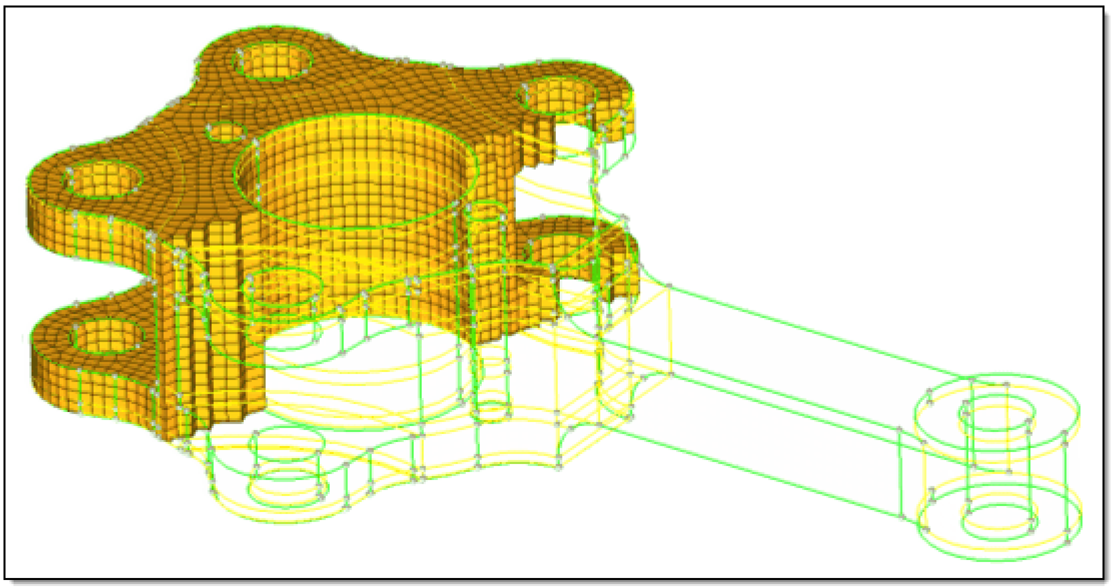


Figure 834:

Create Solid Map Mesh

Before you being, ensure that the solids have been partitioned so that they are either one directional or three directional mappable. Refer to, Partition Solids for Mappability.

Also, make sure you have an existing 2D mesh, which will be used to extrapolate the 3D solid map mesh.

This topic uses the General subpanel, but the Line Drag, Linear Solid, and Ends Only subpanels all draw from the same set of input controls. The Line Drag, Linear Solid, and Ends Only subpanels panels simply filter out the controls that do not apply to their mapping techniques.

- Use the Line Drag subpanel to select a 2D mesh, and then select a line from the model geometry to use as the mapping direction.
- Use the Linear Solid subpanel to select two existing 2D meshes and extrapolate a 3D mesh that connects them.
- Use the Ends Only subpanel to select two opposing surfaces and one 2D mesh, then extrapolate the mesh between the surfaces.

You can omit the source, the destination, or the along geometry by setting either one of the entity selectors to (none). Only one of these selections can be set to (none); the other two selections are then required to define the volume to fill.

1. From the 3D page, click **solid map**.
2. Select the **General** subpanel.
3. For source geom, select a geometry type and then select the source geometry that defines the source face of the 3D volume.
   - Choose **surfs** to select surfaces that define the source face of the volume/solid.
   - Choose **lines** to select lines that define the periphery of the source face.
   - Choose **nodes** to select multiple lists of nodes, each representing the periphery of the source face.
   - Choose **none** if you do not want to define source geometry. The geometry inferred from the elems to drag is considered as source geometry.
4. Use the elems to drag selector to select the elements/mesh that correspond to the source face extruded to create the solid mesh.
5. For dest geom, select a geometry type and then select the destination geometry that defines the destination face of the 3D volume.
   - Choose **surfs** to select surfaces that define the destination face of the volume/solid.
   - Choose **lines** to select lines that define the periphery of the source face.
   - Choose **nodes** to select multiple lists of nodes, each representing the periphery of the source face.
   - Choose **none** if you do not want to define destination geometry.
6. Select elems to match.
   - Choose **elems** to select elements on the desitnation surface that you wish the 3D mesh to match up with.
   - Choose **none** to access the smooth dest checkbox, which when selected, smooths the mesh that is mapped on the destination face when your destination geometry varies greatly from the source geometry.
7. For along geom, select a geometry type and then select the geometry that defines the face of the 3D volume along which you wish to map the mesh.

- Choose **surfs** to select surfaces to define the mapping face of the volume/solid.
- Choose **lines** to select lines that define the periphery of the source face.
- Choose **nodes** to select a node list that defines a line along which to map.
- Choose **mixed** to select any combination of surfaces, lines, 2D elements/shell faces, and/or nodelist/nodepath. When elements are used, the mapped solid mesh maintains the nodal positions with selected elements. They can be equivalenced to have common nodes. While selecting nodelist/nodepath, each selection should represent an edge that connects the source and destination.
- Choose **none** if you do not want to define geometry.

8. For along parameters, define the parameters required for the mesh along the solid map.

   This determines the number of elements along the depth of the mapping. If the size or density is set to "0", the element size/density is calculated based on the average element size of the source elements (elems to drag).
9. For along bias style, choose the type of biasing to use while creating nodes in the along direction.

   The biasing style works in conjunction with biasing intensity. If intensity is "0", biasing is not applied.
10. In the intensity field, enter a biasing intensity.
11. Click **mesh**.

HyperMesh displays the progress of the solid map meshing process in the status bar. Upon completion, HyperMesh displays a report of the mesh quality. The element quality value reported is the worst scaled Jacobian in the mesh. The scaled Jacobian's value may range from 0.0 to 1.0(best). An elem's scaled Jacobian is a ratio of the elem Jacobian over the Jacobian of an ideal elem of the same configuration.

Create Solid Map Mesh from One Volume

Create a new 3D mesh from a single, mappable solid volume.

Before you being, ensure that the solids have been partitioned so that they are either one directional or three directional mappable. Refer to, Partition Solids for Mappability.

Also make sure you have an existing 2D mesh, which will be used to extrapolate the 3D solid map mesh. When creating mesh from the One Volume subpanel, a 3D mesh can be automatically created directly on solids as long as the solids you select are already mappable.

1. From the 3D page, click **solid map**.
2. Select the **One Volume** subpanel.
3. For volume to mesh, select the solid/surface to mesh.
4. Define the direction of mesh mapping.
   a) For source hint, select the “beginning” surface.
   b) For dest hint, select the “ending” surface.
5. For source shells, select the type of elements to use when creating the resulting output solid mesh.

   This defines the 2D mesh on the initial surface of the solid, and will dictate the output element type when meshing the solids.

- Choose **mixed** to use hexa and penta elements.
- Choose **quad** to create hexa elements.
- Choose **trias** or **R-trias** to create only penta elements (right-angle pentas in the case of R-trias).

6. Select which component to put the newly-created elements.
   - Choose **elems to solid/surf comp** to organize elements in the same component that contains the solid and its surfaces.
   - Choose **elems to current comp** to organize new elements in the current component.
7. To smooth the elements on the resulting face of the solid to improve the resulting mesh quality, select the **smooth dest** checkbox.
8. For along parameters, define the parameters required for the mesh along the solid map.

   This determines the number of elements along the depth of the mapping. If the size or density is set to "0", the element size/density is calculated based on the average element size of the source elements (elems to drag).
9. For along bias style, choose the type of biasing to use while creating nodes in the along direction.

   The biasing style works in conjunction with biasing intensity. If intensity is "0", biasing is not applied.
10. In the intensity field, enter a biasing intensity.
11. Click **mesh**.

HyperMesh displays the progress of the solid map meshing process in the status bar. Upon completion, HyperMesh displays a report of the mesh quality. The element quality value reported is the worst scaled Jacobian in the mesh. The scaled Jacobian's value may range from 0.0 to 1.0(best). An elem's scaled Jacobian is a ratio of the elem Jacobian over the Jacobian of an ideal elem of the same configuration.

Create Solid Map Mesh from Multiple Solids

Create a new 3D mesh from multiple, mappable solids.

Before you being, ensure that the solids have been partitioned so that they are either one directional or three directional mappable. Refer to, Partition Solids for Mappability.

Also make sure you have an existing 2D mesh, which will be used to extrapolate the 3D solid map mesh. When creating mesh from the Multi Solids subpanel, a 3D mesh can be automatically created directly on solids as long as the solids you select are already mappable.

HyperMesh can create volume meshes on multiple shapes at the same time, allowing you to mesh solid parts faster.

> **Note:** Complex parts must still be partitioned into multiple simpler solids. Creating volume meshing on multiple shapes simultaneously does not always work for a large numbers of solids, even if they are all mappable. In some cases you may need to mesh them a few at a time or even, in extreme cases, individually.

1. From the 3D page, click **solid map**.

2. Select the **Multi Solids** subpanel.
3. Use the solids selector to select the desired solids.
4. Define the direction of mesh mapping.
   a) For source hint, select the “beginning” surface.
   b) For dest hint, select the “ending” surface.
5. Select the meshing mode.
   - Choose automatic to automatically create the 3D solid mesh.
   - Choose interactive to manually define mesh density and element mesh patterns before creating a final mesh.
6. In the elem size field, enter the size to be used when initially distributing the nodes to the edges.
7. For source shells, select the type of elements to use when creating the resulting output solid mesh.

   This defines the 2D mesh on the initial surface of the solid, and will dictate the output element type when meshing the solids.
   - Choose **mixed** to use hexa and penta elements.
   - Choose **quad** to create hexa elements.
   - Choose **trias** or **R-trias** to create only penta elements (right-angle pentas in the case of R-trias).
8. Select which component to put the newly-created elements.
   - Choose **elems to solid/surf comp** to organize elements in the same component that contains the solid and its surfaces.
   - Choose **elems to current comp** to organize new elements in the current component.
9. To smooth the elements on the resulting face of the solid to improve the resulting mesh quality, select the **smooth dest** checkbox.
10. To keep the solid elements generated more perpendicular to the surface faces in the along direction, select the **apply orthogonality to along** checkbox.
11. To halt the meshing routines upon the creation of a bad jacobian solid element, select the **stop meshing on bad jacobian** checkbox.
12. To force the mesh to honor any prior edge node density settings when creating the temporary surface mesh, select the **previous settings** checkbox.
13. Click **mesh**.
    If the **automatic** mesh mode is selected, a solid mesh is created. If **interactive** mesh mode is selected, temporary 2D shell meshes are created and the node seeding density is assigned to all of the along edges and the Density subpanel opens.

Tetra Meshing

Tetra meshing fills an enclosed volume with first order or second order tetrahedral elements.

A region is considered enclosed if it is entirely bounded by a shell mesh (tria and/or quad elements). Hexahedral, wedge, and pyramid element configurations can also be generated during tetra meshing.

These elements are typically generated when you need boundary layer type meshes on certain areas of the volume surface.

Create Tetra Meshes with the Tetramesh Panel

You can specify some elements to be fixed, and others to be floatable.

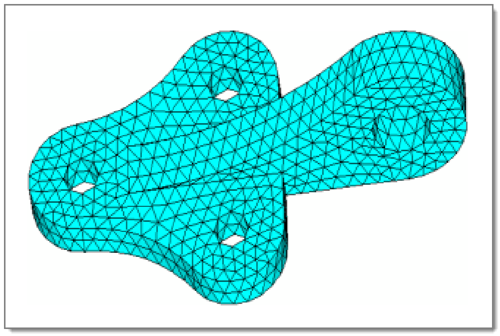
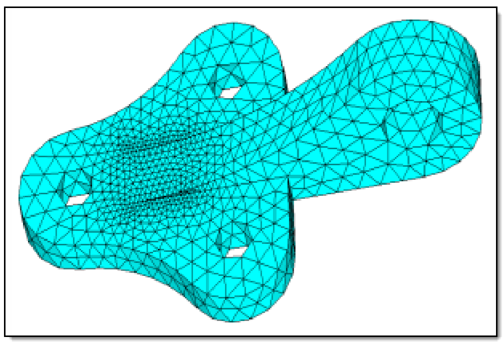
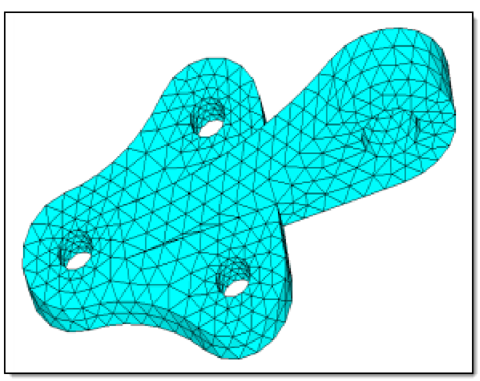
A fixed tria-quad element is one that must be exactly represented as a face of a tetra/penta-pyramid/hexa element in the final mesh. A floatable element is one whose nodes locations are used, but the exact connectivity of those nodes can be modified if it produces a better mesh. Unless you need a special mesh type, such as surface layers of pentas/hexas, you should select as fixed only those elements that must match a pre-existing mesh, leaving the rest floatable. If the bounding surface contains quad elements, and if these quad elements are defined as fixedelements, then a first layer of elements is generated on the boundary, and pyramid elements are generated from the quad faces. However, when quad elements are defined as float elements, they are split into two trias, and the tetra meshing proceeds normally.

You can also specify various growth options in order to control the tradeoff between the number of tetras generated and their quality. Higher, more aggressive growth rates produce fewer elements, but they may be of poorer quality.

In the Tetramesh panel you can choose from three different mesh generation priorities. The **generate mesh normally** option applies in most cases, but if your solver is particularly sensitive to element quality, use the **optimize element quality** option. This directs the tetramesher to spend more time trying to generate better quality elements. In particular, it employs the volumetric ratio (CFD "skew") measurement for rating potential tetras. For some applications, element quality considerations are less important than mesh generation time. In those cases, choose the **optimize meshing speed** option.

1. From the 3D page, click **Tetramesh**.
2. Access different types of tetra meshing from the subpanels.

Subpanel	Description
Tetra mesh	Fills an arbitrary volume, defined by its surface using tria/quad elements, with tetrahedral elements.
Tetra remesh	Regenerates the mesh for a single volume of tetrahedral elements.
Volume tetra	Given a solid entity or a set of surfaces representing a closed volume, this meshing option generates a shell mesh and fills the enclosed volume with solid elements. You can choose to create a shell mesh (2D) using quads, trias, or mixed elements and a solid mesh (3D) using tetrahedral elements only or mixed (tetras and penta) elements. In addition, you can use proximity meshing, which refines the mesh in areas where the features are small and closer together.

Subpanel	Description
	 *Figure 835:* You can also use surface curvature as a function of element density as shown below. This option creates finer mesh in areas of high surface curvature.  *Figure 836:* When you select quads or mixed as your 2D element type, HyperMesh creates quad elements and splits them diagonally into two trias during tetra face creation. This can create tetra elements whose triangular faces are right triangles (90-45-45 angles) instead of equilateral triangles (60-60-60 angles).  *Figure 837:*

Subpanel	Description
	Note: Sometimes the meshing may fail to correctly interpolate from the surface mesh; when this occurs, the shell elements are cleaned up according to the same settings used in the Quick TetraMesh macro on the Utility menu, and a second attempt is made. This means that some of the features in a model may be smoothed over.
Tetramesh parameters	Sets general qualities of the tetrameshing engine, such as a maximum element size, growth rate, the balance between speed and element quality, or whether to perform smoothing operations after initial meshing.
Refinement box	Lets you define a specific box-shaped volume within an existing teramesh in which to generate finer mesh.

Create Tetra Mesh with the Tetramesh Process Manager

Navigate through the different steps necessary to create a high quality tetramesh quickly using the Process Manager.

Not every step in the process is mandatory; you can follow the entire process or just a few steps, based on your requirements.

> **Note:** Execute steps only once, unless a reject option is available within the current step. If you are not sure about the outcome of the next step, save the process before going further.

Effective tetrameshing requires a standard surface mesh to use as a base from which the 3D tetras are built, therefore the beginning steps of the process revolve around achieving quality surface meshes.

Specific geometry cleanup and meshing algorithms are implemented in the Tetramesh Process Manager template to handle complex parts efficiently.

A specific tetramesh process instance can be saved in a process manager template file (`*.pmi`) so its parameters can be reused later on similar parts, allowing for increased efficiency and implementation standards. The Tetramesh Process template can be used for automated geometry and element cleanup (Batchmesher).

Create New Session

1. From the menu bar, click **Mesh** > **Create** > **TetraMesh Process** > **Create New**.
2. In the **Create New Session** dialog, enter a session name and location where the template and related files will be saved and click **Create**.

The Process Manager browser and panel open.

> **Tip:**
> You can go back and forth between template panels and core panels to take advantage of core functionality by undocking the The Process Manager browser and panel open. panel. To dock and undock the The Process Manager browser and panel open. panel, click .

Import Geometry

Before you begin, make sure the Process Manager, Import Geometry panel is open.

1. Select geometry.

To import	Do this
Geometry	**1.** For Import Type, select **Geometry**. **2.** For File Type, select the type of geometry to import. **3.** In the Import Filename field, navigate to the desired file. **4.** To automatically save an imported CAD file as an HyperMesh file after the import completes, select the **Save HM File After Import** checkbox. **5.** In the Scale Factor field, enter a scale factor if the imported model is in a spatial scale other than what you want to work with. This feature can also be used to convert the model's measurements between metric and English units, for example. **Note:** Only spatial units are affected, mass and similar qualities are not. **6.** For Cleanup tol, choose a method to perform cleanup on imported geometry. • Choose **Automatic** to use the global cleanup tolerance specified on the Options panel, Geometry subpanel. • Choose **Manual** to enter your own tolerance. **7.** To move all entities marked as blank to a special component, select the **Import blanked (no show) components** checkbox. By default, the translator ignores all of the blank flags in the file.
HyperModel Model	**1.** For Import Type, select **HM Model**.

To import	Do this
	2. In the Import Filename field, navigate to the desired file.

2. Click **Import**.
3. Click **Next**.

Cleanup Geometry

Before you begin, make sure the Process Manager, Geometry Cleanup panel is open.

1. Cleanup geometry.

To cleanup geometry by	Do this
Equivalencing free edges	1. Click the **Free Edges** tab. 2. In the Tolerance field, enter the maximum distance across which surfaces can be made equivalent. Surfaces with a gap between them greater than this value will not be made equivalent. 3. Click **Equivalence**. Edges and other features that fall within the tolerance value of each other are combined.
Displaying free edges by edge type	1. Click the **Edge Tools** tab. 2. Select the checkboxes (Free Edges / T-junctions) for the edge types to display. 3. Adjust display. • Click **Isolate** to display only the surfaces attached to the selected edge types. • Click **Display All** to remove the mask and view all surfaces.
Create filler surfaces	1. Click the **Edge Tools** tab. 2. Click **Create** to generate filler surfaces for all the free edge loops. This functionality helps to eliminate issues caused from missing surfaces by creating the closed volumes necessary for tetrameshing.

2. Click either **Accept** or **Next** to continue.

Organize and Cleanup Holes

Isolate holes based on diameter ranges. Isolating and meshing holes separately from other geometry features is a very important step during tetra meshing; failing to do so can drastically affect the overall mesh quality. You may also wish to use this step even when there are no specific meshing requirements for holes just to organize your model.

Before you begin, make sure the Process Manager, Cleanup and Organize Holes panel is open.

1. Click **+** to add additional rows to the table if you require multiple sets of hole criteria.
2. In the D< field, enter diameter values to consider when selecting holes.

   Holes with diameters less than the value you specify will be automatically selected. If you add multiple lines to the table, HyperMesh generates non-overlapping ranges from them.

   > **Tip:** If you have no specific requirements for hole meshing, you can use a single diameter value that is larger than the largest holes in the model, in order to select all of the holes.

3. In the Num Circumference Elems field, enter the number of elements to place around the circumference of any holes of the specified diameter range.

   Higher numbers result in a smoother mesh that better approximates a hole's shape and curvature, but could impact tetramesh quality if they result in smaller elements around the hole's edges than the enforced element size for other surfaces.

   > **Tip:** If you have no specific requirements for hole meshing, it is recommend that you use at least 6 elements for each hole's circumference.

4. In the Longitudinal Elem Size field, enter the desired element size in the longitudinal direction.

   Longitudinal direction is the direction traveled along the depth of the hole.

   > **Tip:** If you have no specific requirements for hole meshing, it is best to use the same element size as the rest of the model.

5. Click **Auto Organize** to automatically organize the holes into separate components based on the input parameters, diameter, and so on.

   One component will be created for each row in the table, and the holes are placed into the components that their diameter and other criteria match. The model holes, and corresponding table rows, are color-coded according to the component that they have been sorted into.

   Clicking one of these buttons selects the entire row, making it the "active" row. To deselect an active row, click the colorless button at the very top-left corner of the table (the one in the header row along with the column names).

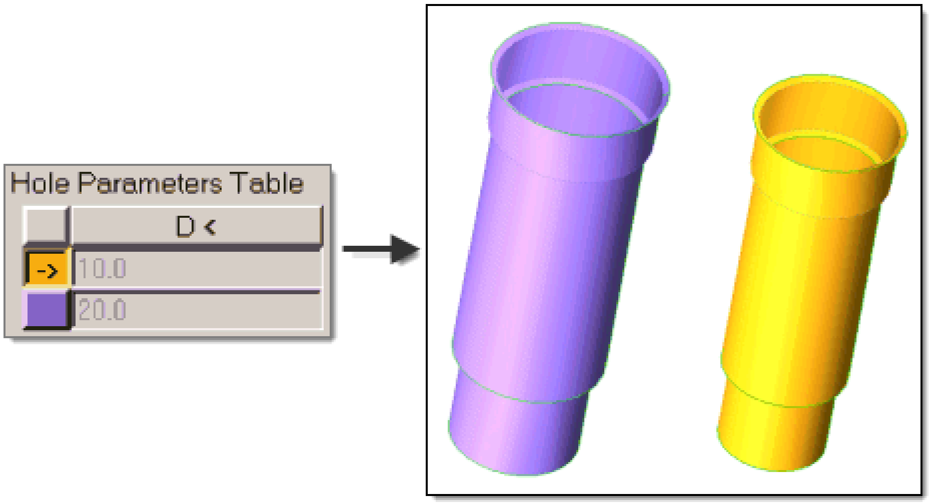


Figure 838:

6. Click **Organize** to manually organize any holes that were missed by Auto Organize into components using the Organize panel.
7. Click either **Accept** or **Next** to continue.

Mesh Holes

Before you begin, make sure the Process Manager, Mesh Holes panel is open.

1. From the Mesh Type column, select the mesh type to use when meshing the holes.

   There are two meshing options available: R-tria regular and R-tria union jack.

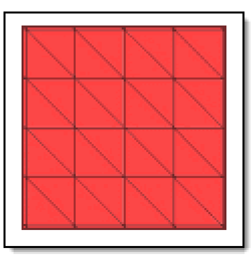


Figure 839: R-tria regular

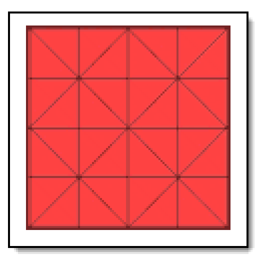


Figure 840: R-tria union jack

2. Mesh holes.

   Each row has a color button on its left which corresponds to the component holding hole surfaces. Clicking one of these buttons selects the entire row, making it the "active" row. To deselect an active row, click the colorless button at the very top-left corner of the table (the one in the header row along with the column names).

   - To mesh a single group of hole, select it's corresponding row and click **Mesh**.

- To mesh all of the holes simultaneously, click **Mesh All**.

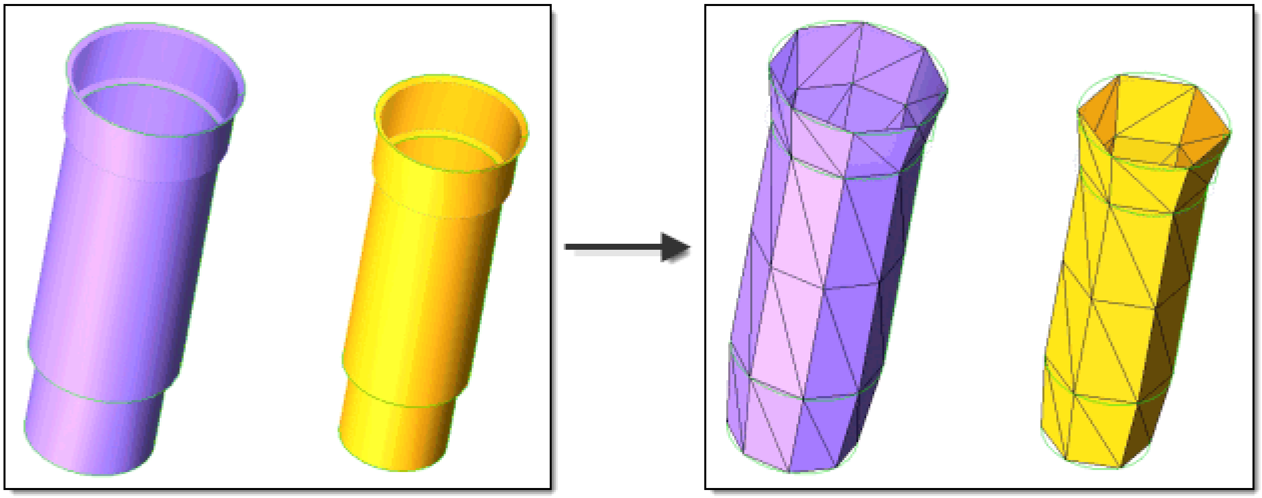


Figure 841:

3. Click either **Accept** or **Next** to continue.

> **Note:** To delete the mesh created for a group of holes, select it's corresponding row and click **Delete Mesh**.

Organize and Cleanup Features

Organize and cleanup user defined features such as water jackets, inlets, outlets and contact surfaces.

Before you begin, make sure the Process Manager, Organize & Cleanup Features panel is open.

Auto cleanup operations are based on the criteria set in the Batchmesher criteria file. Cleanup operations include equivalencing free edges, fixing of small surfaces (relative to the element size), and detection of features such as beads. It also performs specified surface editing/defeaturing operations like removal of pinholes below a specified size, removal of edge fillets, and the addition of a layer of washer elements around holes.

The Autocleanup panel performs the entire geometry cleanup portion of the Batchmesher. Since it performs a variety of geometry cleanup tasks, the results will not be instantaneous and can take a few minutes for large models. Cleanup criteria is determined by the Batchmesher parameter and element quality criteria files, both of which can be edited from within this panel using the Batchmesher Parameter Editor.

1. Create a feature by clicking **+**.
2. In the dialog that opens, enter a name for the feature and click **OK**.
3. Use the surfs selector to select the surfaces associated with the feature, which will be added to a new component.

   When selecting surfaces, surfaces are added "by face" to the selection to allow all of the surfaces associated with the same face in the model to be added simultaneously.

4. Click **proceed** to finish adding surfaces to the component.

   Features are sorted into color-coded components and the corresponding table rows are colored to match.
5. Select a feature's corresponding row to make it active, then click **Auto Cleanup**.
6. In the Autocleanup panel, perform automated geometry cleanup operations on the geometry contained in the selected component.
7. Click either **Accept** or **Next** to continue.

Organize and Cleanup Fillets

Defeature fillets, and replace the rounded corner where two lines come together with a point. Tangents are calculated at the beginning and at the end of each fillet, then those tangents are intersected to create a corner.

Before you begin, make sure the Process Manager, Fillets and Organize panel is open.

1. Use the Components selector to select the components that contain the fillets that require cleanup.
2. In the Min Radius field, enter a minimum fillet radius to cleanup.

   Any fillets in the model with radii of less than this value will be ignored, while fillets with radii greater than this will be cleaned up (removed so that the relevant lines meet at a sharp corner.)
3. In the Max Radius field, enter a maximum fillet radius to cleanup.

   Fillets with radii greater than this value will be ignored and remain.
4. To suppress such edges after the fillet removal process, select the **Suppress fillet tangent edges** checkbox.

   Because fillets are removed by generating tangent lines to replace them, these lines often result in extra edges in model geometry.

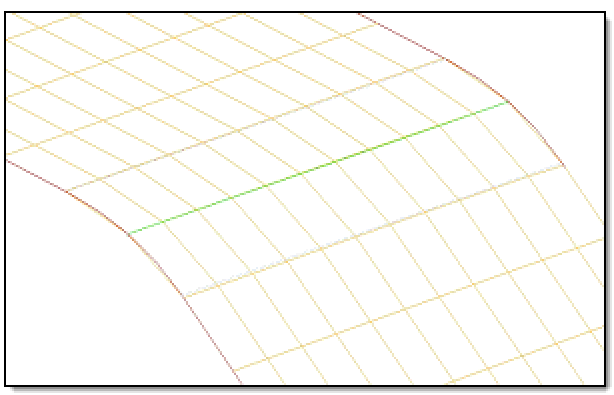


Figure 842: Suppress Fillet Tangent Edges On

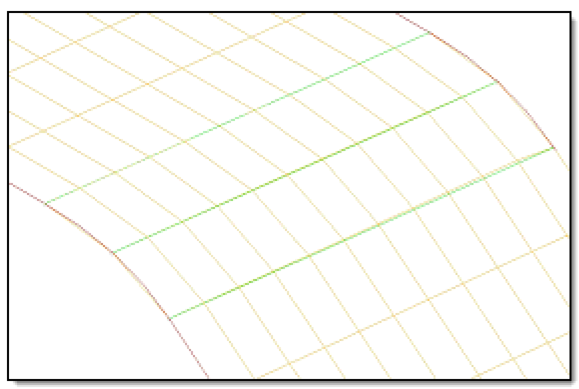


Figure 843: Suppress Fillet Tangent Edges Off

5. Click **Cleanup**.

The fillet midline split routine is executed based on what you have specified for the radius range and tangent suppression.

6. Click either **Accept** or **Next** to continue.

Mesh Features

Before you begin, make sure the Process Manager, Mesh Features panel is open.

1. For Mesh Type, select the type of elements used to generate the mesh for each feature.
2. In the Elem Size field, enter the element size used to generate the mesh for each feature.
3. Create mesh.
   - To create mesh for the selected/active row, click **Mesh**.
   - To create mesh for all rows, click **Mesh All**.
4. Click either **Accept** or **Next** to continue.

> **Note:**
> Delete the mesh from a specific component by selecting the desired row and clicking **Delete Mesh**.

Organize and Cleanup Global Surfaces

Organize and clean up all remaining global surfaces, that is, surfaces that were not already covered by holes and user-defined features.

Before you begin, make sure the Process Manager, Global Organize and Cleanup panel is open.

1. In the Min Element Size field, enter a minimum element size for all such components to prevent the autocleanup process from accidentally suppressing very small surfaces. Smaller values result in more aggressive geometry cleanup, but could potentially result in loss of some features.
2. Optional: Click **Organize** to organize remaining surfaces into appropriate components using the Organize panel.
3. Click **Auto Cleanup**.
   The Autocleanup panel opens, which can be used to clean all remaining surfaces.
4. Click either **Accept** or **Next** to continue.

Mesh/Remesh Global Surfaces

Mesh all remaining global surfaces, that is, surfaces that were not already covered by holes and user-defined features.

Before you begin, make sure the Process Manager, Mesh/Remesh panel is open.

1. In the Element Size field, enter the element size to use when meshing.
2. For Mesh Type, choose the type of element to use when meshing.
3. Click **Mesh**.

A mesh is created using the specified element size and type on all remaining "global" geometry.

> **Note:**
> If you do not like the results, click **Delete Mesh** to remove mesh from all such geometry, without affecting the specific items already meshed in previous stages, such as holes or user-defined features.

4. Click either **Accept** or **Next** to continue.

Cleanup Elements

Optimize the quality of the elements created during each of the previous meshing stages.

Before you begin, make sure the Process Manager, Element Cleanup panel is open.

- Perform automatic cleanup.

> **Note:**
> The nodes of features, such as ridges, will not be preserved if their angle is less than the specified minimum angle. In addition, element quality requirements may override feature angle preservation regardless of the angle specified.

a) Click the **Auto tab**.

b) Use the Components selector to select the components that contain the elements to cleanup.

c) In the Min Size field, enter the minimum size that is acceptable for any given elements.

d) In the Max Feature Angle field, enter the maximum feature angle to preserve.

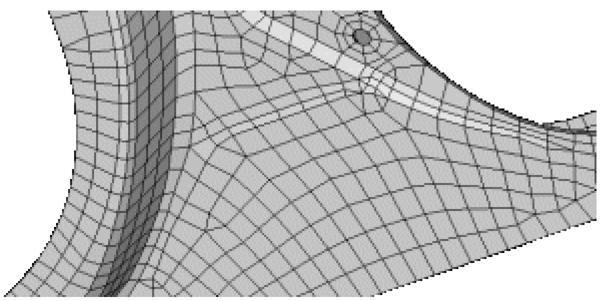


Figure 844: Minimum Feature Angle (Low Value)

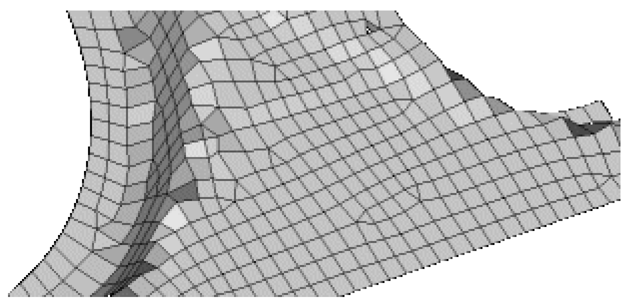


Figure 845: Minimum Feature Angle (High Value)

e) In the Normals Angle field, enter the maximum allowable angle between the normals of adjacent elements.

When possible, adjacent elements whose normals exceed this angle will be split into multiple smaller elements with less-extreme normal angles.

f) Click **AutoCleanup**.

g) Click either **Accept** or **Next** to continue.

The mesh is examined within each component individually, removing elements of poor quality and stitching the mesh together to fill the resulting gaps. Cleaning up the mesh on a per-component basis prevents mesh overlap between adjacent surfaces that belong to different components.

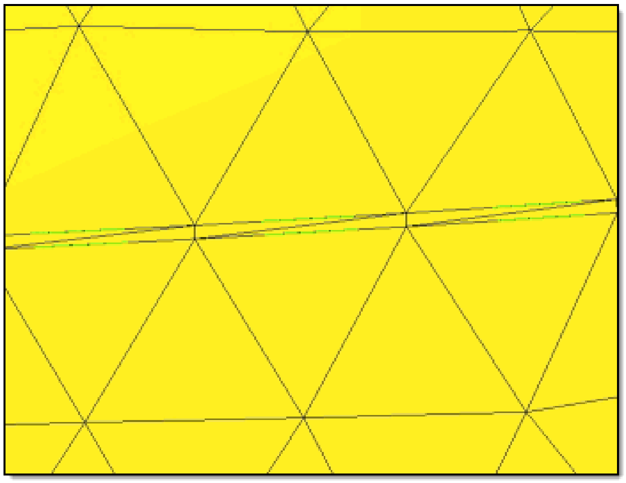


Figure 846: Before Element Cleanup

Notice the "sliver" elements along midline.

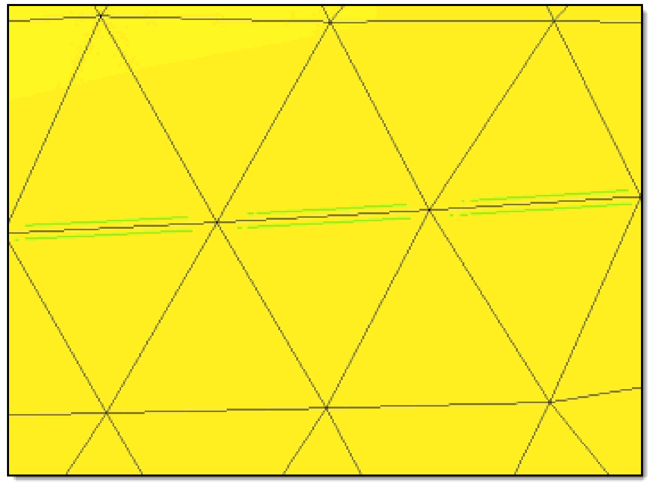


Figure 847: After Element Cleanup

Notice the small, sliver elements are removed.

- Perform manual cleanup.
  a) Click the **Manual** tab.
  b) Choose the type of edges to find.
     - Choose **Free** to find edges which are associated with only 1 surface.
     - Choose **T-Connections** to edges which are associated with three edges.
  c) Click **Find** to find edges.
  d) In the Tolerance field, enter the tolerance to use when identifying nodes associated with free edges and try to stitch them, leaving nodes outside the tolerance unchanged.
  e) Click **Fix** to fix free edegs.
  f) Click **Display Normals** to open the Normals panel, which can then be used to review normals.
  g) Click either **Accept** or **Next** to continue.

Perform Tetra Meshing

Generate the 3D mesh from the existing surface meshes.

Before you begin, make sure the Process Manager, Tetra Meshing panel is open.

By default, all elements belonging to holes and user-defined features will be set to tetramesh as Fixed trias/quads, while the rest of the model will be meshed using the "floatable" method.

Create Tetra Meshes with the Tetramesh Panel

Volume Shrink Wrap Meshing

Produce an all-hexa or all-tetra mesh based on the selected elements or geometry, then use the shrink wrap functionality as a quick mechanism to generate solid meshes.

> **Note:** When generating such a mesh, the Jacobian value has a large effect on the coarseness of the resulting volume mesh, as described below.

Using the shrink wrap mesh to achieve improved FE output from OptiStruct topology runs has also provided very good results which allow for quick tetra-meshing and, therefore, quick re-analysis after the optimization run.

Comparison of Altering the Jacobian Value for Solid Mesh Generation

Within both tight and loose wrap algorithm’s there is an option to generate solid mesh. This will generate an all hexa mesh on completion of the shrink wrap. When the **generate solid mesh** checkbox is selected it will expose a minimum jacobian input, this option essentially will hexa mesh the part with this element quality critieria defined, it controls the hexa quality which is directly linked to the adherence to the topological features of the original component. The jacobian value must be between 0 and 1. The nearer the value is to 1 the cruder the output will appear, the mesh will be more heavily voxelised. When the value is closer to 0, you are allowing the shrink wrap solid mesh algorithm to smooth and adhere to more features while maintaining the solid mesh minimum jacobian element quality. By default the minimum jacobian value is 0.3.

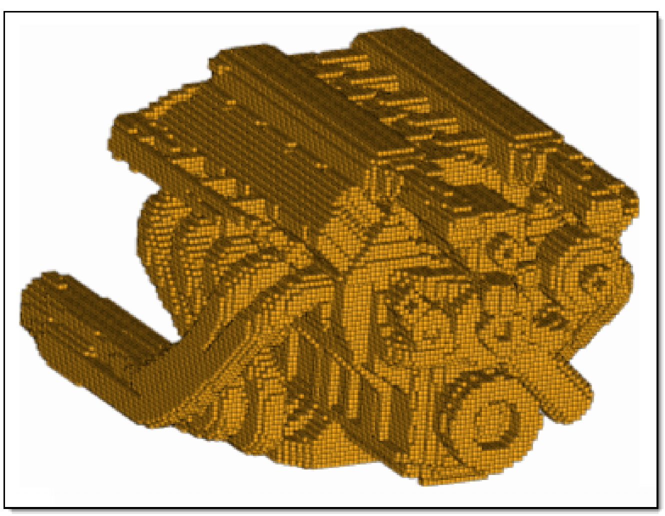


Figure 848: 2mm Solid Mesh, Jacobian=1.0

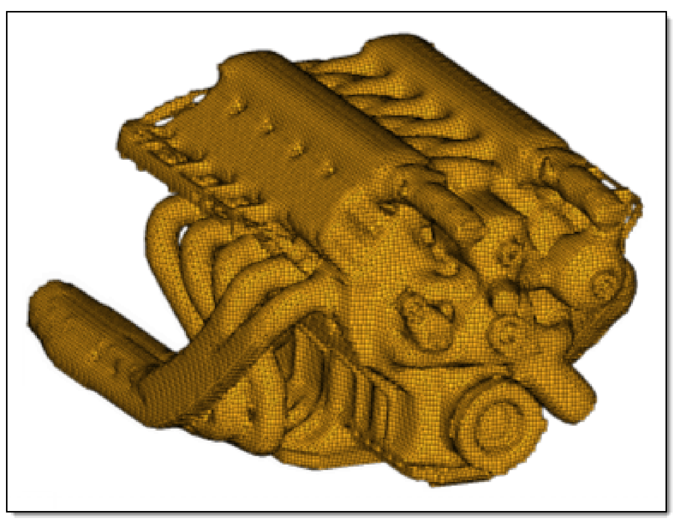


Figure 849: 2mm Solid Mesh, Jacobian=0.3

Acoustic Cavity Meshing

Acoustic Cavity meshing generates a fluid volume mesh used to calculate the acoustic modes (or standing waves) inside the air spaces of a vehicle or similarly enclosed structural model.

Acoustic cavity meshing is primarily used by Noise, Vibration, and Handling (NVH) engineers to design quieter interiors.

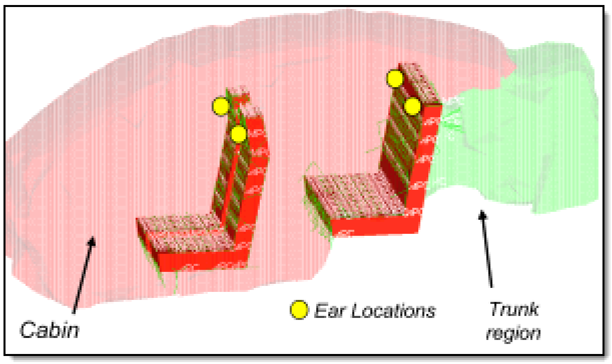


Figure 850:

Acoustic cavity meshing can be a CPU-intensive process, especially with fine and/or complex meshes, but this can be offset by additional CPU cores. The Acoustic Cavity Mesher is multithreaded to take advantage of multi-core environments.

How Cavities are Identified

Hole and gap patching is a critical part of defining cavities enclosed in the structure model. The process refers to patching over inconsequential gaps and holes that inevitably exist in the structure model, such as speaker holes.

Gaps are defined as elongated openings based on their longest dimensions. Holes are openings defined by the radius of a sphere that can pass through them. By specifying the size of the hole and gap patches, you can control how the cavities are defined through auto search.

Typically, a cavity model is intended to be meshed right up to the outer body panel. Plastic and fiber trim panels are often included in a trimmed body model, but not meant to be used for cavity meshing. However, if the trim panels are selected during the AC meshing process they can be confused as outer body panels, leading to incorrect cavity definition. Therefore, it is important to ensure that only the exterior body panels are selected as the structure components to be included in the auto cavity search.

Types of cavities you may wish to model:

Door

Include the cavity between the inner and outer door panels as a part of the interior by specifying a hole patch size smaller than that of the largest opening in the inner panel, so that the opening is not patched. This allows interior mesh to flow into the door cavity.

Instrument Panel (IP)

Model the cavity behind the Instrument Panel as a part of the interior cavity by excluding the IP parts from the structure component selection during the auto cavity search. This forces the auto cavity search to ignore the existence of the sealed IP cluster when determining the interior volume of the passenger compartment. If the IP panel needs to be treated as a radiating source, a fluid boundary needs to be created at the location of the IP, similar to the how a package tray can be included as a structure part.

Pillar

Include large pillar cavities, such as a D pillar cavity, as a part of the interior cavity by ensuring that the gap and hole patch size specified are smaller than that of the largest opening. This prevents the opening from being patched and allows interior mesh to flow into the pillar cavity.

Under Seat

Ensure the under-seat spaces are meshed by specifying an element size smaller than the smallest dimension of the space, thus allowing the interior mesh to fill the cavity.

Trim Component

Special functionalities are required to mesh these cavities.

Factors that Influence Cavity Meshing

The ability to model the acoustic cavity and predict acoustic response inside it is a critical part of NVH analysis, as noise level and quality become key product quality differentiators in the marketplace. A number of factors need to be considered when meshing an acoustic cavity model.

Adequate mesh size.

A rule of thumb is that at least six elements are needed per acoustic wavelength. Based on this rule, the minimum acoustic element sizes at various frequencies are:

500 Hz

114 mm

1000 Hz

57 mm

Smaller elements mean a more complex cavity model, which takes longer to run and generates a larger output file, particularly when fluid grid participation output is requested. Recall, however, that the mesher is multithreaded to take advantage of multiple CPU cores.

Mesh size can also affect whether smaller cavity areas, such as the cavities underneath seats, get filled. Mesh size should be selected by considering the size of the smallest cavity that needs to be filled.

Mesh quality as defined by Jacobian value, Tetra Collapse, and similar measures.

Poor mesh quality may cause problems when submitted to the solver, or lead to less-accurate results.

How closely the cavity shape matches the actual structure.

This impacts how accurately the cavity model captures acoustic modes, and how difficult it is to obtain good coupling between the fluid and structure. It is important to define the structure panels intended to be coupled to the cavity.

Aesthetics of the cavity mesh.

The model may appear too jagged if the cavity mesh matches the structure closely. Some users prefer a smooth looking mesh for presentation purposes, but care must be taken so that this does not adversely affect the modes calculated or the quality of coupling generated when default ACMODL search parameters are used.

Interior response definition.

Interior response points need to be defined so that they become a part of the mesh definition when cavity mesh is generated.

Seat Foam Cavity definition and coupling.

Seat foams are typically modeled as denser air cavities. Their geometry definition can come in either as CAD data or existing FE mesh. Some seat models contain detailed foam curvature definition, while others may just be blocky boxes. The acoustic mesher can generate a new seat foam mesh and use congruent grids to connect to the interior cavity elements, or generate fluid MPCs to connect grids on a existing foam cavity mesh to the interior cavity mesh.

Trunk Cavity separated from the Interior Cavity.

For passenger sedans, the trunk cavity is typically modeled as a separate cavity from the interior, separated by the rear seat back foam cavity.

Package Tray properly coupled to both interior and trunk cavities.

For passenger sedans, the "package tray" or "parcel shelf" is situated behind the rear seat backs, between the interior cavity on the top and the trunk cavity below. Its vibration should be coupled into both cavities. This means a boundary (or gap) needs to exist in the cavity model where the package tray is located. This is typically accomplished by the two cavities not sharing grids at the boundary.

Once generated, fluid cavities must be coupled to the structure. OptiStruct creates this coupling automatically during solver analysis, storing the information in the ACMODL card. In addition, Radioss generates an `.interface` file which can be loaded into HyperMesh to verify fluid surface and structure wetted surfaces.

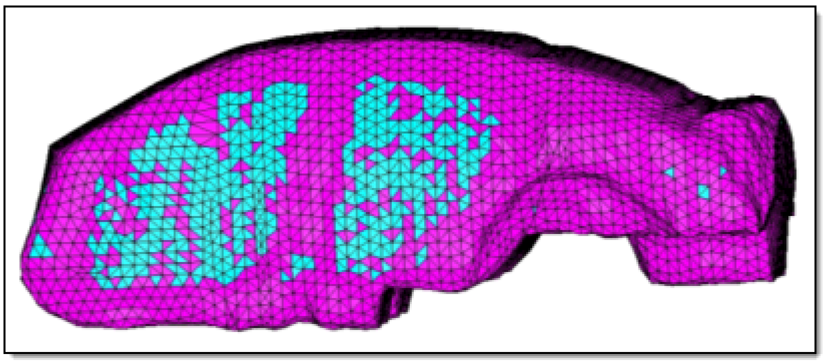


Figure 851:

Create an Acoustic Cavity Mesh

Creating an acoustic cavity mesh is two-staged. First, create a voxel-based preview mesh. Second, select individual volumes, set element quality requirements, and create a smoother, more refined computational mesh for the selected volumes.

Acoustic Cavity meshing begins with the **Acoustic Cavity Mesh** panel. This panel accepts the necessary base input data to generate a voxelated preview mesh for one or more acoustic cavities. Once the preview mesh exists, however, the **Acoustic Cavity** browser displays in the tab area; you can use this tool to modify element quality checks, select specific volumes that have a preview mesh, and create the computational mesh for each selected cavity.

1. Open the **AcousticCavity** browser and **Acoustic Cavity Mesh** panel by selecting **Mesh** > **Create** > **Acoustic Cavity Mesh** from the menu bar.
2. Define acoustic cavity meshing parameters.
   a) In the **Acoustic Cavity** browser, click .
   b) In the **Options** dialog, define acoustic cavity meshing parameters accordingly.
3. Using the **Acoustic Cavity Mesh** panel, generate preview mesh.

a) Use the **structure: comps** selector to select the components that surround the desired cavity.

   Select all of the components that enclose the air spaces. You can create meshes for multiple air spaces simultaneously, depending on which components you initially select.

b) Use the **seats: comps** selector to select the solid bodies representing the seats.

   Acoustic cavity meshing takes seating into account, and generates a separate volume mesh for each seat (bench seats are correctly treated as a single assembly, not broken up into individual seats based on the number of humans who could sit on them). This distinguishes the seats from the mesh that represents the air space.

c) Select a method for **seat coupling**.

   - Choose **node to node remesh** to obtain new node-connected cavities for the seats, matching seat nodes, and cavity nodes on a 1:1 basis.
   - Choose **MPC no remesh** to connect the input seat components to the found cavities using MPCs, which allows node mapping at a ratio other than 1:1. Only use this option if the seat components are composed of solid elements. For proper MPC creation, the seat mesh size must be relatively similar to the size used to create the acoustic cavity mesh; in general, the size difference should be no more than 30%.

d) In the **max element size** field, enter a desired element size.

   The **max frequency** will be populated automatically depending on the specified **max element size** and **No. elements per wavelength** specified in **Options** dialog. It is also possible to specify the **max frequency** value, and then **max element size** will be calculated accordingly.

   > **Note:** A small, fine mesh may take a considerable amount of time to generate.

e) In the **gap patch size** field, enter how large of a gap in the geometry the mesher will ignore.

f) In the **hole patch size** field, enter how large of a hole in the geometry the mesher will ignore.

g) Optional: Create permanent elements from the temporary elements used to patch over the holes, and store them in their own collector called ^patched_holes by selecting the **create hole elements** checkbox.

h) Click **preview**.

A simple preview mesh is generated.

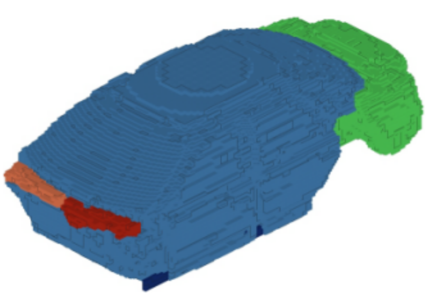


Figure 852: Simple Preview Mesh

4. In the **AcousticCavity** browser, select individual volumes, set element quality requirements, and create a smoother, more refined computational mesh for the selected volumes.

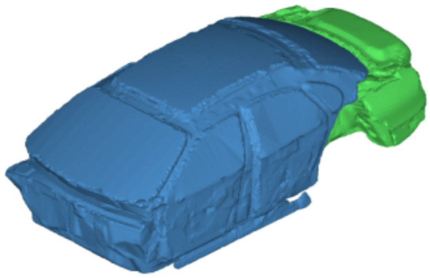


Figure 853: Final Acoustic Cavity Mesh

Voxel Meshing

Voxel meshing fills an enclosed volume with voxels (hexas) of a predefined size.

This type of mesh is only useful in topology optimization. It does not give meaningful results in a stress analysis.

To work properly, the volume must be enclosed completely by shell elements (quads and trias) without T-Connections or free edges. The normals of these elements should point inwards. The voxels (hexa elements) are stored in the component, hexas.

Create Voxel Mesh

1. From the menu bar, click **Mesh** > **Create** > **Voxel Mesh**.
2. Select the components that contain shell elements enclosing one volume.

   > **Note:** If more than one volume is selected, normals should be adjusted manually.

   a) In the panel area, use the comps selector to select components.
   b) Click **proceed**.

   The **Voxelmesh** dialog opens.
3. Define options.
   - To check for T-connections and free edges, select the **Perform element check** checkbox.

     If found, the results are stored in collectors of the corresponding names.
   - To automatically adjust normals (to inward), select the **Adjust normals** checkbox.

     This works if the selection is one connected volume only. The volume may contain internal voids.
   - To fill areas that are hidden in each coordinate direction, even if they are not touching the enclosed volume, select the **Fill undercuts** checkbox.

     These elements are stored in the component, hexasfill.

- To create are store the voxel meshes in nine components (hexas0, hexas1...) depending on the number of nodes that are inside the volume, select the **One component for each number of inner nodes** checkbox.

> **Note:** Zero inner nodes may occur if one edge of the volume intersects the center of a hexa-face.

- To select a coordinate system along which to align the mesh, select the **Use local coordinates** checkbox.

  If no selection is made, the global (basic, screen) coordinate system is used.

4. Set the edge size for hexa elements.

> **Note:** A grid of nodes is created for the box wrapping the volume, therefore the memory usage may be high for unreasonably small values.

- Choose **Cubes** to enter a single value for the edge size.
- Choose **Rectangles** to enter x, y, and z edge lengths.

5. Click **Start**.
6. If you selected the Use local coordinates checkbox, you will be prompted to select a coordinate system. Select the system and then click **proceed**.

Gasket Meshing

Gasket elements are used to model components with a very thin thickness, with their main purpose being to act as a sealing between structural components.

Gasket elements have a different node order compared to first or second order hex and penta elements. Gasket elements have nodes on their top and bottom face, and no mid-side nodes along their thickness.

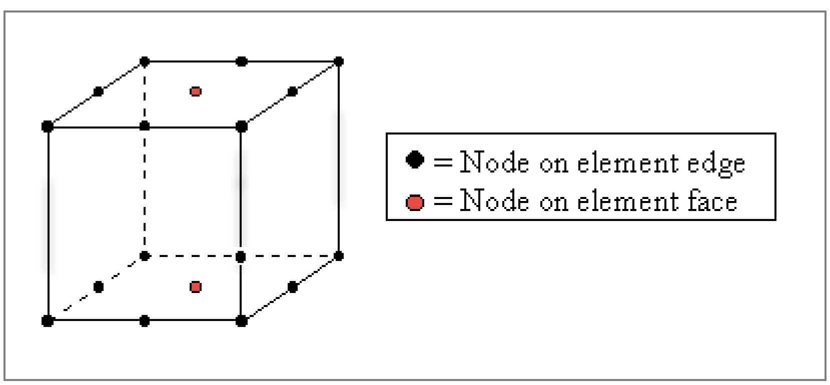


Figure 854: Node Order for Gasket Elements of Type GK3D18

Restriction:

Gasket elements are only supported in the Abaqus solver interface, and you can only create gasket elements of type GK3D18 and GK3D13 with the Gasket Elements tool.

Create Gasket Elements

1. From the menu bar, click **Mesh** > **Create** > **3D Elements** > **Gasket Elements**.
   The **Create Special Elements** dialog opens.
2. Select **Create Mesh**.
3. Use the Source selector to select the face of the 3D volume.
4. Use the Target selector to select target face of the 3D volume.
5. In the Element Size field, enter the average element size to make elements.
6. In the Number of Layers field, enter the number of elements (layers) to create between source and target layer.
7. For Destination Comp, choose where newly created elements are organized.
   - Choose **New Component** to organize elements in a new component.
   - Choose **Current Component** to organize elements in the current, active component.
   - Choose **Elems to Surf Component** to organize elements in the component to which the surface belongs.
8. For Assign Property & Material, choose a method for assigning properties to newly created elements.
   - Choose **New Property** to assign a new property.
   - Choose **Existing Property** to select an existing property.
9. Click **Apply**.
10. Click **OK** to close the dialog.

Tip: Highlight the nodes of the newly created gasket elements by clicking **Review**.

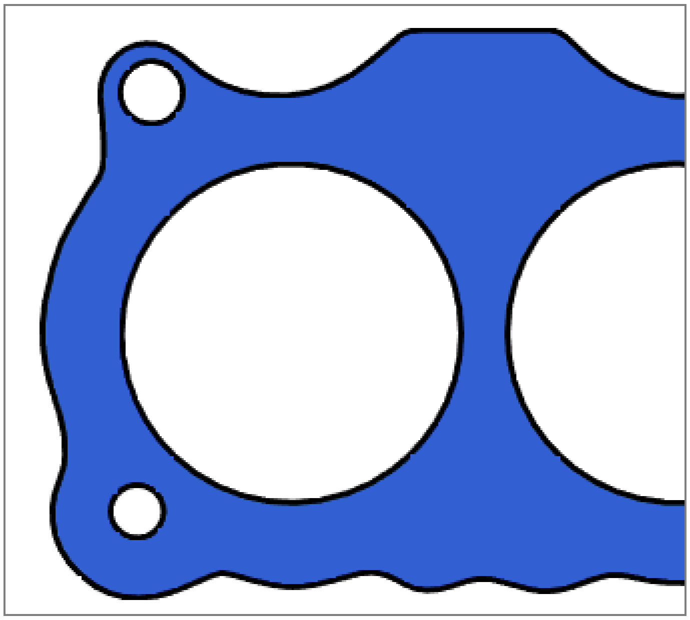


Figure 855: Original Geometry

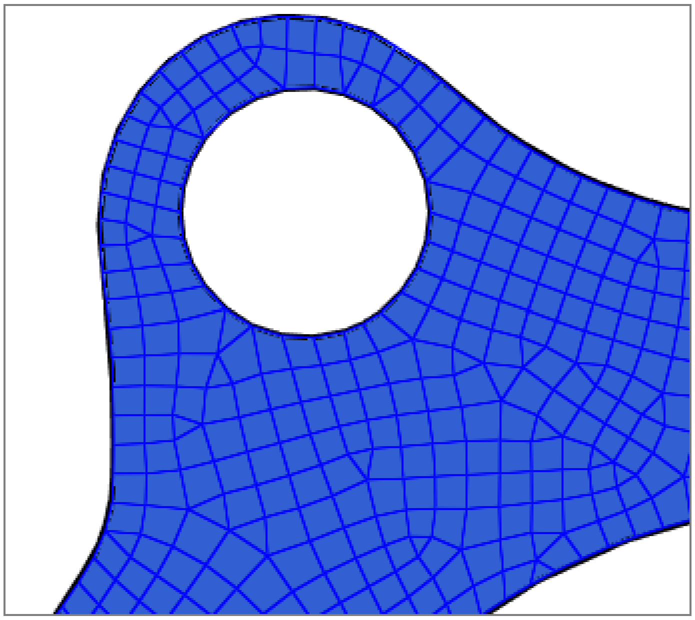


Figure 856: Mesh with Gasket Elements

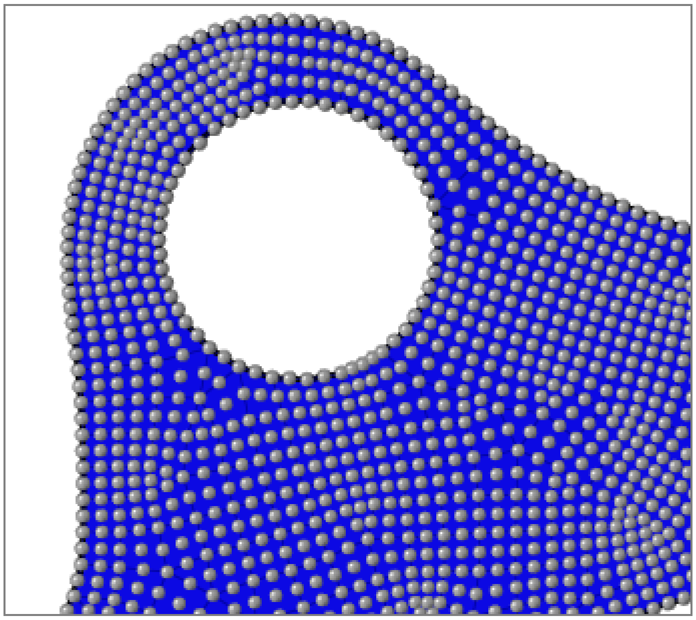


Figure 857: Gasket Elements with Review Enabled

Update Gasket Elements

1. From the menu bar, click **Mesh** > **Create** > **3D Elements** > **Gasket Elements**.

The **Create Special Elements** dialog opens.

2. Select **Update Elements**.
3. Use the Source selector to select the face of the 3D volume.
4. For Element Type, select an element type.
5. Click **Apply**.
6. Click **OK** to close the dialog.

> **Tip:**
> Highlight the nodes of the newly created gasket elements by clicking **Review**.

Mesh Controls

Mesh controls are used to automate and streamline the meshing process.

Manage mesh controls from the Mesh Controls Browser, which can be accessed by clicking **Mesh** > **Mesh Controls** from the menu bar.

You can define one model mesh control, as well as an unlimited number of local mesh controls. Meshing is supported for Batchmesher, surface meshing, adaptive wrap meshing and volume meshing.

Model mesh controls define mesh settings for all of the entities which need to be meshed in one meshing job. In general, model mesh controls reflect the mesh settings on a majority of the model. In order to run a meshing job, at least one model mesh control must be defined. Multiple model mesh controls can be defined, but only one can be active and used to run a meshing job.

Local mesh controls define mesh settings for specific areas of a model, and take precedence over model mesh controls. Local mesh controls are not mandatory, but are useful when local mesh settings need to be defined for a meshing job. An unlimited number of local mesh controls can be defined and used to run a meshing job. Local mesh controls have an on/off state which can be enabled and disabled. When a meshing job is submitted, enabled mesh controls are considered and used to automate the meshing process.

Mesh controls are HyperMesh entities that are stored in the database. Mesh controls can also be saved to an external `.xml` template file for re-use in other models.

Mesh Control Types

You can create mesh controls specific to the different meshing types and refinement zones.

Batchmesher Mesh Controls

Model and local mesh controls for BatchMesh meshing.

> **Note:** In order to run a meshing job, at least one model mesh control must be defined.

When defining model and local mesh controls, select a parameter mode to determine how criteria and parameters are generated. Batchmesher mesh controls are run using a single CPU.

When generating the mesh, local controls are applied first, starting from the smallest target element size, up to the largest. The model control is then applied.

Normal Parameter Mode

The Normal parameter mode generates criteria and parameters using an existing set of files.

Table 195: Parameters

Parameter	Description
Criteria File	Specify a criteria file that will be used to automatically generate criteria.
Parameters File	Specify a parameters file that will be used to automatically generate criteria.
Mesh Connectivity	Determines how newly created elements and any adjacent existing elements are connected. **Keep** Use existing nodes on any shared boundary edges. **Break** Ignore existing adjacent elements and generates mesh according to the element size and type you specified.

Scaled Parameter Mode

The Scaled parameter model generates criteria and parameters on-the-fly by scaling an existing set of files. This parameter mode is useful when you want to evaluate mesh size changes using minimal effort.

Table 196: Parameters

Parameter	Description
Element Size	Element size used when scaling criteria and parameters.
Criteria File	Specify a criteria file that will be used to automatically generate criteria. The "min size" and "max size" thresholds are created by simple proportional scaling of the base thresholds to the requested element size. All other criteria is not changed.
Parameters File	Specify a parameters file that will be used to automatically generate criteria. All dimensionless parameters and options are not changed. All parameters having a length dimension are proportionally scaled using the scale factor target size/original size. • Element size • Max thin solid thickness (for midsurface extraction)

Parameter	Description
	• Surface holes table: radii ranges and washers widths, target radii if given directly (not as a radius fraction) • Solid holes table: radii ranges • Surface fillet table: radii and widths ranges, maximal chordal deviation • Flanges recognition: max and min flanges width, maximal width for narrow surface width allowed to be deleted • Duplicated surfaces search tolerance – scaled if 0.5 < scale_factor < 2 • Edges equivalencing tolerance – scaled if 0.5 < scale_factor < 2 • Narrow fillets merging width threshold • Beads to be removed maximal height • Flanged holes to be removed maximal height • Edge fillets defilleting - max radius • Logos to be removed : maximal height and depth • Threads to be removed max depth • Trias reduction maximal and minimal elements size • Elements warpage fix – maximal offset normally to surfaces
Mesh Connectivity	Determines how newly created elements and any adjacent existing elements are connected. **Keep** Use existing nodes on any shared boundary edges. **Break** Ignore existing adjacent elements and generates mesh according to the element size and type you specified.

Auto Generate Parameter Mode

The Auto Generate parameter mode generates criteria and parameters on-the-fly using very basic and limited input parameters. This parameter mode is useful for basic and simple use cases, where existing parameter and criteria files do not exist and you do not have strict mesh requirements.

Mesh control input parameters are automatically populated within specific Criteria file fields. The Min Size - Good/Warn/Worst and Max Size - Good/Warn/Worst are internally calculated. All other Criteria file settings are hard-coded.

Table 197: Criteria

Mesh Control Input Parameter	Criteria File Field
Element Size	Target element size

Mesh Control Input Parameter	Criteria File Field
	Min Size - Ideal Max Size - Ideal
Minimum Element Size	Min Size - Fail
Maximum Element Size	Max Size - Fail

Mesh control input parameters are automatically populated within specific Parameter file fields. All other parameters and behaviors are kept as default.

Table 198: Parameters

Mesh Control Field	Parameter File Field
Element Size	Target element size
Element Type	Create mesh with element type, with “align” and “size” checkboxes enabled.
Element Order	Element order
Element Feature Angle	Feature angle during element cleanup
Geometry Cleanup	Enables or disables, in its entirety, the Geometry cleanup section

When Geometry Cleanup is enabled, two threshold radii are defined as:

- min_rad = max(0.4*Element Size, 0.8*Minimum Element Size)
- mid_rad = max(0.6*Element Size, min_rad)
    - max_rad = 3.0*Element Size

Enabling the Geometry Cleanup parameters listed below, facilitates the following behaviors.

Remove Small Holes
Removes small surface and solid holes with a radii range of 0 – min_rad.

Create Washers
Create one layer of washers with a minimum of 6 elements and an auto-defined width for surface holes in the min_rad – max_rad range.

Seed Holes
Enable hole seeding for surface and solid holes, per the rules below. If Create Washers is enabled, this is ignored for surface holes.

- min_rad – mid_rad = 4 elements minimum
- mid_rad – max_rad = 6 elements minimum

Allow Movement Across Edges

Enable major post-mesh element cleanup operations to improve the final element quality at the expense of edge capturing and performance.

Advanced Feature Capture

Enable improved feature capturing at the expense of element quality and performance.

In the Parameters file, the following parameters are automatically defined. All other parameters are kept as default.

- Extract midsurfaces is enabled with the Method set to skin offset.
- Geometry Cleanup
    - Surface fillet recognition

      The following thresholds are defined:

        - min_rad = 0.7*max(0.4*Element Size, Minimum Element Size)
        - min_wid = 0.49*Pi*min_rad
        - mid_rad = 1.307*max(0.5*Element Size, Minimum Element Size)
        - mid_wid = 0.5*Pi*mid_rad
        - max_rad = 1.5*Element Size
        - max_wid = 0.51*Pi*max_rad

      The following number of elements rows are enforced:

        - min_rad – mid_rad and min_wid – mid_wid = 1 element across
        - mid_rad – max_rad and mid_wid – max_wid = 2 elements across
    - Flange recognition is enabled with:
        - Elements across flange width = 2
        - Maximum width of flange = 3.5*Element Size
        - Minimum width of flange = max(0.5*Element Size, Minimum Element Size)
        - Delete flange narrow surfaces with width < = auto
    - Other options
        - Suppress beads with height < = min(0.2*Element Size, Minimum Element Size)
        - Suppress flanged holes with height < = min(0.15*Element Size, Minimum Element Size)
        - Remove edge fillets with radius < = 0.9*Element Size
        - Remove logo with size < = 3.0*Element Size, and height < = min(0.15*Element Size, Minimum Element Size)
    - Element Cleanup
        - Apply tria reduction with min elem size > = 0.5*Element Size and max size < 1.5*Element Size

- Move across free edges, max dist < = 0.0025 * element size

Surface Meshing Mesh Controls

Model, local, refinement, and feature mesh controls for surface meshing.

When generating the mesh, the following behaviors apply:

- Feature mesh controls are applied first.
- Model and local mesh controls are applied second, in the following order, according to their configuration. If any mesh controls have the same configuration, meshing is applied starting from the smallest size up to the largest:
  - Rigid body (surface-based only)
  - QI optimize
  - Size and bias
  - Edge deviation
  - Surface deviation (surface-based)/Adaptive (element-based)

Model and Local

Model and local mesh controls define the meshing mode, element size, element type, and additional parameters used to create a mesh of shell elements using surface geometry or existing shell elements.

> **Note:** In order to run a meshing job, at least one model mesh control must be defined.

When defining model and local mesh controls, select a mesh mode to use during automatic meshing to determine the type of meshing to perform. For more information on the different mesh modes, refer to Mesh Modes. The mesh mode for model controls can be changed at any time. However, for local controls, the mesh mode is defined only at the time of creation.

Size and Bias

Generates mesh using elements of a uniform size that you specify.

Main Parameters

Table 199: Parameters

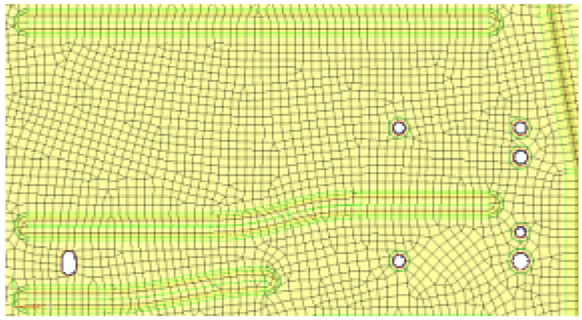
Parameter	Description
Use Model Settings	Set all of the main parameters from the model control. For local controls only.
Element Size	Average element size. The length of any active (shared or free) surface edge divided by the element size determines the number of elements to place along that edge.

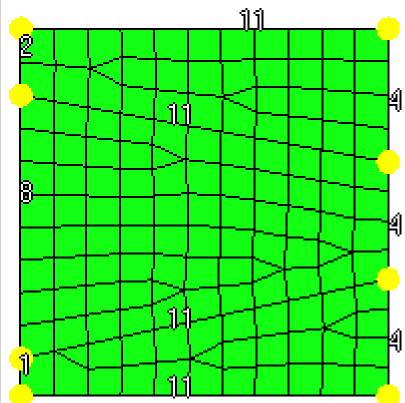
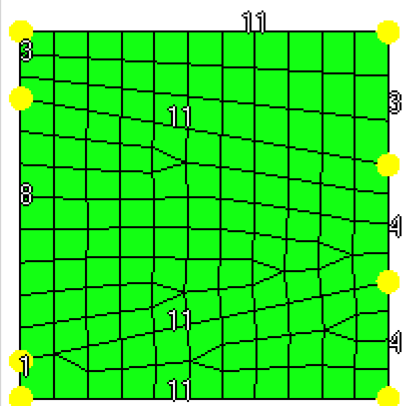
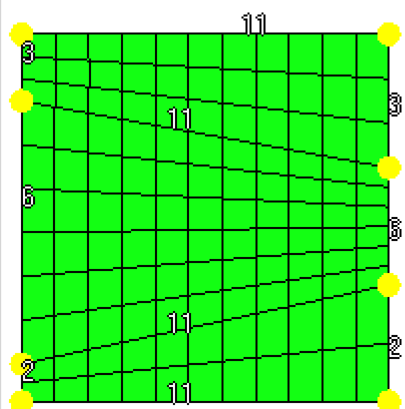
Parameter	Description
Element Type	Type of elements used to create mesh during automatic meshing. For more information, see Mesh Types.
Element Order	**First** Create elements using a linear shape function. **Second** Create elements using a polynomial shape function. For local controls, this is inherited from the model control and cannot be set independently.

Advanced Parameters

Table 200: Parameters

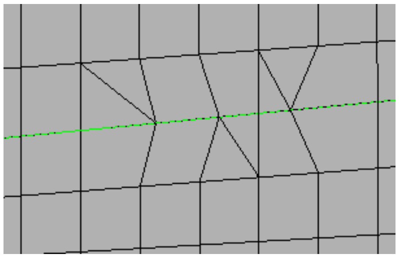
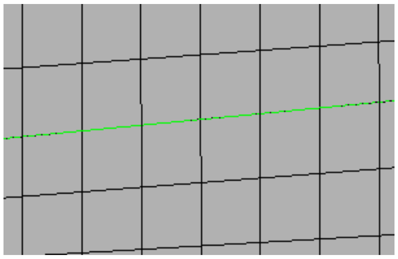
Parameter	Description
Organization	
Destination Component	**Original** Store newly created elements in the component to which the surface belongs. **Current** Store newly created elements in the "current" active component. Valid only for model controls.
Mesh Connectivity	Determines how newly created elements and any adjacent existing elements are connected. **Keep** Use existing nodes on any shared boundary edges. **Redo** Re-seed existing nodes along the boundary of the newly created mesh to optimize mesh quality. **Break** Ignore existing adjacent elements and generates mesh according to the element size and type you specified. Valid only for model controls.
Flow	
Align	Produces a more orthogonal quad-dominant mesh. Only available when the Element Type is set to Mixed.

Parameter	Description
	 *Figure 858: Flow Alignment Off*  *Figure 859: Flow Alignment On* *Notice a straighter row of elements is produced.*
Size	Enforces the global mesh element size with minimal min/max element size variations. Only available when Element Type is set to Mixed and Flow: Align is enabled.
Mapping	
Size	Keeps elements roughly the same size.
Skew	Prevents mesh from producing highly-skewed elements.
Other - Surface Based Automeshing Only	
Link Opposite Edges	Links mesh settings on opposing edges of rectangular surfaces. Changes applied to one linked edge will be applied to the other. By default, the maximum aspect ratio (AR) to allow between large and small edge sizes of linked edges is 2.11. Manually specify an aspect ratio by setting Aspect Ratio Less Than to **User Specified**. Increasing the value adds more surfaces to the linked chain, whereas decreasing the value removes some surfaces from the linked chain.

Parameter	Description
	 *Figure 860: Link Opposite Edges Selected*  *Figure 861: Link Opposite Edges Selected and Aspect Ratio Less Than = Auto*  *Figure 862: Link Opposite Edges Selected, Aspect Ratio Less Than = 8.0*

Features - Element Based Automeshing Only Parameters

Table 201: Parameters

Parameter	Description
Auto Features	Defines logical faces when using existing finite elements as a basis for automeshing. **Connected** Detect features based on the specified feature angle, and make additional effort to void any "orphan" or non-closed feature lines. It works similarly to Auto Detect, but includes a more rigorous check to combine small areas and avoid creating features that end abruptly or do not connect to any other features. **Auto Detect** Detect features based on the specified feature angle. **Surface Edges** Automatically detect and utilize geometric lines associated with selected elements as features in the re-meshing operation.  *Figure 863: Before Re-Meshing*  *Figure 864: After Re-Meshing, Surface Edges Selected* **None** Ignore extra features defined by any of the above methods.
User Features	A selection of user-created 1D elements which define features that need to be captured along with any feature angle-based features. Only 1D elements should be selected here, as any other element selections are ignored.

Parameter	Description
Anchor Nodes	Nodes that will remain and be re-used in the new mesh. Anchor nodes are "fixed" so that the automesher cannot move or replace them; in essence, they are exceptions to the re-meshing operation, and the new mesh must utilize them.

QI Optimize

An iterative automatic mesh generation method driven by element quality criteria.

Main Parameters

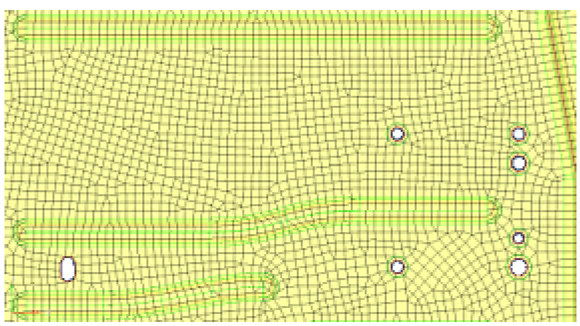
Table 202: Parameters

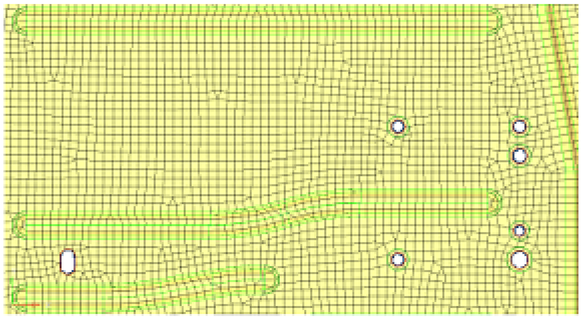
Parameter	Description
Use Model Settings	Set all of the main parameters from the model control. For local controls only.
Element Size	Average element size. The length of any active (shared or free) surface edge divided by the element size determines the number of elements to place along that edge.
Element Type	Type of elements used to create mesh during automatic meshing. For more information, see Mesh Types.
Element Order	**First** Create elements using a linear shape function. **Second** Create elements using a polynomial shape function. For local controls, this is inherited from the model control and cannot be set independently.
Criteria File	Specify a criteria file that will be used to automatically generate criteria.

Advanced Parameters

Table 203: Parameters

Parameter	Description
Organization	

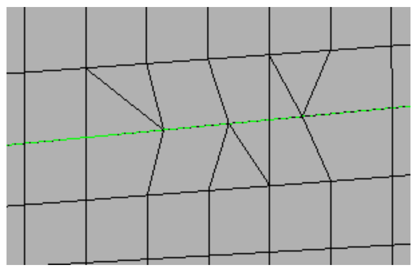
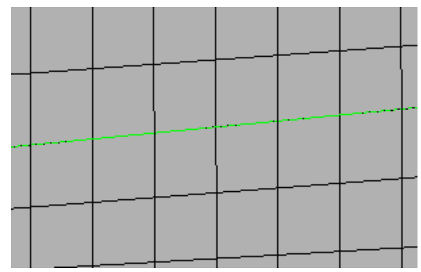
Parameter	Description
Destination Component	**Original** Store newly created elements in the component to which the surface belongs. **Current** Store newly created elements in the "current" active component. Valid only for model controls.
Mesh Connectivity	Determines how newly created elements and any adjacent existing elements are connected. **Keep** Use existing nodes on any shared boundary edges. **Redo** Re-seed existing nodes along the boundary of the newly created mesh to optimize mesh quality. **Break** Ignore existing adjacent elements and generates mesh according to the element size and type you specified. Valid only for model controls.
Flow	
Align	Produces a more orthogonal quad-dominant mesh. Only available when the Element Type is set to Mixed.  *Figure 865: Flow Alignment Off*

Parameter	Description
	 *Figure 866: Flow Alignment On* *Notice a straighter row of elements is produced.*
Size	Enforces the global mesh element size with minimal min/max element size variations. Only available when Element Type is set to Mixed and Flow: Align is enabled.
Mapping	
Size	Keeps elements roughly the same size.
Skew	Prevents mesh from producing highly-skewed elements.
Other - Surface Based Automeshing Only	
Smooth Across Common Edges	Node smoothing moves nodes across adjacent surface edges whose feature angle is less than the value specified. When selected, strict adherence to the geometry of the surface edges in not enforced for non-feature edges; some deviation from the geometry can occur.

Features - Element Based Automeshing Only

Table 204: Parameters

Parameter	Description
Auto Features	Defines logical faces when using existing finite elements as a basis for automeshing. **Connected** Detect features based on the specified feature angle, and make additional effort to void any "orphan" or non-closed feature lines.

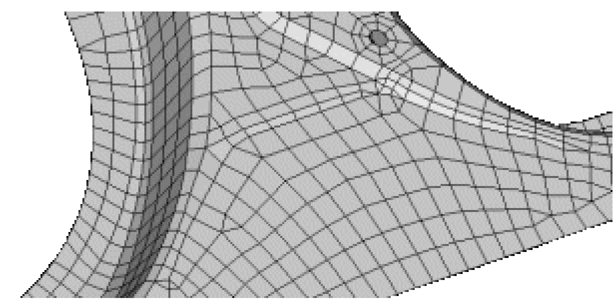
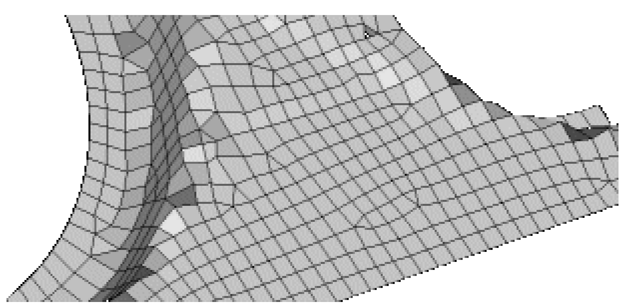
Parameter	Description
	It works similarly to Auto Detect, but includes a more rigorous check to combine small areas and avoid creating features that end abruptly or do not connect to any other features. **Auto Detect** Detect features based on the specified feature angle. **Surface Edges** Automatically detect and utilize geometric lines associated with selected elements as features in the re-meshing operation.  *Figure 867: Before Re-Meshing*  *Figure 868: After Re-Meshing, Surface Edges Selected* **None** Ignore extra features defined by any of the above methods.
User Features	A selection of user-created 1D elements which define features that need to be captured along with any feature angle-based features. Only 1D elements should be selected here, as any other element selections are ignored.
Anchor Nodes	Nodes that will remain and be re-used in the new mesh. Anchor nodes are "fixed" so that the automesher cannot move or replace them; in essence, they are exceptions to the re-meshing operation, and the new mesh must utilize them.

Edge Deviation

Determines how far the mesh elements can deviate from the actual edges of the surfaces meshed, or when in the case of re-meshing elements, deviation from inferred edges based on features.

Main Parameters

Table 205: Parameters

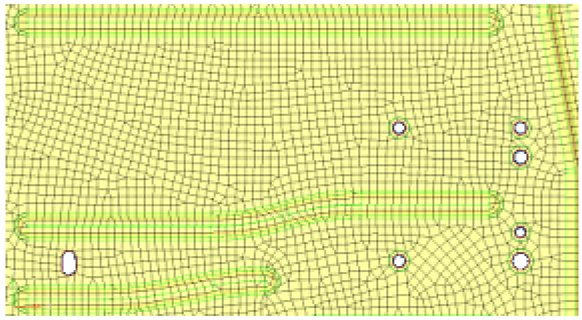
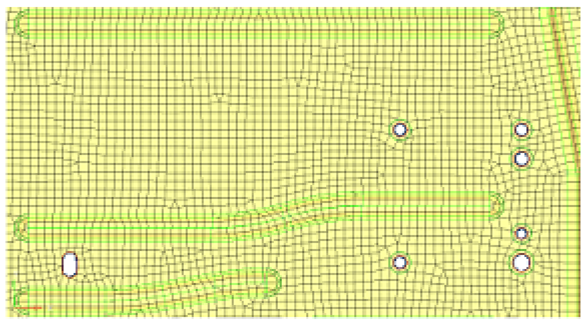
Parameter	Description
Use Model Settings	Set all of the main parameters from the model control. For local controls only.
Minimum Size	Minimum allowable element size.
Maximum Size	Maximum allowable element size.
Maximum Deviation	Allowable deviation between the element edge and the surface edge. To meet this requirement, element edge lengths along a curved surface edge are reduced as needed down to a lower limit set by the minimum element size.
Maximum Feature Angle	Maximum angle across which elements can be maintained. When two adjacent elements' normals exceed this angle, a new set of nodes is created between them to maintain clean feature lines. Using higher value results in elements spanning the feature line.  *Figure 869:* *With an appropriate value, the features lines are preserved.*  *Figure 870:* *If the feature angle is too high, the feature lines are blurred.*
Element Type	Type of elements used to create mesh during automatic meshing.

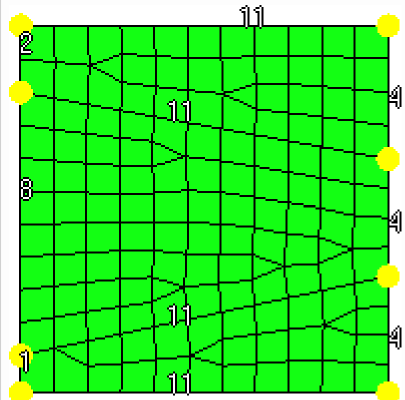
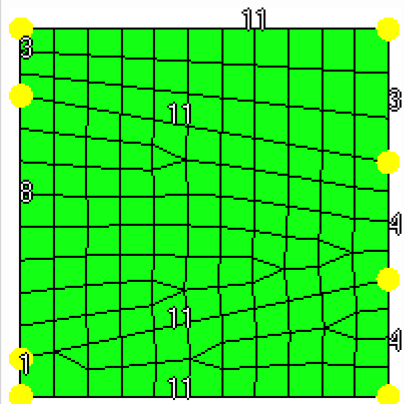
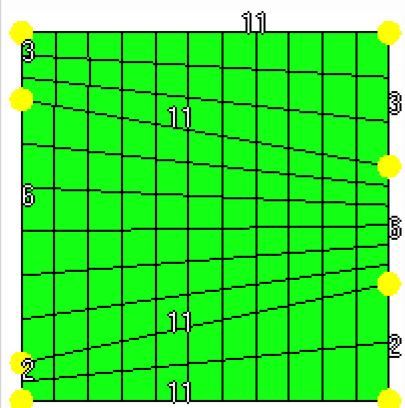
Parameter	Description
	For more information, see Mesh Types.
Element Order	**First** Create elements using a linear shape function. **Second** Create elements using a polynomial shape function. For local controls, this is inherited from the model control and cannot be set independently.

Advanced Parameters

Table 206: Parameters

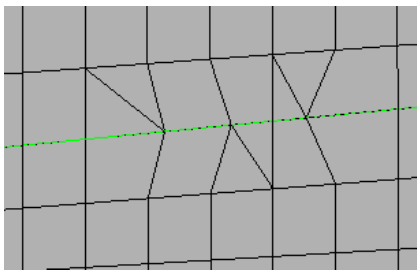
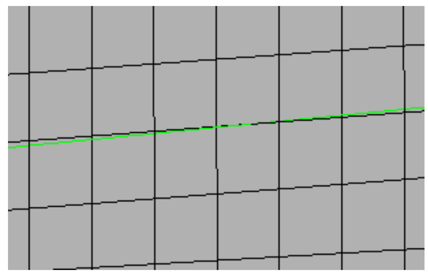
Parameter	Description
Organization	
Destination Component	**Original** Store newly created elements in the component to which the surface belongs. **Current** Store newly created elements in the "current" active component. Valid only for model controls.
Mesh Connectivity	Determines how newly created elements and any adjacent existing elements are connected. **Keep** Use existing nodes on any shared boundary edges. **Redo** Re-seed existing nodes along the boundary of the newly created mesh to optimize mesh quality. **Break** Ignore existing adjacent elements and generates mesh according to the element size and type you specified. Valid only for model controls.
Flow	
Align	Produces a more orthogonal quad-dominant mesh. Only available when the Element Type is set to Mixed.

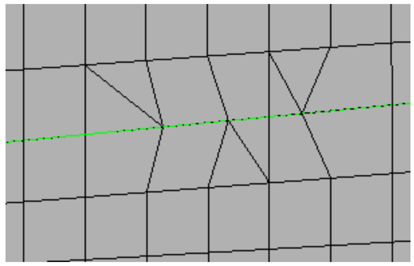
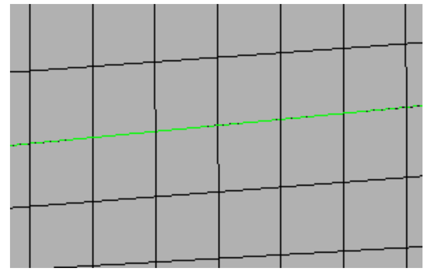
Parameter	Description
	 *Figure 871: Flow Alignment Off*  *Figure 872: Flow Alignment On* *Notice a straighter row of elements is produced.*
Mapping	
Size	Keeps elements roughly the same size.
Skew	Prevents mesh from producing highly-skewed elements.
Other - Surface Based Automeshing Only	
Link Opposite Edges	Links mesh settings on opposing edges of rectangular surfaces. Changes applied to one linked edge will be applied to the other. By default, the maximum aspect ratio (AR) to allow between large and small edge sizes of linked edges is 2.11. Manually specify an aspect ratio by setting Aspect Ratio Less Than to **User Specified**. Increasing the value adds more surfaces to the linked chain, whereas decreasing the value removes some surfaces from the linked chain.

Parameter	Description
	 *Figure 873: Link Opposite Edges Selected*  *Figure 874: Link Opposite Edges Selected and Aspect Ratio Less Than = Auto*  *Figure 875: Link Opposite Edges Selected, Aspect Ratio Less Than = 8.0*

Features - Element Based Automeshing Only

Table 207: Parameters

Parameter	Description
Auto Features	Defines logical faces when using existing finite elements as a basis for automeshing. **Connected** Detect features based on the specified feature angle, and make additional effort to void any "orphan" or non-closed feature lines. It works similarly to Auto Detect, but includes a more rigorous check to combine small areas and avoid creating features that end abruptly or do not connect to any other features.  *Figure 876: Refore Re-Meshing*  *Figure 877: After Re-Meshing, with Connected Features* **Auto Detect** Detect features based on the specified feature angle. **Surface Edges** Automatically detect and utilize geometric lines associated with selected elements as features in the re-meshing operation.

Parameter	Description
	 *Figure 878: Before Re-Meshing*  *Figure 879: After Re-Meshing, with Surface Edges Selected* **None** Ignore extra features defined by any of the above methods.
User Features	A selection of user-created 1D elements which define features that need to be captured along with any feature angle-based features. Only 1D elements should be selected here, as any other element selections are ignored.
Anchor Nodes	Nodes that will remain and be re-used in the new mesh. Anchor nodes are "fixed" so that the automesher cannot move or replace them; in essence, they are exceptions to the re-meshing operation, and the new mesh must utilize them.

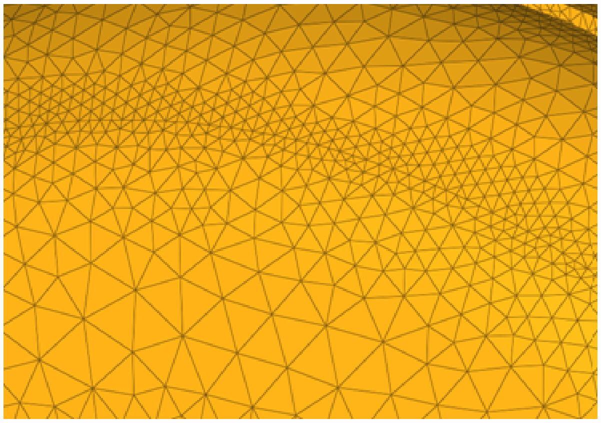
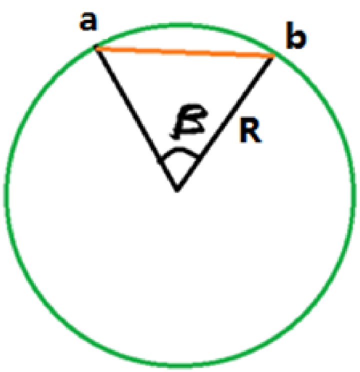
Surface Deviation

Generates mesh within the limits of element deviation from input elements.

Main Parameters

Table 208: Parameters

Parameter	Description
Use Model Settings	Set all of the main parameters from the model control. For local controls only.
Minimum Size	Minimum allowable element size.

Parameter	Description
	When there are model and local mesh controls for surface deviation, and the minimum size of neighboring mesh controls are different, the minimum size will automatically propagate from the smaller mesh size to neighboring surfaces.
Maximum Size	Maximum allowable element size.
Growth Rate	Determines how rapidly elements can increase in size as they are created further and further away from features.  *Figure 880: Growth Rate* *Elements further from the features grow larger with each row.*
Span Angle	Controls the element size at curve input. The smaller the angle, the more refined curvature will be and the more preserved the input shape will be. By default span angle is 25.0 degrees. Valid only for element-based controls  *Figure 881: Span Angle* *β is the span angle of the edge ab. The length of ab is less than 2R(sin β/2).*

Parameter	Description
Feature Angle	Determines which features to preserve. The mesher identifies features internally and preserves/refine them based on the defined feature angle. Valid only for element-based controls.
Element Order	**First** Create elements using a linear shape function. **Second** Create elements using a polynomial shape function. For local controls, this is inherited from the model control and cannot be set independently.

Advanced Parameters

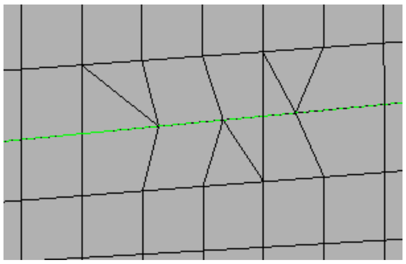
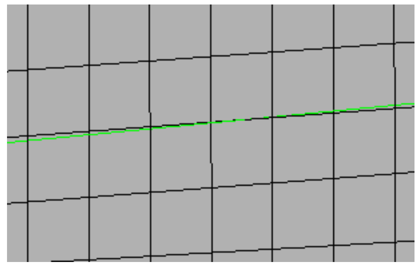
Table 209: Parameters

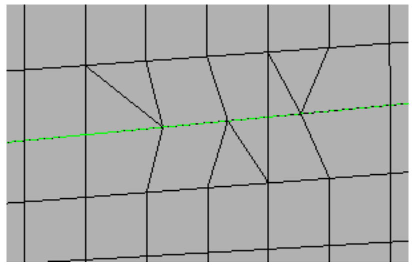
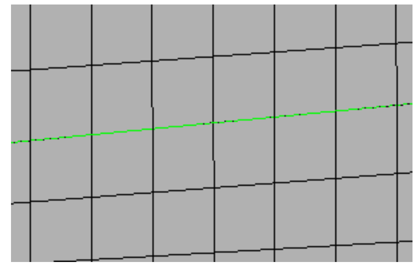
Parameter	Description
Organization	
Mesh Connectivity	Determines how newly created elements and any adjacent existing elements are connected. **Keep** Use existing nodes on any shared boundary edges. **Redo** Re-seed existing nodes along the boundary of the newly created mesh to optimize mesh quality. **Break** Ignore existing adjacent elements and generates mesh according to the element size and type you specified. Valid only for model controls.

Features - Element Based Automeshing Only

Table 210: Parameters

Parameter	Description
Auto Features	Defines logical faces when using existing finite elements as a basis for automeshing.

Parameter	Description
	Connected Detect features based on the specified feature angle, and make additional effort to void any "orphan" or non-closed feature lines. It works similarly to Auto Detect, but includes a more rigorous check to combine small areas and avoid creating features that end abruptly or do not connect to any other features.  *Figure 882: Refore Re-Meshing*  *Figure 883: After Re-Meshing, with Connected Features* **Auto Detect** Detect features based on the specified feature angle. **Surface Edges** Automatically detect and utilize geometric lines associated with selected elements as features in the re-meshing operation.

Parameter	Description
	 *Figure 884: Before Re-Meshing*  *Figure 885: After Re-Meshing, with Surface Edges Selected* **None** Ignore extra features defined by any of the above methods.
User Features	A selection of user-created 1D elements which define features that need to be captured along with any feature angle-based features. Only 1D elements should be selected here, as any other element selections are ignored.
Refine Features	Apart from capturing the features if refine feature is turned enabled, you can define the mesh size to be applied to features selected/found by any of the above methods.

Rigid Body Mesh

Generates mesh to represent the topology of a rigid object. Available only for surface-based meshing.

Main Parameters

Table 211: Parameters

Parameter	Description
Use Model Settings	Set all of the main parameters from the model control. For local controls only.
Minimum Size	Minimum allowable element size.

<table>
<tr><th>Parameter</th><th>Description</th></tr>
<tr><td>Maximum Size</td><td>Maximum allowable element size.</td></tr>
<tr><td>Maximum Deviation</td><td>Allowable deviation between the element edge and the surface edge. To meet this requirement, element edge lengths along a curved surface edge are reduced as needed down to a lower limit set by the minimum element size.</td></tr>
<tr><td>Maximum Feature Angle</td><td>Maximum angle across which elements can be maintained. When two adjacent elements’ normals exceed this angle, a new set of nodes is created between them to maintain clean feature lines. Using higher value results in elements spanning the feature line.

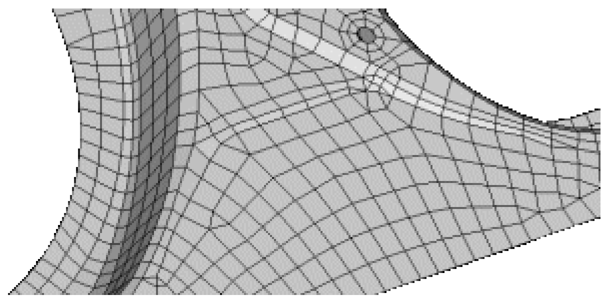



Figure 886:
With an appropriate value, the features lines are preserved.

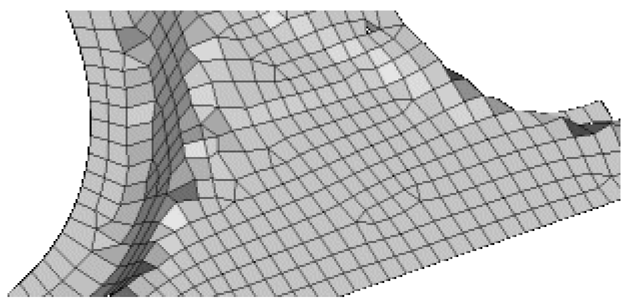



Figure 887:
If the feature angle is too high, the feature lines are blurred.</td></tr>
<tr><td>Element Type</td><td>Type of elements used to create mesh during automatic meshing.
For more information, see Mesh Types.</td></tr>
<tr><td>Element Order</td><td>First
Create elements using a linear shape function.
Second
Create elements using a polynomial shape function.</td></tr>
</table>

Parameter	Description
	For local controls, this is inherited from the model control and cannot be set independently.

Advanced Parameters

Table 212: Parameters

Parameter	Description
Organization	
Destination Component	**Original** Store newly created elements in the component to which the surface belongs. **Current** Store newly created elements in the "current" active component. Valid only for model controls.

Refinement

Refinement controls allow the definition of element refinement based on geometry, proximity and angle.

This is supported only for surface deviation meshing.

Geometric

Geometric refinement mesh controls define the refinement size of selected entities on the following types of geometry: points, lines and surfaces.

> **Note:** Geometric refinement mesh controls only apply to surface deviation meshing (surface-based only).

Table 213: Parameters

Parameter	Description
Refinement Size	Element size applied to selected entities.

Parameter	Description
	If the refinement size is less than the minimum size defined by the model mesh control, then the minimum size will override the refinement size.

Proximity

Proximity refinement mesh controls define refinement based on the proximity of surfaces and elements.

> **Note:** Proximity refinement mesh controls only apply to Surface Deviation meshing (both surface and element based).

Table 214: Parameters

Parameter	Description
Maximum Search Angle	Defines the maximum normal angle between elements and surfaces to be considered for the proximity candidate search.
Maximum Proximity Distance	Performs refinement if the surfaces/elements proximity is below this value.
Minimum Proximity Distance	Performs refinement if the surfaces/elements proximity is above this value.
Proximity Within	Defines how the proximity search should be performed. **Local Selection Only** Check the proximity refinement candidates within the selection of this mesh control only. **Local and Model Selections** Check the proximity refinement candidates within the selection of this mesh control and the model mesh control.
Proximity Check Direction	Defines the direction in which to consider proximity. **Along Normal** Consider only surfaces/ elements which facing same normal for the proximity candidate search. **Inverse Normal** Consider only surfaces/ elements which facing opposite normal for the proximity candidate search. **Both Sides** Consider any normal orientation of surfaces/elements for the proximity candidate search.

Parameter	Description
	Volume Outward Consider outwards direction of the solids for the proximity candidate search. **Volume Inward** Consider inwards direction of the solids for the proximity candidate search.
Refinement Method	Method used to refine proximity elements. **Constant Size** Refine the close proximity candidates with the defined constant size. **Size/Proximity Ratio** Refine the close proximity candidates based on the proximity of the input. Refinement size = ratio * proximity distance. For example, if the proximity distance between two surface/element faces is 10, and a ratio of 0.1 has been defined, then a refinement size of 1 will be applied to the proximity area. Thus refinement will vary based on the variation of the proximity distance. This is very useful when you want a certain number of tetra layers in close proximity areas. **Proximity Curve** Populate a table of proximity verses refinement sizes. All the intermediate values will be interpolated.

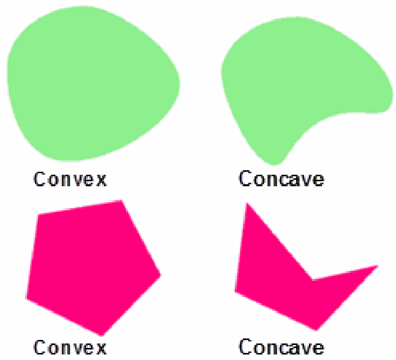
Angle-Based

Angle-based refinement mesh controls define refinement based on the angle between entities. Surfaces/elements selected to be defined by this mesh control will be refined if they fall within the threshold angle limit.

> **Note:** Angle-based refinement mesh controls only apply to surface deviation meshing (both surface and element-based).

Table 215: Parameters

Parameter	Description
Direction	Defines the concave or convex angle with any direction to be refined. Convex and concave angles are related to a polygon and measured between surfaces/element faces. If Both is selected, both types of angles will be refined.

Parameter	Description
	 *Figure 888:*
Minimum Angle Limit	Minimum angle limit. The Refinement Size specified below this parameter in the Entity Editor will be assigned this angle.
Maximum Angle Limit	Maximum angle limit. The Refinement Size specified above this parameter in the Entity Editor will be assigned this angle.

Feature

Feature mesh controls define the mesh density and quality of specific geometric features. These can be used to generate a high quality tria mesh on supported feature types.

Feature controls can be run standalone by right-clicking on the Feature folder and selecting the Mesh operation from the context menu. In this scenario:

- If a feature global control only exists, on-the-fly selection is prompted.
- If a feature body control only exists, it must have a pre-defined selection and on-the-fly selection is not allowed.
- If both a feature global and body control exist, the body control will always be used and the global control will be ignored.

Feature controls can also be utilized along with other Surface Mesh controls by right-clicking on the Surface Mesh folder and selecting the Mesh operation from the context menu. In this scenario:

- A surface mesh model control is required.
- The selection strictly comes from the surface mesh model control, either as a pre-defined selection or on-the-fly.
- Only the surface mesh model control selection that overlaps with any feature selection (including body or local controls) is considered.
- Feature global control will always be ignored in this case.

It is recommended that feature controls be run standalone. Combined usage with other surface mesh controls may result in a non-conformal mesh.

Global and Body

Global mesh controls define the mesh for all geometry in the model, whereas body mesh controls define the mesh of selected local bodies.

Mesh Size Parameters

Table 216: Parameters

Parameter	Description
Average Element Size	The 2D element size in the model which controls the average element size.
Minimum Element Size	Minimum length of elements to generate in the mesh.
Coarse Mesh	Reduces the number of solid elements and the number of nodes. On an average, coarse mesh generates about 20 percent less nodes/ elements compared to a regular volume mesher.
Use Maximum Element Size	Define the maximum element size instead of the average element size. Maximum element size = 1.4 * Average element size. Only available for Global mesh controls.

Geometry Curvature Parameters

Table 217: Parameters

Parameter	Description
Maximum Angle per Element	The approximation of a geometry is related to the angle made by an element edge on a perfect circle. For example, if the angle is 45 degrees this will approximate a circle by 8 elements. This same measurement is extended to an arbitrary curved surface.
Curvature Minimum Element Size	The mesh approximates the geometry by varying the mesh size as a function of the curvature. This geometry approximation cannot result in a mesh size smaller than the Curvature minimum element size.

Mesh Quality Parameters

Table 218: Parameters

Parameter	Description
Aspect Ratio	Determines the quality of the mesh. The value varies from 1 (equilateral element) to infinity (flat element). The default aspect ratio value is 10.
Mesh Grading	The variation of the mesh in the face from the boundary to the interior is controlled by the surface mesh grading. The default value is 1.5.

Mesh Type Parameters

Table 219: Parameters

Parameter	Description
Element Type	Use Tria3 or Tria6 elements to create mesh.
Project Mid-node	Enables the projection of mid-nodes onto the faces and edges of the geometry.
Mesh Pattern	Determines the mesh pattern. **Auto** Determine the faces that will be isomeshed automatically. All regular four edged faces and faces that are four sided will be isomeshed. **Regular 4-Sided Faces** Isomesh all regular four edged faces, even if the edges are not of similar size. **None** Force a free mesh on all faces in the body, even if it is a regular four edged face.

Cylinder

Cylinder mesh controls define the mesh both axially and in circular direction on selected cylindrical surfaces.

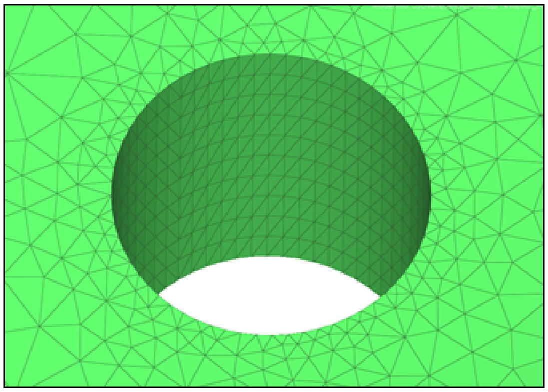


Figure 889: Cylinder

Table 220: Parameters

Parameter	Description
Axial Mesh Size	Define the element size along the length of the selected cylindrical faces.
Divisions	The number of nodes along the circular direction of the cylindrical face can be controlled by specifying the number of circular mesh seeds.

Edge

Edge mesh controls define the mesh on an edge.

Table 221: Parameters

Parameter	Description
Divisions	Edge seeds are applied by specifying the number of elements.

Face

Face mesh controls define the mesh on the selected local faces. A uniform mesh is generated on the face that has the face mesh control specified. The curvature is ignored.

Mesh Size Parameters

Table 222: Parameters

Parameter	Description
Average Element Size	The 2D element size in the model which controls the average element size.

Parameter	Description
Minimum Element Size	Minimum length of elements to generate in the mesh.

Geometry Curvature Parameters

Table 223: Parameters

Parameter	Description
Maximum Angle per Element	The approximation of a geometry is related to the angle made by an element edge on a perfect circle. For example, if the angle is 45 degrees this will approximate a circle by 8 elements. This same measurement is extended to an arbitrary curved surface.
Curvature Minimum Element Size	The mesh approximates the geometry by varying the mesh size as a function of the curvature. This geometry approximation cannot result in a mesh size smaller than the Curvature minimum element size.

Mesh Quality Paraemters

Table 224: Parameters

Parameter	Description
Aspect Ratio	Determines the quality of the mesh. The value varies from 1 (equilateral element) to infinity (flat element). The default aspect ratio value is 10.
Mesh Grading	The variation of the mesh in the face from the boundary to the interior is controlled by the surface mesh grading. The default value is 1.5.
Use Local	Uses the local settings when enabled. The settings from the global control will be used when disabled.

Fillet

Fillet mesh controls define the mesh over fillets along the length and the curve direction. You can apply fillet mesh controls to selected faces or bodies.

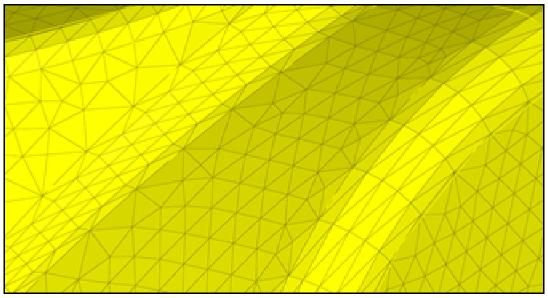


Figure 890: Fillet

Axial Mesh Size Parameters

Table 225: Parameters

Parameter	Description
Length Along Fillet	Defines the element length along the length of the fillet.

Geometry Curvature Parameters

Table 226: Parameters

Parameter	Description
Specification	Defines the element size along the fillet curvature direction. **Number** Enables the Number of Elements option to explicitly indicate the number of elements along the curvature direction. **Curvature** Enables the Maximum Angle per Element and Curvature Minimum Element Size options for defining the mesh based on these values.

Mesh Quality Parameters

Table 227: Parameters

Parameter	Description
Ignore Fillet Width Less Than	Specifies the minimum fillet width to consider. Fillets with a width below this value are ignored.
Aspect Ratio	Determines the quality of the mesh. The value varies from 1 (equilateral element) to infinity (flat element). The default aspect ratio value is 10.

Parameter	Description
Merge Small Fillets	Merges adjacent small fillets before meshing in order to improve the mesh quality.
Adjust Axial Length to Aspect Ratio	Adjusts the length along the fillet to make sure that the aspect ratio is satisfied.

Imprint Circle

Imprint circle mesh controls create a circular edge on a face with specified radius and seeds.

Mesh Seeds Parameters

Table 228: Parameters

Parameter	Description
Radius	Calculated using radius and scale factor values.
Scale	Scales the user defined radius by this factor.
Divisions	Number of seeds along the circle.

Center Parameters

Table 229: Parameters

Parameter	Description
Circle Center	Defines the center of the circular edge.

Isoline

Isoline mesh controls place mapped mesh on cylindrical/conical faces that are trimmed. The mapped mesh is created for a given size/seed in axial and circular directions.

Axial Mesh Size Parameters

Table 230: Parameters

Parameter	Description
Specification	Defines the given size/seed in axial and circular directions to create the mapped mesh.

Geometry Curvature Parameters

Table 231: Parameters

Parameter	Description
Specification	The number of nodes along the circular direction can be controlled by using maximum angle or mesh seeds. The maximum angle refers to the angle subtended by a circular spanning element edge from the axis of cylindrical face.
Merge Faces	Merges selected faces to a single face. Use this parameter to avoid the creation of a thin layer of elements when meshing.

Mesh Quality Parameters

Table 232: Parameters

Parameter	Description
Aspect Ratio	Determines the quality of the mesh. The value varies from 1 (equilateral element) to infinity (flat element). The default aspect ratio value is 10.
Curvature Minimum Element Size	The mesh approximates the geometry by varying the mesh size as a function of the curvature. This geometry approximation cannot result in a mesh size smaller than the Curvature minimum element size.

Preserved Entity

Preserved Entity mesh controls define the features to preserve while meshing. Tiny features on edges or surfaces may be collapsed if the minimum element size is larger than the feature. Entities selected here will be preserved from such collapsing.

Symmetry

Symmetry mesh controls are used to get identical mesh between the master face and symmetry face. If there are any discontinuous edges, select three nodes/vertices for both the master and symmetry face.

Table 233: Parameters

Parameter	Description
Master Surface	Defines the surface to considered as the master/source for reflecting the mesh.

Parameter	Description
Symmetry Surface	Defines the surface to considered as the slave/target for reflecting the mesh.
Master Points	Defines points on the Master surface to be mapped in case there are any discontinuous edges.
Symmetry Points	Defines points on the Symmetry surface to be mapped in case there are any discontinuous edges.

Washer

Washer mesh controls define rings around a circle. The circle has to be inside of a face.

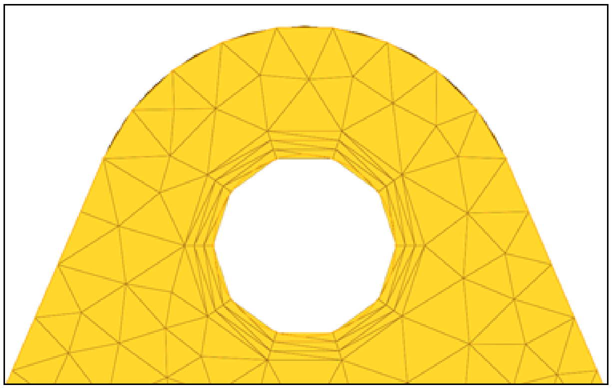


Figure 891: Washer

Table 234: Parameters

Parameter	Description
Washer Width	Define the width of the washer around the circle.

Parameter	Description
Number of Layers	Defines the number of mesh layers to create around the circle.

Adaptive Wrap Mesh Controls

Model, local, proximity and leak detection mesh controls for adaptive wrap meshing.

Optional automatic surface re-meshing of the wrapping result ensures good surface mesh quality.

Model and Local

Model and local controls define the wrap type, wrap selection, and wrap settings. Local controls defines wrap settings and will override those defined in the model control.

> **Note:** In order to run a meshing job, at least one model mesh control must be defined. For surface/region selection, all mesh controls created for adaptive wrapper should have surface/region selection only. Mesh controls with component and surface selection will not be compatible.

Wrap Selection

Table 235: Parameters

Parameter	Description
Wrap Type	Specifies whether to wrap the exterior or cavity of the selected entities.
Wrap Selection	Specifies which volumes to wrap. **All** Wrap all of the volumes in the model. **Wrap Enclosed** Wrap the volumes enclosed by the defined nodes and ignores the remaining volumes. **Wrap Nth Largest** Specify the volume to be wrapped by specifying the wrap size index in terms of volume size. To wrap the largest volume, enter 1 in the wrap size index field; to wrap the second largest volume, enter 2 in the wrap size index field.

Parameter	Description
	Exclude Enclosed Ignore the volume(s) enclosed by the defined nodes and wraps the remaining volumes.

Wrap Mesh

Table 236: Parameters

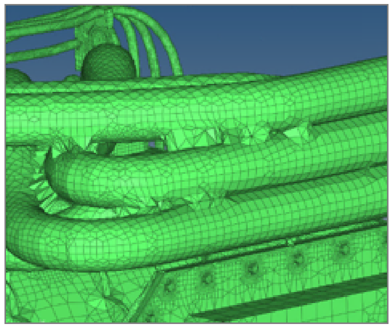
Parameter	Description
Average Element Size	The 2D element size in the model which controls the average element size.
Minimum Element Size	Minimum element size to be used to control how features are refined. Define the minimum element size accordingly to precisely preserve features.
Feature Edges	Specifies which features in the model to refine. **All** Extract features based on the defined feature angle. Open or closed loop features are considered and can be extracted. **Connected** Extract features based on the defined feature angle. Only closed loop features are considered and can be extracted. **User Defined** Manually select features to refine. If you have a large model and do not want to refine feature everything, this option is useful.
Hole Patch Tolerance	Determine which holes to patch. Closed loop free edges spanned within one or more components are considered.
Gap Patch Tolerance	Determine which gaps to patch. If two components/element groups have a gap between them and are within the gap patch tolerance, then the gap between them will be patched. Override the functionality of the gap patch at specific components/element groups by defining a proximity control for them.
Baffle Thickness	Define the thickness assigned to baffles in the model. A baffle represents open – zero thickness shell face. Baffle thickness parameters inflate the baffle with defined thickness during wrapping.

Parameter	Description
Organize as Input Components	Split wrapped elements so that they are organized based on input components. Duplicate naming is not allowed, therefore wrapped components will be named the original component name followed by _Wrap. If this option is not enabled, a component called Adaptive Wrapper will be created and will contain all wrapped elements.
Remesh after Wrapping	Remesh the entire wrapped component with the defined growth rate and span angle. The element size used for remeshing will be automatically detected. If this parameter is enabled along with the parameter organize wrap elements by base comps, then wrapped components will be remeshed and named with the original component name followed by _Wrap_Remesh. Separate assemblies will be created for wrapped and remeshed components.

Refinement

Table 237: Parameters

Parameter	Description
Element Average Size	Refine based on the average element size in the model.
Curvature Based	Specify a maximum chordal deviation, which determines which features are refined. When this parameter is disabled, features will not be refined even though features are defined in wrapper parameters.
Global Proximity Refinement	Enable proximity refinement at a global level. All input elements for proximity refinement within the defined tolerance are considered. If this parameter is disabled, proximity refinement at a global level will not take.
Intersection Line	Refine wrapped mesh at the intersecting lines of input elements.
Rough patch smoothing	Smooth out rough/zig-zag patches which are a result of the wrapper. This option will only operate on rough patches, and preserves features as much as possible.

<table>
<tr><th>Parameter</th><th>Description</th></tr>
<tr><td></td><td>
Figure 892: Rough Patch Smoothing Off

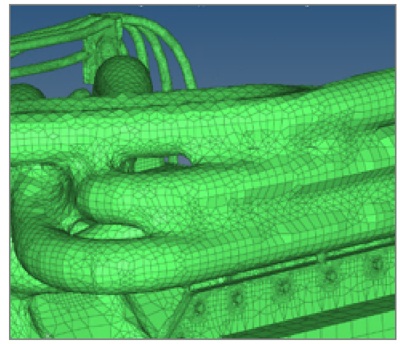



Figure 893: Rough Patch Smoothing On</td></tr>
</table>

Proximity

Proximity mesh controls define element refinement based on proximity. Use proximity mesh controls to avoid contact between selected components and any refinement gaps between them.

Global and local proximity mesh controls can be defined. Global proximity mesh controls can only be created from the Adaptive Wrap > Model > Refinement section, whereas local proximity mesh controls can only be created from the Adaptive Wrap > Proximity section.

Table 238: Parameters

Parameter	Description
Group Selection	Determines how proximity is defined. **Within Group** Consider proximity refinement for all of the components/surfaces/regions you selected as long as the gap or proximity between the components is above the specified Proximity Tolerance. **Across Group** Only consider proximity refinement for all components/surfaces/regions across Group 1 and Group 2 as long as the gap or proximity between the components is above the specified Proximity Tolerance. Proximity for the components within one group will not be considered.

Parameter	Description
	Note: For surface/region selection, all mesh controls created for adaptive wrapper should have surface/region selection only.
Proximity Tolerance	Resolve all proximity above this value.
Self Proximity	Refine the proximity of elements within one component above the Proximity Tolerance. Only available when Group Selection is set to Within Group.

Leak Detection

Leak detection mesh controls check if the resulting wrapped surface mesh creates a desired cavity before the wrapping process starts. For example, for an external aero/underhood analysis, leak detection mesh controls can help make sure a cabin, engine, and fuel tank are properly sealed, prohibiting mesh from being created there.

Once leak detection mesh controls are defined, right-click on the Adaptive Wrap folder and select **Run Leak Detection**. Depending upon the model size and parameters defined, leak detection can take several minutes.

Table 239: Parameters

Parameter	Description
Enclosed Node	Define the node within shells to be enclosed by the wrapper. This node should be enclosed by a possible volume.
Destination Nodes	Destination nodes enable leak detection paths to be found between the enclosed node and destination nodes. 1D elements are generated to show

Parameter	Description
	the leak detection path. If a leak detection path is not found, 1D elements will not be generated.

Volume Meshing Mesh Controls

Model, local and volume selector mesh controls for volume meshing.

Model

Model mesh controls define boundary layer and/or tetrameshing parameters.

BL + Tetra

BL + Tetra model mesh controls define boundary layer and tetrameshing parameters.

Entity Selection Parameters

In the Entities field, use the entity selector to select the entities that the mesh control applies to. The following entities can be selected using the entity selector:

- Components
- Elements
- Regions (solid selection only)
- Solids

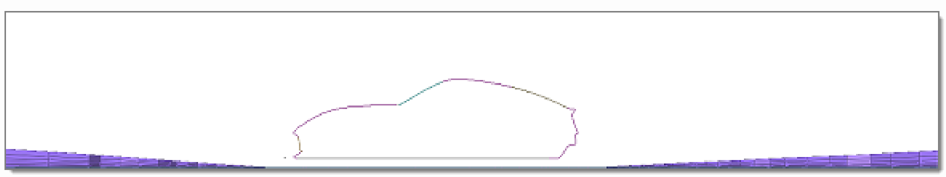
> **Note:** If you have changed your selection to solid or region in model volume mesh controls, existing local controls that have elements or components selected will be made inactive. Any new local mesh controls will have surfaces set for their default selection.
>
> If regions are selected, final volume mesh controls will be placed in a component with the same name as the region.
>
> Meshing will only work if surfaces or solids have mesh associated with them.

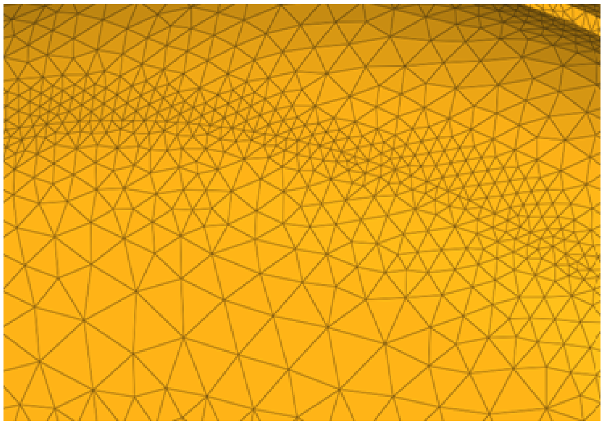
Boundary Layer Parameters

Available parameters vary depending on the Method you select: Simple, Advanced, User Defined. For Simple and Advance mesh controls, refer to Link to CFD Tetramesh Panel.

Table 240: Parameters

Parameter	Description
Basic Surface Mesh Treatment	**Fixed** Prohibit selected elements from being modified.

Parameter	Description
	Float Enable 2D base elements to be modified, if necessary. Generally 2D base elements with NoBL are modified when refinement zones are defined and/or when the BL imprints on them.
BL Definition	When enabled, parameters will be editable and will be applied to all selections. When disabled, a BL definition will not be applied at model level. Local controls will override the BL definition at the model level.
First Layer Thickness	Specify the thickness of the first boundary layer.
First Layer Thickness Method (A)	**Constant** Enable a constant thickness to be defined for the first boundary layer of the selection. **As Factor of Base 2D Elements** Enable a factor, which will be multiplied by the average element size, to be defined. The first layer height for each element equals the average element size multiplied by the factor. This option is useful when the size of 2D elements varies significantly and a constant first layer height is not needed. With this factor, a smooth BL to tetramesh transition for all elements can be achieved.  *Figure 894: First Layer Thickness Method (A)*
Growth Rate	Determine how rapidly elements can increase in size as they are created further and further away from features.

<table>
<tr><th>Parameter</th><th>Description</th></tr>
<tr><td></td><td>
Figure 895: Growth Rate

Elements further from the features grow larger with each row.</td></tr>
<tr><td>BL Growth Rate Method (A)</td><td>Constant
Enable a constant ratio to be defined, which determines how boundary layers grow.

Acceleration
Enable a growth acceleration for boundary layers to be defined beyond the first few layers. This option acts as a growth rate on the growth rate, but only after the first few initial boundary layers. A Start Acceleration from Layer must be defined first, and then from that layer the acceleration will be started. An Acceleration to the initial growth rate and a Maximum Growth Rate must also be defined.
By default, the first two boundary layers grow by the growth rate described above. However, subsequent layers grow by the growth rate multiplied by the acceleration factor. Thus, if d is the initial thickness, r is the initial growth rate, and a is the acceleration rate, then the thicknesses of the successive layers are d, d*r, d*r*(r*a), d*r*(r*a)^2, and so on.

Aspect Ratio Based
Enable the growth rate definition for boundary layers to be based on the defined aspect ratio of the final layer. After the first few initial boundary layers, if this type of growth rate method is selected, the rest of the BL will grow to achieve the user defined Final layer height / base ratio.</td></tr>
<tr><td>BL Thickness Control (UD)</td><td>Enable this option to enter either the Number of layers or the Total BL thickness.</td></tr>
</table>

Parameter	Description
Second Group (UD)	Help to get a smooth transition between BL layers and the tet core more quickly, by defining a higher growth rate.
Final Layer Height/Base Ratio	Define the ratio between the total boundary layer thickness and the average element size of the base surface elements.
BL Stopping Criteria (A)	Determine what to do when BL has reached the defined criteria for Final Layer Height/Base Ratio. **Chop Off Layers** Chop off the BL if elements reach the aspect ratio criteria. **Keep Growing Gr=1** Allow the BL to grow until the neighboring elements begin to grow, even if elements reach the aspect ratio criteria with GR =1.
Number of Layers (S)	Define the total number of layers to be generated using the specified first layer thickness and growth rate.
Hexa Transition Mode	**Simple Pyramid** Transition from a BL hexahedral's quad face to a tetrahedral core mesh using one pyramid element. The height of pyramid elements is controlled by a simple transition ratio parameter, which represents the ratio between the transition pyramid height and the characteristic size of the base quad. **All Prism** Split quad elements in the surface mesh into two trias each so that there will be no need to transition from quad faces to tria faces when transitioning from the last boundary layer to the tetrahedral core. This mode is very important when there are quad elements on areas with (low) distributed BL thickness ratios, because in such areas the thickness of the transition elements, for example simple pyramid, was not taken into account when doing the interference study to assign distributed BL thickness ratio to those elements. **All Tetras** Generate tetra elements only in the boundary layer and splits the quad elements of the surface mesh into tria elements.
Boundary Layer Only	Generate only the boundary layer, stopping before the tetrahedral core is generated. Adjacent surface meshes are also modified to reflect changes introduced by the boundary layer thickness. A collector named ^CFD_trias_for_tetramesh is created and is typically used to generate the inner core tetrahedral mesh using the Tetramesh parameters subpanel.

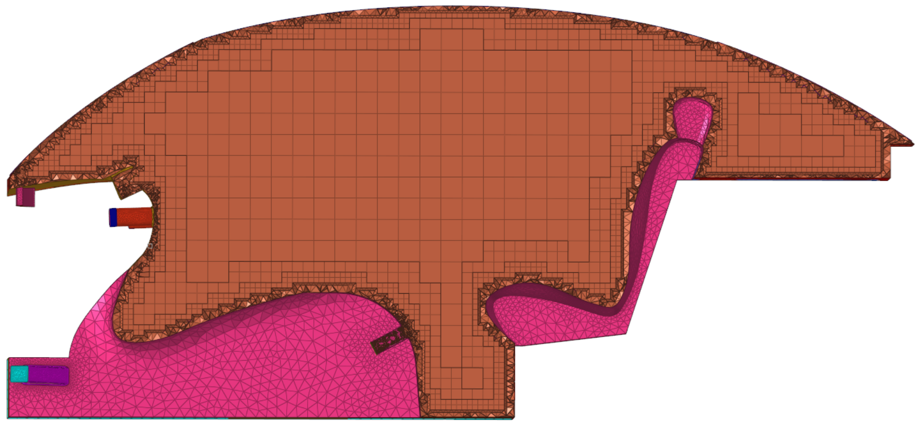
Parameter	Description
	Boundary layer elements are placed in a collector named CFD_boundary_layer; core tetrahedral elements are placed in a collector named CFD_Tetramesh_core. Both collectors are automatically created if they do not exist. However, if these collectors do exist, it is recommended that you empty them before meshing; otherwise there will be more than one set of elements occupying the same physical volume. If you mesh the volume in several steps (multi-volume meshing), it is recommended that you empty the collector before generating the mesh for the next adjacent volume.

Core Mesher Parameters

Available options will direct how the core domain should be meshed. All options will also rely on option in “Tetra Mesh” section.

Table 241: Parameters

Parameter	Description
Core Mesh	**Tetra Mesh** Will create tetra mesh in core. **Hex Dominant** Will create hex elements in core and create pyramids/ tetrahedral in transition region. All elements will have conformal connectivity. Core Hex elements will be created based on user defined Hexa Size. Height of tetra/pyramid transition region can be controlled using Tet-core Layer Height Factor. **Octree Dominant** Will create octree elements in core and create pyramids/ tetrahedral in transition region. All elements except core octree elements will have conformal connectivity. Core Octree elements will be creates based on user defined Max Octant Size and Min Octant Size. Height of tetra/pyramid transition region can be controlled using Tet-core Layer Height Factor.

Parameter	Description
	 *Figure 896: Core Octree Element*

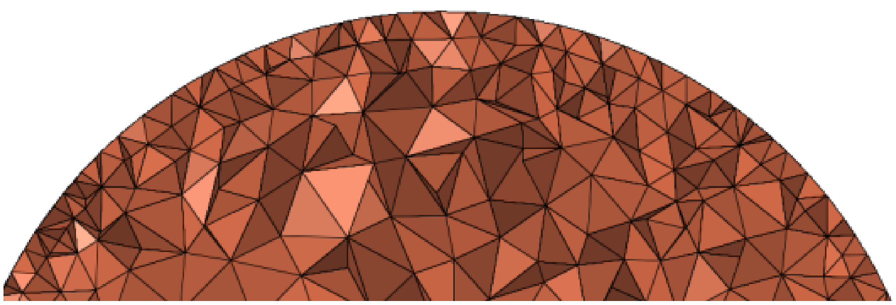
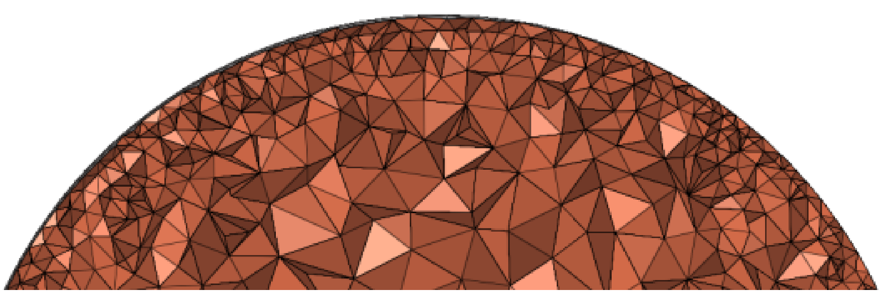
Tetra Mesh Parameters

The tetramesher is multi-threaded and will utilize available threads for meshing similar. This behavior is similar to that of boundary layer meshing.

Table 242: Parameters

Parameter	Description
Element Size Limits	Specify the average, minimum, maximum, or minimum/maximum size of the tetramesh. **None** The maximum element size will be determined by the input 2D elements size and growth rate. **Average Size** Enter the average element size for the tetramesh. If you enter 10, the element sizes will range between 6.6 and 14. **Maximum Size** Tetra element will not be above this size. **Minimum Size** Tetra elements will not be below this size.

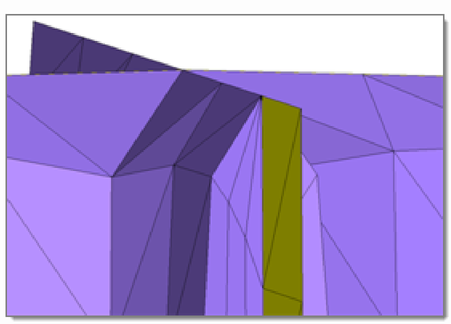
<table>
<tr><th>Parameter</th><th>Description</th></tr>
<tr><td></td><td>Minimum/Maximum Size
Tetra elements will not be below or above this size.
Maximum element size guidelines:
When the input shell element size is close to the user defined maximum tetra size, then the maximum tetra size is used in averaged sense (therefore the actual maximum size may be larger than defined). This prevents a large number of elements from being created.
When the input shell element size is sufficiently different then the user defined maximum tetra size, the maximum size will be enforced.</td></tr>
<tr><td>Quality</td><td>Normal
Use the standard tetra-meshing algorithm.
Optimize Speed
Use an algorithm for faster meshing. Use this option if element quality considerations are less important than mesh generation time.
Optimize Quality
Spend more time optimizing element quality, and employs the volumetric ratio, or CFD skew measurement for tetras as a quality measure. Use this option if your solver is sensitive to element quality.</td></tr>
<tr><td>Tetra Mesh Method</td><td>Select a tetra mesh method:
Delaunay
Enable a mesher, which is implemented based on the delaunay approach. This method is recommended for improved performance.
Advancing Front
Enablesthe legacy tetra mesher.
Octree Based
Enable an octree structured based tetrameshing. Smoothing near boundaries will be performed with this method.</td></tr>
<tr><td>Growth Rate</td><td>The Growth rate parameter works as follows: if d is the initial thickness and r is the initial growth rate, then the thicknesses of the successive layers are d, d*r, d*r^2, d*r^3, d*r^4, and so on.
If element quality is important and you are not concerned with the total number of elements being created, then Interpolate will produce the best results because the element size changes smoothly and therefore the element quality is better.</td></tr>
</table>

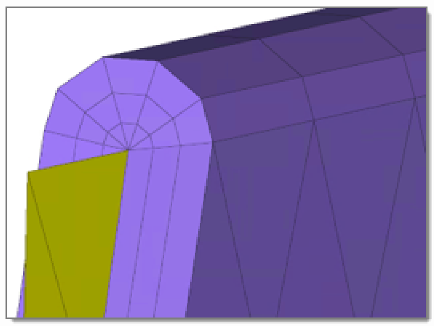
Parameter	Description
	Different default values are specified for the various growth rate options: **Standard** 1.2 **Aggressive** 1.35 **Gradual** 1.08 **Interpolate** 1.08 **User Controlled, Octree based, Delaunay** Define your own value when you select this option.
Tetramesh Height Factor Near Boundary	The Delaunay method allows options to control the height of tetra mesh near boundary. Tetra transition from boundary layer or surfaces can be controlled using this factor.  *Figure 897: Height Factor = 1*  *Figure 898: Height Factor = .5*

Parameter	Description
Pyramid Transition Ratio	Define the relative height of pyramid elements used for the transition from hexa elements in the boundary layer to the tetra elements in the core.
Smoothing	Apply an extra stage of calculation to improve overall mesh quality. Additional smoothing and swapping steps will be performed and tetra elements will be split to achieve a smoother mesh transition. If tetra elements are used in the boundary layer, then those elements will be excluded from smoothing to maintain the original distribution.
Use Number of Layers	Define the number of tetrahedral layers to generate. When enabled, the Tetramesher ensures the tetracore contains, at minimum, the specified number of tetra layers in the model. This functionality ensures a certain mesh resolution in case of close proximity or thin channels. When generating multiple tetrahedral layers, keep the following restrictions in mind: • Do not generate more than three or four layers, unless you refine the surfaces to have a fine mesh at close proximity areas. • Layer meshes will not be created near narrow strip surfaces, as the current algorithm does not alter the surface mesh given.

Advanced Parameters

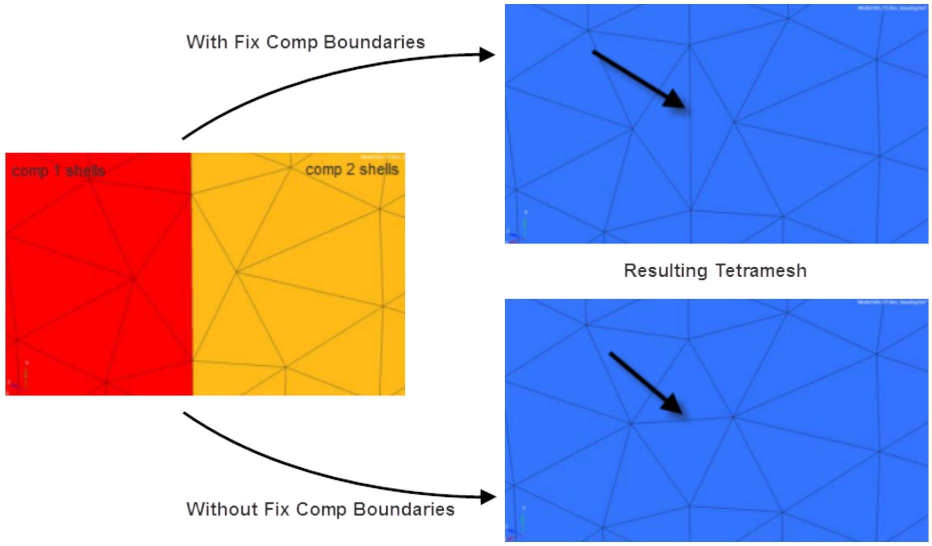
Table 243: Parameters

Parameter	Description
Boundary Layer Propagation	
Treatment at Sharp Edges	**Node collapse** Collapse BL at baffles or sharp corners below the defined angle.  *Figure 899: Node Collapse*

Parameter	Description
	The baffle is colored yellow. **Multiple normal** Grow multiple normals around baffles or sharp corners below the defined angle. If two adjacent elements enclose an angle smaller than the threshold (sharp edge pointing into the volume), normals will be computed on that edge and the boundary layer will consider those normals.  *Figure 900: Multiple Normal* *The baffle is colored yellow.*
Sharp Edge Angle	Define the threshold of angles below which the BL should be collapsed.
Minimum Imprint Angle from BL to Non-BL	Control which cases to imprint BL entities on without BL components. If the angle between the BL component and the non BL component is high, imprinting will create high aspect ratio elements. If the angle between BL and Non-BL entities (component elements) is less than the imprint angle or greater than (180-imprint angle), then the BL will collapse rather than imprinting on non-BL entities. Recommended range: 6-10
Max Layer Difference Between Neighbors	Control the maximum layer difference between neighboring elements. This parameter helps avoid situations where all BLs collapse at once, and also provides smooth BL transitions in cases of BL truncation. A good value for this parameter is 1/4 of the total BL layers. The value specified also depends on layer height. Recommended range: Depends on how many layers you are growing.
Proximity	
Maximum BL Compression	Enable BL compression, or squeezing, when there is not enough space available for the BL to grow. The BL will try to compress by the max BL compression factor first. For example, if the original total BL height is defined as 1, with a 0.4 max BL compression, the BL layers will try

Parameter	Description
	to be compressed until 0.6 of the total height is reached. Once the BL is compressed to this value, the mesher will start chopping off layers if there is not enough space. A value of zero enforces no BL compression, which is useful when you want to maintain the BL height; a value of one enables the maximum possible compression. Recommended range: 0-0.6
Minimum BL Thickness/ Base Ratio	Due to close proximity, the BL will sometimes only be able to generate one to two layers (a very small total BL height at that location). At that location, it might be possible that the transition between BL layers and the tetra core is bad. With this factor, if the total BL height is less than the defined factor base size, all of the BL layers will be chopped off. By default, this value is zero, which disables the effects of this parameter.
Minimum Tetcore/Final Layer Height Ratio	Control the minimum height of the tet core as a factor of the final layer height. After creating the BL in close proximity, there will be a small space available for tetramesh. This results in high aspect ratio tetra elements. Recommend value: 1.3 (default)
Boundary Layer Quality	
Generation Method	Controls the growth of boundary layers. **Optimize Quality** Use a set of meshing parameters, which ensures a good quality boundary layer in most cases. **Optimize Speed** Choose meshing parameters in a way that the meshing time is minimized and an acceptable boundary layer quality is achieved.
Maximum Cell Skewness	Chop off BL cells exceeding the defined maximum cell skewness. This parameter prevents the generation of highly skewed elements. The tetra mesher sometimes creates better quality elements compared to the BL mesher. If your input 2D mesh has bad element quality and topology, it is recommended that you define a higher value. Recommended range: 0.8 - 0.95

Parameter	Description
Minimum Normalized Jacobian	Chop off BL cells exceeding the defined minimum normalized Jacobian. This parameter prevents the generation of negative elements. Recommended range: 0.05 - 0.2
Tetra Quality	
Element Quality Target	Select an element criteria and threshold. After the tetrameshing step, a mesh optimization step will be performed to fulfill the defined threshold for the selected element criteria. Available quality criteria include: Volume Skew, Tetra Collapse, and Cell Squish.
Volume Setup	
Validate 2D Input	Check BL elements before tetrameshing to rectify if there is anything wrong with the input (intersecting elements) provided for the tetramesh.
Fix Invalid 2D Element	Fix invalid elements (at present only unoffsettable nodes) before volume meshing. For unoffsettable nodes (where BL collapses if not smooth), the connected elements will be smoothed where the BL will be generated.
Fix Component Boundaries	Anchor nodes are maintained during CFD tetrameshing, so that the new mesh adheres to them. 1D elements can be selected instead of nodes if you need a tetra element edge at a certain location. Select this option when certain mesh nodes or edges are required on a certain location, such as for post-processing purposes. If the Float option is selected for some boundary regions, surface shell edges will be swapped during mesh generation. However, this prevents the swapping of edges between two components.

Parameter	Description
	 *Figure 901:*
Update Input Shells	Automatically update the shells on all boundaries after meshing. Updated shell elements are placed in the initial boundary shell components.
Fill Void	Mesh all volumes. If your geometry includes volumes inside of another volume, enable this parameter. For example, if you had a sphere inside of a larger sphere, enabling this parameter would cause the volume of the inner sphere as well as the volume between the two spheres to be meshed.
Other	
Anchor Node	Node that will remain and be re-used in the new mesh. Anchor nodes are "fixed" so that the automesher cannot move or replace them; in essence, they are exceptions to the re-meshing operation, and the new mesh must utilize them.

Tetra

Tetra mesh controls define tetrameshing parameters.

Entity Selection Parameters

In the Entities field, use the entity selector to select the entities that the mesh control applies to. The following entities can be selected using the entity selector:

- Components
- Elements
- Regions (solid selection only)
- Solids

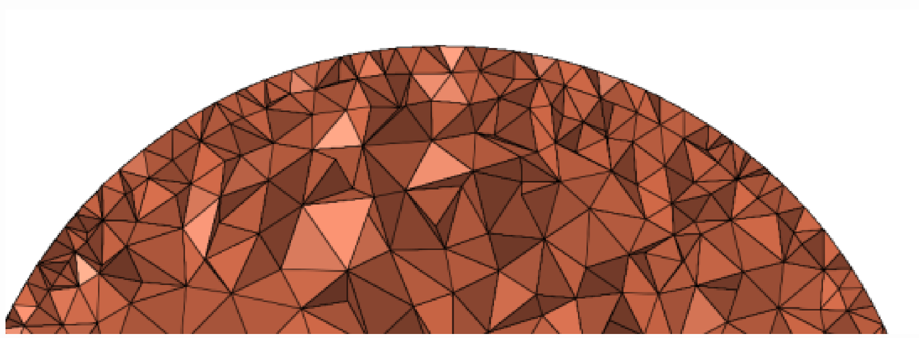
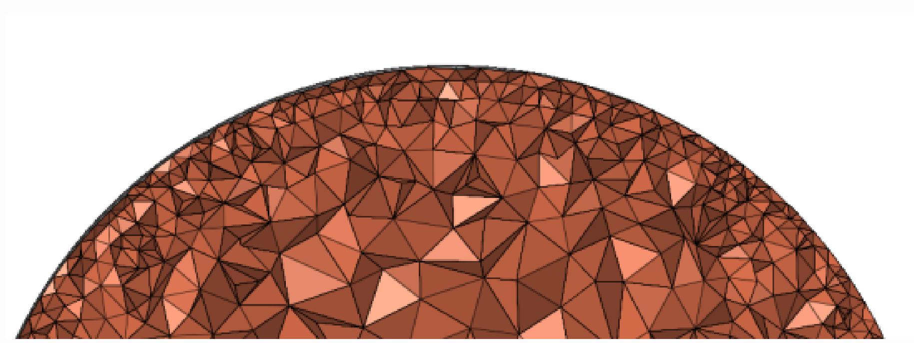
> **Note:**
>
> If you have changed your selection to solid or region in model volume mesh controls, existing local controls that have elements or components selected will be made inactive. Any new local mesh controls will have surfaces set for their default selection.
>
> If regions are selected, final volume mesh controls will be placed in a component with the same name as the region.
>
> Meshing will only work if surfaces or solids have mesh associated with them.

Tetra Mesh Parameters

Table 244: Parameters

Parameter	Description
Base Surface Mesh Treatment	**Fixed** Prohibit selected elements from being modified. **Float** Modify 2D base elements, if necessary. Generally 2D base elements with NoBL are modified when refinement zones are defined and/or when the BL imprints on them.
Element Size Limits	Specify the average, minimum, maximum, or minimum/maximum size of the tetramesh. **None** The maximum element size will be determined by the input 2D elements size and growth rate. **Average Size** Enter the average element size for the tetramesh. If you enter 10, the element sizes will range between 6.6 and 14. **Maximum Size** Tetra element will not be above this size. **Minimum Size** Tetra elements will not be below this size. **Minimum/Maximum Size** Tetra elements will not be below or above this size. Maximum element size guidelines: • When the input shell element size is close to the user defined maximum tetra size, then the maximum tetra size is used in

Parameter	Description
	averaged sense (therefore the actual maximum size may be larger than defined). This prevents a large number of elements from being created. • When the input shell element size is sufficiently different then the user defined maximum tetra size, the maximum size will be enforced.
Quality	**Normal** Use the standard tetra-meshing algorithm. **Optimize Speed** Use an algorithm for faster meshing. Use this option if element quality considerations are less important than mesh generation time. **Optimize Quality** Spend more time optimizing element quality, and employs the volumetric ratio, or CFD skew measurement for tetras as a quality measure. Use this option if your solver is sensitive to element quality.
Tetra Mesh Method	Select a tetra mesh method: **Delaunay** Enable a mesher, which is implemented based on the delaunay approach. This method is recommended for improved performance. **Advancing Front** Enablesthe legacy tetra mesher. **Octree Based** Enable an octree structured based tetrameshing. Smoothing near boundaries will be performed with this method.
Growth Rate	The Growth rate parameter works as follows: if d is the initial thickness and r is the initial growth rate, then the thicknesses of the successive layers are d, d*r, d*r^2, d*r^3, d*r^4, and so on. If element quality is important and you are not concerned with the total number of elements being created, then Interpolate will produce the best results because the element size changes smoothly and therefore the element quality is better. Different default values are specified for the various growth rate options: **Standard** 1.2

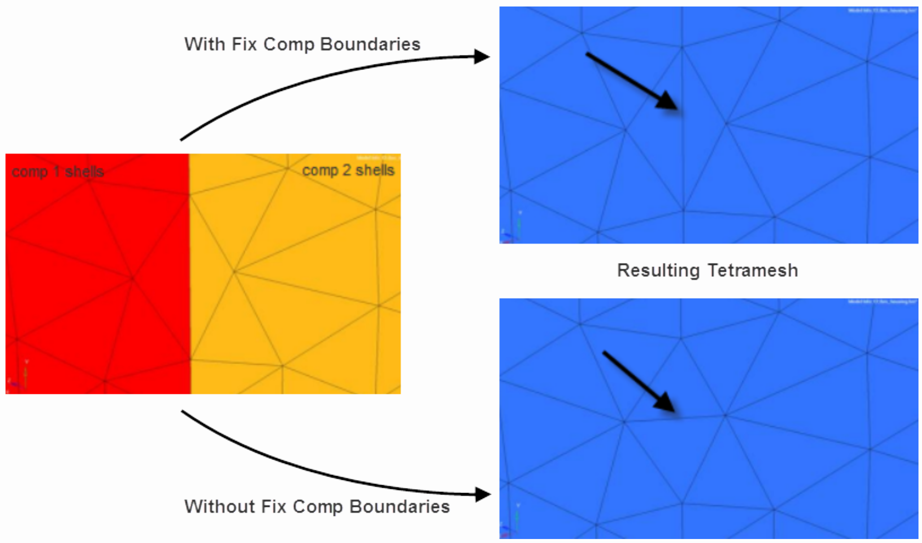
Parameter	Description
	Aggressive 1.35 **Gradual** 1.08 **Interpolate** 1.08 **User Controlled, Octree based, Delaunay** Define your own value when you select this option.
Tetramesh Height Factor Near Boundary	Delaunay method allow option to control height of tetra mesh near boundary. Tetra transition from boundary layer or surfaces can be controlled using this factor.  *Figure 902: Height Factor = 1*  *Figure 903: Height Factor = 0.5*
Pyramid Transition Ratio	Define the relative height of pyramid elements used for the transition from hexa elements in the boundary layer to the tetra elements in the core.

Parameter	Description
Smoothing	Apply an extra stage of calculation to improve overall mesh quality. Additional smoothing and swapping steps will be performed and tetra elements will be split to achieve a smoother mesh transition. If tetra elements are used in the boundary layer, then those elements will be excluded from smoothing to maintain the original distribution.
Use Number of Layers	Define the number of tetrahedral layers to generate. When enabled, the Tetramesher ensures the tetracore contains, at minimum, the specified number of tetra layers in the model. This functionality ensures a certain mesh resolution in case of close proximity or thin channels. When generating multiple tetrahedral layers, keep the following restrictions in mind: • Do not generate more than three or four layers, unless you refine the surfaces to have a fine mesh at close proximity areas. • Layer meshes will not be created near narrow strip surfaces, as the current algorithm does not alter the surface mesh given.

Advanced Parameters

Table 245: Parameters

Parameter	Description
Tetra Quality	
Element Quality Target	Select an element criteria and threshold. After the tetrameshing step, a mesh optimization step will be performed to fulfill the defined threshold for the selected element criteria. Available quality criteria include: Volume Skew, Tetra Collapse, and Cell Squish.
Volume Setup	
Validate 2D Input	Check BL elements before tetrameshing to rectify if there is anything wrong with the input (intersecting elements) provided for the tetramesh.
Fix Invalid 2D Element	Fix invalid elements (at present only unoffsettable nodes) before volume meshing. For unoffsettable nodes (where BL collapses if not smooth), the connected elements will be smoothed where the BL will be generated.

Parameter	Description
Fix Component Boundaries	Anchor nodes are maintained during CFD tetrameshing, so that the new mesh adheres to them. 1D elements can be selected instead of nodes if you need a tetra element edge at a certain location. Select this option when certain mesh nodes or edges are required on a certain location, such as for post-processing purposes. If the Float option is selected for some boundary regions, surface shell edges will be swapped during mesh generation. However, this prevents the swapping of edges between two components.  *Figure 904:*
Update Input Shells	Automatically update the shells on all boundaries after meshing. Updated shell elements are placed in the initial boundary shell components.
Fill Void	Mesh all volumes. If your geometry includes volumes inside of another volume, enable this parameter. For example, if you had a sphere inside of a larger sphere, enabling this parameter would cause the volume of the inner sphere as well as the volume between the two spheres to be meshed.
Other	
Anchor Node	Node that will remain and be re-used in the new mesh. Anchor nodes are "fixed" so that the automesher cannot move or replace them; in

Parameter	Description
	essence, they are exceptions to the re-meshing operation, and the new mesh must utilize them.

Local

Local mesh controls define regions where boundary layers are desired, or are not desired.

No BL

No BL local mesh controls define components/elements on which boundary layers mesh is not required.

Entity Selection Parameters

In the Entities field, use the entity selector to select the entities that the mesh control applies to. The following entities can be selected using the entity selector:

- Components
- Elements
- Regions (solid selection only)
- Solids

Note:

If you have changed your selection to solid or region in model volume mesh controls, existing local controls that have elements or components selected will be made inactive. Any new local mesh controls will have surfaces set for their default selection.

If regions are selected, final volume mesh controls will be placed in a component with the same name as the region.

Meshing will only work if surfaces or solids have mesh associated with them.

Boundary Layer Parameters

Table 246: Parameters

Parameter	Description
Basic Surface Mesh Treatment	**Fixed** Prohibit selected elements from being modified.

Parameter	Description
	Float Enable 2D base elements to be modified, if necessary. Generally 2D base elements with NoBL are modified when refinement zones are defined and/or when the BL imprints on them.

Local BL

Local BL local mesh controls define local boundary layer settings. Any settings defined in the model mesh controls will be overridden with the BL settings defined in local mesh controls.

Entity Selection Parameters

In the Entities field, use the entity selector to select the entities that the mesh control applies to. The following entities can be selected using the entity selector:

- Components
- Elements
- Regions (solid selection only)
- Solids

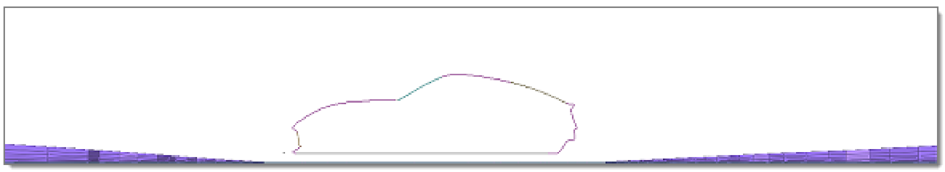
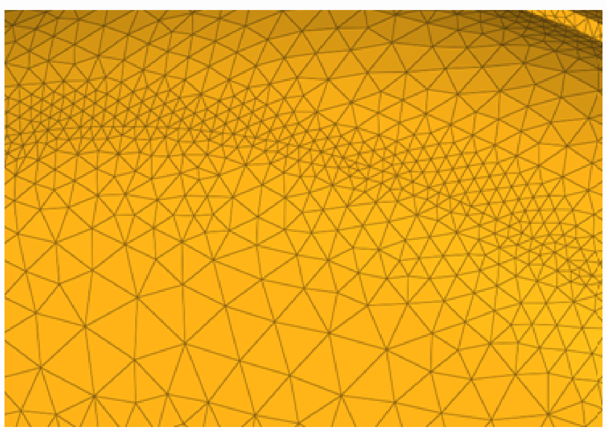
> **Note:**
>
> If you have changed your selection to solid or region in model volume mesh controls, existing local controls that have elements or components selected will be made inactive. Any new local mesh controls will have surfaces set for their default selection.
>
> If regions are selected, final volume mesh controls will be placed in a component with the same name as the region.
>
> Meshing will only work if surfaces or solids have mesh associated with them.

Boundary Layer Parameters

Available parameters vary depending on the Method you select: Simple, Advanced, User Defined.

Table 247: Parameters

Parameter	Description
Base Surface Mesh Treatment	**Fixed** Prohibit selected elements from being modified. **Float** Modify 2D base elements, if necessary. Generally 2D base elements with NoBL are modified when refinement zones are defined and/or when the BL imprints on them.
First Layer Thickness	Specify the thickness of the first boundary layer.

Parameter	Description
First Layer Thickness Method	**Constant** Define a constant thickness for the first boundary layer of the selection. **As Factor of Base 2D Elements** Enable a factor, which will be multiplied by the average element size, to be defined. The first layer height for each element equals the average element size multiplied by the factor. This option is useful when the size of 2D elements varies significantly and a constant first layer height is not needed. With this factor, a smooth BL to tetramesh transition for all elements can be achieved.  *Figure 905: First Layer Thickness Method*
Growth Rate	Determines how rapidly elements can increase in size as they are created further and further away from features.  *Figure 906: Growth Rate* Elements further from the features grow larger with each row.
BL Growth Rate Method	**Constant** Define a constant ratio, which determines how boundary layers grow.

Parameter	Description
	Acceleration Define a growth acceleration for boundary layers beyond the first few layers. This option acts as a growth rate on the growth rate, but only after the first few initial boundary layers. A Start Acceleration from Layer must be defined first, and then from that layer the acceleration will be started. An Acceleration to the initial growth rate and a Maximum Growth Rate must also be defined. By default, the first two boundary layers grow by the growth rate described above. However, subsequent layers grow by the growth rate multiplied by the acceleration factor. Thus, if d is the initial thickness, r is the initial growth rate, and a is the acceleration rate, then the thicknesses of the successive layers are d, d*r, d*r*(r*a), d*r*(r*a)^2, and so on. **Aspect Ratio Based** Define the growth rate definition for boundary layers based on the defined aspect ratio of the final layer. After the first few initial boundary layers, if this type of growth rate method is selected, the rest of the BL will grow to achieve the user defined Final layer height / base ratio.
BL Thickness Control	Enables this option to enter either the Number of layers or the Total BL thickness.
Second Group	Help to get a smooth transition between BL layers and the tet core more quickly, by defining a higher growth rate.
Final Layer Height / Base Ratio	Define the ratio between the total boundary layer thickness and the average element size of the base surface elements.
Number of Layers	Define the total number of layers to be generated using the specified first layer thickness and growth rate.
BL Stopping Criteria	Determine what to do when BL has reached the defined criteria for Final Layer Height/Base Ratio. **Chop Off Layers** Chop off the BL if elements reach the aspect ratio criteria. **Keep Growing Gr=1** Grow BL until the neighboring elements begin to grow, even if elements reach the aspect ratio criteria with GR =1.

Advanced Parameters

Table 248: Parameters

Parameter	Description
Use Global Values	Values defined for advanced parameters will be taken from the advanced parameters defined for the model mesh control.
Maximum BL Compression	Enable BL compression, or squeezing, when there is not enough space available for the BL to grow. The BL will try to compress by the max BL compression factor first. For example, if the original total BL height is defined as 1, with a 0.4 max BL compression, the BL layers will try to be compressed until 0.6 of the total height is reached. Once the BL is compressed to this value, the mesher will start chopping off layers if there is not enough space. A value of zero enforces no BL compression, which is useful when you want to maintain the BL height; a value of one enables the maximum possible compression. Recommended range: 0-0.6
Minimum BL Thickness / Base Ratio	Due to close proximity, the BL will sometimes only be able to generate one to two layers (a very small total BL height at that location). At that location, it might be possible that the transition between BL layers and the tetra core is bad. With this factor, if the total BL height is less than the defined factor base size, all of the BL layers will be chopped off. By default, this value is zero, which disables the effects of this parameter.

Volume Selector

Volume Selector mesh controls define which volumes should be meshed and how mesh should be generated. Only one instance of a volume selector mesh control is allowed.

The parameters defined for Volume Selector mesh controls are applicable to both BL + Tetra and Tetra model mesh controls.

Table 249: Parameters

Parameter	Description
Select Volumes	Defines which volumes to mesh.

Parameter	Description
	All Volumes Mesh all of the volumes in the model. This option is also affected by the parameter Fill Void, which fills of the voids (volume completely enclosed in another volume) when enabled. Example: When this option is enabled, and there is a sphere inside of a larger sphere, the volume of the inner sphere as well as the volume between the two spheres will be meshed. **Exclude Enclosed** Mesh all of the volume except for the volumes enclosed by the defined seed node. The seed node should be enclosed in the volume. **Nth Largest** Select volumes to mesh based on size. Specify whether to select the 1st largest, 2nd largest, ... using the volume index "N", which is volume number. If you do not specify N, the smallest volume will be meshed by default. **By Seed Nodes/Elems** Select volumes to be meshed by either specifying a seed node (the seed node should be enclosed in the volume) or touching elements or geometric solids (if input is solid). All can be defined at same time. If there is a conflict between fluid and solid volumes, the fluid volume will take precedence.
Mesh to File	Store the generated mesh in a `.nas` or `.hmx` file after meshing is finished. When enabled, specify a location to export the mesh.
BL and Tetras in One Component	Store BL elements and tetra elements in one component. When disabled, BL elements and tetra elements will be stored in separate components, which is useful when you need to define morphing constraints on BL elements.
Generate BL Contours	Generate a `.res` file in your working directory of BL result contours (Number of layers, first layer height, total BL thickness) for each input element after volume meshing is finished. This file will automatically be assigned the same name as the HyperMesh model file, but it will have a `.res`extension. BL contours help you visualize how BLs are generated. View this file in the Contour panel, or by clicking **File** > **Load** > **Results** from the menu bar. Scroll through the available results.

Parameter	Description
	Only applicable to BL + Tetra model mesh controls.

Volume Mesh Control Options

Check mesh quality using tools accessed from the context menu that opens when you right-click on a Volume Mesh folder in the Mesh Controls browser.

Check 2D Mesh

Validate the input surface mesh before performing volume mesh generation using the Boundary Shell Checker tool.

Solid Mesh Optimization

Fix 3D elements in the following ways using the Solid Mesh Optimization tool.

- Fix hexa and tetra element quality with respect to several element criteria.
- Fix second order element's maximum angle, and minimum and maximum length ratio and Jacobian.

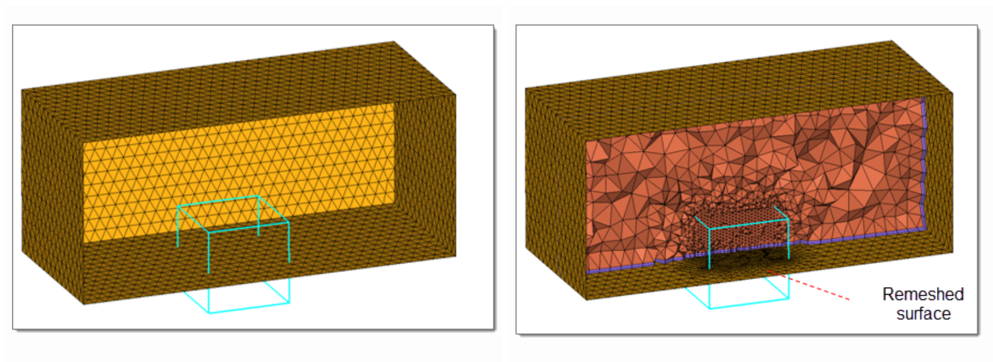
Refinement Zone Mesh Controls

Refinement zones are used to refine mesh with a defined size.

Create refinement zones for surface meshing (surface deviation only), adaptive wrap meshing, and volume meshing (BL+ Tetra and Tetra).

Table 250: Parameters

Parameter	Description
Enabled for Volume Mesh / Surface Mesh / Wrapping Mesh	Enable refinement zones for volume meshing, surface meshing, and/or adaptive wrap meshing.
Element Size	Define the target element size for mesh inside of the refinement box. **Note:** The actual mesh size will vary in order to maintain mesh connectivity at the edges of the box. In the example below, the boundary region has been selected as With BL (float) and remesh, therefore the region included in the refinement box has been remeshed with the elements size assigned to the refinement box.

Parameter	Description
	 *Figure 907:*
Shape	Define the refinement zone shape. **Box** Is defined by: **By Center and Size** Use the selector to select a center node and specify the x, y, and z dimensions of each side of the box. Example: A size of 5 creates a 5 x 5 x 5 box centered around the center node. **By Two Nodes** Use the selector to select two nodes which represent opposite corners of a cubic volume. **By Four Nodes** Use the selector to select a base node and three additional nodes. The four selected nodes cannot be coplanar. The base node, N1, and N2 form a triangle, which is then flipped 180 degrees to form a rectangular base for the refinement box. The vector from base node to N3 defines the refinement box's height and direction from the base. **By Eight Nodes** Use the selector to select all eight nodes of the refinement box. The order of the box will be corrected internally if defined incorrectly. **By Element Volume** Use the elems selector to select the elements that define the volume. Define a scaling factor to scale the elements enclosed in the box. A scale factor of 1 creates the smallest possible box

Parameter	Description
	that can still enclose the selected elements, whereas a factor of 2 creates a box twice as large in every dimension. **Cone** Defined by the lower circular face center (as a node or coordinates), lower circular face radius, cylinder height, and normal axis of the cylinder. **Cylinder** Defined by the lower circular face center (as a node or coordinates), lower circular face radius, cylinder height, and normal axis of the cylinder. **Ellipsoid** Defined by four nodes: the center node and three other nodes which define the radius in each direction and also the orientation. **Frustum** Defined by the lower circular face center (as a node or coordinates), lower circular face radius, top circular face radius, cylinder height, and normal axis of the cylinder. **Sphere** Defined by the center (as a node or coordinates) and radius.

Regions

Regions store information used to facilitate and automate modeling practices and processes.

Regions enable a selection which can be common across design changes or other models, provided region data is the same. Regions currently support two configurations of input: By ID and By Metadata. Both inputs are supported for geometry only, and can be used for selection purposes.

For example, when using mesh controls, it is possible to mesh a surface which has been tagged with metadata (or ID). If the design changes and a new version is authored, then you can quickly reapply the same mesh controls as long as the metadata (or ID) is still applied to the new CAD version. Regions enables an automated re-meshing process that is consistently repeatable.

Create Basic Regions

Create basic regions and define them in the Entity Editor.

1. In the Mesh Controls Browser, right-click on the **Region** folder and select **Create** > **Basic** from the context menu.

2. In the Entity Editor, select region parameters accordingly.

Create Regions from Features

Create regions by selecting entities based on their feature type using the Regions from Feature Selection tool.

Regions will be created with the given region name, and will reference the specific entities found by the tool based on the defined feature type.

1. In the Mesh Controls Browser, right-click on the **Region** folder and select **Create** > **From Feature Selection** from the context menu.
2. In the **Regions from Feature Selection** dialog, enter a region name.
3. Select a feature type.
    - Choose **Circles** to select surface edges that represent closed circles within a min and max radius range, optionally limited to only internal edges.

      The edges are found from either the selected surface edges, or from all of the edges of the selected surfaces.
    - Choose **Cylinders** to select surfaces that represent cylinders within a min and max radius range.

      Cylinders may be both closed and partial, or can be optionally limited to only closed.
    - Choose **Fillets** to select surfaces that represent fillets within a min and max radius range.

Create Regions from Metadata to CAE

Create regions from points, lines and surfaces with metadata.

1. In the Mesh Controls Browser, right-click on the **Region** folder and select **Create** > **From Metadata to CAE** from the context menu.
   The **Metadata to CAE** dialog opens.
2. Set Create to **Regions**.
3. Set From to **Points**, **Lines** or **Surfaces**.
4. In the Metadata field, select a specific metadata name.
   A list of entities is generated.
5. In the table, select entities from which to create regions.

6. Click **Apply**.

Import/Export Mesh Controls

Mesh controls can be saved and reused across different models by exporting them to a template file (`*.xml`) and then importing them into a new HyperMesh session from the Mesh Controls Browser.

All defined mesh controls are exported to a single `.xml` file, which can be utilized as a template. The mesh control parameters are saved to the file, while any entity selection is not.

From the Mesh Controls Browser, right-click on the Mesh Controls folder and select **Import** or **Export** from the context menu.

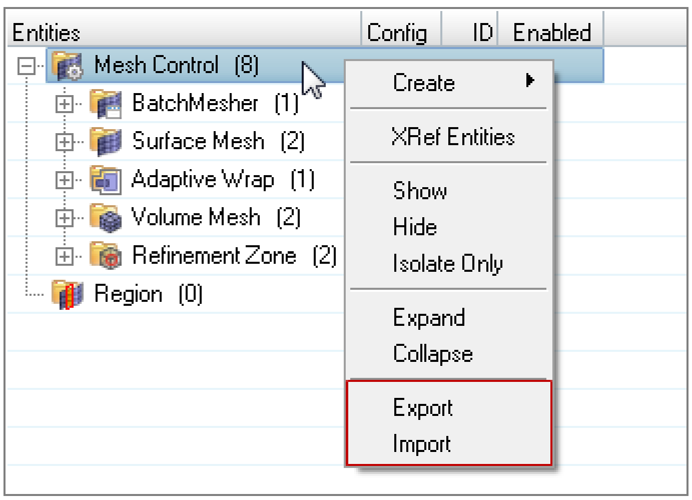


Figure 908:

Create and Edit Mesh Controls

Mesh controls can be created for batch meshing, surface meshing, adaptive wrap meshing, volume meshing and refinement zones.

Model and local mesh control entity selection behaves as follows:

- If entities are selected in the model control, they will be directly used and you will not be prompted for any additional selection.
- If no entities are selected in the model control, you will be prompted for an "on-the-fly" entity selection:
  - The entity type set for the model control is the type of entity you will be prompted to select.

    For example, a size and bias control for surfaces can refer to surfaces, components, and regions. Whatever is set as the current type in the active model control is what you will be promoted to select on-the-fly.
- Entities must be defined for local controls. Entities that are part of the local control selection must also be part of the model/on-the-fly selection. Any local entities that are not part of the model/on-the-fly selection are ignored.

- The use of regions for selection is recommended wherever possible for ease of use and best compatibility with future workflows.

1. From the Mesh Controls Browser, right-click and select **Create** from the context menu.

The mesh control is created, and its corresponding parameters are displayed in the Entity Editor.

2. In the Entity Editor, define mesh control parameters.

a) In the Name field, enter a name for the mesh control.

b) Under Entity Selection, use the entities selector to select entities to associate with the mesh control.

c) Define additional parameters accordingly.

You can edit mesh controls at any time by selecting them in the Mesh Controls Browser and modifying them in the Entity Editor.

Tip:

- Create mesh controls specific for the different meshing types and refinement zones by right-clicking on a corresponding folder in the Mesh Controls tab.
- Duplicate mesh controls by right-clicking on a mesh control and selecting **Duplicate** from the context menu. You can only duplicate one mesh control at a time.

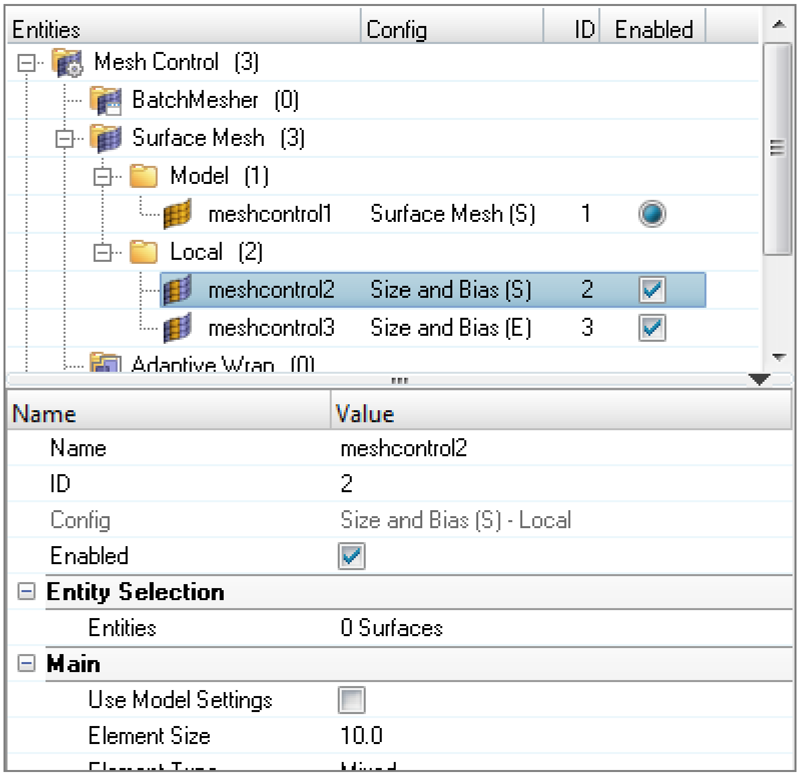


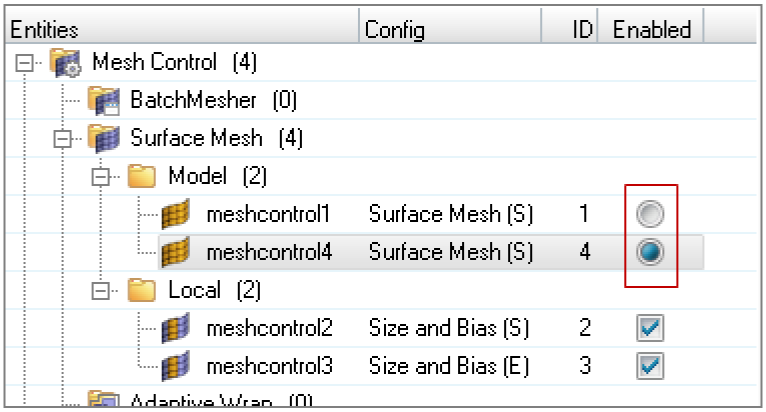
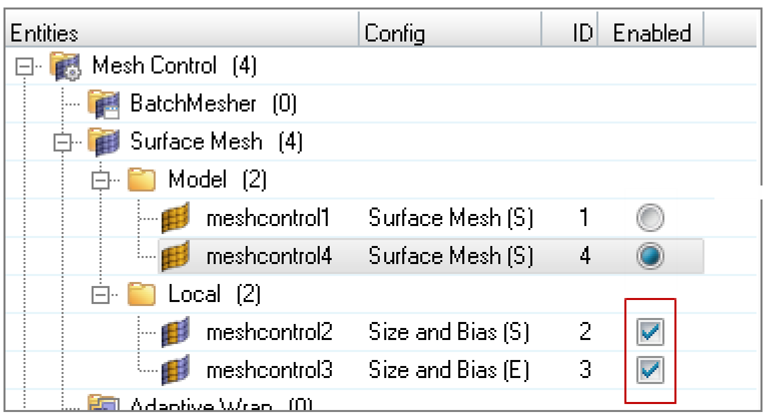
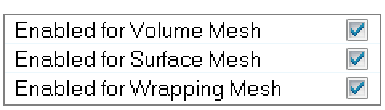
Figure 909:

Enable Mesh Controls

When a meshing job is submitted, enabled mesh controls are considered and used to automate the meshing process.

In the Mesh Controls Browser, enable and disable mesh controls.

Option	Description
Model mesh controls	Multiple instances of model mesh controls can be defined, but only one can be enabled. **1.** Enable a model mesh control by selecting its corresponding radio button in the Enabled column.

Option	Description
	 *Figure 910: Enabled Model Mesh Controls*
Local mesh controls	An unlimited number of local mesh controls can be defined and enabled. • Enable a local mesh control by selecting its corresponding checkbox in the Enabled column. • Disable a local mesh control by clearing its corresponding checkbox.  *Figure 911: Enabled Local Mesh Controls*
Refinement Zone controls	An unlimited number of refinement zone mesh controls can be defined and enabled. Each refinement zone can be independently enabled or disabled for surface meshing (surface deviation only), adaptive wrap meshing, and volume meshing as required.  *Figure 912: Refinement Zone Controls* • Enable a refinement zone for all meshers by selecting its corresponding checkbox in the Enabled column.

Option	Description
	• Disable a refinement zone for all meshers by clearing its corresponding checkbox in the Enabled column. If a refinement zone is disabled for any mesher but enabled for others, a tristate checkbox is displayed.

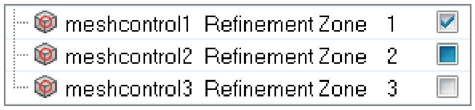


Figure 913: Refinement Zone Control with Tristate Status

Create Mesh

Once you are finished creating mesh controls, generate the mesh from the Mesh Controls Browser.

In the Mesh Controls Browser, right-click on the BatchMesher, Surface Mesh, Adaptive Mesh, or Volume Mesh folders and select **Mesh** from the context menu.

> **Note:**
> A mesh cannot be independently created for Refinement Zone controls.

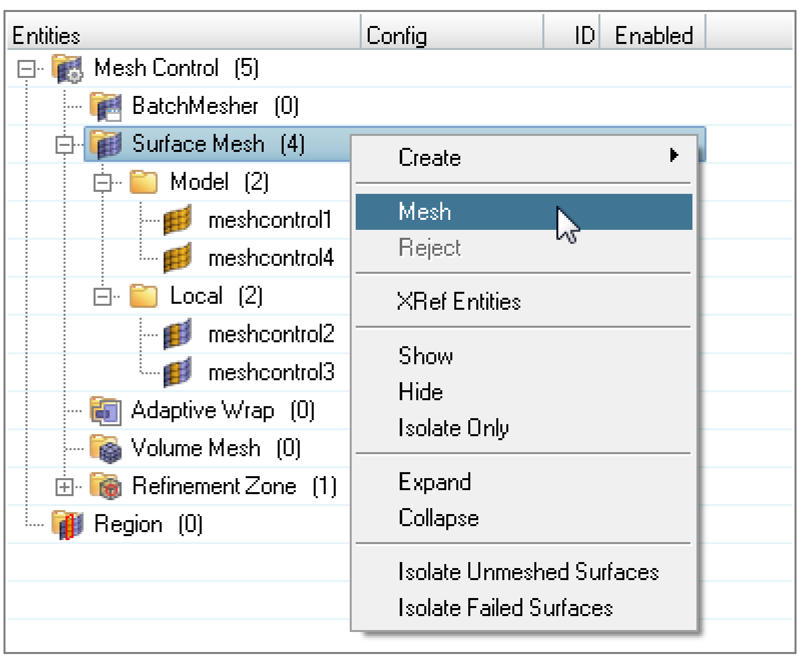


Figure 914:

Check Mesh Quality

Evaluate the overall quality of a mesh, and resolve criteria violations.

Element Quality Calculations

After you build your model, calculate the quality of elements in a mesh.

How Element Quality is Calculated

The quality of elements in a mesh can be gauged in many ways, and the methods used often depend not only on the element type, but also on the individual solver used.

When possible, the most common or standard methods are used, but there is no truly standardized set of element quality checks. When a solver does not support a specific check within HyperMesh, HyperMesh uses its own method to perform the check.

HyperMesh

When possible, HyperMesh checks strive to maintain compatibility with popular solvers.

2D and 3D Element Checks

The following checks apply to both types of elements, but when applied to 3D elements they are generally applied to each face of the element. The value of the worst face is reported as the 3D element's overall quality value.

Aspect Ratio

Ratio of the longest edge of an element to either its shortest edge or the shortest distance from a corner node to the opposing edge ("minimal normalized height"). HyperMesh uses the same method used for the Length (min) check.
For 3D elements, each face of the element is treated as a 2D element and its aspect ratio determined. The largest aspect ratio among these faces is returned as the 3D element's aspect ratio.
Aspect ratios should rarely exceed 5:1

Chordal Deviation

Largest distance between the centers of element edges and the associated surface.
Second order elements return the same chordal deviation as first order, when the corner nodes are used due to the expensive nature of the calculations.

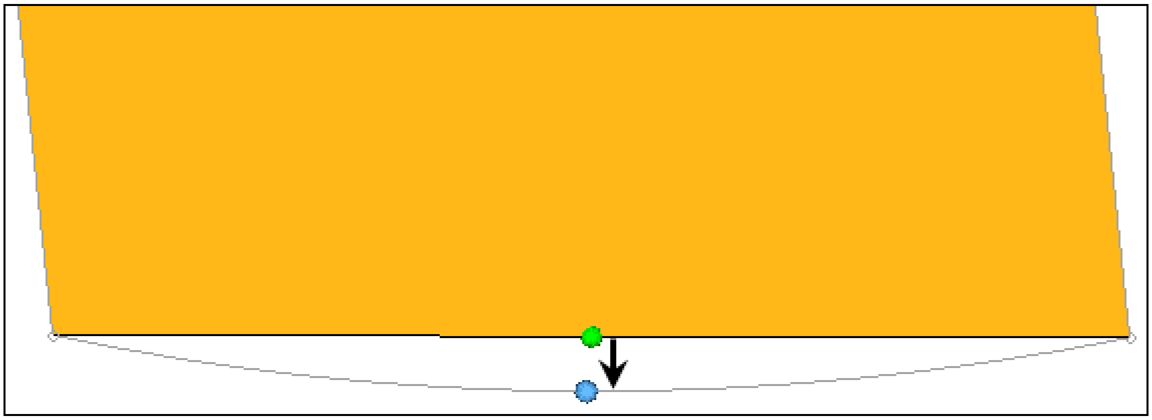


Figure 915: Chordal Deviation

Interior Angles

Maximum and minimum interior angles are evaluated independently for triangles and quadrilaterals.

Jacobian

Deviation of an element from its ideal or "perfect" shape, such as a triangle's deviation from equilateral.

The Jacobian value ranges from 0.0 to 1.0, where 1.0 represents a perfectly shaped element. The determinant of the Jacobian relates the local stretching of the parametric space which is required to fit it onto the global coordinate space.

HyperMesh evaluates the determinant of the Jacobian matrix at each of the element's integration points (also called Gauss points) or at the element's corner nodes, and reports the ratio between the smallest and the largest. In the case of Jacobian evaluation at the Gauss points, values of 0.7 and above are generally acceptable. You can select which method of evaluation to use (Gauss point or corner node) from the Check Element settings.

Length (min)

Minimum element lengths are calculated using one of two methods.

- The shortest edge of the element. This method is used for non-tetrahedral 3D elements.
- The shortest distance from a corner node to its opposing edge (or face, in the case of tetra elements); referred to as "minimal normalized height".

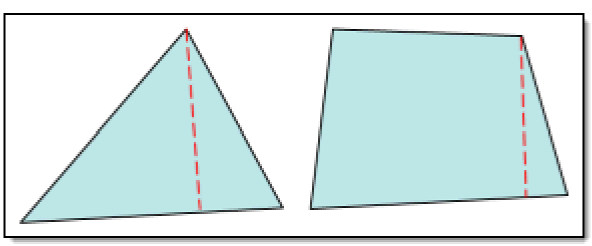


Figure 916: Length Check

You can choose which method to use in the Check Element settings.

> **Note:** This setting affects the calculation of the Aspect Ratio check.

Minimum Length / Size

Minimum element size is calculated using:

Shortest edge

Length of the shortest edge of each element is used.

Minimal normalized height

Is a more accurate, but more complex height.
For triangular elements, each corner node (i) HyperMeshHyperMesh calculates the closest (perpendicular) distance to the ray including the opposite leg of the triangle, h(i). MNH = min(hi) * 2/sqrt(3.0). The scaling factor 2/sqrt(3.0) ensures that for equilateral triangles, the MNH is the length of the minimum side.

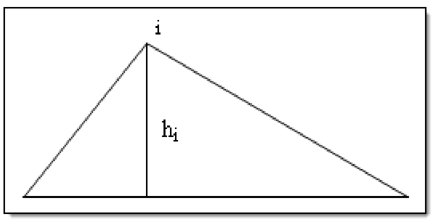


Figure 917: Minimal Normalized Height for Triangular Elements

For quadrilateral elements, each corner node, HyperMesh calculates the closest (perpendicular) distances to the rays containing the legs of the quadrilateral that do not include this node. The figure above depicts these lengths as red lines. Minimal normalized height is taken to be the minimum of all eight lines and the four edge lengths, thus, the minimum of 12 possible lengths.

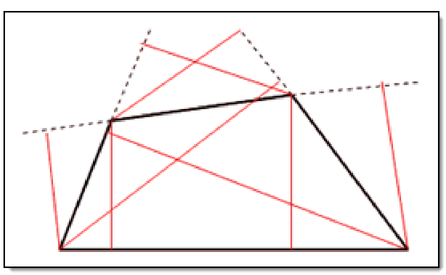


Figure 918: Minimal Normalized Height for Quadrilateral Elements

Minimal height

The same as minimal normalized height, but without a scaling factor.

Skew

Skew of triangular elements is calculated by finding the minimum angle between the vector from each node to the opposing mid-side, and the vector between the two adjacent mid-sides at each node of the element.

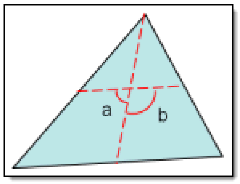


Figure 919: Skew of Triangular Elements

The minimum angle found is subtracted from ninety degrees and reported as the element's skew.

> **Note:** Skew for quads is part of the HyperMesh-Alt quality check.

Taper

Taper ratio for the quadrilateral element is defined by first finding the area of the triangle formed at each corner grid point.

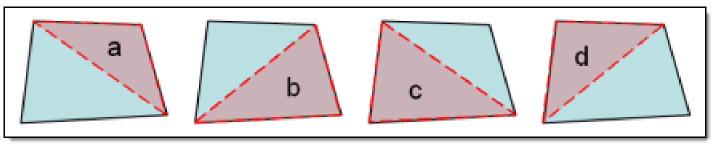


Figure 920: Taper for Quadrilateral Element

These areas are then compared to one half of the area of the quadrilateral.
HyperMesh then finds the smallest ratio of each of these triangular areas to ½ the quad element's total area (in the diagram above, "a" is smallest). The resulting value is subtracted from 1, and the result reported as the element taper. This means that as the taper approaches 0, the shape approaches a rectangle.

$$taper = 1 - \left(\frac{A_{tri}}{0.5 \times A_{quad}}\right)_{\min}$$

Triangles are assigned a value of 0, in order to prevent HyperMesh from mistaking them for highly-tapered quadrilaterals and reporting them as "failed".

Warpage

Amount by which an element, or in the case of solid elements, an element face, deviates from being planar. Since three points define a plane, this check only applies to quads. The quad is divided into two trias along its diagonal, and the angle between the trias' normals is measured. Warpage of up to five degrees is generally acceptable.

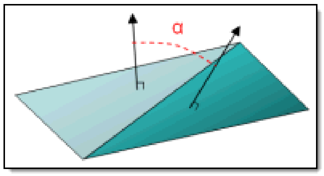


Figure 921: Warpage

3D Element Only Checks

Minimum Length / Size

Two methods are used to calculate the minimum element size.

Shortest edge

Length of the shortest edge of each element is used.

Minimal normalized height

More accurate, but more complex.

HyperMesh calculates the closest (perpendicular) distances to the planes formed by the opposite faces for each corner node.

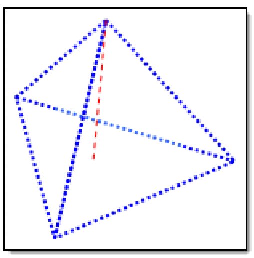


Figure 922:

The resulting minimum length/size is the minimum of all such measured distances.

Tetra Collapse

The height of the tetra element is measured from each of the four nodes to its opposite face, and then divided by the square root of the face's area.

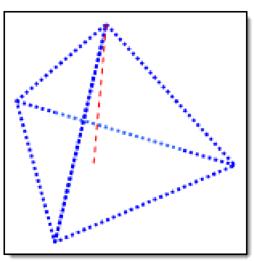


Figure 923:

The minimum of the four resulting values (one per node) is then normalized by dividing it by 1.24. As the tetra collapses, the value approaches 0.0, while a perfect tetra has a value of 1.0. Non-tetrahedral elements are given values of 1 so that HyperMesh will not mistake them for bad tetra elements.

Vol. Aspect Ratio

Tetrahedral elements are evaluated by finding the longest edge length and dividing it by the shortest height (measured from a node to its opposing face). Other 3D elements, such as hex elements, are evaluated based on the ratio of their longest edge to their shortest edge.

Volume Skew

Only applicable to tetrahedral elements; all others are assigned values of zero. Volume Skew is defined as 1-shape factor, so a skew of 0 is perfect and a skew of 1 is the worst possible value. The shape factor for a tetrahedral element is determined by dividing the element's volume by the volume of an ideal (equilateral) tetrahedron of the same circumradius. In the case of tetrahedral elements, the circumradius is the radius of a sphere passing through the four vertices of the tetrahedron.

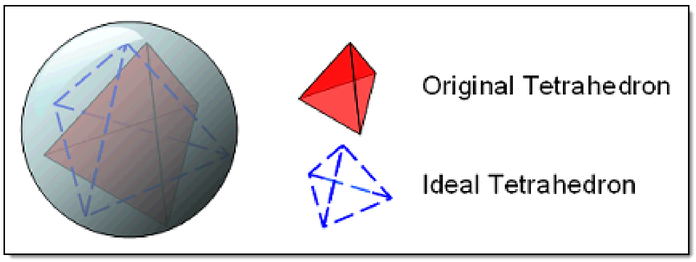


Figure 924:

HyperMesh-Alt

HyperMesh includes some alternate methods of calculating certain element types, which only apply to quads or rectangular faces of solids, and only include alternate checks for Aspect Ratio, Skew, Taper and Warpage.

> **Note:** Because these methods apply only to certain quality checks, in order to use them you must choose the **set individually** option in the Check Element settings.

Aspect Ratio

ratio1 = V1/H1
ratio2 = V2/H2
Skew value is larger of ratio1 or ratio2.

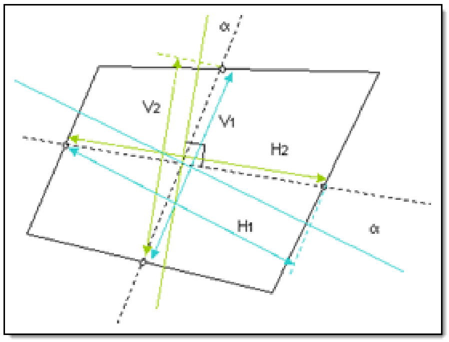


Figure 925: Aspect Ratio

Skew

First, HyperMesh constructs lines connecting the midpoints of each edge of the quad, dotted in the picture below. Next, HyperMesh constructs a third line, green in the picture below, perpendicular to one of the initial lines, then finds the angle between this third line and the remaining initial line – with which is it most likely not perpendicular, unless the quad is a perfect rectangle.

α is the skew (angle) value.

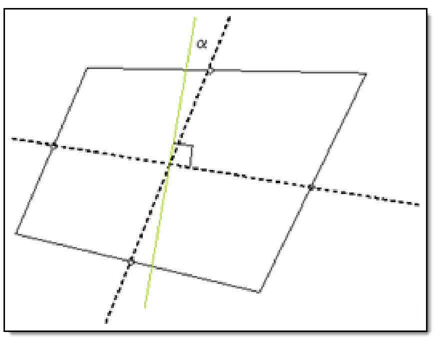


Figure 926: Skew

Taper

First, the quad's nodes are projected to plane defined by the orthonormal vectors U-V found as follows:

$Z = X \times Y$

$V = Z \times X$

$U = X$

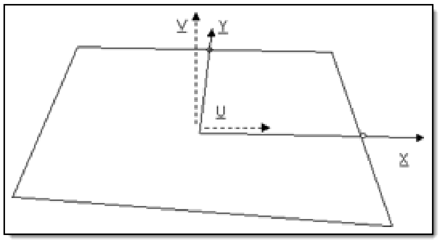


Figure 927:

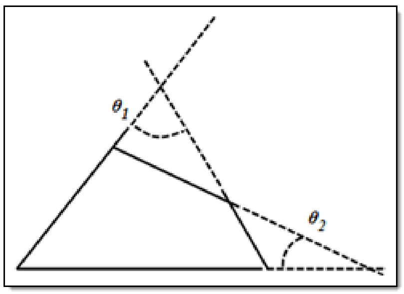


Figure 928:

In HyperMesh Taper angle is defined as: $\theta = \max\left(\frac{\theta_1}{2}; \frac{\theta_2}{2}\right)$.

The optimal value is 0°, and a generally acceptable limit is. <= 30°. The The ultimate limit, which the Taper angle cannot exceed is 45°.

Warpage

Only applies to quads or rectangular faces of solids.

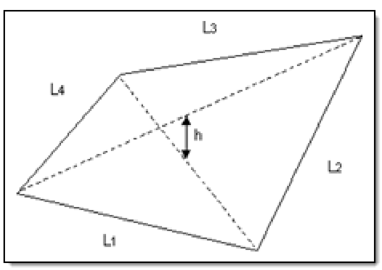


Figure 929:

Warpage = 100 * h / max { Li }, where h is the minimum distance between the diagonals.

OptiStruct

For the most part, OptiStruct uses the same checks as HyperMesh. However, OptiStruct uses its own method of calculating Aspect Ratio, and it does not support 3D element checks.

Aspect Ratio

Ratio between the minimum and maximum side lengths.

3D elements are evaluated by treating each face of the element as a 2D element, finding the aspect ratio of each face, and then returning the most extreme aspect ratio found.

Chordal Deviation

Chordal deviation of an element is calculated as the largest distance between the centers of element edges and the associated surface. 2nd order elements return the same chordal deviation as 1st order, when the corner nodes are used due to the expensive nature of the calculations.

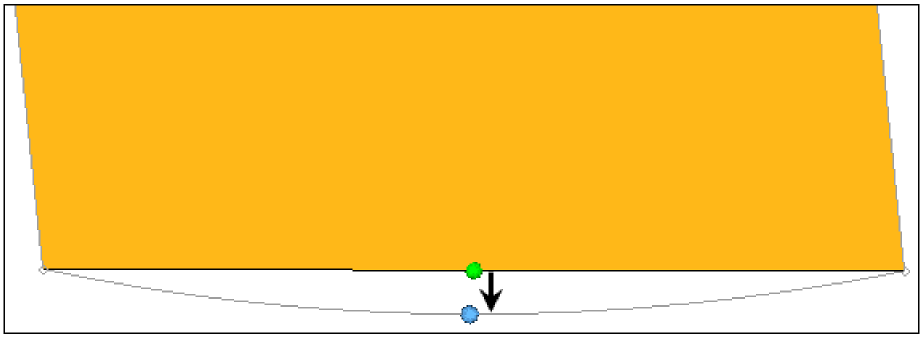


Figure 930: Chordal Deviation

Interior Angles

Maximum and minimum values are evaluated independently for triangles and quadrilaterals.

Jacobian

Deviation of an element from its ideal or "perfect" shape, such as a triangle's deviation from equilateral. The Jacobian value ranges from 0.0 to 1.0, where 1.0 represents a perfectly shaped element. The determinant of the Jacobian relates the local stretching of the parametric space which is required to fit it onto the global coordinate space.
HyperMesh evaluates the determinant of the Jacobian matrix at each of the element's integration points, also called Gauss points, or at the element's corner nodes, and reports the ratio between the smallest and the largest. In the case of Jacobian evaluation at the Gauss points, values of 0.7 and above are generally acceptable. You can select which method of evaluation to use, Gauss point or corner node, from the Check Element settings.

Length (min)

Minimum element lengths are calculated using one of two methods:

- The shortest edge of the element. This method is used for non-tetrahedral 3D elements.
- The shortest distance from a corner node to its opposing edge (or face, in the case of tetra elements); referred to as "minimal normalized height".

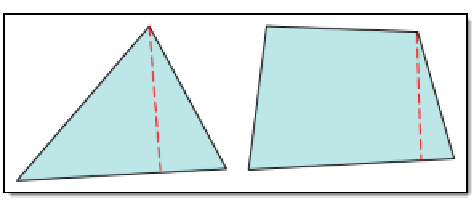


Figure 931: Length (Min)

Skew

Skew of triangular elements is calculated by finding the minimum angle between the vector from each node to the opposing mid-side, and the vector between the two adjacent mid-sides at each node of the element.

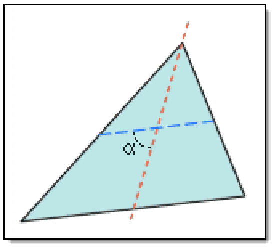


Figure 932: Skew of Triangular Element

The minimum angle found is subtracted from ninety degrees and reported as its skew.

Warpage

Amount by which an element, or in the case of solid elements, an element face, deviates from being planar. Since three points define a plane, this check only applies to quads. The quad is divided into two trias along its diagonal, and the angle between the trias' normals is measured. Warpage of up to five degrees is generally acceptable.

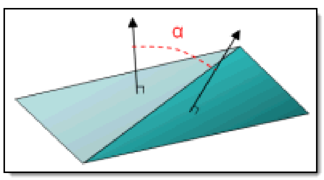


Figure 933: Warpage

Abaqus

Abaqus specific checks used to calculate element quality for 2D and 3D elements.

2D and 3D Element Checks

These checks apply to both types of elements, but when applied to 3D elements they are generally applied to each face of the element. The value of the worst face is reported as the 3D element's overall quality value.

Additional element checks not listed here are not part of the solver's normal set of checks, and therefore use HyperMesh check methods.

Aspect Ratio

Ratio of the longest edge of an element to its shortest edge.
When applied to 3D elements, the same method is used (longest edge divided by shortest edge) rather than evaluating each face individually and taking the worst face result.

Interior Angles

Maximum and minimum values are evaluated independently for triangles and quadrilaterals.

Jacobian

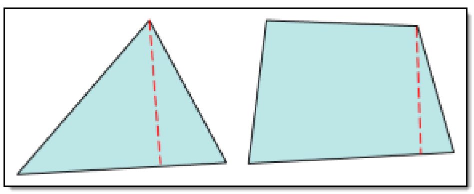
Deviation of an element from its ideal or "perfect" shape, such as a triangle's deviation from equilateral. The Jacobian value ranges from 0.0 to 1.0, where 1.0 represents a perfectly shaped element. The determinant of the Jacobian relates the local stretching of the parametric space which is required to fit it onto the global coordinate space.

HyperMesh evaluates the determinant of the Jacobian matrix at each of the element's integration points, also called Gauss points, or at the element's corner nodes, and reports the ratio between the smallest and the largest. In the case of Jacobian evaluation at the Gauss points, values of 0.7 and above are generally acceptable. You can select which method of evaluation to use (Gauss point or corner node) from the Check Element settings.

Length (min)

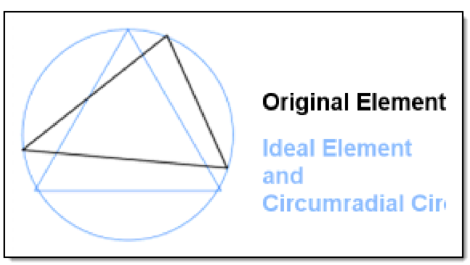
Minimum element lengths are calculated using one of two methods:

- The shortest edge of the element. This method is used for non-tetrahedral 3D elements.
- The shortest distance from a corner node to its opposing edge (or face, in the case of tetra elements); referred to as "minimal normalized height".



Skew (tria only)

Defined by shape factor. Abaqus determines triangular element shape factor by dividing the element's area by the area of an ideally shaped element. The ideally shaped element is defined as an equilateral triangle with the same circumradius—the radius of a circle that passes through the three vertices of the triangle—as the element.



$$SF = \frac{A_{actual}}{A_{ideal}}$$

Figure 934:

This shape factor converts to skew by subtracting it from 1. Thus, a perfect equilateral tria element has a skew of 0 and the worst tria has a value of 1.0.

Quadrilaterals are simply assigned a value of 0.

3D Element Only Checks

Volume Skew

Only applicable to tetrahedral elements; all others are assigned values of zero.

Volume Skew is defined as 1 minus the shape factor, so a skew of 0 is perfect and a skew of 1 is the worst possible value.

The shape factor for a tetrahedral element is determined by dividing the element's volume by the volume of an ideal (equilateral) tetrahedron of the same circumradius. In the case of tetrahedral elements, the circumradius is the radius of a sphere passing through the four vertices of the tetrahedron.

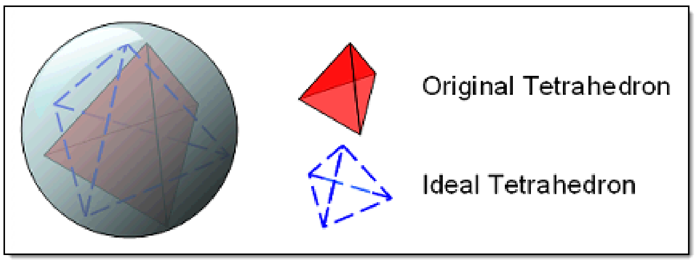


Figure 935: Volume Skew

ANSYS

ANSYS specific checks used to calculate element quality for 2D and 3D elements.

2D and 3D Element Checks

These checks apply to both types of elements, but when applied to 3D elements they are generally applied to each face of the element. The value of the worst face is reported as the 3D element's overall quality value.

Additional element checks not listed here are not part of the solver's normal set of checks, and therefore use HyperMesh check methods.

Aspect Ratio (tria)

For tria elements, a line is drawn from one node to the midpoint of the opposite edge. Next, another line is drawn between the midpoints of the remaining two sides. These lines are typically not perpendicular to each other or to any of the element edges, but provide four points (three midpoints plus the vertex).

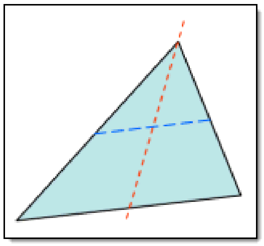


Figure 936:

Then, a rectangle is created for each of these two lines, such that one line perpendicularly meets the midpoints of two opposing edges of the rectangle, and the remaining edges of the rectangle pass through the end points of the remaining line. This results in two rectangles, one perpendicular to each of the two lines.

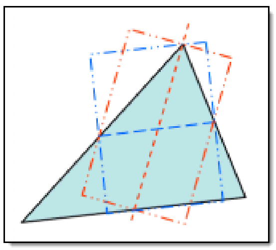


Figure 937:

Third, this process is repeated for each of the remaining two nodes of the tria element, resulting in the construction of four additional rectangles (six in total).

Finally, each rectangle is examined to find the ratio of its longest side to its shortest side. Of these six values—one for each rectangle—the most extreme value is then divided by the square root of three to produce the tria aspect ratio.

The best aspect ratio (an equilateral tria) is 1. Higher numbers indicate greater deviation from equilateral.

Aspect Ratio (quad)

If the element is not flat, it's projected to a plane which is based on the average of the element's corner normals. All subsequent calculations are based on this projected element rather than the original (curved) element.

Next, two lines are created which bisect opposite edges of the element. These lines are typically not perpendicular to each other or to any of the element edges, but they provide four midpoints.

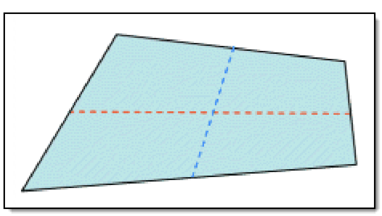


Figure 938:

Third, a rectangle is created for each line, such that the line perpendicularly bisects two opposing edges of the created rectangle, and the remaining two edges of the rectangle pass through the remaining line's endpoints. This creates two rectangles—one perpendicular to each line.

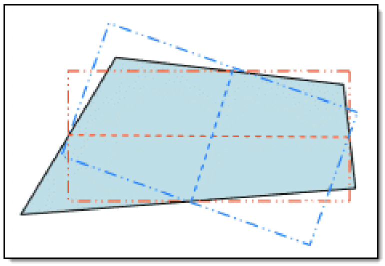


Figure 939:

Finally, the rectangles are compared to find the one with the greatest length ratio of longest side to shortest side. This value is reported as the quad's aspect ratio. A value of one indicates a perfectly equilateral element, while higher numbers indicate increasingly greater deviation from equilateral.

Interior Angles

Maximum and minimum values are evaluated independently for triangles and quadrilaterals.

Jacobian

Deviation of an element from its ideal or "perfect" shape, such as a triangle's deviation from equilateral. The Jacobian value ranges from 0.0 to 1.0, where 1.0 represents a perfectly shaped element. The determinant of the Jacobian relates the local stretching of the parametric space which is required to fit it onto the global coordinate space.
HyperMesh evaluates the determinant of the Jacobian matrix at each of the element's integration points, also called Gauss points, or at the element's corner nodes, and reports the ratio between the smallest and the largest. In the case of Jacobian evaluation at the Gauss points, values of 0.7 and above are generally acceptable. You can select which method of evaluation to use (Gauss point or corner node) from the Check Element settings.

Length (min)

Minimum element lengths are calculated using one of two methods:

- The shortest edge of the element. This method is used for non-tetrahedral 3D elements.
- The shortest distance from a corner node to its opposing edge (or face, in the case of tetra elements); referred to as "minimal normalized height".

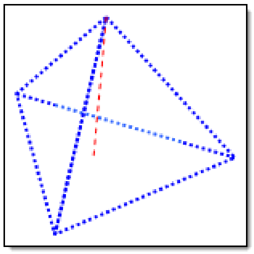


Figure 940:

Angle Deviation (Skew)

Only applicable to quadrilateral elements, and relies upon the angles between adjacent legs at each corner node (that is, the interior angles at each corner). Each angle is compared to a base of 90 degrees, and the one with the largest deviation from 90 is reported as the angle deviation. Triangular elements are given a value of zero.

Warping Factor

Only applicable to quadrilateral elements as well as the quadrilateral faces of 3D bricks, wedges, and pyramids.

Calculated by creating a normal from the vector product of the element's two diagonals. Next, the element's area is projected to a plane through the average normal. Finally, the difference in height is measured between each node of the original element and its corresponding node on the projection. For flat elements, this is always zero, but for warped elements one or more nodes will deviate from the plane. The greater the difference, the more warped the element is.

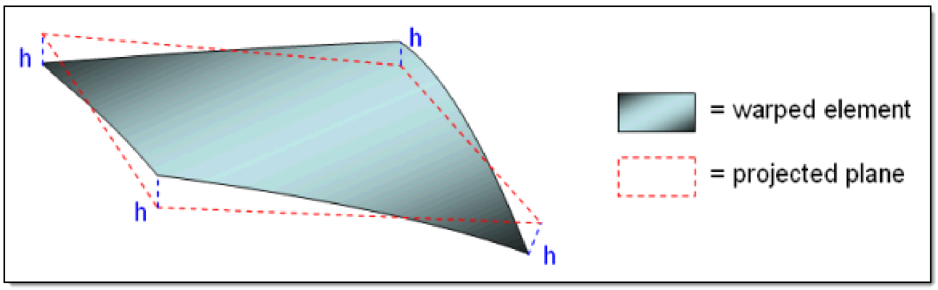


Figure 941:

The warping factor is calculated as the edge height difference divided by the square root of the projected area.

3D Element Only Checks

ANSYS does not use any exclusively 3D checks within HyperMesh, but HyperMesh does use its own when ANSYS is set as the solver. For details on 3D checks, refer to HyperMesh.

I-deas

I-deas specific checks used to calculate element quality for 2D and 3D elements.

Additional element checks not listed here are not part of the solver's normal set of checks, and therefore use HyperMesh check methods.

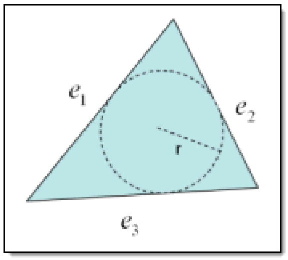
2D and 3D Element Checks

These checks apply to both types of elements, but when applied to 3D elements they are generally applied to each face of the element. The value of the worst face is reported as the 3D element's overall quality value.

Stretch (Aspect Ratio)

Stretch is evaluated differently depending on whether the element is triangular or quadrilateral:

- For trias, the radius of the largest circle that fits within the element is divided by the longest edge, then multiplied by the square root of 12.



$$stretch = \sqrt{12} \times \frac{r}{e_{max}}$$

Figure 942: Stretch for Trias

- For quads, the minimum edge length is divided by the maximum diagonal length. The result is multiplied by the square root of 2.

> **Note:** The inverse of stretch displays on-screen in HyperMesh as the aspect.

Chordal Deviation

Largest distance between the centers of element edges and the associated surface. Second order elements return the same chordal deviation as first order, when the corner nodes are used due to the expensive nature of the calculations.

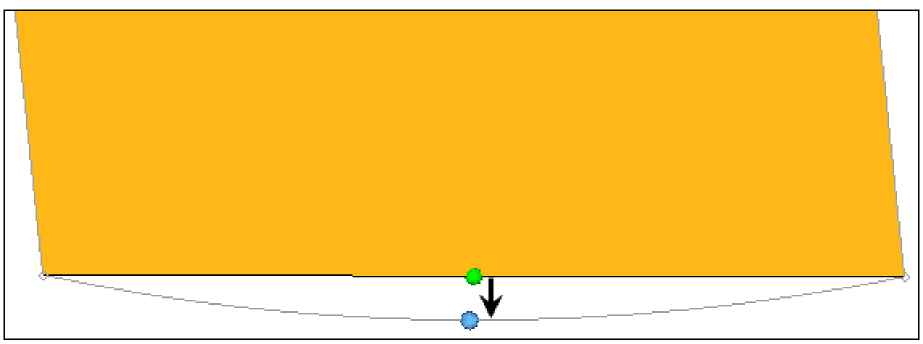


Figure 943: Chordal Deviation

Jacobian

Deviation of an element from its ideal or "perfect" shape, such as a triangle's deviation from equilateral. The Jacobian value ranges from 0.0 to 1.0, where 1.0 represents a perfectly shaped element. The determinant of the Jacobian relates the local stretching of the parametric space which is required to fit it onto the global coordinate space.

HyperMesh evaluates the determinant of the Jacobian matrix at each of the element's integration points, also called Gauss points, or at the element's corner nodes, and reports the ratio between the smallest and the largest. In the case of Jacobian evaluation at the Gauss points, values of 0.7 and above are generally acceptable. You can select which method of evaluation to use (Gauss point or corner node) from the Check Element settings.

Length (min)

Minimum element lengths are calculated using one of two methods:

- The shortest edge of the element. This method is used for non-tetrahedral 3D elements.
- The shortest distance from a corner node to its opposing edge (or face, in the case of tetra elements); referred to as "minimal normalized height".

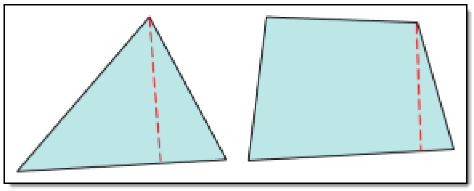


Figure 944: Length (min)

Skew

Deviation of an element’s corners from 90 degrees (for quads) or 60 degrees (for trias).
The check calculates skew by finding:

$= \sum_{i=1}^{4} |90 - \alpha_i|$ for quadrilaterals

$= \sum_{i=1}^{3} |90 - \alpha_i|$ for triangular elements

Where alpha is the angle of each corner. An ideal/equilateral element has a skew of zero, as none of its corners deviate from the target (90 or 60 degrees).

Taper

Taper ratio for the quadrilateral element is defined by first finding the area of the triangle formed at each corner grid point.

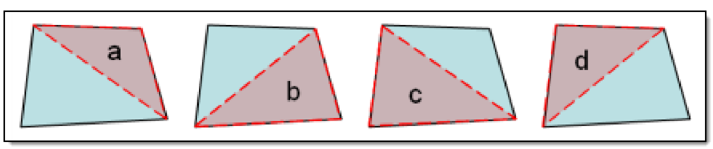


Figure 945: Taper

These areas are then compared to one half of the area of the quadrilateral.
HyperMesh then finds the smallest ratio of each of these triangular areas to ½ the quad element’s total area. In the diagram above, "a" is smallest. The resulting value is subtracted from 1, and the result reported as the element taper. This means that as the taper approaches 0, the shape approaches a rectangle.

$$taper = 1 - \left(\frac{A_{tri}}{0.5 \times A_{quad}} \right)_{\min}$$

Triangles are assigned a value of 0, in order to prevent HyperMesh from mistaking them for highly-tapered quadrilaterals and reporting them as "failed".

Warpage

The amount by which an element, or in the case of solid elements, an element face, deviates from being planar. Since three points define a plane, this check only applies to quads. The quad is divided into two trias along its diagonal, and the angle between the trias' normals is measured.

3D Element Only Checks

Stretch (volume aspect ratio)

Stretch is evaluated differently depending on whether the element is a tetrahedron, Wedge, Brick, or Pyramid.

Tetras

The radius of the largest sphere that fits within the element is divided by the longest edge. This value is then multiplied by the square root of 24.

Wedges

Each face is evaluated for its 2D stretch, and the worst value is reported. This means that the value reported for vol AR should always be the same as that reported for aspect.

Bricks

The minimum edge length is divided by the maximum diagonal length. The result is multiplied by the square root of 3.

Pyramids

No check is defined, so HyperMesh performs its standard check in which each face is evaluated as a 2D object and the worst result reported.

Medina

Medina specific checks used to calculate element quality for 2D and 3D elements.

Additional element checks not listed here are not part of the solver's normal set of checks, and therefore use HyperMesh check methods.

2D and 3D Element Checks

These checks apply to both types of elements, but when applied to 3D elements they are generally applied to each face of the element. The value of the worst face is reported as the 3D element's overall quality value.

Aspect Ratio (Edge Ratio)

Edge Ratio is calculated as the ratio between an element's shortest edge and its longest edge; For the sake of consistency, HyperMesh inverts this result, effectively making it the ratio of longest to shortest, and reports the result as the element's aspect ratio.

Interior Angles

Maximum and minimum values are evaluated independently for triangles and quadrilaterals.

Jacobian

Deviation of an element from its ideal or "perfect" shape, such as a triangle's deviation from equilateral. The Jacobian value ranges from 0.0 to 1.0, where 1.0 represents a perfectly shaped element. The determinant of the Jacobian relates the local stretching of the parametric space which is required to fit it onto the global coordinate space.

HyperMesh evaluates the determinant of the Jacobian matrix at each of the element’s integration points, also called Gauss points, or at the element’s corner nodes, and reports the ratio between the smallest and the largest. In the case of Jacobian evaluation at the Gauss points, values of 0.7 and above are generally acceptable. You can select which method of evaluation to use (Gauss point or corner node) from the Check Element settings.

Length (min)

Minimum element lengths are calculated using one of two methods:

- The shortest edge of the element. This method is used for non-tetrahedral 3D elements.
- The shortest distance from a corner node to its opposing edge (or face, in the case of tetra elements); referred to as "minimal normalized height".

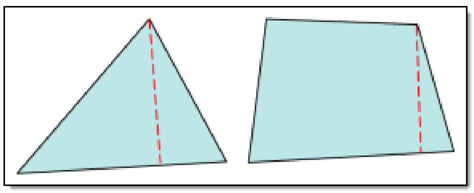


Figure 946: Length (min)

Maximum Angle

Largest angle between adjacent edges of the element is reported.

Minimum Angle

Smallest angle between adjacent edges of the element is reported.

Skew

Element’s interior corner angles are compared to 90 degrees (for quads) or 60 degrees (for trias). The absolute values of these deviations are summed and reported.

Taper

Quadrilateral elements are split into two triangles.

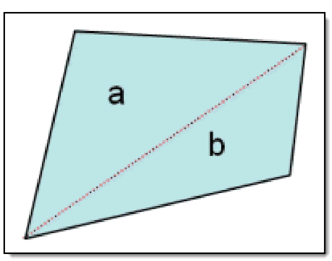


Figure 947: Taper

The area of the smaller of the two triangles is compared to the total area of the quadrilateral. In Figure 947, $taper = \frac{A_b}{A_{quad}}$.

> **Note:** To improve consistency with other taper checks, HyperMesh displays a value of 0.5 minus this value so that 0 implies no taper. However, this is not completely consistent with other taper checks, because in this case taper ranges from 0 (no taper) to 0.5 (full taper), whereas HyperMesh's own taper check reports a 1.0 for full taper.

Warpage

Elements with more than three nodes are split into triangles. The largest angle between the normals of triangle pairs is reported as the warpage.

3D Element Only Checks

Medina does not use any 3D specific checks. HyperMesh uses its own checks instead.

Moldflow

Moldflow specific checks used to calculate element quality for 2D and 3D elements.

Additional element checks not listed here are not part of the solver's normal set of checks, and therefore use HyperMesh check methods.

2D and 3D Element Checks

These checks apply to both types of elements, but when applied to 3D elements they are generally applied to each face of the element. The value of the worst face is reported as the 3D element's overall quality value.

Aspect Ratio

Only applied to triangles, with quadrilaterals given a value of:

$$\frac{2.0}{\sqrt{3}}$$

This is the same value obtained from an equilateral triangle, and is assigned to quads to prevent HyperMesh from misinterpreting a quad as a badly formed triangular element.
MoldFlow calculates a triangle's aspect ratio by squaring the longest edge of the triangle, and dividing the result by twice the triangle's area. 1.0 denotes a perfect equilateral triangle.
When applied to 3D elements, the aspect ratio is the ratio between the longest and shortest edges of the tetrahedral element.

3D Element Only Checks

Vol. Aspect Ratio

Finds the perpendicular height h of each node, and then dividing the longest edge length L by the shortest height h and multiplying by the square root of 1.5:

$$\frac{\sqrt{1.5} \times L}{h}$$

This results in an equilateral tetrahedron having a volume aspect ratio of 1.5. Non-tetrahedral elements are assigned a value of 1.0.

Nastran

Nastran specific checks used to calculate element quality for 2D and 3D elements.

Additional element checks not listed here are not part of the solver's normal set of checks, and therefore use HyperMesh check methods.

2D and 3D Element Checks

Interior Angles

Maximum and minimum values are evaluated independently for triangles and quadrilaterals.

Jacobian

Deviation of an element from its ideal or "perfect" shape, such as a triangle's deviation from equilateral. The Jacobian value ranges from 0.0 to 1.0, where 1.0 represents a perfectly shaped element. The determinant of the Jacobian relates the local stretching of the parametric space which is required to fit it onto the global coordinate space.
HyperMesh evaluates the determinant of the Jacobian matrix at each of the element's integration points, also called Gauss points, or at the element's corner nodes, and reports the ratio between the smallest and the largest. In the case of Jacobian evaluation at the Gauss points, values of 0.7 and above are generally acceptable. You can select which method of evaluation to use (Gauss point or corner node) from the Check Element settings.

Skew

Nastran creates lines between the midpoints of opposite sides of the element, then measures the angles between these lines. The angle with the greatest deviation from the ideal value is used to determine skew.

Taper

Nastran finds the taper of quadrilateral elements by treating each node as the corner of a triangle, using one of the quad's diagonals as the triangle's third leg. The areas of each of these four "virtual" triangles are compared to one half of the total area of the quadrilateral element to produce a ratio; the largest of these ratios is then compared to the tolerance value. A value of 1.0 is a perfect quadrilateral, and higher numbers denote greater taper.
However, for the sake of consistency within HyperMesh, an equivalent taper is reported instead. This means that the smallest area ratio found (instead of the largest ratio) is subtracted from 1, so that 0 represents a perfect quadrilateral element instead of 1.0, and greater deviation from 0 indicates greater taper. Triangle elements are simply assigned a value of 0 to prevent HyperMesh from incorrectly identifying them as failed (highly-tapered) quads.

Warpage

First, Nastran constructs a plane based on the mean of the quad's four points. This means that the corner points of a warped quad are alternately H units above and below the constructed plane. This value is then used along with the length of the element's diagonals in the following equation:

$$WC = 2H \big/ (D1 + D2)$$

Where WC is the Warping Coefficient, H is the "height" or distance of the nodes from the constructed plane, and D1 and D2 are the lengths of the diagonals. Thus, a perfect quad has a WC of zero.

3D Element Only Checks

Vol. Aspect Ratio

Nastran evaluates Tetrahedral elements by finding the longest edge length and dividing it by the shortest height, measured from a node to its opposing face. Other 3D elements, such as hex elements, are evaluated based on the ratio of their longest edge to their shortest edge.

Warpage

Nastran evaluates warpage on solid element faces by dividing the quad face into two trias along its diagonal, and measuring the cosine of the angle between the trias' normals. This value will be 1.0 for a face where all nodes lie on the same plane.

Patran

Patran specific checks used to calculate element quality for 2D and 3D elements.

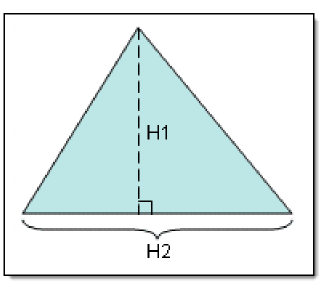
Additional element checks not listed here are not part of the solver's normal set of checks, and therefore use HyperMesh check methods.

2D and 3D Element Checks

These checks apply to both types of elements, but when applied to 3D elements they are generally applied to each face of the element. The value of the worst face is reported as the 3D element's overall quality value.

Aspect Ratio (triangle)

The length of a side is divided by the height of the triangle from that side to its opposite node, then multiplied by ½ of the square root of 3. In a perfect equilateral triangle, this formula produces a value of 1. The process is performed for each of the three sides, and the largest value of the three is reported as the aspect ratio.



$$Aspect = \frac{\sqrt{3}\,h_2}{2h_1}$$

Figure 948: Aspect Ratio for Triangles

Aspect Ratio (quads)

If the element is not flat, it is projected to a plane which is based on the average of the element's corner normals. All subsequent calculations are based on this projected element rather than the original (curved) element.

Next, two lines are created which bisect opposite edges of the element. These lines are typically not perpendicular to each other or to any of the element edges, but they provide four midpoints.

Third, a rectangle is created for each line, such that the line perpendicularly bisects two opposing edges of the created rectangle, and the remaining two edges of the rectangle pass through the remaining line's endpoints. This creates two rectangles—one perpendicular to each line.

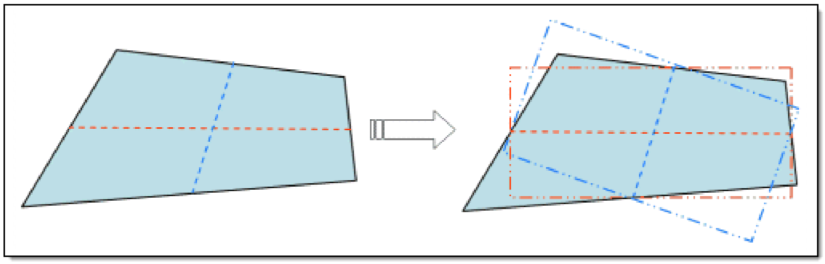


Figure 949: Aspect Ratio for Quads

Finally, the rectangles are compared to find the one with the greatest length ratio of longest side to shortest side. This value is reported as the quad's aspect ratio. A value of 1 indicates a perfectly equilateral element, while higher numbers indicate increasingly greater deviation from equilateral.

Interior Angles

Maximum and minimum values are evaluated independently for triangles and quadrilaterals.

Jacobian

Deviation of an element from its ideal or "perfect" shape, such as a triangle's deviation from equilateral. The Jacobian value ranges from 0.0 to 1.0, where 1.0 represents a perfectly shaped element. The determinant of the Jacobian relates the local stretching of the parametric space which is required to fit it onto the global coordinate space.

HyperMesh evaluates the determinant of the Jacobian matrix at each of the element's integration points, also called Gauss points, or at the element's corner nodes, and reports the ratio between the smallest and the largest. In the case of Jacobian evaluation at the Gauss points, values of 0.7 and above are generally acceptable. You can select which method of evaluation to use (Gauss point or corner node) from the Check Element settings.

Length (min)

Minimum element lengths are calculated using one of two methods:

- The shortest edge of the element. This method is used for non-tetrahedral 3D elements.
- The shortest distance from a corner node to its opposing edge (or face, in the case of tetra elements); referred to as "minimal normalized height".

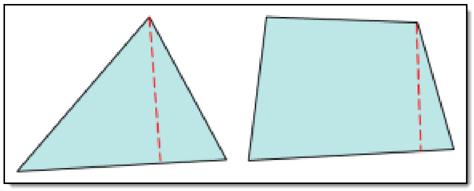


Figure 950: Length (min)

Skew (triangle)

Patran evaluates triangular skew by constructing a line from one of the triangle's nodes to the midpoint of its opposite side, and another line connecting the midpoints of the remaining two sides.

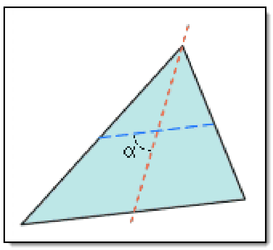


Figure 951: Skew for Triangles

An angle between these created lines is compared to 90 degrees to find its deviation from square. This process is then repeated for each of the remaining two nodes, and the largest of the three computed angle deviations is reported as the element's skew.

Skew (Quad)

The skew test begins by bisecting the four element edges. This creates an origin at the vector average of the four corners, with the x-axis extending from the origin to the bisector on edge 2. Next, finding the cross-product of the x-axis and the vector that stretches from the origin to the midpoint of edge 3 defines the z-axis. With the x and z axes defined, their cross-product defines the y-axis.

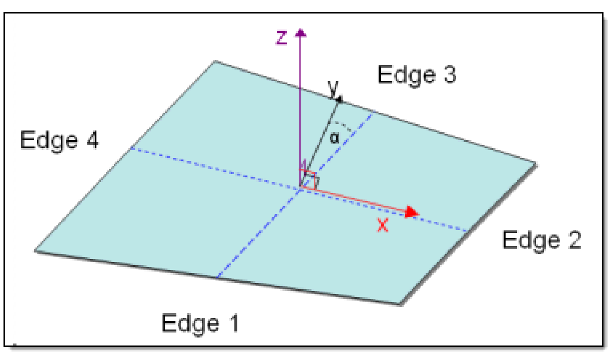


Figure 952: Skew for Quads

Finally, subtracting the angle α (located between the y axis and the line bisecting edges 1 and 3) from 90 degrees reveals the element skew.

Taper

Patran calculates taper by first averaging the corner nodes to find the element center, and creating lines between this center and the corner nodes to split the element into four triangles.

$$taper = \frac{4\alpha_{smallest}}{\alpha 1 + \alpha 2 + \alpha 3 + \alpha 4}$$

The taper calculation is simply the smallest triangle's area divided by the average of all the triangle areas—or, put another way, the taper is quadruple the area of the smallest triangle, divided by the sum of the areas of all four triangles:

$$taper = \frac{4\alpha_{smallest}}{\alpha 1 + \alpha 2 + \alpha 3 + \alpha 4}$$

> **Note:** For the sake of display compatibility, HyperMesh reports an equivalent value for Taper. Taper is determined as above, but is then subtracted from 1 to produce a number between zero and one. Thus, as the element taper decreases, the reported value approaches zero (a perfect square). Triangles are assigned a value of zero to prevent them from showing up as failed quads.

Warpage

The warpage test bisects the element edges, creating a point at the vector average of the element corners. This point serves as the base node for a plane, with the plane's x-axis extending from the base node to the bisector on edge 2 of the element. The plane normal (z-axis) is in the direction of the cross-product of this x-axis and the vector from the origin to the bisector of edge 3. Each corner of the quad is then the same distance, h, from the plane. Next, Patran measures the length of each half-edge, and calculates the arcsine of the ratio of h to the shortest half-edge length (L):

$$\Theta = \sin^{-1}\frac{h}{L}$$

3D Element Only Checks

Vol. Aspect Ratio (Tetrahedron)

Patran finds the aspect ratio of Tetra elements by finding the ratio between a vertex height and ½ the area of the opposing face. This process is repeated for each vertex, and the largest ratio found.

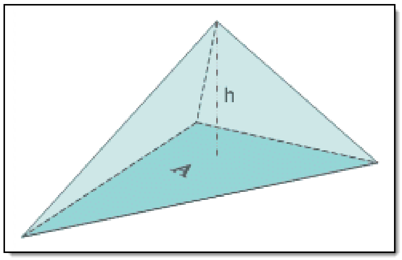


Figure 953: Vol. Aspect Ratio for Tetrahedrons

Next, Patran multiplies the largest ratio found by 0.805927, the corresponding ratio of an equilateral tetrahedron. The result is reported as the element's aspect ratio, with a value of 1 representing a perfect equilateral tetrahedron.

Vol. Aspect Ratio (pyramid)

Ratio of the element's longest edge length to its shortest edge length.

Vol. Aspect Ratio (wedge)

This test begins by averaging the triangular faces of the element to create a triangular mid-surface. Next, it finds the aspect ratio of the mid-surface, as for a tria element. Then it compares the average height (h1) of the wedge element to the mid-surface's maximum edge length (h2).

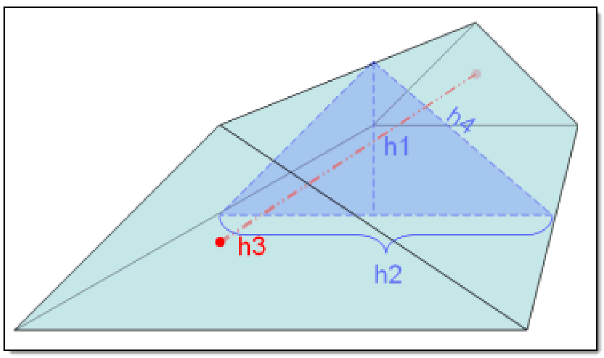


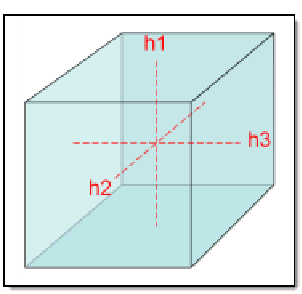
Figure 954: Vol. Aspect Ratio for Wedges

If the wedge height h1 exceeds the edge length h2, the wedge's aspect ratio equals the mid-surface aspect ratio multiplied by h2, then divided by the average distance between the triangular faces (h3).
If the wedge height h1 is less than the edge length h2, the wedge aspect ratio equals either the mid-surface aspect ratio, or the maximum edge length h2 divided by the average distance between the triangular faces (h3), whichever is greater.

$$Aspect\ Ratio = \frac{h_4}{h_3}\frac{\sqrt{3}\,h_2}{2h_1}$$

Vol. Aspect Ratio (hexahedron)

Each face of the hex element is treated as a warped quadrilateral, and its center point found. The volume aspect ratio is simply the ratio of the largest distance h between the center points of any two opposing faces, to the smallest such distance.



$$Aspect\ Ratio = \frac{\max(h_1, h_2, h_3)}{\min(h_1, h_2, h_3)}$$

Figure 955: Vol. Aspect Ratio for Hexahedrons

Quality Index Calculations

The Quality Index value is a function of twelve criteria with user-defined weight factors. Each criterion has five rating levels.

HyperMesh assigns a penalty value to each element according to its rating for individual criteria. The elements that fail a criterion are assigned a penalty of 1.0 to 10.0 as a linear function of how far the element is from satisfying the criterion. The elements that pass a criterion are assigned a penalty value of 0.0 to 1.0 for that criterion. The quality index (Q.I.) is a function of individual criteria penalty values. Each element is assigned the corresponding element Q.I. color.

element Q.I.
: (weighted average of penalties that pass) + (weighted sum of penalties that fail)

criteria Q.I.
: (weighted average of penalties of elements that pass) + (weighted sum of penalties of elements that fail)

compound Q.I
: (weighted average of criteria Q.I. that pass) + (weighted sum of criteria Q.I. that fail)

All of this means that higher compound Q.I. values indicate worse quality.

Each criterion has five levels. The elements are assigned a penalty depending on where they fall in these levels.

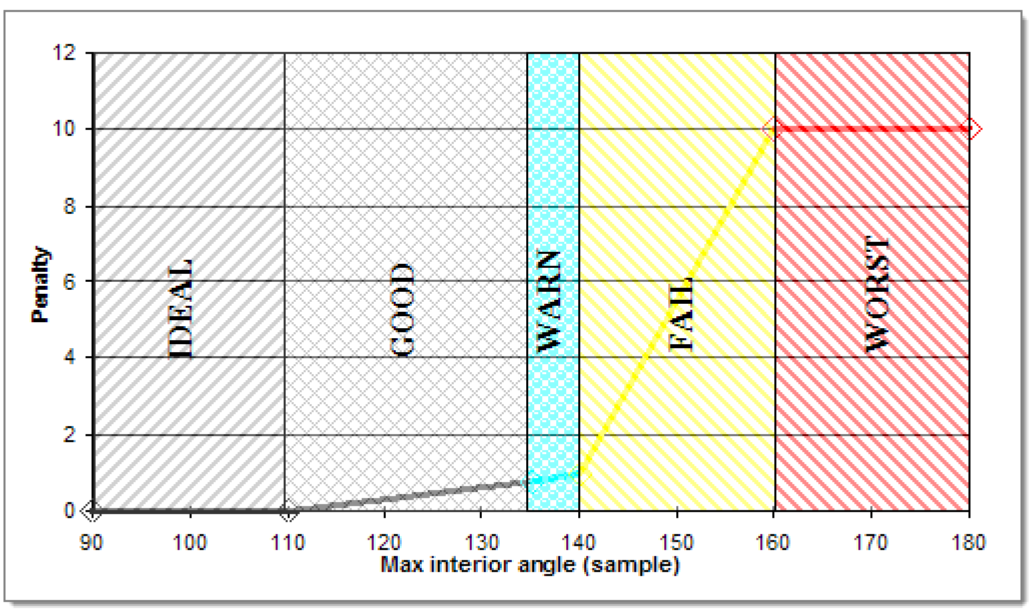


Figure 956: Criterion Levels

Ideal

The absolute best/ideal value that an element can achieve. For example, an ideal element would have an aspect ratio of 1.0, warpage of 0.0, jacobian of 1.0, and so on. Some criteria may not have an ideal, for example the ideal minimum element size is the same as average element size. Similarly, for simulations that require all triangular mesh, "% of trias" is not applicable. Thus the ideal "% of trias" depends on the analysis type, and should be set by you. Elements that fall in this level are drawn in their default color, not highlighted. Ideal elements have no penalty assigned to them.

Good

This level is slightly worse than ideal, but is still considered good for the required analysis. All elements whose criteria are equal to or better than this level are considered good and no penalty is assigned to them. You set all the good level thresholds. Elements that fall in this level, between good and warn, are drawn in their default color, not highlighted. The elements that fall between good and warn are assigned a penalty between 0-0.79.

Warn

An intermediate level between good and fail. This level is used to highlight the elements that have not failed the criteria, but are close to it. HyperMesh sets these values at 80 percent between good and fail levels. The elements that are in this level, that is, falling between warn and fail, are drawn in cyan by default and are assigned a penalty value between 0.8 – 0.99.

Fail

Determines the elements that are considered to be unacceptable for analysis, thus failed. It is recommended that you fix these elements before performing the analysis. You can specify the fail levels. All the elements that fail are given a penalty greater than 1.0. The penalty value is calculated depending on the severity of the failure. The elements that have failed, between fail and worst, are drawn in yellow by default and are given a penalty value between 1.0-10.0. Therefore, elements that passed all criteria have a penalty less than 1.0.

Worst

Highlights elements that failed the criteria by a large margin, and which require immediate attention. The worst levels are set by HyperMesh as a factor of good and fail values. The elements that fall in and beyond this worst level are drawn in red and given a flat penalty value of 10.0.

In addition to these levels, you can also turn on/off individual criteria according to your analysis requirements. You can also set different weights for individual criteria. For example, if jacobian is relatively more important than warpage, you can choose to set jacobian comp weight to 2.0. The Comp QI calculated will then give jacobian twice the weight as the remaining criteria.

Element Quality View

The Element Quality view allows you to investigate each specific element criteria, view a breakdown of all failed and worst elements, resolve all criteria violations at one time, and evaluate the over all quality of a mesh.

Element Quality View is a permanent visualization mode that HyperMesh displays in the upper left-hand corner of the graphics area when you select **By Element Quality** on the Visualization toolbar.

Multiple Criteria Legend

When you select Element Quality View, HyperMesh displays the Multiple Criteria legend by default.

In this legend, you can:

- Review the different 2D element criteria's.
- Adjust the initial threshold values assigned to each 2D element criteria.
- Select specific criteria's to investigate further.

The Element Quality View tool bases the initial threshold values on the ideal, good, warn, fail (default), and worst values that are defined in the current 2D element criteria settings. By default, this tool bases the initial threshold values on the fail column. You can directly edit these values from the Element Quality View or you can edit them in the Criteria File Editor.

> **Note:** Any changes that you make in the Element Quality View will impact all of the other Element Quality View settings. The changes that you make to the threshold values in the Element Quality View will not effect the values in the criteria file, but any changes that you make to the criteria file will effect the Element Quality View.

When you click a criteria in the Multiple Criteria legend, a single criteria specific legend appears to the right of the threshold values. The elements are color coded according to how well they adhere to the quality requirements in the graphics area.

Single Criteria Legend

Each 2D element criteria, in the Multiple Criteria legend, has its own single criteria legend. The single criteria legends consists of a color coded sliding scale, which you can use to evaluate the elements in the graphics area and resolve all criteria violations.

The colors exhibited in the sliding scale reflect the quality of each element in the graphics area. The elements that are of the best quality will always display in blue, whereas, the elements that are of the worst quality will always display in red. The Element Quality View tool determines the quality of each element using the 2D element criteria that you defined in the Multiple Criteria legend.

The Element Quality View tool always lists the values in the sliding scale from low to high, with the lowest value always being at the bottom of the scale and the highest value always being at the top. This tool always defines the first and last values in the legend with the minimum and maximum values. You cannot edit the minimum and maximum values.

The second and second to last values are the initial legend range coverage, based on the good to worst, warn to worst (default), fail to worst, and min to max values. These values are taken from the criteria file, and can be edited. The Element Quality View tool interpolates the values in between these.

Each single criteria legend contains a slider that is located, by default, at the exact, current threshold position. If the initial legend range coverage does not provide the threshold value set defined in the settings, then the slider will be positioned to the closest available value. To view a breakdown of all of the failed and worse elements, move the slider up and down.

Element Quality View Options and Settings

Overview of the options and settings used to control the Element Quality view.

Multiple Criteria Context Menu

To access the Multiple Criteria context menu, right-click on the Multiple Criteria legend.

No Result As

Change how elements that fail all of the 2D element criteria appear in the graphics area. By default, these elements are shaded in gray. To make these elements invisible, click transparent.

Set Threshold Values

Set the threshold values to one of the following levels: worst, fail (default), warn, good, and ideal.

> **Note:** When you select a different level to set the threshold values to, the values will reset to the values that are read from the current criteria file settings.

Edit Criteria

Opens the Criteria File Editor, from which you can edit the current 2D element criteria in the criteria file.

> **Note:** As long as the Criteria File Editor is open, you cannot modify the legends in the Element Quality View.

Configure Quality View

Opens the **Element Quality View Configuration** dialog, from which you can:

- Change the color assigned to each criteria.
- Select which element criteria you would like to display in the Multiple Criteria legend.
- Rearrange the order of the element criteria in the Multiple Criteria legend

Single Criteria Context Menu

To access the Single Criteria context menu, right-click on a single criteria legend.

Close XXX Legend

Closes the single criteria legend that is currently open, and activates the Multiple Criteria legend.

Set Legend Range

Set the legend range of the threshold values by selecting one of the following options: good to worst, warn to worst, fail to worst, and min to max. This option is not enabled when the Quality Index legend is open.

Beyond Threshold As

Changes how the elements that have no results or have higher/lower values are displayed in the graphics area. The following display options are available: Transparent, Feature Lines, and Off (default).

Edit Criteria

Opens the Criteria File Editor, from which you can edit the current 2D element criteria in the criteria file.

> **Note:** As long as the Criteria File Editor is open, you cannot modify the legends in the Element Quality View.

Configure Quality View

Opens the **Element Quality View Configuration** dialog, where you can:

- Change the color assigned to each criteria.
- Select which element criteria you would to display in the Multiple Criteria legend.
- Rearrange the order of the element criteria in the Multiple Criteria legend

Element Quality Configuration Dialog

In the **Element Quality View Configuration** dialog, you can customize the appearance of the Element Quality View tool, and select the criteria that appears in the Multiple Criteria legend. Open this dialog by clicking **Configure Quality View** from the right-click element quality view context menu.

The **Element Quality View Configuration** dialog contains a list of all possible 2D element criteria that you can display in the Multiple Criteria legend. To select which 2D element criteria you would like to display in the Multiple Criteria legend, select or clear each element criteria's checkbox. Click the blue up and down arrows to rearrange the order the 2D element criteria will appear in, in the Multiple Criteria legend.

> **Note:** If you change the order of the element criteria after you have adjusted the threshold values in the Multiple Criteria legend, the threshold values will reset to their fail values. It is best to change the order of the element criteria first, and then change the threshold values in the Multiple Criteria legend.

You can also change the display color of each criteria in this dialog by selecting a new color from the color pallet.

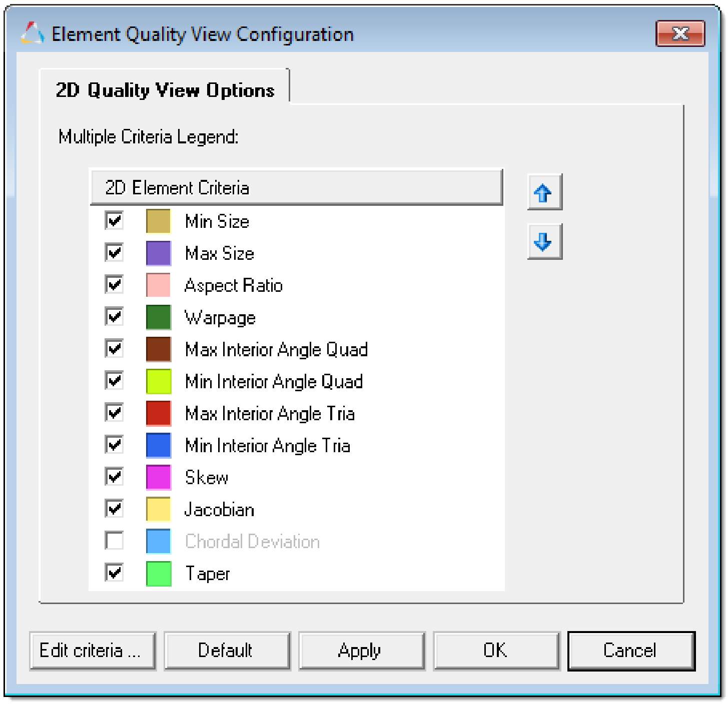


Figure 961:

Check Element Quality

After you build your model, check the quality of the elements in the model.

You can check your model for connectivity and duplicate elements.

- Use the Check Elems panel, 1D subpanel to:
  - Check one-dimensional elements for free ends
  - Determine if a group of rigid elements form a loop
  - Check weld and rigid elements for double dependency
  - Check all elements for a minimum length of a side
- Use the Check Elems, 2D panel to:
  - Check elements for warpage, aspect ratio, skew, and jacobian ratio
  - Check the maximum and minimum interior angles of quad and tria elements
  - Check all elements for a minimum length of a side
  - Check a mesh of elements for its maximum chordal deviation from a real or inferred surface
- Use the Check Elems, 3D panel to:
  - Check elements for warpage, aspect ratio, skew, and jacobian ratio
  - Check the maximum and minimum interior angles of quad and tria elements
  - Check all elements for a minimum length of a side
  - Check tetra elements for collapse, CFD-style volumetric skew, and Nastran-style aspect ratio

- Use the Check Elems, Time subpanel to check for elements whose small size might cause problems for an explicit solver.
- Use the Check Elems, Group subpanel to check for and eliminate group or interface elements whose underlying structural element has changed and left them detached.
- Use the Check Elems, User subpanel to verify element quality by using a template file that checks for user-specified conditions.

Create Mesh Quality Report

Create an HTML mesh quality report for the current model or for multiple models.

The standard HyperMesh criteria file is used as input. The mesh quality report summarizes the Quality Index (QI) calculations, along with information on the number of elements that fall into each QI range.

A report consists of:

- A single summary HTML page that gives a brief summary of several key metrics for all selected models. This page links to pages with detailed information about the mesh quality of each model.
- A directory containing HTML pages for each model. These pages contain detailed mesh quality information. This page contains a link to the input HyperMesh file as well as the criteria file used.

The HTML files and directory are named based on the input file and the date/time the report was generated.

1. From the menu bar, click **Mesh** > **Checks** > **Elements** > **Quality Report** and select:
   - Choose **Current Model** to create a report for the current model.
   - Choose **Multiple Models** to create a report for multiple models.
2. In the **Browser for Folder** dialog, select the output directory where the report will be located and click **OK**.
3. If you are creating a report for multiple Files, in the **Select model file(s)** dialog, select the database files to generate the report for and click **Open**.
4. In the **Select criteria file** dialog, select a criteria file and click **Open**.

If you are creating a report for the current model, when the current model is not saved as an `.hm` file, instead of using the HyperMesh model name in the HTML, the name CurrentModel is used. There is no link from the detailed report to the model file, as one does not exist.

If you are creating a report for multiple models, each HyperMesh file is loaded one at a time into the current HyperMesh session, and the report is generated for each file.

> **Restriction:** HyperMesh files should not contain any special HTML characters like #. This will prevent the HTML links from working properly.

Review Shell Thickness

The Thickness view allows you to investigate the shell thicknesses in your model.

The Thickness view is a permanent visualization mode that HyperMesh displays in the upper left-hand corner of the graphics area when you select **By Thickness** on the Visualization toolbar. Both element thickness as well as node thickness is supported. The thickness colors can be shown along two different legend styles, discrete and ranged.

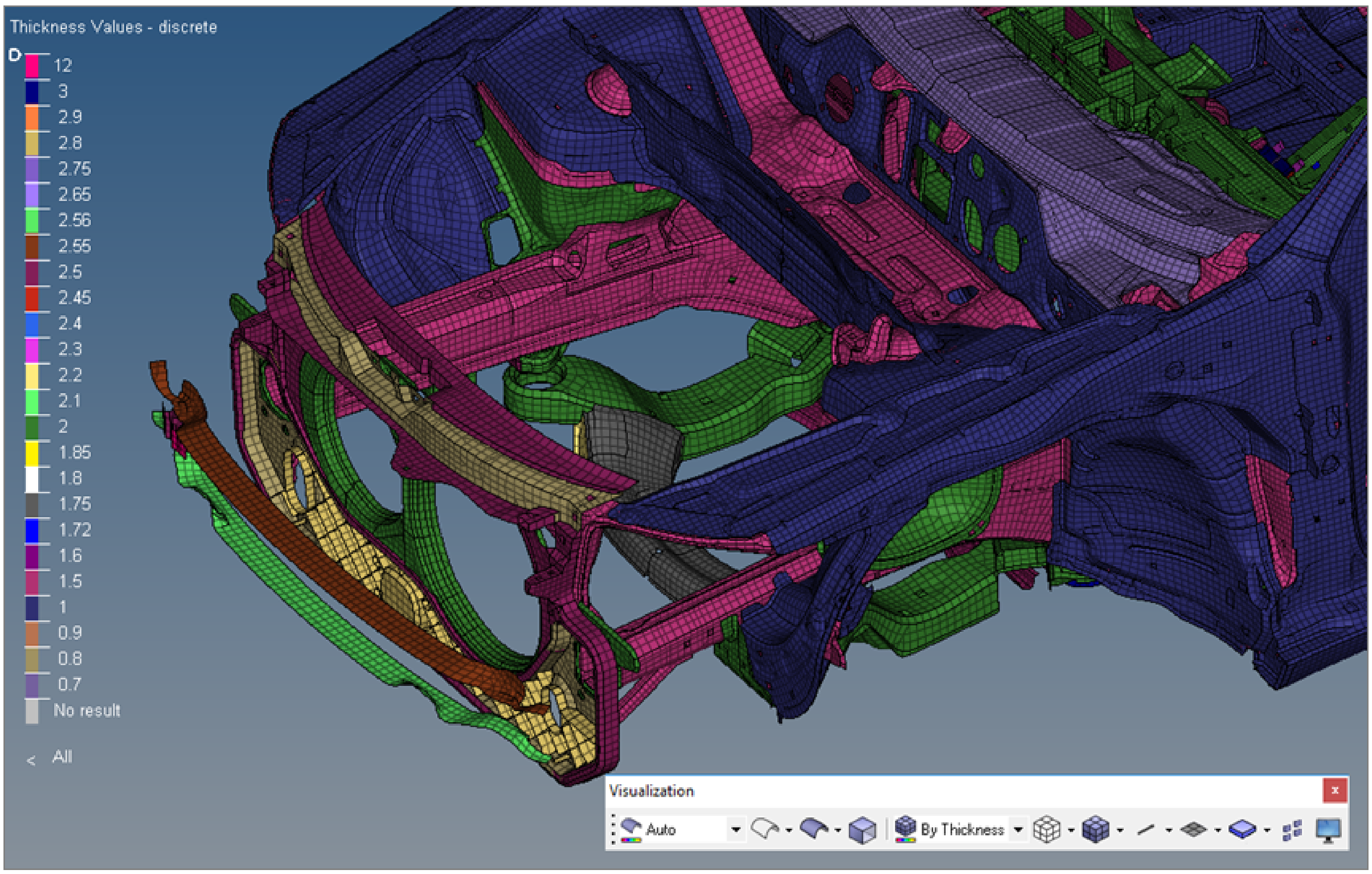


Figure 962:

Discrete Thickness Legend

When you select Thickness View, the Discrete Thickness legend displays by default.

In the Discrete Thickness legend each individual thickness value has its own exclusive color and hence the thickness values are not editable. This legend allows the evaluation of the element thicknesses in the graphics area.

The Discrete Thickness legend always lists the thickness values from lowest to highest. If further elements are displayed and these elements have a thickness assigned, which is not yet shown in the legend, the legend will be updated appropriately and the new thickness value will be properly sorted among the other thickness values.

> **Note:** When the number of different thicknesses to be displayed exceeds 25, HyperMesh automatically switches to the Ranged Thickness legend.

The slider, in the upper-left corner of the Discrete Thickness legend, can be used to change the display of elements beyond the threshold value. Elements can be hidden, or can be shown as transparent or wireframe. Clicking on the Operator Sign below the color bar changes it from 'smaller than' to 'larger than' and vice versa.

Reposition the slider by clicking-and-dragging it to the desired value on the legend. Same can be reached by clicking onto the thickness threshold value next to the operator sign and entering a new value. The value will be rounded to the next available thickness value.

Modify colors by clicking a color in the legend and selecting a new one from the color palette.

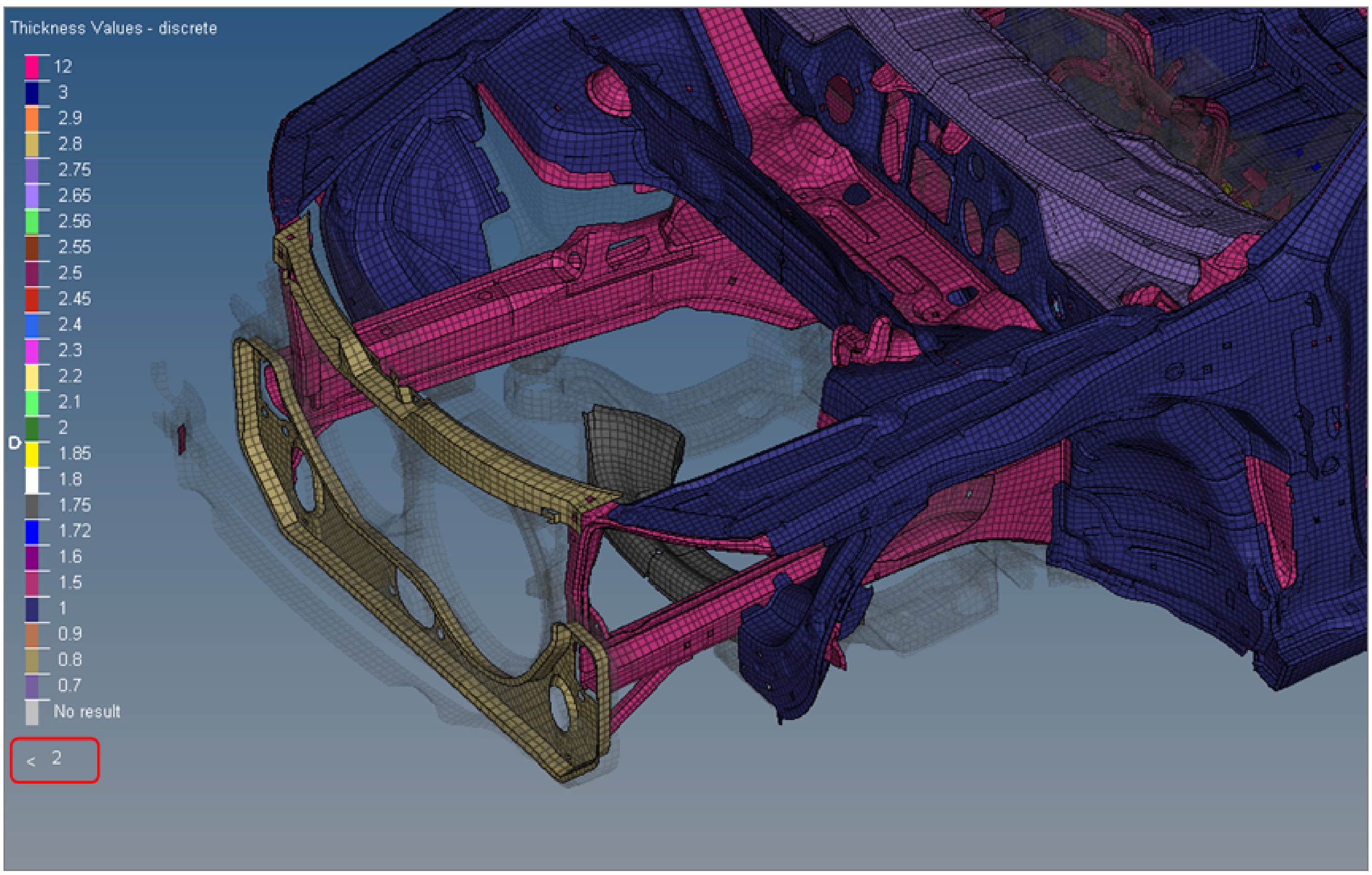


Figure 963:

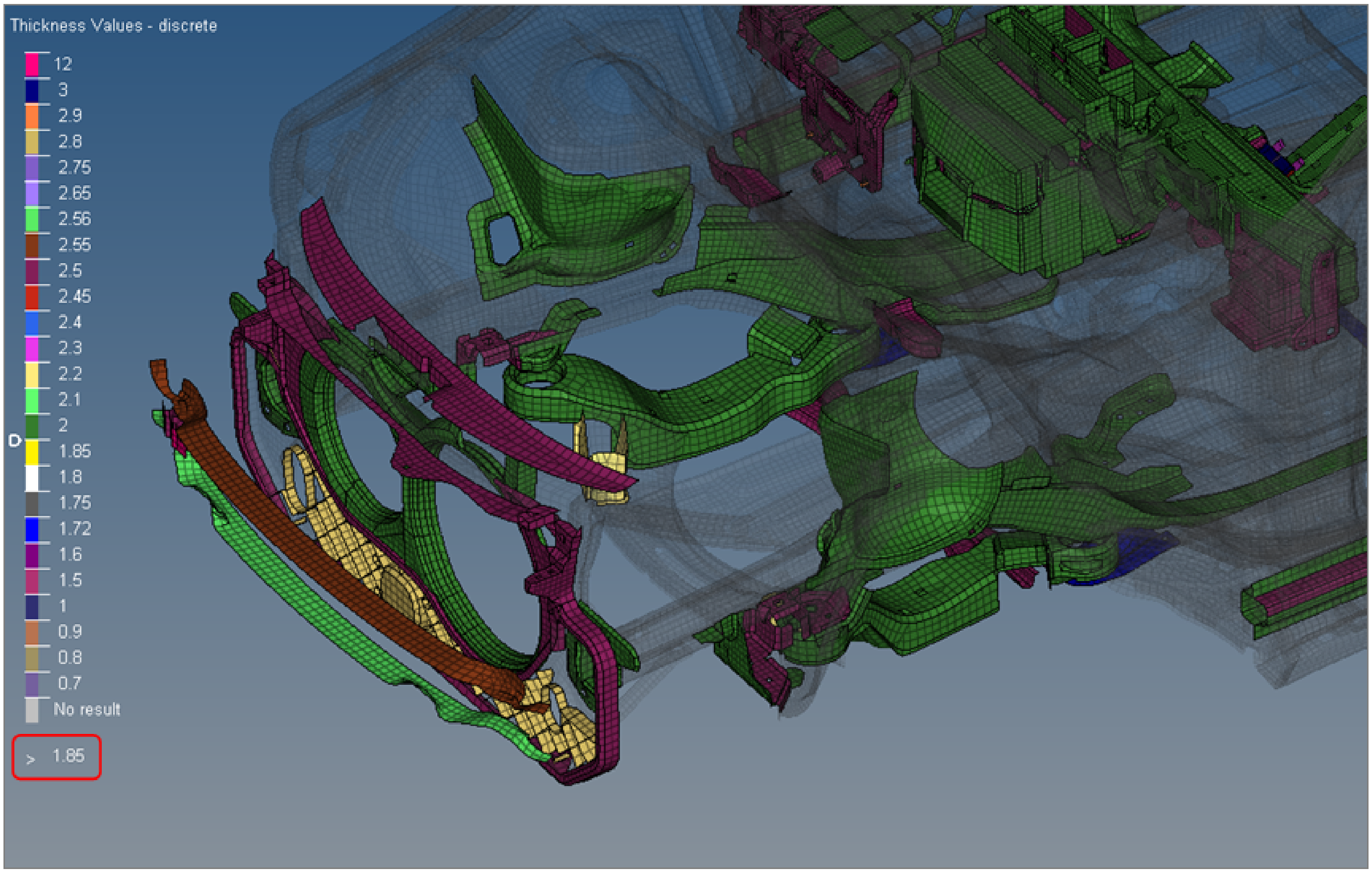


Figure 964:

Ranged Thickness Legend

The Ranged Thickness legend consists of a color coded sliding scale. The colors exhibited reflect the thicknesses of each 2D element in the graphics area.

The Ranged Thickness legend always lists the thickness values from lowest to highest. Edit these values by clicking on them and entering a new value. The remaining thickness values are interpolated between other edited values and the lowest and highest value in the legend.

> **Note:** Reset a value by clicking it and leaving the field empty. The value will be re-interpolated.

You can also edit the lowest and highest values, but the values cannot exceed the minimum and maximum thicknesses in the range.

The slider, in the upper-left corner of the Ranged Thickness legend, can be used to change the display of elements beyond the threshold value. Elements can be hidden, or can be shown as transparent or wireframe. Clicking on the Operator Sign below the color bar changes it from 'equal or smaller than' to 'equal or larger than' and vice versa.

Reposition the slider by clicking-and-dragging it to the desired value on the legend. Same can be reached by clicking onto the thickness threshold value next to the operator sign and entering a new value. The value will be rounded to the next available thickness value.

Modify colors by clicking a color in the legend and selecting a new one from the color palette.

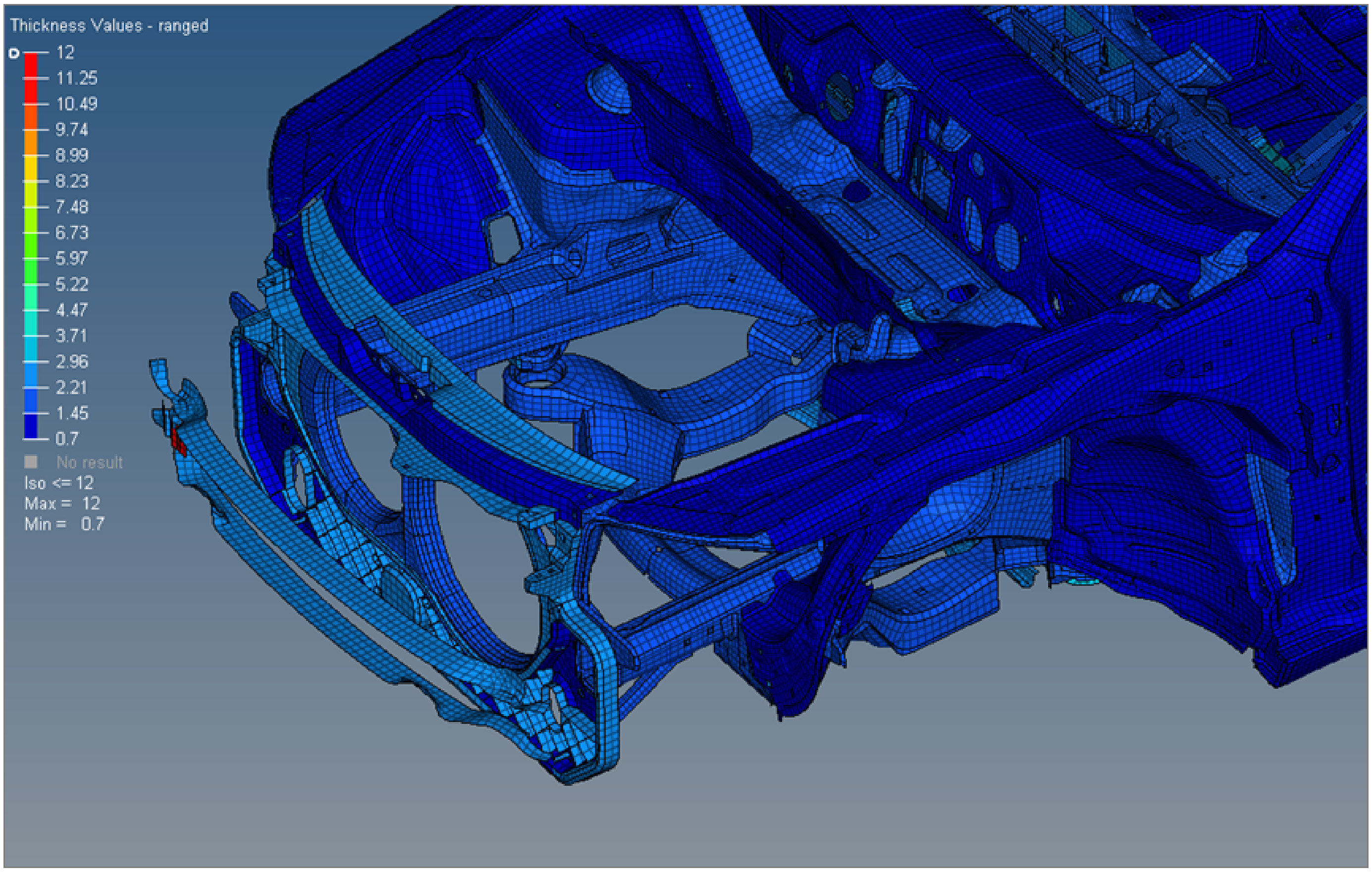


Figure 965:

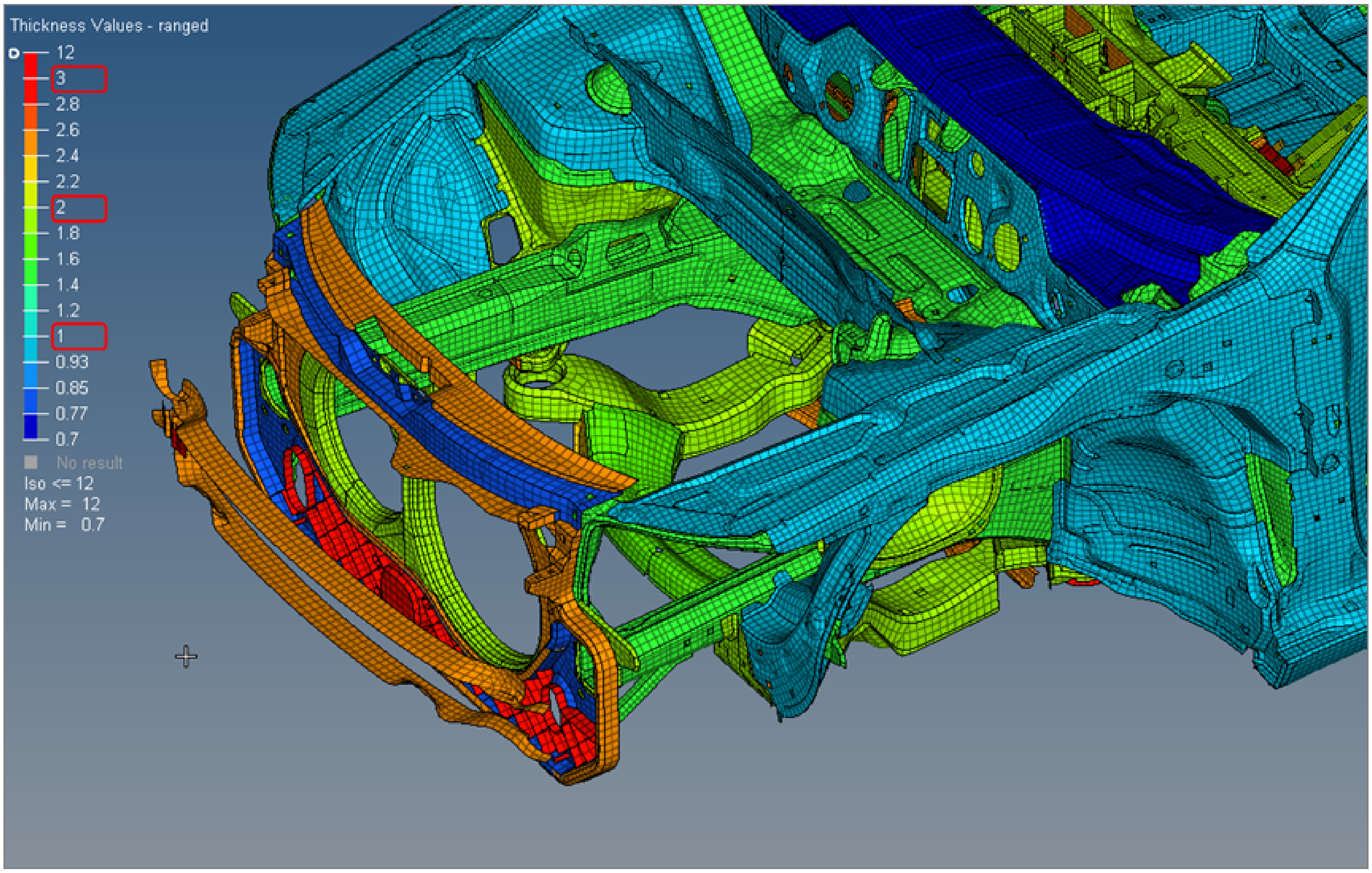

Thickness Values - ranged
12
3
2.8
2.6
2.4
2.2
2
1.8
1.6
1.4
1.2
1
0.93
0.85
0.77
0.7
No result
Iso <= 12
Max = 12
Min = 0.7

Figure 966:

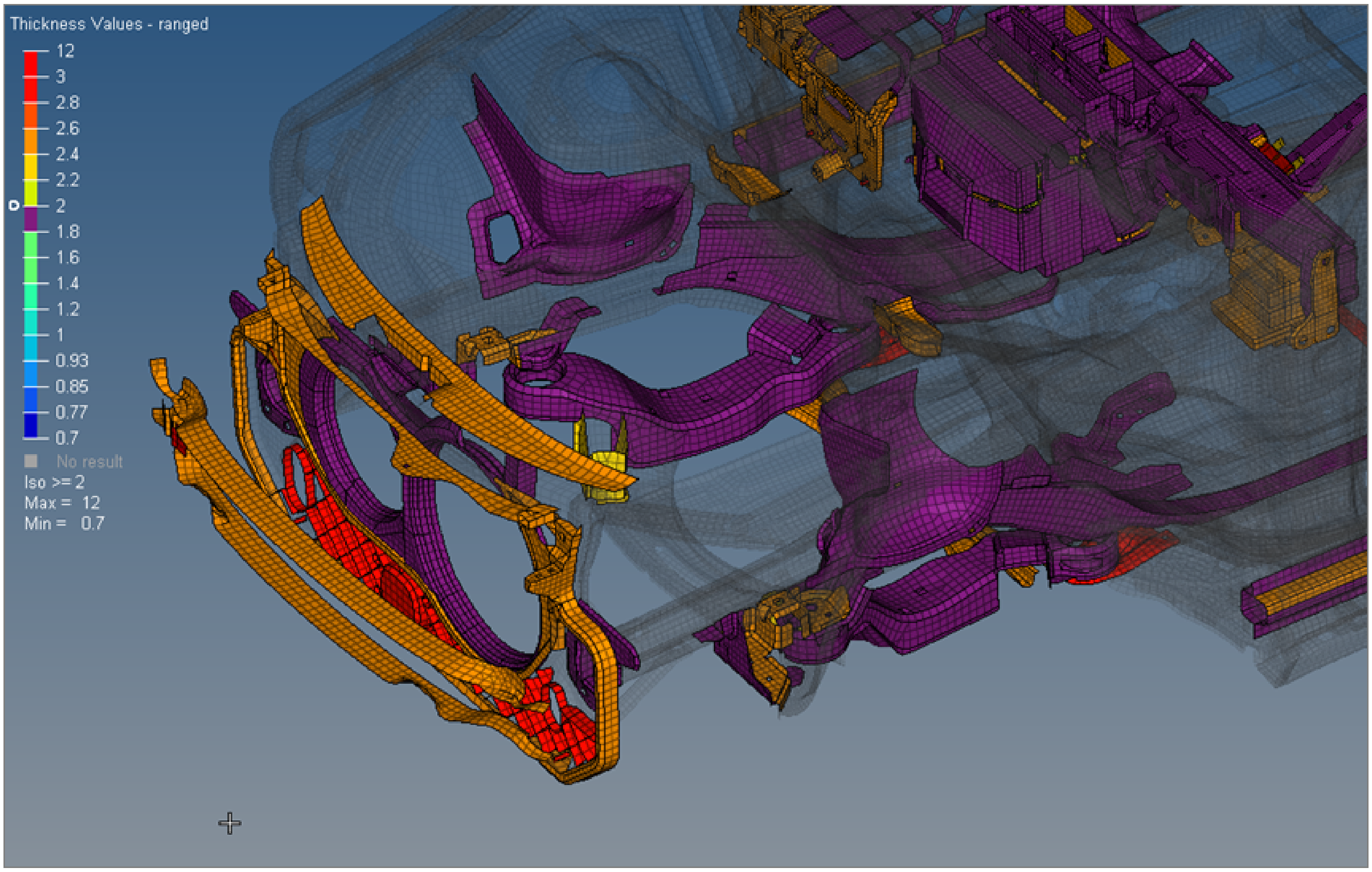


Figure 967:

Thickness View and Plies

The Discrete Thickness legend and the Ranged Thickness legend are both capable of showing ply thicknesses. Any combination of 2D Detailed Element Representation and Composite Layer Visualization is supported.

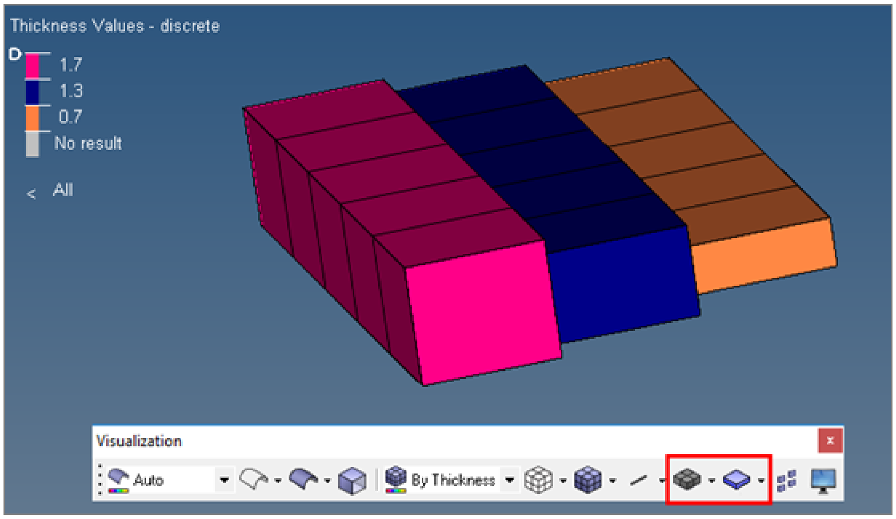


Figure 968:

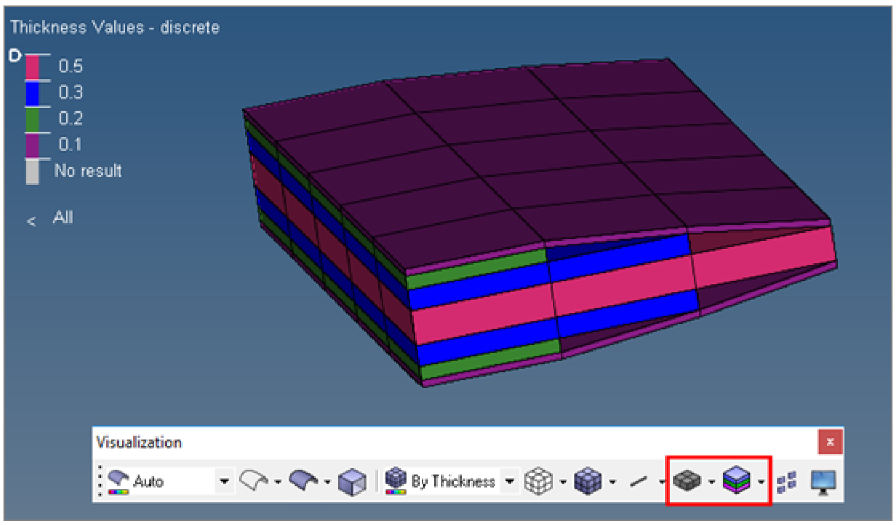


Figure 969:

Thickness View Context Menu

Use the right-click context menu to dictate the Thickness View. To access the Thickness View context menu, right-click on the ranged or discrete legend.

Beyond Threshold As

Change how the elements that have no results or have higher/lower values are displayed in the graphics area. The following display options are available: Transparent, Feature Lines, and Off (default).

Ranged Legend

Set/unset the check for the Ranged Legend.
When this option is set or the number of thicknesses to be shown exceeds 25, the Ranged Legend is used. When Ranged Legend is unset, the Discrete Legend is used.
This option is only available when the number of visible thicknesses is equal or less than 25.

Reset

Restores the legends to their initial default settings, that is colors, values, and number of levels.

Number of Levels

The number of levels describe how many colors the colorbars are displayed. It can be set to any value between 2 and 15 (Ranged legend only).

Check and Fix Boundary Shell Intersections

Validate the input surface mesh before performing volume mesh generation using the **Boundary Shell Checker**.

Before you can invoke the **Boundary Shell Checker**, select the input shell mesh, forming the closed volume.

1. Open the **Boundary Shell Checker**.
    - In the **CFD tetramesh** panel, click **check 2D mesh**.
    - In the **tetramesh** panel, click **check 2D mesh**.
    - In the **Mesh Controls** browser, right-click on a Volume Mesh control and select **Check 2D Mesh** from the context menu.
2. Check your mesh by defining check options and clicking **Check**.

Option	Description
Proximity tolerance	Perform a proximity check for the selected shell elements forming one volume. This check will detect a penetration (for example two elements cutting each other) as well as an "almost" penetration (for example one node is very close to an element). To detect the latter situation, a proximity tolerance can be specified. The actual search tolerance is relative to the local minimum edge length and is defined by search tol. = (proximity tol) * (local min. edge length). The failed elements are duplicated and are placed in the component ^error_elems.

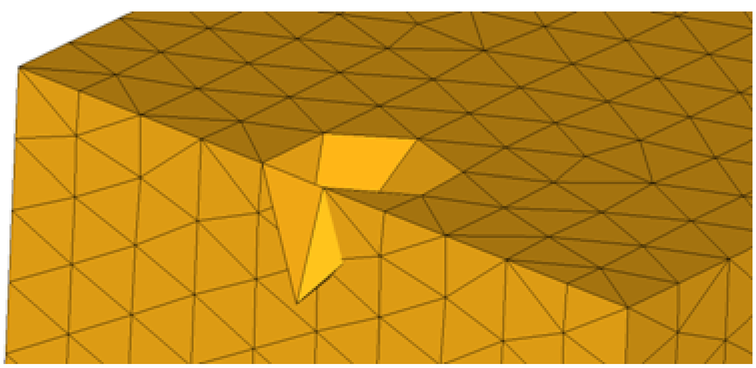
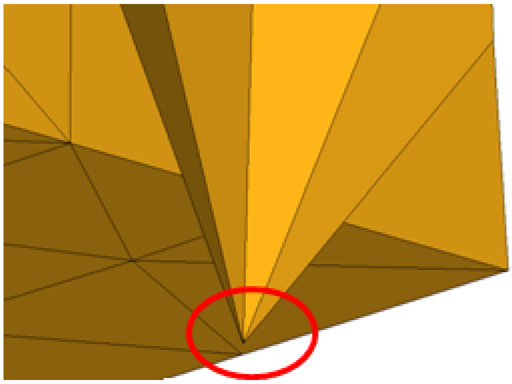
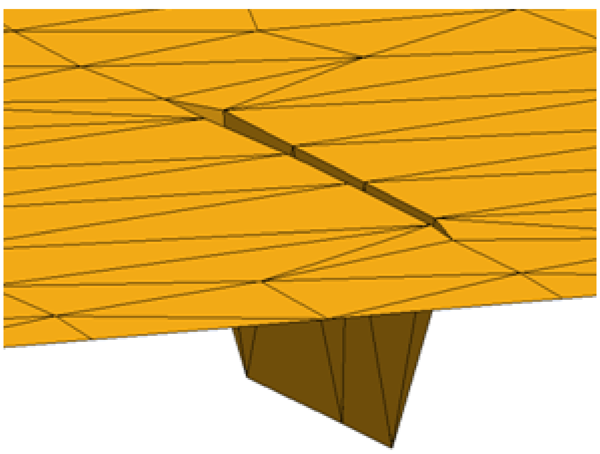


Figure 970: Penetration Example

A penetration, where two or more elements are cutting each other.

Option	Description
	 *Figure 971: Almost Intersection* *A "almost" interaction, where one mesh node is very close to the shell elements. The node and shell elements belong to the same volume.*
Dihedral angle <	Check the selected input mesh for adjacent elements which form a sharp angle, for example a needle pocket. The threshold angle is user defined. The failed elements are duplicated and are placed in the component ^error_elems.  *Figure 972: Needle Pocket* *The involved elements form a sharp pocket pointing outwards of the volume.*
Per vol check	Find the intersections that are only within the shell volumes. Intersections across volumes and open shells will not be found. If this checkbox is disabled, intersections will be checked for all of the input elements/ components even if the input shells do not form closed volume. Along with intersecting and diehedral elements, all duplicate elements and close proximity elements within the user defined tolerance will also be found. All of the failed elements will be placed into a separate component for review.

Option	Description
	^err_x_elems Contains intersecting elements. **^err_agl_elems** Contains failed dihedral angle check elements. **^err_prx_elems** Contains close proximity elements within the user defined tolerance. **^minus_prx_elems** Contains close proximity elements for a per volume check which has opposite orientation. These elements are safe for volume meshing. **^minus_agl_elems** Contains dihedral angle check elements for a per volume check which has opposite orientation. These elements are safe for volume meshing. **^err_dup_elems** Contains duplicate elements.
Results to sets	Find issues in sets, and create new components.
Results by cluster	Create separate error element components/sets according to how they are clustered.

3. Fix your mesh by defining fix options and clicking **Fix**.

Option	Description
Fix intersection	Resolve intersections identified within selected components. This option is useful when you want to remove localized intersections within your model before tetrameshing
Fix aspect ratio >	Fix elements organized within selected components that are higher than the defined aspect ratio. Surface elements that have a high aspect ratio can result in a poor quality tetramesh.
Fix surface element	Fix elements organized within selected components that are higher than user defined height.

Option	Description
minimum height <	

Solid Mesh Optimization

The Solid Mesh Optimization tool can be used to improve the quality of a tetra, hexas, and second order meshes with respect to several element criteria.

Solid Mesh Optimization Tetras

This topic covers solid mesh optimization for tetras.

In this task you will learn about tetra meshing.

1. To access this utility, click **Mesh > Check > Elements > Solid Mesh Optimization**.
2. Select a set of elements to fix using the Input selection collector.
3. Complete the Tetras table items by entering the appropriate values.

   For more information on table items, refer to the Altair HyperMesh Element Quality Calculation section.
4. Select a Triangles option.

   Fix All
   All triangles are fixed during optimization.

   Edge Swap
   Edges of boundary triangles can be swapped during optimization. Node locations of boundary triangles are not modified.

   Remesh
   Boundary triangles can be remeshed during optimization. This option usually yields the best results.
5. Set Constraints options according to the following table.

Constraint	Action
Fixed Trias	Set this constraint to **Elements** if you do not want to modify tria elements during the optimization step. This constraint is only available if Edge Swap or Remesh is selected under Boundary triangles.
Feature Line	Set this constraint to **Elements** to mark feature lines by 1D plot elements that coincide with element edges. Selecting those 1D edges ensures that the corresponding element edges are not modified during the optimization step. This option is only

Constraint	Action
	available if Edge Swap or Remesh is selected under Boundary triangles.
Anchor Nodes	Set this constraint to **Nodes** so that selected mesh nodes are not modified during the optimization process.
Refinement Box	Set this constraint to **Elements** so that elements inside the refinement box are remeshed and optimized with respect to the defined thresholds.

6. Check the **Fix Shell Comp Boundaries** option to maintain the edges between two components during the optimization step.

   This is useful when the interface between a solid and a fluid component has to be maintained.
7. Check the **Maintain Geometry Edges** option so that nodes on a geometry edge are not modified during optimization.
8. Check the **Save to Current Comp** option to save elements generated during optimization to the current component.
9. Check the **Update Input Shells** option to update shell elements attached to modified tetra elements.

   This option is only available if Edge Swap or Remesh is selected under Boundary triangles.
10. Check the **Optimize Tetras by Force** option to include a node insertion algorithm.

    This usually yields a higher element quality in the resulting mesh, but in return might also increase the computation time.
11. Enter the maximum number of optimization steps in the **Maximize Iteration** field.
12. Enter an angle in the **Feature Angle** field that will be considered a feature edge and that will be preserved during the optimization step.

    If the angle of two adjacent shell elements is greater than this angle, the edge will be preserved.
13. To show elements that have failed, click **Show Failed**.

    Only the elements that have failed the selected element criteria will be displayed. Click this button again to redisplay all elements.
14. To perform an element check based on the defined parameters in the **Criteria Editor**, click **Check**.

    Mesh statistics will be displayed in the table.
15. To start the optimization process, click **Fix**.
16. To re-establish the initial tetra mesh and reject the modified mesh, click **Reject**.

17. When you are finished, click **Close**.

Solid Mesh Optimization Hexas

This topic covers solid mesh optimization for hexas.

In this task you will learn about hexas meshing.

1. To access this utility, click **Mesh > Check > Elements > Solid Mesh Optimization**. Then click the **Hexas** tab.
2. Select a set of elements to fix using the Input selection collector.
3. Complete the Hexas table items by entering the appropriate values.
4. Define the relative weight for the active quality criteria.

   Criteria with a higher weight will get higher preference in case of conflict.
5. Check the **Allow Boundary Node Movement** option to allow for node movement on or normal to the boundary.

   If you check this option, you can define the following fields:

Option	Description
On Geometry: Maximum Node Movement/ Element Size	Enter a value in this field that defines the maximum node movement allowed *on geometry* to fix hex quality as a factor of the hex element.
Away From Geometry: Maximum Node Movement/Element Size	Enter a value in this field that defines the maximum node movement allowed *away from geometry* to fix hex quality as a factor of the hex element.
Feature Angle	Enter a value in this field that identifies the feature edges on which node movement will be restricted along the edges only.

6. Enter a value in the **Maximum Iterations** field that defines the number of maximum iterations to go through to fix quality.

   Hex quality improvement will terminate if quality is not fixed within the defined iterations.
7. Check the **Show Original Face Node Location** option to create temporary nodes showing input node positions that were moved during optimization.
8. To show elements that have failed, click **Show Failed**.

   Only the elements that have failed the selected element criteria will be displayed. Click this button again to redisplay all elements.
9. To perform an element check based on the defined parameters in the **Criteria Editor**, click **Check**.

   Mesh statistics will be displayed in the table.

10. To start the optimization process, click **Fix**.
11. To re-establish the initial hexa mesh and reject the modified mesh, click **Reject**.
12. When you are finished, click **Close**.

Solid Mesh Optimization Second Order

This topic covers second order mesh optimization.

In this task you will learn about second order meshing.

1. To access this utility, click **Mesh > Check > Elements > Solid Mesh Optimization**. Then click the **Second Order Elements** tab.
2. Select a set of elements to fix using the Input selection collector.
3. Define the element **maximum angle**, **minimum and maximum length ratio**, and **Jacobian thresholds**.
4. Select the appropriate option in the **Evaluate Jacobian at** field.
5. Select the appropriate options in the **Middle Nodes Repositioning** field:
    a) Check **Internal Nodes** to allow mid tetra nodes located interior to volume mesh.
    b) Check **Boundary Node** options to allow mid tetra nodes located on the boundary. **Move Along Geometry First** enables tetra mid nodes along geometry to fix quality. **Move Off From Geometry** enables tetra mid nodes away from geometry to fix quality.

       Tetra mid nodes will only be moved away from geometry, if moving nodes along geometry does not fix quality.
6. To perform an element check based on the defined parameters in the **Criteria Editor**, click **Check**.

   Mesh statistics will be displayed in the table.
7. To start the optimization process, click **Fix**.
8. To re-establish the initial second order mesh and reject the modified mesh, click **Reject**.
9. When you are finished, click **Close**.

Check for Element Penetrations/Intersections

Penetration is defined as the overlap of the material thickness of shell elements, while intersection is defined as elements that actually pass completely through one another.

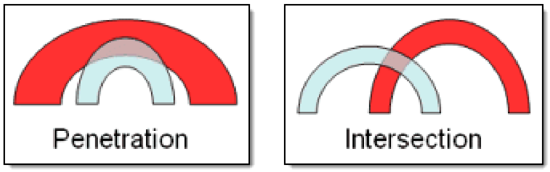


Figure 973: Penetration and Intersection

> **Restriction:** Penetration checking is supported by all of the impact solver interfaces, such as LS-DYNA, Radioss and PAM-CRASH, and works best with a solver interface that supports thickness data for modeled shell elements. The default HyperMesh solver interface does not support modeled element thickness, but the penetration checking tools can still be used to specify a uniform thickness when performing a check.

- You can only set up and initiate the check in the Penetration panel; the majority of the checking tool actually resides in a special tab that opens in the tab area. However, this tab only displays after you complete the panel and run an initial check.
- When the penetration check runs, it automatically masks (hides) everything in your model except for the penetrating or intersecting elements. It then fits the view to these elements' components. You can show or hide additional elements using toolbar buttons located in the Penetration tab, and you can make other entity types, such as ellipsoids, visible again via the Display panel or the Mask panel.
- Solid entities never register penetrations between each other; instead, any overlap between solids registers as intersections, because one or more of each the solid's faces intersect. A solid that is completely contained within another solid will not be detected as an intersection or penetration, because its faces will not intersect any of the larger (containing) solid's faces. In addition, only surface elements register penetrations; the tool cannot find penetrations between internal, that is, tetra- or hexa-, elements.

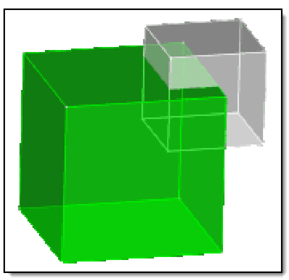


Figure 974: Intersections Found

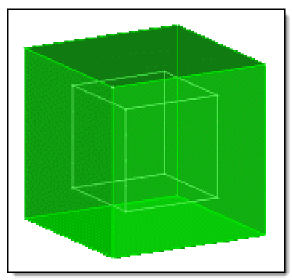


Figure 975: No Intersections

- Solids can register penetrations with respect to adjacent shell elements, based on the thickness of the shell elements.

1. From the Tools page, click **Penetration**.

2. Use the entity selector to select entities to be checked for penetrations or intersections.

In any case, the penetrating elements will be found; for example, picking two components locates elements from each component that penetrate elements of the other.

3. Select the type of interferences to check for.

Choose **all interferences**, **intersections only**, or **penetrations only**.

> **Note:** Solid entities only register penetrations in conjunction with shell elements. With other solids, they only register intersections.

4. Select the type of elements to check.

   Choose **2D and 3D elements**, **2D only elements**, or **3D only elements**.

   If you choose **2D only elements** or **3D only elements**, HyperMesh only checks elements of the specified type for penetration. Elements in the selected entities which are not of that dimensional type will be ignored, even if they penetrate or intersect another entity.
5. To enter a tolerance for penetration checking, select the **allowable interference depth** checkbox.

   For example, if you check a model of a part measured in millimeters, and are not concerned about penetrations of less than a tenth of a millimeter, you could set this field to a value of 0.1. The penetration tool would then ignore any elements that penetrated each other by 1/10 of a millimeter or less, but still locate and highlight elements with a penetration depth greater than 1/10 mm.

   Some solvers will not permit direct, adjacent contact between elements, for example a penetration depth of exactly zero, with no space between elements. For such solvers, you should set this field to any negative value, such as -0.1. This allows the penetration tool to locate and display elements that are exactly adjacent to one another as if they were penetrating each other, so that you can use the penetration fixing tools to add some space between them.
6. Select a method adjusting thickness.

   Not available when you run the check on groups, because a group’s thickness is defined by its control card.

   - Choose **Component thickness** to use the thickness value specified in a component’s property card for each element within that component.
   - Choose **thickness multiplier** to enter a value to multiply the selected entities’ thickness by for purposes of the penetration check. Fractional numbers are acceptable, but negative ones are not.
   - Choose **uniform thickness** to enter a thickness value.

     This can be used to work around the lack of thickness information in the default HyperMesh user profile, or when working with models that do not have a thickness specified.

     You can also use this option to determine the proximity between non-penetrating parts by specifying a thickness greater than the minimum distance between the selected elements.
7. To check for components that intersect or penetrate themselves, for instance, due to high curvature in the component, select the **include self interference** checkbox.

   This option is computationally intensive, therefore it is not recommend that you use it when checking large numbers of elements for penetration.
8. Click **check**.

The selected components, elements, or groups are checked for penetration and/or intersection. A message in the status bar displays the percentage progress of each step in the check.

If for some reason you wish to abort a check, you can do so by right-clicking in the graphics area and holding the button down. The exact length of time that you must hold the button depends on the size and complexity of the check you are running; the check will cancel once its completion percentage increments. When the check aborts, the status bar turns red and displays a message stating that the check was canceled.

Once the penetration/intersection check is complete, view the results of the check and make adjustments.

Edit Mesh

Modify your mesh by detect holes, locating edges or features, refining mesh pattern, and so on.

Detect Holes

Locate holes in a model, and potentially all of them, define them, and add them as geometry to a new component or to the current one.

You can specify many types of criteria to define specific types of holes that you wish to find.

1. From the menu bar, click **Check** > **Components** > **Hole Detection**.
   The **Hole Detection** dialog opens.
2. Prepare for hole detection.
   a) Click the **Preparation** tab.
   b) For Select entities, use the selector to select the components to scan for holes.
   c) For Detect holes in, select the types of elements to find holes.
   d) In the 2D adjacent face deviation angle field, enter the maximal angle deviation of the holes axial direction of adjacent element normals.

      Acceptable values range between 0.0 and 90.0 degree. Lesser values will result in a default of 45.0 degrees.
   e) For 3D solid hole feature detection, select a method for detecting holes.
      - Choose **By specified angle** to enter the exact angle of holes you wish to detect.
      - Choose **Auto using angle** to enter upper and lower limits of hole angles that you wish to detect. Holes with feature angles beyond either of these numbers will be ignored.

      Holes in 3D solids are assumed to have an opening on one or more faces of the solid. You can base detection on each hole's feature angle, that is, the angle at which the hole deviates from the face in which its opening appears.

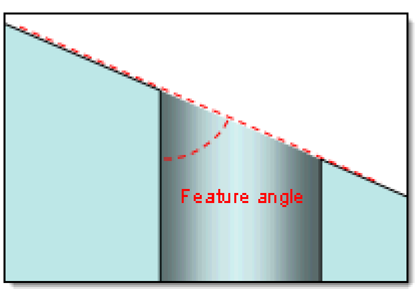
      

      Figure 976:

      In either case the values must be more than zero (zero would be perfectly collinear with the face) but no greater than 90 degrees, which represents a hole that runs perfectly perpendicular to the face.
   f) Click **Run**.

3. Define the type of holes to find in 2D mesh.

a) Click the **2D Holes** tab.

b) For Hole type, select the type of opening for the 2D shape.

Options include circular, including ovals, square, rectangular, and general (all shapes).

c) To define a specific hole width to find, regardless of shape, select the **Minimum dimension** and/or **Maximum dimension** checkboxes and enter a dimension.

If set at or below zero, these checks are not run.

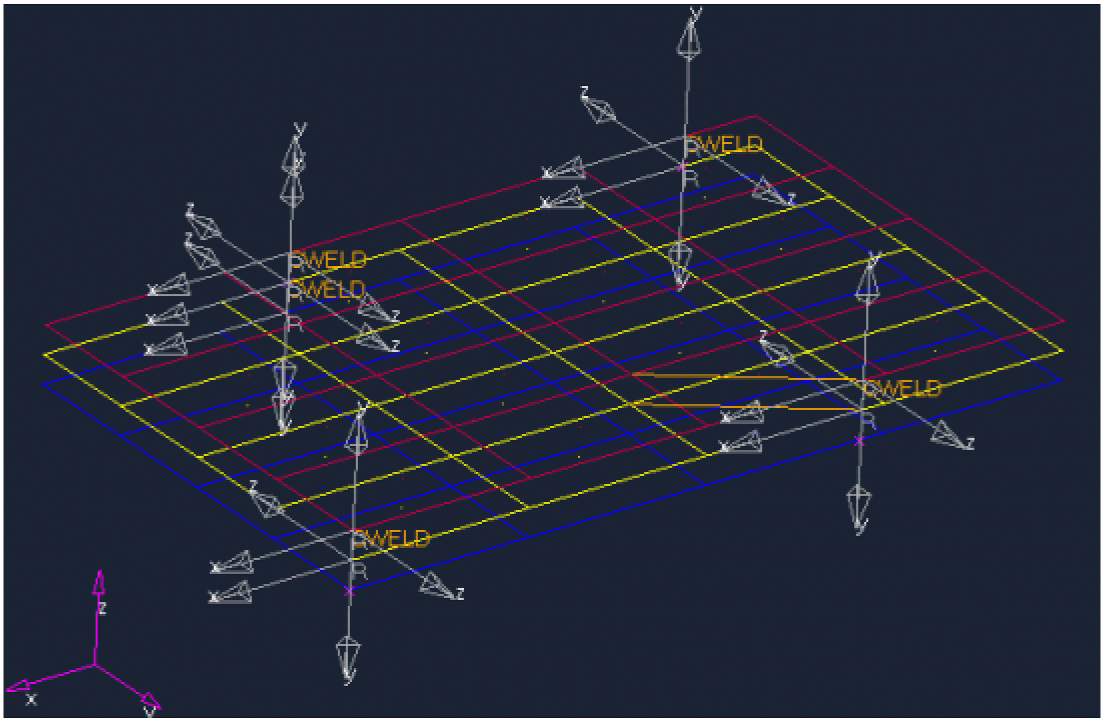


Figure 977:

d) To check each node on the edge of a hole, relative to the plane that best approximates all of the nodes on the hole's edge, select the **Offset plane deviation** checkbox and enter a distance measurement.

Any nodes further than this distance from the midplane of the bounding box will cause the tool to ignore the hole. If this value is set to zero or less, the check is not run at all on any holes.

e) Choose which component the found holes are placed into.

- Choose **Auto** to organize holes in the ^edges_holes_shell component.
- Choose **Elems to current comp** to organize holes in the current component.

f) Click **Find**.

All 2D holes matching the criteria are located.

4. Define the type of holes to find in 3D mesh.

a) Click the **3D Holes** tab.

b) For Hole type, select the type of opening for the 3D shape.

Options include circular, square, rectangular, and general (all shapes).

c) To define a specific width of the hole's opening to find, regardless of shape, select the **Minimum dimension** and/or **Maximum dimension** checkboxes and enter a dimension.

This carries over from the 2D tab because the openings themselves are 2D edges. If set at or below zero, these checks are not run.

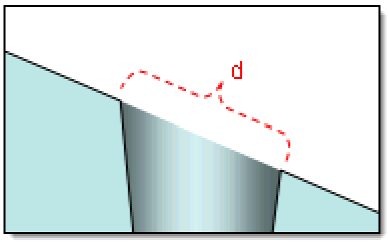


Figure 978:

d) To define the depth of the hole, regardless of shape, select the **Minimum height** and/or **Maximum height** checkboxes and enter a height.

If set at or below zero, these checks are not run.

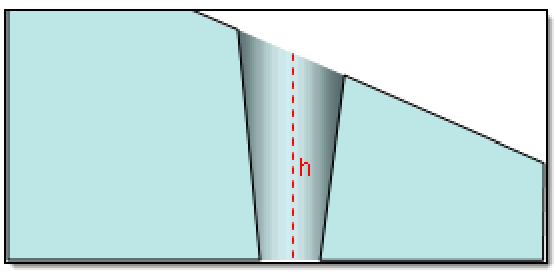


Figure 979:

e) To check each node on the edge of a hole, relative to the best-fit bounding box that encompasses all of the nodes on the hole's edge, select the **Offset plane deviation** checkbox and enter a distance measurement.

Any nodes further than this distance from the midplane of the bounding box will cause the tool to ignore the hole. If this value is set to zero or less, the check is not run at all on any holes.

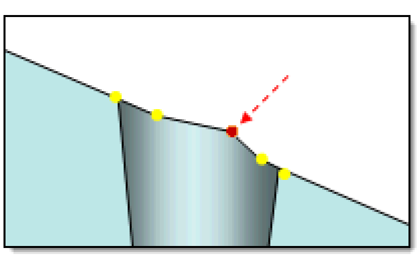


Figure 980:

With very low plane deviation, the red node might invalidate this hole.

f) To search for specific tapered holes; this is the maximum angle between the hole's sides, and a planar cross-section that is perpendicular to its length, select the **Cone angle** checkbox and enter an angle.

A value of 90 represents a hole that does not taper at all. Holes with a taper at or below the specified angle, that is, tapers sharper than the specified angle, will be found, while tapers above it, that is, closer to being a straight shaft, will be ignored. The default value is 80.0 degrees; if less than or equal to 0.0 the cone angle check is not run.

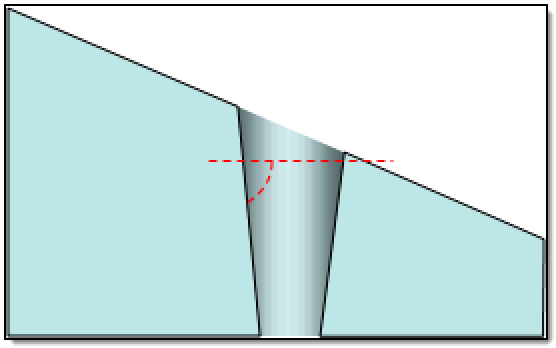


Figure 981:

g) Choose which component the found holes are placed into.
- Choose **Auto** to organize holes in the ^edges_faces_solid component.
- Choose **Elems to current comp** to organize holes in the current component.

h) To generate elements around the perimeter of the hole edge, select the **Create edges** checkbox.

These new elements are organized into a component called ^edges_holes_shell.

i) For Hole Handling, choose whether to find **Open** holes, **Capped** holes, or **All** holes.

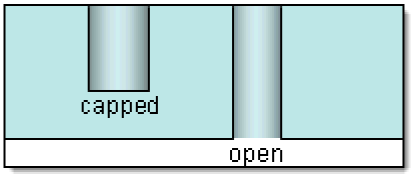


Figure 982:

j) Click **Find**.

All 3D holes matching the criteria are located.

Refine Mesh by Pattern

Create a regular orthogonal mesh.

Before you begin, make sure the initial mesh is a regular mapped quad mesh, and not a free quad mesh.

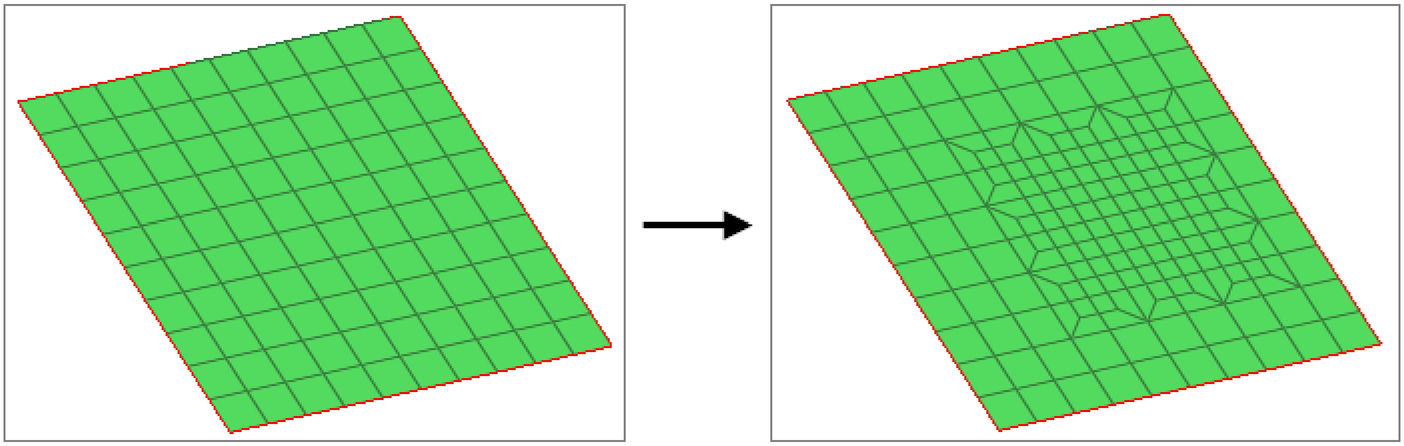


Figure 983: Pattern Based Mesh Refinement

1. From the menu bar, click **Mesh** > **Edit** > **Elements** > **Refine by Pattern**.
   The **Mesh Refinement** dialog opens.
2. Use the Select node: Nodes selector to select the center node to start the uniform mesh refinement.

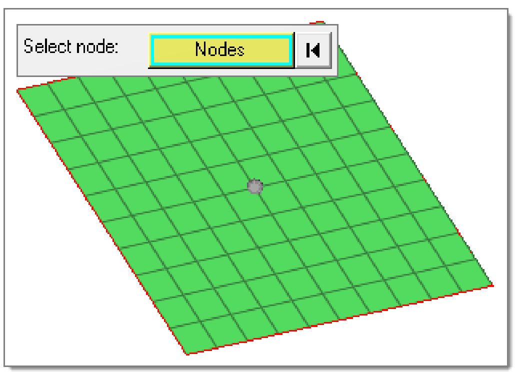


Figure 984:

3. For Select system, select a global or local system to be used for mesh propagation and box alignment directions.
4. Define the Refinement box, which determines the constant refinement zone.

   Elements inside this box should be the same size.

   a) In the mesh size field, enter the fine mesh size.

      The size should be smaller than the initial mesh.

   b) In the Length field, enter the length of the refinement box.

   c) To preview the box, click **Review**.

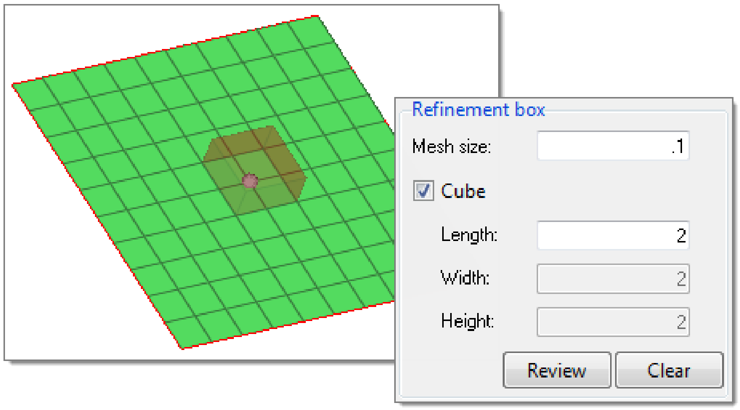


Figure 985:

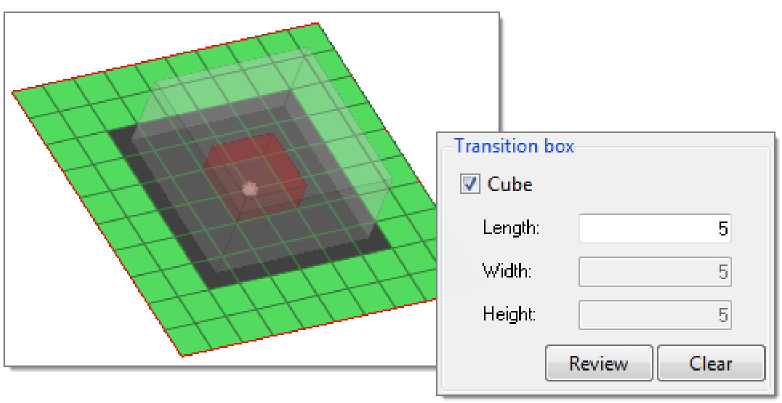
5. Define the Transition box.

a) In the Length field, enter the length of the transition box.

The mesh will be transitioned to match the initial mesh.

> **Note:** The transition box should enclose the refinement box. The size of the transition box determines how aggressive the transition will be. It is recommended that you limit the transition to 1 element connection 3 elements (1:3).

b) To preview the box, click **Review**.



6. Click **Mesh**.
Attached 1D elements are refined.

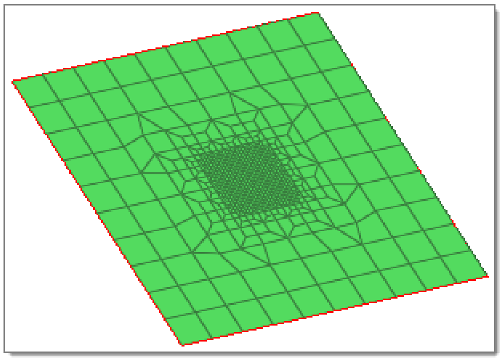


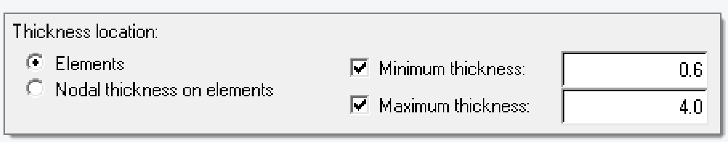
Figure 986:

Scale Element Thickness

Scale the thickness of elements in your model.

> **Restriction:** Only available in the LS-DYNA, Radioss and PAM-CRASH 2G solver interfaces.

1. From the menu bar, click **Mesh** > **Edit** > **Elements** > **Scale Thickness**.
   The **Scale Thickness** dialog opens.
2. Under Thickness location, define the thickness location of the entities you want to scale. By defining the **minimum thickness** and/or the **maximum thickness**, the scaled thickness will not go below the minimum and beyond the maximum thickness mentioned.
   a) In the Minimum thickness field, enter a thickness the scaled thickness will not go below.
   b) In the Maximum thickness field, enter a thickness the scaled thickness will not exceed.

   > **Note:** Nodal thickness on elements is only available in the LS-DYNA solver interface.
   >
   > 
   >
   >
   > *Figure 987:*

3. Under Selection, select entities to scale.
   - Choose **All** to select all of the entities in your model.
   - Choose **Entities** to use the Elements selector to manually select entities.

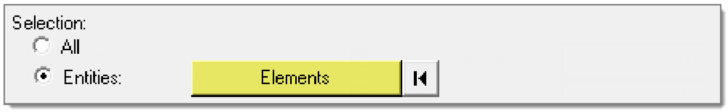


Figure 988:

4. Under Scale by, choose whether to multiply, add, or subtract the existing thickness with a desired value.

   To subtract from the thickness, select **Add** and enter a negative value.

   > **Note:** You cannot multiply with a negative value.

Figure 989:

5. Click **Calculate.**

   Thickness is scaled.

Scale Thickness Behavior

Solver interface specific behavior for scaling the thickness of elements.

LS-DYNA

You can only perform thickness operations on *ELEMENT_SHELL_THICKNESS and *ELEMENT_SHELL_COMPOSITES elements. If the card image of elements is *ELEMENT_SHELL_COMPOSITES, then the total thickness will be calculated using the sum of all the THICK attributes. Scaling will be done in proportion with all the thickness attributes.

For example, consider the number of plies is four and the THICK attributes are: THICK(1) = 2, THICK(2) = 4, THICK(3) = 6, THICK(4) = 8. The total thickness will equal 20. If you add two to this thickness, then after scaling the thickness of these attributes will be: THICK(1) = 2.2, THICK(2) = 4.4, THICK(3) = 6.6, and THICK(4) = 8.8. After scaling, the total thickness will become 22.

Radioss

You can only perform thickness operations on `/SHELL`/ and `/INISHE` cards. `/INISHE` supports the following three cards: `/INISHE/EPSP_F`, `/NISHE/STRA_F`, and `/INISHE/STRS_F`. If the thickness attribute of `/SHELL` is not defined and `/INISHE` is not available, then the thickness will either be defined by the Property or Component. As of now, the thickness of Property and Component entities cannot be assigned using the Scale Thickness tool, therefore thickness will be assigned to the `/SHELL` thickness attribute.

PAM-CRASH 2G

You can perform thickness operations on elements with card image SHELL and TSHEL for the attribute "T". You also perform thickness operations on components that have the card image SHELL, TSHEL, or MEMBR for the attribute H. There is no impact on thickness for the TCONT contact.

Calculate and Assign Midmesh Thicknesses

Calculate and assign the thickness of a midmesh from solid geometry using the **Midmesh Thickness** tool.

The thickness will be assigned to the midmesh either on node card, element card, nodal thickness on element card, or as properties on elements or components depending on the solver profile you are using.

> **Restriction:** Only available in the OptiStruct, Radioss, Abaqus, LS-DYNA, PAM-CRASH 2G, or Nastran solver profiles.

From the menu bar, click **Mesh** > **Edit** > **Elements** > **Midmesh Thickness**.

Select Midmesh and Solid Inputs

The midmesh and solid can be specified by manually selecting entities from the current HyperMesh session, or by selecting external geometry or FE solver decks as input.

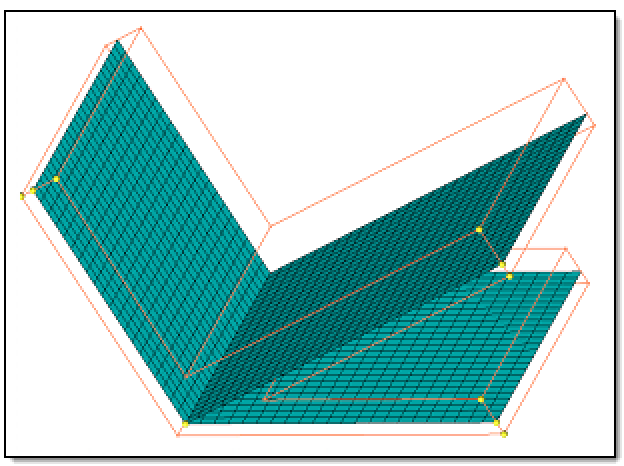


Figure 990: FE and Geometry with Traditional Element Visualization

Select midmesh and solid inputs.

To select input from	Do this
External geometry/FE solver decks	1. For input, choose **From file**. 2. For **Solid geom/mesh**, choose whether to select geometry or FE (mesh), then navigate to the file that contains the input. 3. For **Midmesh**, navigate to the file that contains the midmesh.
Existing entities	1. For input, choose **Select entities**. 2. Use the **Solid geom/mesh** selector to select input. 3. Use the **Midmesh** selector to select input.

Define Thickness Output

Calculate and assign a thickness on the midmesh.

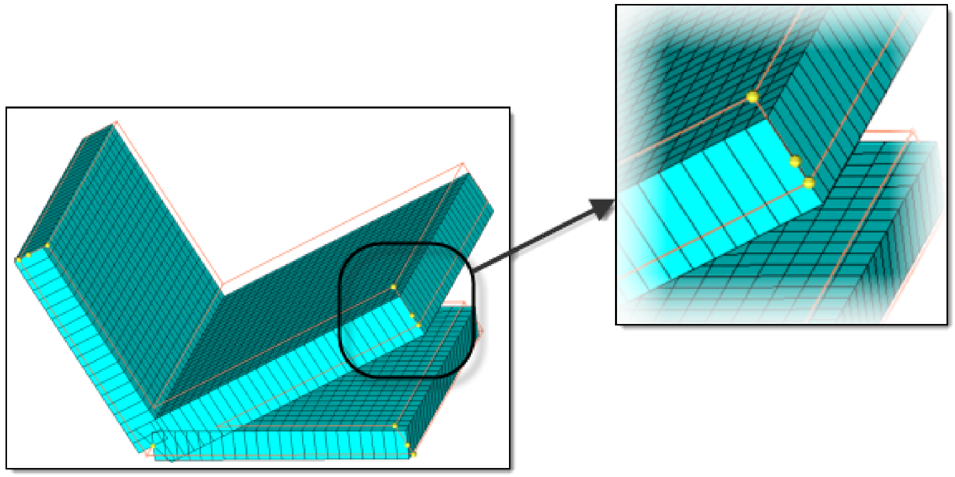
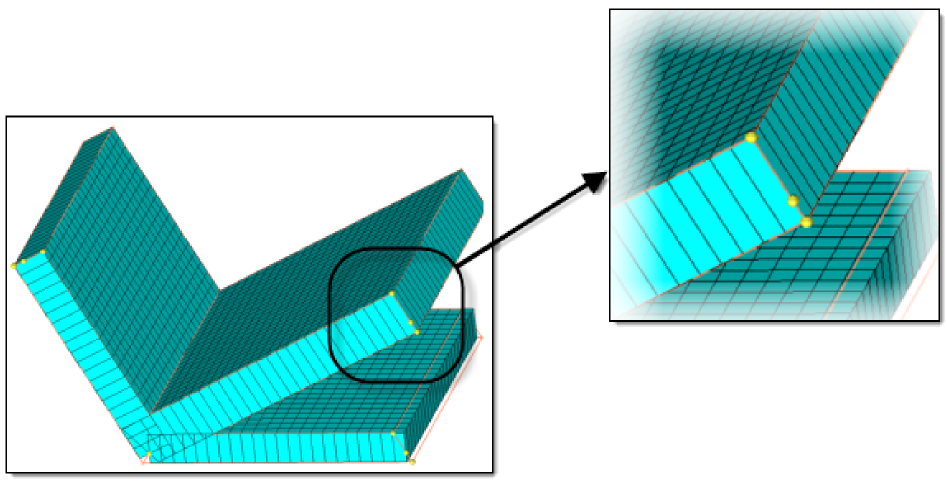
1. Select a Thickness output.

Output	Description	Solver Profile
Nodes	Assign a thickness on node cards.	Abaqus
Elements	Assign a single thickness on the element card.	Abaqus LS-DYNA PAM-CRASH 2G (TSHELL and SHELL card image) Radioss
Nodal thickness on elements	Assign multiple thicknesses on elements for each node.	LS-DYNA Nastran OptiStruct
Properties on elements	Create properties for groups of elements, and assign a thickness to each property.	Abaqus Nastran OptiStruct Samcef
Properties on components	Create components and properties for groups of elements, assign a thickness to each	Abaqus

Output	Description	Solver Profile
	property, and then assign the property to the corresponding component. The original components are duplicated, thus maintaining relevant information and attributes.	ANSYS LS-DYNA Nastran OptiStruct Samcef
Select card image Select property/section card image Select property card image	Select the card image to use for the relevant thickness assignment. This varies depending on the solver profile and the type of thickness output selected.	Abaqus ANSYS LS-DYNA OptiStruct Radioss
Components Contact thickness	Assign thickness/contact thickness on PART.	PAM-CRASH 2G
Prefix	Define the prefix for generating property names.	Abaqus ANSYS LS-DYNA Nastran OptiStruct PAM-CRASH 2G

2. Define thickness options.

Option	Action
Minimum thickness/ Maximum thickness	Define a thickness to assign calculated thicknesses. • To enter a minimum thickness to assign calculated thicknesses below a specified value, select the **Minimum thickness** checkbox. • To enter a maximum thickness to assign calculated thicknesses above a specified value, select the **Maximum thickness** checkbox.
Assign offset to elements/sections	To assign an offset value to elements if they are not in the middle of the selected geometry, select the **Assign offset to elements/ sections** checkbox.

Option	Action
	 *Figure 991: 2D Detailed Element Representation On, No Element Offset On* *The mesh deviates from the geometry; the midmesh node is equidistant from mesh corners, but not from the nodes on the geometry corners.*
	 *Figure 992: 2D Detailed Element Representation On, Element Offset On* *Offset has caused the mesh to match the geometry, though the initial midmesh node remains unchanged, such that it now matches the geometry.*
Assign average thickness to element groups	To approximate the thickness values on individual elements to be an average value representing a group of elements, select the **Assign average thickness to element groups** checkbox.

Option	Action
	Note: Only available when Thickness output is set to **Elements**.

Consider a model which has a thickness variation as shown in Figure 993. If you assign thickness on elements, by default you will get a highly variable distribution.

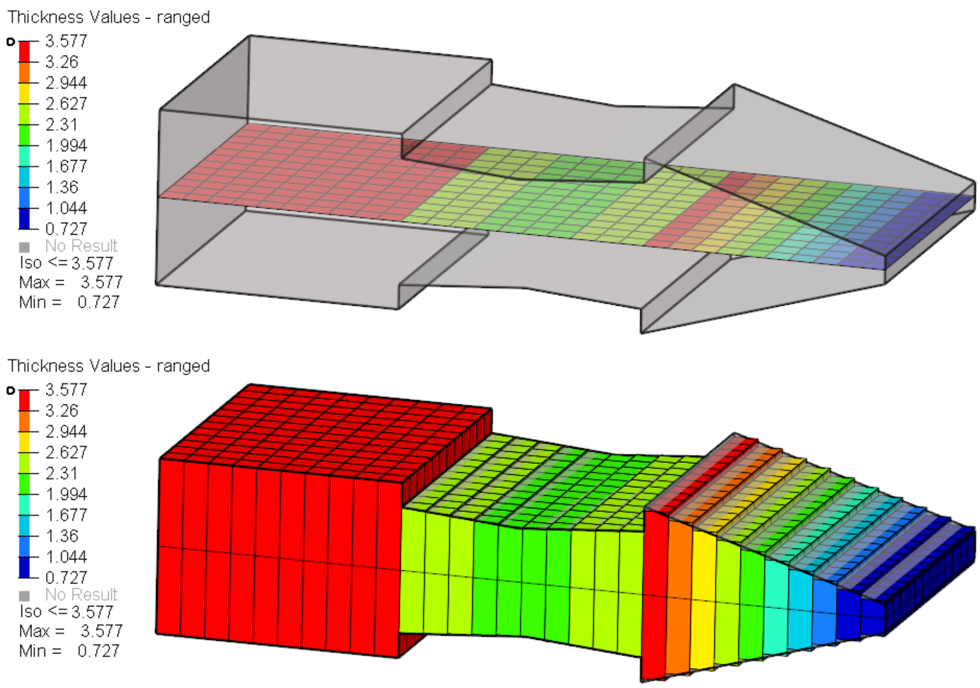


Figure 993:

Each row of elements is assigned a different thickness, resulting in various thicknesses, even if the solid was captured accurately.

Enabling **Assign average thickness to element groups** groups elements with similar thicknesses together, and assigns an average thickness to the groups. In Figure 994, the middle region of the

Option | **Action**

model is assigned a single thickness and the right section is assigned fewer steps. The accuracy of the model captured is reduced.

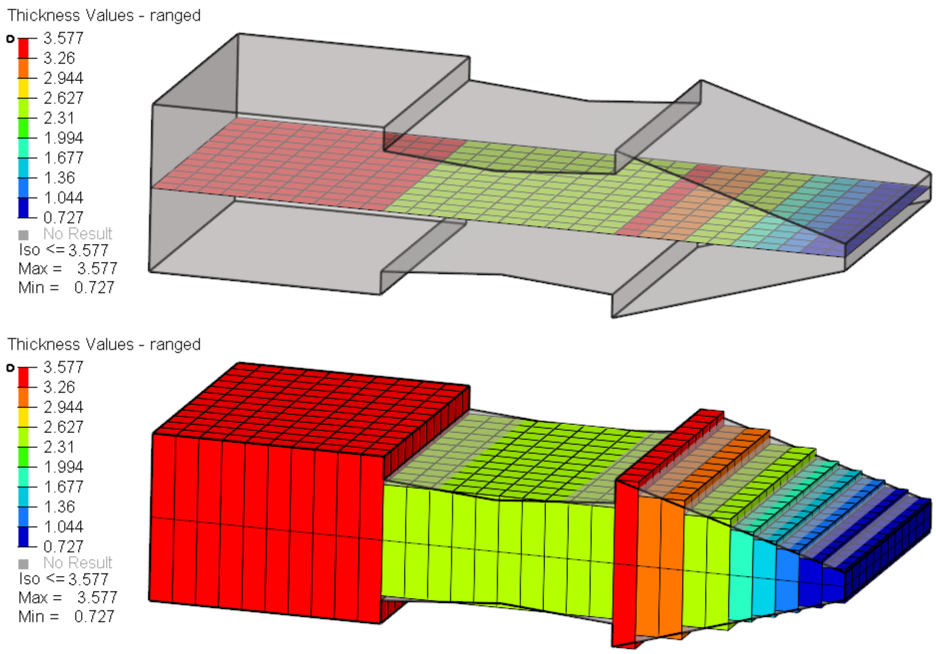


Figure 994:

> **Note:** This is the default behavior if the **Thickness output** option is set to **Properties on elements** or **Properties on components**.

Maximum thickness range interval

Control how similar thickness elements are grouped together.

- Choose **Rel** (relative interval value) to specify a relative interval value, which defines the relative width of each thickness band.

  For example, suppose you enter a Rel value of 0.2. Starting from the element with the smallest thickness, elements with

Option	Action
	similar thickness are grouped together, such that the difference between maximum and minimum thickness in one group does not exceed 0.2 times the maximum thickness in the group. The thickness values of elements in a group is averaged and assigned to all of the elements in the group (or to the property.)

- Choose **Abs** (absolute interval value) to specify a value to use as the cut off for the maximum range of the thickness in a group.

  If this value is 0.5, the difference between maximum and minimum of the thickness values in a group / property will be a maximum of 0.5.

  Smaller values result in a larger number of properties that will be created, allowing the thickness variation to be captured more accurately.

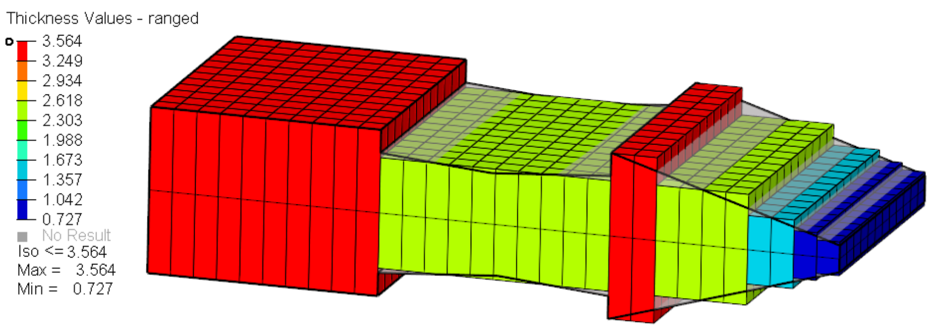
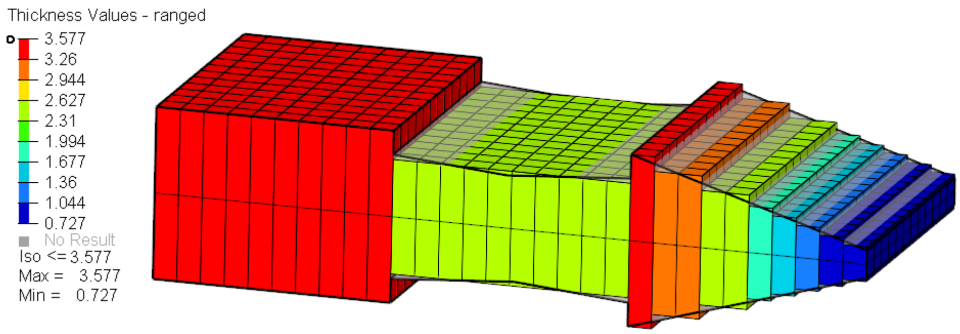


Figure 995:

Assign average thickness to element groups: On / Maximum thickness range interval: Rel = 0.5.

Option	Action
	 *Figure 996:* *Assign average thickness to element groups: On / Maximum thickness range interval: Rel = 0.2.* **Note:** Available when **Assign average thickness to element groups** is enabled, or if **thickness output** is set to **Properties on Elements** or **Properties on Components**.
Fixed interval with start thickness	Control thickness properties at constant intervals. Enter a desired thickness value in the **Start thickness** field, and a thickness interval value in the **Interval** field. All elements will be grouped into properties/components with fixed increments to the thickness specified by the interval value, beginning with the start thickness value.
Truncate thickness to range interval	Round the thickness values up or down to have the same number of decimal digits as the input value in the Maximum thickness range interval field.. There are three modes available. Automatic determines an optimal truncation.

3. To save the information logged during the computation in a specified file, select the **Save log file** checkbox.
4. Define advanced options.

Option	Action
Correction method	Select a correction method to perform on elements or nodes where the thickness could not be computed correctly. • Choose **Interpolate from Neighbors** to interpolate an appropriate thickness value from the neighboring element/node thickness. • Choose **Adjust to min/max values** to assign either minimum or maximum thickness, if they are defined, to the corrected elements.
Scaling at corners [0,10]	Enter a scaling factor to use when interpolating thickness values near t-junctions or corners. A value less than 1 will result in linear decrease in the thickness values nearer to the junction/corners. A value of zero will result in an approximate mass-conserved thickness estimation. When equal to 1, the thickness is extrapolated / interpolated without any scaling. When greater than 1, the thickness will increase as you go closer to the junction. Only available when the Correction method is set to **Interpolate from Neighbors**.

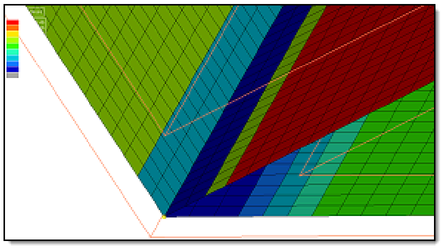


Figure 997: Scaling at corners = 0, Thickness Contour Applied, Traditional Element Visualization

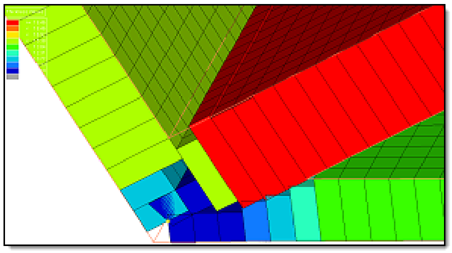
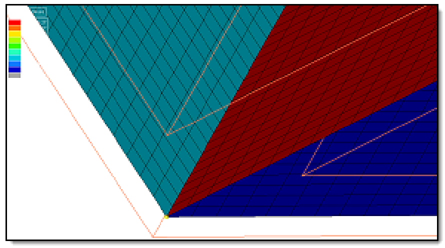
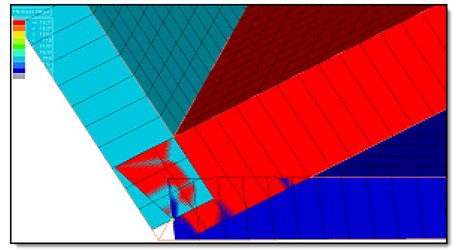
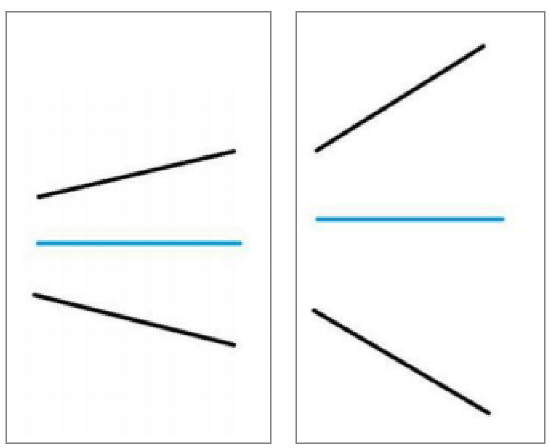
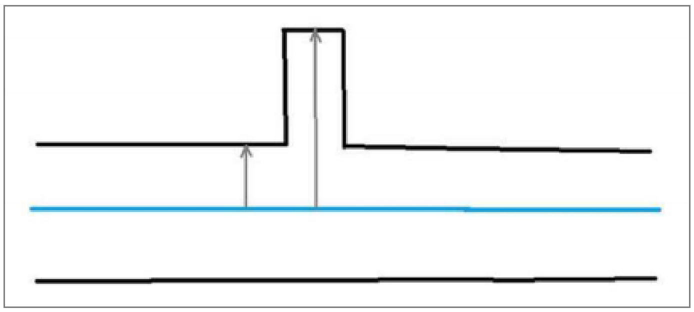


Figure 998: Scaling at corners = 0, Thickness Contour Applied, 2D Detailed Element Representation

Option	Action
	 *Figure 999: Scaling at corners = 1, Thickness Contour Applied, Traditional Element Visualization*  *Figure 1000: Scaling at corners = 1, Thickness Contour Applied, 2D Detailed Element Representation*
Max midmesh / solid angle [0.1,90]	Enter the cut-off angle in degrees. Enter the maximum angle (0 to 90) that the mid-mesh makes with the solid, beyond which the estimated thickness will be ignored and corrected. For example, the images below show the midmesh in blue, and the solid in black. In the first image the midmesh and the solid form a moderate angle. However in the second image the midmesh and the solid form a larger angle. It is possible that this area is a junction, located at the solid edge, or the solid may contain some noisy features, therefore you may want to ignore the calculated thickness and interpolate that value from surroundings.  *Figure 1001:*

Option	Action
Max thickness gradient [0, 10]	Measures the change in adjacent thickness values, and restricts the gradient. Enter the maximum change in thickness allowed across two adjacent measurement locations, as a factor of the distance between the locations. Allowable factors range from 0 to 10. For example, the image below shows a tiny projected boss in an otherwise planar area, which you may want to ignore. Defining the **Max thickness gradient** specifies the cut-off for the ratio of difference in thickness at two locations to the distance between the two locations. If the ratio is more than the given value, the measured thickness is discarded, and interpolated from its neighbors.  *Figure 1002:*
Max relative faceting error	Enter a maximum relative faceting error to use during faceting calculations. When HyperMesh computes the proximity with a solid, it does not actually compute with the surface geometry. Instead, HyperMesh computes with the tessellated approximation (facets) of the geometry. In case of curved geometry, the facets may not exactly match the geometry, and hence the thickness is computed, causing faceting errors. Specifying the **Max relative faceting error** enables you to control the accuracy of the tessellation used, as a factor of the estimated thickness. The **Max relative faceting error** is the ratio of the maximum error of facets to the estimated thickness. If it exceeds this value, facets are further refined to better capture the geometry.
Max search distance	Enter the maximum distance which will be considered for searching solid proximity. This value is also used to restrict the maximum distance from which a good thickness value is considered for correcting an incorrect thickness estimate. You can also used this

Option	Action
	option to restrict the algorithm from assigning incorrect thicknesses for the mid-mesh that is out of the solid proximity.
Do not assign thickness to midmesh outside solid	Do not assign a thickness to the midmesh outside of the solid.

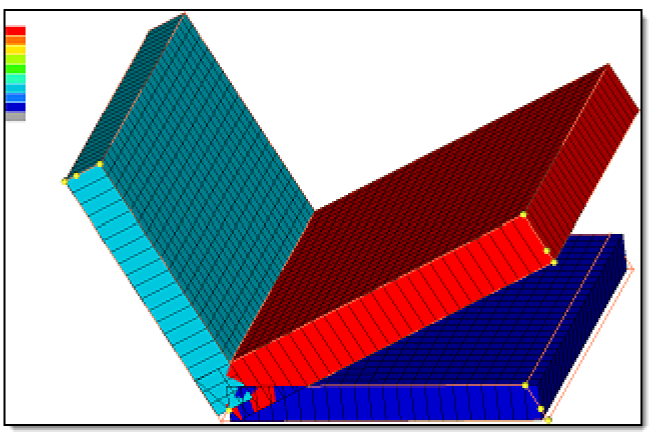
5. Click **Calculate Thickness**.

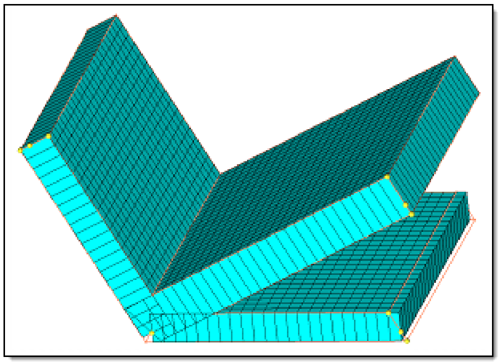
The thickness is computed and assigned on the midmesh.

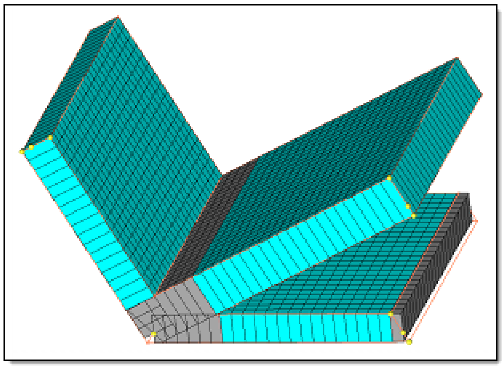
Define Visualization Settings

Define visualization settings for the midmesh.

Define visualization settings for the midmesh.

Option	Action
Element coloring by thickness	Set the **Element Color mode** to **By Thickness** resulting in a contour plot of thickness.  *Figure 1003: Thickness Contour Applied, 2D Detailed Element Representation*
2D detailed element representation	Display midmesh elements as 2D with the assigned thickness as depth.

Option	Action
	 *Figure 1004: 2D Detailed Element Representation* Assigning a thickness for each individual node or element, enables you to assign a variable thickness to elements. You can visualize both the element and nodal thickness in the graphics area by selecting **2D Detailed Element Representation** from the Visualization toolbar. The behavior of 2D Detailed Element Representation depends on what you assigned a thickness to. For example, if you assign a component an element thickness, then the element thickness representation will be displayed. If you assign a nodal thickness to an element node, then each individual node thickness will be plotted and you will be able to see the first and second order trias and quad elements. When you assign a nodal thickness, you can select the **By Thickness** option in the Visualization toolbar to display each node value. If you assign a nodal thickness, and offset is on, then the element thickness from the nodes will be calculated, the element will be offset, and the nodal thickness 2D Detailed Element representation on the offset elements will be plotted. **Restriction:** Composites with variable thickness are not added for this feature.
Highlight corrected elements	Highlight the elements whereby the thickness could not be calculated from solid, and deploy the correction method.

Option	Action
	 *Figure 1005: 2D Detailed Element Representation, Corrected Elements Highlighted*

Midmesh Thickness Assignment Behaviors

Common and solver interface specific behavior for assigning thicknesses to the midmesh.

Abaqus

In Abaqus, you can use the **Map Midmesh Thickness** tool to estimate and assign a midmesh thickness to nodes, elements, or properties. The midmesh thickness behavior is very similar amongst all of the entity types except elements. In cases of elements, tables with element IDs and thicknesses are created.

Nodes

- Creates a single node set named HM_NodalThickness with a Nodal_Thickness card image.
- You need to create a property and assign it to the selected 2D elements.
- Adds the relevant nodes to the node set.
- Assigns a single thickness to each node using the attribute ThicknessValue.

Elements

- Creates a single table named HM_ElementThicknesss with a Nodal_Thickness card image Distribution_Table.
- You need to create a property and assign it to the selected 2D elements.
- Assigns a single thickness to each element using the attribute ThicknessValue.

Properties on elements

- Based on the thickness, multiple properties with either SHELLSECTION or SHELLGENRALSECT card images are created.
- Creates corresponding materials and properties named thickness_t.

> **Note:** During creation, you are able to enter a name for the property and material.

- Assigns a property to the selected 2D elements.

Properties on components

- All the options mentioned above, are applicable for Properties on components, except the properties are assigned to components instead of elements.

ANSYS

In Ansys, you can use the **Map Midmesh Thickness** tool to estimate and assign a midmesh thickness to properties.

Properties on components

- Based on the thickness, multiple properties/sections with the REAL SET or SECTYPE card images are created.
- Creates corresponding properties/sections named thickness_t.

> **Note:** During creation, you are able to enter a prefix for the property/section.

- Assigns a property to the components.

LS-DYNA

In LS-DYNA, you can use the **Map Midmesh Thickness** tool to estimate and assign a midmesh thickness to elements or to each node on corresponding elements.

Nodal Thickness onn Elements

- If you select **Nodal thickness on elements**, a thickness is assigned to each node on corresponding elements. A thickness will be assigned to nodes that have the following THIC attributes of the *ELEMENT_SHELL_THICKNESS card: THIC1, THIC2, THIC3 and THIC4.

Elements

- If you select **Elements**, a thickness is assigned to the THIC1, THIC2, THIC3, and THIC4 fields of the card *ELEMENT_SHELL_THICKNESS. A single value thickness will be assigned to all of the thickness attributes of the element.

> **Note:** If the element is of type TRIA, then a THIC3 thickness will be assigned to THIC4. If you change the view to 2D Detailed Element Representation from the **Visualization** toolbar, then THIC4 will be ignored for TRIA elements.

Properties on Components

- Based on the thickness, multiple properties with SHELLSECTION card images are created.
- Creates corresponding materials and components named thickness_t.

> **Note:** During creation, you are able to enter a name for the property and component.

- Assign a property to the components that were created.

Write INCLUDE_STAMPED_PART

- If you select this checkbox, an include file is created for each component selected for the run. A system collector is also created with the card image includeStampedPart, and associate it to the created include file.

> **Note:** Only valid for the **Thickness output** options **Elements** and **Nodal thickness on elements**. It is not supported for **Properties on Components**.

Nastran

In Nastran, you can use the Map Midmesh Thickness tool to estimate and assign a midmesh thickness to properties or each node on corresponding elements.

Nodal Thickness on Elements

- If you select **Nodal thickness on elements**, a thickness is assigned to each node on corresponding elements. A thickness will be assigned to the T1, T2, T3 and T4 attributes.

Properties on elements

- Based on the thickness, multiple properties with either PSHELL or PCOMP card images are created.
- Creates corresponding materials and properties named thickness_t.

> **Note:** During creation, you are able to enter a name for the property and material.

- Assigns a property to the selected 2D elements.

Properties on components

- All the options mentioned above, are applicable for Properties on components, except the properties are assigned to components instead of elements.

Altair OptiStruct

In OptiStruct, you can use the Map Midmesh Thickness tool to estimate and assign a midmesh thickness to properties or each node on corresponding elements.

Nodal Thickness on Elements

- If you select **Nodal thickness on elements**, a thickness is assigned to each node on corresponding elements. A thickness will be assigned to the T1, T2, T3 and T4 attributes.

Properties on elements

- Based on the thickness, multiple properties with either PSHELL or PCOMP card images are created.

- Creates corresponding materials and properties named thickness_t.

> **Note:** During creation, you are able to enter a name for the property and material.

- Assigns a property to the selected 2D elements.

Properties on components

- All the options mentioned above, are applicable for Properties on components, except the properties are assigned to components instead of elements.

PAM-CRASH 2G

In PAM-CRASH 2G, use the **Map Midmesh Thickness** tool to estimate and assign a midmesh thickness to elements or components. You can also maintain the thickness for a contact (TCONT) to be the same as element thickness (H).

Elements

- If you select **Elements**, a thickness is assigned to the attribute T of the card SHELL and TSHEL.

Components

- If you select **Components**, a thickness is assigned to the attribute H of the card PART based on SHELL, TSHEL, and MEMBR.

Contact thickness

- If you enable the **Contact thickness** checkbox under Components, a thickness value is assigned to the TCONT attribute, which is the same as the thickness value of H of the card PART on SHELL, TSHEL, and MEMBR.

Altair Radioss

In RADIOSS, you can use the **Map Midmesh Thickness** tool to estimate and assign a midmesh thickness to elements only. This option requires you to select one of the following options from the **Select Card Image** list.

Shell

- Assigns a thickness on the thickness attribute of the /SHELL keyword for quads or on the /SH3N keyword for trias.

EPSP_F / STRA_F / STRS_F

- When you select any of these options, a table is created that contains element IDs and corresponding thickness information.

> **Note:** Depending on the type of elements in your model, there may be two tables created, one for tria elements and one for quad elements. The naming convention for tria tables will be tkTable_INISH3_{optionName}_{tableIndx}, and the naming convention for quad tables will be tkTable_INISHE_{optionName}_{tableIndx}.

For example, if you select **EPSP_F** from the **Select Card Image** list, the **Map Midmesh Thickness** tool creates and names the first table tkTable_INISH3_EPSP_F_1 and tkTable_INISHE_EPSP_F_1.

Samcef

In Samcef, you can use the **Map Midmesh Thickness** tool to estimate and assign a midmesh thickness to properties.

Properties on elements

- Based on the thickness, multiple properties with the SHELLPHP card images are created.
- Creates corresponding materials and properties named thickness_t.

> **Note:** During creation, you are able to enter a prefix for the property and material.

- Assigns a property to the selected 2D elements.

Properties on components

- All the options mentioned above, are applicable for Properties on components, except the properties are assigned to components instead of elements.

Coarsen Mesh

Simplify a mesh by combining many small elements into a smaller number of larger ones.

An existing mesh may be finer and more complex than your simulation requires. This can result in the simulation, or other utilities that depend on existing elements, taking an unnecessarily long time to run, especially when your goal is to view real-time animations for NVH (Noise, Vibration, and Handling) or similar analyses.

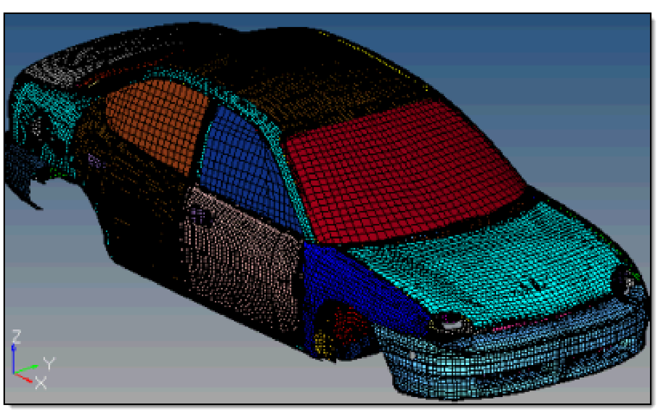


Figure 1006: Before Coarsening

The mesh size is 30.

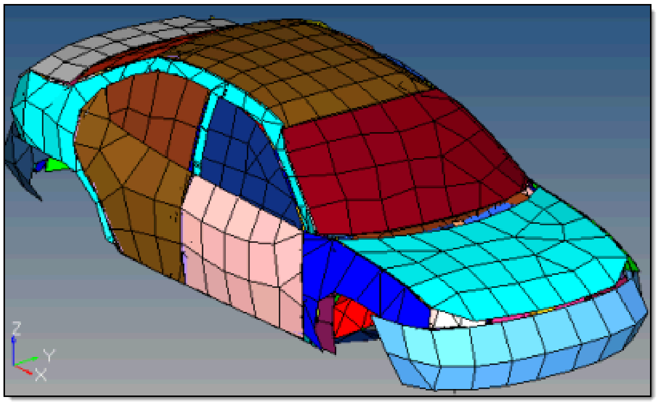


Figure 1007: After Coarsening

The mesh size is 200.

When a mesh is coarsened like this, it is important to note that every node in the coarse mesh corresponds exactly to one of the nodes in the original mesh, although many nodes are removed, the ones that remain are still the same nodes from the original model. No locations or qualities are changed. Similarly, any nodes or points with special information, such as comments, in the 10th column of their deck will be preserved.

It is best to filter out components that are not relevant to your analysis. In the example above, the wheels and suspension were removed from consideration.

1. From the menu bar, click **Mesh** > **Create** > **Coarsen Mesh**.
   The **Coarsen Mesh** dialog opens.
2. Use the **Select entities: Components** selector to select the components to simplify.
3. Use the **Select hard points: Nodes** selector to select any hard points, such as those defining a hole or ridge, that must be preserved.
4. In the **Element size** field, enter a new element size.
5. To prevent the coarsened mesh from exceeding a target number of nodes, select the **Maximum Node Count** checkbox.

   This is an iterative process and will adjust the mesh size you specified accordingly.

   > **Tip:** This option is helpful when you have large models and want to better control the size of the output for NVH models. The maximum node count will apply to all of the selected components. If you would like greater control of a specific component, coarsen that component separately.

6. For **Mesh type**, select a mesh type.
7. To eliminate and mesh over features with angles smaller than the specified value, while preserving features with angles greater than the specified value so that the mesh aligns with the feature line rather than allowing elements to cross over it, select the **Feature angle** checkbox.

   The coarsening process uses two stages; if the first stage fails on some elements, the second stage is run. The feature angle setting only applies to the second stage; it is irrelevant to the initial stage.

8. To mesh over holes smaller than the specified value, but preserve holes that are at least the specified value, select the **Minimum fill hole diameter** checkbox.
9. To remove most features that become detached during the coarsening process, which are underneath the target input size that you specified, select the **Delete small parts** checkbox.

   The entire feature must be within this limit. For example, a free bolt hole that is 3 units in diameter but 25 units in length will remain if this amount is less than 25.
10. Define advanced options.

Option	Description
Delete 1Ds before meshing	Consider any 1D elements that are part of the input selection for this operation. If those 1D elements are attached to a hard point or are part of a 1D path back to the 2D/3D structure, those elements are not deleted. Any other 1D elements that are part of the input selection are deleted
Retain input mesh	Generate a new component containing the coarsened elements, leaving the original component and its elements untouched. The coarsened elements will share nodes with the original input mesh.
Delete masses before meshing	Consider any 0D elements that are part of the input selection for this operation. If those 0D elements are attached to a hard point or are part of a 1D path back to the 2D/3D structure, then those elements are not deleted. Any other 1D elements that are part of the input selection are deleted.
Delete free 1Ds and masses after meshing	The coarsening operation itself includes making sure the relevant rigidlink/RBE3/mass elements are connected back to the structure accordingly. For this option rigidlink/RBE3/mass elements are considered free only if all legs are free after coarsening/ reconnection is complete.
Delete free rigidlink/ RBE3 legs after meshing	Delete any free legs after coarsening and reconnection are complete. As mentioned above, a rigidlink/RBE3 element is only free if all legs are free for the purpose of this tool.
Convert existing 1D to PLOTELs	Convert remaining 1D elements that were part of the input selection to PLOTEL, once all of the above rules have been completed. This includes all legs of rigidlink/RBE3 elements. Every leg of these elements needs to be converted to a separate PLOTEL (for example, a 10 leg RBE3 will be 10 PLOTELs).
Create PLOTELs along 2D edges	Create PLOTELs along all 2D element edges and 2D mesh will be deleted. If **Convert coarsened 2Ds to PLOTEL3/4** is enabled, then PLOTEL3/4 are also created.

Option	Description
Convert coarsened 2D to PLOTEL 3/4	Convert all Tria and Quad elements to PLOTEL3 or PLOTEL4 element types configurations. **Restriction:** Only available in the OptiStruct user profile.
Component size factor	Component size factor to convert.
Auto reduce factor	Auto reduce factor to convert.

11. Click **Mesh**.
    The selected components are meshed.

If the results are not satisfactory, you can Reject the new coarse mesh, change the options, and try again.

Connect Intersecting 2D Elements

Connect intersecting triangular and quadrilaaterial 2D elements.

Several components\elements can be connected simultaneously. Components defined as master entities will be preserved. It is suggested that you ensure that 2D elements are properly intersecting.

1. From the menu bar, click **Mesh** > **Boolean Operation**.
   The **Boolean Operation** dialog opens.
2. Choose the type of boolean operation to perform.
   - Choose **Union**.

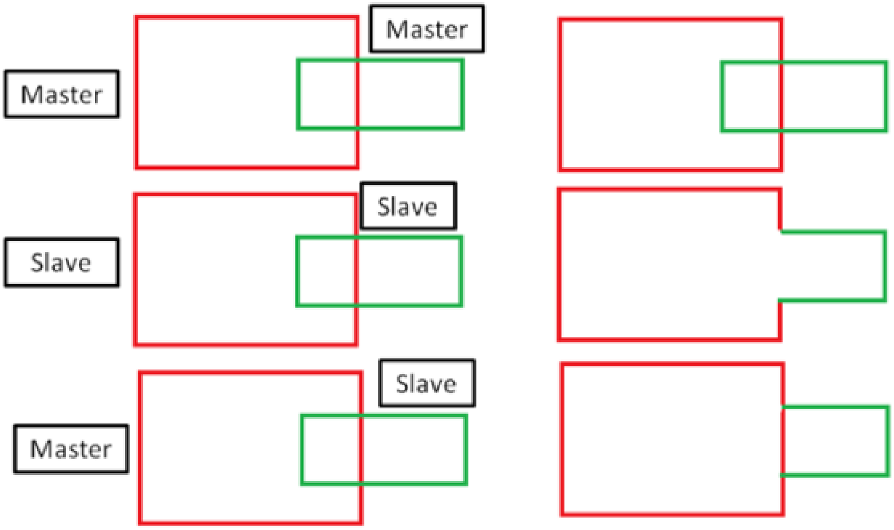


Figure 1008: Union

- Choose **Subtract**.

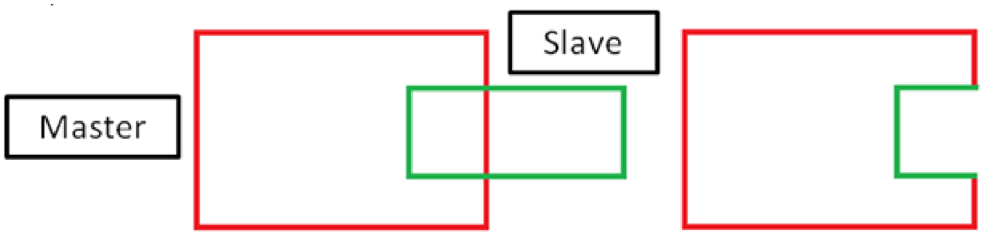


Figure 1009: Subtract

3. Use the All entities to Boolean selector to select all components or elements that will be involved in the boolean operation.

   Entities defined that are not present in master entities to preserve are considered as slaves. If you define all entities for boolean as master entities to preserve, entities will intersect and connect components but will not remove any internal shells. The boolean operation will only remove internal shells if they are enclosed and if they belong to an entity not defined as master entities to preserve. If both of these conditions are not satisfied the internal shells will not be removed.
4. Use the Master entities to preserve selector to select components that you do not want to remove internal shells for.
5. If you have components with open shells but they are connected to other components and create closed shells together, select the **Consider connected components as one shell** checkbox to consider the closeness of the connected shells while deciding which internal shells will be removed.
6. To create connection edges, select the **Keep intersect edges** checkbox.
7. To remesh elements near the intersection of inputs based on the defined parameters, select the **Local remesh at contacts** checkbox.

In a case of tria element input, tria elements of first order will be remeshed. In a case of quad elements input, remeshing will be done with mixed elements.

8. In the Number of layers field, enter the number of additional layers next to the intersect edge to remesh.
9. In the Feature angle field, enter the feature angle to use for the remeshed elements to be captured.
10. In the Growth rate field, enter a growth rate to use for remeshing.
11. Click **Run**.

Fuse Mesh

Connect close proximity, overlapping and intersecting parts.

Input assemblies can have intersecting/partial intersecting parts, overlapping parts, or close proximity parts.

Use the fuse tool to connect close proximity, overlapping, and intersecting parts.

1. Open the Fuse tool by clicking **Mesh > Fuse** from themenu bar.
2. Use the Source entities selector to select source entities.

   Source entities will move to the target.
3. Use the Target entities selector to select target entities.

   Target entities will not move.
4. Decide whether to perform Open Shell or Close Shell fusing.

   The fuse algorithm treats open shells (not closed input) and closed shells (water tight shells) differently.
5. Define additional Open Shell and Close Shell fusing options accordingly.

   Options vary based on the fusing type selected.
6. Click **Fuse** to start the fuse operation.

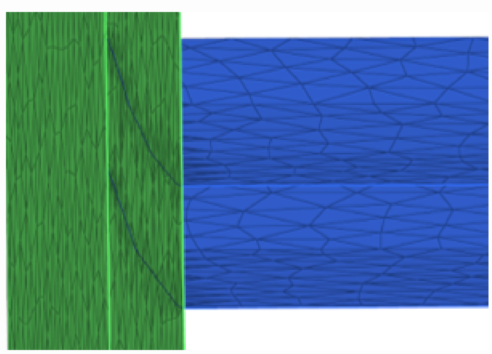
Open and Close Shell Fusing Options

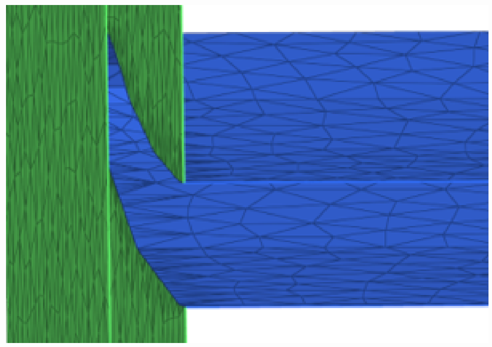
The following tables list options and actions for the **Mesh > Fuse** dialog box.

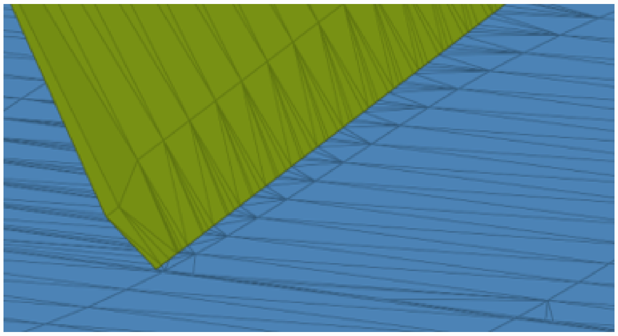
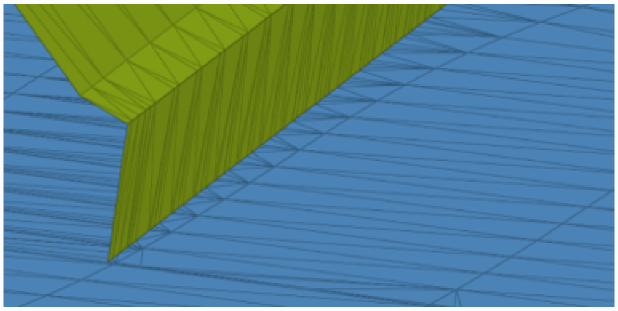
Open Shell Fusing	
Option	**Action**
Proximity Search Threshold	Type the maximum search distance to find proximity elements between source and target into this field. Fusing will be performed between them.

Open Shell Fusing	
Option	**Action**
	Note: You are not required to enter a precise distance, but it is recommended that you do not define too big of a value compared to the average distance between the source and target in order to avoid performance issues.
Perform Remesh	Select this checkbox to remesh elements near the connection based on the user-defined parameters. Triangle element inputs of the first order will be remeshed. Quad element inputs will be remeshed with mixer elements.
Feature Angle	Type the feature angle for the remeshed elements to be captured into this field.
Connect Source Free Edge Only	Select this checkbox to connect only the free edges within the user-defined proximity threshold.
Fusing Direction	Select a direction to which the source will be projected to the target: • **Along Source Normal:** Choose this option to project source elements to the target based on the source element normal or the shortest distance direction from the source. • **Along Source Edge Tangent:** Choose this option to project proximity source elements in a tangential direction to the source. Tangency is only calculated for free edges. For other inputs, it will project to the target based on shortest distance direction from source.
Create Patch at Connection	Select this checkbox to create new patch elements between source and target. If this option is unchecked, source element nodes will be projected and stitched to the target.
Remove Redundant Patches from Target	Select this checkbox to remove redundant patches after fusing is performed that are within the user-defined patch maximum width factor. The value of the maximum width factor is calculated by multiplying the factor with the proximity search threshold.
Snap to Features	Select this checkbox to fuse source elements to target elements if the features fall within the user-defined feature snapping tolerance factor. The value of the snapping tolerance

Open Shell Fusing	
Option	**Action**
	is calculated by multiplying the factor with the proximity search threshold. This feature includes free edge, non-manifold edges (t-connection / x-connection), and features based on the feature angle defined in preferences.

Close Shell Fusing	
Option	**Action**
Proximity Search Threshold	Type the maximum search distance to find proximity elements between source and target into this field. Fusing will be performed between them. **Note:** You are not required to enter a precise distance, but it is recommended that you do not define too big of a value compared to the average distance between the source and target in order to avoid performance issues.
Keep Interface	Select this checkbox to keep common shells between source and target as they are. Uncheck to remove common shells.  *Figure 1010: Keep Interface Enabled*

Close Shell Fusing	
Option	**Action**
	 *Figure 1011: Keep Interface Disabled*
Maximum Search Angle	Type the maximum angle value that is used to detect close proximity between the source and target into this field.
Fusing Direction	Select a direction to which the source will be projected to the target: • **Along Source Normal:** Choose this option to project source elements to the target based on the source element normal or the shortest distance direction from the source. • **Along Source Edge Tangent:** Choose this option to project proximity source elements in a tangential direction to the source. Tangency is only calculated for free edges. For other inputs, it will project to the target based on shortest distance direction from source.
Remove Redundant Patches from Target	Select this checkbox to remove redundant patches after fusing is performed that are within the user-defined patch maximum width factor. **Note:** This is only useful in cases of open shells.
Collapse Patches	Select this checkbox to collapse created patches and directly connect the source nodes to the target nodes. When enabled, you can select a destination component for patch: Source Component, Current Component, or New Component.

Close Shell Fusing	
Option	**Action**
	 *Figure 1012: Collapse Patches Enabled*  *Figure 1013: Collapse Patches Disabled*
Perform Remesh	Select this checkbox to remesh elements near the connection based on the user-defined parameters. Triangle element inputs of the first order will be remeshed. Quad element inputs will be remeshed with mixer elements.
Number of Layers	Type the number of additional layers next to the intersect edge that need to be remeshed into this field.
Feature Angle	Type the feature angle for the remeshed elements to be captured into this field.

Close Shell Fusing	
Option	**Action**
Growth Rate	Type the growth rate for remeshing into this field.

Fuse Mesh Examples

The Fuse tool can be used with several types of analyses where mesh connections are required.

CFD Fluid Analysis Model Preparation

Create a water tight, fluid volume for CFD analysis by connecting shell and solid parts.

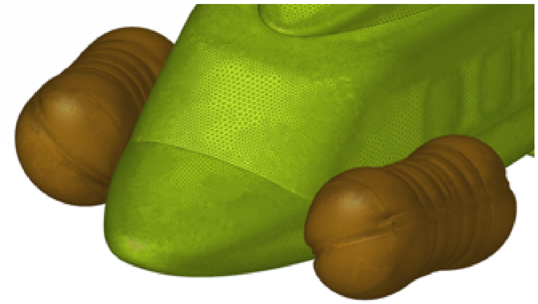


Figure 1014:

Thermal Analysis Model Preparation

Prepare the model for thermal analysis by connecting closed shells across the assembly.

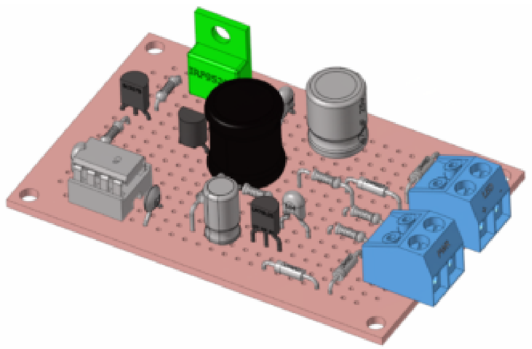


Figure 1015:

Electromagnetic Analysis Model Preparation

Define connections across part assemblies by connecting the midmesh of shell parts.

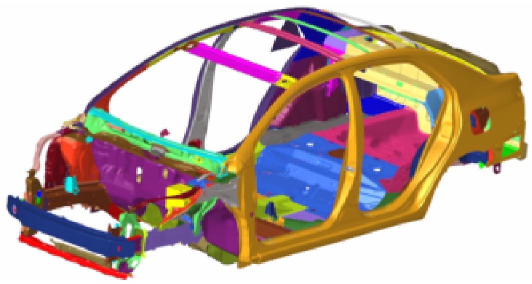


Figure 1016:

Fill Holes, Gaps, Patches

Fill holes, gaps, and patches in first order and second order elements.

1. From the menu bar, click **Mesh** > **Hole/Gap Fill**.
   The **Hole/Gap Fill** dialog opens.
2. Select a Fill Type.
   - Choose **Hole fills** to fills holes and edge loops. Holes can be free edge loops or feature edge loops.

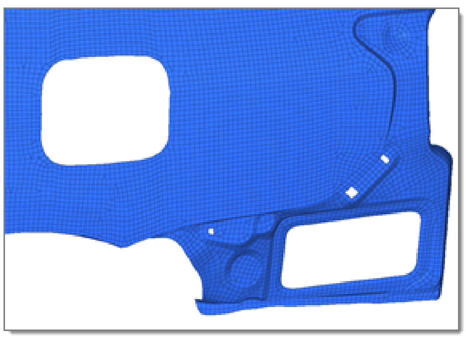


Figure 1017: Free Edge Holes

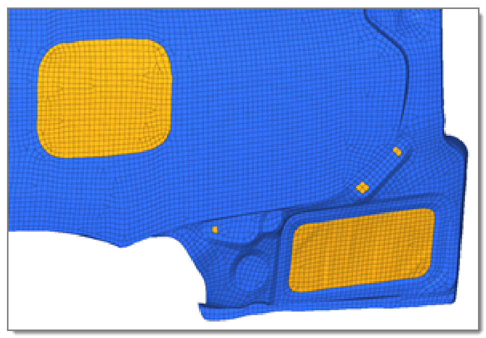


Figure 1018: Free Edge Holes Filled

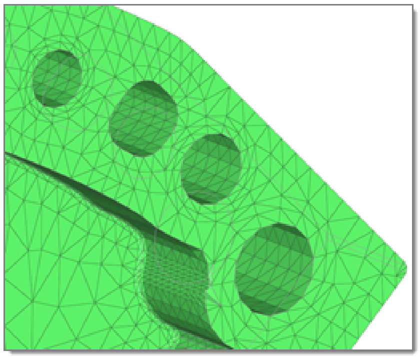


Figure 1019: Feature Edge Holes

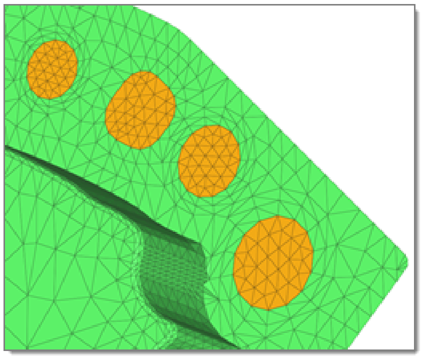


Figure 1020: Feature Edge Holes Filled

- Choose **Gap fills** to fill the gaps between a set of elements.

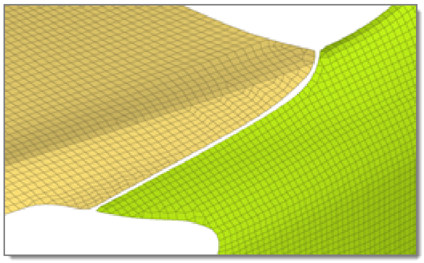


Figure 1021: Gap between Element Sets

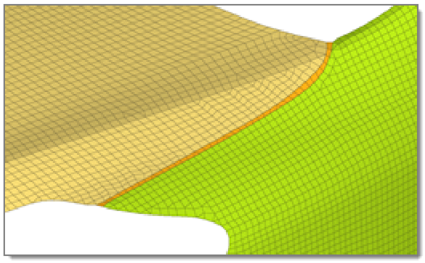


Figure 1022: Gap Filled between Element Sets

- Choose **Patch fills** to fill partial edge loops; fills gaps and holes as a pre step for wrapper, where a non-conformal patch can be created to provide a proper input to wrapper.

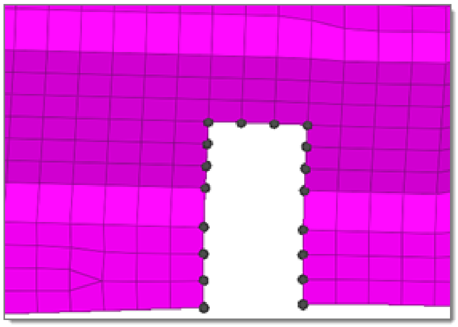


Figure 1023: Partial Edge Loop

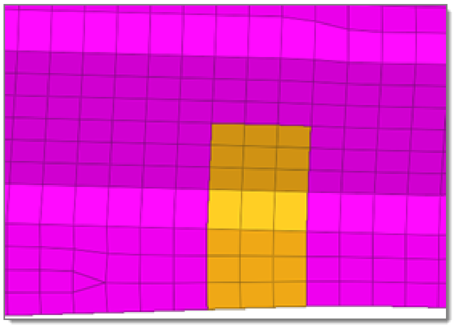


Figure 1024: Partial Edge Loop Filled

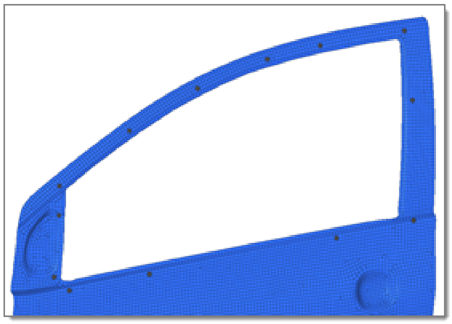


Figure 1025: Gaps and Holes

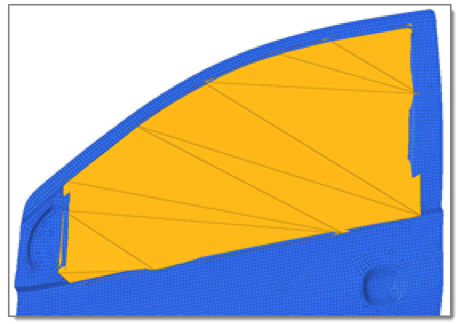
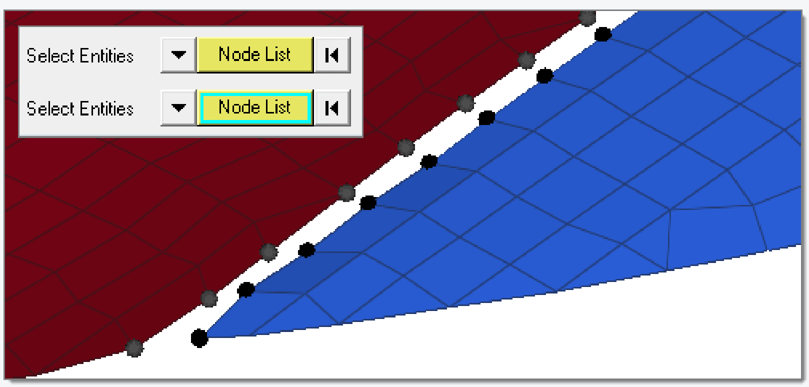


Figure 1026: Gaps and Holes Filled (Non-Conformal Patch)

3. Use the selector(s) to select the entities which surround holes, gaps, or patches to fill.

For gaps use the first selector to select entities along one side of the gap, then use the second selector to select entities along the other side of the gap.

> **Note:** Selection options change based on the Fill Mode selected.
>
> 
>
>
> *Figure 1027: Node Lists to Detect the Gap*

4. In the Max width field, enter the maximum hole width or gap width to fill.

Holes/gaps that have a width greater than the specified value will not be filled. Only available for Hole Fill and Gap Fill. In gap fill when node lists is selected, the maximum gap will try to be detected based on the average distance between selected node groups.

5. Choose a destination component.

- Choose **Current Component** to organize newly created patches in the current component. If no current component is defined, a new component "auto" will make created and set as the current component.
- Choose **Adjacent Component** to organize the filled/patched elements to the adjacent components. If the hole or gap is shared by more than two components, then newly created

elements will be organized in the component with the higher number of nodes around hole/gap.

- Choose **New Component** to organize newly created patches in a new component named ^patch.

6. Define additional options.
    - To remesh the hole/gap/patch around the surrounding elements, select the **Remesh** checkbox.

      During remeshing, element size, type, and order is determined by the surrounding element's size and type.
    - To mesh/remesh feature loops in holes/gaps, select the **Consider Feature** checkbox.

      While filling holes, feature holes are also consider and filled. In the Auto detect fill mode, filled feature/free edge holes create intersecting elements with their neighbors, therefore they are discarded and the operation is marked as failed.
    - To fill holes/gaps based on neighboring elements normal, select the **Curved Fill** checkbox.
7. Use the Guide Node Pair selector to select two nodes to determine along which direction to fill the hole or gap.

   This is useful when the dimensions of a hole/gap are in two different directions.
8. Click **Fill**.

.

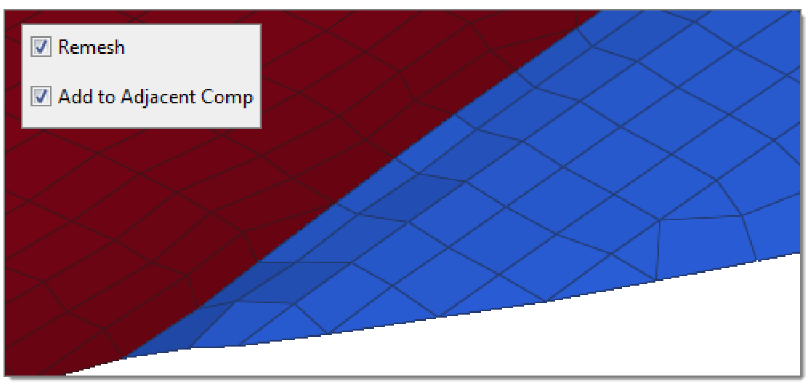


Figure 1028: Gap F and Remeshed with Adjacent Components

Batchmesher

Batchmesher is a tool that performs geometry feature recognition, cleanup and automatic meshing in batch mode for given CAD files.

Batchmesher can read geometry files and perform a variety of geometry cleanup operations to facilitate better mesh creation for the selected element size and type. Cleanup operations include, but are not limited to equivalencing free edges, fixing small surfaces, relative to the element size, and detecting features such as beads, fillets and flanges. Batchmesher also performs surface editing/defeaturing operations like removing pinholes smaller than a specified size, removing edge fillets, and adding washer layers around holes.

Batchmesher uses criteria set by you to determine the quality index (QI) of a model, uses this QI rating to assess the potential value of each geometry cleanup and meshing tool, and then applies the tools accordingly. QI optimized meshing and node placement optimization are performed to obtain the best quality meshing. Final results are stored in a HyperMesh binary database file containing both the cleaned-up geometry and the resulting finite element mesh.

The required inputs are set within a parameter file and a criteria file. The parameter file contains the average element size and type, as well as any special handling of geometry features. The criteria file contains the target element quality requirements for tests like Jacobian, warpage, and more.

User-defined Tcl procedures can also be supplied to perform both run-based (pre-run, post-run) as well as model-based (pre-geometry load, post-geometry load, pre-batchmesh, post-batchmesh) customizations.

On output, Batchmesher creates a unique directory for each run in the results directory where it stores output files. The directories are named bm_date_001, bm_date_002, and so on.

For each CAD/HM input file, there are several output files generated:

`modelname_critername_paramname.hm`
: This is the main output of Batchmesher and contains both the cleaned-up geometry and the resulting finite element mesh.

`modelname_critername_paramname_res.txt`
: This is a text file that reports the progress and status of Batchmesher at various steps in the meshing process. It reports information such as the number of surfaces (total, unmeshable, and so on), the number of elements, the percentage of trias, the mesh QI value, and so forth. COMPLETE at the end of this file indicates successful completion of the Batchmesher process for the model.

For each Batchmesher run, there are also several output files generated:

`run_results.txt`
: This is a text file that reports the progress and status of the jobs submitted to the batch meshing process. It reports the number of jobs submitted, any waiting in the queue, whether the job is complete, and similar details. For completed jobs, it provides summary information such as the time taken to complete the job, the number of surfaces in the model, the number of elements created, and so forth.

RunView.log

This file maintains a log of submitted runs. This `.log` file can be loaded into the Batchmesher GUI to review the results at a later time.

In addition to the files mentioned above, additional output files may be created due to customization procedures performed at various stages of the Batchmesher process.

Start Batchmesher

Batchmesher can be started on various platforms.

Start Batchmesher on Windows or Linux

Start Batchmesher on Windows

Start Batchmesher on Windows from the Start menu or system command prompt.

- Start Batchmesher from Start menu.

  Startup options can be supplied to the application by modifying the shortcut, or creating a new one, and adding the required options.

  a) From the Start menu, select **BatchMesher 2019**.
- Start Batchmesher from command prompt.
  a) Launch the system command prompt.
  b) `cd` to the directory from which the Batchmesher application should be run.
  c) Enter the full path of the Batchmesher application start script, along with any startup options.

     For example, enter `<altair_home>\hm\batchmesh\hw_batchmesh.bat`.

Start Batchmesher on Linux

1. Launch the system terminal.
2. `cd` to the directory from which the Batchmesher application should be run.
3. At the prompt, enter the full path of the Batchmesher application start script, along with any startup options.

   For example, enter `<altair_home>/altair/scripts/hw_batchmesh`.

Batchmesher Startup Options

Startup options allow Batchmesher to be launched with a specific behavior or with specific settings defined.

The options are provided as command line options to the application startup script.

For example, on Windows:

```
<altair_home>\hm\batchmesh\hw_batchmesh.bat -nogui -cad_translator catia -cad_model_dir
        C:\work\models -cad_model_ext CATPart -criteria_file C:\work\10mm.criteria -
param_file
          C:\work\10mm.param -user_procedure PRE_GEOMETRY_LOAD C:\work\mytcl.tcl
        myprocedure "myarg1 myarg2" -user_procedure POST_BATCHMESH C:\work
\NastranOutput.tcl
        nastranexport
```

```
<altair_home>\hm\batchmesh\hw_batchmesh.bat -config_file C:\work\bm.cfg -multicpu 4
        -batch
```

For example, on Linux:

```
<altair_home>/altair/scripts/hw_batchmesh -nogui -cad_translator catia -cad_model_dir
        /work/models -cad_model_ext CATPart -criteria_file /work/10mm.criteria -
param_file
          /work/10mm.param -user_procedure PRE_GEOMETRY_LOAD /work/mytcl.tcl
        myprocedure "myarg1 myarg2" -user_procedure POST_BATCHMESH /work/
NastranOutput.tcl
        nastranexport
```

```
<altair_home>/altair/scripts/hw_batchmesh -config_file /work/bm.cfg -multicpu 4
      -batch
```

Batchmesher GUI Startup Options

-batch

Run jobs automatically via a config file.

> **Restriction:** Must be used with the -config_file option.

-config_file <path>

Specifies the full path and file name of the config file to use.
If used without -batch, this configures the Batchmesher GUI.
If used with -batch, this configures the Batchmesher GUI and automatically runs the jobs.

-file_wait_timeout <minutes>

Model loading and PRE_GEOMETRY_LOAD user procedure run time.
May be required if some models need more than the default time (20 minutes) to load. Also may be required if the user specified PRE_GEOMETRY_LOAD procedure has a long run time. See also -timeout_scale.

-help

Print out the Batchmesher usage message.

-multicpu <number>

Number of CPUs to use for simultaneous Batchmesher jobs.
Overrides any value set in -config_file.

-relocate_to_input

Move all `.hm` output files to the corresponding directory of the input model.

-submit_time <time>

Define the future submission time of the current GUI configuration.
Time format depends on the OS and localization settings on the machine. Refer to the value of the Submit At button on the Batch Mesh tab to see an example of the format required for your environment. An example is "8/8/11 12:27:00 PM".

-time_limit_default <minutes>

Model batch meshing step timeout.

May be required if some steps need more than the default time (15 minutes). The actual timeout is the maximum of -time_limit_default and the value read from the time_limit.txt file (updated automatically after each step). This is ignored if <=0. See also -timeout_scale.

-timeout_scale <scale>

Proportionally increase/decrease -file_wait_timeout, -time_limit_default and the value read from the `time_limit.txt` file.

Default value is 1.0. Ignored if <=0.0.

-total_timeout <minutes>

Total timeout for whole batch mesh job.

May be required if model batch meshing require more than the default time (240 minutes).

-work_dir <path>

Specify the directory where the output files are to be written.

Batchmesher "Watchdog" Startup Options

This mode runs a single Batchmesher job with no GUI. This is useful for grid computing job submission.

-cad_model_dir <directory>

Specify the directory where CAD/HM input files are located. This is used for multiple file jobs.

> **Restriction:** Must be used with the -cad_model_ext option, and cannot be used with the -cad_model_file option. This is a mandatory argument if -cad_model_file is not specified.

-cad_model_ext <extension>

Specify the file extension to use when scanning the -cad_model_dir for CAD/HM input files. All input files found with this extension will be batch meshed. This should be simply the extension, not including the period (.) For example, CATPart instead of .CATPart.

> **Restriction:** Must be used with the -cad_model_dir option, and cannot be used with the -cad_model_file option. This is a mandatory argument if -cad_model_file is not specified.

-cad_model_file <path>

Specify the full path and file name of the CAD/HM input file.

> **Restriction:** Cannot be used with the -cad_model_dir and -cad_model_ext options. This is a mandatory argument if -cad_model_dir is not specified.

-cad_translator <type>

Specify the CAD file type being used.

Valid values include:

acis

catia

ct-ug

Detect
dxf
hm
iges
inspire
intergraph
jt
pdgs
parasolid
proe
step
solidworks
tribon
ug
vdafs

-criteria_file <path>
Specify the full path and file name of the criteria input file. This is a mandatory argument.

-file_wait_timeout <minutes>
Model loading and PRE_GEOMETRY_LOAD user procedure run time. May be required if some models need more than the default time (20 minutes) to load. Also may be required if the user specified PRE_GEOMETRY_LOAD procedure has a long run time. See also -timeout_scale.

-help
Print out the Batchmesher usage message.

-nobg
Force Batchmesher to run in the foreground on Linux.

-nogui
Run Batchmesher in no GUI mode. This is a mandatory argument.

-param_file <path>
Specify the full path and file name of the parameter input file. This is a mandatory argument.

-recurse <value>
A true|false value that specifies whether to include sub-directories when using the -cad_model_dir option.

> **Tip:** Not recommended for use in grid computing job submission.

-qi_post_procedure <value>
A true|false value that specifies whether to generate the HTML quality reports for the run.

-run_results <path>
Specify the full path and file name of the results output file to write/append to.

-run_tcl_file <path>

Specify the full path and file name of the Tcl script containing a procedure to run. If this option is used, then all other options except -run_tcl_proc are ignored.

> **Restriction:** Must be used in conjunction with the -run_tcl_proc option.

-run_tcl_proc <proc>

Specify the name of the Tcl procedure to run.

> **Restriction:** Must be used in conjunction with the -run_tcl_file option.

-time_limit_default <minutes>

Model batch meshing step timeout. May be required if some steps need more than the default time (15 minutes). The actual timeout is the maximum of -time_limit_default and the value read from the `time_limit.txt` file (updated automatically after each step). This is ignored if ≤ 0. See also -timeout_scale.

-timeout_scale <scale>

Proportionally increase/decrease -file_wait_timeout, -time_limit_default and the value read from the `time_limit.txt` file.
Default value is 1.0. Ignored if ≤0.0.

-total_timeout <minutes>

Total timeout for whole batch mesh job. May be required if model batch meshing require more than the default time (240 minutes).

-user_procedure <type> <path> <proc> <args>

Specify user-registered procedures. This option can be used multiple times, once for each type.

type

Type of procedure to register.
Valid values are PRE_GEOMETRY_LOAD, PRE_BATCHMESH, POST_BATCHMESH.

path

Full path and file name of the Tcl script containing a procedure to run.

proc

Name of the Tcl procedure to run.

args

List of arguments to pass into the procedure. This can be empty. If there are multiple arguments, they must be enclosed in quotes.

-work_dir <path>

Specify the directory where the output files are to be written.

Enable Grid Computing

Batchmesher supports grid-based computing.

The default grid is "PBS Pro".

By default, the "Grid" option is disabled in Batchmesher's base configuration (loaded from the `<altair_home>\hm\batchmesh\hw_batchmesh.cfg` file).

1. Enable Batchmesher grid computing by selecting **File** > **Load Config** from the Batchmesher menu bar, and loading the `<altair_home>\hm\batchmesh\hw_batchmesh_grid.cfg` file.
   After loading `hw_batchmesh_grid.cfg`, a new Grid option displays alongside Local in the File menu's Run Options sub-menu. Once this option is activated, it will remain even if you load subsequent configuration files.
2. To use the Grid option, you must configure the `qsub.tcl`, `qstat.tcl`, and `qdel.tcl` default scripts.

   All default scripts are located in `<altair_home>\hm\batchmesh`.

   The exact script configuration depends on the grid system you use, and requires detailed knowledge of your current grid system.

Grid Computing Default Scripts

Overview of the default scripts used for grid computing.

The three default scripts were created for use with Unix PBS Pro clusters and will work without modification if your cluster configuration is similar to the default configuration.

qsub.tcl

```
qsub.tcl -batch_args {args} -work_dir dir
```

The `qsub.tcl` script creates a node-side script and submits the job to the computing grid. If an error is encountered at job submission, this script returns the word "error". Otherwise, it returns the unique JobID for the submitted job.

args
: Command line for one Batchmesher job, contained in curly braces. This line is created by the Batchmesher GUI, and has to be written to the node-side script.

dir
: Specify the directory where the output files are to be written.

```
qsub.tcl -batch_args
        {/soft/hw/altair/scripts/hw_batchmesh -nogui -cad_translator hm -criteria_file
        /homes/username/configs/nvh10.criteria -param_file /homes/username/configs/
nvh10.param
        -cad_model_file /homes/username/models/model.hm -nobg} -work_dir
        /homes/username/results/bm_060209_001/
```

Returns JobID: 1234

Figure 1029: Example: qsub.tcl

qstat.tcl

```
qstat.tcl JobIDList
```

The `qstat.tcl` script obtains status information for jobs with specified JobIDs. It returns a list of JobIDs paired with status mnemonics.

R
: Job is running

Q
: Job is queued, eligible to run

E
: Job is exiting after having run

W
: Job is waiting for idle resource

U
: Status undefined (if status not R, Q, E or W)

none
: Information about job was not found

The list of unique JobIDs for submitted jobs:

```
qstat.tcl 1234 1235 1236 1239
```

Returns JobID status list of:

```
1234 none
1235 R
1236 R
1239 Q
```

Figure 1030: Example: JobIDList

qdel.tcl

```
qstat.tcl JobIDList
```

The `qdel.tcl` script terminates jobs with specified JobIDs. It returns 0 if the jobs terminated without errors or "none" if the jobs cannot be terminated or there was a termination error.

The List of unique JobIDs for submitted jobs:

```
qdel.tcl 1234 1235
```

Returns termination status:

```
0
```

Figure 1031: Example: JobIDList

Error Codes

Error codes encountered in Batchmesher.

101
: Wrong number of arguments provided to hw_batchmesh. Used only in the command line.

102
: Missing required arguments for hw_batchmesh. Used only in the command line.

103
: The specified output directory does not exist. (See –work_dir option in hw_batchmesh).

104

Undefined required environment variable.

105

The specified input directory contains no model files (see -cad_model_dir in hw_batchmesh).

106

The custom pre-run or post-run Tcl procedure generated an error.

107

At least three critical errors occurred during the meshing of one model.

111

The HyperMesh executable (hmopengl) is in an incorrect path or is inaccessible.

112

The input geometry file is in an incorrect path or is inaccessible from hw_batchmesh.

113

The criteria file is in an incorrect path or is inaccessible from hw_batchmesh.

114

The parameter file is in an incorrect path or is inaccessible from hw_batchmesh.

121

Either the `time_limit.txt` or result (`*_res.txt`) file was not created after the specified timeout.

132

Result file (`*_res.txt`) not found.

133

Error while reading the `time_limit.txt` file.

134

Abnormal termination of the HyperMesh process.

135

The HyperMesh process is frozen, possibly waiting for user input.

141

The input geometry file is in an incorrect path or is inaccessible from HyperMesh.

142

The criteria file is in an incorrect path or is inaccessible from HyperMesh.

143

The parameter file is in an incorrect path or is inaccessible from HyperMesh.

144

An error occurred while reading the input file (see HyperMesh `*readfile` command).

145

An error occurred while importing the input file (see HyperMesh `*feinputwithdata2` command).

146

An error occurred while reading the criteria file (see HyperMesh `*readqualitycriteria` command).

147
An error occurred while running hw_batchmesh.

148
The custom pre-geom, pre-mesh or post-mesh Tcl procedure generated an error.

151
Licensing error.

Customize with User Procedures

Customize Batchmesher by providing user-defined Tcl procedures that run at specific times during the Batchmesher process.

Procedures can be customized for pre-run, post-run, pre-geometry load, post-geometry load, pre-batchmesh, and post-batchmesh.

These scripts can perform a wide range of tasks, such as:

- Exporting a mesh in solver format.
- Generating the midsurface of a thin solid geometry.
- Performing a surface offset to move a sheet geometry to a midsurface location.
- Naming and numbering parts to user-specific requirements.

Batchmesher has the ability to specify user-specified procedures at the following steps for each job:

Pre-geometry load
Executed immediately after the job begins, before the input model is imported.

Pre-batch mesh
Executed immediately after the input model is imported, before the batch mesh begins. Examples include extracting a midsurface or performing a surface offset.

Post-batch mesh
Executed immediately after the batch mesh process is complete. Examples include creating solver specific cards, or exporting the mesh in a specific format.

Access to model-specific variables is possible within user-registered procedures. The array `::hmbm::gVarArray` contains variables accessible by you.

`CADImportOpt`
String of CAD import options.

`cadtype`
Type of model file.

`critername`
Criteria file name, excluding the path.

`criterpath`
Criteria file name, including the path.

`CurrentEventName`

modelname
Input model file name, excluding the path.

modelpath
Input model file name, including the path.

outmodelname
Output model file name, excluding the path.

outpath
Output file path.

paramname
Parameter file name, excluding the path.

parampath
Parameter file name, including the path.

POST_BATCHMESH,tclparameters
Parameters passed to the post-batch mesh procedure. Only available when such a procedure is defined.

POST_BATCHMESH,tclprocedure
Post-batch mesh procedure name. Only available when such a procedure is defined.

POST_BATCHMESH,tclscriptpath
Post-batch mesh script name, including the path. Only available when such a procedure is defined.

PRE_BATCHMESH,tclparameters
Parameters passed to the pre-batch mesh procedure. Only available when such a procedure is defined.

PRE_BATCHMESH,tclprocedure
Pre-batch mesh procedure name. Only available when such a procedure is defined.

PRE_BATCHMESH,tclscriptpath
Pre-batch mesh script name, including the path. Only available when such a procedure is defined.

PRE_GEOMETRY_LOAD,tclparameters
Parameters passed to the pre-geometry load procedure. Only available when such a procedure is defined.

PRE_GEOMETRY_LOAD,tclprocedure
Pre-geometry load procedure name. Only available when such a procedure is defined.

PRE_GEOMETRY_LOAD,tclscriptpath
Pre-geometry load script name, including the path. Only available when such a procedure is defined.

resfilename
Output result file name, excluding the path.

By default, Batchmesher additionally passes in the name of the model file as the last argument to the Tcl procedures.

Note:

Tcl scripts must not automatically run any procedures, as this is handled by Batchmesher. Doing so will generate an error or unexpected results.

Save the model after running user procedures, as this is not done automatically. For example:

```
hm_answernext "yes"
*writefile "$::hmbm::gVarArray(outmodelname)" 0
```

```
proc nastran_export { modelName args }  {
set outputDir $::hwbm::gVarArray(modelpath)
set template_dir [ hm_info -appinfo SPECIFIEDPATH TEMPLATES_DIR]
set template [file join $template_dir "feoutput" "nastran" "general"]
*feoutput "$template" ${outputDir}/${modelName}.dat 1 1 1
}
```

Figure 1032: Example: Post-Mesh User Procedure

This example exports the generated mesh to a NASTRAN file named `<modelname>.dat` *in the same directory as the original input model.*

Procedures can also be specified at the run level to enables you to perform operations such as reading all the batch-meshed parts into a single model, creating properties or materials, or creating connections such as welds between the parts. Customization options at the run level can be set to execute at two points in the batch mesh process:

Pre-run
: Executed before the first model/job starts.

Post-run
: Executed after the last model/job completes.

In the pre-run and post-run scripts, the model-specific variables are not available.

Batch Mesh

Typical workflow for creating a mesh using Batchmesher.

1. Define a mesh type.

   A mesh type consists of a Criteria File and a Parameter File.

   a) Click the **Configurations** tab.
   b) Click to add a new configuration entry to the table.
   c) In the Mesh Type field, enter a name.
   d) In the Criteria File and Parameter File fields, browse to select the criteria and parameter files that define the mesh type.

2. Choose the models to batch mesh.

   a) Click the **Run Setup** tab.
   b) In the Input model directory field, browse to select the directory that contains the geometry to batch mesh.

c) Click to select model files.

d) In the **Select Model Files** dialog, select all of the relevant geometry files and click **Select**.

> **Tip:**
> In the Type of geometry field, select a geometry type to only list the files of that type.

The selected geometry files display in the table along with their geometry type.

3. Click the **Mesh Type** field for each file and select a relevant mesh type.

> **Tip:** Quickly apply the same mesh type to all files above or below the current row by right-clicking and selecting **Propagate Up** or **Propagate Down**.

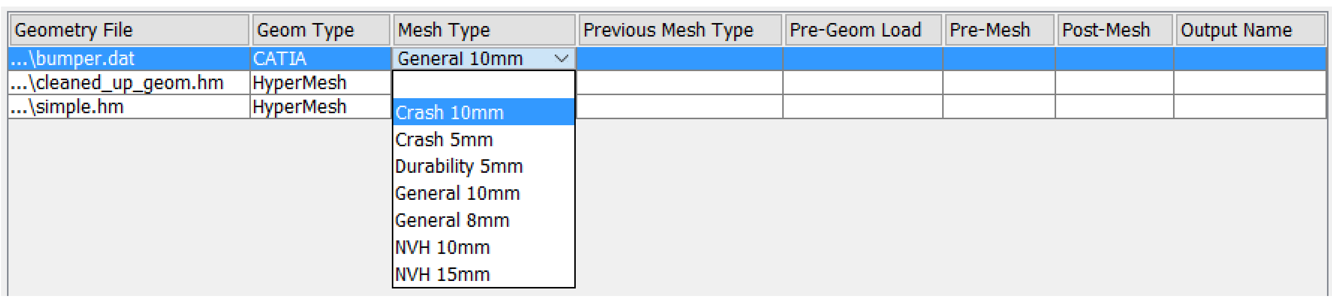


Figure 1033:

4. In the Output directory field, browse to select the directory where Batchmesher will save all results files.

If no output directory is specified, the results will be saved to the Input model directory.

5. Click **Submit** to start the Batchmesher run.

6. In the Run Status tab, monitor the job and run statuses.

All runs are listed in the Run Status tab, along with the status of job in the run. A Batchmesher run creates a unique directory inside of the Output directory where it stores its meshed results. This unique directory name displays in the tree for each run.

> **Tip:**
> Advanced details of a job with a COMPLETE or WORKING status can be monitored by highlighting the item of interest and clicking **Details**.
>
> Advanced details for a run can be monitored by clicking **Run Details**.

7. Once a job is complete in the Run Status tab, load the final mesh into HyperMesh by selecting the job in the tree and clicking **Load Mesh**.

Run Setup

Define settings for a batch mesh run and the jobs it contains in the Run Setup tab.

In the Run Setup tab, a table layout is used to specify each job, along with options that apply to the entire run.

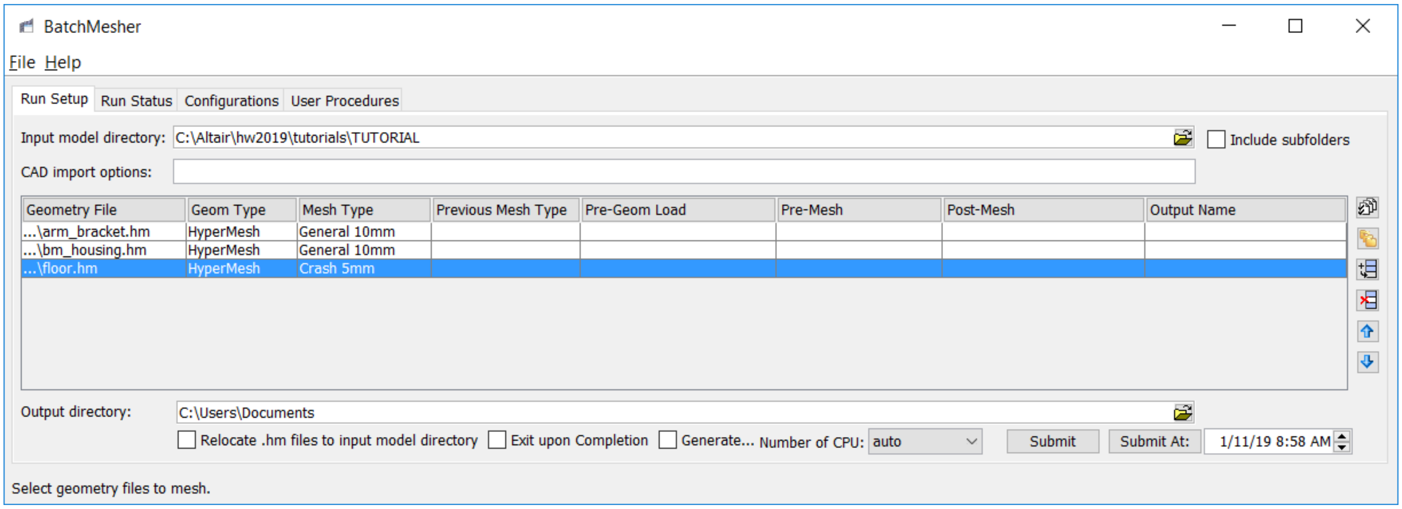


Figure 1034:

For each geometry file in the table, a geometry type must be chosen in the Geom Type field. If you select files using the **Select Model Files** dialog, than the geometry type is automatically set based on the value in the dialog. Manually select a geometry type by left-clicking the **Geom Type** field. Select the **Auto** geometry type to automatically attempt to detect the model type during import.

Additionally, each file requires a mesh type to be chosen in the Mesh Type field. The available mesh types are defined in the Configurations tab. Select a mesh type by left-clicking the **Mesh Type** field. Clear the mesh type by selecting the empty row in the drop down list.

Job dependencies can be defined, where the output of one model is used as the input to another. This is useful, for example, when creating a common mid-surface geometry model that is then used as input to generate domain specific mesh models. Such dependencies can be defined using the Previous Mesh Type field. When a previous mesh type value is selected, Batchmesher will find another Geometry File with the same name and specified Mesh Type, and run that model first. The output from that model will then be used as the input for the dependent model. The order of the definitions does not matter, as Batchmesher will decide the appropriate order to run any dependent jobs. If there is no other Geometry File that has the specified Previous Mesh Type, Batchmesher will not execute the dependent model.

Further, if there is an error in generating the previous model, Batchmesher will also not execute the dependent model.

Each entry can have user-defined procedures specified. The available procedures are defined in the User Procedures tab. To set a procedure, left click in the corresponding Pre-geom Load, Pre-Mesh or Post-Mesh field and select from the drop down list. To clear the procedure, select the empty row in the drop down list. Additional procedures for pre- and post-run can be specified in the User Procedures tab as well.

The name of the `.hm` file that is generated by Batchmesher can be customized using the Output Name cell. If this field is empty, the default output name is used, which is a concatenation of the Geometry File name, the criteria file name, and the parameter file name. If the value is set to Input model name the input model name is taken, and any file extension is replaced with `.hm`. Finally, a user-defined value can be entered. The name will be taken exactly as defined, including any file extension. If multiple files are given the same output name, the files will be overwritten and only the last one saved will remain.

> **Tip:** Quickly assign the same option to all of preceding or following entries in the table by right-clicking in the corresponding field and selecting either **Propagate Up** or **Propagate Down**.

Table 251: Run Setup Options

Option	Description
Input model directory	Directory that contains the geometry/CAD files for batch meshing. **Note:** You must select the directory containing the CAD files and not the CAD files themselves. The Include subfolders option additionally lists CAD files in all the subfolders of the selected directory.
CAD import options	CAD import strings can be provided to override default import behaviors, for example "SplitComponentsBy=Parts". The provided strings apply to all CAD import, regardless of format. Valid strings can be found in Supported CAD Readers.
	Launch the **Select Model Files** dialog to selecting the individual files to be batch meshed. **Tip:** Use the Shift and Ctrl keys to select or deselect multiple files from the list. If the Include subfolders checkbox is selected, matching files in subdirectories are also listed. Once the selection is complete, click **Select** to add the selected files to the table.

Option	Description
	All files of the selected geometry type in the source directory are selected by default. The process can be repeated to add more files from different directories or to add the same files multiple times to generate different sizes/types of mesh. A new directory can also be specified and files can be selected from that location. In addition, double clicking in a row in the Geometry File column allows direct editing of the file name and path.
	Launch the **Select Folder with Model Files** dialog to select an entire directory to be batch meshed. Once a directory is selected, all of the files matching the specified geometry type are listed. If the Include subfolders checkbox is selected, matching files in subfolders are also listed. No selection of individual files needs to be made. After review, click **Select** to add the selected directory to the table. All files of the selected geometry type in the directory are included. The process can be repeated to add more directories or to add the same directories multiple times to generate different sizes/types of mesh. In addition, double clicking in a row in the Geometry File column allows direct editing of the directory name and path.
	Add an empty row to the end of the table.
	Remove the selected row/rows from the table.
	Move the selected rows up in the table.
	Move the selected rows down in the table.
Output directory	Directory where the Batchmesher output files should be written. The results of the run are saved in a subdirectory named bm_<date>_<run>. For example, a first run on Aug. 4, 2011 will be named bm_110804_001. The next run on the same day would be bm_110804_002. If no output directory is specified, the Input model directory will be used.
Relocate .hm files to input model directory	Move all .hm output files to the corresponding directory of the input model.
Generate quality report	Generate HTML quality reports for each model, and a summary report for each job. The reports can be loaded from the Run Status tab.
Exit upon completion	Close the Batchmesher GUI after all jobs are complete.

Option	Description
Number of CPU	Maximum number of simultaneous jobs that can be run. If set to "auto" for multi-CPU machines, this will default to the number of CPUs minus 1.
Submit	Start the Batchmesher run and automatically open the Run Status tab. A run can also be started at a later time using the Submit At option.
Submit At	Start the Batchmesher run at a specified time, and automatically open the Run Status tab. The job status is listed as "Waiting" until the time the run starts.

Run Status

Once an Batchmesher run is initialized, the Run Status tab reports on the status of the run and its jobs.

Each run is listed as a sub-folder in a tree, along with the exact path of the results location. Each job, corresponding to one geometry file, is listed as a node of the tree, along with its geometry and mesh types, and its current status. Additional details can be queried for each job and for the entire run.

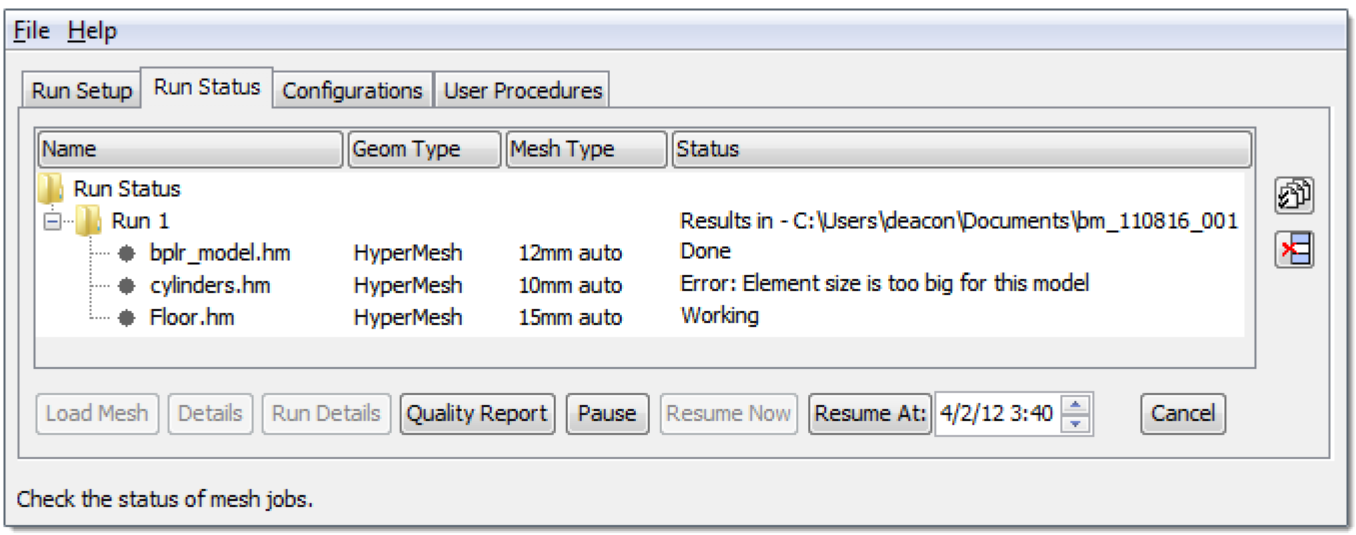


Figure 1035:

The Status field will display one of the following:

Working
: Batch meshing is currently being performed on this model.

Pending
: This model is currently in the queue and has not started the Batchmesher process yet. Any models with the status can be canceled.

Waiting
: The job will begin automatically at a user-specified date and time.

Done

The batch meshing process is complete, and results can be reviewed.

Table 252: Run Status Options

Option	Description
	Load a `.log` file from a previous run to be reviewed.
	Remove a run from the tree. **Note:** This only affects the user interface; it does not delete any files.
Load Mesh	Select a single job from the tree and use this button to invoke interactive HyperMesh and load the final batch meshed model. The corresponding criteria file is also loaded in the QI panel so that the quality checks represent the meshing requirements set in Batchmesher. **Note:** This can only be performed on models that have a status of Done.
Details	Select a single job from the tree and use this button to obtain more details on the status. The details include: • Date and time the job began. • Complete path of the geometry model file. • Complete path of corresponding criteria file. • Complete path of corresponding parameter file. • Time taken to load the geometry model. • Table containing information on relevant steps in the batch meshing process. • Status of the run. A value of COMPLETE indicates success. A value of ERROR indicates a failure. Click **Refresh** to manually refresh the window with the latest details. Click **Auto Refresh** to automatically update the window with the latest details while the job is running.
Run Details	Select a run from the tree and use this button to obtain more details on the status. The details include: • Date and time the run began.

Option	Description
	• For completed jobs, statistics such as the time required to complete the job, the final number of faces/elems/failures, and the quality index. • Number of jobs completed, in process and waiting. Click **Refresh** to manually refresh the window with the latest details. Click **Auto Refresh** to automatically update the window with the latest details while the job is running.
Quality Report	Generate HTML quality reports for each job, and a summary report for each run. **Restriction:** Only valid if the Generate quality report checkbox was enabled on the Run Setup tab for the run.
Pause	Pause all pending jobs. This does not affect currently running jobs, which cannot be paused.
Resume Now	Resume all paused jobs.
Resume At	Resume all paused jobs at a future date and time.
Cancel	Cancel a single run. Runs that are Pending or Working can be canceled.

Configurations

Define the available mesh types that can be used in Batchmesher jobs in the Configurations tab.

A mesh type is a name given to a set composed of one criteria file and one parameter file.

A table layout is used to specify each mesh type and its files.

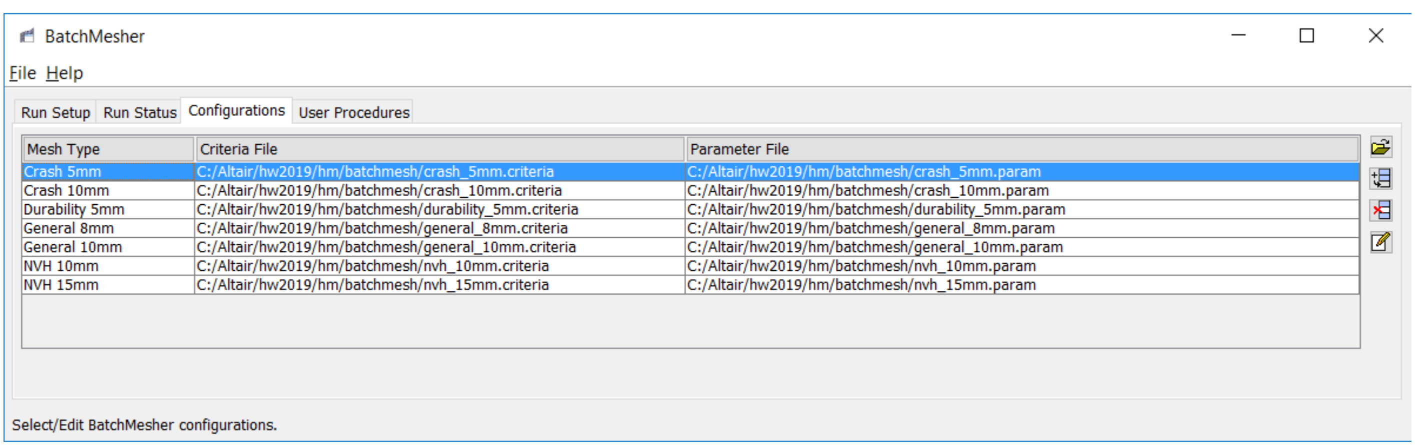


Figure 1036:

There are several pre-defined mesh types provided in the installation.

crash 5mm
Sample template file for crash analysis with average element size of 5mm.

crash 10mm
Sample template file for crash analysis with average element size of 10mm.

durability 5mm
Sample template file for durability analysis with average element size of 5mm.

general 8mm
Sample template file for general use cases with average element size of 8mm.

general 10mm
Sample template file for general use cases with average element size of 10mm.

NVH 10mm
Sample template file for NVH analysis with average element size of 10mm.

NVH 15mm
Sample template file for NVH analysis with average element size of 15mm.

Option	Description
	Launch a dialog, from which you can select a criteria or parameter file, depending on the selected cell. In addition, double click a row in the Criteria File or Parameter File columns to edit the file name and path.
	Add an empty row to the end of the table.
	Remove the selected row/rows from the table.
	Launch the Criteria and Parameters Editors, from which you can edit the criteria and parameters files in the selected row. **Note:** It may not be possible to edit the default parameter and criteria files provided in the installation due to file permissions.

User Procedures

User procedures that can be specified in Batchmesher for jobs and runs in the User Procedures tab.

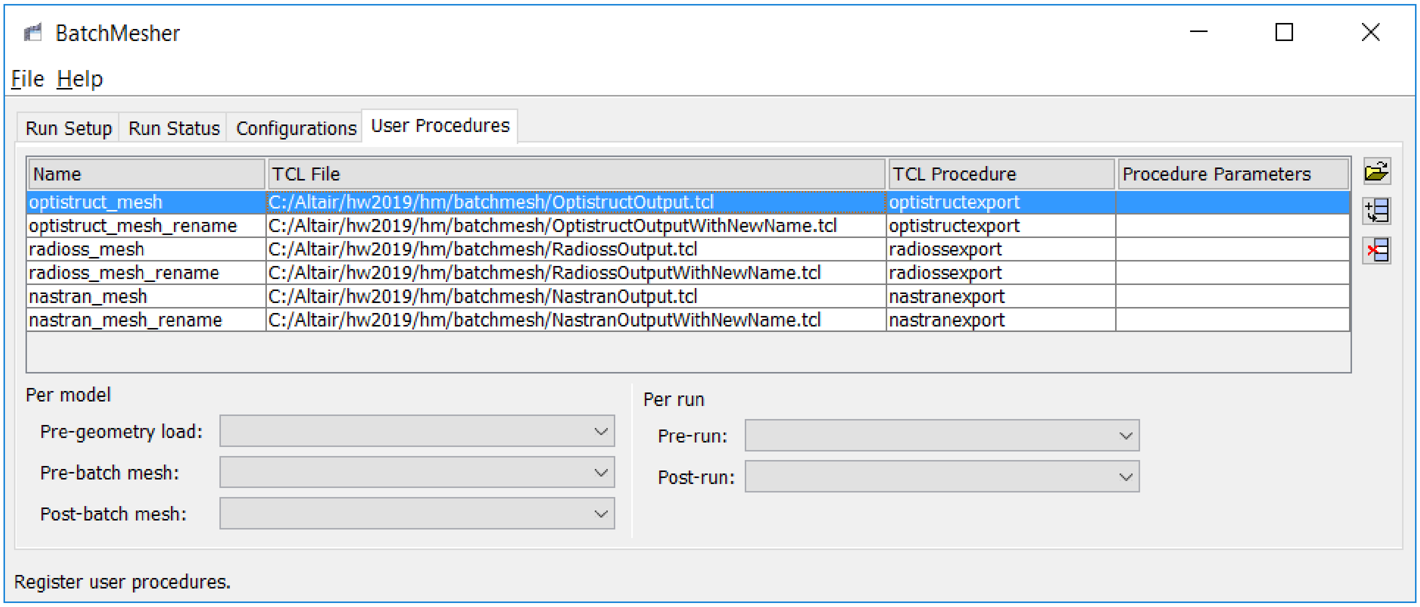


Figure 1037:

There are several pre-defined procedures provided in the installation.

nastran_mesh

Write out the resulting mesh in Nastran format.

nastran_mesh_rename

Write out the resulting mesh in Nastran format, but renames the output file.

For each user procedure in the table, the Name must be specified after selecting the file. Double-click in the cell to edit the name.

Note: It is recommended to use a unique name for each entry, but it is not required.

Additionally, each user procedure requires a Tcl Procedure to be selected. To set the procedure, left-click in the **Tcl Procedure** cell and select from the drop down list. To clear the procedure, select the empty row in the drop down list.

Procedures can be specified that apply to all jobs by default. These can then be manually overridden on the Batch Mesh tab.

Pre-geometry load
: Select a user procedure to execute immediately after the job begins, before the input model is imported.

Pre-batch mesh
: Select a user procedure to execute immediately after the input model is imported, before the batch mesh begins. Examples include extracting a midsurface or performing a surface offset.

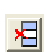
Post-batch mesh
: Select a user procedure to execute immediately after the batch mesh process is complete. Examples include creating solver specific cards, or exporting the mesh in a specific format.

Procedures can also be specified that apply to the entire run. These cannot be manually overridden elsewhere.

Pre-Run
: Select a user procedure to execute before the first model/job starts.

Post-Run
: Select a user procedure to execute after the last model/job completes.

Table 253: User Procedure Options

Task	Description
	Open a dialog, from which you can select a `.tcl` file. In addition, double-click a row in the Tcl File column to edit the file name and path.
	Add a empty row to the end of the table.
	Remove the selected row/rows from the table.

Edit Criteria and Parameter Files

Set the appropriate parameters/options to obtain the desired mesh from Batchmesher by editing the criteria and parameter files.

Use the Criteria and Parameters Editor to create and modify mesh criteria files as well as geometry cleanup parameter files.

HyperMesh uses the same editor for both types of files; criteria and parameters display on separate tabs within the editor.

1. In Batchmesher, click the **Configurations** tab.
2. Right-click the entry of the configuration to modify and select **Edit File** from the context menu.
3. In the **Criteria and Parameters Files Editor**, edit criteria and parameters.
   - Click the **Criteria** tab to setup the quality index (QI) mesh criteria.
   - Click the **Parameters** tab to setup geometry cleanup and defeaturing parameters.
4. Save your changes by clicking **File** > **Save/Save As** from the menu bar.
   - Click **Save** to saves the currently loaded file. If no file is loaded, a prompt will be given for a new name.
   - Click **Save As** save the current settings to a new or different file name.

See Also
Criteria and Parameter Settings

Connectors

8

Create connections between parts of your model.

This chapter covers the following:

Connectors define which parts of a model have to be fastened to each other and how the connections have to be performed.

Connectors are geometric entities (not FE) used to create connections between components. Connectors are used to realize FE idealizations of the physical connection. Just as you create an FE mesh on a surface, you create FE connections by realizing a connector.

In HyperMesh, connectors can take into account the mechanical and structural conditions of the model.

Connectors, in the form of mass connectors, are used to distribute masses on certain areas in a model. These connectors define both the area and the type of mass in such cases.

Connector Entity

Connectors are geometric entities (not FE) used to create connections between components. Connectors are used to realize FE idealizations of the physical connection. Just as you create an FE mesh on a surface, you create FE connections by realizing a connector.

Connector Definition

A connector definition describes the connection between multiple entities at a specific location.

Entities that are to be connected are referred to as link entities. The connector location can be defined as:

- node
- node list
- geometric point
- line
- line list
- surface
- elements
- tags

Any number of link entities of differing types can be added to a connector in any order. The connector sets the order of link entities during the realization process.

Example: Connect Assemblies

The projection behavior for connecting assemblies.

In this example, the model file has two assemblies that each have two components:

Assem 1 contains Comp 1 and Comp 2

Assem 2 contains Comp 3 and Comp 4

A connector is created with the Links as Assem 1 and Assem 2, and the number of layers set to 3.

During realization the closest found components residing inside the 2 assemblies will be retained as the component links.

Inside each assembly, the closest component to the connector is determined to satisfy a 2T connection, and for the third layer (3T) the closest component to the connector in either of the assembly links is utilized.

Note: The same projection behavior is used for parts containing components.

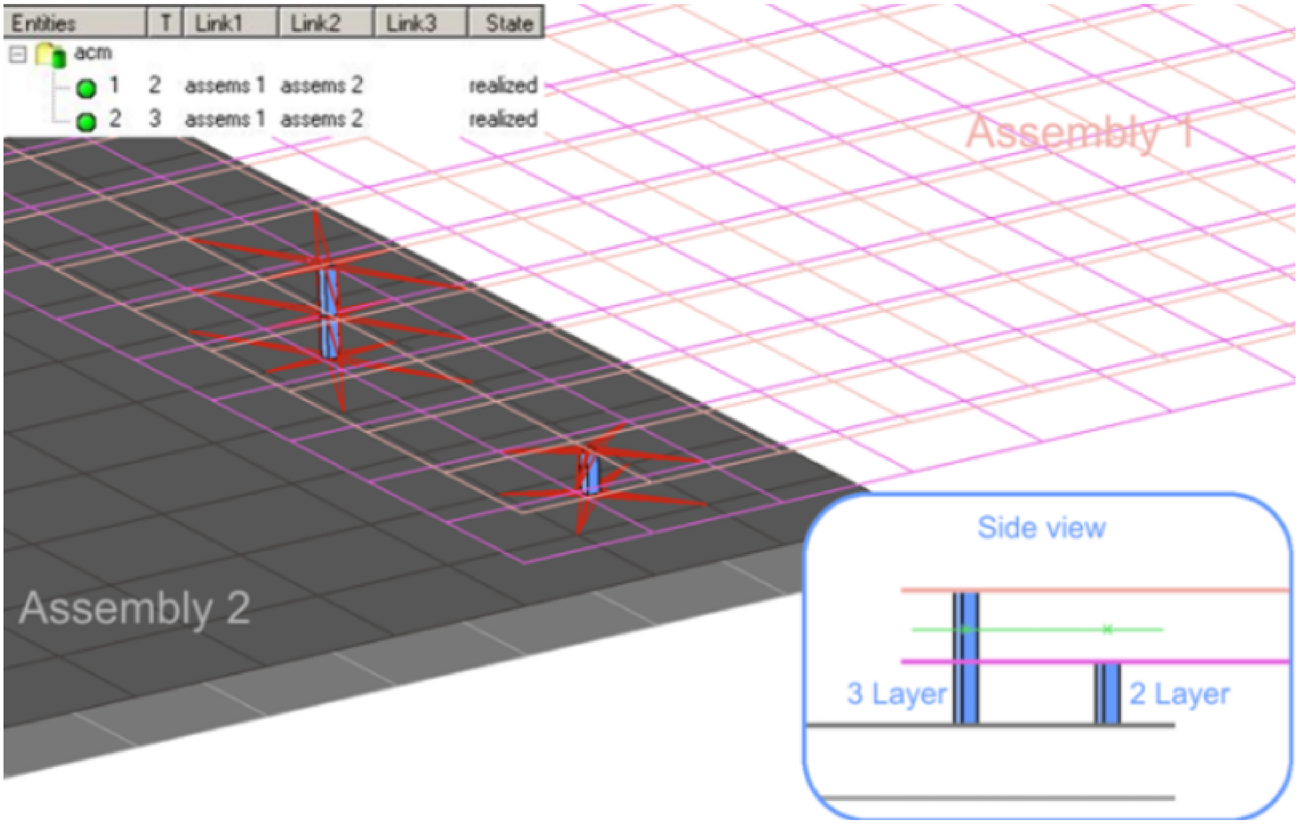

Entities
T
Link1
Link2
Link3
State
acm
1 2 assems 1 assems 2 realized
2 3 assems 1 assems 2 realized
Assembly 1
Assembly 2
Side view
3 Layer
2 Layer

Figure 1038:

Connector Terminology

Overview of connector terminology.

Connector Location

The position in space at which a connector entity is created.

Entities that can be used to define the connector location depend on the connector type.

Spots

Nodes
Connector is created at the node location.

Points
Connector is created at the point location.

Lines
Connector is created at the center of the selected line.
Only one connector is created for each line, but the line may be split into multiple projection locations as specified by the offset, spacing, and density values.

Nodelist
The nodelist can be considered as to be a line. The treatment is the same.

Bolts

Nodes
Connector is created at the node location.

Points
Connector is created at the point location.

Lines
Connector is created at the center of the selected line.
Only one connector is created for each line, but the line may be split into multiple projection locations as specified by the offset, spacing, and density values.
This connector is only used for repetitive holes in a certain, constant distance along the selected line.

Seams

Lines, Linelist
Connector is created at the center of the selected line.
Only one connector is created for each line, but the line may be split into multiple projection locations as specified by the offset, spacing, and density values.

Nodelist

The nodelist can be considered as to be a line. The treatment is the same.

Areas

Elems

Connector is created at the elements location.
Only one connector is created for each group of elements, but the area may be subdivided into multiple projection locations as specified by the nodes of the selected elements.
The area can be re-meshed to get different projection locations.

Surfs

Connector is created at the surface location.
Only one connector is created for each surface, but the area may be subdivided into multiple projection locations as specified by the mesh type and element size values.
The area can be re-meshed to get different projection locations.

Linelists/Lines

One connector is created for each line. The line is extruded to an area considering the width and the offset values. The area may be subdivided into multiple projection locations as specified by the mesh type and element size values.
If **line combine** is enabled, adjacent lines are treated as one line and produce one area connector.
The area can be re-meshed to get different projection locations.

Nodepath

The nodelist can be considered as to be a linelist. The treatment is absolutely the same.

Masses

Nodes

Connector is created at the node location.

Points

Connector is created at the point location.

Connector Realization

During connector realization, welds are created using the connector definition.

In HyperMesh, the only form of realization currently supported is FE realization (weld creation). For successful realization, the connector must be populated with all the relevant details required for its realization type. For example, FE realization requires the connector to be populated with a projection tolerance and an FE configuration type.

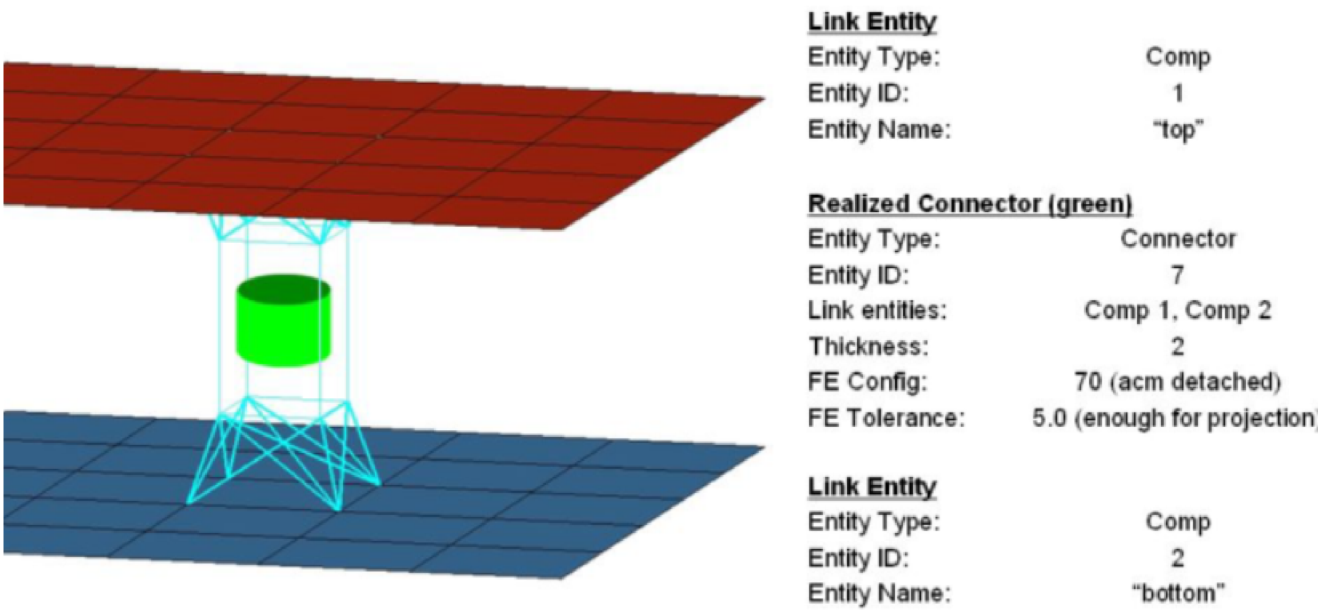


Figure 1039: Connector Realization

Connector seven realized with a valid tolerance value, and a config value of type 70 (acm detached).

One advantage of separating weld FE realization from the connector definition is that a connector can be re-realized as a weld of a different configuration, or possibly, a user-defined weld, without having to redefine the connector. If you edit the connector definition, for example, add or delete a link entity from the connector, the connector removes the welds it created, and reverts back to an unrealized state. The connector is unrealized only if its user-control mode is turned off. By default, the connector mode is off but it can be turned on by registering custom FE with a connector. Connectors store all FE information that they create, allowing advanced find, mask, delete, and organizational functionality in a number of common panels. If the weld creation is unsuccessful, due to low tolerance, insufficient link entities, and so on, the connector icon is displayed as failed (red). An unrealized connector is yellow, a realized connector is green, and a failed connector is red.

Connector State

Status of the connector before and after realization.

The color of the connector changes based on their state to allow you to quickly visualize and filter connectors.

Unrealized

The initial definition of the connector entity after it is created.
The connector is displayed in yellow.

Realized

The connector is considered realized only if weld creation at the connector was successful.
The connector is displayed in green.

Modified

The connector is considered modified when one or more of its corresponding attributes have been edited in the Connector Entity Editor.
The connector is displayed blue.

Failed

The connector is considered failed if the weld creation at the connector was not successful.
The connector is displayed in red.

Link Entity State

The link entity state specifies if the entity referenced by the link entity is meshed or unmeshed.

Geom

Specifies that the entity needs to be connected (welded) using its geometry (connect surfaces only).

Elems

Specifies that the entity needs to be connected (welded) using its mesh.

Both states are applicable to assemblies, components and surfaces only. The elems state connects the mesh on the assembly, component, or surface. The geom state connects the geometry on the assembly, component, or surface. For all other link entities only the elems state is applicable. The states are added to the connector entity.

After establishing link entities, they can only be modified and edited in the lower part of the Connector Browser. The functions to add link, update links and remove links is found in the browser's right-click context menu.

The link entity state options for assemblies, component, and surfaces are set when creating the connector, or on the Add Links panel. The state can be edited/updated in the lower part of the Connector Browser, as well. Therefore, the extended information has to be activated in the browser configuration.

Link Entity

A link entity is a reference to a separate entity that can be added to a connector.

The entities, or a subset of them, to which the link entities refer are welded together during realization.

Supported link types include:

Assemblies

Connect elements or surfaces.
A group of parts that needs to be welded is often represented as an assembly.

Components

Connect elements or surfaces.
A part that needs to be welded is often represented as a component.

Elements
Facilitates a patch-patch weld connector.

Nodes
Facilitates a node-node weld connector.

Parts
Connect elements or surfaces.

Properties
Connect elements.

Surfaces
Create welds to connect geometry before meshing; the welds create fixed points for the mesh. The connected surfaces may be either meshed or unmeshed.

Tags
Define a weld connector for a node or an element that it holds.

> **Note:** Only nodes, tags, elements, surfaces, components, properties, and assemblies can be added to connectors. The connectors can hold a single entity or a combination of these entities.

After link entities are established, they can only be modified and edited in the lower part of the Connector Browser. The functions to add link, update links and remove links is found in the browser's right-click context menu.

The link entity options are set when creating the connector, or on the Add Links panel. The link entity can be edited/updated in the lower part of the Connector Browser as well.

Number of Layers

The number of layers defines how many thicknesses (layers) have to be connected at the connector position.

For seam and area connectors the number of layers is predefined as two.

For most spot connectors you will set the number of layers to two or three; though any higher number is possible. For spot connectors it is also possible to set the number of layers to auto. Then the exact number of layers is identified during the link detection and is written to each individual connector.

Bolt connectors can additionally be set to unlimited. This is for cases when the exact number of layers is not known. In this case, the number of layers is limited by other conditions like tolerance and cylinder dimensions.

For apply mass connectors a limit for entities can be set; this is optional.

The number of layers has to be defined for:

- Combined connector creation/realization in the top subpanel for spots, bolts, seams and areas.
- Pure connector creation in the create subpanel for spots, bolts, seams and areas.
- Adding links in the Add links panel.

If connect when is set to now, a link detection is performed during connector creation. With respect to the given tolerance, the selected link candidates and options like non-normal projection the valid connector links are established. By default, if excess link candidates have been selected, the links are reduced to the minimum needed to fulfill the request for the number of layers. The request for the number of layers has to be fulfilled for each connector test point; this leads, for example, to seam connectors typically having more than just two link entities.

> **Note:** The number of layers should not be mistaken for number of referenced links of a connector. In many cases the number of layers and the number of referenced links are equal, but that's not mandatory. The number of referenced links can be less, equal or more than the number of layers, although the connector might be realized successfully if the links are provided in a manner that the request for the number of layers can be fulfilled. That means not all referenced links necessarily need to be used for the connection, and remain unused, or sometimes only one link is used twice, for example in the case of a flap. For spot and seam connectors extra links can be stored. This can be set individually on the appropriate option subpanels below links conservation.

The link entity options are set when creating the connector, or on the Add Links panel. The link entity can be edited/updated in the lower part of the Connector Browser, as well.

Re-Connect Rules

Defines how a connector should protect its link entity information.

The re-connect rules can be defined in different ways:

- During pure connector creation in the create subpanel for spots, bolts, seams and areas, if connect when is set to now.
- During adding links to an existing connector in the Add links panel, if connect when is set to now.
- During adding links to an existing connector in the lower part of the Connector Browser.

In the Connector Browser, link entities can be precisely edited, provided that the extended information option is activated in the **Browser Configuration** dialog.

None

If a link entity references an entity that is removed from the database, the link entity is then removed from the connector.

by id

If a link entity references a entity that is removed from the database, the link entity retains the ID of the entity. The link entity remains in the connector.

by name

Same as the by id rule except that the entity name is retained.

by UID

Works the same as the other rules except that the part unique identifier is retained.
Only available for part links.

The link entity options are set when creating the connector, or on the Add Links panel. The link entity can be edited/updated in the lower part of the Connector Browser, as well; therefore, the extended information has to be activated in the browser configuration.

Realization Methods

Overview of the process used to select the best routine for realization.

Spot Realization

Overview of the spot connector realization process and methods.

Spot Realization Process

Overview of the spot realization process.

1. Select the realization type.

   mesh independent
   Use for realizations that do not need a node connection, and the connection is primarily defined via a solver-specific card, such as `CWELDs` for Nastran.

   mesh dependent
   Use for all other cases.

2. If mesh dependent is selected, you must decided whether or not to adjust the mesh or the realization.

   Adjust mesh
   Projection is done in a perpendicular way, and the mesh has to be adapted to the projection points.

   Adjust realizations
   The mesh will not be modified, at the expense of non-normal or incomplete realizations. Many realization types are defined with head elements attached to body elements. In the case of these realization types, the head elements realize the connection without modifying the mesh. Then the body element is still created in a normal direction.

3. Select a method for performing adjustments.

   Adjust mesh
   Sub-options include: quad transition and remesh.

   Adjust realizations
   Sub-options include: find nearest nodes, project and find nodes, and ensure projection.

4. Choose whether or not the imprint should be skipped for quad transition.

In stage 1, select the type of realization:

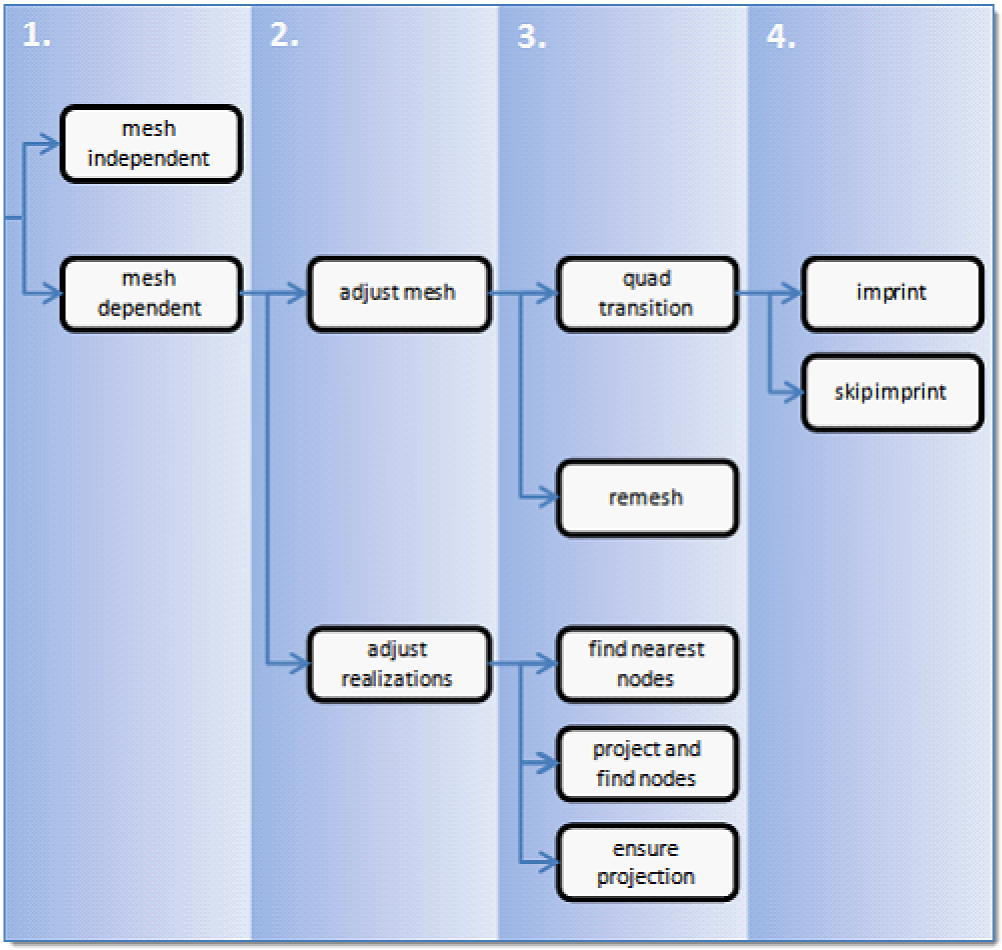


Figure 1040: Spot Realization Process

Spot Realization Methods

Overview of the different options for spot realization methods.

Mesh Independent

The mesh independent option is normally used for solver-specific realization types. A post-script is performed during realization to define the solver specific connection. For example, for the Nastran CWELD of ELEMID option, the shells which are in contact are observed and defined in the CWELD card.

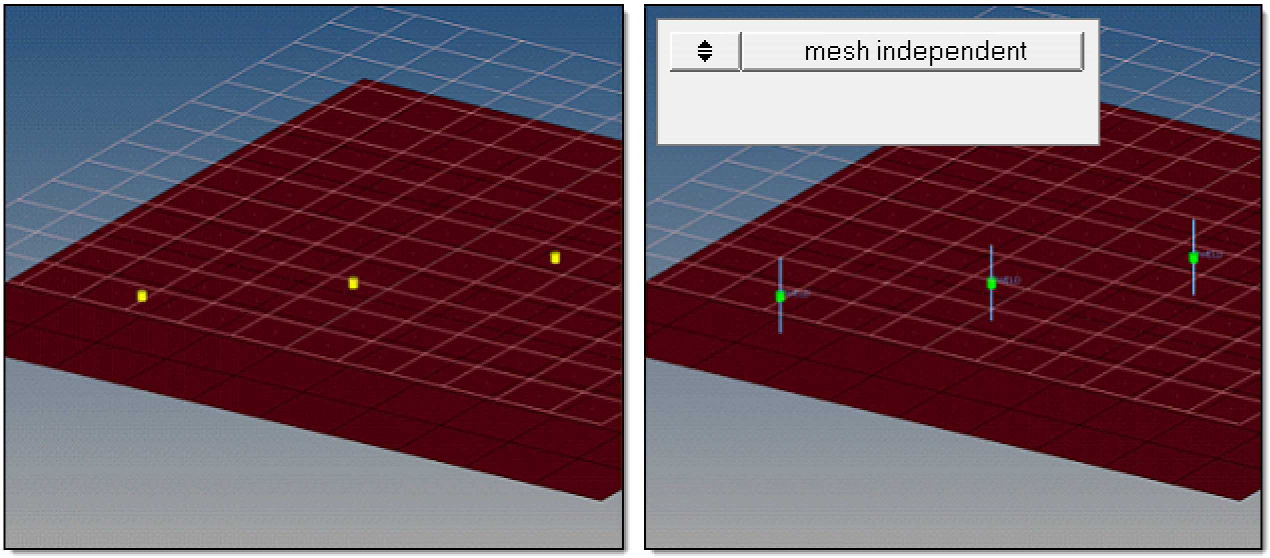


Figure 1041: Mesh Independent

Mesh Dependent – Adjust Mesh – Quad Transition – Imprint

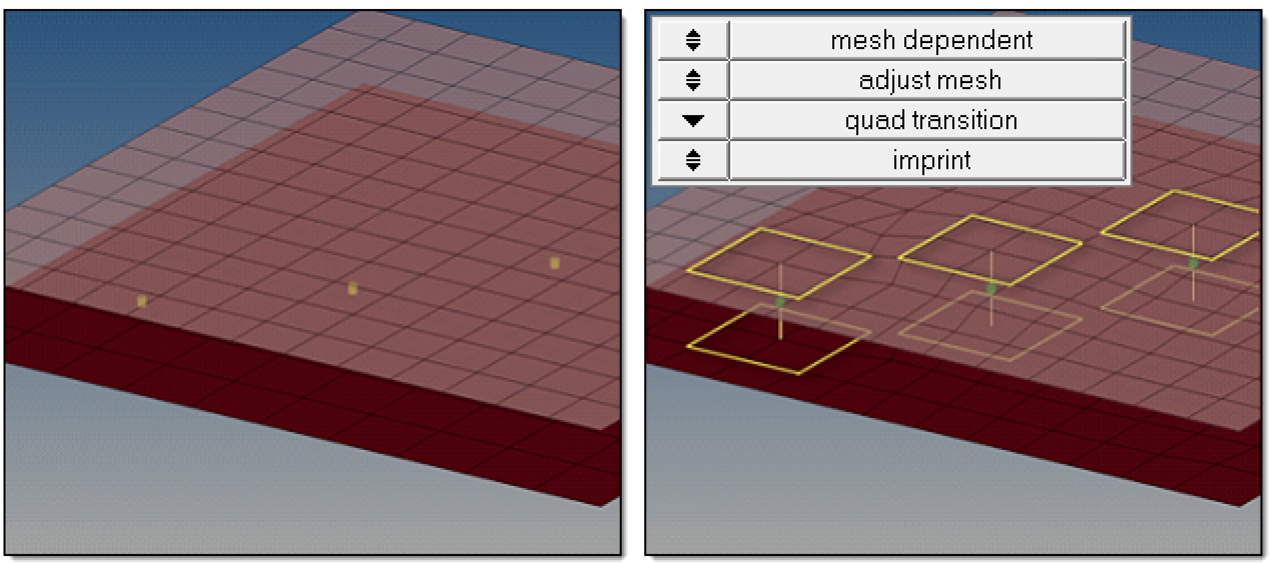


Figure 1042: Mesh Dependent, Adjust Mesh, Quad Transition, Imprint

Quad Transition

The quad transition option creates perfectly shaped quad elements around the projection points. By default, the quad size is determined by the average mesh size. Alternatively, you can specify a specific quad size in the quad size field.

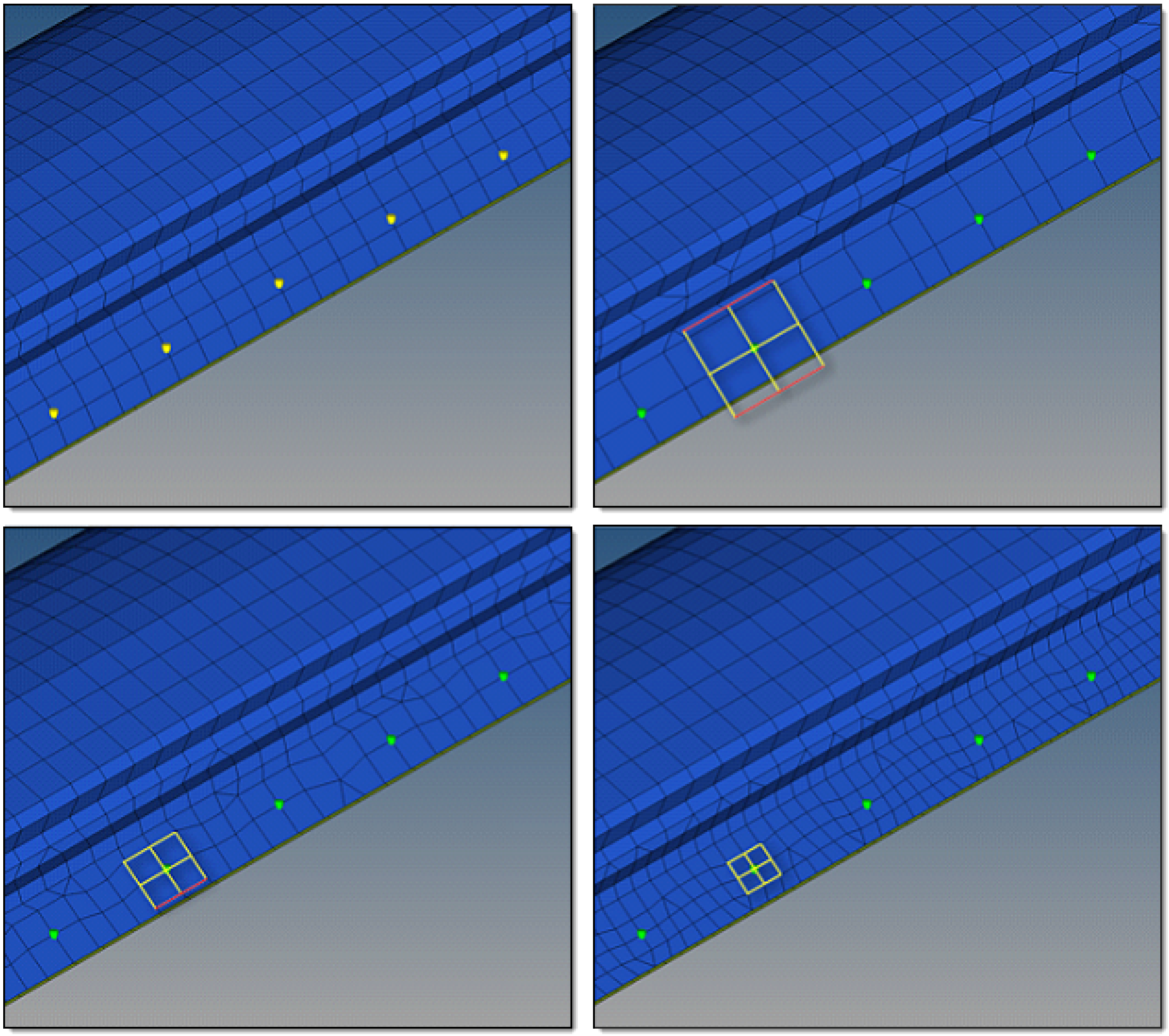


Figure 1043: Connectors Realized with Quad Transition using Different Quad Pattern Sizes
The top, left image illustrates the initial model situation. The remaining images illustrate connectors that have been realized with quad transition using different quad pattern sizes: average, coarse, small. The regular quad pattern size is highlighted and the red lines illustrate which nodes have been snapped to a relevant feature or free edge.

For spot quad transition, automatic snapping and feature detection is enabled by default via the **allow snapping** checkbox. This prevents the creation of elements that are too small, and ensures that the geometry is not modified too much.
Free edges and features with an angle greater than 25° are always taken into account. If smaller feature angles should be considered, decrease the value in the feature angle field (**Preferences** > **Meshing Options**). Feature angles smaller than 5° will not be considered.
By default, snapping is allowed by a distance of one third of the quad pattern element size. In the case of a predefined quad pattern element size of 10.0, the outer nodes can snap to features in a distance of 3.3. The algorithm also tries to snap all three nodes of a quad pattern or none.

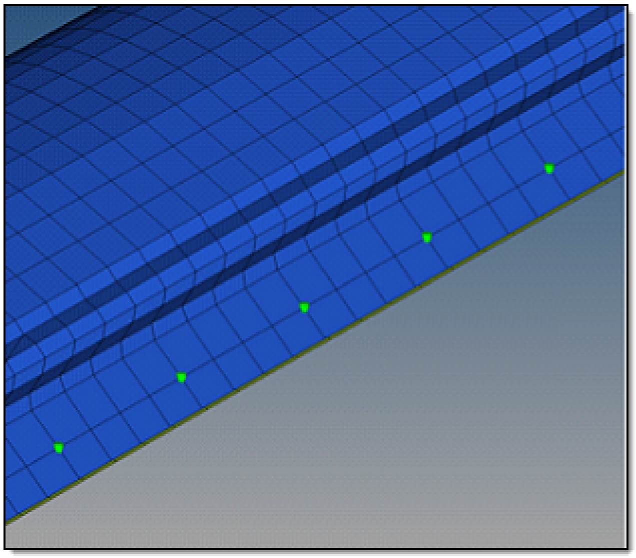


Figure 1044: Connector Realized with Quad Transition using Adequate Quad Pattern

A spot connector line is created when quad transition is used and a line or a node list is selected as the connector position, unless the **split to points** checkbox is activated.
In Figure 1045, spot connectors are at the same exact position, though there is a notable difference. In both images, the connectors have been created along a line, but in the left image the **split to points** checkbox was enabled. In the left image, the quad transition pattern is aligned to the mesh; in the right image the quad transition pattern is oriented along the spot connector line. All elements around the spot connector line belong to the regular pattern. The number of element pairs created along the spot connector line between the spot positions depends on the average or selected mesh size, which can be from one to many. The quad elements are distributed equidistant along the line.
In curved regions the inner and outer lengths of the element edges differ.

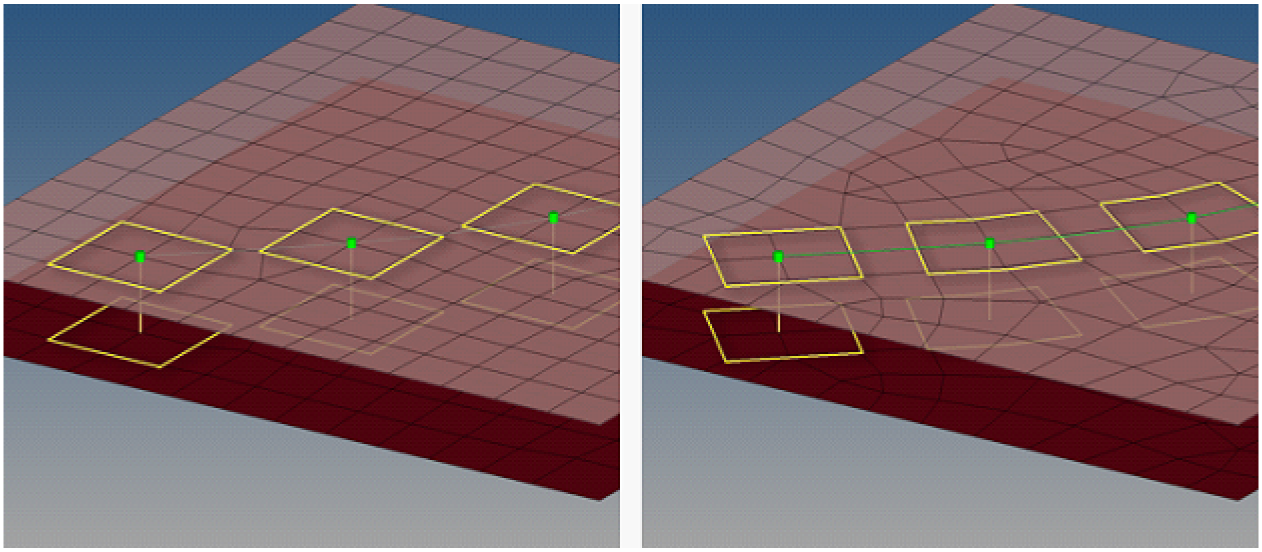


Figure 1045: Split to Points Example

Imprint

When creating mesh-dependent realizations with quad transitions, the quad transition meshes can overlap and disturb each other if more than one set of connectors is created too close to each other. Select the **imprint** option to reconcile such transitions with each other and modify the underlying mesh to match the results, creating a final result that is seamless and properly meshed.

To allow smaller imprint conflicts to be automatically resolved when connectors are realized, the **resolve conflicting imprints** checkbox is enabled by default. Overlapping elements are released, and a normal remesh of that area is performed as long as the overlapping area is smaller than half the regular quad transition element size. Larger conflicts may require a manual imprint.

Mesh dependent – Adjust Mesh – Quad Transition – Skip Imprint

Quad Transition

The quad transition option creates perfectly shaped quad elements around the projection points. By default, the quad size is determined by the average mesh size. Alternatively, you can specify a specific quad size in the quad size field.

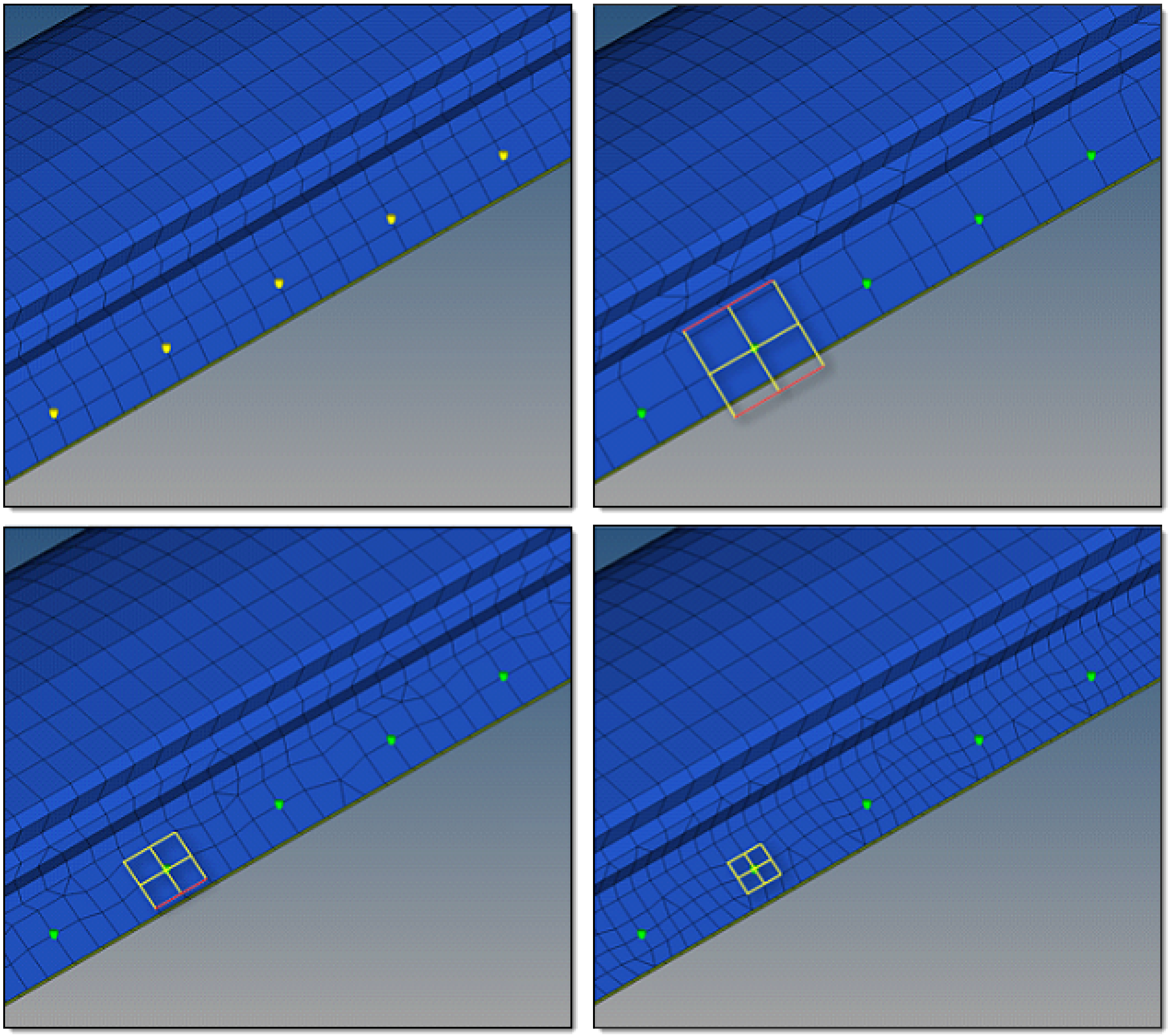


Figure 1046: Connector Realized with Quad Transition using Adequate Quad Pattern

The top, left image illustrates the initial model situation. The remaining images illustrate connectors that have been realized with quad transition using different quad pattern sizes: average, coarse, small. The regular quad pattern size is highlighted and the red lines illustrate which nodes have been snapped to a relevant feature or free edge.

For spot quad transition, automatic snapping and feature detection is enabled by default via the **allow snapping** checkbox. This prevents the creation of elements that are too small, and ensures that the geometry is not modified too much.

Free edges and features with an angle greater than 25° are always taken into account. If smaller feature angles should be considered, decrease the value in the feature angle field (**Preferences** > **Meshing Options**). Feature angles smaller than 5° will not be considered.

By default, snapping is allowed by a distance of one third of the quad pattern element size. In the case of a predefined quad pattern element size of 10.0, the outer nodes can snap to features in a distance of 3.3. The algorithm also tries to snap all three nodes of a quad pattern or none.

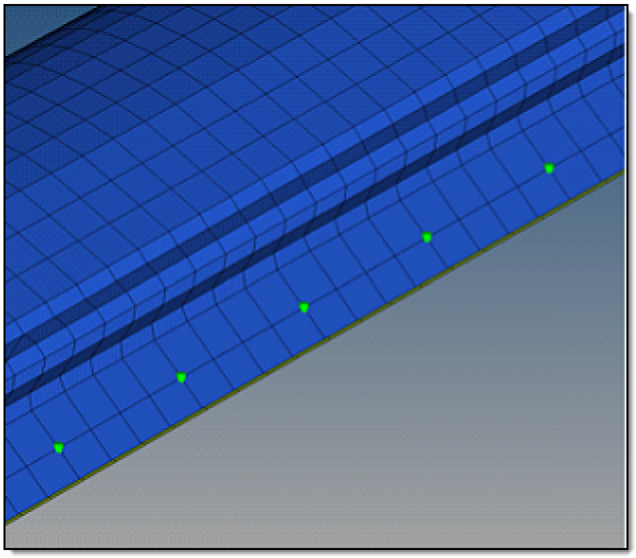


Figure 1047: Connector Realized with Quad Transition using Adequate Quad Pattern

A spot connector line is created when quad transition is used and a line or a node list is selected as the connector position, unless the **split to points** checkbox is activated.
In Figure 1048, spot connectors are at the same exact position, though there is a notable difference. In both images, the connectors have been created along a line, but in the left image the **split to points** option was enabled. In the left image, the quad transition pattern is aligned to the mesh; in the right image the quad transition pattern is oriented along the spot connector line. All elements around the spot connector line belong to the regular pattern. The number of element pairs created along the spot connector line between the spot positions depends on the average or selected mesh size, which can be from one to many. The quad elements are distributed equidistant along the line.
In curved regions the inner and outer lengths of the element edges differ.

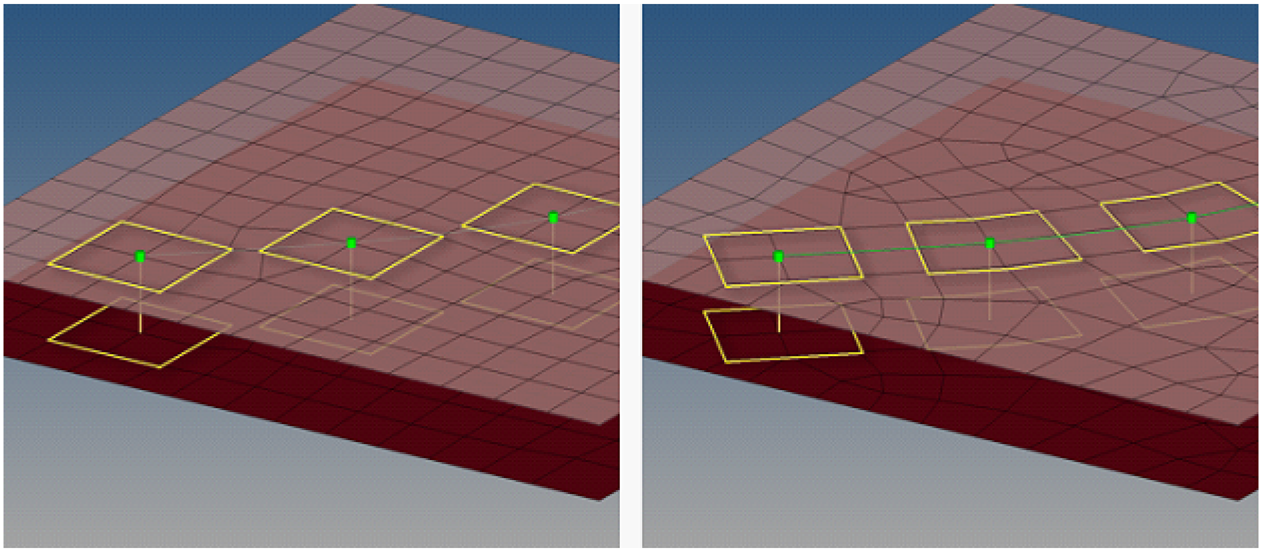


Figure 1048: Split to Points Example

Skip Imprint

The skip imprint option prevents the last step of quad transition from being performed. The component ^conn_imprint is created instead, which contains the element pattern. These elements can be modified and manually imprinted later using the Connector Imprint panel. Skip imprint allows you to realize such mesh-dependent realizations in very complex areas of the model where the automatic imprint fails because of issues such as conflicting spots.

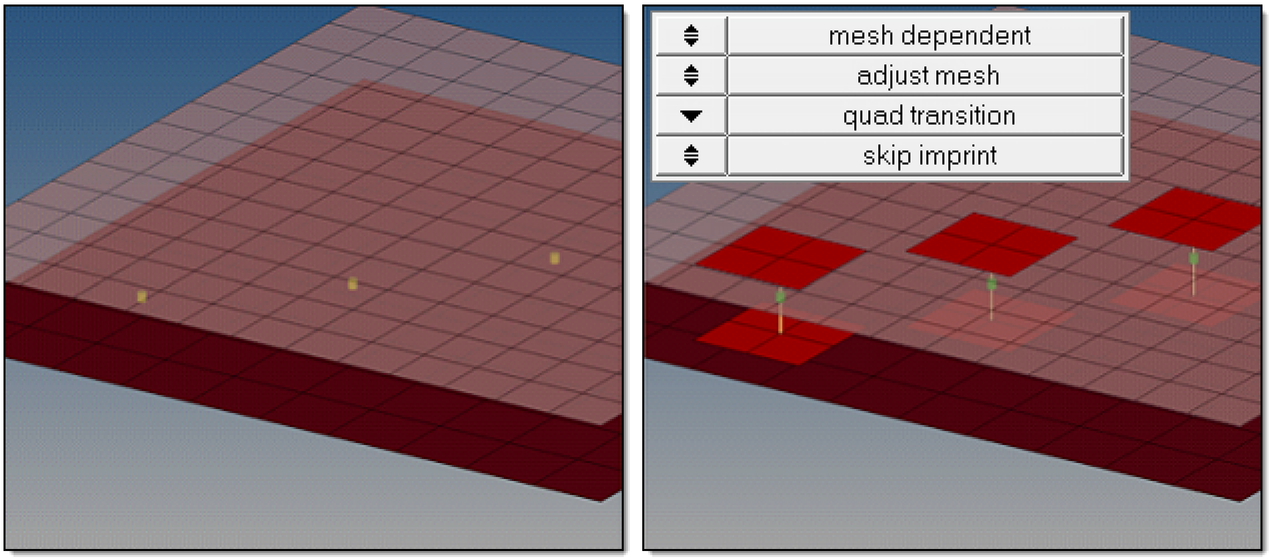


Figure 1049: Skip Imprint

Mesh Dependent – Adjust Mesh – Remesh

The remesh option takes the projection points into account and uses snap and split capabilities to connect the weld to the links.

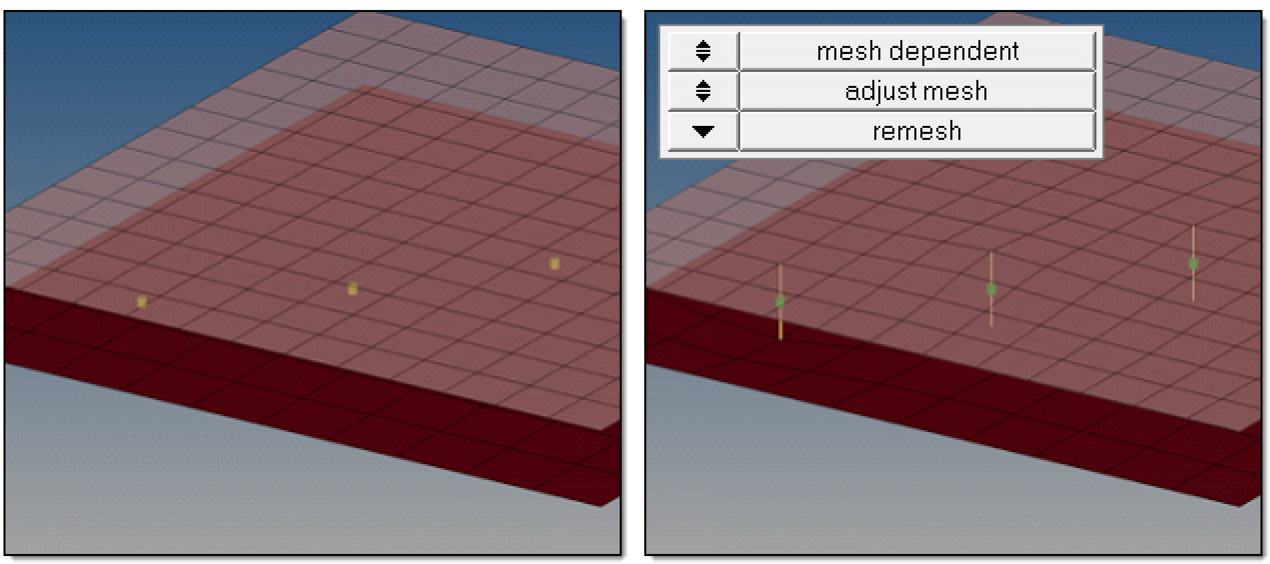


Figure 1050: Mesh Dependent, Adjust Mesh, Remesh

Mesh Dependent – Adjust Realization – Find Nearest Nodes

The find nearest nodes option searches for the nearest nodes within the given tolerance only, making it possible to connect t-joints and similar areas. This option is also very useful in situations where the connectors are not positioned perfectly. The realizations are allowed to be non-normal.

Find nearest node does not perform projections.

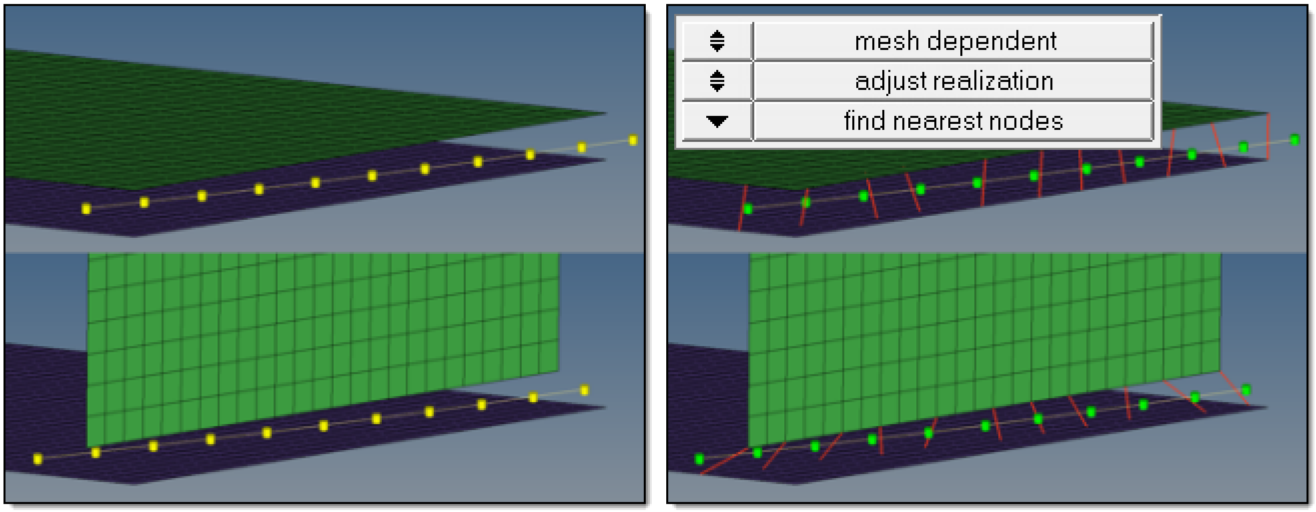


Figure 1051: Mesh Dependent, Adjust Realization, Find Nearest Nodes

Mesh Dependent – Adjust Realization – Project and Find Nodes

The project and find nodes option requires a valid normal projection onto the link entities in the first step. In the second step, the nodes closest to the projection points will be used for the connection. If the normal projection is not possible, the realization fails.

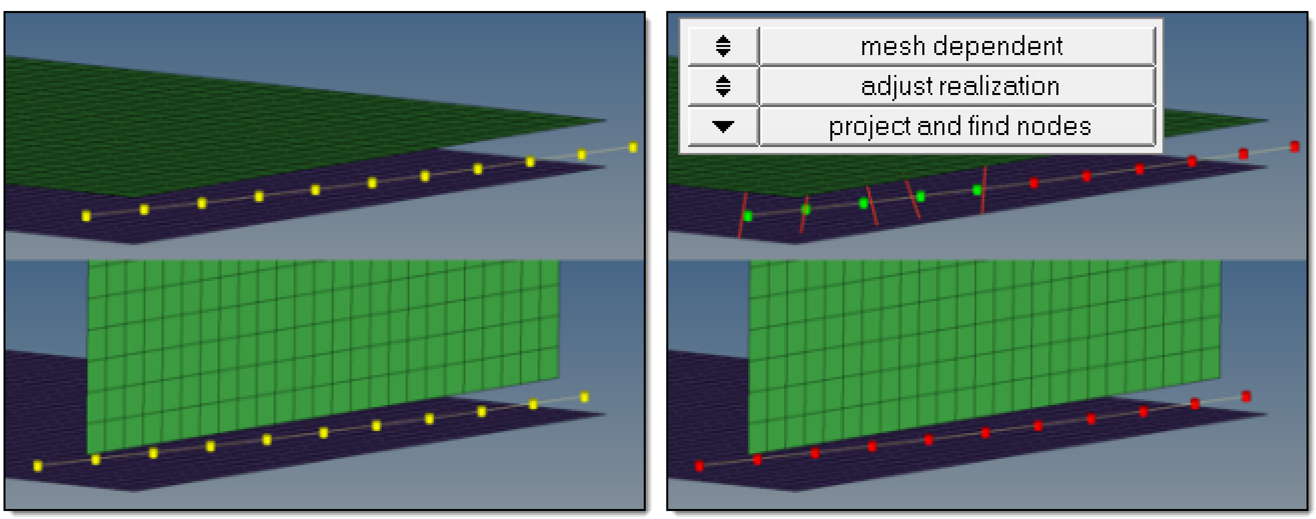


Figure 1052: Failed Realization

An angle of less than five degrees is considered normal. Activating the **non-normal projection** checkbox omits the requirement for a normal projection, and permits links to only be found in the connector tolerance. The result is exactly the same as it is for the find nearest node option.

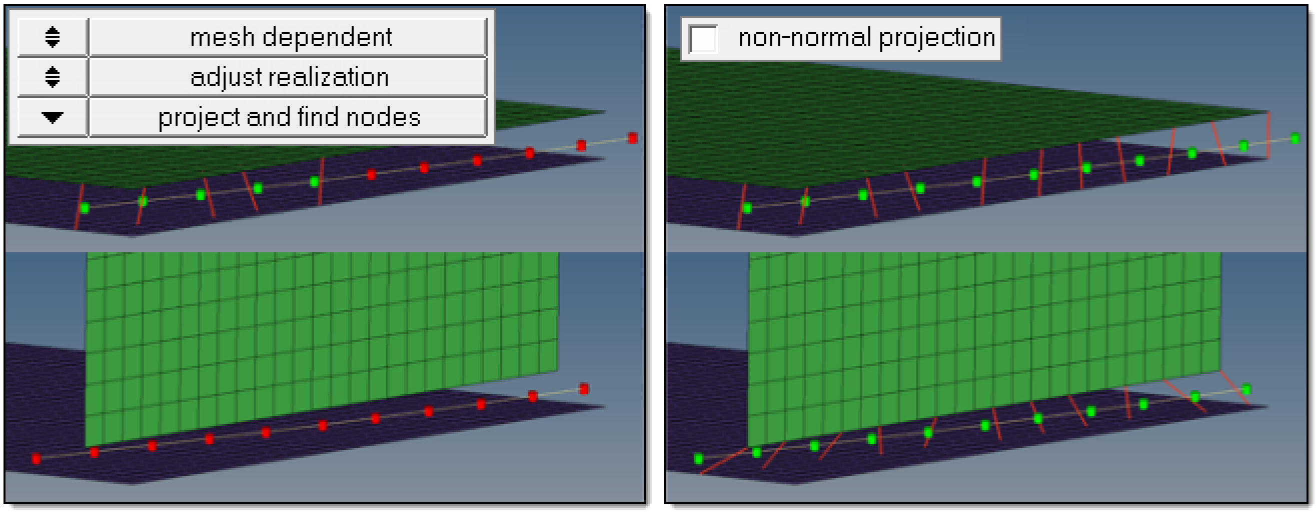


Figure 1053: Non-Normal Projection

Mesh Dependent – Adjust Realization – Ensure Projection

When the **ensure projection** option is selected, the minimum condition for the realization is a possible projection. The realization will be performed in the direction from one projection point to the next. If the projection point is coincident with a shell node they will be equivalenced.

Ensure projection is comparable to the older use shell node option, which is no longer available.

> **Note:** Ensure projection can lead to incompletely defined connections from a solver perspective unless the connector positions are not aligned to the mesh. The advantage of this projection method is the exact determination of the projection points.

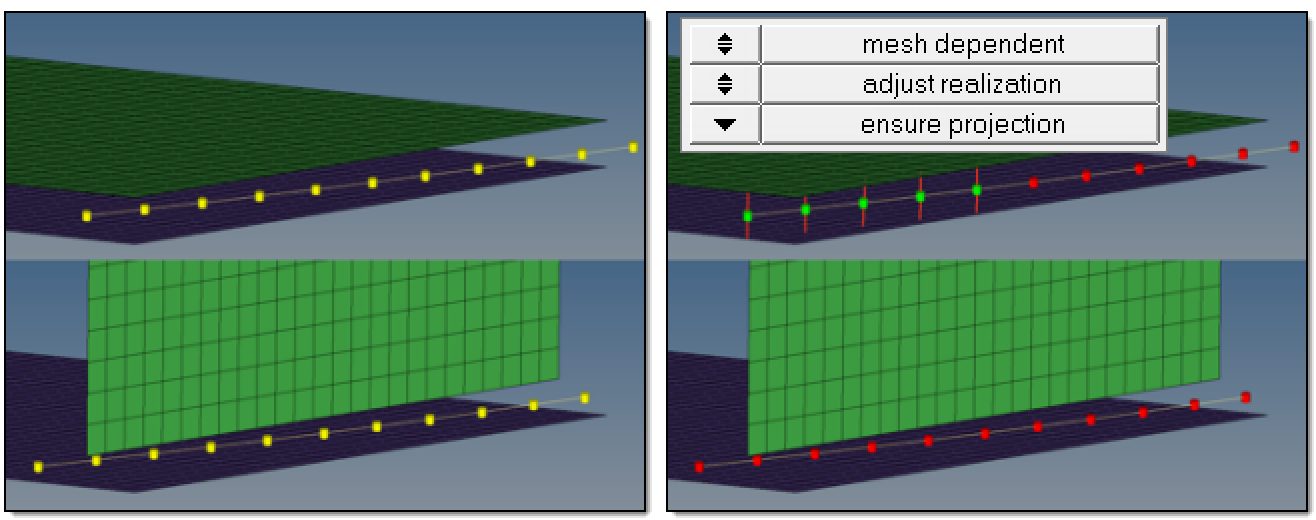


Figure 1054: Ensure Projection

Enabling the **non-normal projection** checkbox allows the realization to be performed from one projection point to the next.

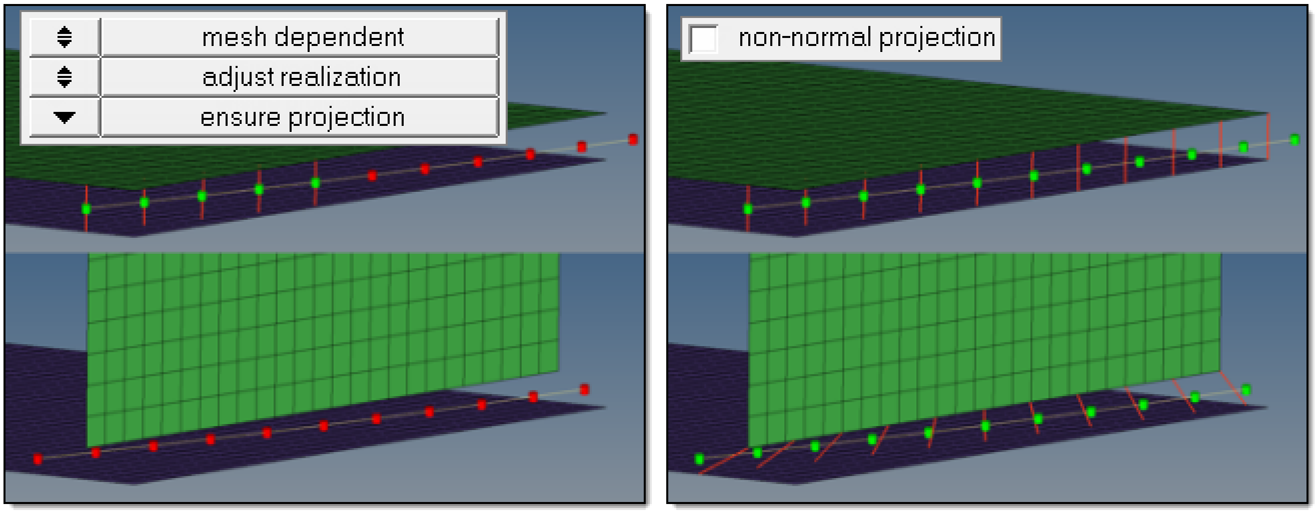


Figure 1055: Ensure Projection with Non-Normal Projection

Bolt Realization

Overview of the spot connector realization process and methods.

Bolt Realization Process

Overview of the bolt realization process.

1. Select the realization type.

mesh independent

Use for realizations which do not need a node connection, and the connection is primarily defined via a solver-specific card or the nodes which need to be connected are defined by a cylinder, such as Bolt (cylinder spring) for Radioss.

For all mesh independent bolt realization types a cylinder is defined. Bolt hole detection is not performed because holes are not required. All of the nodes inside the cylinder are considered part of the bolt realization unless they also belong to a link defined on the connector.

The cylinder dimension is primarily defined by its diameter, length L1 and L2.

- L1 points in the same direction as the connector vector and describes the distance from the connector position to the first end of the cylinder.
- L2 points in the opposite direction and measures the distance between the connector position and the second end of the cylinder.

Therefore, the connector vector is essential for these types of realizations.

If a vector is not predefined and is determined dynamically, the vector will always point from the connector position to the projection point on the farthest link.

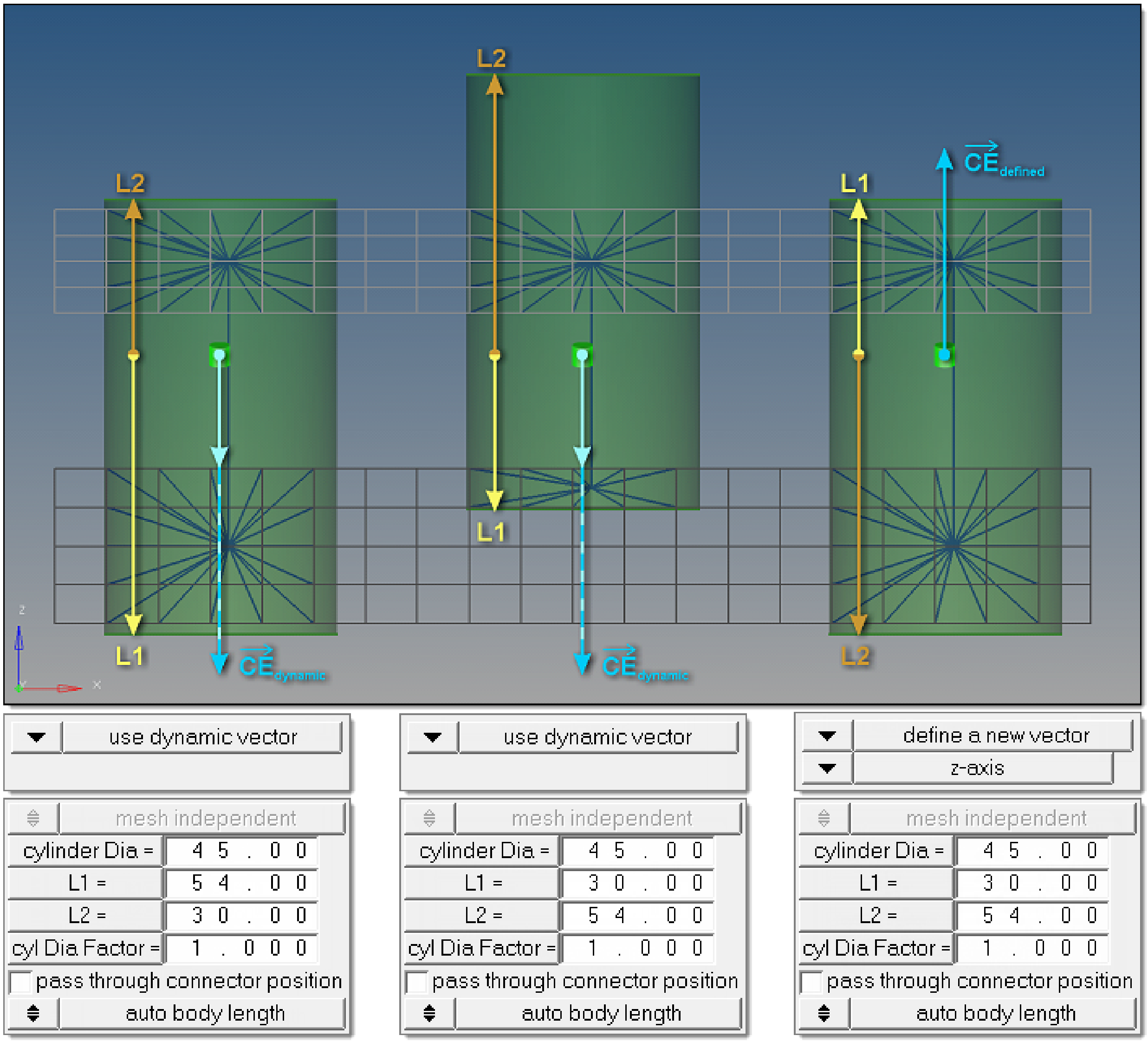


Figure 1056: Connector Vector

Dynamic vectors have been used in the first two cylinder. Interchanging the values for L1 and L2 leads to very different realizations. By comparison, the third cylinder uses the same values for L1 and L2 as in the second cylinder, but the result looks better because of the predefined connector vector in the global z-direction.

In some automated processes the cylinder diameter is automatically set to the bolt shaft diameter. This leads to failed cylinder bolts when the holes are properly modeled because the defined cylinder does not contain nodes. In such cases the cylinder diameter factor has been introduced; this factor is set to 1.0 by default and is a multiplier for the cylinder diameter to increase the final cylinder diameter when necessary.

> **Note:** The cylinder diameter as well as the cylinder diameter factor can be reviewed and modified in the Connector Browser. Connectors can also be rerealized with modified cylinder diameters and cylinder diameter factors without updating L1 or L2 in the browser. This is not possible from the panel.

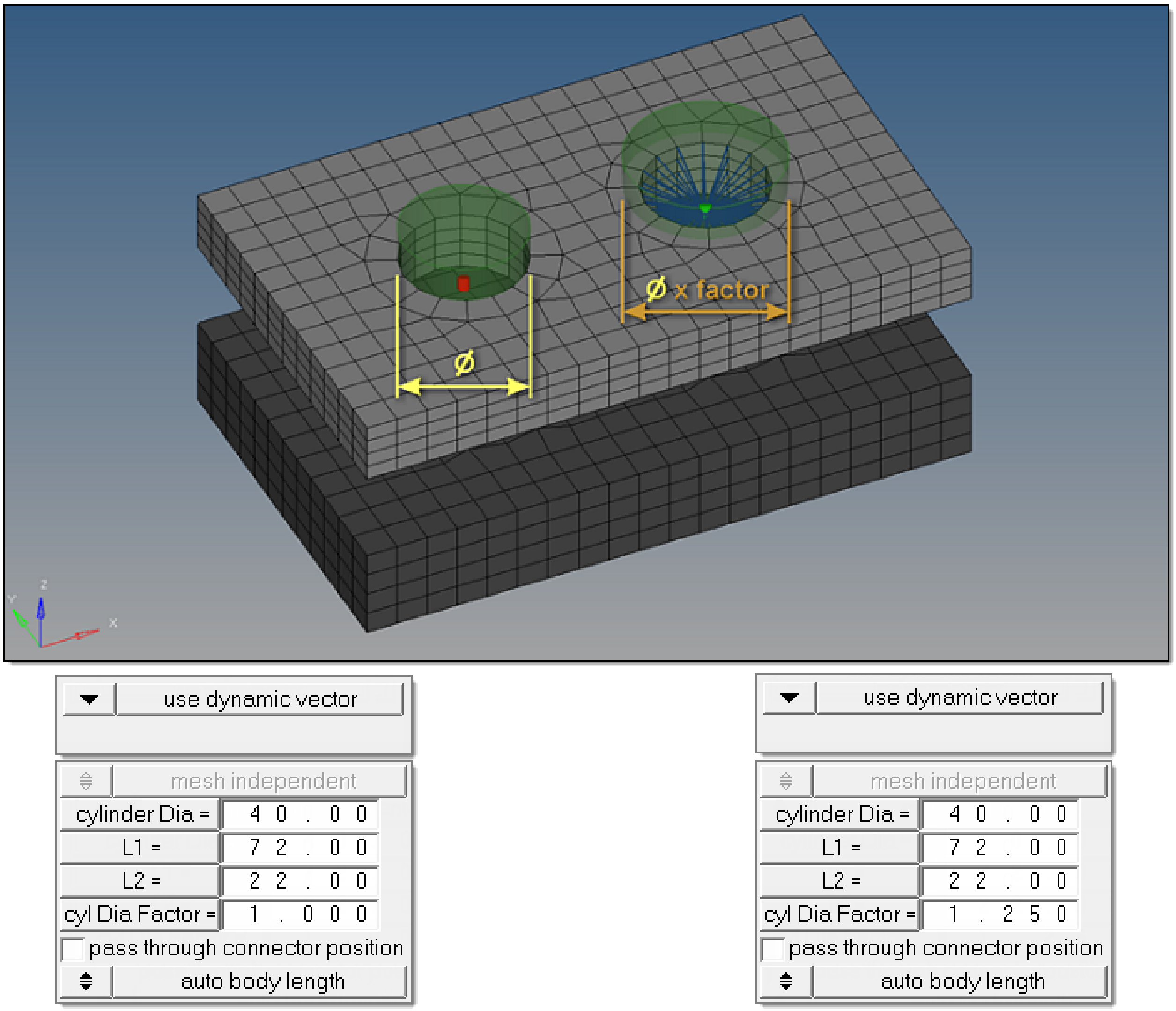


Figure 1057: Cylinder Diameter Factor

mesh dependent

Use for all other cases.

2. If you selected mesh dependent, you must determine:
   - Is the existence of holes in each layer requested upfront?
   - How should the 2D mesh be prepared before the final realization is performed?

In the past a bolt realization always required a hole for each layer in the initial mesh. This is no longer necessary for 2D meshes because the imprint capability punches the needed holes into the mesh before the final realization is performed, enabling the mesh to be manipulated in a pre-step. This makes it possible to punch holes, move holes, close holes, create washers, and so on.

Select one of the following:

Consider existing holes only

A minimum of one hole per layer must be available in the origin mesh. If holes do not exist the realization will fail. This is the default method and must be used for any type of solid meshes.

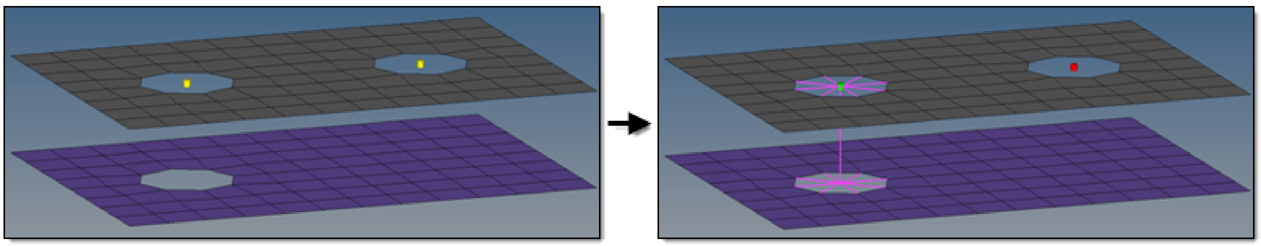


Figure 1058: Consider Existing Holes Only

Create hole, if none

If there are not any holes on a certain layer, they will be punched into that layer at the position of the projection point. The diameter of the new hole is defined on the Realization Details subpanel. This method is used if the model does not contain the appropriate amount of holes per bolt, but holes are required for specific realization types.

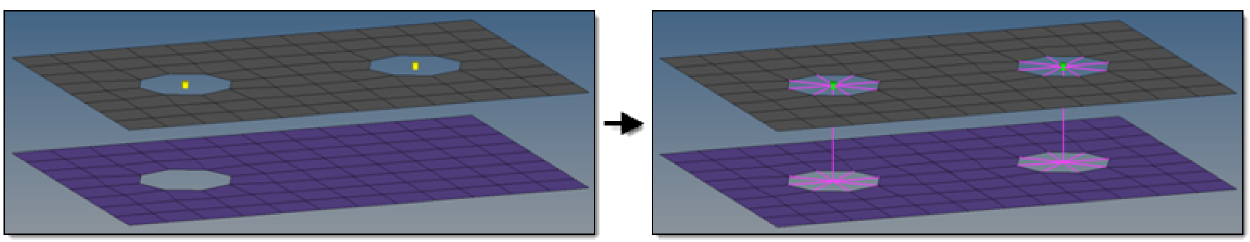


Figure 1059: Create Hole, if None

Use hole, if available

Creates hybrids (hole on one side only), but other combinations are allowed. On the mesh side (no hole), the connection is realized via the head elements defined in the chosen realization type. The head element(s) is/are created between the appropriate body element node and the nodes inside the diameter (no hole connection dia) defined on the Realization Details subpanel. This option is used for realization types which are not eager for holes.

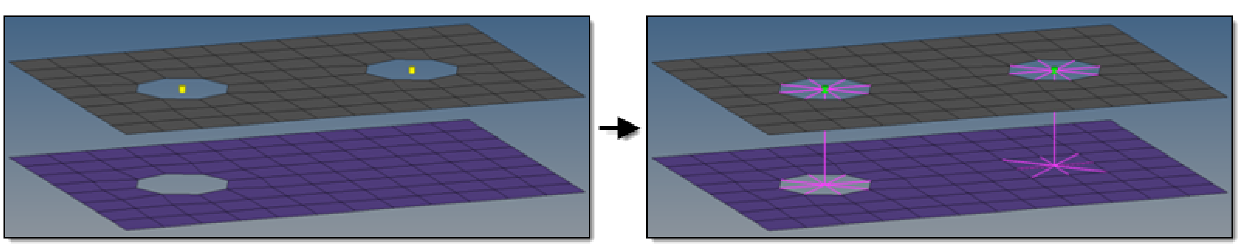


Figure 1060: Use Hole, if Available

Fill & remesh hole, if available

Use this option when you do not want the shape of holes to interfere with the mesh flow in the bolt region. The detected holes are closed and a remesh of the new elements and a few additional rows of adjacent elements is performed. The connection is realized via the head elements defined in the chosen realization type. The head elements are created between the appropriate body element nodes and the nodes inside the diameter (no hole connection dia) defined on the Realization Details subpanel.

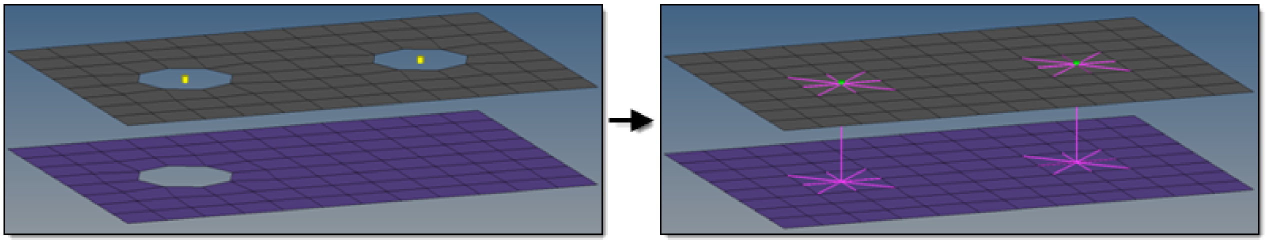


Figure 1061: Fill & Remesh Hole, if Available

> **Note:** The hole detection mechanism for the last three options takes into account a cylinder which is defined in the Hole Detection Details subpanel. The hole consideration cylinder option can be defined as a factor of the no hole connection dia option or as an exact diameter. Principally holes which are inside or touching the cylinder can be detected, but the hole consideration is reduced to one hole per link per connector. Additionally holes need to fit to the requested dimensions, which are also defined in the Hole Detection Details subpanel.

3. For the create hole, if none, and use hole options, a mesh modification including a hole movement is allowed. Therefore, you can determine whether to adjust the hole(s) or to adjust the realization.

Adjust hole position (2D)

Moves the center of a hole into the position of the projection point.

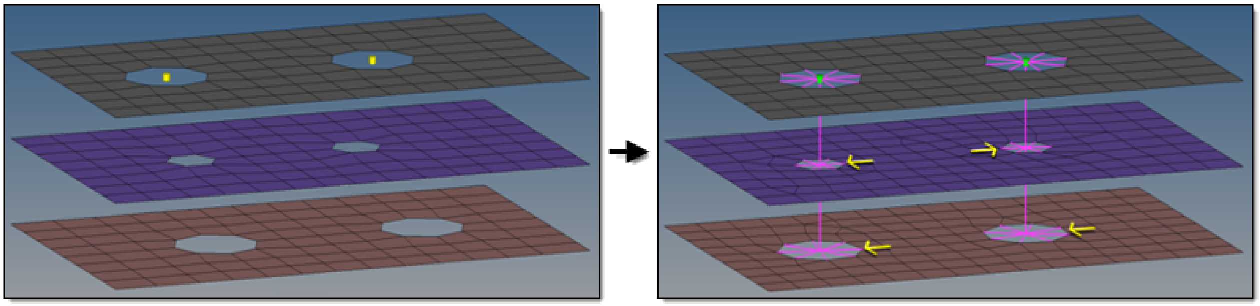


Figure 1062: Adjust Hole Position (2D)

Adjust realization

Hole positions are not modified, enabling the realized elements to compensate the nonaligned centers of holes.

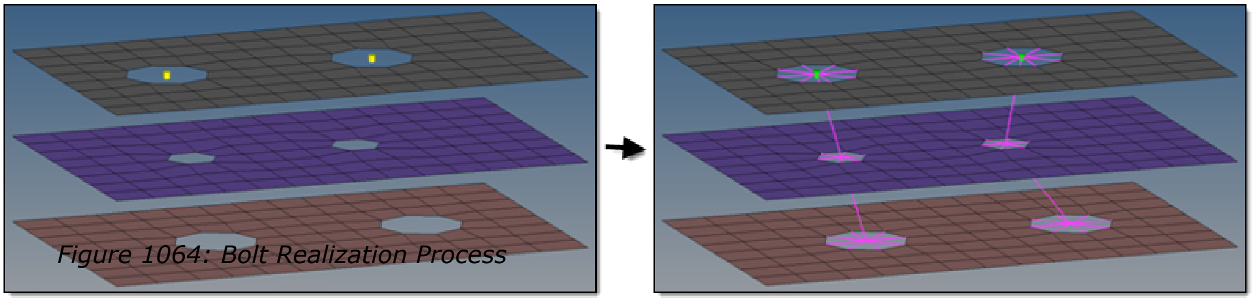


Figure 1063: Adjust Realization

4. This stage cannot be skipped, as it has great influence on stages 2 and 3, which is expressed by the large arrow underlying those stages in the flow chart.

   This stage contains all realize and hole detection details which need to be known before the final realization is done. The realize and hole detection subpanel can be accessed by clicking **realize & hole detect details** in the bolt and realize subpanels. This subpanel is organized into two additional subpanels: hole detection details and realization details.

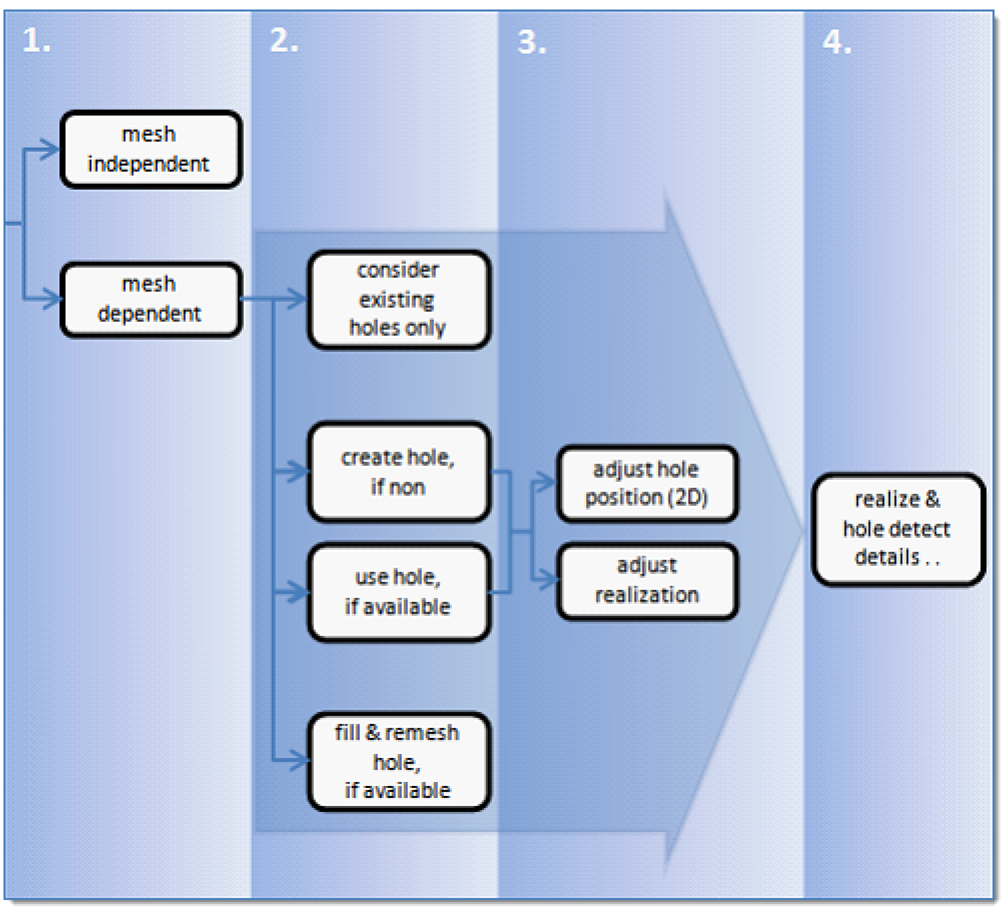


Figure 1064: Bolt Realization Process

Bolt Realization Methods

Overview of the different options for bolt realization methods.

Hole Detection Details

Dimension and Feature Angle

The minimum and maximum dimensions define which holes should be considered during bolt realization. The minimum and maximum feature angle define the features to be considered as hole edges for solid elements.

Hole Consideration Cylinder

Not all of the holes found in the given connector tolerance can be considered for the various bolt realizations.

- The connector tolerance, especially when set to a large value, detects many holes. To prevent detecting holes which are far away from the connector position and are not aligned with the other hole(s), the consideration cylinder excludes outer holes from the detection.
- Since the existence of a hole is not necessarily requested anymore, a space has to be defined where the holes are expected to be. It is no longer sufficient to use just the connector tolerance, therefore the hole consideration cylinder option performed along the projection path becomes necessary. All of the holes the cylinder touches or contains can be considered for the various bolt realizations.

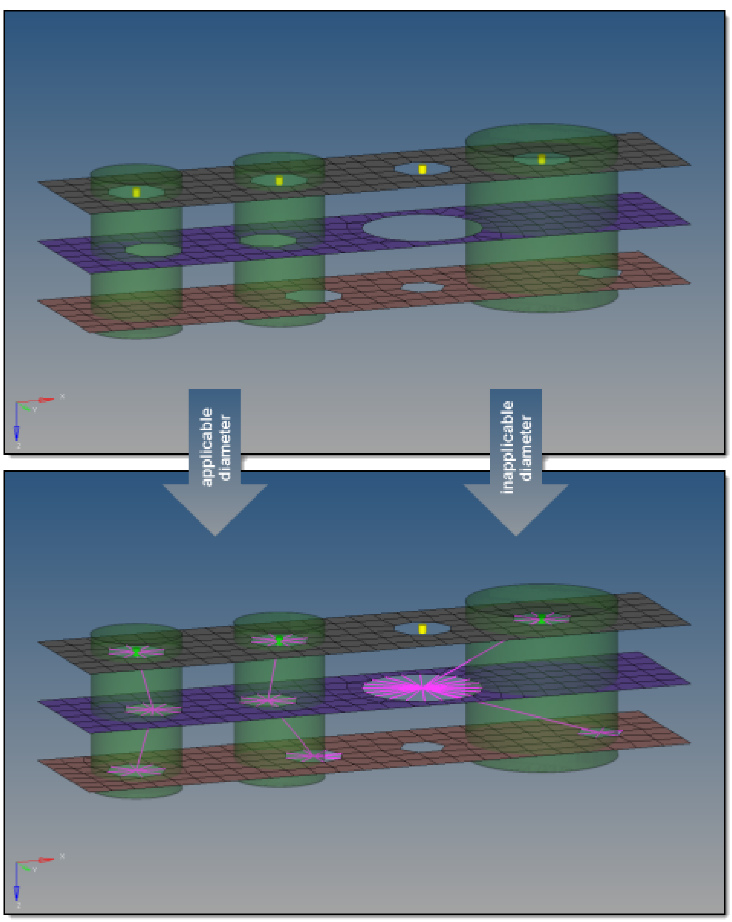


Figure 1065: Hole Consideration Cylinder

Define cylinder diameters in the following ways:

auto cylinder diameter (factor)
Factors given diameters, which include: create hole diameter (2D), create/adjust hole diameter (2D), adjust hole diameter (2D), and no hole connection diameter. The first available diameter is used. The default factor is 1.5.

exact cylinder diameter
Specify an exact diameter. The default diameter is 15.0

> **Note:** The hole consideration cylinder option is not offered when using the **consider existing holes only** option.

Realization Details

The realization details subpanel contains additional information about the exact treatment of holes and no holes, if bolt realization is performed. Dependent on the options selected in stage 2, varying subsets of the options are offered.

Diameter and adjustments options include:

create hole diameter (2D)
Create new holes with the specified diameter. Used if holes are required by the create hole, if none option.

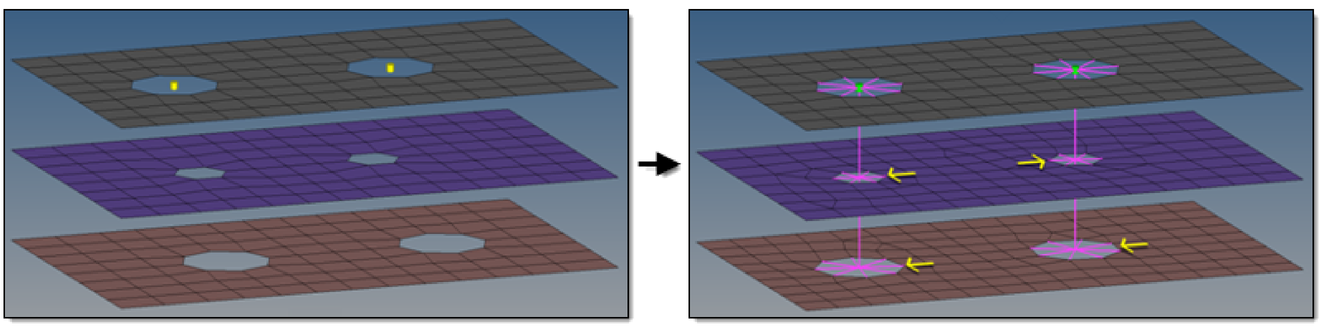


Figure 1066: Create Hole Diameter (2D

create and adjust hole diameter (2D)
Create new holes with the specified diameter and adjust existing holes with the specified diameter, which leads to bolt realizations with the exact same diameter on all links. Used if holes are required by the create hole, if none option.

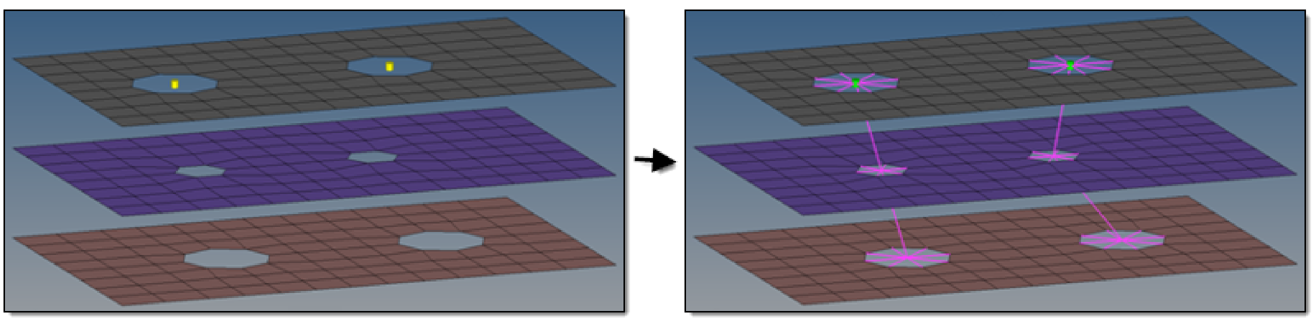


Figure 1067: Create and Adjust Hole Diameter (2D

adjust hole diameter (2D)

Adjust existing holes to the specified diameter. The do not adjust hole diameter option switches off the adjustment and uses the holes with their origin size. Used if holes are not necessarily required when using the use hole, if available option, but the existing holes need to have a specific diameter.

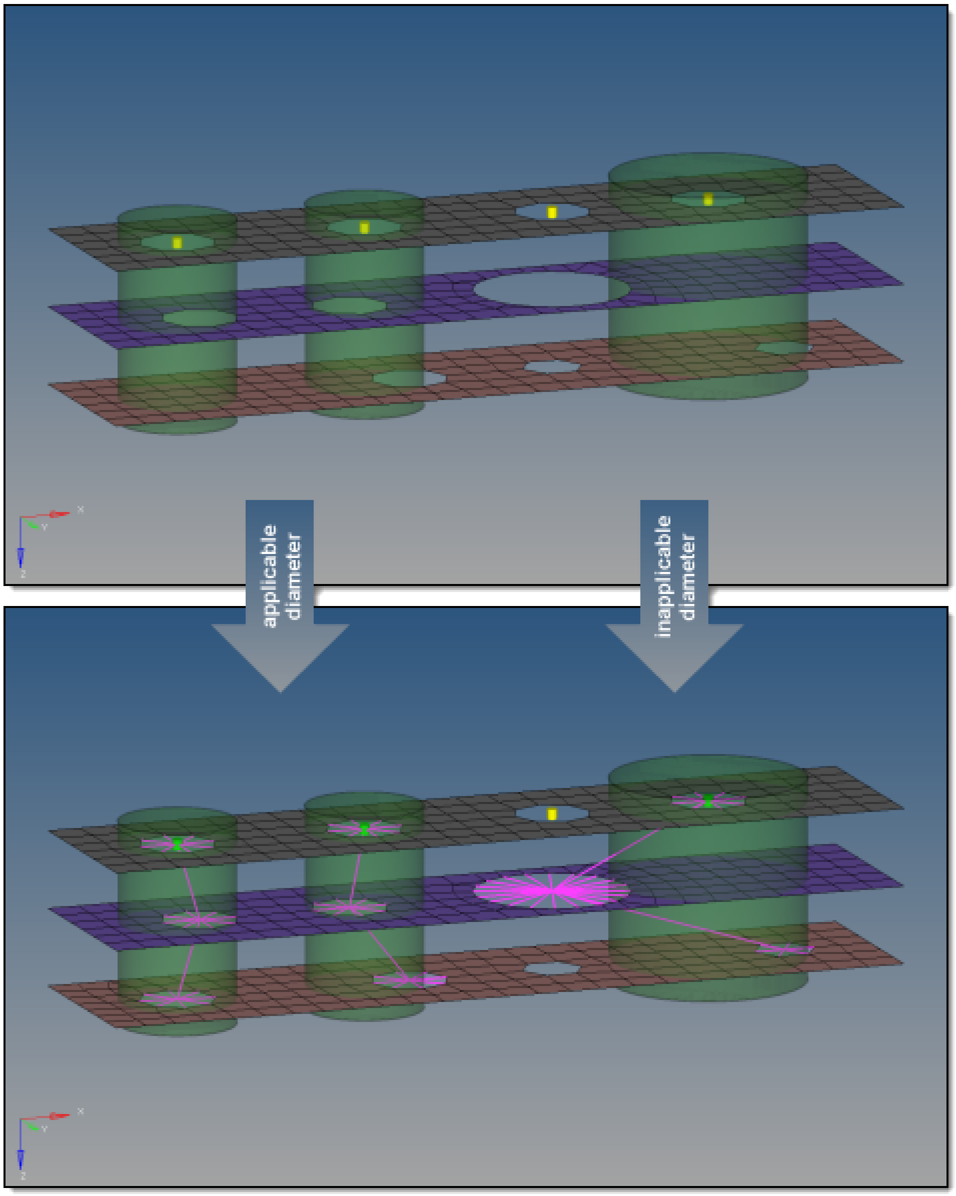


Figure 1068: Adjust Hole Diameter (2D)

no hole connection diameter

Connect a link without an available hole by joining the nodes found inside the circle with the specified diameter around the projection point via head elements. Used if holes are not required when using the use hole, if available option or fill and remesh hole, if available option.

Hole filling and number of nodes around holes options include:

fill holes (2D)

Fill detected holes during bolt realization. There are various quad patterns available, which cause a remeshing of the area around the hole.

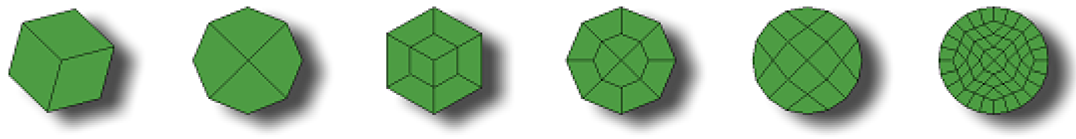


Figure 1069: Quad Patterns

Holes can also be filled with pie pieces. If the number of pie pieces is defined, the surrounding mesh is remeshed. The pie pieces preserve option also creates pie pieces, but takes the existing hole nodes into account and prevents the remeshing.

> **Note:** Activating the fill holes (2D) option deactivates the no. of nodes around hole option.

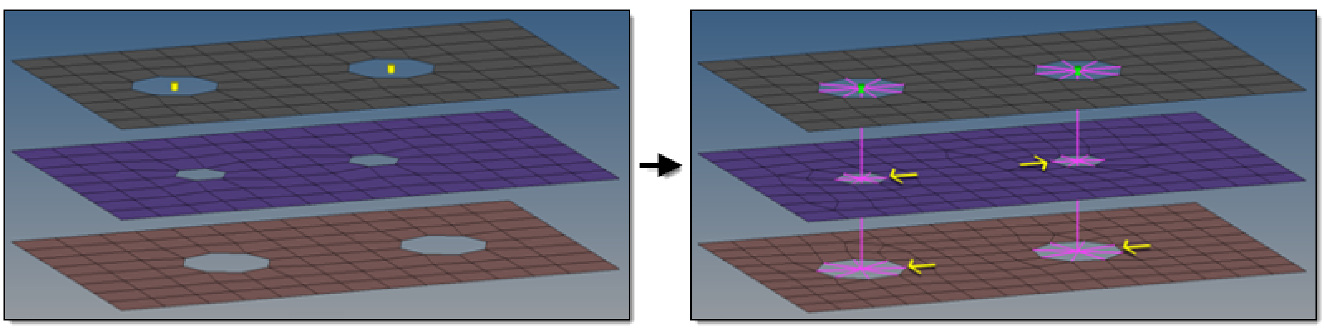


Figure 1070: Pie Pieces Preserve

no. of nodes around hole

preserve

Use the number of nodes of the appropriate origin hole. This is the default option which prevents the surrounding mesh from being remeshed. For new holes, the auto option is used.

density

Specify the exact number of nodes (default is 8). The surrounding mesh gets remeshed.

elem size

Specify an element size (default is 5.0). The number of elements around the hole is calculated based on this size. This is the preferred option for extremely different hole diameters. The surrounding mesh gets remeshed.

auto

Perform a node distribution based on the underlying mesh size. The number of nodes is always rounded to an equal number.

For 2D holes, one or two washer layers can be created, after which the surrounding mesh gets remeshed.

The width of the washer can be defined by:

- Factoring the hole radius.
- Directly specifying the exact width.

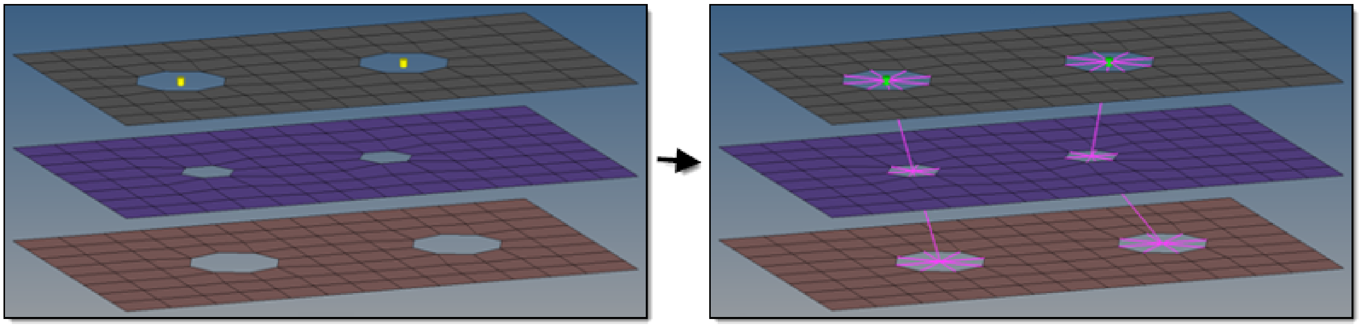


Figure 1071: Define Washer Width

Seam Realization

Overview of the spot connector realization process and methods.

Seam Realization Process

Overview of the seam realization process.

1. Select the realization type.

   mesh independent

   Use for realizations which do not need a node connection and the connection is primarily defined via a solver-specific card, such as LLINKs for PAM-CRASH.

   mesh dependent

   Use for all other cases.

2. If **mesh dependent** is selected, you must decide whether or not to adjust the mesh or the realization.

   Adjust mesh

   Projection is done in a perpendicular way, and the mesh has to be adapted to the projection points.

   Adjust realization

   The mesh will not be modified, at the expense of non-normal or incomplete realizations. Many realization types are defined with head elements attached to body elements. In the case of these realization types, the head elements realize the connection without modifying the mesh, and the body elements are created in a normal direction.

Create washer layer options:

3. Choose how the adjustments should take place.

   Adjust mesh

   Sub-options include: quad transition and remesh.

   Adjust realizations

   Sub-options include: find nearest nodes, project and find nodes, and ensure projection.

4. Choose whether or not the imprint should be skipped for quad transition.

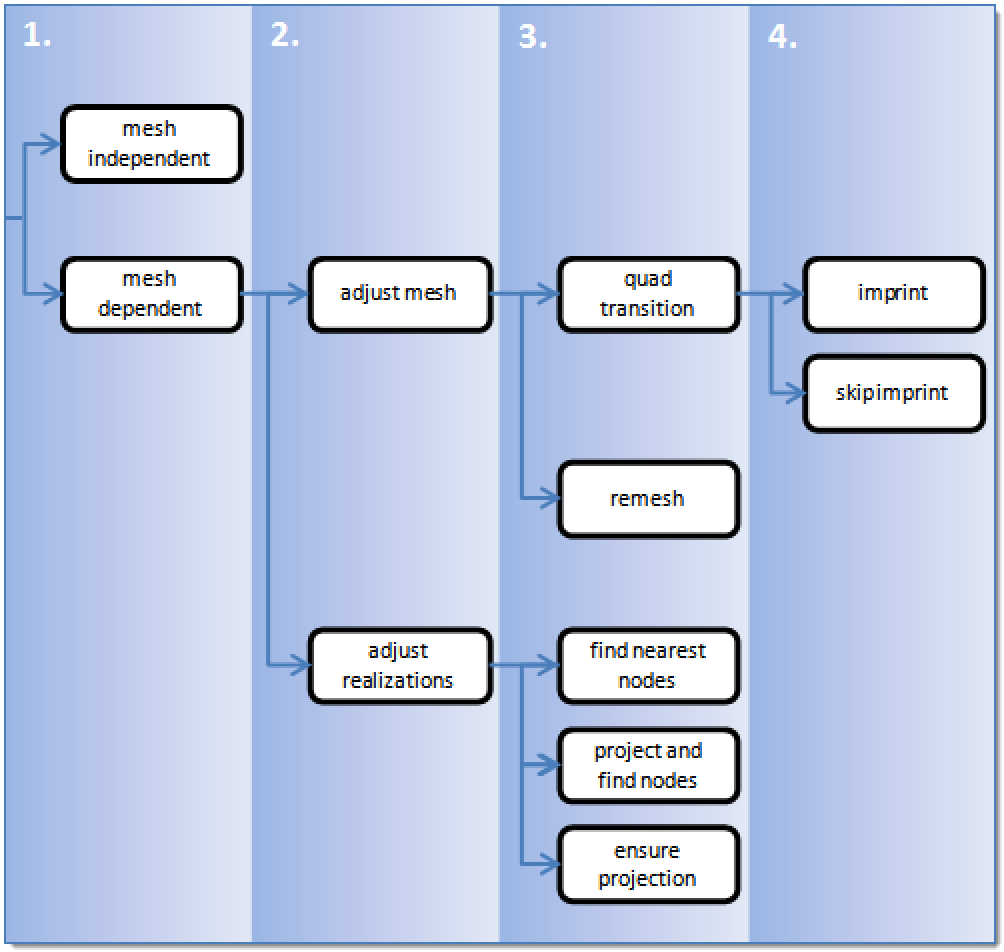


Figure 1072: Seam Realization Process

Seam Realization Methods

Overview of the different options for seam realization methods.

Mesh Independent

The mesh independent option is normally used for solver-specific realization types, then a post script is performed during realization to define the solver specific connection. For example, for the PAM-CRASH LLINK all necessary solver specific cards are created along with the realization.

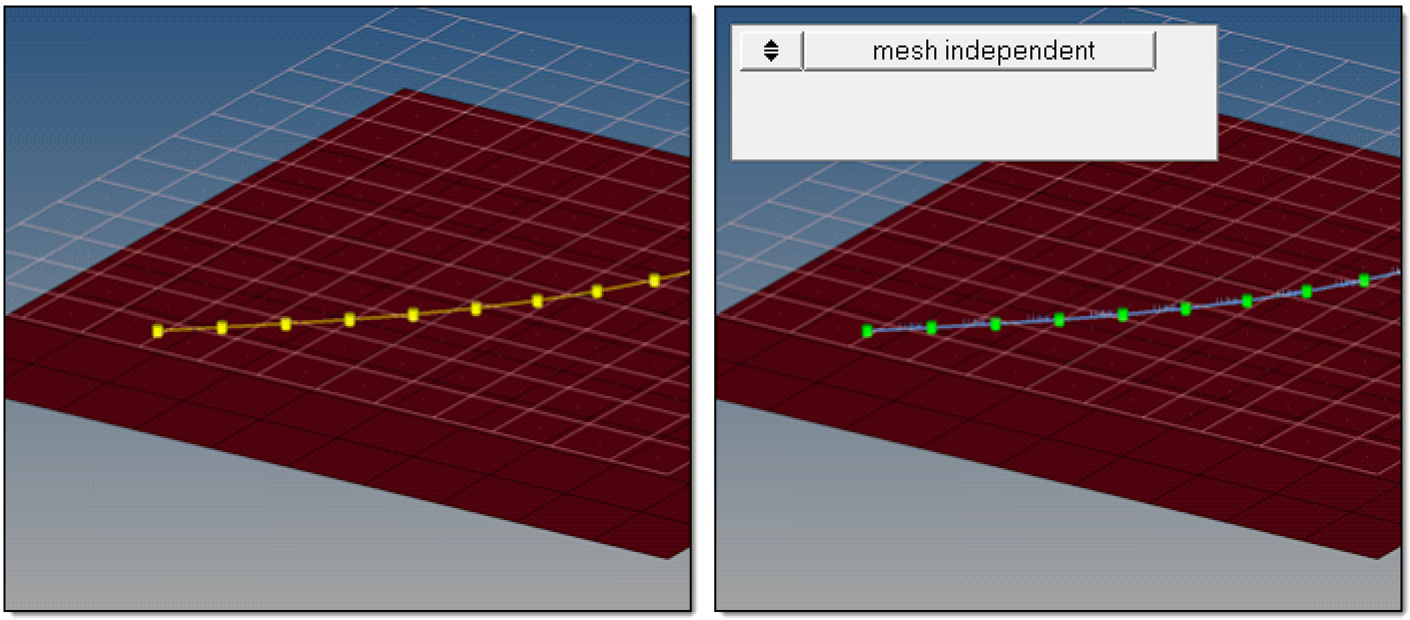


Figure 1073: Mesh Independent

Mesh Dependent – Adjust Mesh – Quad Transition - Imprint

Quad Transition

The quad transition option creates perfectly shaped quad elements around the projection line. The quad size is determined by the average mesh size. From one projection point to the next, exactly one pair of elements is created. You can also use this option to create seams from quad elements, and realize the connections to the links through perfectly modeled t-edges.
In certain limits, the mesh automatically snaps to important features. This prevents the creation of elements that are too small, and ensures that the geometry is not modified too much.
The considered feature angle can be defined individually for each connector. Feature edges below 10.0° will not be taken into account, whereas features above 25.0° and free edges will always be taken into account.
By default, snapping is allowed by a distance of one third of the quad pattern element size. In the case of a predefined quad pattern element size of 10.0, the outer nodes can snap to features in a distance of 3.3. The algorithm also tries to snap all three nodes of a quad pattern or none.

Imprint

When creating mesh-dependent realizations with quad transitions, the quad transition meshes can overlap and disturb each other if more than one set of connectors is created too close to each other. The imprint option reconciles such transitions with each other and modifies the underlying mesh to match the results to create a final result that is seamless and properly meshed.

To enable smaller imprint conflicts to be automatically resolved when connectors are realized, the resolve conflicting imprints option is activated by default. Overlapping elements are released, and a normal remesh of that area is performed as long as the overlapping area is smaller than half the regular quad transition element size. Larger conflicts may require a manual imprint.

To allow smaller imprint conflicts to be automatically resolved when connectors are realized, the **resolve conflicting imprints** checkbox is enabled by default. Overlapping elements are released, and a normal remesh of that area is performed as long as the overlapping area is smaller than half the regular quad transition element size. Larger conflicts may require a manual imprint.

The size of the imprint can be determined using the pitch size (use pitch size to imprint) or using the average size of the underlying mesh (use avg. mesh size to imprint). If you want to define a specific imprint size, select size to imprint.

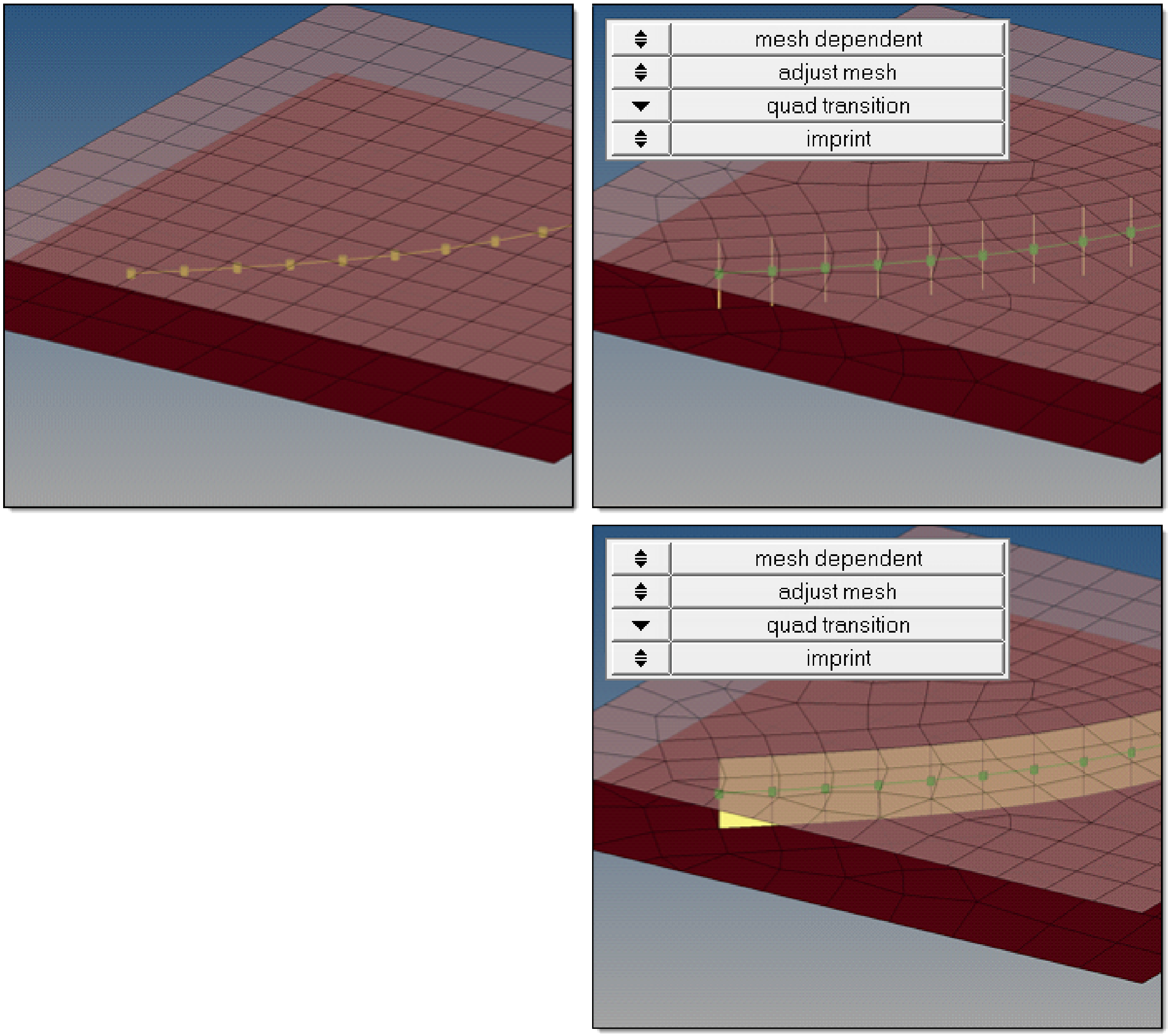


Figure 1074: Imprint

Mesh Dependent – Adjust Mesh – Quad Transition – Skip Imprint

Quad Transition

The quad transition option creates perfectly shaped quad elements around the projection line. The quad size is determined by the average mesh size. From one projection point to the next, exactly one pair of elements is created. You can also use this option to create seams from quad elements, and realize the connections to the links through perfectly modeled t-edges.

In certain limits, the mesh automatically snaps to important features to prevent the creation of elements that are too small, and to ensure that the geometry is not modified too much.

The considered feature angle can be defined individually for each connector. Feature edges below 10.0° are not taken into account, whereas features above 25.0° and free edges are always taken into account.

By default, snapping is allowed by a distance of one third of the quad pattern element size. In the case of a predefined quad pattern element size of 10.0, the outer nodes can snap to features in a distance of 3.3. The algorithm also tries to snap all three nodes of a quad pattern or none.

Skip Imprint

The skip imprint option prevents the last step of quad transition from being performed. The component ^conn_imprint is created instead, which contains the element pattern. These elements can be modified and manually imprinted later using the Connector Imprint panel.

Skip imprint enables you to realize such mesh-dependent realizations, even in very complex areas of the model where the automatic imprint fails because of issues such as conflicting seams.

The size of the imprint can be determined using the pitch size (use pitch size to imprint) or using the average size of the underlying mesh (use avg. mesh size to imprint). If you want to define a specific imprint size, select size to imprint.

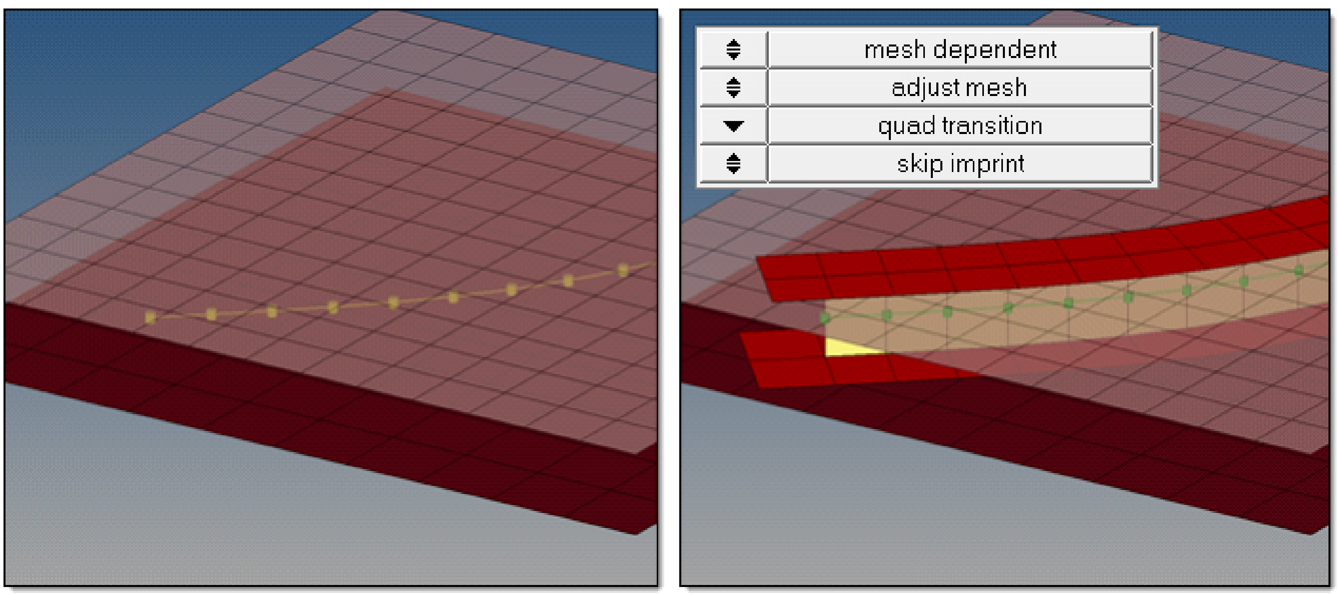


Figure 1075: Skip Imprint

Mesh Dependent – Adjust Mesh – Remesh

The remesh option uses snap and split capabilities to connect 1D welds to the links in the position of the projection points. In the case of a quad realization, remesh looks for a correct t-edge.

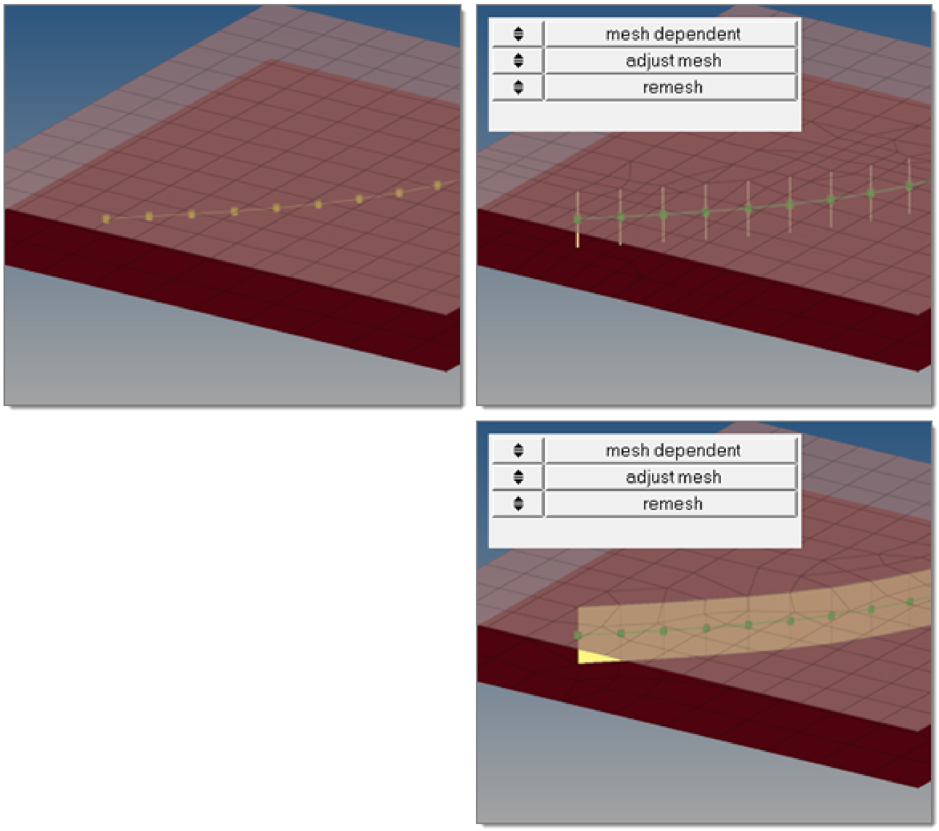


Figure 1076: Remesh

Mesh Dependent – Adjust Realization – Find Nearest Nodes

The find nearest node option searches for the nearest nodes within the given tolerance, making it possible to connect t-joints and similar areas. This option is very useful in situations where the connectors are not positioned perfectly. These realizations are allowed to be non-normal. Find nearest nodes does not do any projection.

> **Note:** If the connector points are close to each other and two of these points find the same closest nodes the connector fails.

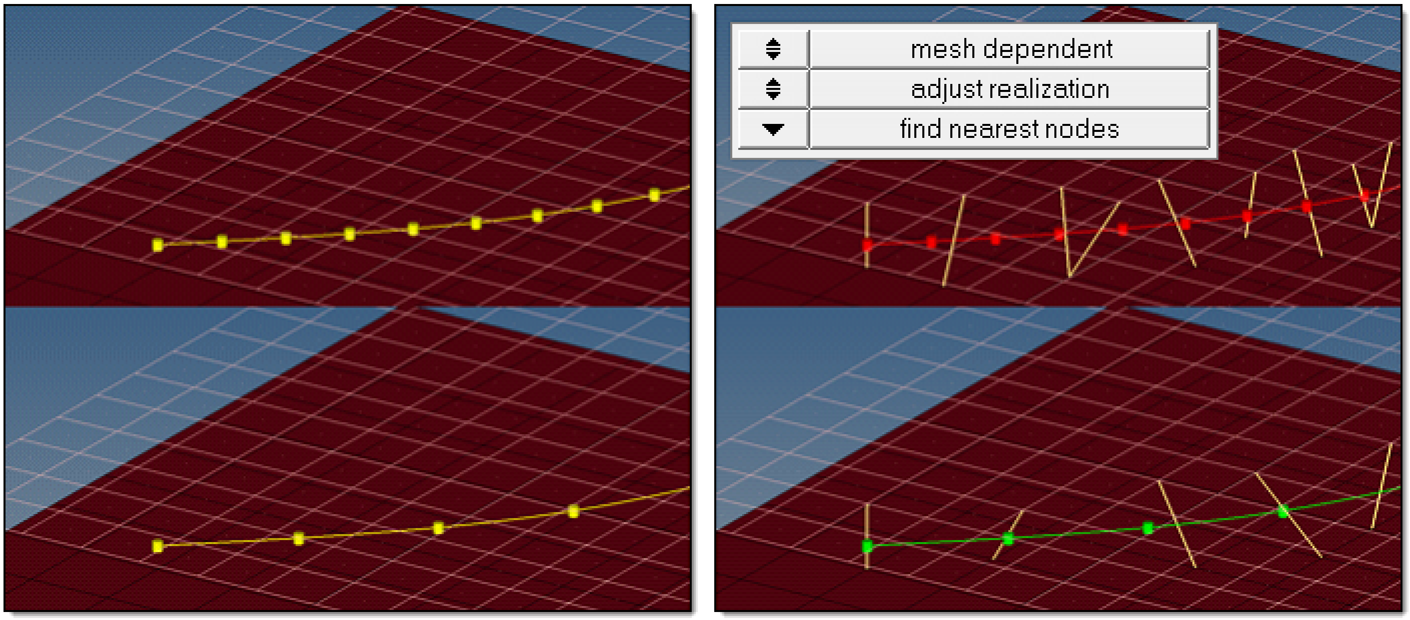


Figure 1077: Find Nearest Nodes

Mesh Dependent – Adjust Realization – Project and Find Nodes

The project and find nodes option produces the same exact result as find nearest node because a non-normal projection for seams is always allowed. Principally, project and find nodes requires a valid projection onto the link entities in the first step. In the second step, the nodes closest to the projection points will be used for the connection. If the projection (connector tolerance) is not possible, the realization fails.

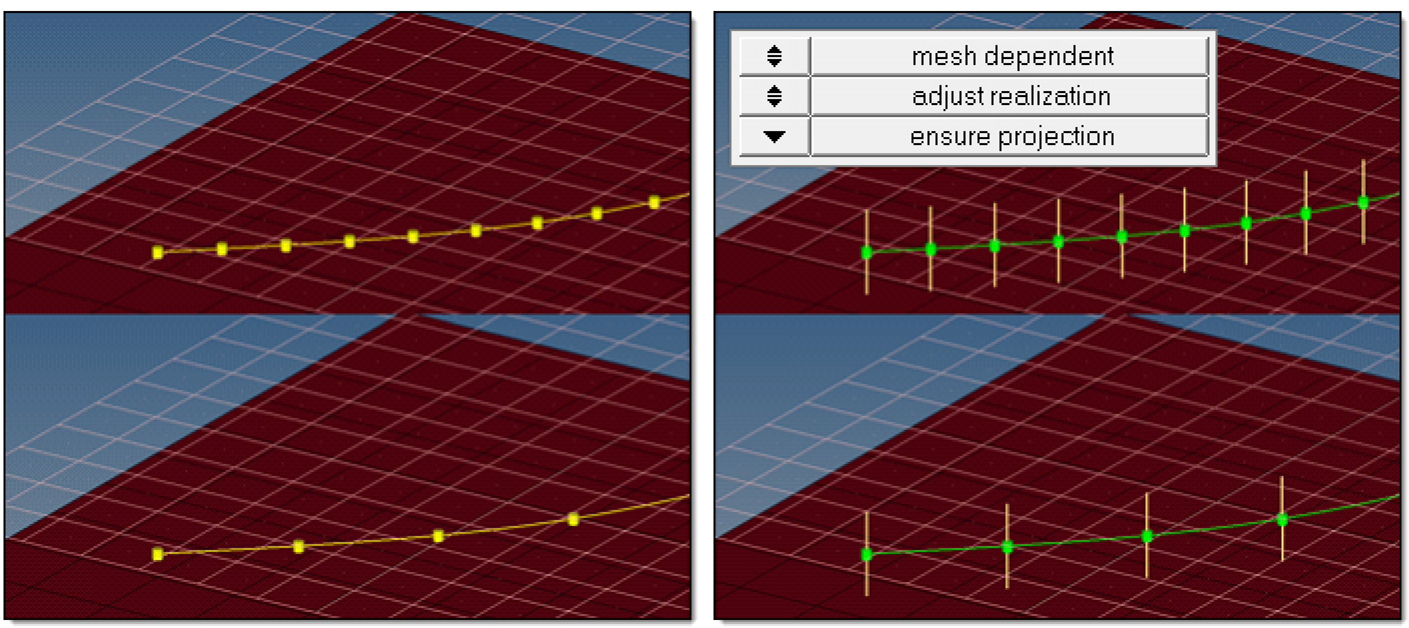


Figure 1079: Ensure Projection

Mesh Dependent – Adjust Realization – Ensure Projection

When using the ensure projection option, the minimum condition for the realization is a possible normal projection. The realization will be performed in the direction from one projection point to the next. If the projection point is coincident with a shell node they will be equivalenced.

Ensure projection can lead to incompletely defined connections from a solver perspective unless the connector positions are not aligned to the mesh. The advantage of this projection method is the exact determination of the projection points.

> **Note:** The realization fails when the connector points are close to each other, and two of these points find the same nodes.

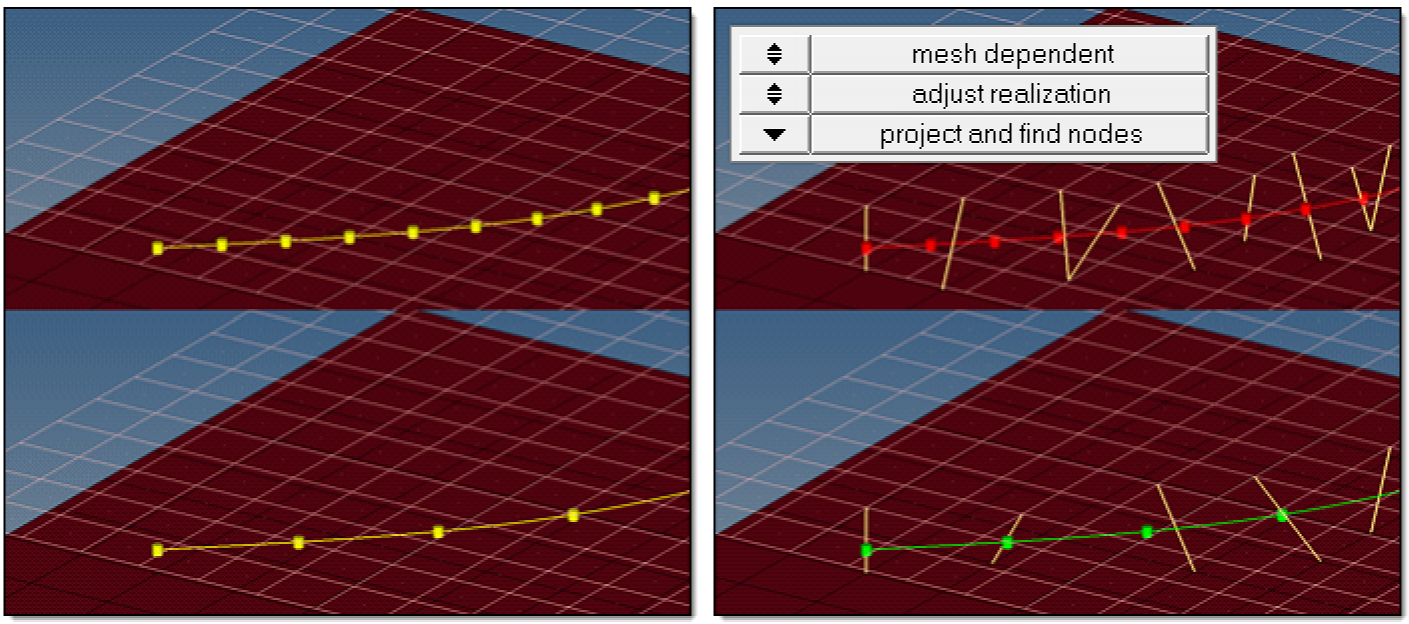


Figure 1078: Project and Find Nodes

Area Realization

Overview of the area connector realization process and methods.

Area Realization Process

Overview of the area realization process.

Each test point can be considered as a single spot. The methods are the same and the results look identical.

> **Note:** The quad transition is not yet implemented for area connectors, even though the panel offers these options. Instead of quad transition, a simple adjust realization – ensure projection is performed.

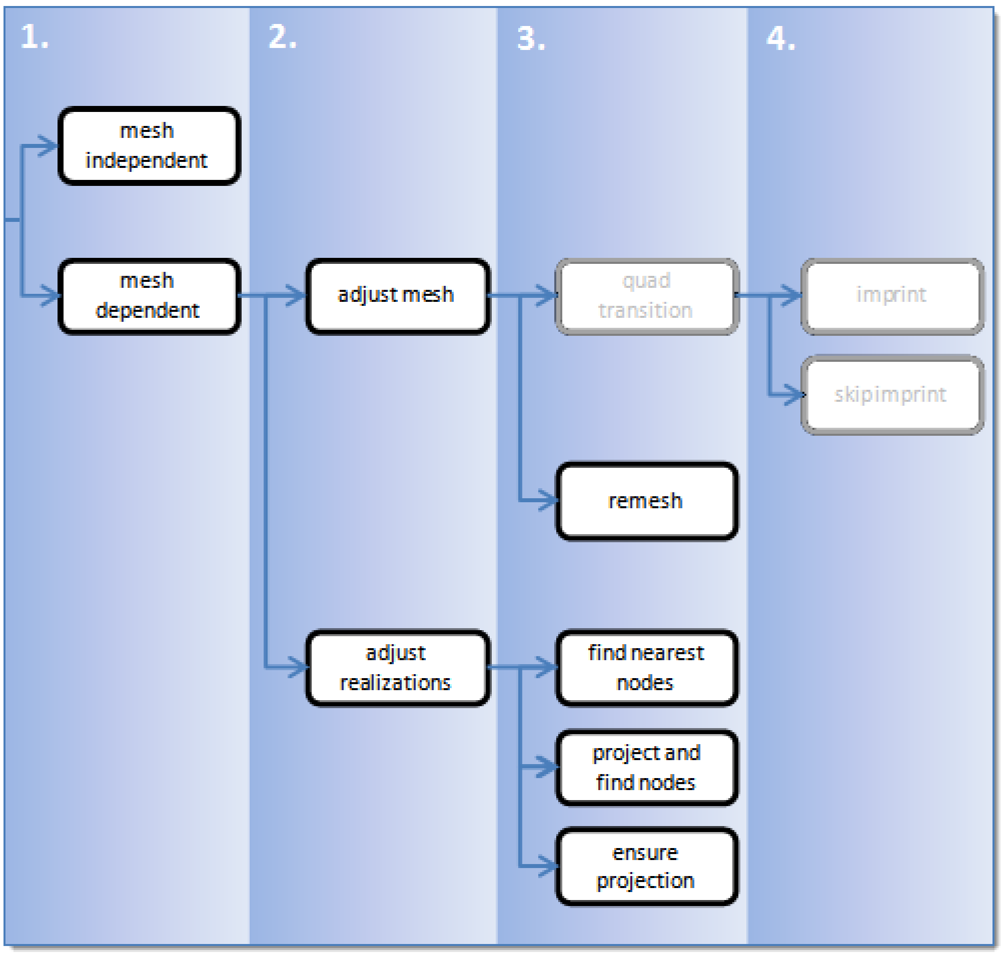


Figure 1080: Area Realization Process

Special Realization Types

Learn about special connector realization types.

Hexa Nugget

Use the Hexa Nugget realization to create hexa clusters between shell components.

Figure 1081: HAZ Dimensions

Contacts are defined between the shell components and the appropriate hexa nodes. A heat affected zone for the shells from ultra-high strength steel material is also created.

Hexa Nugget realizations can be used for any amount of parallel combinations of shell components.

> **Note:** The Hexa Nugget realization is only available in Radioss and LS-DYNA user profiles, and can only be selected and defined in the Connector Entity Editor on existing spot connectors.

Define the heat affected zone dimensions using the parameters available in the Connector Entity Editor. The dimension and property of each heat affected zone (HAZ) can be separately defined. Each setting is stored and considered per individual connector.

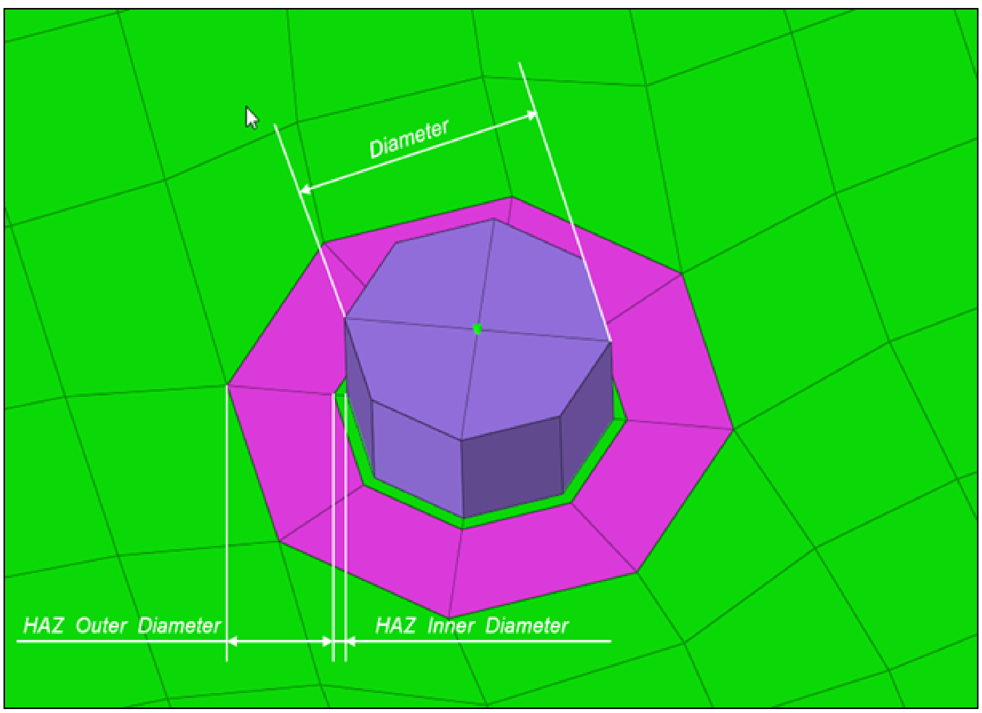


Figure 1081: HAZ Dimensions

General Info

Table 254: General Info Parameters

Parameter	Action
Tolerance	Specify a distance from the connector location. Only the entities within this tolerance can be taken into account for the final realization. The tolerance is used to verify whether adequate link candidates are available to be connected with respect to the number of layers.

Weld Shape

Table 255: Weld Shape Parameters

Parameter	Action
Hexa Number	Create a hexa cluster of 4 hexas, arranged in a predefined pattern. **Note:** By default, Hexa Number is set to 4 hexa. You cannot edit this field.
Coats	Specify the number of hexa elements required along the thickness. **Note:** By default, Hexa Coats is set to 1 for a hexa nugget connector. You cannot edit this field.

Realization Details

Use the Realization Detail parameters to define the dimension of the welds.

Table 256: Realization Details Parameters

Parameter	Action
Diameter Option	Choose how the diameter is defined. This field is used for realizations based on hexa elements, where the size of the realized element (hexa) is created based on the diameter value. **diameter** Specify a single diameter value. **diameter mapping file** Use a diameter mapping file to calculate the diameter. The diameter mapping file obtains diameter values that you assigned to a range of flange thicknesses in the Diameter Table. Along with flange thickness ranges, you can also specify the main flange thicknesses to consider when assigning diameter values.
Hexa Thickness Option	Project the hexa spot to touch the shell elements. The position is independent from any thickness. **Note:** By default, Hexa Thickness Option is set to shell gap. You cannot edit this field.

Connectivity Info

Table 257: Connectivity Info Parameters

Parameter	Option
Connectivity	**Note:** By default, Connectivity is set to contact. You cannot edit this field.

Parameter	Option
HAZ	Create the quad weld elements and stitch them to both links by adjusting their mesh. Perform all required HAZ.
Enable Condition HAZ	Control which parts need to receive heat affected zones during the realization.
Skip Imprint	Create the quad weld elements, but do not change the meshes of the links. Instead, create additional elements to represent the requested HAZ. These elements are organized in the ^conn_imprint component. You can use these elements for a later manual imprint after they have been manipulated to your need. This can be helpful in more complex areas, where the standard imprint functionality fails, for example, when working with conflicting connectors. **Note:** The quad weld elements are not attached to the links. The connection needs further attention.

Contact Info

Table 258: Contact Info Parameters

Parameter	Action
Contact Creation Option	Choose a method for creating contacts. **one contact per link** Create one exclusive contact with two exclusive sets per each link. Create the contact between the link and the hexa nugget nodes laying on this link. The nodes of one hexa nugget belong to two different contact definitions. • Groupname(s): hexa_nugget_contact_Linkname • Setname(s) Master: hexa_nugget_master_part_set_Linkname • Setname(s) Slave: hexa_nugget_slave_node_set_Linkname **one contact per link pairing** All hexa nugget connectors using the same links are sharing the same contact and sets. • Groupname(s): hexa_nugget_contact_Linkname1_Linkname2_Linkname3(when 3 layer)

Parameter	Action
	• Setname(s) Master: hexa_nugget_master_part_set_Linkname1_Linkname2_Linkname3 • Setname(s) Slave: hexa_nugget_slave_node_set_Linkname1_Linkname2_Linkname3 **single contact** Create one contact and two sets for this realization type in the model. • Groupname: hexa_nugget_singlecontact • Setname Master: hexa_nugget_single_master_part_set • Setname Slave: hexa_nugget_single_slave_node_set

Auto Correction

Table 259: Auto Correction Parameters

Parameter	Option
Allow Reposition	Reposition hexa elements and HAZ elements. If the option is deselected, the connector will fail when the connector is too close to free or feature edges where the nugget and HAZ elements cannot be accommodated.
Reposition Tolerance	Specify a maximum allowable distance by which hexa elements and HAZ elements are repositioned based on connector location to edge.
Control Distance	Choose a method for controlling distance. **for close free edges** Only consider the connectors which are positioned close to free edges for repositioning at the time of realization. **for close free and feature edges** Only consider connectors for repositioning at the time of realization which are positioned close to free edges as well as feature edges.
Requested Distance	Choose whether the distance to reposition is based on minimum distance to imprint or a minimum distance to connector projection.

Parameter	Option
Distance	Choose whether to manually specify a distance or use the minimum element size from the element criteria file.

HAZ Info

Table 260: HAZ Info Parameters

<table>
<tr><th>Parameter</th><th>Option</th></tr>
<tr><td>HAZ Material Condition</td><td>Enable the selection of UHSS material which controls which parts need to receive heat affected zones during the realization.</td></tr>
<tr><td>UHSS Material Option</td><td>Choose a method for selecting UHSS material.
From current model
Select an existing material from current model.
In the Connector Entity Editor, the UHSS Material Entity ID’s field is populated with the number of the materials selected.
From search file (default)
Select material from a search file.
For Hexa Nugget Connector realizations the material search file name is <code>materialsnippets.txt</code>. HyperMesh searches for this file in the following locations and in the following order:
1. Installation: <code>[hm_scripts_dir]/ connectors/ materialsnippets.txt</code>
2. User directory: <code>[USER_HOME]/ materialsnippets.txt</code>
3. HyperWorks configuration path folder: <code>[HW_CONFIG_PATH]/ materialsnippets.txt</code>
4. Current working folder: <code>[CURRENTWORKINGDIR]/ materialsnippets.txt</code>
In the Connector Entity Editor, the UHSS Material File field is populated with the name of the file that was found last.
The text file contains snippets from the material names, which need to receive heat affected zones during the realization.
From connector metadata
Once a connector is realized with the UHSS Material Option “From search file”, the folder name is written as metadata to the connector in a relative manner to allow the exact same rerealization in a different work environment as long as the same <code>materialsnippets.txt</code> files are saved in according folders.</td></tr>
<tr><td>HAZ Dimension Scheme</td><td>Choose a setting for defining HAZ diameters.</td></tr>
</table>

Parameter	Option
HAZ Inner Diameter	**values** Specify an absolute value for the HAZ Inner Diameter. **Note:** This value must be greater than the value used for defining the nugget diameter. **scale factors** Calculated as a scale factor from the nugget diameter. **Note:** This value must be greater than or equal to 1.0. **offsets** Specify a HAZ Inner Diameter which is offset from the nugget diameter.
HAZ Outer Diameter	**values** Specify an absolute value for the width of the first layer around the HAZ Inner Diameter. **scale factors** Specify a width for the first layer around the HAZ Inner Diameter. Calculated as a scale factor from the nugget diameter. **offsets** Specify a width for the first layer around the HAZ Inner Diameter which is offset from the HAZ Inner Diameter. **Note:** For values and scale factors, this value must be greater than the value used for defining the HAZ Inner Diameter.
HAZ Layer 2	Enables you to specify the HAZ Outer Diameter 2. By default, this checkbox is cleared and the HAZ Outer Diameter 2 cannot be defined.
HAZ Outer Diameter 2	**values** Specify an absolute value for the width of the second layer around the HAZ Outer Diameter. **scale factors** Specify a width for the second layer around the HAZ Outer Diameter. Calculated as a scale factor from the nugget diameter. **offsets** Specify a width of the second layer around the HAZ Outer Diameter, which is offset from the HAZ Outer Diameter.

Parameter	Option
	Note: For values and scale factors, this value must be greater than the value used for defining the HAZ Outer Diameter.

Property and Material Info

Use the Property and Material Info parameters to define the properties and materials of the welds and the heat affected zones (HAZ).

Table 261: Property and Material Info Parameters

Parameter	Option
HAZ Material Option	Choose a method for assigning material for the HAZ. **Use original material** Assign a new HAZ component with the same material that was assigned to the original components. **Use copy of original material** Duplicate the original materials and assign them to new components. The original material is duplicated. The new material name will be the same as the original material with _HAZ_mat as a postfix. **Select a material** Select a material from the current model via the select material for HAZ option. HAZ components are created with the same name as the selected material with property ID as a postfix. When set to <unspecified>, a default material is taken from the `[hm_scripts_dir]/connectors/hexa_nugget/dyna/hexa_nugget_HAZ_material_default.ini` file. HAZ components are created with name hexa_nugget_HAZ_mat_default_ and with the property ID as a postfix.
HAZ Property Option	Choose a method for assigning a property to HAZ. **Use original property** Assign a new HAZ component with the same property that was assigned to the original components. **Use copy of original material** Duplicate the original properties and assign them to new components.

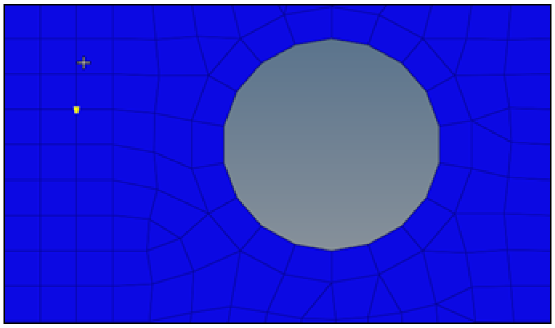
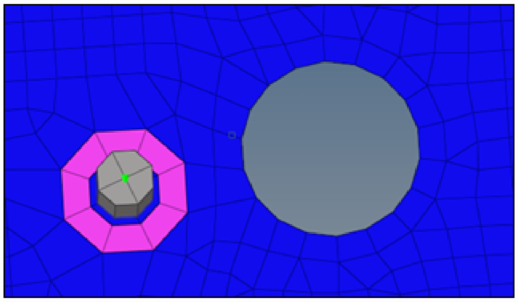
Parameter	Option
	The original property is duplicated. The new property name will be the same as the original property with _HAZ_prop as a postfix. **Scale original thickness** Create a new property and component for each link that has a HAZ imprinted. The property is a copy of the original. Properties are named as <originalname>_<HAZ_prop>_<scaled thickness>, and components are named as <matname>_<scaled property name>. **Input thickness** Create a new property and component for each link that has a HAZ imprinted. The property is a copy of the original. Property is named as <hexa_nugget_HAZ_prop>_<thickness_value>, and components are named as <matname>_<Input thichness prop name>. **Select a property** Select a property from the current model via the select property for HAZ option. HAZ components are created and named using the UHSS material with the property ID as a postfix. For example, <UHSS matname>_<prop ID>.
Nugget Material Option	Choose a method for creating and assigning material to nuggets. **Create one material per each link combination** Create one material for each link combination. The weld material is named <hexa_nugget_weld_mat>_<link 1 ID/name>_<Link 2 ID/name>. If the link with ID 1 and the link with ID 2 are connected, than the weld material which gets created will be named hexa_nugget_weld_mat_1_2. **Create one material per each number of layer** Create one material based on the number of layers the weld is connecting to. The weld material is named <hexa_nugget_weld_mat>_<link 1 ID/ name>_<No. of layers>. If it is a 2T weld, than the mat name will be hexa_nugget_weld_mat_2L. If it is a 3T weld, than it will be named hexa_nugget_weld_mat_3L.

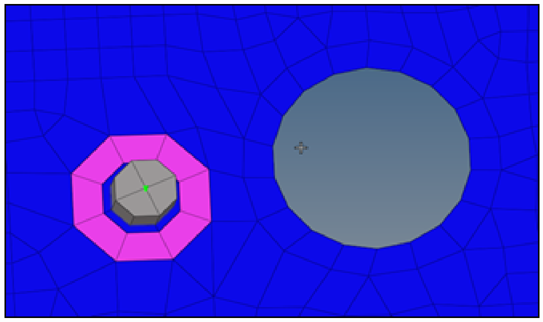
<table>
<tr><th>Parameter</th><th>Option</th></tr>
<tr><td></td><td>Use one single weld material
Select a weld material from the current model using Select Nugget Material Option. The selected material will be assigned to all weld elements.</td></tr>
<tr><td>Itemize Multilayer Connections</td><td>Itemize a multilayer connection. For example, in the case of a three layer connection, the two hexa clusters will be handled separately and will be assigned two different materials. At the same time, a two layer connection between the same two links will be assigned another material.
Names are created as following:
hexa_nugget_weld_material_link1_link2_(link3)
Ideally the links are ordered as <code><outer involved link>_<inner involved link>_(<uninvolved comp>)</code>, so that the two links involved come first, and the link(s) that are not involved are written in brackets.
hexa_nugget_weld_material_link3_link2_(link1)
Ideally the links are ordered as <code><outer involved link>_<inner involved link>_(<uninvolved comp>)</code>, so that the two links involved come first, and the link(s) not involved are written in brackets.
hexa_nugget_weld_material_link1_link2, hexa_nugget_weld_material_link2_link3
All materials are derivatives from <code>[hm_scripts_dir]/connectors/hexa_nugget/radioss/hexa_nugget_weld_material_default.ini</code>
Note: Only available when Nugget Material Option is set to Create one material per each link combination.</td></tr>
<tr><td>Select Nugget Property</td><td>Select a property from the current model.</td></tr>
</table>

Behavior

Table 262: Behavior Parameters

Parameter	Option
Non-normal	Attempt to realize connectors if the shell layers are not normal to each other.

Parameter	Option
Project Hexa Faces To Shell	Investigate and project each individual node, enabling you to expect and accept penetrations. In some border cases, during realization not all nodes of the appropriate hexa(cluster)-side are positioned exactly on the shell. This is due to some simplifications during the projection routine because it is estimated that the shells will not be too curvy and will be pretty plain. Therefore, not every node is individually investigated and projected to the same plane as the first one.
Ensure Hexa Projection	When enabled, all hexa node projections to a link that are beyond the specified tolerance (0.01*diameter) are marked as an error. **Tip:** Enable this checkbox when working with connections that use a contact definition.
Preserve Washer	Choose a method for preserving washers during the realization. Initial situation with perfectly meshed washers.  *Figure 1082: Original* After realization, with no washer preservation. The washers have been opened.  *Figure 1083: No Washer Preservation* After realization, with washer preservation and remesh allowed. The washers are still intact, but the mesh seeding has been changed.

<table>
<tr><th>Parameter</th><th>Option</th></tr>
<tr><td></td><td>
Figure 1084: Preserve Washer, Allow Remesh
After realization, with washer preservation and no remesh allowed. The washers have been fully preserved.

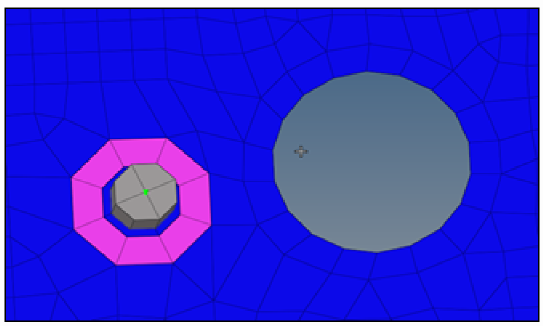



Figure 1085: Preserve Washer, No Remesh</td></tr>
<tr><td>Feature Angle</td><td>Specify a value used to determine important features, which need to be kept when doing the imprint.
Note: Features crossing the HAZ and close by cannot be kept.</td></tr>
<tr><td>Post Collector Name Setting</td><td>While realizing a hexa nugget connector, choose to name the collectors created after the realization with link names or link IDs.</td></tr>
</table>

Parameter	Option
Post Collector Name Separator	Choose a separator to use between collector names.

HiLock

The HiLock realization type can be used for any more or less parallel combination of `PSHELL` and `PCOMP` elements, and creates a 1D element construct consisting of `RBAR`, `CBAR` and `CBUSH` elements.

Restriction: Available for Nastran and OptiStruct solver interfaces.

The outer extensions represent the thicknesses of the outer shell elements. The inner nodes of the `RBAR` element are connected to the shell elements whereas the inner nodes of the `CBAR` elements are coincident to the shell nodes. Between the appropriate connected and coincident nodes `CBUSH` elements are created. Each outer node connects one `CBAR` and one `RBAR`.

Each HiLock connection gets its own coordinate system with the z-axis collinear to the HiLock direction. All affected nodes are assigned to this coordinate system, which is taken into account for the DOF definition of the `CBAR` elements, the stiffness calculation of the `CBUSH` elements, and the DOF of the node constraint.

This realization uses the shell properties and materials (`PSHELL` or `PCOMP`) and a definable HiLock material to calculate the exact position of the outer nodes and the stiffness of the `PBUSH` elements.

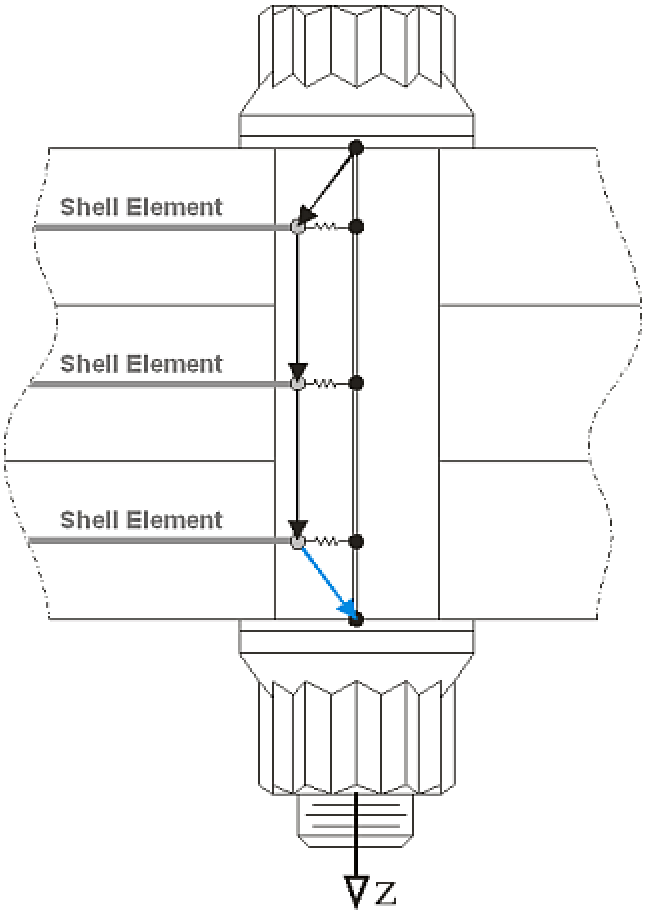


Figure 1086:

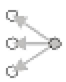
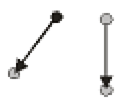
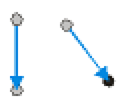
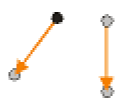
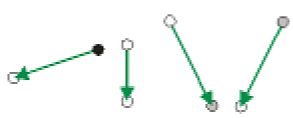
Component		DOF's for Rigid Elements and Linear Gaps for Fastener Axis in z-Direction
	CBAR	
	CBUSH	1245
	RBE2	12
	RBAR	345
	RBAR	45
	RBAR	3456
	RBAR	3

Figure 1087:

Details and Requirements

Certain conditions must be met for reliable realization.

- In the case of composites, only `PCOMP` cards in which the laminate option is either blank or set to SYM (symmetric) are supported. MEM, BEND, SMEAR, and SMCORE options are not supported for HiLock realization, and will cause realization to fail if used.
- The joined shells should be parallel to each other and planar.
- The fastener should be perpendicular to the shells.
- The z-axis of the element system, the material system, and the fastener system should be collinear.
- Stiffness is calculated assuming that the shells are perfectly planar and parallel. Small deviations will produce insignificant changes in predicted stiffness, but larger ones would require a system transformation.
- The shell elements which share a node with the HiLock (separate for each layer) should have the same properties, same materials, same material orientations, and similar sizes. The attributes of the element upon which the projection falls is assumed to be the same as the other surrounding elements (no averaging method is used.)

If attributes necessary for the stiffness calculation are missing, the connector fails and an error message displays in the status bar.

The realization requires available nodes near the connector position. If sufficient nodes are not available, the created elements are not collinear anymore and the HiLock gets a questionable geometry.

Organization and Definition of HiLock Realization

All HiLock elements (`RBARs`, `CBARs`, `CBUSHs`) created during the realization process are organized into a component named HiLock.

The following property collectors are created:

HiLock_PBAR_<diameter>
: Created with the `PBAR` card associated with it. The `RBAR` elements reference this property. The attributes are calculated depending on the diameter specified in the Spot panel during realization.

HiLock_PBUSH_<translational stiffness>_<rotational stiffness>
: Created with the `PBUSH` card associated with them. The `CBUSH` elements reference this property. The attributes are calculated depending on the chosen HiLock material and the properties and materials of the connected shells (`PSHELL` and/or `PCOMP`).

The following load collector is created:

HiLock_SPC6
: SPCs are created for each HiLock are moved into it.

The following system collector is created:

HiLock
: Systems created during the realizations will be moved into this collector. If this system collector already exists, any newly created systems will be moved into the same collector.

If a HiLock material is not selected, a default material is created.

HiLock_MAT1
: This material will be assigned to the `PBAR` cards, and can be found in the following folder of the installation directory: `[hm_scripts_dir]/connectors/HiLock_Mats`.
The predefined values are:

set E 1.8+07

set G 4.7e+04

set NU 0.330

set RHO 8.9e-09set A 1.7e-05

HiLock Material Option

When defining a HiLock connector, the HiLock Material Option can be selected for individual connectors using the Connector Entity Editor.

From current model
: Select an existing material from the current model.

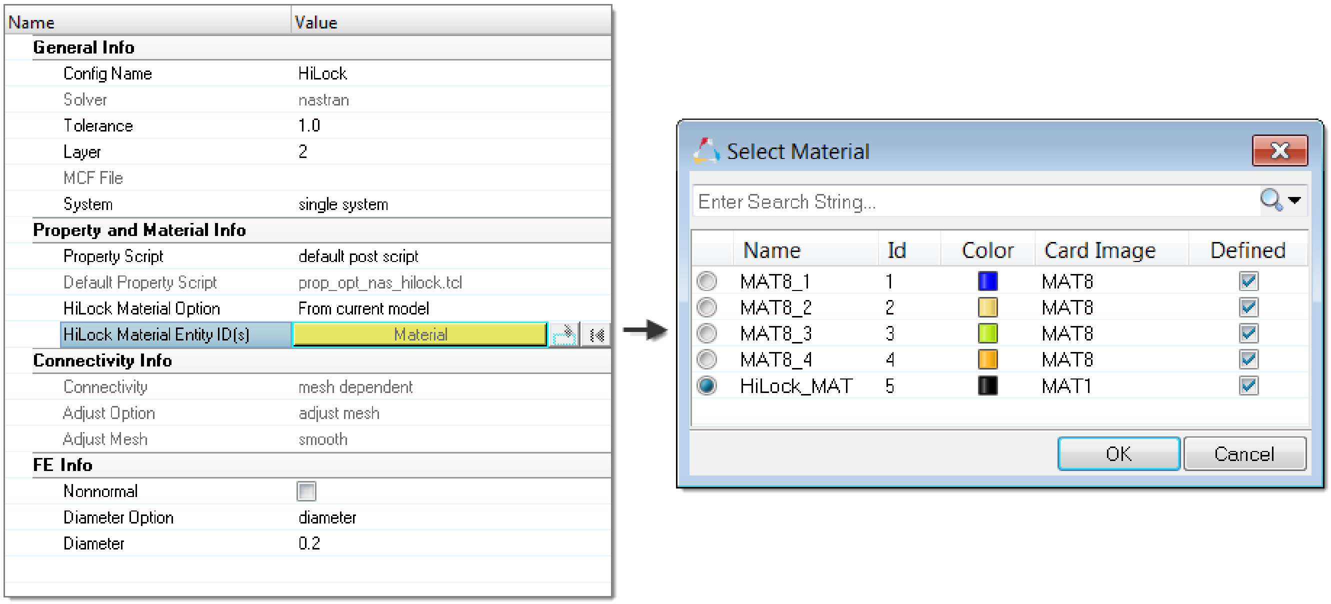


Figure 1088:

From search folder (default)

For HiLock realizations the material search folder is HiLock_Mats. HyperMesh searches for this folder in the following locations and in the following order:

1. Installation: `[hm_scripts_dir]/connectors/Hilock_Mats`
2. User directory: `[USER_HOME]/HiLock_Mats`
3. HyperWorks Configuration Path folder: `[HW_CONFIG_PATH]/Hilock_Mats`
4. Current working folder: `[CURRENTWORKINGDIR]/HiLock_Mats`

In the Connector Entity Editor, the HiLock Material Folder field is populated with the name of the folder that was found last. Only the files in this HiLock_Mats folder are considered and can be selected in the HiLock Material File field.
By default, the first file listed (alphabetical order) in the folder is automatically populated in the HiLock Material File field, and will be used when realizing a connector from the panel. For this reason, it is important that you only keep valid material files in the HiLock_Mats folders.

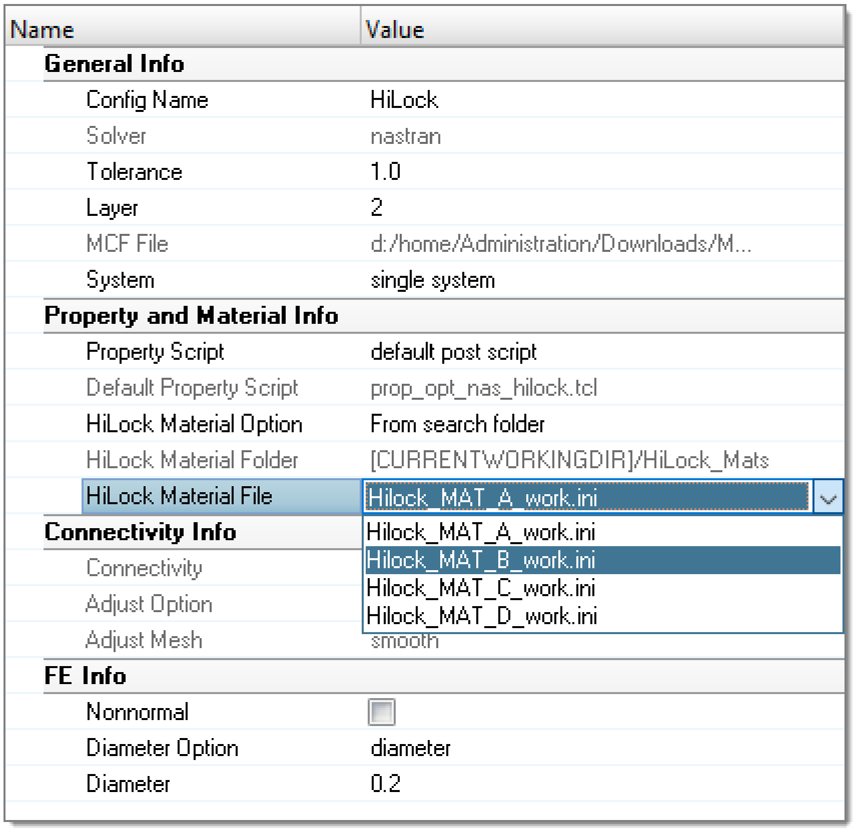
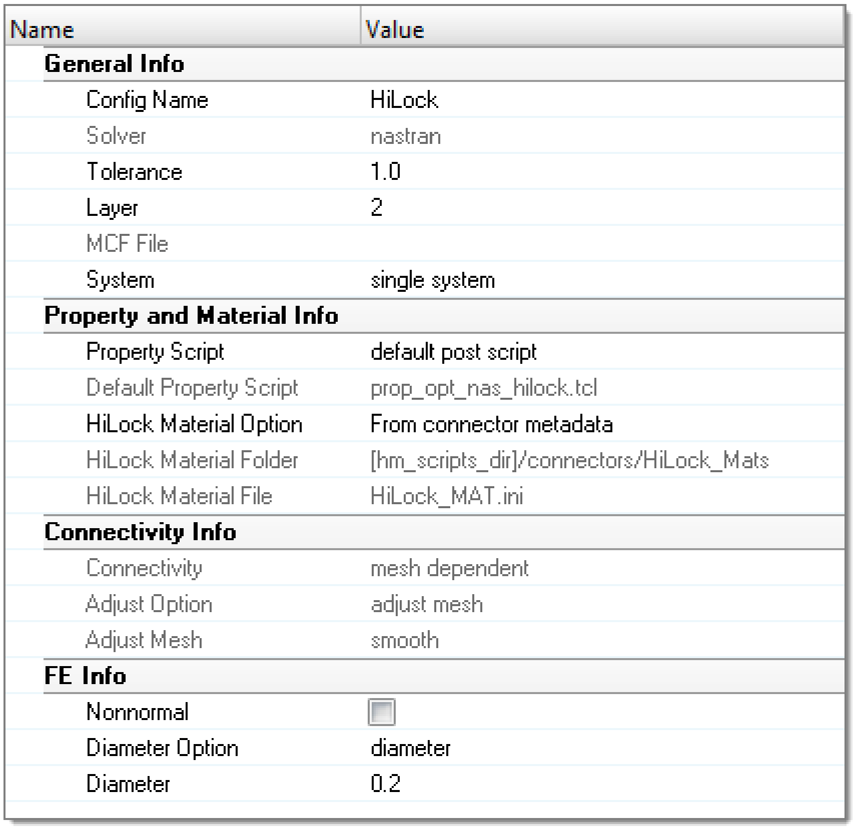


Figure 1089:

From connector metadata

Once a connector is realized with the HiLock Material Option “From Search Folder”, the folder and file name is written as metadata to the connector. Folder data is saved in a relative manner to allow the exact same rerealization in a different work environment as long as the materials are saved in according folders.

If the materials are not available as the metadata states, the realization will fail and the following message with be displayed: Material file/id not found.



Name	Value
General Info	
Config Name	HiLock
Solver	nastran
Tolerance	1.0
Layer	2
MCF File	
System	single system
Property and Material Info	
Property Script	default post script
Default Property Script	prop_opt_nas_hilock.tcl
HiLock Material Option	From connector metadata
HiLock Material Folder	[hm_scripts_dir]/connectors/HiLock_Mats
HiLock Material File	HiLock_MAT.ini
Connectivity Info	
Connectivity	mesh dependent
Adjust Option	adjust mesh
Adjust Mesh	smooth
FE Info	
Nonnormal	☐
Diameter Option	diameter
Diameter	0.2

Figure 1090:

Calculation of Bearing Stiffness in Composite Parts

Combined translational bearing stiffness at composite plate with the fastener contact

These values are defined for every composite plate in the joint.

After summation of bearing stiffness of plies where n = number of plies in the composite plate:

$$S_{xbt} = \sum_{i=1}^{n} S_{ixbt} = \sum_{i=1}^{n} \frac{t_i}{\frac{1}{Q_{11}^{(i)}} + \frac{1}{E_{cf}}}$$

$$S_{ybt} = \sum_{i=1}^{n} S_{iybt} = \sum_{i=1}^{n} \frac{t_i}{\frac{1}{Q_{22}^{(i)}} + \frac{1}{E_{cf}}}$$

Combined translational bearing stiffness of the joint at ply i location in directions x and y

$$S_{ixbt} = \frac{1}{C_{ixbt}} = \frac{t_i}{\frac{1}{\overline{Q}_{11}^{(i)}} + \frac{1}{E_{cf}}}$$

$$S_{iybt} = \frac{1}{C_{iybt}} = \frac{t_i}{\frac{1}{\overline{Q}_{22}^{(i)}} + \frac{1}{E_{cf}}}$$

Transformed reduced stiffness in x and y-direction for ply i

$$\overline{Q_{11}^{(i)}} = Q_{11}^{(i)} m_i^4 + 2\left(Q_{12}^{(i)} + Q_{66}^{(i)}\right) m_i^2 n_i^2 + Q_{22}^{(i)} n_i^4$$

$$\overline{Q_{22}^{(i)}} = Q_{11}^{(i)} n_i^4 + 2\left(Q_{12}^{(i)} + Q_{66}^{(i)}\right) m_i^2 n_i^2 + Q_{22}^{(i)} m_i^4$$

Where,

$$m_i = \cos\theta_i$$
$$n_i = \sin\theta_i$$

(theta = angle of orientation for ply i)

Nonzero components of the reduced stiffness matrix for ply i

(1)

$$Q_{11}^{(i)} = \frac{E_1^{(i)}}{1 - \nu_{12}^{(i)} \nu_{21}^{(i)}}$$

$$Q_{22}^{(i)} = \frac{E_2^{(i)}}{1 - \nu_{12}^{(i)} \nu_{21}^{(i)}}$$

$$Q_{12}^{(i)} = \frac{\nu_{12}^{(i)} E_2^{(i)}}{1 - \nu_{12}^{(i)} \nu_{21}^{(i)}} = \frac{\nu_{21}^{(i)} E_1^{(i)}}{1 - \nu_{12}^{(i)} \nu_{21}^{(i)}}$$

$$Q_{66}^{(i)} = Q_{12}^{(i)}$$

$E_1^{(i)}$, $E_2^{(i)}$ Elastic moduli of ply i in material principal 1 and 2 directions

$G_{12}^{(i)}$ Shear modulus in the lamina i plane (plane 1-2)

$\nu_{12}^{(i)}$, $\nu_{21}^{(i)} = \nu_{12}^{(i)} \frac{E_2^{(i)}}{E_1^{(i)}}$ Poisson's ratio in the lamina i plane

Angle of ply i orientation in a CQUAD4 element

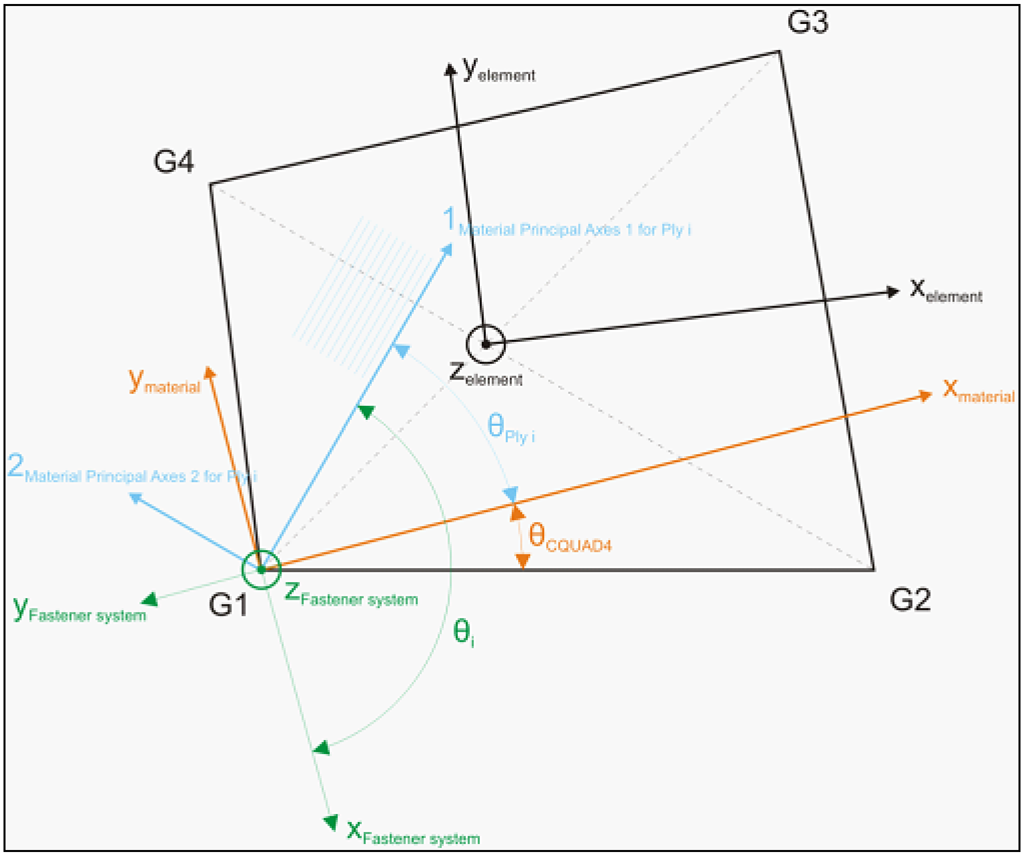


Figure 1091:

Material orientation of a CQUAD4 defined by a coordinate system

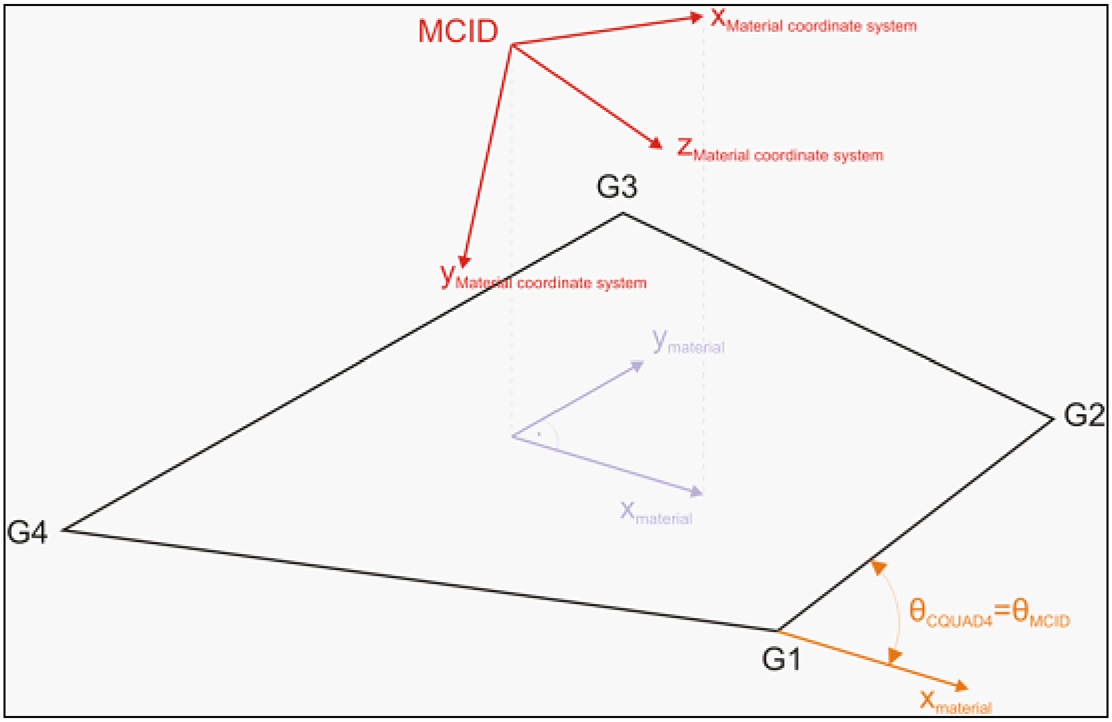
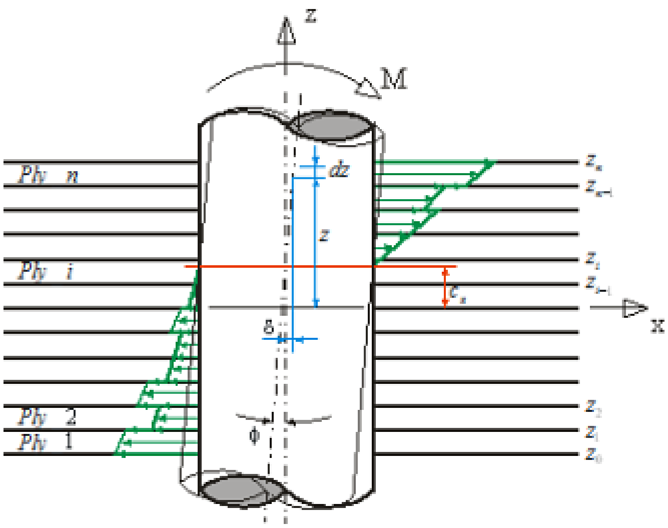


Figure 1092:

Rotational bearing stiffness in plate-fastener contact



$$S_{xbr} = \frac{1}{3}\sum_{i=1}^{n} \frac{(z_i - c_x)^3 - (z_{i-1} - c_x)^3}{\dfrac{1}{\overline{\overline{Q}}{}_{11}^{(i)}} + \dfrac{1}{E_{cf}}}$$

$$S_{ybr} = \sum_{i=1}^{n} \frac{(z_i - c_y)^3 - (z_{i-1} - c_y)^3}{\dfrac{1}{\overline{\overline{Q}}{}_{22}^{(i)}} + \dfrac{1}{E_{cf}}}$$

Figure 1093:

Calculation of Bearing Stiffness in Metallic Parts

Combined translational bearing stiffness at metallic plate with the fastener contact

After summation of bearing stiffness of plies where n = number of plies in the composite plate:

$$S_{xbt} = S_{ybt} = \frac{1}{\frac{1-v2}{E} + \frac{1}{E_{cf}}}$$

Where,

t

Thickness of metallic part

E

Elastic compression modulus of metallic (isotropic) part

v

Poisson's ratio

Rotational bearing stiffness at metallic plate with the fastener contact

$$S_{xbr} = S_{ybr} = \frac{1}{12} \cdot \frac{1^3}{\frac{1-v2}{E} + \frac{1}{E_{cf}}}$$

Where,

t

Thickness of metallic part

E

Elastic compression modulus of metallic (isotropic) part

v

Poisson's ratio

RBE3 Load Transfer

Use the RBE3 Load Transfer realization to create MPC’s using RBE3 elements between the nodes of shell-shell, shell-solid or solid-solid groups by using spot connectors.

Restriction: Available for OptiStruct, Nastran and Abaqus solver interfaces.

Solid-Solid

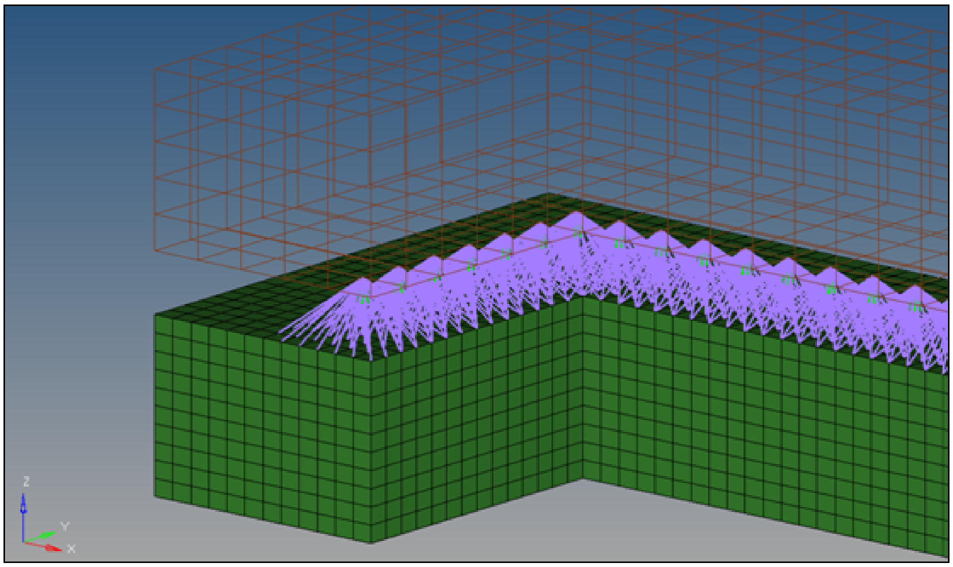
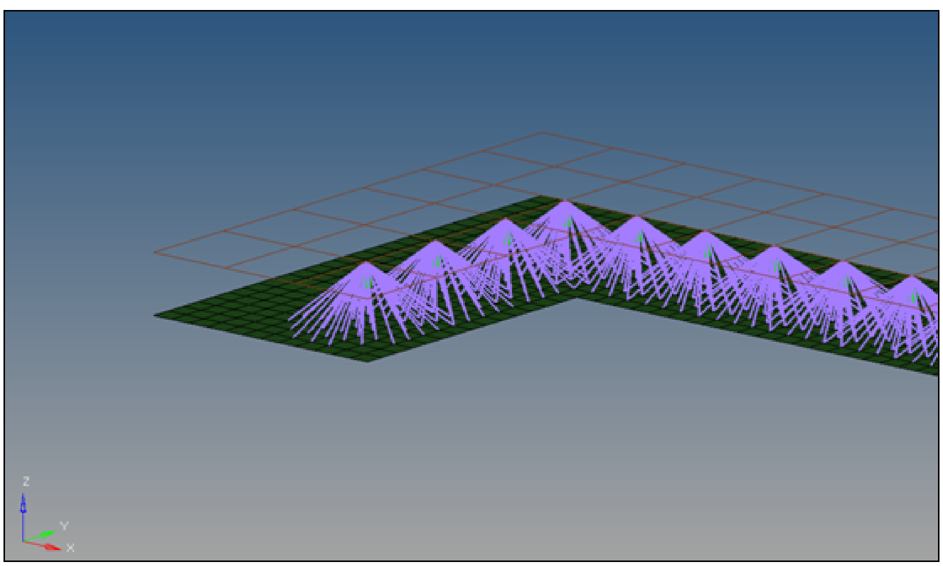


Figure 1094:

Shell-Shell (Face to Face)



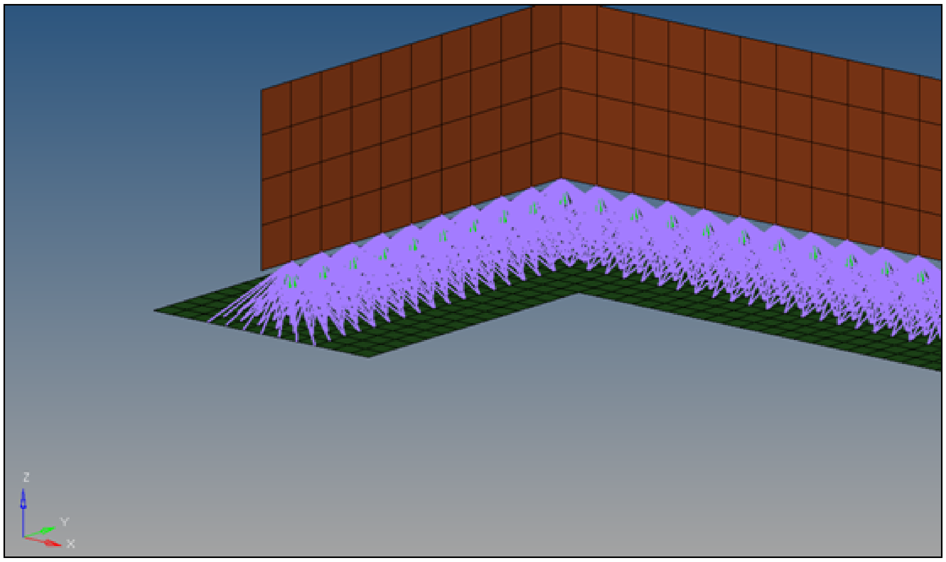


Figure 1096:

Shell-Shell (Edge to Edge

For successful realization of these connectors, the non-normal projection option needs to be active. Otherwise the projection onto an edge does not work.

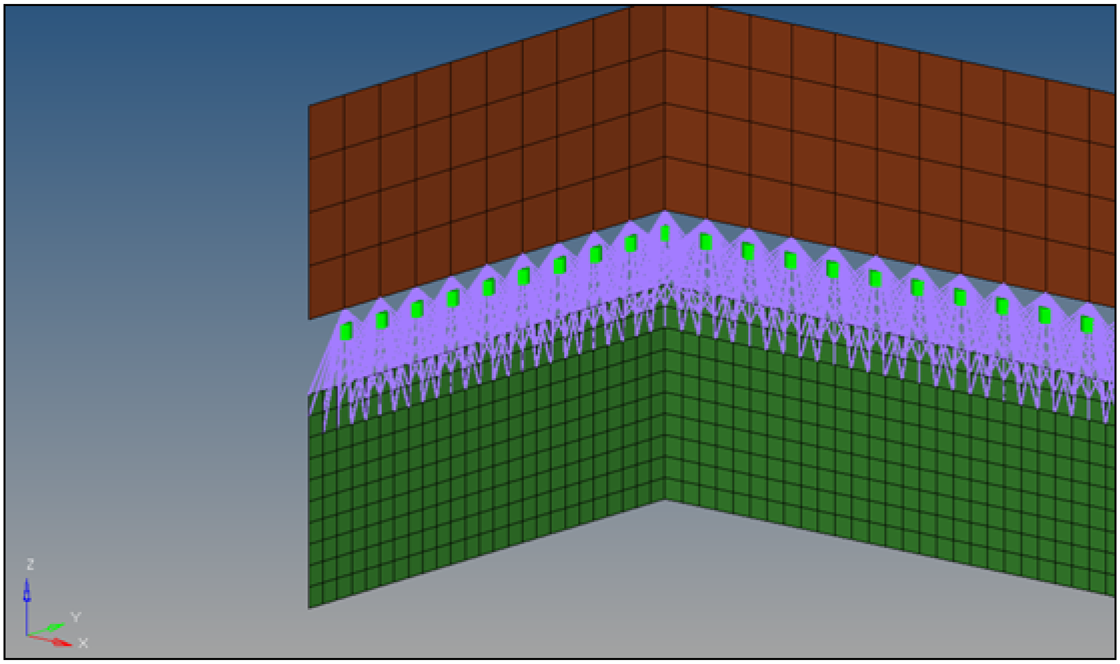


Figure 1097:

Shell-Beam

This situation is a very specific and requires some preparation to be successful, since the projection onto 1D elements is not supported. In this situation, you need to enable the **non-normal projection** checkbox for the projection onto the edge. In addition, the node of the 1D element needs to be defined directly as a link. Normally this is done during connector creation by activating add node location as link.

> **Note:** This option is only available for nodes as connector location, and only if the center definition is set to use connector position for center.

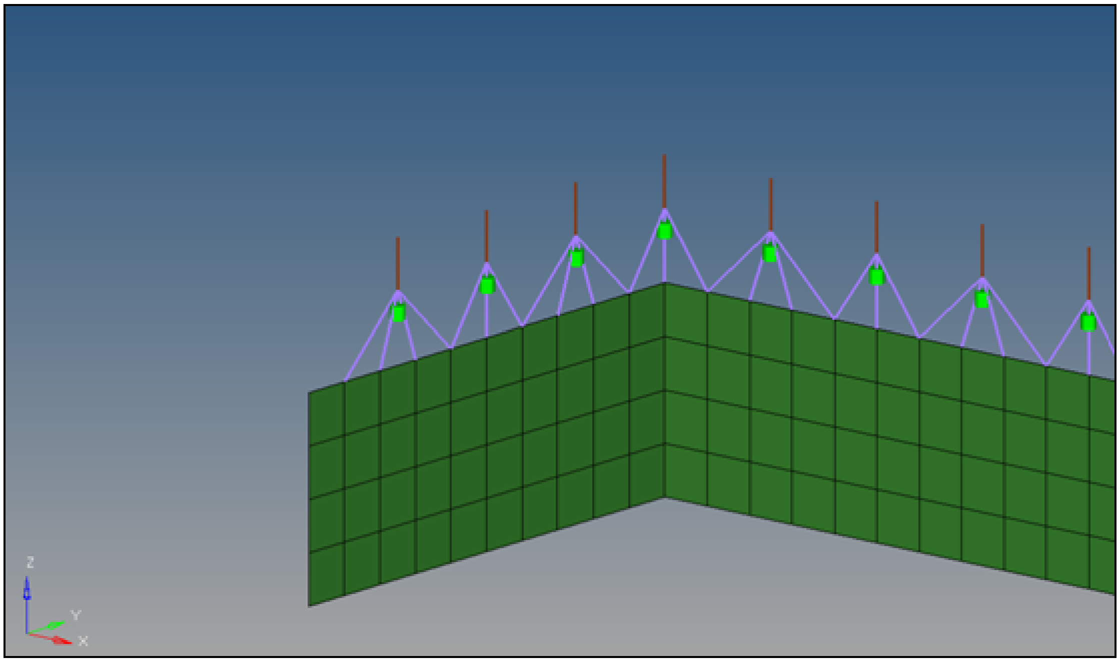


Figure 1098: Shell-Beam

Center definition options include:

use shortest projection for center

The closest node becomes the center of the RBE3 element.

During the realization, based on the connector position and the tolerance, the closest links are determined up to the number of required layers (num layer). All other link candidates are not taken into account for the next step. The closest node is also determined and becomes the center of the RBE3 element. Based on this center position, all nodes within the given tolerance (distance center to node) and belonging to the remained links are attached to the RBE3 element.

> **Note:** If the connector has been created with the option add location node as link, the use shortest projection for center option is ignored and the linked node becomes the center of the RBE3 element.

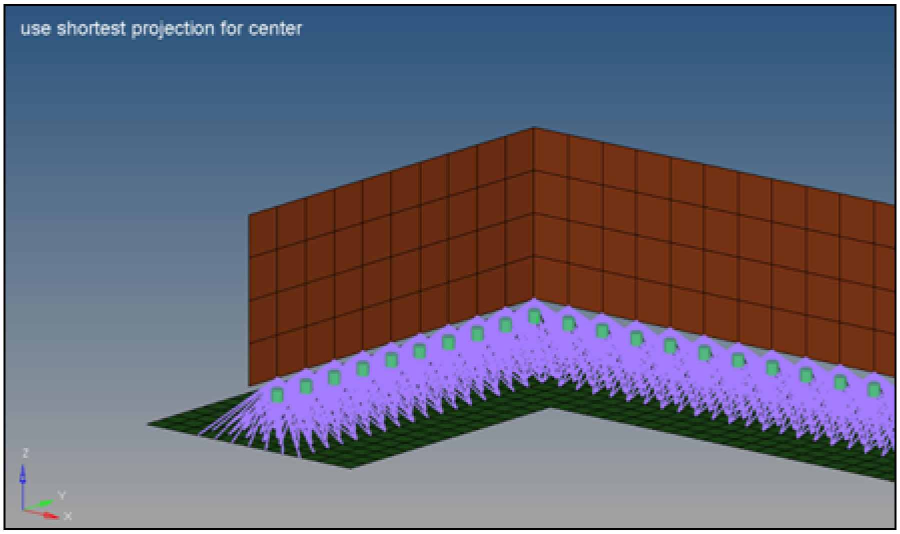


Figure 1099: Use Shortest Projection For Center

use connector position for center

The exact position of the connector becomes the center of the RBE3 element.

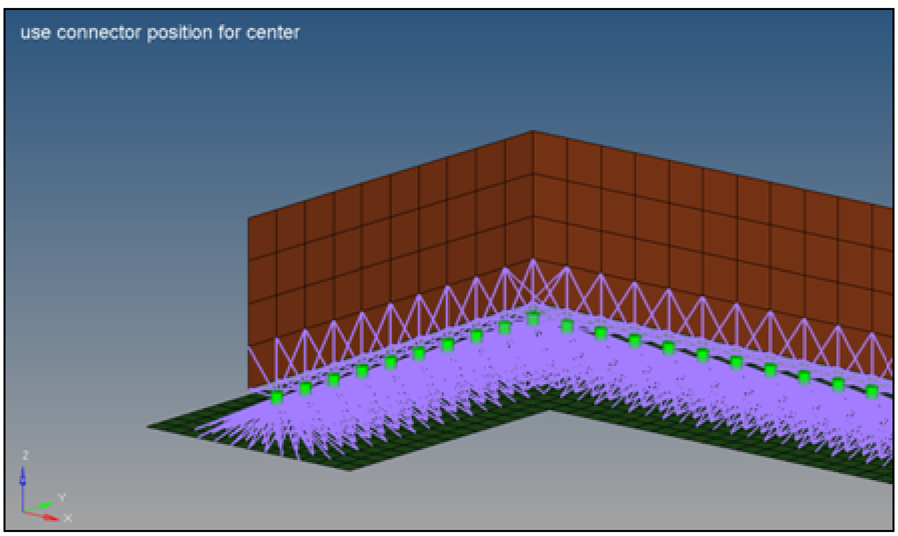


Figure 1100: Use Connector Position For Center

use coarse mesh for center

During the realization, based on the connector position and tolerance, the closest links are determined up to the number of required layers (num layer). All other link candidates are not taken into account for the next step.

From the remaining links, the one with the coarsest mesh is identified and a node on this mesh (close to the perpendicular connector projection) becomes the center of the RBE3 element.

Based on this center position, all nodes within the given tolerance (distance center to node) and belonging to the remaining links are attached to the RBE3 element.

Note: If the connector has been created with the add location node as link option, the use shortest projection for center option is ignored and the linked node becomes the center of the RBE3 element.

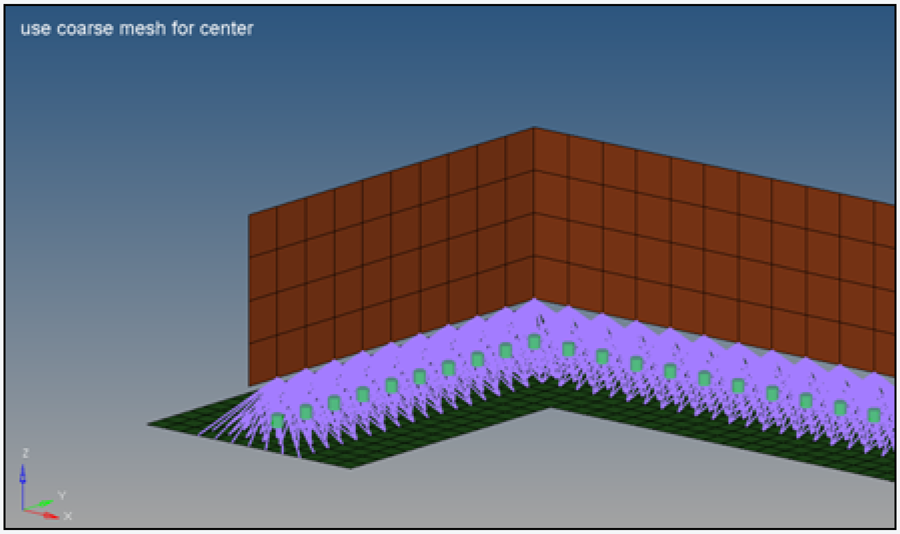


Figure 1101: Use Coarse Mesh For Center

Seam Hexa Adhesives

The Seam Hexa Adhesives realization creates a continuous or discontinuous hexa weld with a predefined pattern.

All defined information is stored on the connector, and can be exported into the connector `.xml` file.

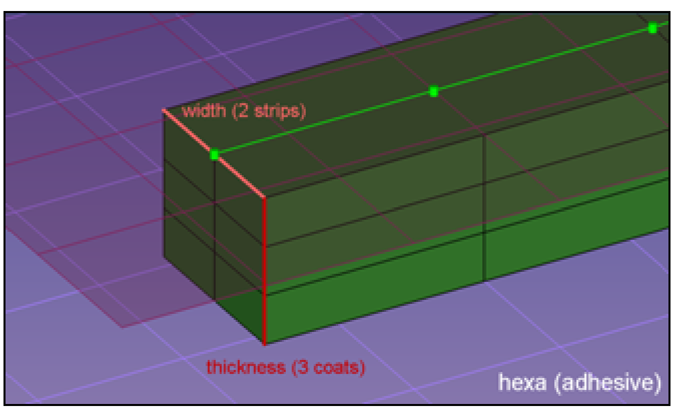


Figure 1102: Seam Hexa Adhesives

Seam Hexas are created from the Seam panel.

The HEXA elements will be centered about the seam connector if the seam connector is not close to a free edge. If the distance between the seam connector and free edge of a component is less than half the width of the HEXA, then the realization of HEXA elements will start from the seam connector and will be extruded in the direction away from the edge.

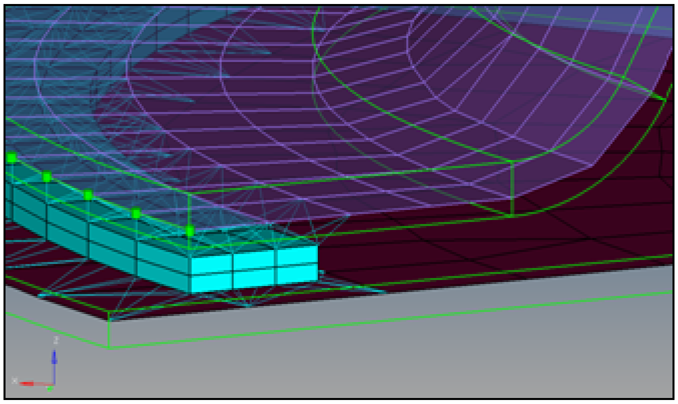


Figure 1103:

For OptiStruct and Nastran solvers, the HEXA elements are tied to a shell using RBE3's at locations where the HEXA nodes and shell nodes are non-coincident. If the HEXA nodes and shell nodes are coincident then RBE2's will be used to tie them.

For the LS-DYNA solver only, the shell gap thickness option is supported. If the HEXA nodes are coincident with shell nodes, then those shell nodes will be used to create HEXA elements. The HEXA elements at some or all nodes will be tied directly to the shells.

This realization type is intended to work on meshes, both shells and solids.

The hexa dimension depends on the following settings:

- The length of a hexa is predefined by the distribution of the test points along the seam connector. This is defined by spacing or density during the connector creation.
- The width of the single hexa depends on the number of strips and the defined total width of the seam, which is measured perpendicular to the seam direction.
- The thickness of a single hexa depends on the number of defined coats and the selected thickness option.

The available thickness options interact with the consider shell thickness option and offset for hexa positioning option. In the figures below, the green seams on the left take into account the thickness as well as the shell offset. This information is used for dimensioning and positioning the hexas. For the pink seams on the right side, the hexas are always positioned around the exact middle between the current shell positions. The shell thicknesses are taken into account only for the hexa height, but not for the positioning. The orange lines and arrows in the figures below illustrate the dependencies for the positioning.

shell gap

The seam completely and exactly fills the gap between the two shells. Shell thicknesses and offsets are not considered.

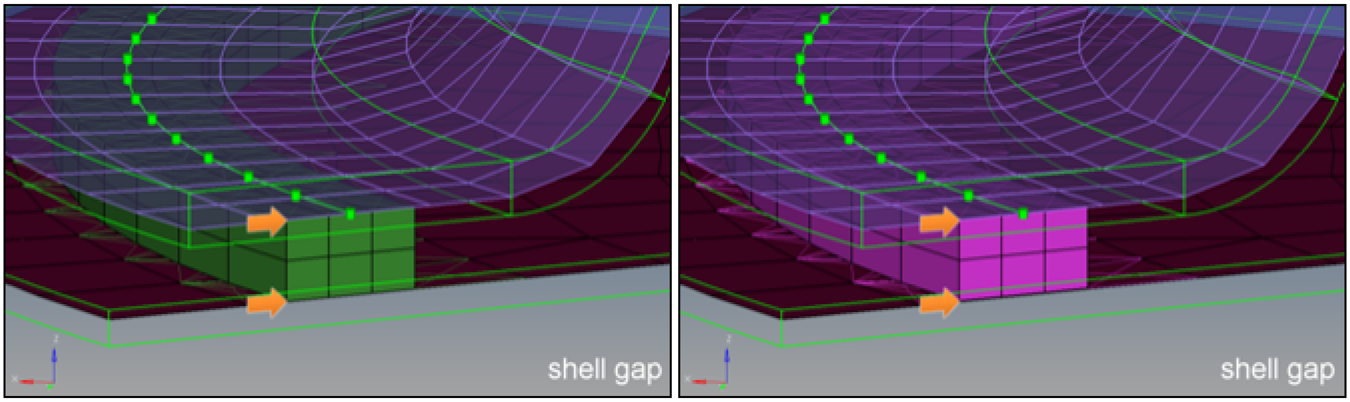


Figure 1104: Shell Gap

maintain gap

The seam is positioned in the exact middle between the shells. The seam thickness is adjusted, that on both sides the gap between shell and seam fits the defined gap size. Shell thicknesses and offsets are not considered.

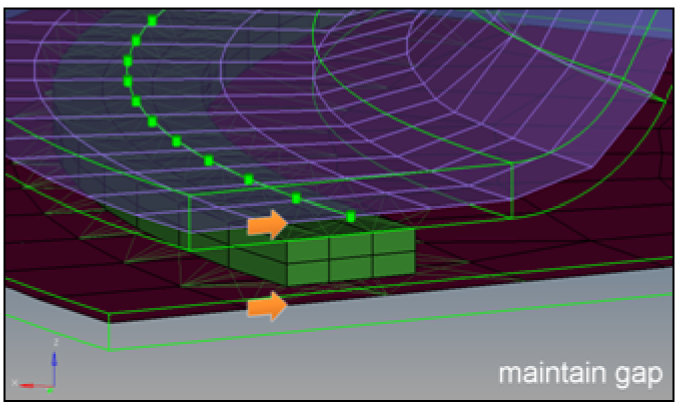


Figure 1105: Maintain Gap

(t1+t2)/2

The seam thickness is calculated by averaging both shell thicknesses. On the left side the offsets and thicknesses are taken into account, so that the seam is positioned around the middle of the air gap. On the right side the seam is just positioned around the middle of the shell positions.

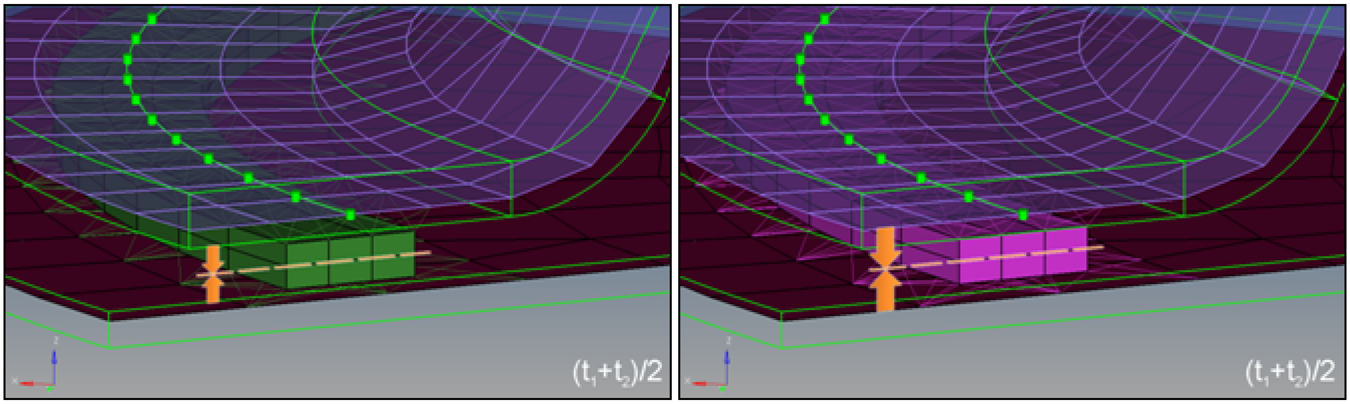


Figure 1106: (t1+t2)/2

midthickness

On the left side the exact air gap is determined and filled with the seam. On the right side the seam thickness is calculated by subtracting half the thickness of both shells from the total distance of the shells.

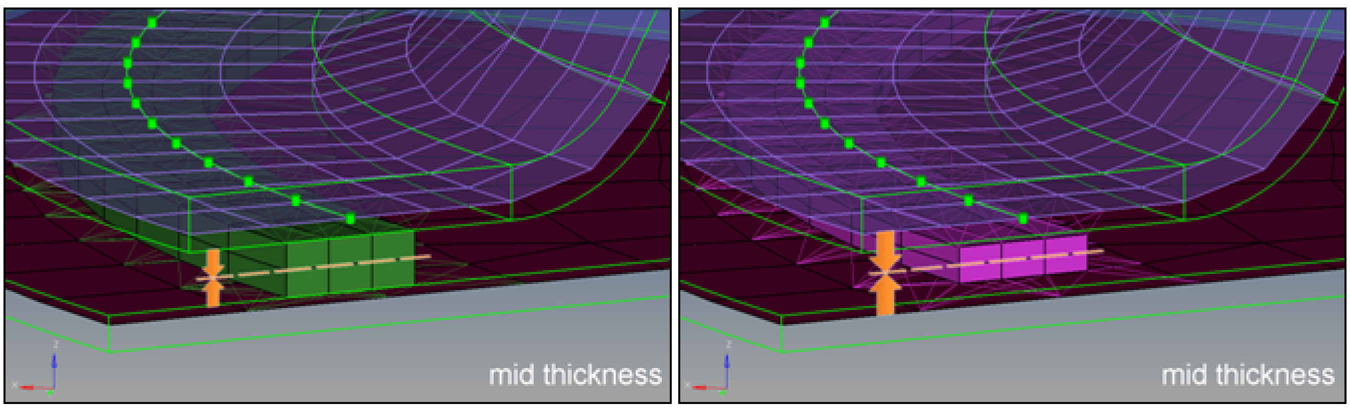


Figure 1107: Midthickness

const. thickness

The thickness of the seam is predefined for both; on the left side the seam is positioned around the middle of the air gap, on the right side around the middle of the two shells.

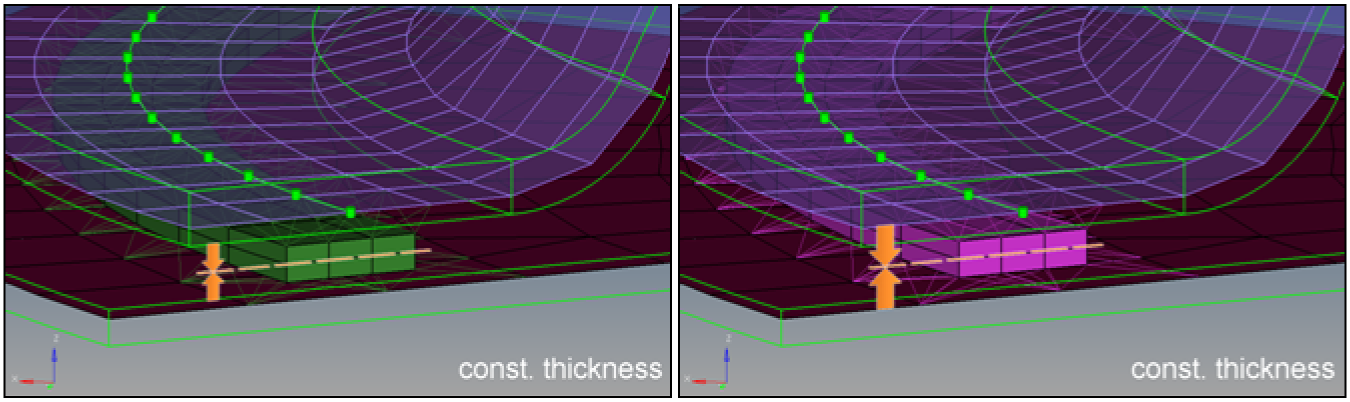


Figure 1108: Const. Thickness

Radioss ACM (Shell Gap Contact and Coating)

The Radioss acm (shell gap contact and coating) realization creates hexa clusters between shell components. Contacts get defined between the shell components and the appropriate hexa nodes. A heat affected zone for the shells from ultra high strength steel material is also created.

The Radioss acm (shell gap contact and coating) realization can be used for any amount of parallel combinations of shell components.

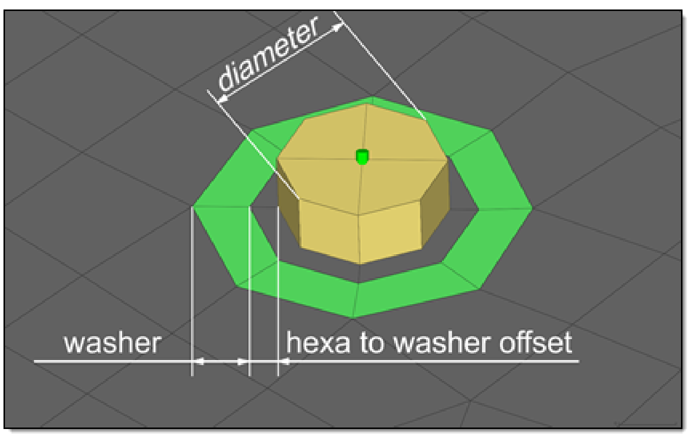


Figure 1109: Heat Affected Zone Dimensions

Organization and Definition of ACM (Shell Gap Contact and Coating)

1. For each connected link the contact /inter/TYPE2/ gets created and is named TYPE2_CONTACT_PID_<link ID>. The following sets are created and referenced.

- MASTERPART_SET_PID_<link ID>: In this set, which is referenced as the master by the above mentioned contact, the link entities like the component get organized.
- SLAVENODE_SET_PID_<link ID>: In this set, which is referenced as the slave by the above mentioned contact, the hexa nodes projecting onto the master entities get organized.

2. For each link combination the hexa clusters are organized into separate components and named RAD_SOLID_SPOTWELD_PID_<link1 ID>_<link2 ID>. All components are assigned the following material and property:
    - RAD_SOLID_SPOTWELD_DEFAULT_MAT. This material is defined as /MAT/LAW59/.
    - RAD_SOLID_SPOTWELD_DEFAULT_PROP. This property is defined as /PROP/CONNECT/.

    The default values are read from `uhss_washersolid_matprop.rad` in the installation.
3. The heat affected zone elements (washer) are organized into one separate component for each link from the ultra high strength steel material and named RAD_WASHER_PID_<link ID>. All components are assigned the following material and property:
    - RAD_WASHER_MAT. This material is defined as `/MAT/PLAS_JOHNS/`.
    - RAD_WASHER_PROP. This property is defined as `/PROP/SHELL/`.

    The material and property values are read from `uhss_washersolid_matprop.rad` in the installation.

Defining Materials for Heat Affected Zone Treatment

You must specify which materials are considered as ultra high strength materials. When defining an acm (shell gap contact and coating) connector, the UHSS Material Option can be selected for individual connectors using the Connector Entity Editor.

From current model
: Select an existing material from the current model.

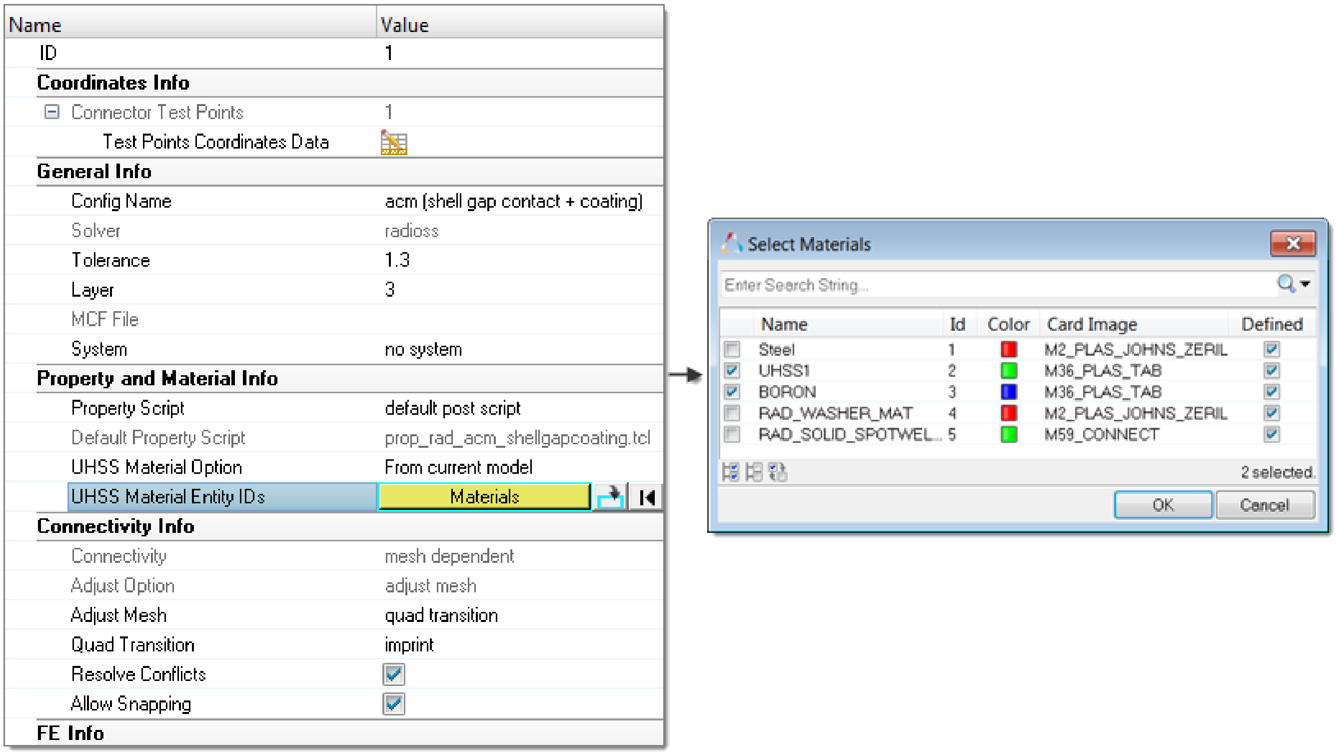
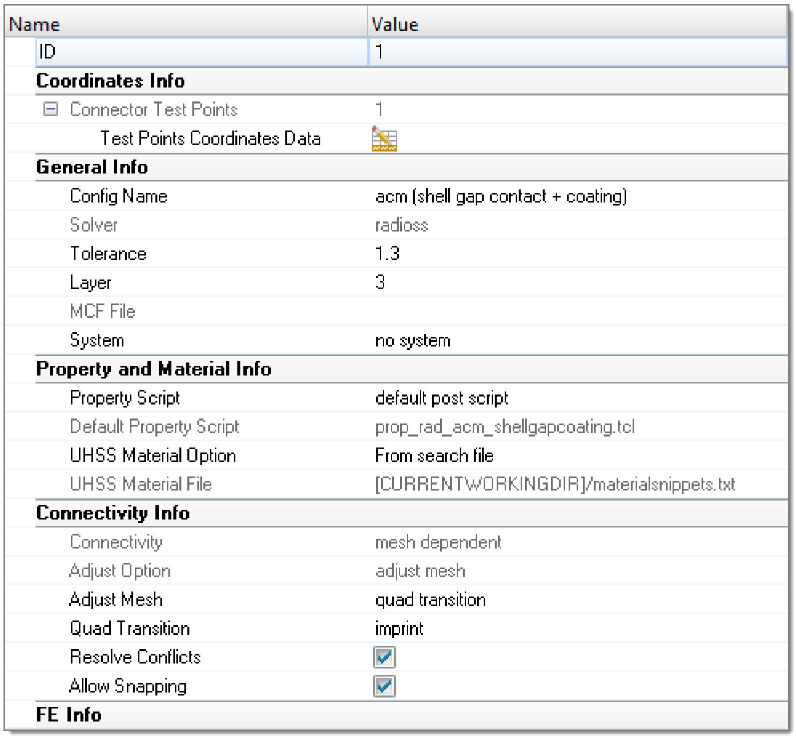


Figure 1110: From Current Model

From search file (default)

For acm (shell gap contact and coating) realizations the material search file name is `materialsnippets.txt`. HyperMesh searches for this file in the following locations and in the following order:

1. Installation: `[hm_scripts_dir]/connectors/materialsnippets.txt`
2. User directory: `[USER_HOME]/materialsnippets.txt`
3. HyperWorks Configuration Path folder: `[HW_CONFIG_PATH]/materialsnippets.txt`
4. Current working folder: `[CURRENTWORKINGDIR]/materialsnippets.txtv`



Name	Value
ID	1
Coordinates Info	
Connector Test Points	1
Test Points Coordinates Data	
General Info	
Config Name	acm (shell gap contact + coating)
Solver	radioss
Tolerance	1.3
Layer	3
MCF File	
System	no system
Property and Material Info	
Property Script	default post script
Default Property Script	prop_rad_acm_shellgapcoating.tcl
UHSS Material Option	From search file
UHSS Material File	[CURRENTWORKINGDIR]/materialsnippets.txt
Connectivity Info	
Connectivity	mesh dependent
Adjust Option	adjust mesh
Adjust Mesh	quad transition
Quad Transition	imprint
Resolve Conflicts	☑
Allow Snapping	☑
FE Info	

Figure 1111: From Search File

In the Connector Entity Editor, the UHSS Material File field is populated with the name of the file that was found last.
The text file contains snippets from the materialnames, which need to recieve heat affected zones during the realization.

From connector metadata

Once a connector is realized with the UHSS Material Option "From search file", the folder name is written as metadata to the connector in a relative manner to allow the exact same rerealization in a different work environment as long as the same `materialsnippets.txt` files are saved in according folders.

Name	Value
ID	1
Coordinates Info	
⊟ Connector Test Points	1
Test Points Coordinates Data	
General Info	
Config Name	acm (shell gap contact + coating)
Solver	radioss
Tolerance	1.3
Layer	3
MCF File	
System	no system
Property and Material Info	
Property Script	default post script
Default Property Script	prop_rad_acm_shellgapcoating.tcl
UHSS Material Option	From connector metadata
UHSS Material File	[CURRENTWORKINGDIR]/materialsnippets.txt
Connectivity Info	
Connectivity	mesh dependent
Adjust Option	adjust mesh
Adjust Mesh	quad transition
Quad Transition	imprint
Resolve Conflicts	☑
Allow Snapping	☑
FE Info	

Figure 1112: From Connector Metadata

Seam-Quad LTB

The seam-quad LTB realization serves and realizes t-welds, lap-welds, and butt-welds simultaneously.

This weld type is identified automatically based on the orientation of the links to each other.

The dimensions and properties assigned to all heat affected zones (HAZ) can be defined separately. Normal directions of quad weld elements and HAZ elements can be controlled. An edge treatment can be defined for t-welds and butt-welds to move the edge a precise distance from the opposite link.

> **Restriction:** Available in the OptiStruct and Nastran solver interfaces, and can only be selected and defined in the Connector Entity Editor.

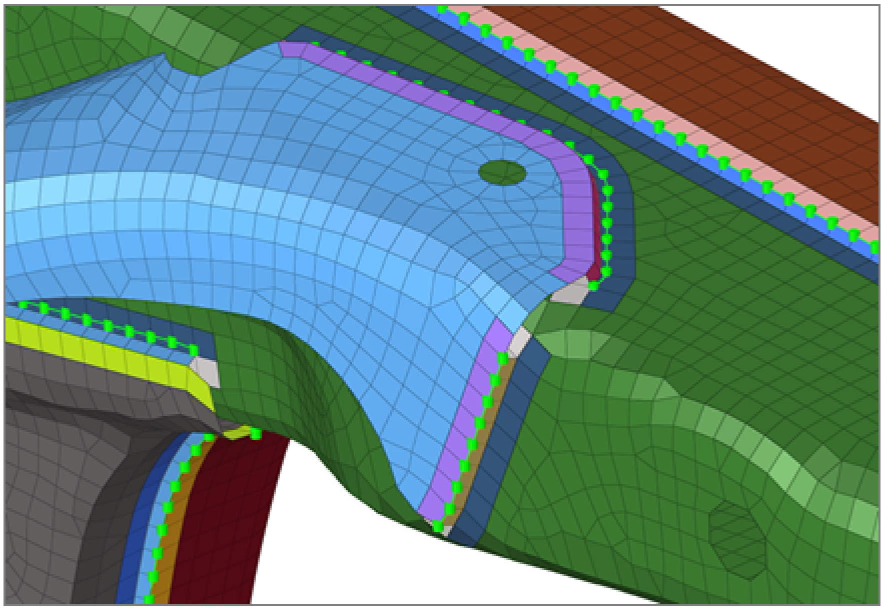


Figure 1113: Seam-Quad LTB

General Info

Weld Type

Defines weather to setup a configuration exclusively for a T, L, or B connection, or automatically setup a configuration for each connection based on the angle.

In any case, the connection type is dependent on the:

- B/L classification angle
- L/T classification angle

Both types of angles are defined in the Behavior section.

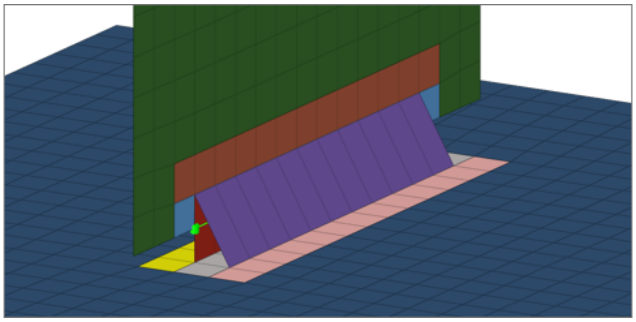


Figure 1114: T Connection

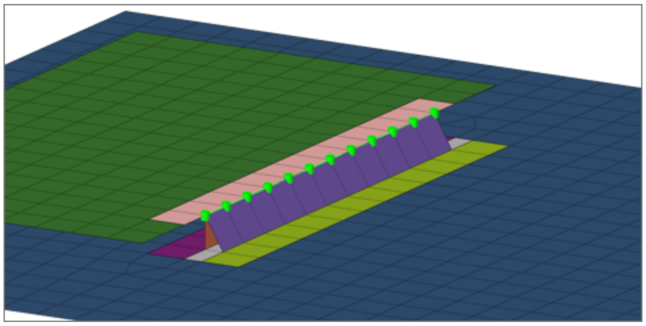


Figure 1115: L Connection

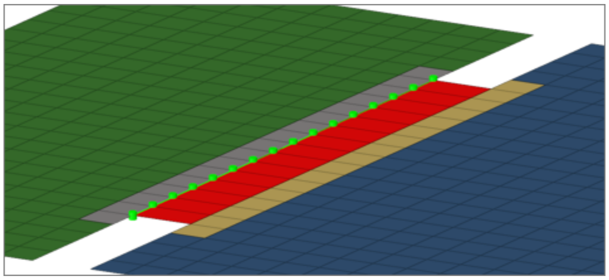


Figure 1116: B Connection

Tolerance

Defines the distance from the connector location.

Only entities within this tolerance can be taken into account for the final realization. The tolerance is used to verify whether adequate link candidates are available to be connected with respect to the number of layers.

Weld Shape

T Weld Shape

Defines how the T weld is created.

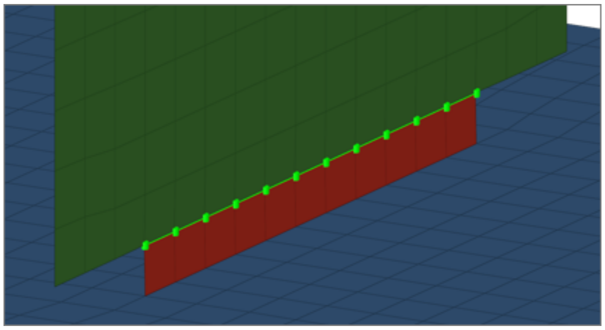


Figure 1117: Vertical T weld

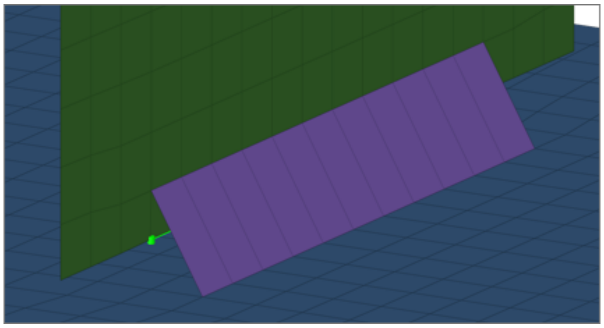


Figure 1118: Angled T weld

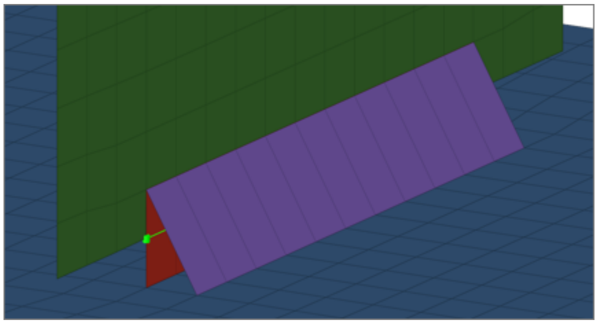


Figure 1119: Vertical and angled T weld

L Weld Shape

Defines how the L weld is created.

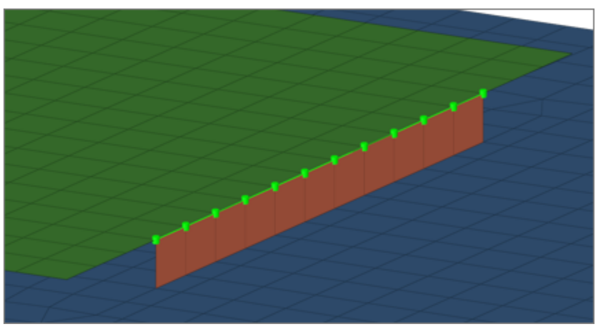


Figure 1120: Vertical L weld

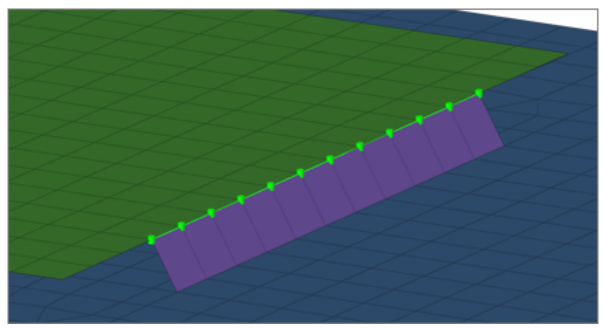


Figure 1121: Angled L weld

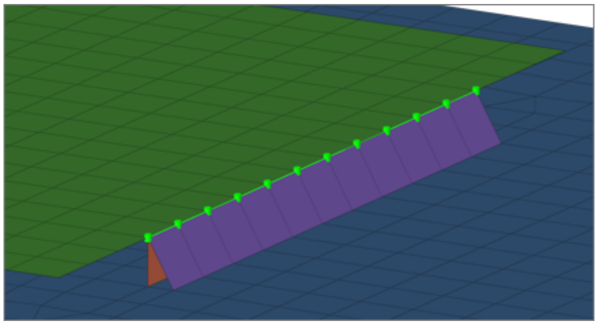


Figure 1122: Vertical and angled L weld

B Weld Shape

B welds are always created in a straight manner.

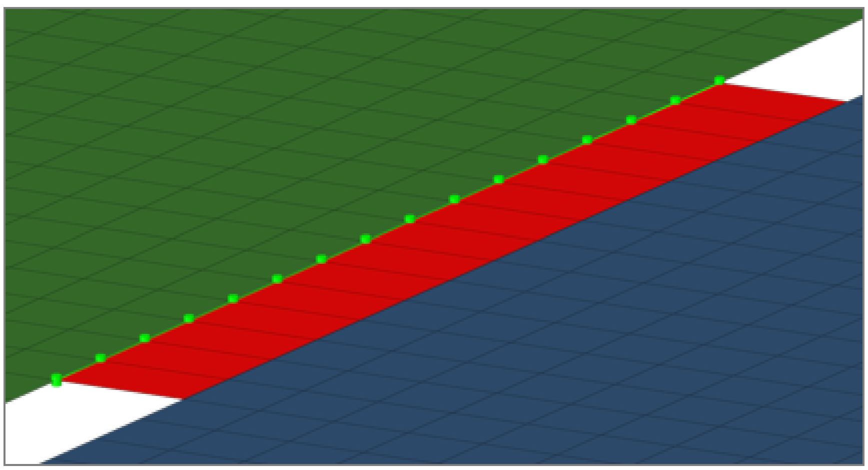


Figure 1123: Straight B weld

With Caps

When enabled, seams are closed with a tria element.

The Caps settings determines how the caps are created.

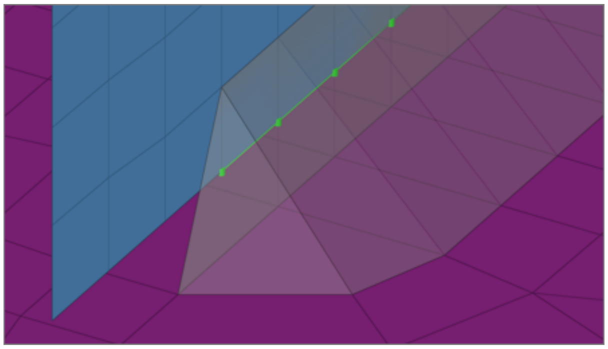


Figure 1124: Tria cap element for T weld

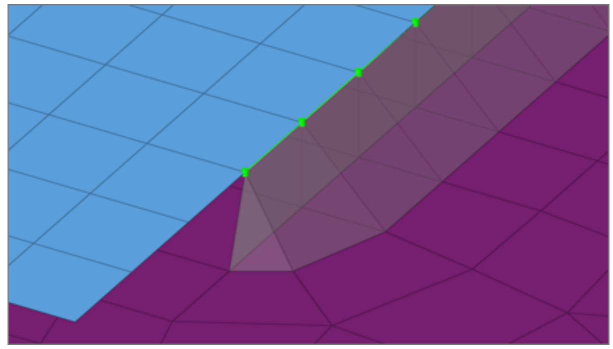


Figure 1125: Tria cap element for L weld

Realization Details

The Realization Details settings position the yellow marked nodes in the Figure 1126, Figure 1127, and Figure 1128.

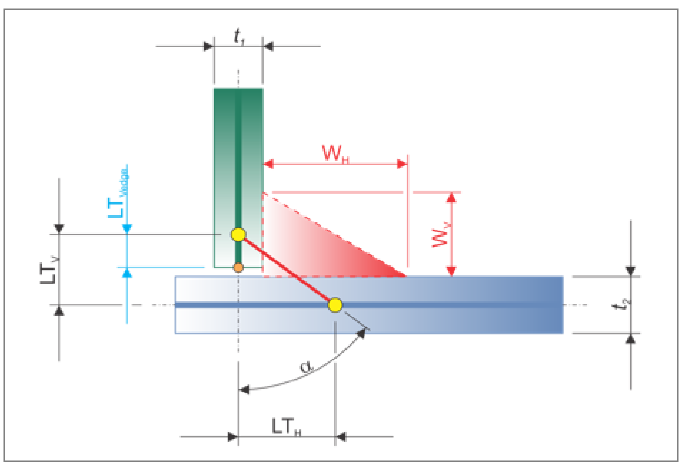


Figure 1126: T Dimensions

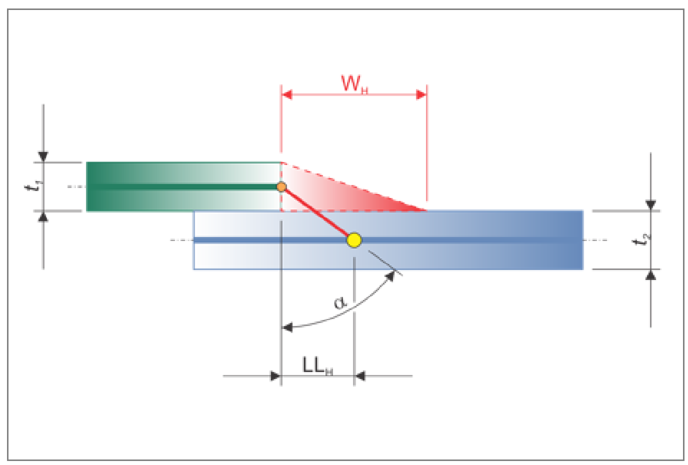


Figure 1127: L Dimensions

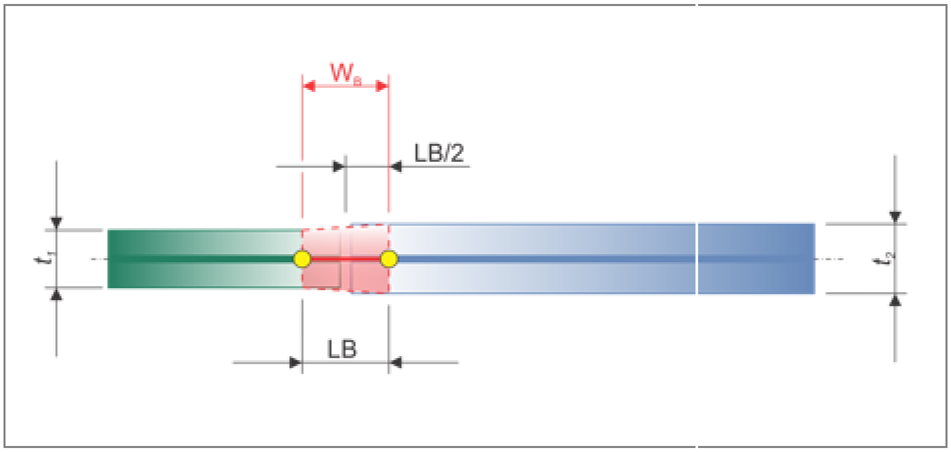


Figure 1128: B Dimensions

The dimension of the welds are dependent on the Weld Shape settings.

Dimensioning Scheme

Defines the dimensioning scheme for the dimensions of the T weld, L weld, and B weld connections.

input

Manually define discrete values for the weld dimensions, shown in black in Figure 1126, Figure 1127, and Figure 1128, with the exception of thickness. The horizontal dimensions can be defined using a length or an angle.

thickness dependent

Choose a formula to define the weld dimensions, shown in black in Figure 1126, Figure 1127, and Figure 1128, with the exception of thickness. The provided formulas are all dependent on the thicknesses t1 and t2. A formula can be chosen individually for each verticalV and horizontalH distance, or the same formula can be used for T, L and B.

weldsize dependent

Manually define discrete values for the weld dimensions, shown in red in the Figure 1126, Figure 1127, and Figure 1128. The verticalV and horizontalH distances are defined with formulas reflecting the weld sizes and the t1 and t2 thicknesses.

DIM T (Dimensioning T)

	Input	**Thickness dependent**	**Weldsize dependent**
Horizontal Lengths LTH	by angle by length	(t1+t2)/2 3*(t1+t2)/2 t1+t2	t1/2+wh/2 (fix)

	Input	Thickness dependent	Weldsize dependent
Vertical Length LTV	by length by edge	(t1+t2)/2 3*(t1+t2)/2 t1+t2 by edge	t2/2+wv/2 by edge

DIM L (Dimensioning L)

	Input	Thickness dependent	Weldsize dependent
Horizontal Lengths LTH	by length by angle	(t1+t2)/2 3*(t1+t2)/2 t1+t2	wh/2 (fix)

DIM B (Dimensioning B)

	Input	Thickness dependent	Weldsize dependent
Lengths LB	by length by angle	(t1+t2)/2 3*(t1+t2)/2 t1+t2 by edge	wb by edges

Edge Treatment (T/B)

When discrete lengths are requested for T and B connections, it is sometimes necessary to move the edges.

Edge treatment is not needed when the different length dimension settings are set to by edge.

When enabled, edges are allowed to move. See Edge Treatment Options for more information.

Max Length Value

Defines the maximum length value.

This setting is useful when lengths are calculated based on thicknesses. If a length is greater than the Max Length Value, then the Max Length Value will be used instead.

Caps

Determines how caps are created.

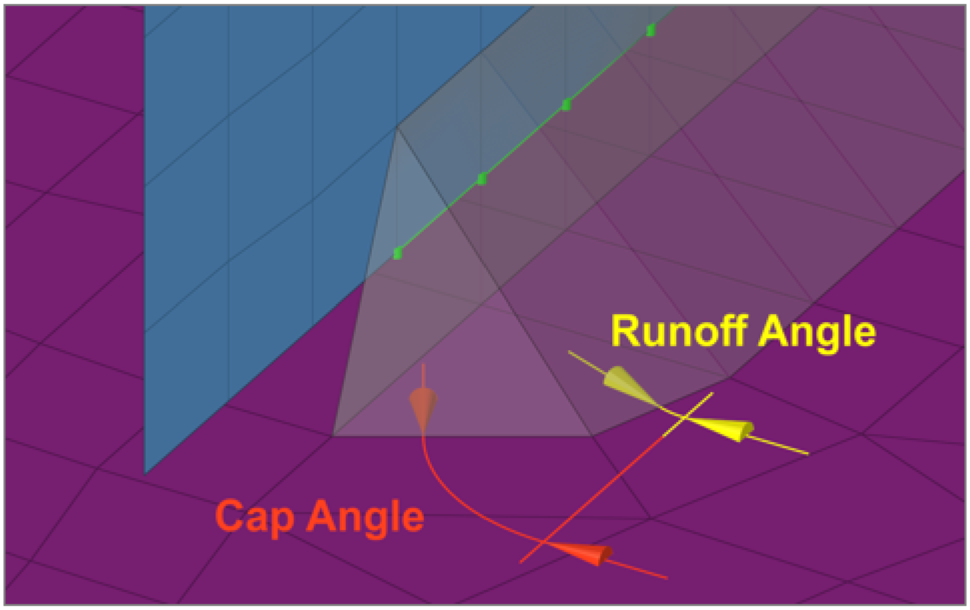


Figure 1129: Cap Angle and Runoff Angle

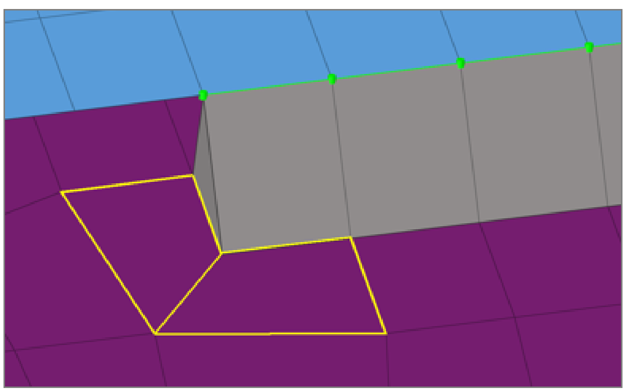


Figure 1130: Sharp Corner Enabled

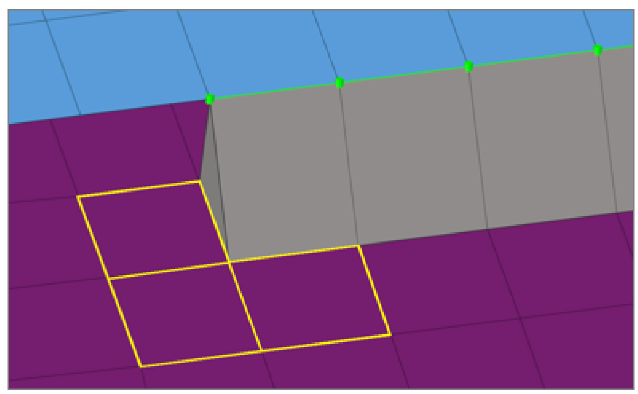


Figure 1131: Sharp Corner Disabled

Element Details

The Element Details settings control the normal directions of the weld, as well as the HAZ elements.

Vertical Element Normal, Angled Element Normal, and HAZ Element Normal can be set to either:

Towards welder

Shows the normal directions

Away from welder

Shows the exact opposite.

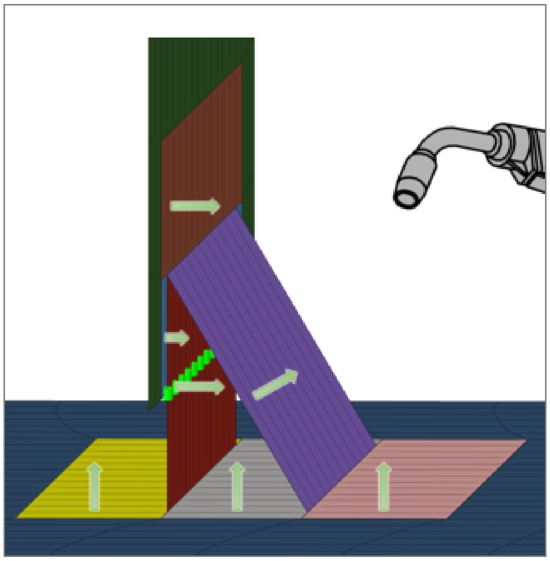


Figure 1132: Towards welder

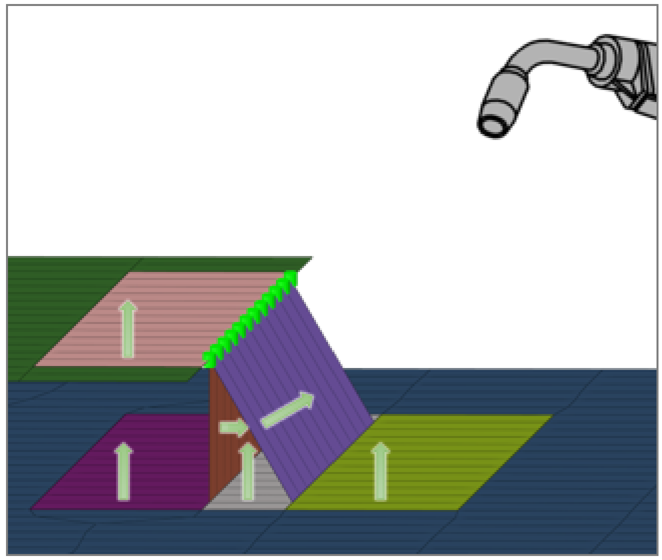


Figure 1133: Away from welder

Connectivity Info

imprint (default)

Creates quad weld elements, and stitches them to both links by adjusting their mesh. All required HAZ are performed.

skip imprint

Creates quad weld elements, but does not change the meshes of the links. Instead, additional elements are created to represent the requested HAZ. These elements are organized in the ^conn_imprint component, and can later be used for a manual imprint after they have been manipulated to the users needs. This option can be helpful when working with more complex areas, where the standard imprint functionality fails, for example, conflicting connectors.

imprint/no HAZ

Creates quad weld elements, and stitches them to both links by adjusting their mesh. Mesh modifications are as minimal as possible, and no HAZ are performed.

none

Creates quad weld elements only. Quad weld elements will not be attached to the links. The connection will need further attention.

HAZ Info

The HAZ Info settings define the lengths of the different heat affected zones (HAZ), which are dependent on the HAZ lengths for T, L and B (defined in Realization Details). The HAZ length settings vary depending on the defined weld shapes (vertical, angled, vertical and angled, caps).

HAZ Scheme

Choose a dimensioning scheme for the HAZ lengths of T, L, and B.

input

Enables you to decide if the HAZ lengths should be defined individually, or if all HAZ lengths are determined using the same approach (same as all).

weldsize dependent

Only available if weldsize dependent has been chosen for the Dimensioning Scheme as well.

HAZ Lengths

same as all

Assigns the same length to all HAZ lengths.

individual

Assign HAZ lengths individually.

HAZ Lengths (various)

The following options are available in the various HAZ length settings.

input

Requires a discrete value be specified for the length.

average meshsize

Length is dependent on the average mesh size in the local area where the imprint is performed.

by thickness

Sets the length to the same value as the thickness of the link getting the HAZ.

LTH

Horizontal length for T connections, which is the length between the foot points of the vertical and angled part of a seam.

LLH

Horizontal length for L connections, which is the length between the foot points of the vertical and angled part of a seam.

LB

Butt weld length.

skip HAZ

Skips individual HAZ that are not required.

same as positive side

Assigns the same length as the positive side to the negative side.

wh or wh/2

Length is dependent on the horizontal weld size. Only available when HAZ Scheme is set to weldsize dependent.

wv or wv/2

Length is dependent on the vertical weld size. Only available when HAZ Scheme is set to weldsize dependent.

wb or wb/2

Length is dependent on the butt weld size. Only available when HAZ Scheme is set to weldsize dependent.

LTVedge

Choose between skip HAZ and LTVedge. Only available for the HAZ_{Tvedge} length.

HAZ Length Factor (Avg. Meshsize/Thickness)

Factor that increases or decreases the HAZ lengths, which have been defined using the average meshsize or by thickness length options.

Max HAZ Length

Maximum length for all HAZ lengths. If the HAZ length is greater than this value, then the Max HAZ Length is used.

Dimensioning and Heat Affected Zones (HAZ):

Dimensioning T

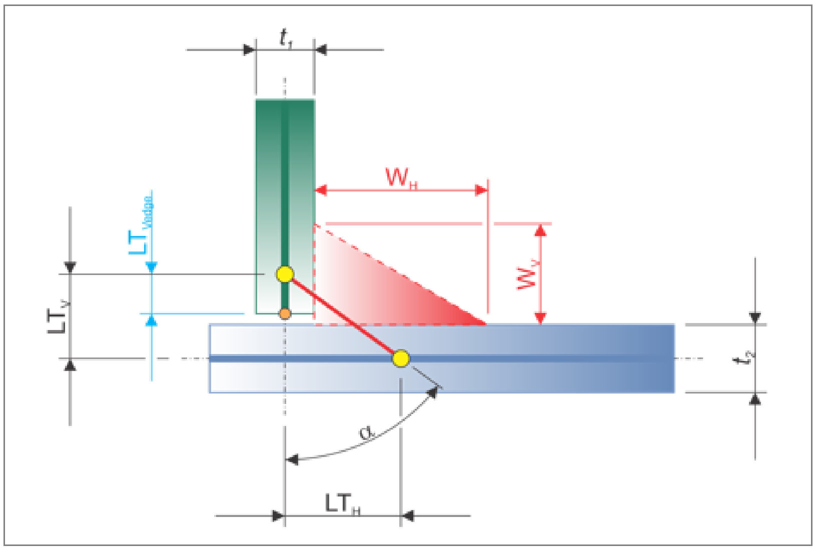


Figure 1134: Dimensioning T

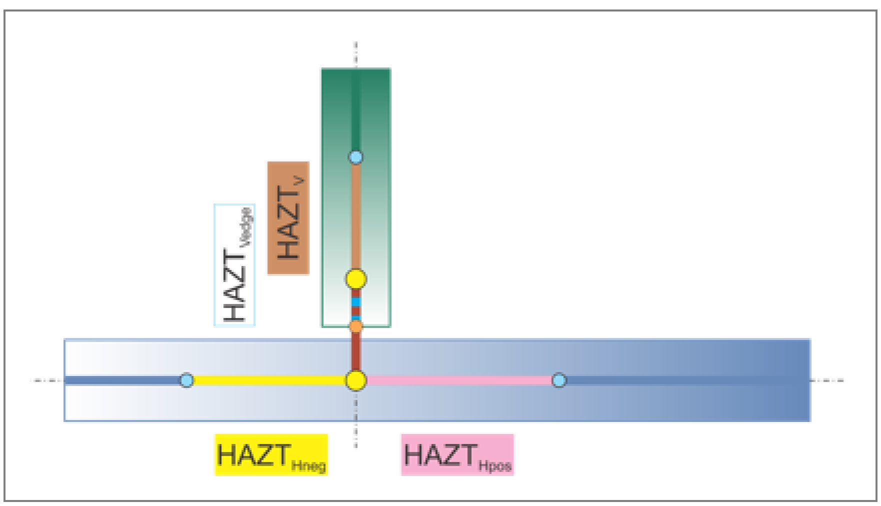


Figure 1135: Vertical T Weld HAZ

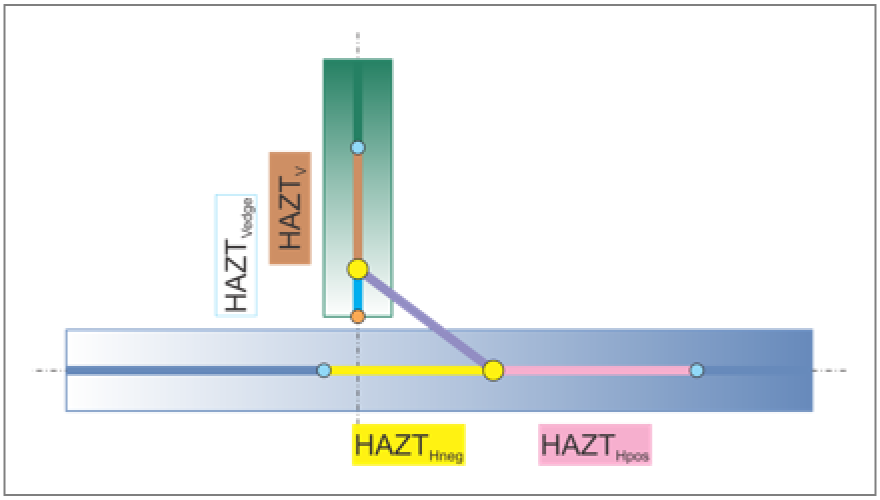


Figure 1136: Angled T Weld HAZ

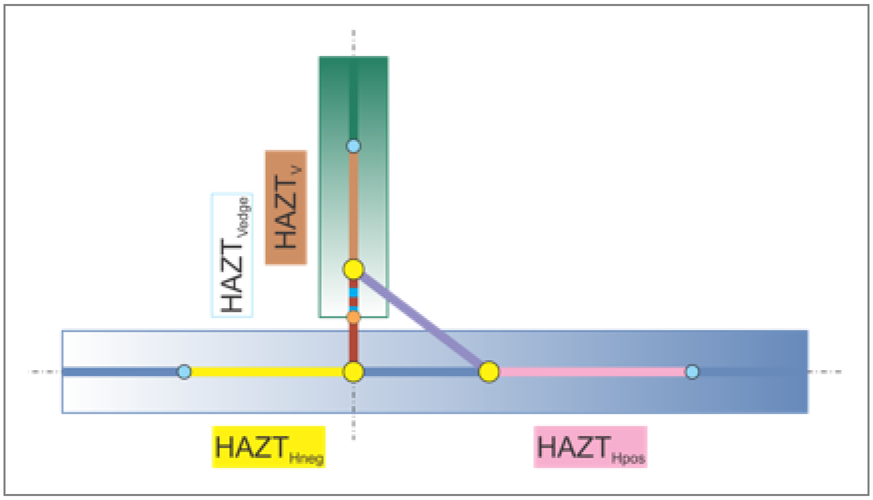


Figure 1137: Vertical and Angled T Weld HAZ

Dimensioning L

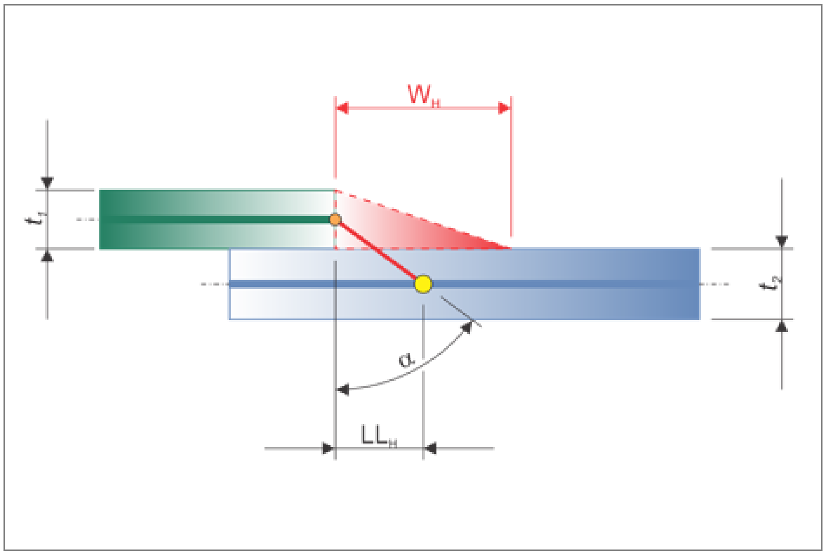


Figure 1138: Dimensioning L

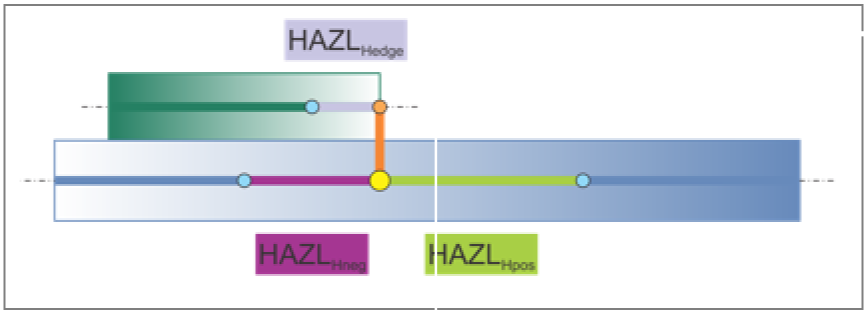


Figure 1139: Vertical L Weld HAZ

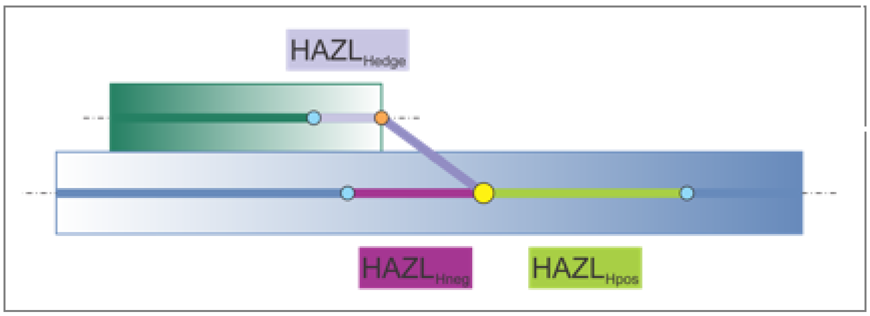


Figure 1140: Angled L Weld HAZ

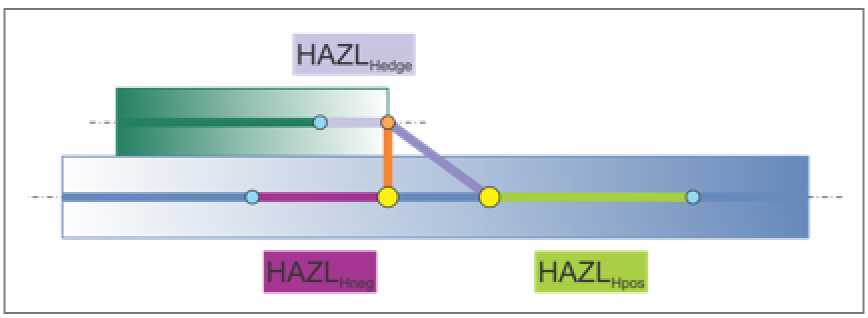


Figure 1141: Vertical and Angled L Weld HAZ

Dimensioning B

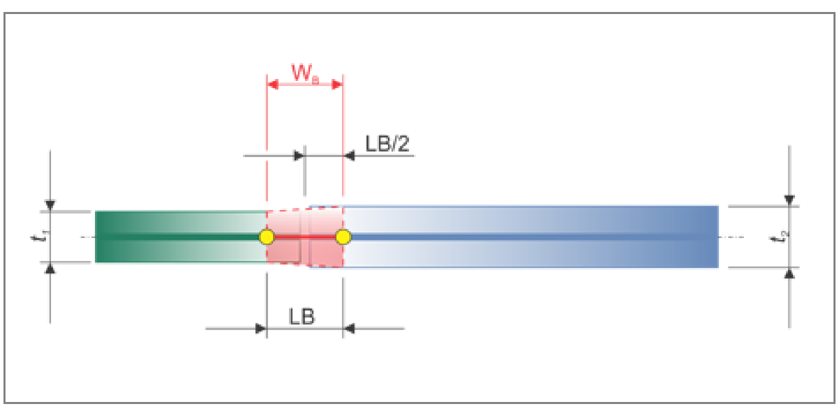


Figure 1142: Dimensioning B

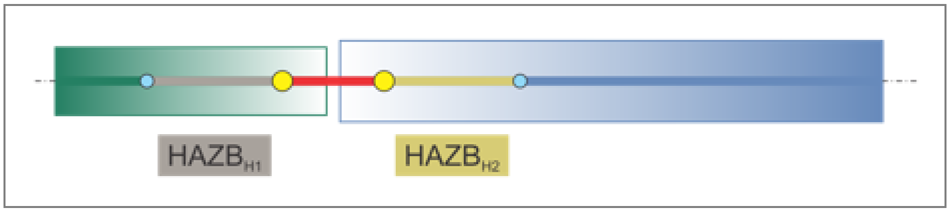


Figure 1143: Straight B Weld HAZ

Property and Material Info

The Property and Material Info parameters define the properties and materials of the welds and the heat affected zones (HAZ).

HAZ Organize Scheme

Choose a HAZ organize scheme:

inherit property

Inherits the elements of the HAZ from the links in which the HAZ elements are imprinted.

general property

Assigns the same HAZ property throughout one link, or throughout all links.

Use the subsequent options to define how the properties are determined.

individual property

Assigns individual properties to each HAZ.

HAZ Component Option

stay in original

Keeps HAZ elements in the component they were imprinted into. No additional properties get created.

new component per original one

Creates a new component for each component that gets a HAZ imprinted. The direct property assignment setting is ignored when this option is selected.

HAZ Property Option

The options available are dependent on the HAZ Organize Scheme selected.

assign original property

Assigns the same property that was assigned to the original components to new components.

assign duplicated property

Duplicates the original properties and assigns them to new components.

select

Select a property from the current model via the Select Property For HAZ option. Unless direct property assignment is activated, a component named ltb_seam_quad_haz_ with the property ID as a postfix is created.

same as original

Assigns HAZ elements the same property as the original. No further properties are created. HAZ elements are organized into components named ltb_seam_quad_haz_ with the property ID as postfix.

scaled original thickness

Creates a new property and component for each link that has a HAZ imprinted. The property is a copy of the original. Properties are named as ltb_seam_quad_haz_<linkname>_<scaled thickness>, and components are named the same as the properties.

In addition, you can define the following:

HAZ thickness factor

Enables entering a factor to scale the thickness.

HAZ Property Grouping

Groups properties in order to reduce the amount of properties created.

Do not group

Prevents grouping.

group same thickness

Groups HAZ elements with the same thickness into one property and component. HAZ elements of T, L, and B welds are also grouped together if they have the same thickness.

Properties are named ltb_seam_quad_haz_<scaled thickness> or ltb_seam_quad_haz_<property ID>, and components use the same name as properties.

group same thickness within T, L, and B

Groups all HAZ elements with the same thickness into one property and component, as long as they have the same weld type of T, L, B.

Properties are named ltb_seam_quad_<t or l or b>_<thickness>, and components use the same name as properties.

input thickness

Creates a new property and component for each link that has a HAZ imprinted. The property is a copy of the original. Properties are named ltb_seam_quad_haz_<linkname>_<scaled thickness>, and components are named the same as the properties.

In addition, you can define the following:

HAZ thickness

Enables a factor for thickness to be entered.

HAZ Property Grouping

Groups properties in order to reduce the amount of properties created.

do not group

Prevents grouping.

group same thickness

Groups all HAZ elements with the same thickness into one property and component. HAZ elements of T, L, and B welds are also grouped together if they have the same thickness.

Properties are named as ltb_seam_quad_haz_<scaled thickness> or ltb_seam_quad_haz_<property ID>, and components use the same name as properties.

group same thickness

Within T, L, and B groups all HAZ elements with the same thickness into one property and component as long as they have the same weld type of T, L, B. Properties are named as ltb_seam_quad_<t or l or b>_<thickness>, and components use the same name as properties.

same as positive side

Guarantees the HAZ on the positive and negative side of the T or L weld are assigned the same property.

same as the other size

Guarantees the HAZ on both sides of the B weld are assigned the same property.

Weld Property

Define how the thicknesses for the different parts of the weld are determined. Appropriate PSHELL properties are created.

Property Option For Vertical Quads

select

Select a property from the current model via the Select Property For Vertical Quad field. Unless direct property assignment is activated, a component with the the name ltb_seam_quad_weld_ and the property ID as postfix is created.

Lh/sqrt(2)

Determines the thicknesses of welds.

0.5*Lh/sqrt(2)

Options are dependent on the weld type (T, L, B) and the selected weld shapes (vertical, angled, vertical and angled).

(Lh/sqrt(2)+Lv/sqrt(2))/4

Properties are named ltb_seam_quad_weld_<weldshape>_<link1>_<link2>_<thickness>, and components are named the same as the properties and host the weld elements.
Lh is the superset of LLH and LTH (see dimensions above). Lv is the superset of LLV and LTV (see dimensions above).

same as edge

Inherits the property of the link with the free edge for the vertical weld. Unless direct property assignment is activated, a component with the name ltb_seam_quad_weld_ with the property ID as postfix is created.

input thickness

Creates properties with the required thicknesses for each link combination and weld shape (vertical, angled, straight).
The properties are named
ltb_seam_quad_weld_<weldshape>_<link1>_<link2>_<thickness>, and the corresponding components are named the same as the properties and host the weld elements.

Property Option for Angled Quads

select
Select a property from the current model via the Select Property For Vertical Quad field. Unless direct property assignment is activated, a component with the the name ltb_seam_quad_weld_ and the property ID as postfix is created.

Lh/sqrt(2)
Determines the thicknesses of welds.

0.5*Lh/sqrt(2)
Options are dependent on the weld type (T, L, B) and the selected weld shapes (vertical, angled, vertical and angled).

(Lh/sqrt(2)+Lv/sqrt(2))/4
Properties are named ltb_seam_quad_weld_<weldshape>_<link1>_<link2>_<thickness>, and components are named the same as the properties and host the weld elements.

(t1+t2)/2
Lh is the superset of LLH and LTH (see dimensions above). Lv is the superset of LLV and LTV (see dimensions above).

input thickness
Creates properties with the required thicknesses for each link combination and weld shape (vertical, angled, straight).
The properties are named ltb_seam_quad_weld_<weldshape>_<link1>_<link2>_<thickness>, and the corresponding components are named the same as the properties and host the weld elements.

Property Option for Capped Quads

select
Select a property from the current model via the Select Property For Vertical Quad field. Unless direct property assignment is activated, a component with the the name ltb_seam_quad_weld_ and the property ID as postfix is created.

Lh/sqrt(2)
Determines the thicknesses of welds.

0.5*Lh/sqrt(2)
Options are dependent on the weld type (T, L, B) and the selected weld shapes (vertical, angled, vertical and angled).

(t1+t2)/2
Properties are named ltb_seam_quad_weld_<weldshape>_<link1>_<link2>_<thickness>, and components are named the same as the properties and host the weld elements.
Lh is the superset of LLH and LTH (see dimensions above). Lv is the superset of LLV and LTV (see dimensions above).

input thickness

Creates properties with the required thicknesses for each link combination and weld shape (vertical, angled, straight).

The properties are named ltb_seam_quad_weld_<weldshape>_<link1>_<link2>_<thickness>, and the corresponding components are named the same as the properties and host the weld elements.

Property Option for Straight Quads

(t1+t2)/2

Determines the thicknesses of welds.

Options are dependent on the weld type (T, L, B) and the selected weld shapes (vertical, angled, vertical and angled).

Properties are named ltb_seam_quad_weld_<weldshape>_<link1>_<link2>_<thickness>, and components are named the same as the properties and host the weld elements.

Lh is the superset of LLH and LTH (see dimensions above). Lv is the superset of LLV and LTV (see dimensions above).

input thickness

Creates properties with the required thicknesses for each link combination and weld shape (vertical, angled, straight).

The properties are named ltb_seam_quad_weld_<weldshape>_<link1>_<link2>_<thickness>, and the corresponding components are named the same as the properties and host the weld elements.

Weld Property Grouping

Reduce the number of properties created by grouping them, except when using the select and same as edge options.

do not group

No grouping will take place. Properties are created as described in previous options.

group same thickness

Groups all weld elements with the same thickness into one property and one associated component. Vertical, angled, and straight weld elements that have the same thickness are also grouped together.

Properties are named ltb_seam_quad_weld_<thickness>, and their associated components are named the same as the property.

group same thickness within vertical, angled + capped and straight quads

Groups all weld elements with the same thickness and weld shape (vertical, angled + capped, straight) into one property and one associated component.

Properties are named ltb_seam_quad_<vertical or angled_capped or straight>_<thickness>, and their associated components are named the same as the property.

Direct Property Assignment

Stops additional components from being created, and directly assigns created or selected properties to individual weld or HAZ elements.

Behavior

B/L classification angle

Angle that is automatically determined for each individual seam connector, whether it is to be considered a butt weld or a lap weld. Default is set to 10.0°.
If the angle of the two links is smaller than the B/L classification angle, then it will be considered a butt weld and a lap weld; a further check determines whether the links overlap. If the links do not overlap, a butt-weld is performed.

L/T classification angle

Angle that is automatically determined for each individual seam connector, whether it is to be considered a lap weld or a t-weld. Default is set to 10.0°.

Angle Direction

Defines which side the angled weld elements are created.

connector side

Angled weld elements are created on the side where the connector is located, as long as the connector is not perfectly on the free edge.
If the connector is on the free edge, the edge quad normal option will be automatically used.

positive side/negative side

The positive and negative side can be determined as long as the links are not perfectly perpendicular to each other.

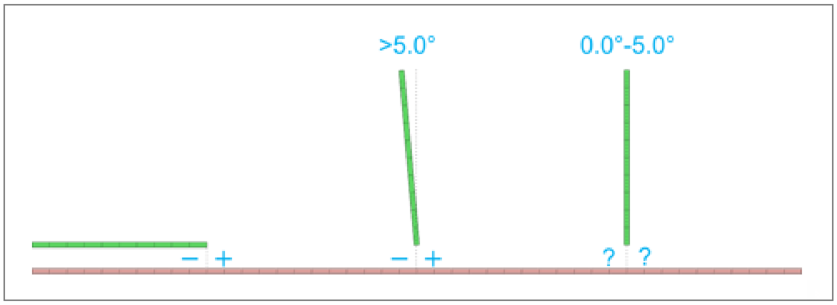


Figure 1144:

Overview of how the positive and negative side is determined. When links are perfectly perpendicular, the edge quad normal option is automatically used.

edge quad normal

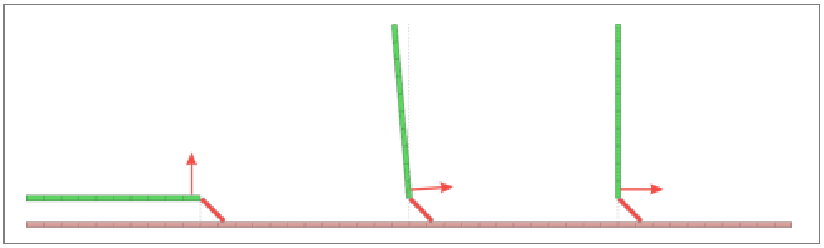


Figure 1145:

Overview of how the side for the angled weld is determined. If the normal directions are reversed, the side of the angled weld changes.

Snapping to Edge

Automatic edge snapping can be used to precisely position connectors. First, the connector snaps to, for example, the closest free edge, then the projection and FE creation starts.
The snapping distance can be defined separately for T, L and B connections.
You can choose whether to snap to:

- maximum 1 element row
- maximum 2 element rows
- no (connector does not snap)

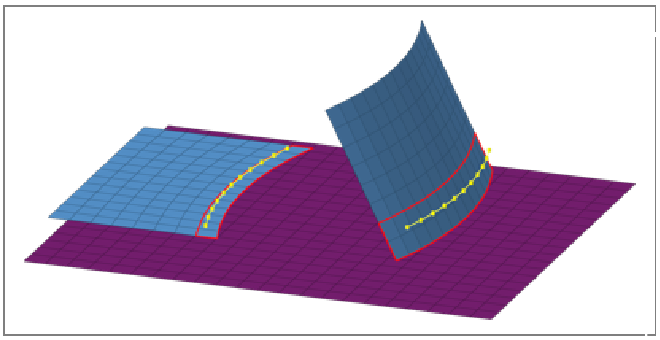


Figure 1146: Original Model before Realization Initial situation with one element row marked for the lap weld and two element rows for the t weld.

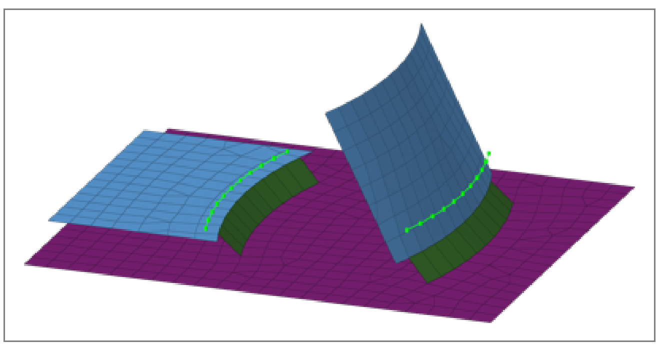


Figure 1147: Edge Snapping Enabled

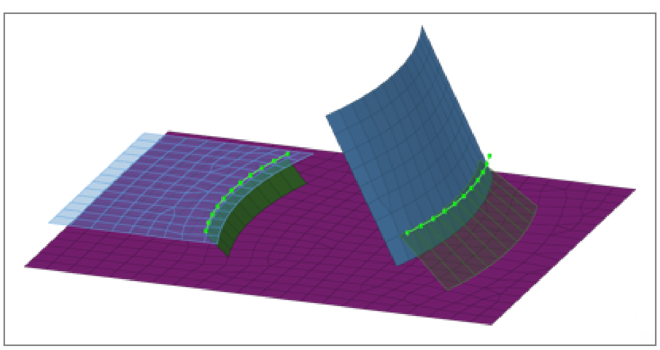


Figure 1148: Edge Snapping Disabled

Edge Treatment (T/B)

Attempts to create specific vertical lengths for T connections LTV, and specific lengths for B connections LB.

Only enabled when the Edge Treatment setting is enabled from the Realization Details settings.

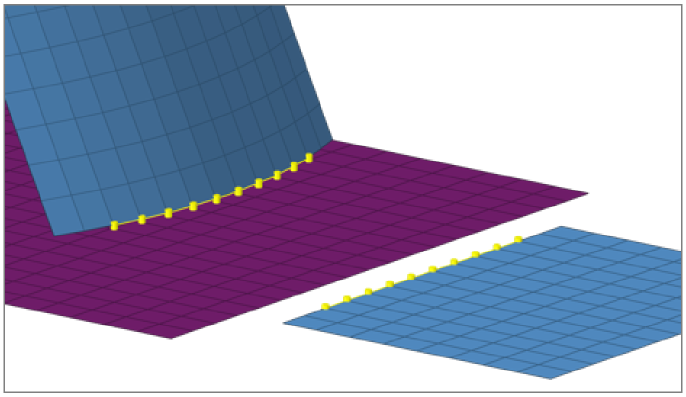


Figure 1149: Original Model before Realization

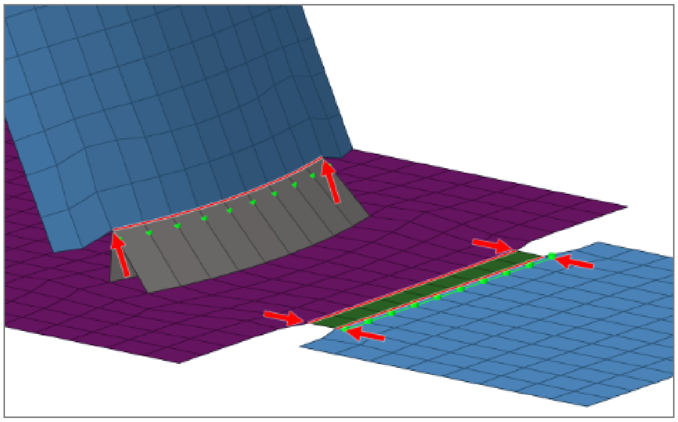


Figure 1150: Realization using Edge Treatment

Free edges were contracted or extended.

Edge Treatment Options

Choose whether to:

- extension and contraction
- extension
- contraction

Edge Treatment Limit

Edge treatment is a pure node movement; therefore, the maximum movement needs to be limited to prevent the elements at the edge from being destroyed. Movement is limited to a maximum of 0.5 times the element size at the edge. 0.5 is the maximum allowed value and default value.

Preserve Washer

Controls how washers are preserved during the seam imprint realization.

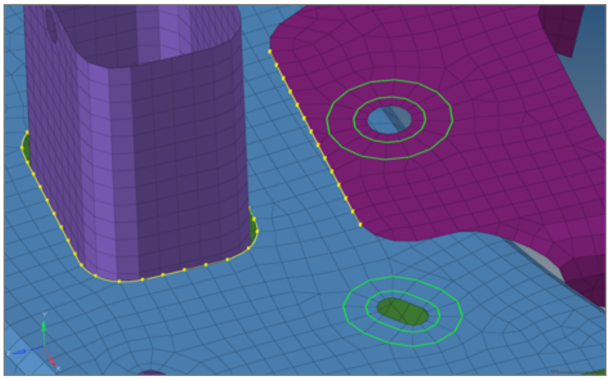


Figure 1151: Original Mode with Perfectly Meshed Washers

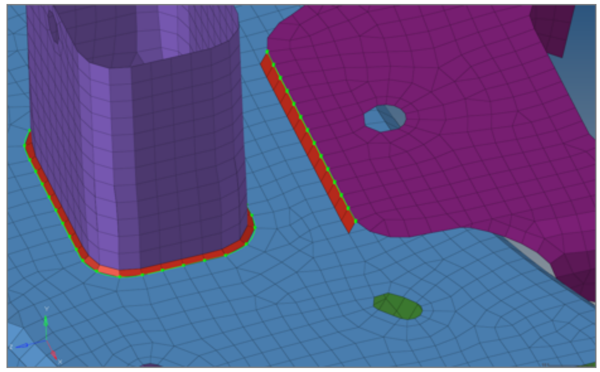


Figure 1152: No Washer Preservation Enabled

Washers have been opened.

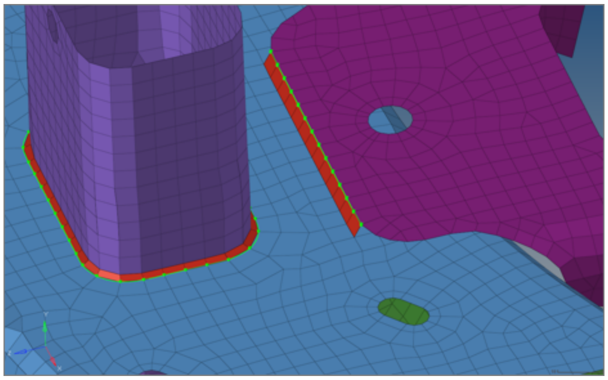


Figure 1153: Washer Preservation and Remesh Enabled

Washers are still intact, but the mesh seeding has been modified.

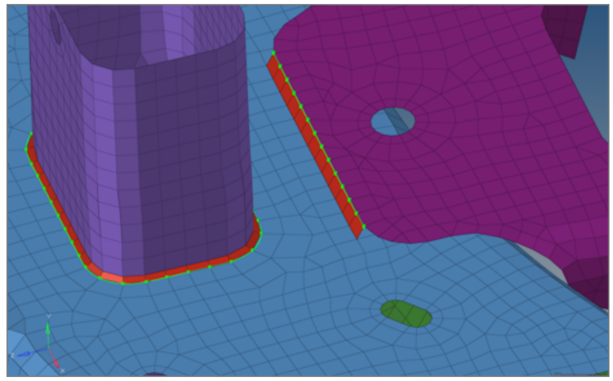


Figure 1154: Washer Preservation and No Remesh Enabled

The washers have been fully preserved.

Don't Share Zone Elements

Seam imprint allows heat affected zones (HAZ) to be merged in close areas. In this situation, one element might touch the weld elements from two different connectors. Do not share zone elements prevents zone elements from being shared.

Quad In Corner

Controls whether a single or double element is created in corners of quad seam connectors with a certain vertex.

A angle must be defined for a single quad corner. If the corner angle is greater than the defined angle, a double quad corner is created.

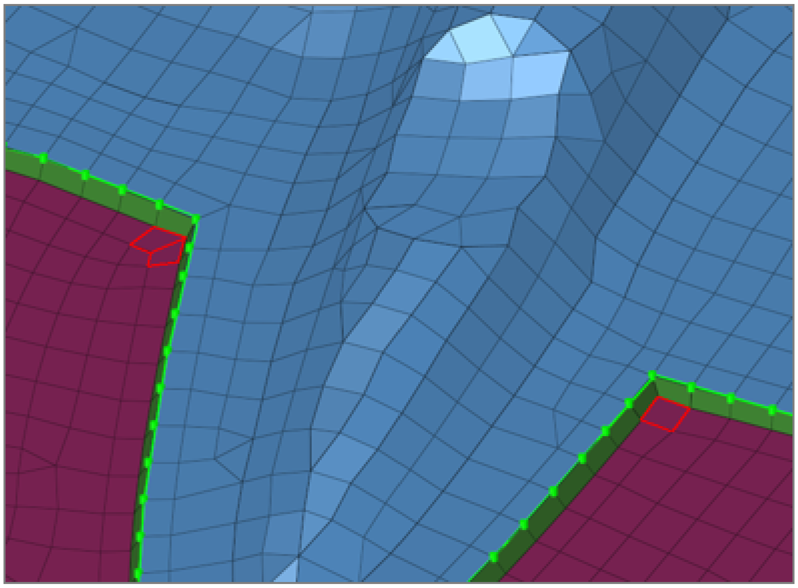


Figure 1155: Quad in Corner

A double quad corner is shown on the right, and a single quad corner is shown on the left.

Quad Control

Controls the maximum deviation from the perfect quad element for the heat affected zone (HAZ). It can be controlled, if the element size or the element skew is more important to retain.

Max Quadsize Reduction In % / Max Quad Skew In Degrees

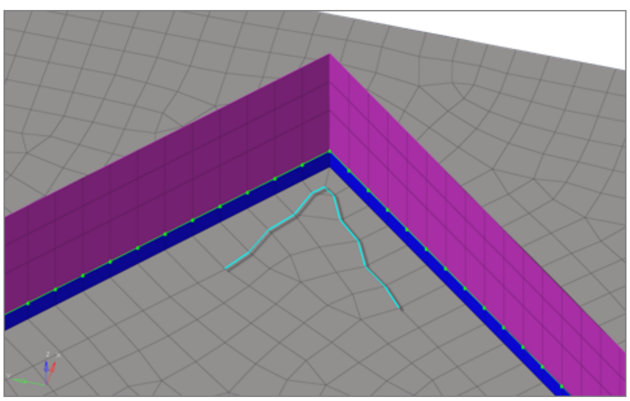


Figure 1156: Max Quad Size Reduction: 80.0 / Max Quad Skew: 5.0

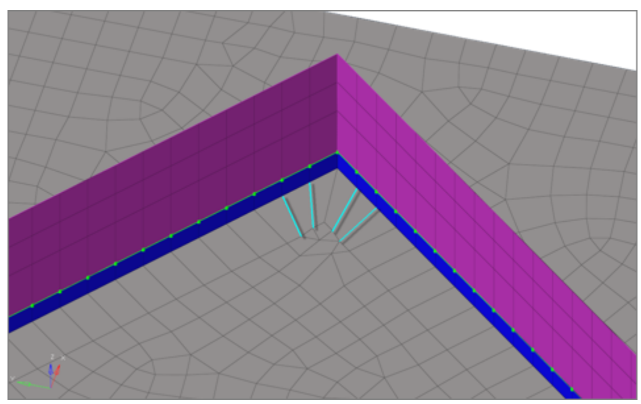


Figure 1157: Max Quad Size Reduction: 5.0 / Max Quad Skew: 45.0

Silver Elements

Sliver elements are small elements that you may not want in your model. In the images below, a perfect perpendicular projection resulted in sliver elements. The Sliver Elements setting can be used to manage sliver elements in your model. In the images below, the red elements represent the HAZ elements.

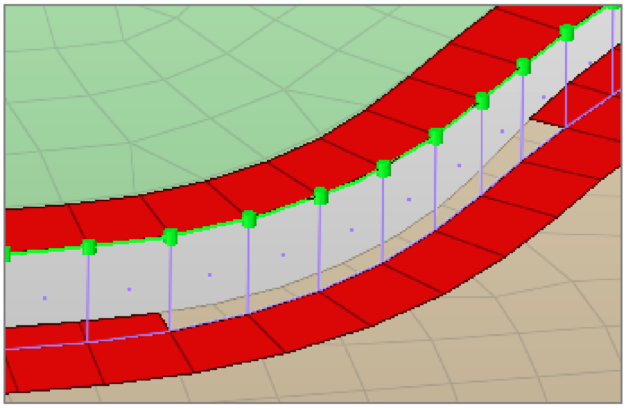


Figure 1158: Allow

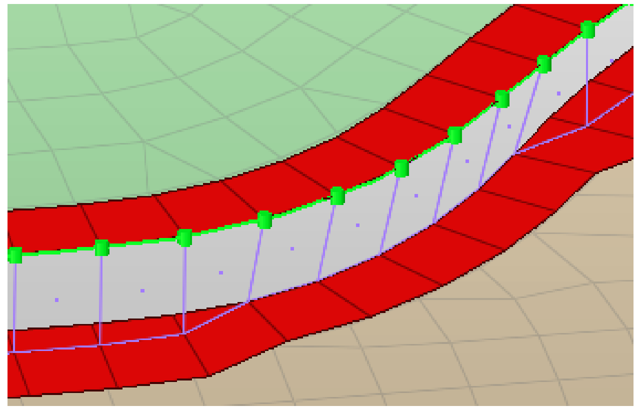


Figure 1159: Prevent by Moving Projection Points

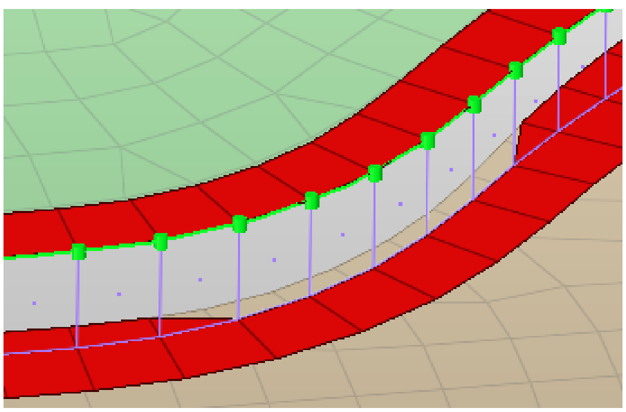


Figure 1160: Prevent by Moving Edge

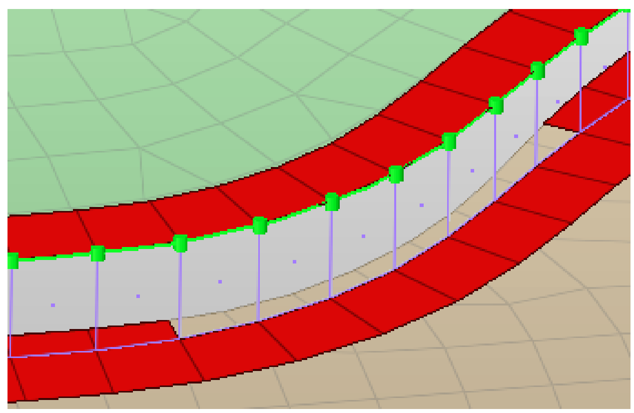


Figure 1161: Delete Sliver Elements

Element length<

This length controls which elements to treat as sliver elements.

Feature Angle

Determines important features to retain during the imprint. Features that cross the HAZ, as well as near by features cannot be retained.

Seam-Rigid LTB

The Seam-Rigid LTB realization serves and realizes t-welds, lap-welds and butt-welds at the same time.

The weld type is identified automatically based on the orientation of the links to each other.

The dimensions and property for all heat affected zones (HAZ) can be defined separately. An edge treatment can be defined for t-welds and butt-welds to move the edge a precise distance from the opposite link.

Restriction: Available in the OptiStruct and Nastran solver interfaces, and can only be selected and defined in the Connector Entity Editor.

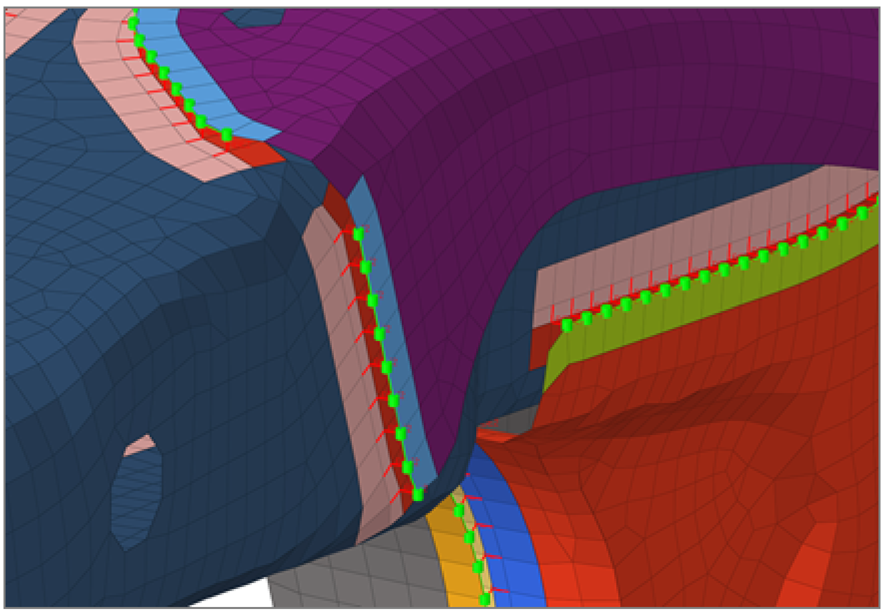


Figure 1162: Seam-Rigid LTB

General Info

Weld Type

Defines weather to setup a configuration exclusively for a T, L, or B connection, or automatically setup a configuration for each connection based on the angle.
In any case, the connection type is dependent on the:

- B/L classification angle
- L/T classification angle

Both types of angles are defined in the Behavior section.

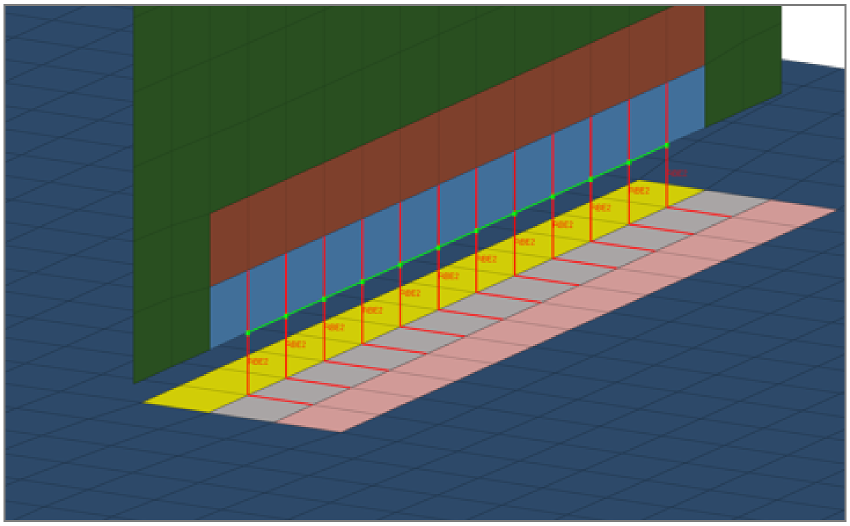


Figure 1163: T Connection

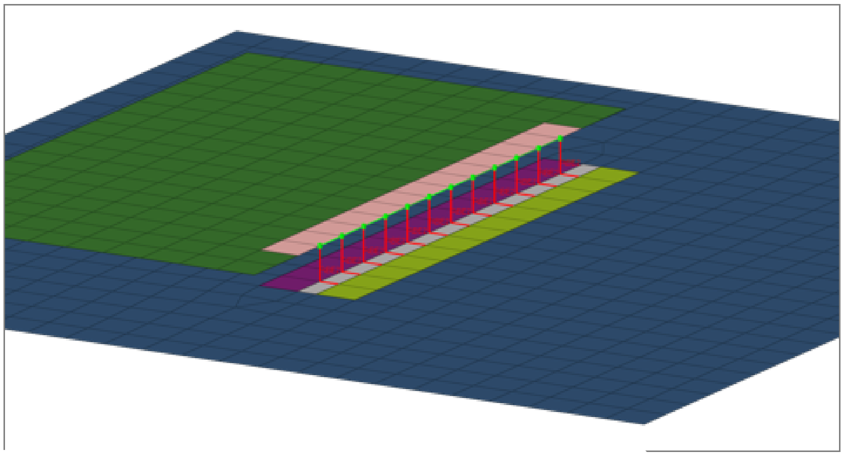


Figure 1164: L Connection

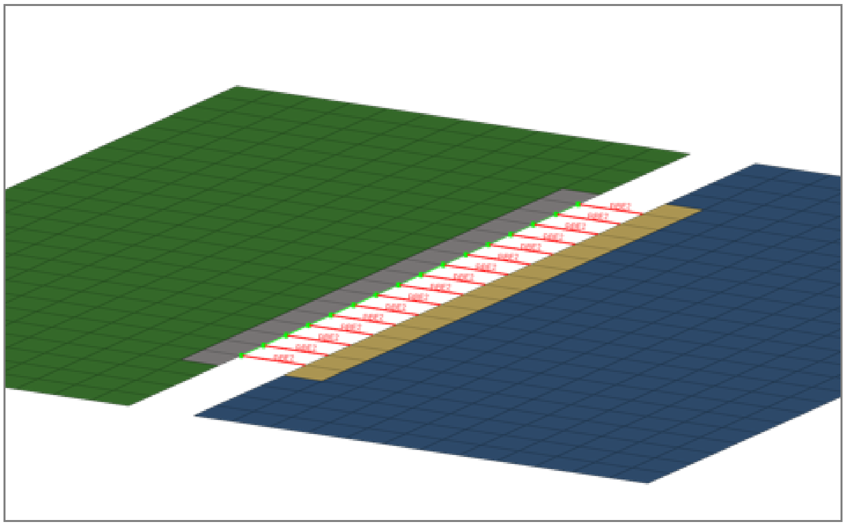


Figure 1165: B Connection

Tolerance

Defines the distance from the connector location.

Only entities within this tolerance can be taken into account for the final realization. The tolerance is used to verify whether adequate link candidates are available to be connected with respect to the number of layers.

Weld Shape

T Weld Shape

Defines how the T weld is created.

The image below shows where the master and slave nodes of the rigid elements will be placed.

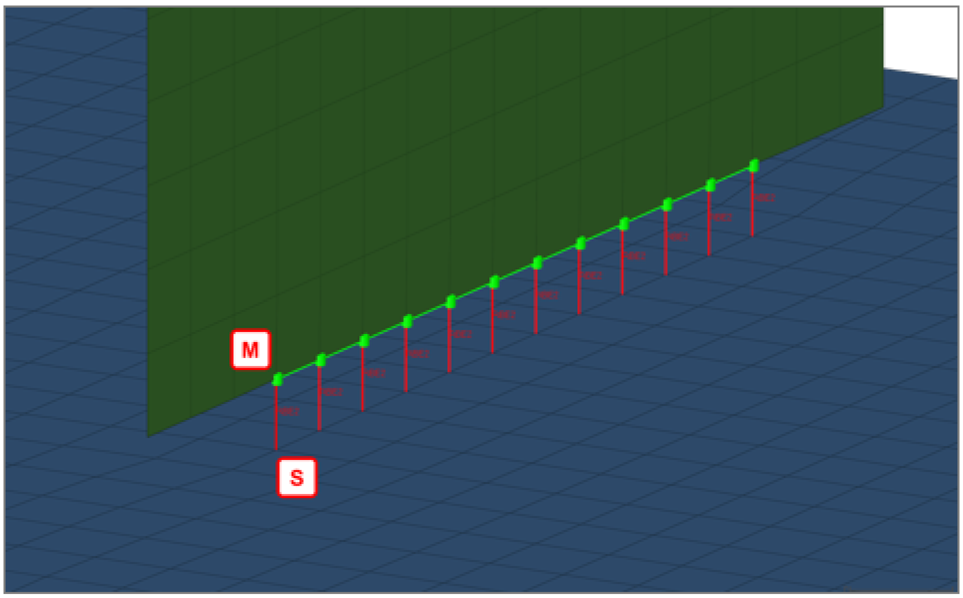


Figure 1166: Vertical T Weld

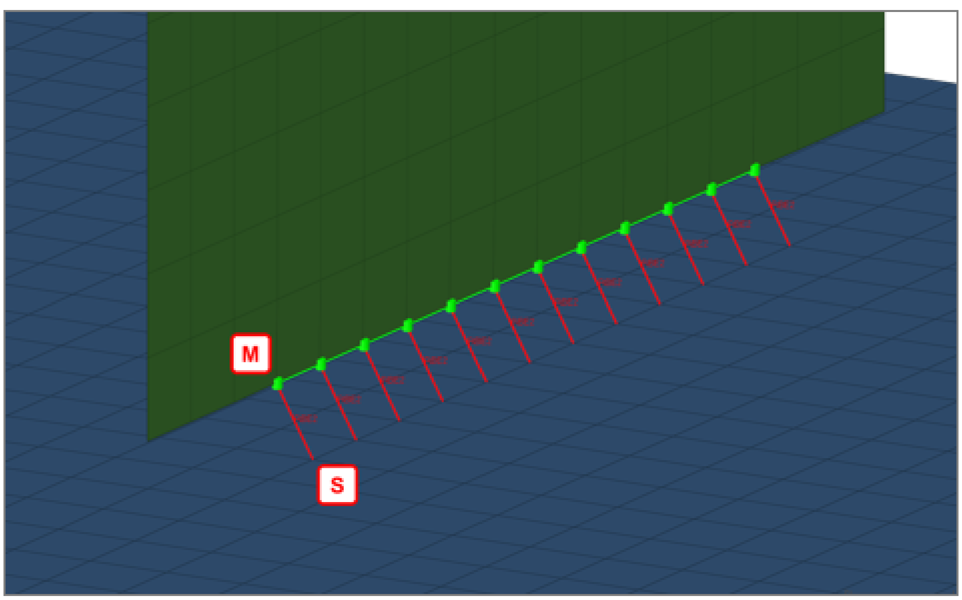


Figure 1167: Angled T Weld, Connected to Edge

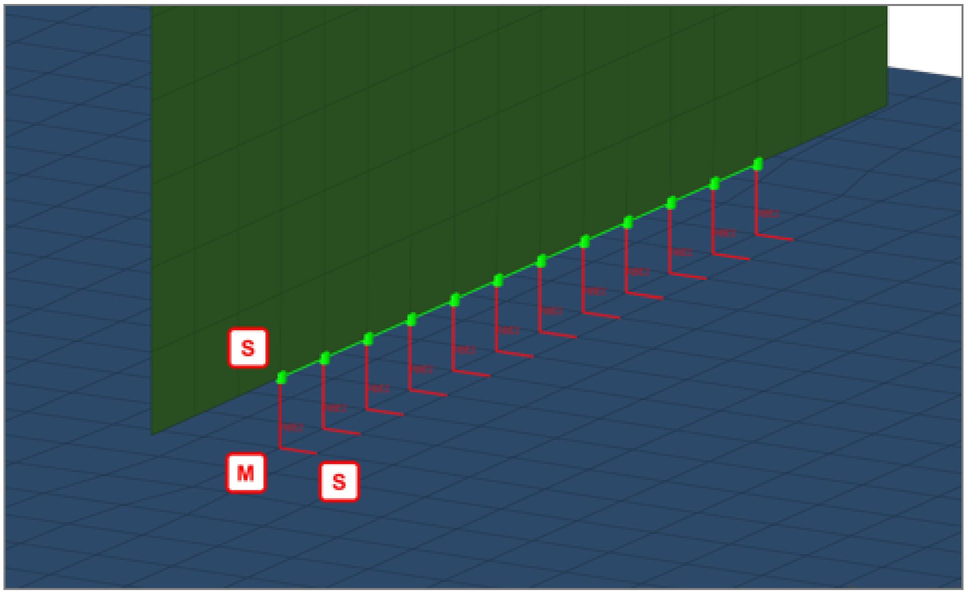


Figure 1168: Vertical and Horizontal T Weld

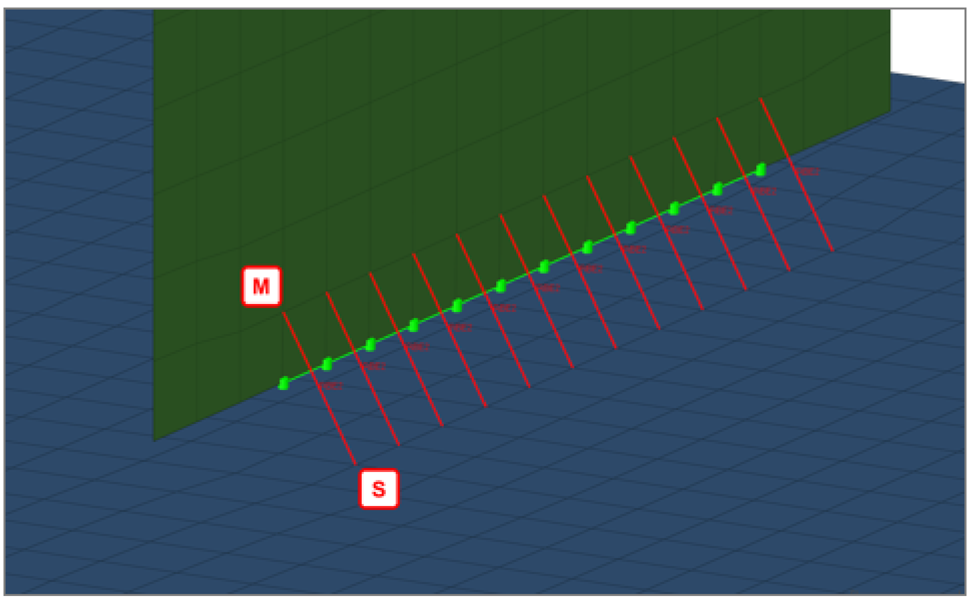


Figure 1169: Angled T Weld, Connected at a Defined Vertical Height LTV

L Weld Shape

Defines how the L weld is created.

The image below shows where the master and slave nodes of the rigid elements will be placed.

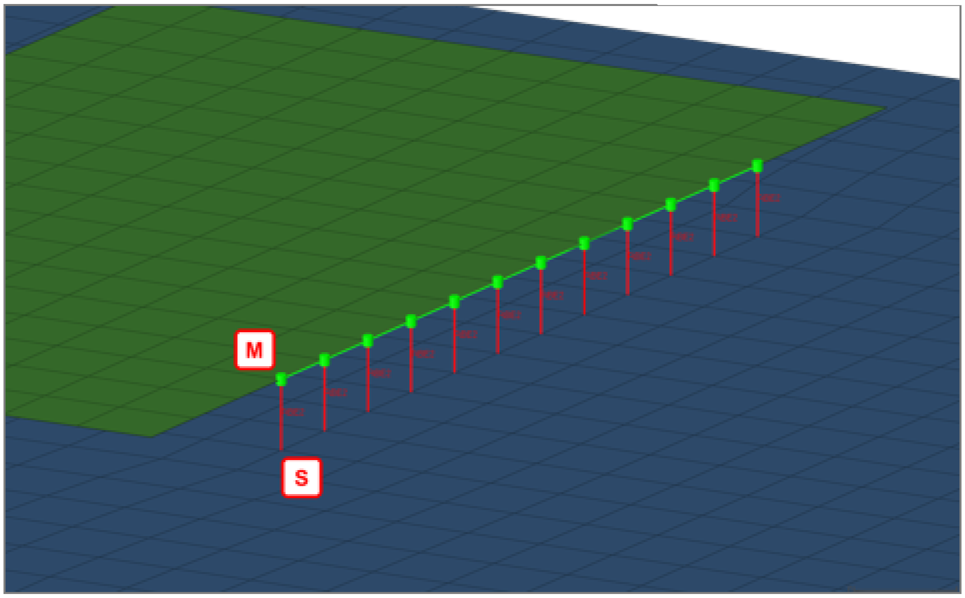


Figure 1170: Vertical L Weld

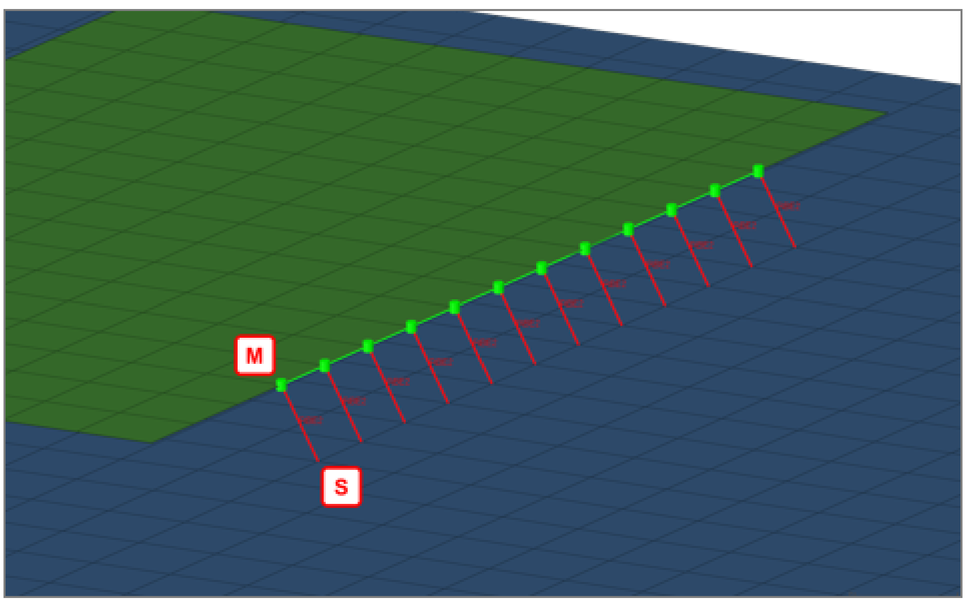


Figure 1171: Angled L Weld

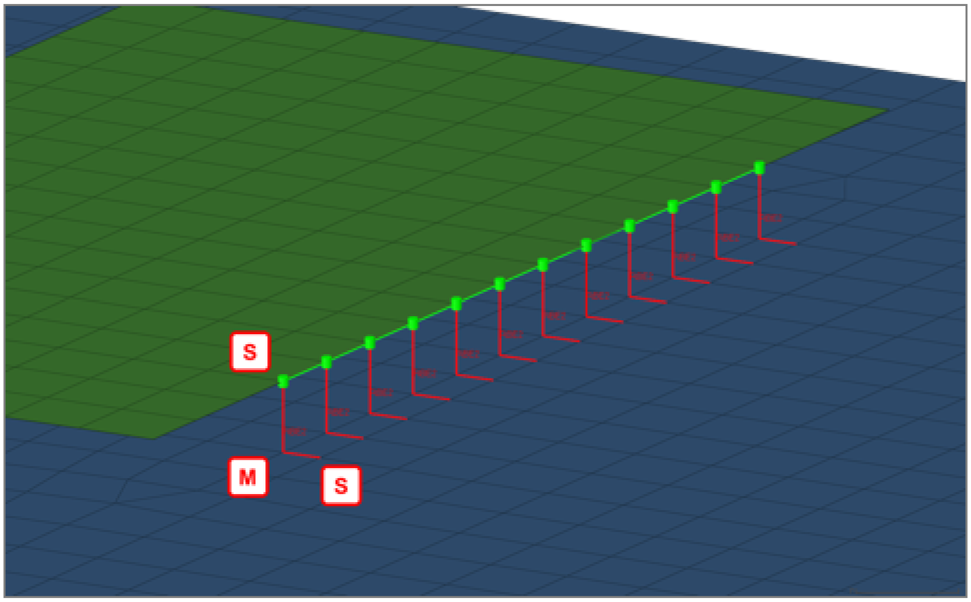


Figure 1172: Vertical and Angled L Weld

B Weld Shape

B welds are always created in a straight manner.

The image below shows where the master and slave nodes of the rigid elements will be placed. The nodes can also be placed the opposite direction, but their position will always be consistent throughout each seam.

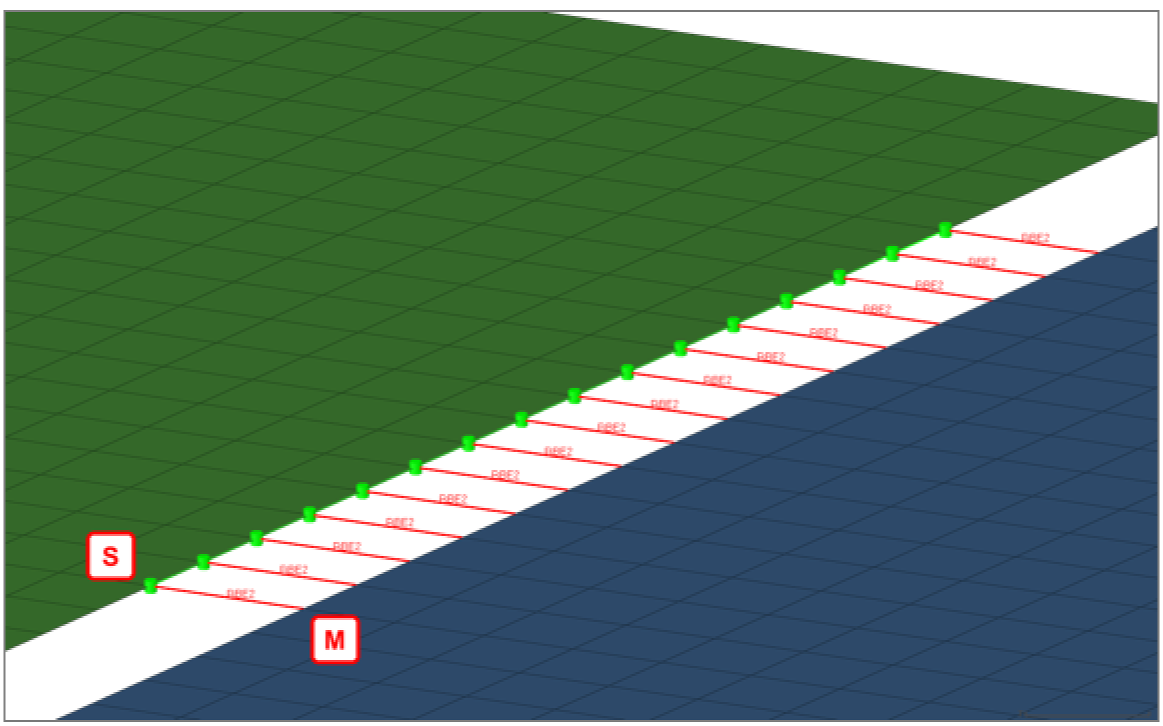


Figure 1173: Straight B Weld

Realization Details

The Realization Details settings position the yellow marked nodes in Figure 1174, Figure 1175, and Figure 1176.

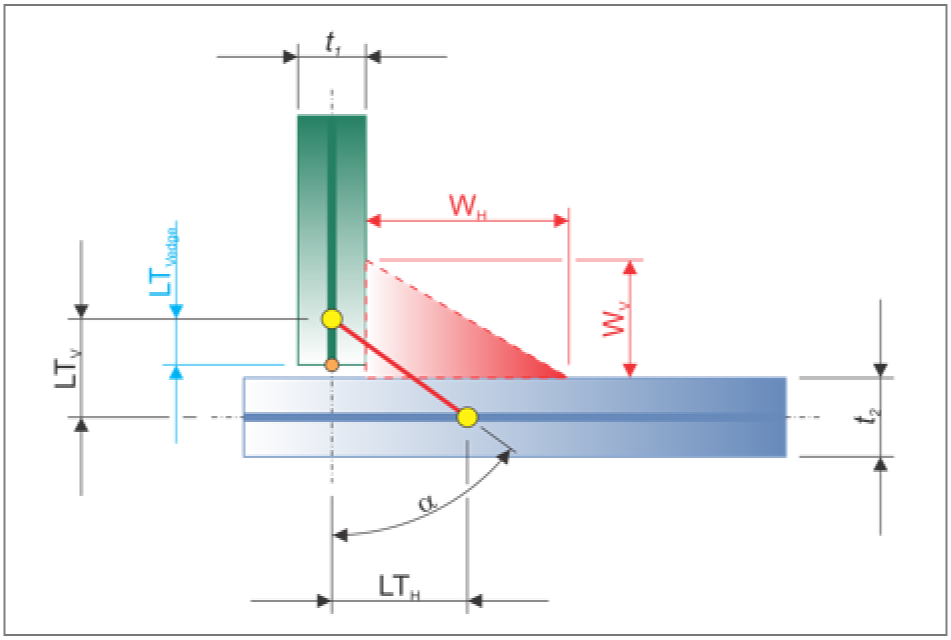


Figure 1174: T Dimensions

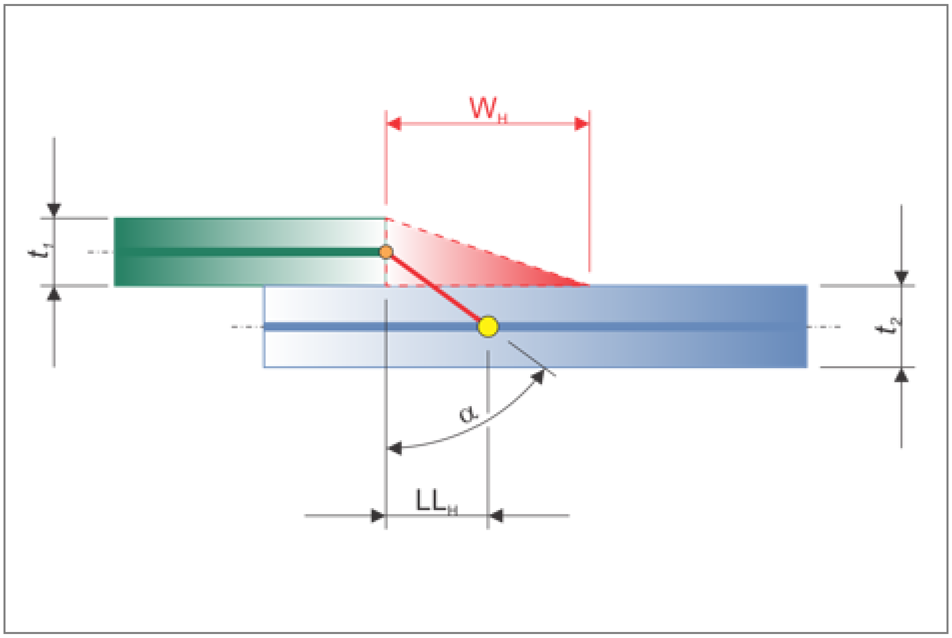


Figure 1175: L Dimensions

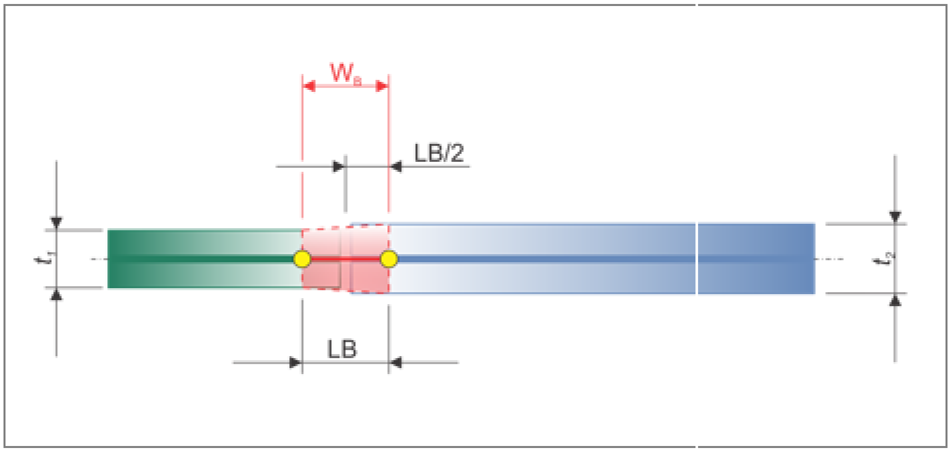


Figure 1176: B Dimensions

The dimension of the welds are dependent on the Weld Shape settings.

Dimensioning Scheme

Defines the dimensioning scheme for the dimensions of the T weld, L weld, and B weld connections.

input

Manually define discrete values for the weld dimensions, shown in black in Figure 1174, Figure 1175, and Figure 1176, with the exception of thickness. The horizontal dimensions can be defined using a length or an angle.

thickness dependent

Choose a formula to define the weld dimensions, shown in black in Figure 1174, Figure 1175, and Figure 1176, with the exception of thickness. The provided formulas are all dependent on the thicknesses t1 and t2. A formula can be chosen individually for each verticalV and horizontalH distance, or the same formula can be used for T, L and B.

weldsize dependent

Manually define discrete values for the weld dimensions, shown in red in Figure 1174, Figure 1175, and Figure 1176. The verticalV and horizontalH distances are defined with formulas reflecting the weld sizes and the t1 and t2 thicknesses.

DIM T (Dimensioning T)

	Input	Thickness dependent	Weldsize dependent
Horizontal Lengths LTH	by angle by length	(t1+t2)/2 3*(t1+t2)/2 t1+t2	t1/2+wh/2 (fix)
Vertical Length LTV	by length by edge	(t1+t2)/2 3*(t1+t2)/2 t1+t2 by edge	t2/2+wv/2 by edge

DIM L (Dimensioning L)

	Input	Thickness dependent	Weldsize dependent
Horizontal Lengths LTH	by length by angle	(t1+t2)/2 3*(t1+t2)/2 t1+t2	wh/2 (fix)

DIM B (Dimensioning B)

	Input	Thickness dependent	Weldsize dependent
Lengths LB	by length by angle	(t1+t2)/2 3*(t1+t2)/2 t1+t2 by edge	wb by edges

Edge Treatment (T/B)

When discrete lengths are requested for T and B connections, it is sometimes necessary to move the edges.

Edge treatment is not needed when the different length dimension settings are set to by edge. When enabled, edges are allowed to move. See Edge Treatment Options for more information.

Max Length Value

Defines the maximum length value.

This setting is useful when lengths are calculated based on thicknesses. If a length is greater than the Max Length Value, then the Max Length Value will be used instead.

Element Details

The DOF parameter controls the setting for the Degrees of Freedom for the RBE2 elements, which are created during the realization.

Connectivity Info

imprint (default)

Creates quad weld elements, and stitches them to both links by adjusting their mesh. All required HAZ are performed.

skip imprint

Creates quad weld elements, but does not change the meshes of the links. Instead, additional elements are created to represent the requested HAZ. These elements are organized in the ^conn_imprint component, and can later be used for a manual imprint after they have been manipulated to your needs. This option can be helpful when working with more complex areas, where the standard imprint functionality fails, for example, conflicting connectors.

imprint/no HAZ

Creates quad weld elements, and stitches them to both links by adjusting their mesh. Mesh modifications are as minimal as possible, and no HAZ are performed.

none

Creates quad weld elements only. Quad weld elements will not be attached to the links. The connection will need further attention.

HAZ Info

The HAZ Info settings define the lengths of the different heat affected zones (HAZ), which are dependent on the HAZ lengths for T, L and B, defined in the Realization Details parameters. The HAZ length settings vary depending on the defined weld shapes (vertical, angled, vertical and angled, caps).

HAZ Scheme

Choose a dimensioning scheme for the HAZ lengths of T, L, and B.

input

Enables you to decide if the HAZ lengths should be defined individually, or if all HAZ lengths are determined using the same approach (same as all).

weldsize dependent

Only available if weldsize dependent has been chosen for the Dimensioning Scheme, as well.

HAZ Lengths

same as all

Assigns the same length to all HAZ lengths.

individual

Assign HAZ lengths individually.

Assign HAZ lengths individually

The following options are available in the various HAZ length settings.

input

Requires a discrete value be specified for the length.

average meshsize

Length is dependent on the average mesh size in the local area where the imprint is performed.

by thickness

Sets the length to the same value as the thickness of the link getting the HAZ.

LTH

Horizontal length for T connections, which is the length between the foot points of the vertical and angled part of a seam.

LLH

Horizontal length for L connections, which is the length between the foot points of the vertical and angled part of a seam.

LB

Butt weld length.

skip HAZ

Skips individual HAZ that are not required.

same as positive side

Assigns the same length as the positive side to the negative side.

wh or wh/2

Length is dependent on the horizontal weld size. Only available when HAZ Scheme is set to weldsize dependent.

wv or wv/2

Length is dependent on the vertical weld size. Only available when HAZ Scheme is set to weldsize dependent.

wb or wb/2

Length is dependent on the butt weld size. Only available when HAZ Scheme is set to weldsize dependent.

LTVedge

Choose between skip HAZ and LTVedge. Only available for the HAZ_{Tvedge} length.

HAZ Length Factor (Avg. Meshsize/Thickness)

Factor that increases or decreases the HAZ lengths, which have been defined using the average meshsize or by thickness length options.

Max HAZ Length

Maximum length for all HAZ lengths. If the HAZ length is greater than this value, then the Max HAZ Length is used.

Dimensioning and Heat Affected Zones (HAZ):

Dimensioning T

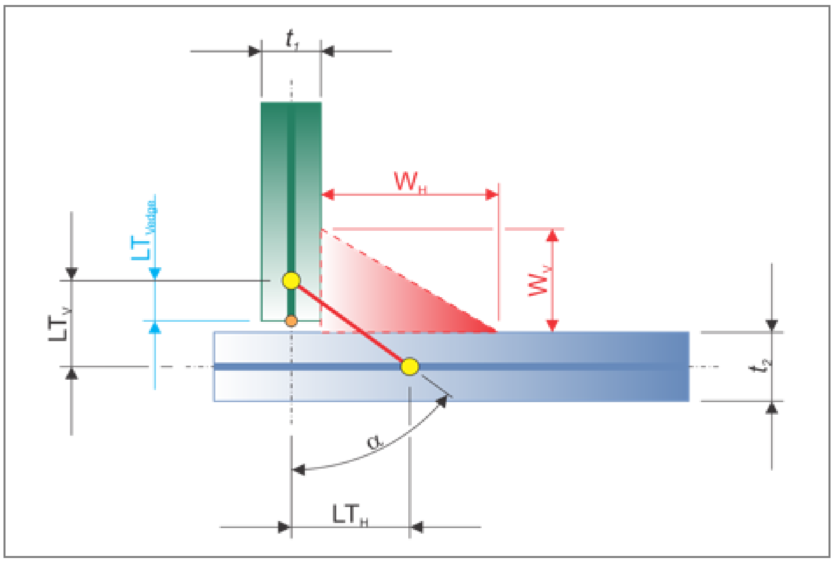


Figure 1177: Dimensioning T

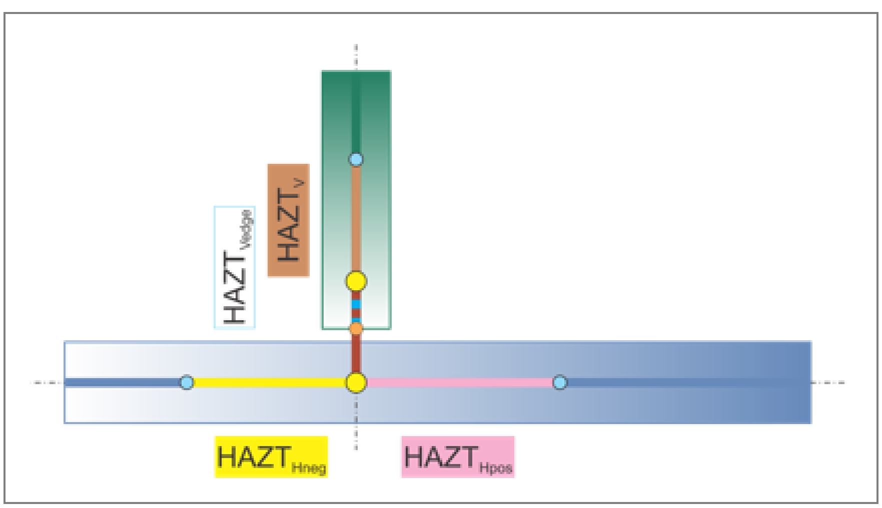


Figure 1178: Vertical T Weld HAZ

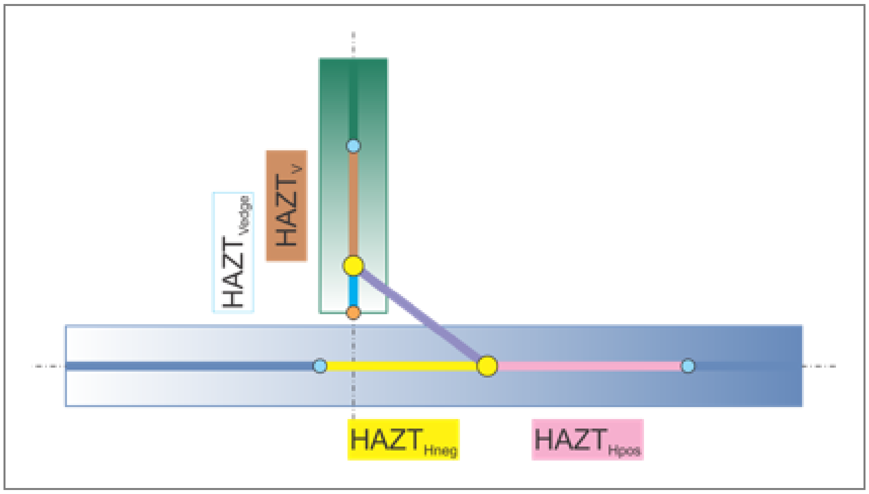


Figure 1179: Angled T Weld HAZ

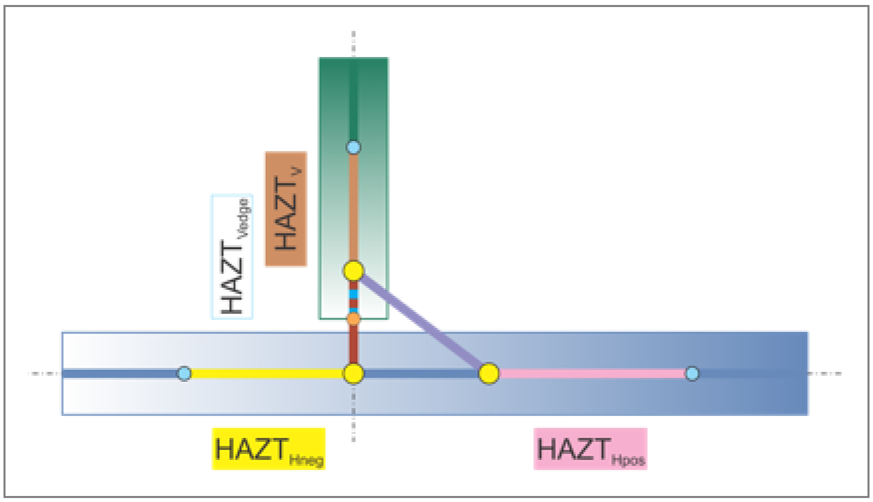


Figure 1180: Vertical and Angled T Weld HAZ

Dimensioning L

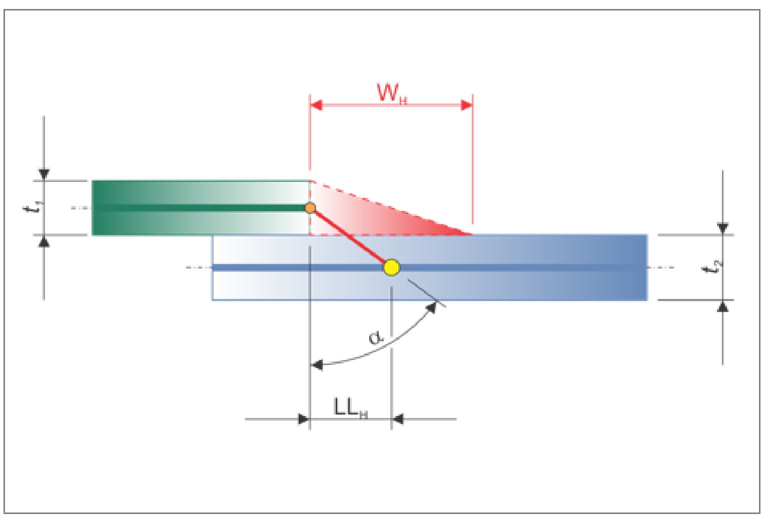


Figure 1181: Dimensioning L

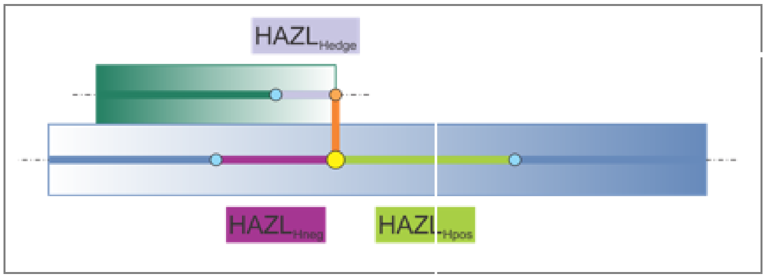


Figure 1182: Vertical L Weld HAZ

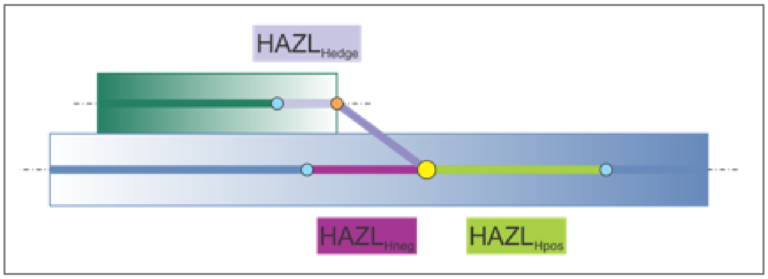


Figure 1183: Angled L Weld HAZ

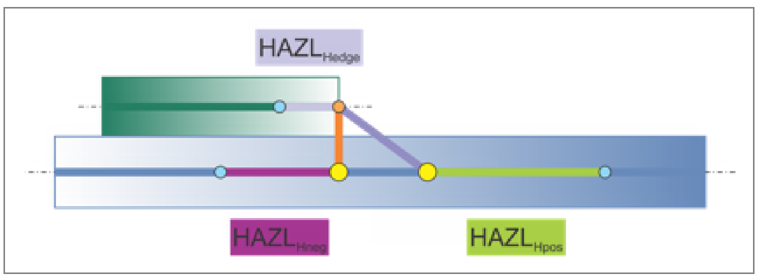


Figure 1184: Vertical and Angled L Weld HAZ

Dimensioning B

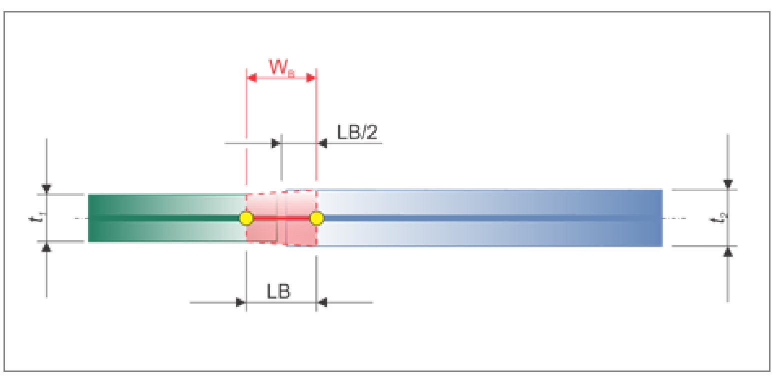


Figure 1185: Dimensioning B

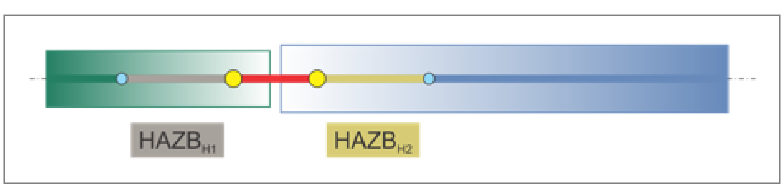


Figure 1186: Straight B Weld HAZ

Property and Material Info

The Property and Material Info settings define the properties and materials of the welds and the heat affected zones (HAZ).

HAZ Organize Scheme

Choose a HAZ organize scheme:

inherit property

Inherits the elements of the HAZ from the links in which the HAZ elements are imprinted.

general property

Assigns the same HAZ property throughout one link, or throughout all links.
Use the subsequent options to define how the properties are determined.

individual property

Assigns individual properties to each HAZ.

HAZ Component Option

stay in original

Keeps HAZ elements in the component they were imprinted into. No additional properties get created.

new component per original one

Creates a new component for each component that gets a HAZ imprinted. The direct property assignment setting is ignored when this option is selected.

HAZ Property Option

The options available are dependent on the HAZ Organize Scheme selected.

assign original property

Assigns the same property that was assigned to the original components to new components.

assign duplicated property

Duplicates the original properties and assigns them to new components.

select

Select a property from the current model via the Select Property For HAZ option. Unless direct property assignment is activated, a component named ltb_rigid_quad_haz_ with the property ID as a postfix is created.

same as original

Assigns HAZ elements the same property as the original. No further properties are created. HAZ elements are organized into components named ltb_rigid_quad_haz_ with the property ID as postfix.

scaled original thickness

Creates a new property and component for each link that has a HAZ imprinted.
The property is a copy of the original. Properties are named as ltb_rigid_quad_haz_<linkname>_<scaled thickness>, and components are named the same as the properties.

In addition, you can define the following:

HAZ thickness factor

Enables you to enter a factor to scale the thickness.

HAZ Property Grouping

Groups properties in order to reduce the amount of properties created.

do not group

group same thickness

Groups HAZ elements with the same thickness into one property and component. HAZ elements of T, L, and B welds are also grouped together if they have the same thickness.
Properties are named ltb_rigid_quad_haz_<scaled thickness> or ltb_rigid_quad_haz_<property ID>, and components use the same name as properties.

group same thickness within T, L, and B

Groups all HAZ elements with the same thickness into one property and component, as long as they have the same weld type of T, L, B.
Properties are named ltb_rigid_quad_<t or l or b>_<thickness>, and components use the same name as properties.

input thickness

Creates a new property and component for each link that has a HAZ imprinted.
The property is a copy of the original. Properties are named ltb_rigid_quad_haz_<linkname>_<scaled thickness>, and components are named the same as the properties.
In addition, you can define the following:

HAZ thickness

Enables you to enter a factor for thickness.

HAZ Property Grouping

Groups properties in order to reduce the amount of properties created.

do not group

Prevents grouping.

group same thickness

groups all HAZ elements with the same thickness into one property and component. HAZ elements of T, L, and B welds are also grouped together if they have the same thickness.
Properties are named as ltb_rigid_quad_haz_<scaled thickness> or ltb_rigid_quad_haz_<property ID>, and components use the same name as properties.

group same thickness

Within T, L, and B groups all HAZ elements with the same thickness into one property and component as long as they have the same weld type of T, L, B.
Properties are named as ltb_rigid_quad_<t or l or b>_<thickness>, and components use the same name as properties.

same as positive side

Guarantees the HAZ on the positive and negative side of the T or L weld are assigned the same property.

same as the other size

Guarantees the HAZ on both sides of the B weld are assigned the same property.

Direct Property Assignment

When activated, additional components will not be created, and created or selected properties will be directly assigned to individual weld or HAZ elements.
Used for HAZ and weld property assignment.

Behavior

B/L classification angle

Angle that is automatically determined for each individual seam connector, whether it is to be considered a butt weld or a lap weld. Default is set to 10.0°.
If the angle of the two links is smaller than the B/L classification angle, then it will be considered a butt weld and a lap weld; a further check determines whether the links overlap. If the links do not overlap, a butt-weld is performed.

L/T classification angle

Angle that is automatically determined for each individual seam connector, whether it is to be considered a lap weld or a t-weld. Default is set to 10.0°.

Angle Direction

Defines which side the angled weld elements are created.

connector side

Angled weld elements are created on the side where the connector is located, as long as the connector is not perfectly on the free edge.
If the connector is on the free edge, the edge quad normal option will be automatically used.

positive side/negative side

The positive and negative side can be determined as long as the links are not perfectly perpendicular to each other.

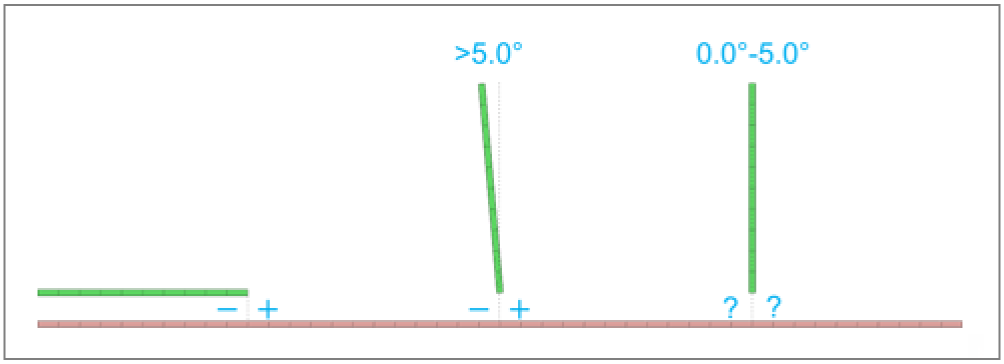


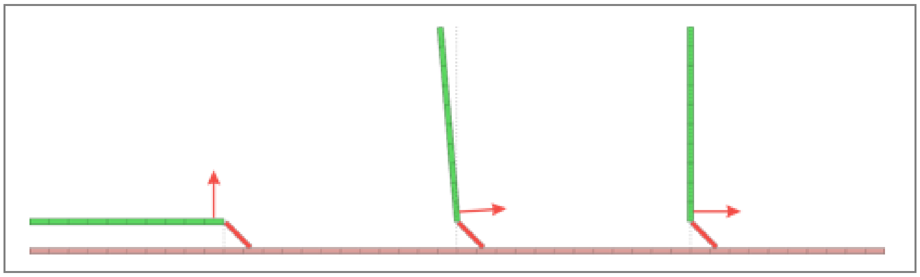
Figure 1187:

Overview of how the positive and negative side is determined. When links are perfectly perpendicular, the edge quad normal option is automatically used.

edge quad normal

Figure 1188:

Overview of how the side for the angled weld is determined. If the normal directions are reversed, the side of the angled weld changes.



Snapping To Edge

Automatic edge snapping can be used to precisely position connectors. First, the connector snaps to, for example, the closest free edge, then the projection and FE creation starts.

The snapping distance can be defined separately for T, L and B connections.

You can choose whether to snap to:

- maximum 1 element row
- maximum 2 element rows
- no (connector does not snap)

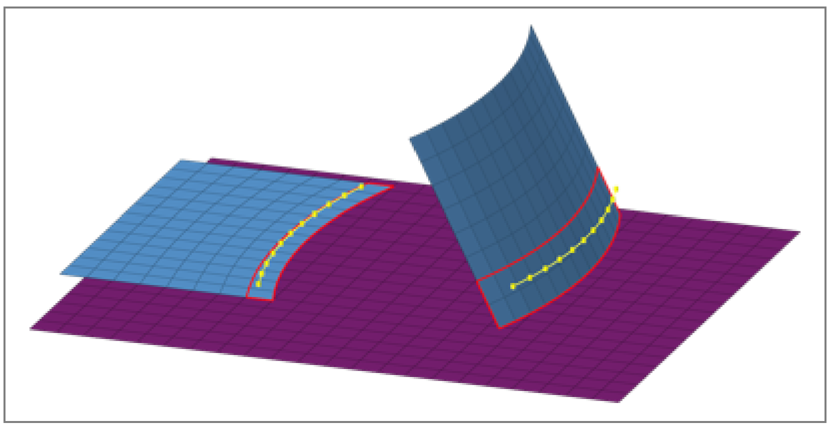


Figure 1189: Original Model before Realization *Initial situation with one element row marked for the lap weld and two element rows for the t weld.*

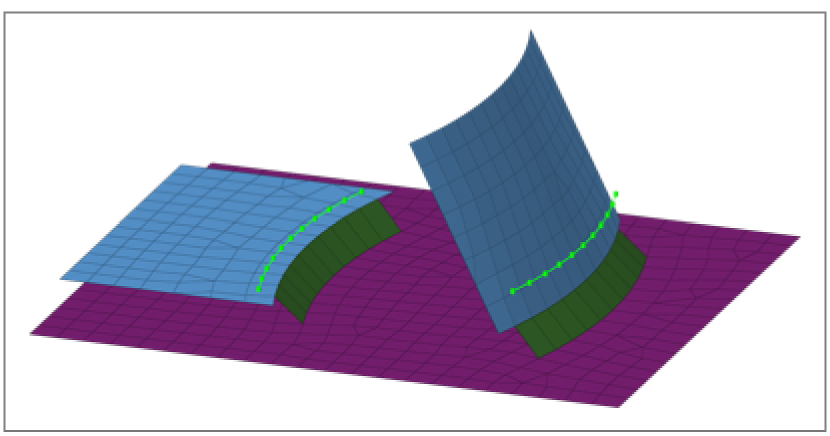


Figure 1190: Edge Snapping Enabled

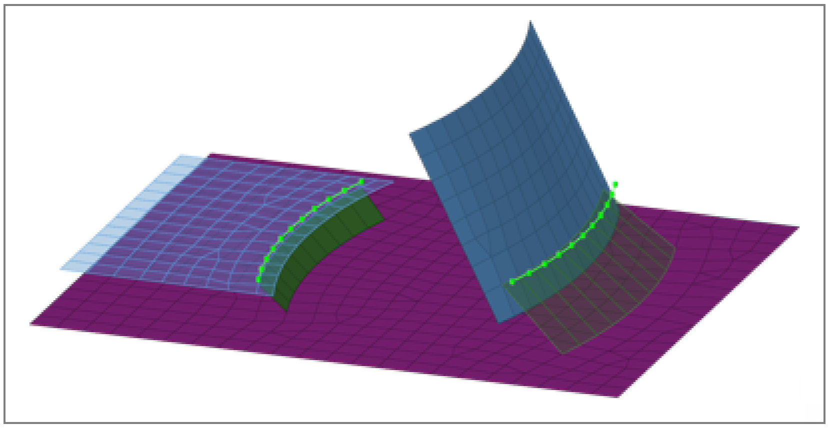


Figure 1191: Edge Snapping Disabled

Edge Treatment (T/B)

The Edge Treatment(T/B) setting attempts to create specific vertical lengths for T connections LTV, and specific lengths for B connections LB.

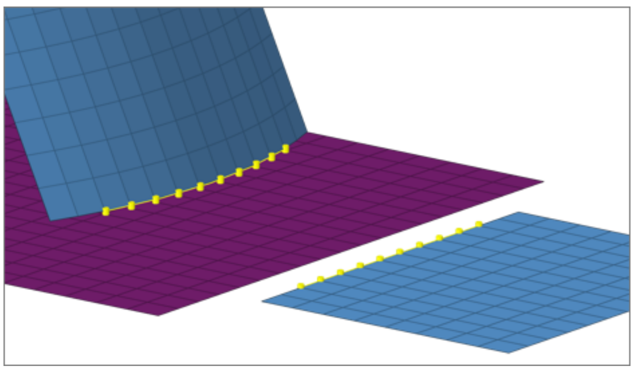


Figure 1192: Original Model before Realization

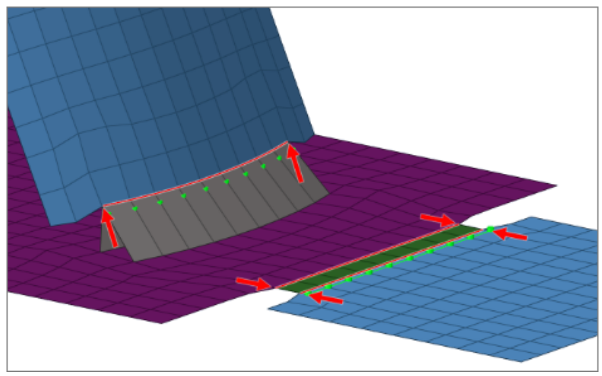


Figure 1193: Realization using Edge Treatment

Free edges were contracted or extended.

Edge Treatment Options

Choose whether to:

extension and contraction
extension
contraction

Edge Treatment Limit

Edge treatment is a pure node movement; therefore, the maximum movement needs to be limited to prevent the elements at the edge from being destroyed. Movement is limited to a maximum of 0.5 times the element size at the edge. 0.5 is the maximum allowed value and default value.

Preserve Washer

Controls how washers are preserved during the seam imprint realization.

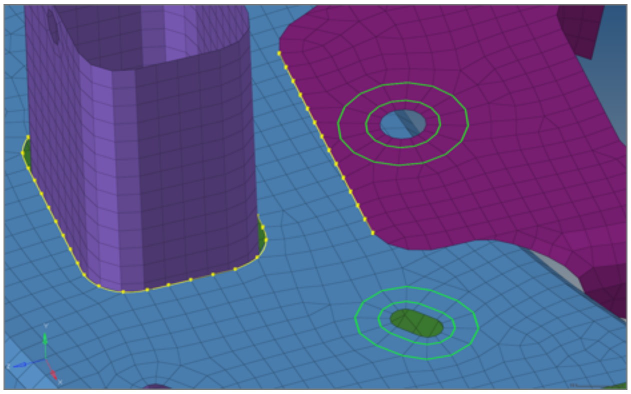


Figure 1194: Original Mode with Perfectly Meshed Washers

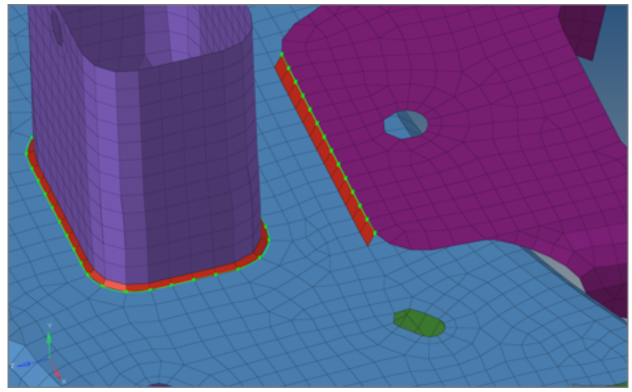


Figure 1195: No Washer Preservation Enabled

Washers have been opened.

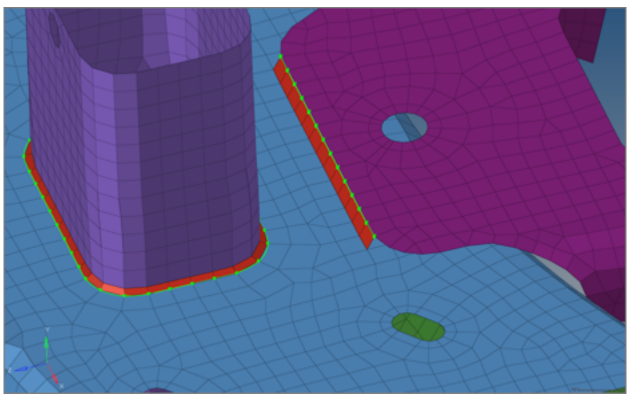


Figure 1196: Washer Preservation and Remesh Enabled

Washers are still intact, but the mesh seeding has been modified.

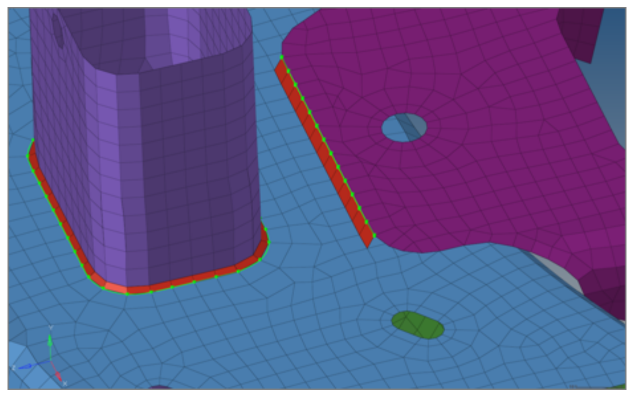


Figure 1197: Washer Preservation and No Remesh Enabled

The washers have been fully preserved.

Don't Share Zone Elements

Seam imprint allows heat affected zones (HAZ) to be merged in close areas. In this situation, one element might touch the weld elements from two different connectors. Don't share zone elements prevents zone elements from being shared.

Quad in Corner

Controls whether a single or double element is created in corners of quad seam connectors with a certain vertex.
A angle must be defined for a single quad corner. If the corner angle is greater than the defined angle, a double quad corner is created.

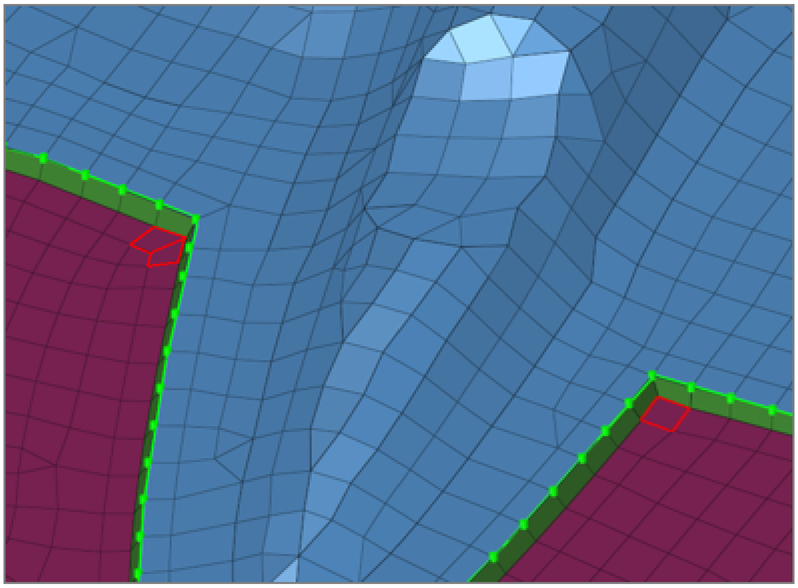


Figure 1198: Quad in Corner

A double quad corner is shown on the right, and a single quad corner is shown on the left.

Quad Control

Controls the maximum deviation from the perfect quad element for the heat affected zone (HAZ). It can be controlled, if the element size or the element skew is more important to retain.

Max Quadsize Reduction In % / Max Quad Skew In Degrees

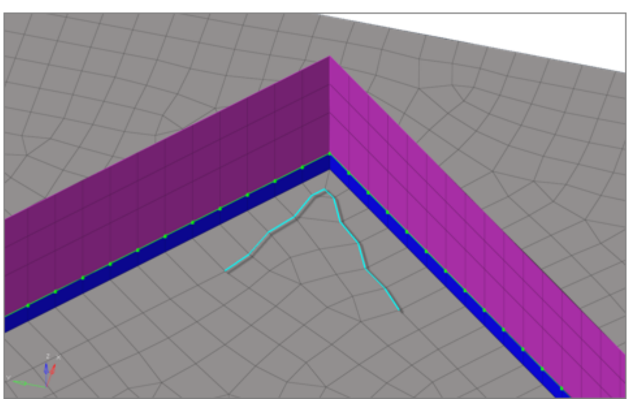


Figure 1199: Max Quad Size Reduction: 80.0 / Max Quad Skew: 5.0

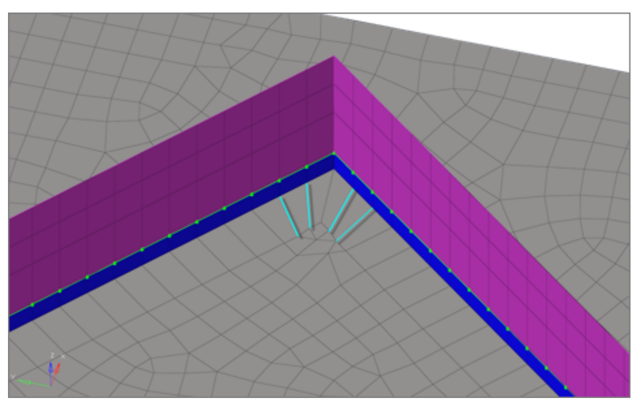


Figure 1200: Max Quad Size Reduction: 5.0 / Max Quad Skew: 45.0

Sliver Elements

Sliver elements are small elements that you may not want in your model. In the images below, a perfect perpendicular projection resulted in sliver elements. The Sliver Elements setting can be used to manage sliver elements in your model. In the images below, the red elements represent the HAZ elements.

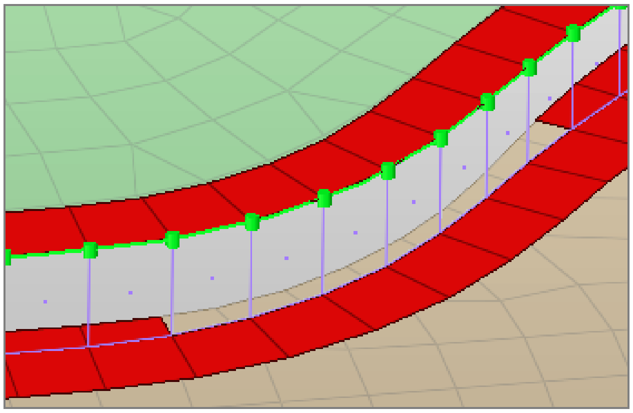


Figure 1201: Allow

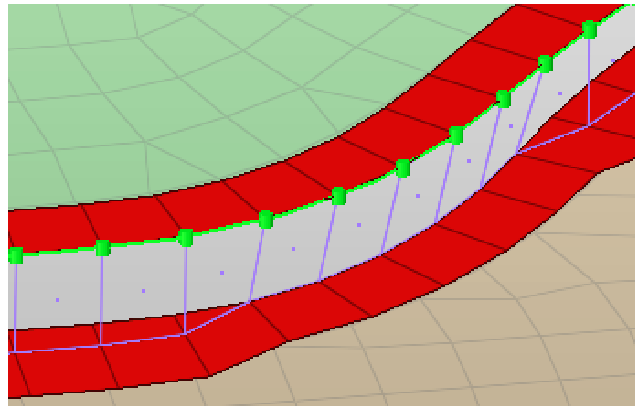


Figure 1202: Prevent by Moving Projection Points

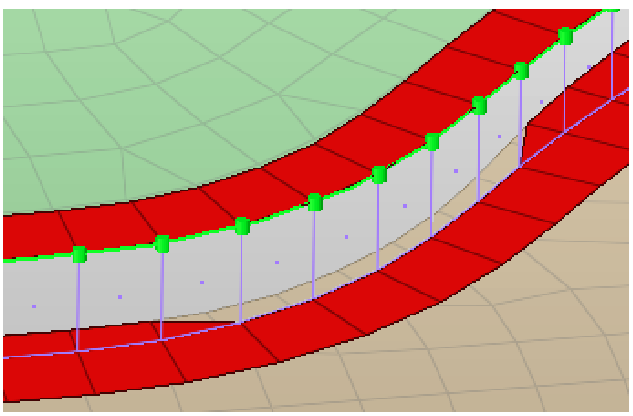


Figure 1203: Prevent by Moving Edge

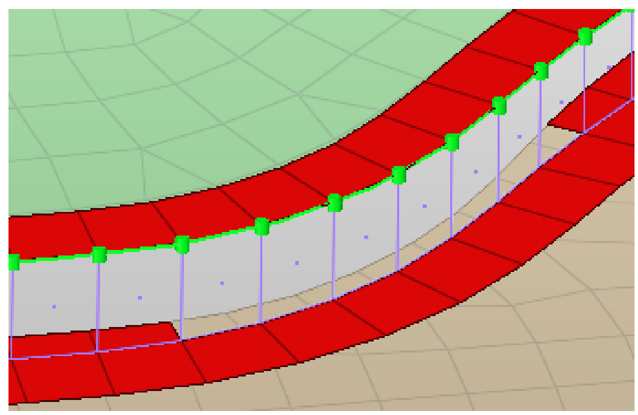


Figure 1204: Delete Sliver Elements

Element length<

This length controls which elements to treat as sliver elements.

Feature Angle

Determines important features to retain during the imprint. Features that cross the HAZ, as well as near by features cannot be retained.

Projection Control Methods for Area Connectors

Use the projection control methods to control how projection should be performed for area connectors.

You can define the projection control method in the Connector Entity Editor. With these controls you can easily realize adhesive beads.

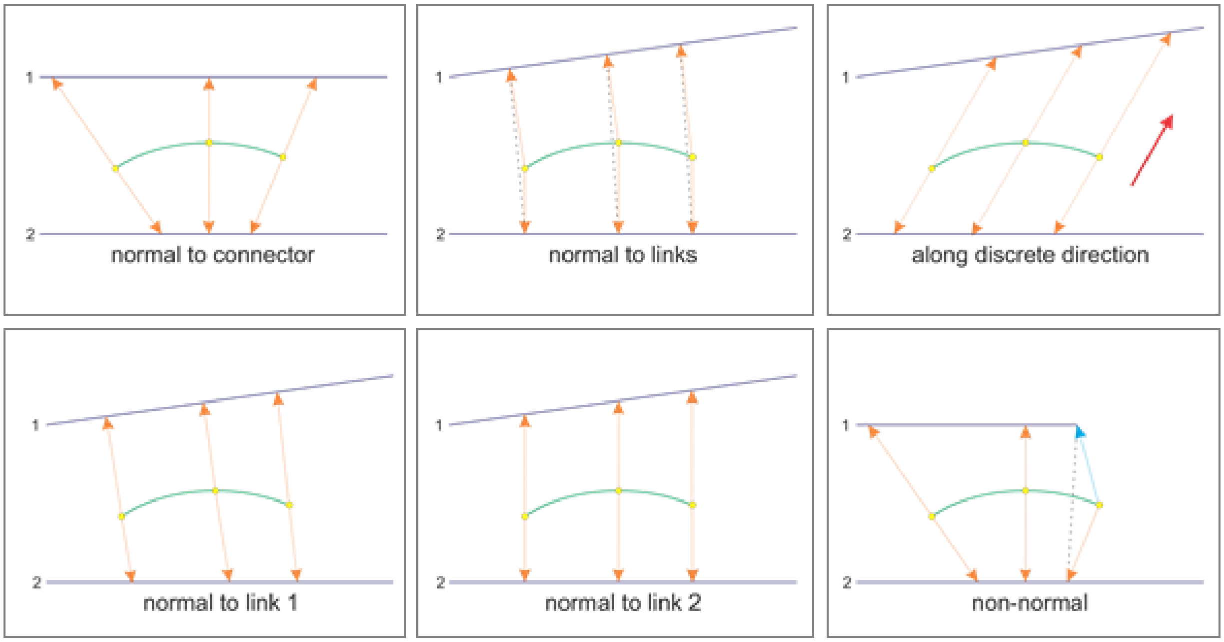


Figure 1205:

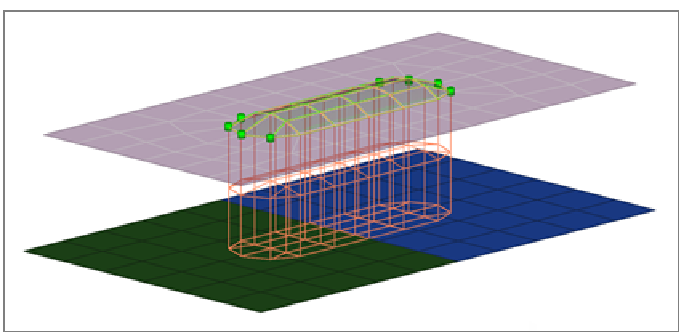


Figure 1206:

Connector Review

There are many advantages to the way connectors store information. Not only does this local storage allow you to edit the connector definition, it also allows you to review connector details and the quality of the realization.

There are a number of tools that can be useful in the review process. Use the connector visualization controls to update the visual appearance of a connector based on its state, thickness (number of layers), style (connector types), or the component in which it is located. In addition, you can use the visualization controls to filter the displayed connectors by various criteria, such as thickness. This filter can then be used store the "displayed" connectors for use in other functions.

The Connector Browser contains a list of all of connectors and their definitions, as well as a list of connector links. Use the Quality panel to check the quality of welds created from the connectors. The connector database can also be queried through Tcl functions.

Connectors User Control Mode

Each individual connector can be placed in a user control mode using either the `*CE_SetSpecificDetailById` or `*CE_SetSpecificDetail` commands.

This user control mode is most useful for automated Tcl scripts. Once in user control mode, the following procedures are possible for a given connector:

- Pre-existing FE can be registered as a given connector's realization by using the `*CE_FE_Register` command.
- Connectors can be edited without automatically unrealizing, as happens most notably when a link is added or removed from a connector, or when an FE realization entity is deleted.
- A connector's state can be manually changed from realized to failed, or from failed to realized by using either the `*CE_SetSpecificDetailById` or `*CE_SetSpecificDetail` commands. A connector's state will not change to or from the unrealized state using this method.

Once a connector is placed into your control mode, your control mode remains active until an unrealize command is called, such as `*CE_Unrealize`, an already realized connector is re-realized, or your control mode is manually turned off with either the `*CE_SetSpecificDetailById` or `*CE_SetSpecificDetail` commands. While a given connector is in user control mode, it may not behave the same as a normal connector. Specifically, there are a number of scenarios where a user-controlled connector will not auto unrealize in response to database changes that would cause a normal connector to auto unrealize.

> **Note:** It is strongly recommended that when FE is registered to a user-controlled connector, that the connector links and other necessary details should also be set with a given connector, so that the connector can properly re-realize if you interactively request it to. At the bare minimum, connectors should know which links they are to connect.

Master Connectors File

Most of the information stored in the connector entity can be exported to a master connectors file, which contains connector entity information such as location, link entity, link entity state, link entity rules. The exported file may also contain metadata information stored in the connector.

The master connectors file contains welding information at a given location and also assists in the weld automation process. An exported master connectors file can be re-imported using the connectors reader to re-create connectors.

The master connectors file is exported in a single format. Master connectors files can have comments beginning with the characters # or $, or there can be blank lines in between. The format of the file is fixed and the order of heading definitions cannot be changed. The column information includes:

- Index (ID)
- Number of Layers
- X coordinate
- Y coordinate
- Z coordinate
- FE Config
- FE Type
- Number of Links
- Link Type
- Link ID and Link Name
- Link State
- Link Rule
- METADATA

The header at the beginning of the file specifies information about the column data.

Number of layers defines the thickness to connect at the specified location (X, Y, Z). The data between the brackets are repeated for each link entity.

For standard FE types such as ACM and CWELD, the FE Config will have a number of 1001, which defines your defined type number specified in FE Config File. The FE Type will be the number defined in the FE Config File. For CWELD it is 72.

The data between the brackets (link entity information) in the table are repeated for the number of links (NumLinks). The NumLinks variable must be equal to the number of link entities.

Metadata is an attribute type that can be stored on a entity. User-defined information, such as Station ID or Gun ID, can be stored on the connector entity as Metadata. The Metadata is defined by a name-value pair and is supported for multiple data types, such as int, double and string. The Metadata name is written to the master connectors file in the following format ~<Struct><DataType>Name. <Struct> represents whether the value associated is a single variable or an array. <DataType> represents the type of data stored in the value. For example, a Metadata of name Assembly containing an array of integers is written out as ~AIAssembly.

The only delimiter supported in the entire file is the double semicolon "::".

The entire column of data in the file should be of the same type.

The connectors reader uses the Templex template to read the master connectors file.

By default, the file is read through the HMIN function call, HMIN_CE_CreateDefined.

The connector entity is created with the information specified in the master connectors file and displayed as unrealized (yellow).

Multiple Weld File Format

In addition to the master connectors file, the connectors reader also supports master weld file formats previously supported by the spotweld reader.

Spotweld Interface

The spotweld feinput translator reads weld information from an ASCII file, such as a Master Weld File.

HyperMesh supports multiple formats for master weld files through weld templates. The weld template is specific for a given format. The spotweld translator registers the template through a spotweld configuration file. The currently supported master weld file format templates and the configuration file can be found in the spotweld_format directory. In order to use a different format you must create a weld template and add its name and path to the configuration file. Existing weld templates can be copied and modified to support the new format.

The outline of a generic master weld file is:

- Point ID
- Layer information (the number of thickness it connects 2T/3T/4T. Max layers supported 4T)
- Spotweld location (X, Y, Z)
- Connector part IDs (HyperMesh Component/Part IDs)
- The delimiter between fields can be ":", ",", " ".
- ASCII files can have comments beginning with the characters # or $, or there can be blank lines in between.

The spotweld translator reads information from the master weld file and stores it in the database. At each of the weld locations, an HM_POINT is created.

Example: Weld Template and Master Weld File

The example below helps create a weld template for specific formats.

```
#Weld format 1.
#Point Id::  T::         X::         Y::         Z::         PID1::        PID2::
      PID3::
        1::      2::        2.000::        3.000::  4.000: :         12::
  14::        20::
```

Figure 1207: Master Weld File

```
int num
header
{
    type "SPOTWELDS"
    set mark
    find "[0-9]+::"
    rewind
    set num = 0
    if
    {
        do 1000000
        {
            if
            {
                isdigit
            }
            then
            {
```

```
                set num = sum(num, 1)
            }
            readln null
        }
    }
    set numrecords = num
    set numrequests = 9
    requests "ID/T/X/Y/Z/PID1/PID2/PID3/PID4"
    set numcomponents = 1
    components "Value"
}
record
 if
    {
        do 1000000
        {
            if
            {
                isdigit
            }
            then
         isalpha
            }
     readln null
        }
    }
    read request // ID
    qfind "::"
 set mark
    read request  //T
 rewind
 read num
    qfind "::"
    read request // X
    qfind "::"
    read request // Y
    qfind "::"
    read request // Z
    do num
    {
        qfind "::"
        read request // PID
    }
    set num = diff(4, num)
    do num
    {
        read constant 0 // fake PID
    }
    readln null
}
```

Figure 1208: Weld Template

FE Configuration File

The FE configuration file (`feconfig.cfg`) is used to define custom welds such as ACM (Area Contact Method) and other special types.

Weld definitions are solver dependant (Nastran, LS-DYNA, and so on). The weld definition in the file includes the type of weld to create and the surrounding connector to shells. The specific solver template for the type of weld must be loaded before the welds can be created using a connector entity. When HyperWorks Desktop or HyperMesh launches, it searches for `feconfig.cfg` in the following locations, listed in search order:

1. The product installation path
2. HW_CONFIG_PATH (a specified environment variable)
3. Your home directory (in UNIX, for example)
4. The current working directory.

By default, the `feconfig.cfg` file from the `<install_directory>/hm/bin` directory is loaded in each of the panels related to each connector type, such as Spot, Seam and Area. It is not recommended to have more than one config file of the same name, even in different directories, as the results can be unpredictable as to which one will be used by default.

Weld Definition Template

```
CFG <SOLVER> <USER_FE_TYPE> <USER_FE_NAME>
*filter <FILTER_TYPE > [<FILTER_TYPE>]
*style < STYLE_TYPE> <STYLE_NUM>
*head
<HM_FE_CONFIG> <HM_FE_TYPE> <RIGID_FLAG>
*body <BODY_FLAG>
<HM_FE_CONFIG> <HM_FE_TYPE> <LENGTH_LOCATION_FLAG> [<DOFS>]
[<HM_FE_CONFIG> <HM_FE_TYPE> <LENGTH_LOCATION_FLAG> [<DOFS>]]
*post <POST_SCRIPT_NAME>
```

Figure 1209: Weld Definition Template

Where,

CFG

Keyword to start a custom weld definition.

SOLVER

The solver template for which FE needs to be created.
Supported solvers are: Abaqus, ANSYS, LS-DYNA, Nastran, OptiStruct, PAM-CRASH, or PAM-CRASH 2G.

USER_FE_TYPE

A unique (with respect to a solver) user defined configuration type ID. Customer-defined CFGs should use numbers greater than 10,000 to ensure no collisions with future native HM CFGs.

USER_FE_NAME

The user-specified name for the FE configuration. The specified name is saved and displayed in the Connector Browser.

> **Note:** This should be the first line in the custom weld definition.

***filter**

Allows only the specified connector types to realize the configuration.
For example, *FILTER spot seam indicates that this configuration can be realized only by the spot and seam connector types. In addition, this option is used as a filter when displaying FE configurations in the type = field of respective realize panels.
*filter lines also set which panel the CFG is visible in.

CE_TYPE

The connector type, spot, bolt, seam, area, and so on.

***style**

Indicates that the configurations have specific behaviors associated during realization, and that they are native types.

> **Note:** The style definition line for these configurations must not be edited.

For example, *style bolt 1 indicates that this is a bolt connection of type 1 that creates a specific bolted connection between the parts.

STYLE_TYPE

The connector style name, such as "adhesive", "bolt", "acm", "quad", "continuous", and so on.

STYLE_NUM

The connector style number:

- Adhesive:

  "1" mesh independent adhesive nodes tie to shells with RBE3/RBE2.

  "2" force shell gap length on. Adhesive (HEXA element) shares nodes with shell at co-incident locations.

- Bolt:

  "0" normal bolt: "wagon wheels" in the holes.

  "1" symmetrical spider bolt.

  "2" unsymmetrical spider bolt: the middle node is biased towards one hole.

  "3" cylinder bolt: ties together all nodes within virtual cylinder.

- ACM:

  "1" share the nodes of HEXA element for consecutive layers (> 2T) and the length of HEXA is average of part thickness.

  "2" HEXA elements in consecutive layers have unique nodes and the length of HEXA is average of part thickness.

"3" share the nodes of HEXA elements for consecutive layers and the length of HEXA is the gap distance between parts.

"5" HEXA nodes are not equivalenced with the shell nodes/washer nodes, if a washer is present.

- Quad

  "1" create two sets of QUAD4 elements, first along projection direction and second at an orientation determined by average part thickness.

  "2" create one set of QUAD4 elements at an orientation determined by average part thickness.

***head**

The string head is required to specify that a rigid is to be created to connect the weld node to the surrounding shell element.
*head lines must be followed with at most one HM_FE_CONFIG line.

HM_FE_CONFIG

The config for the rigid currently supported.
The various types supported are "bar2", "bar3", "equations", "gap", "hex8"(3D), "plot", "mass"(0D), "rigid", "rigidlink", "rbe3", "rod", "spring", "weld", "quad4"(2D seam only), or "penta6"(3D adhesive only).

HM_FE_TYPE

Unique (with respect to a solver) user defined configuration type id defined in the solver template.

RIGID_FLAG

Defines the number and arrangement of rigids.

"0" is a single rigid

"1" is multiple rigids

"2" is multiple rigids to outer shell nodes (for 2D bolt washers only)

"3" is multiple rigids to outer alternate shell nodes (for 2D bolt washers only)

"10" is multiple rigids with a 0 length leg connecting with body (for bolt only)

"12" is multiple rigids to inner and outer shell nodes (for 2D bolt washers only)

"13" is multiple rigids to inner and outer alternate shell nodes (for 2D bolt washers only)

DOFS (Optional)

Degrees of freedom of the rigid (1-6).

***body**

Specifies that a weld is to be created to connect the link entities added to the connector.
*body lines may be followed by one or more HM_FE_CONFIG lines

BODY_FLAG

The body flag is used to calculate the length of the weld. If the body flag = 0, the length is calculated based on the distance between the connecting layers (link entities). If the body flag = 1, the length is calculated based on the average thickness of the connecting layers (link entities).

HM_FE_CONFIG

The config for the rigid currently supported. The various types supported are "bar2", "bar3", "equations", "gap", "hex8"(3D), "plot", "mass"(0D), "rigid", "rigidlink", "rbe3", "rod", "spring", "weld", "quad4"(2D seam only), or "penta6"(3D adhesive only).

HM_FE_TYPE

The solver defined type for the HyperMesh config. For example, CBUSH is of config spring and type 6. The type number is defined in respective solver templates and differs, based on the solver.

LENGTH_LOCATION_FLAG

0D element details.

"0" place the 0D element along the proposed 1D element path. If this 0D element is the only config given in the *body, then it is placed at the center of the proposed 1D element path.

"1" has the same behavior as "0" except only a single 0D element is created even if multiple bodies are created (as happens in >2T welds)

"2" place the 0D element at the connector location.

1D element details.

"0" force zero length welds.

>"0" but <"1" denote a percentage of the distance between shells the length of a given weld should be. To create series welds, all the PERCENT_LENGTH_FLAG variables for a given *body must add up less than or equal to 1.0. Example: 0.33 or 0.50

"1" force each body weld to have a length equal to the distance between the shells, which can be used for parallel welds.

"2" place the 1D element at the connector location, with both nodes coincident.

"3" place multiple (thickness-1) 1D elements at the connector location connected end-to-end, with all nodes coincident.

3D element details.

"0" force a floating hexa element to have a length equal to half the distance specified by the BODY_FLAG.

"1" force the hexa element's length to be equal to the full distance specified by the BODY_FLAG.

DOFS (Optional)

The degrees of freedom (1-6) of the rigid.

***post**

*post lines are optional, but if specified, they must be followed by the name, excluding path, of a valid TCL script with a `.tcl` extension. HyperMesh searches for the TLC script in the locations and order specified below:

1. Current working directory
2. Users home directory
3. Paths in HW_CONFIG_PATH environment variable
4. Installation directory

5. `hm/bin` directory
6. `hm/scripts` directory
7. `hm/scripts/connectors/` directory

This post script will be automatically executed post FE realization, and can be used to edit weld properties, attributes, and other solver specific details.

FE Specification Rules

Each solver will have a specific definition so the same user-defined types can be repeated for each solver.

The head and the body definition must begin with a "*" to define rigid and weld definitions.

Multiple solid element combinations are not currently supported. Therefore, an ACM can have only one hexa weld element specified in the definition.

1D and 3D element combinations are not supported.

The total length of series welds cannot exceed 1.0 (100 percent). Hence there cannot be three welds specified in series having a length factor of 0.5 (50 percent) each.

Series and parallel weld element combinations are not supported.

Series welds are not supported where the link entities are coincident. Series welds are not created when the distance between the connecting link entities is zero.

User comments should start with a hash character "#".

FE Configuration Examples

FE configuration examples for washers, ACM welds, series welds, parallel welds, and 0D welds.

Washers

```
CFG nastran 56 bolts
*filter bolt
*style bolt 0
*head
rigidlink 1 1 dofs=123
rigidlink 1 3
*body 0
rigid 1 1 dofs=456
```

ACM Welds

ACMs with HEXA8 solid elements as welds and RBE3 elements as rigids are created. The length of the hexa is equal to the distance between the connecting shell elements.

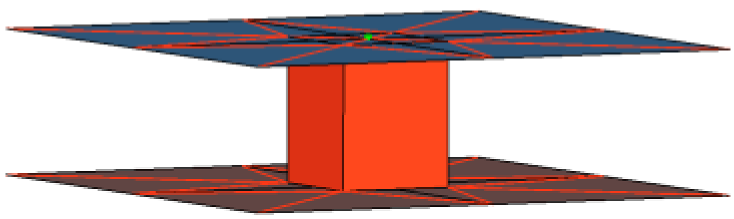


Figure 1210:

```
CFG nastran 71 acm
*head
rbe3 0 0
*body 1
hex8 1 1
```

Series Welds

Two series welds are created with a length equal to half the distance between the link entities.

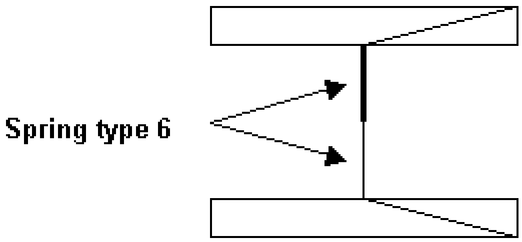


Figure 1211:

```
CFG nastran 101 series
*head
plot 0 0
*body 0
spring 6 0.5
spring 6 0.5
```

Series Welds

The series weld is created at the center with length equal to half the distance between the link entities.

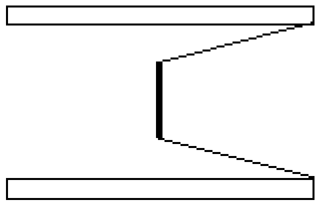


Figure 1212:

```
CFG nastran 101 series
*head
plot 0 0
*body 0
spring 6 0.5
```

Parallel Welds

Bar elements are created at the same location and connect the same link entities.

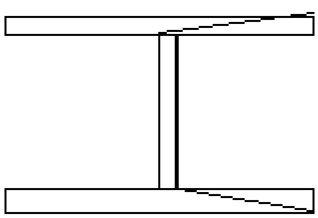


Figure 1213:

```
CFG dyna 101 parallel
*head
plot 0 0
*body 0
bar2 1 1
bar2 1 1
```

0-D Welds

Supported values for the length location flag are "0", "1", or "2". The behavior for each value is as follows, "0" places the 0D element along the proposed 1D element path. If this 0D element is the only config given in the *body, then it is placed at the center of the proposed 1D element path. "1" has the same behavior as "0" except only a single 0D element is created even if multiple bodies are created (as happens in >2T welds) and "2" places the 0D element at the connector location.

```
CFG pamcrash2g 1 plink (ce loc)
*head
plot 0 0
*body 0
mass 5 2
plot 0 1
```

```
*post prop_plink.tcl
```

Abaqus Connector Types

Supported Abaqus connector types and property scripts.

Connector Types

Abaqus Fastener

Creates a CONN3D2 element.

This realization uses the `prop_fastener.tcl`[5] property script.

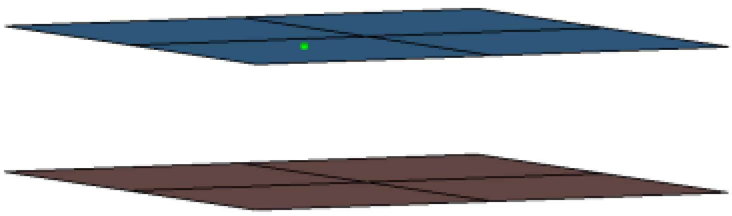


Figure 1214:

```
CFG abaqus 3 fastener
*filter spot
*head
*body 0
rod 13 3
*post prop_fastener.tcl
```

Abaqus acm (equivalenced-(T1+T2)/2)

Creates hexa element with DCOUP3D elements projecting and connecting to the surrounding shell elements. This realization uses the shell thickness to calculate the hexa offset from the shell elements. In the case where the model is a 3T connection, the acm (equivalenced-(T1+T2)/2) realization will join the hexa elements.

This realization uses the `prop_abaqus_acm.tcl`[4] property script.

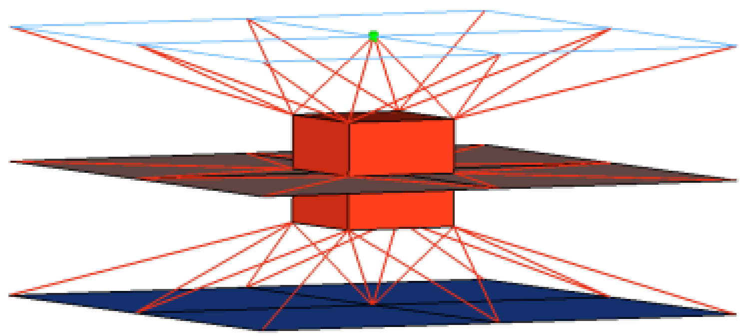


Figure 1215:

```
CFG abaqus 4 acm (equivalenced-(T1+T2)/2)
*filter spot
*style acm 1
*head
rbe3 1 0
*body 0
hex8 1 1
*post prop_abaqus_acm.tcl
cfg_abaqus_4_acm
```

Abaqus sealing

Creates DCOUP3D elements for the head and element for the body. The head elements project and connect to the nodes of the adjoining shell elements.

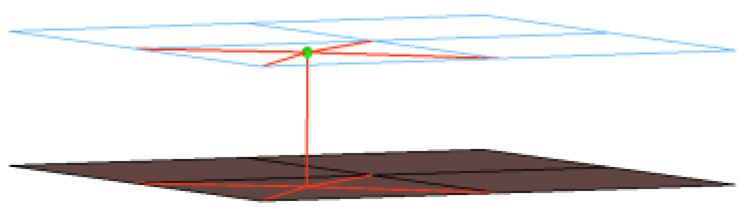


Figure 1216:

```
CFG abaqus 5 sealing
*filter spot
*head
rbe3 1 0
*body 0
rod 13 1
```

Abaqus bush

Creates KINCOUP elements for the head and element for the body. The head elements project and connect to the nodes of the adjoining shell elements.

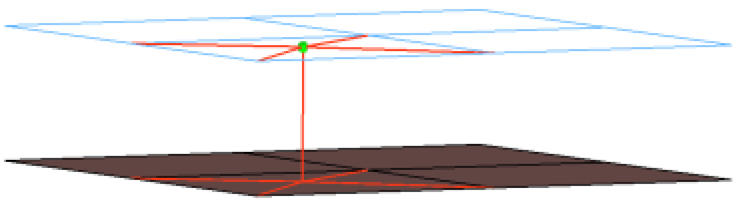


Figure 1217:

```
CFG abaqus 6 bush
*filter spot
*head
rigidlink 1 1
*body 0
rod 13 1
```

Abaqus bolt (b31)

Creates KINCOUP elements for the head and B31 element for the body. The head elements project and connect to the nodes of the adjoining shell elements that form the hole, and also to the second row of nodes to form the washer layer. The connector location can be on the edge of the hole, center of the hole, midpoint in between the two, holes or on the second row of nodes which form the washer layer.

This connector also uses the script `prop_abaqus_b31.tcl`[3].

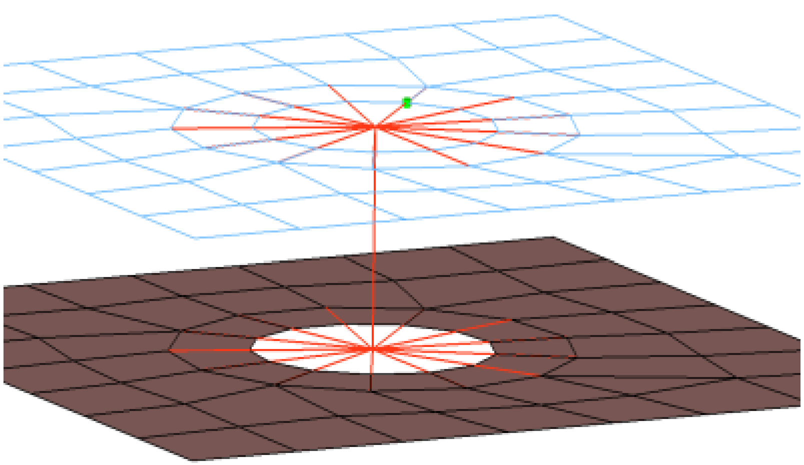


Figure 1218:

```
CFG abaqus 7 bolt (b31)
*filter bolt
*style bolt 0
*head
```

```
rigidlink 1 12
*body 0
bar2 9 1
*post prop_abaqus_b31.tcl
cfg_abaqus_7_bolt
```

Abaqus hinge (b31)

Creates KINCOUP elements for the head and B31 element for the body. The rot x degree of freedom is constrained. The head elements project and connect to the nodes of the adjoining shell elements that form the hole, and also to the second row of nodes to form the washer layer. The connector location can be on the edge of the hole, center of the hole, midpoint in between the two holes, or on the second row of nodes which form the washer layer.

This connector also uses the script `prop_abaqus_b31.tcl`[3].

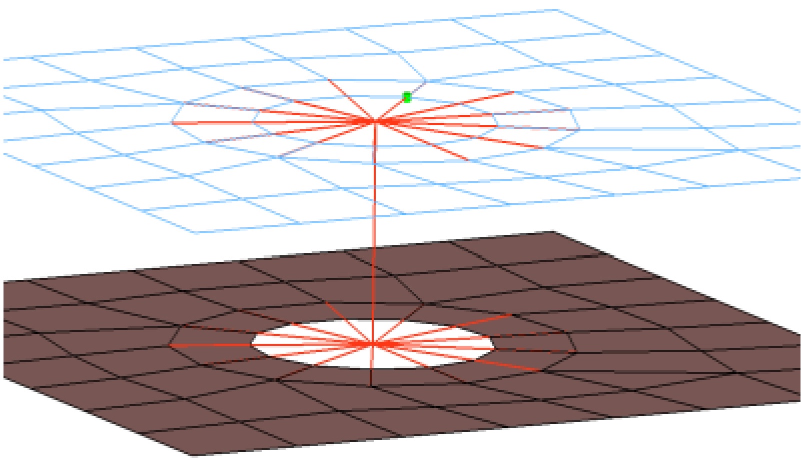


Figure 1219:

```
CFG abaqus 8 hinge (b31)
*filter bolt
*style bolt 0
*head
rigidlink 1 12
*body 0
bar2 9 1 dofs=4
*post prop_abaqus_b31.tcl
```

Abaqus Adhesives

Creates a row of hexa/penta elements for the body and numerous DCOUP3D/KINCOUP elements for the head. The head elements project and connect to the nodes of the adjoining shell elements. If there is a direct normal projection then a KINCOUP element will be used, if there are only non-normal projections then DCOUP3D elements will be created. The size (thickness) for the hexa and/or penta elements depends on the chosen option: shell gap, (T1+T2)/2, mid thickness, const. thickness, maintain gap.

> **Note:** The exact hexa position is also influenced by the option consider shell thickness and offset for hexa positioning. See hexa positioning for hexa adhesives and ACMs for details.

This realization uses the `prop_abaqus_acm.tcl`[4] property script.

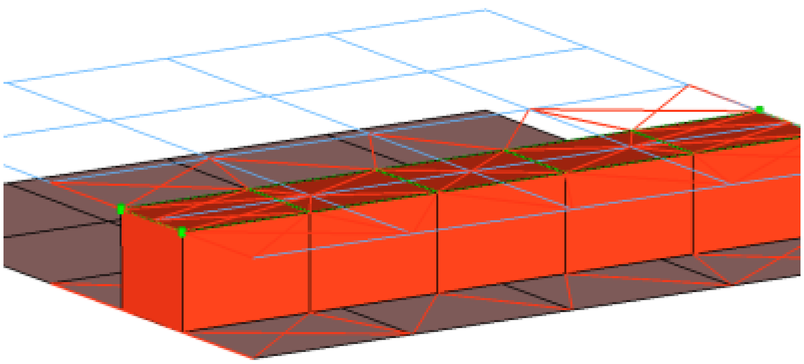


Figure 1220:

```
CFG abaqus 9 adhesives
*filter area
*style adhesive 1
*head
rbe3 1 0
rigid 1 0
*body 1
hex8 1 1
penta6 1 1
*post prop_abaqus_acm.tcl
cfg_abaqus_9_adhesive
```

Abaqus rbe3 (load transfer)

Creates DCOUP3D elements for the body. The degrees of freedom are constrained in the x, y, and z axes for the dependant nodes.

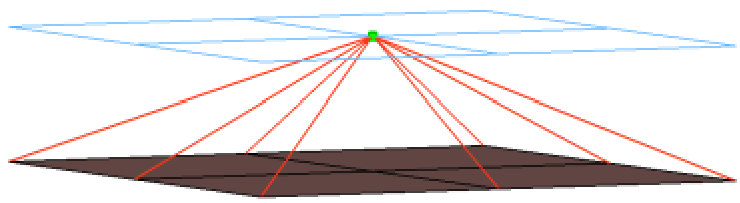


Figure 1221:

```
CFG abaqus 31 rbe3 (load transfer)
*filter spot
*style mpc 1
*head
*body 0
rbe3 1 1 dofs=123
cfg_abaqus_31_rbe3
```

Abaqus clip

Creates a KINCOUP element. The element projects and connects to the nodes of the adjoining shell elements that form the hole, and also the nodes that form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes, or on the second row of nodes which form the washer layer.

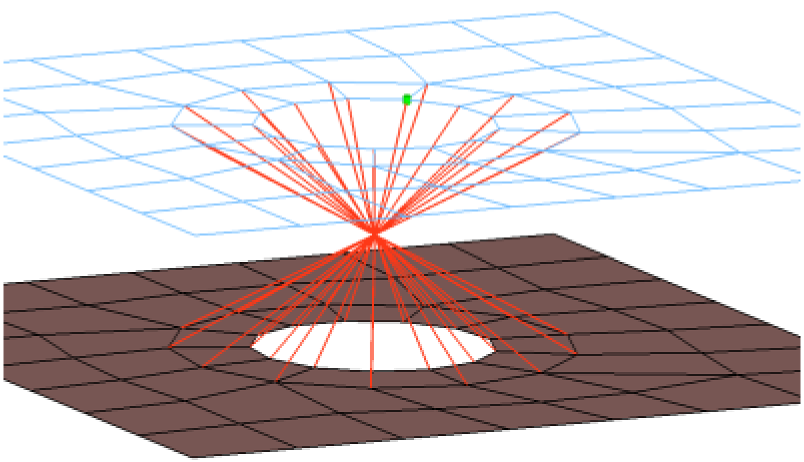


Figure 1222:

```
CFG abaqus 50 clip
*filter bolt
*style bolt 1
*head
*body 0
rigidlink 1 2
```

Abaqus bolt (washer 1) cbar

Creates KINCOUP elements for the head and B31 element for the body. The head elements project and connect to the nodes of the adjoining elements, forming the hole and also the second row of nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes, or on the second row of nodes which form the washer layer.

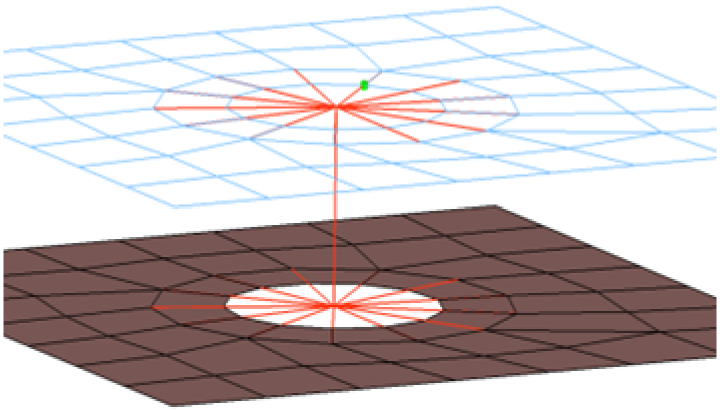


Figure 1223:

```
CFG abaqus 51 bolt (washer 1) cbar
*filter bolt
*style bolt 0
*head
rigidlink 1 12
*body 0
bar2 9 1
```

Abaqus bolt (spider)

Creates a KINCOUP element, which projects and connect to the nodes of the adjoining elements which form the hole. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes, or on the second row of nodes which form the washer layer.

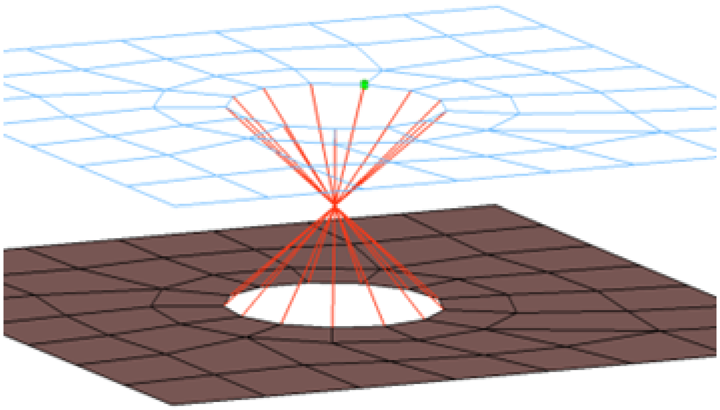


Figure 1224:

```
CFG abaqus 54 bolt (spider)
```

```
*filter bolt
*style bolt 1
*head
*body 0
rigidlink 1 1
```

Abaqus bolt (washer 2)

Creates KINCOUP elements for the head and the body. There are two individual KINCOUP elements at the head of the connection, one to connect to the inner row of nodes, and a second to connect to the washer layer nodes. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes, or on the second row of nodes which form the washer layer.

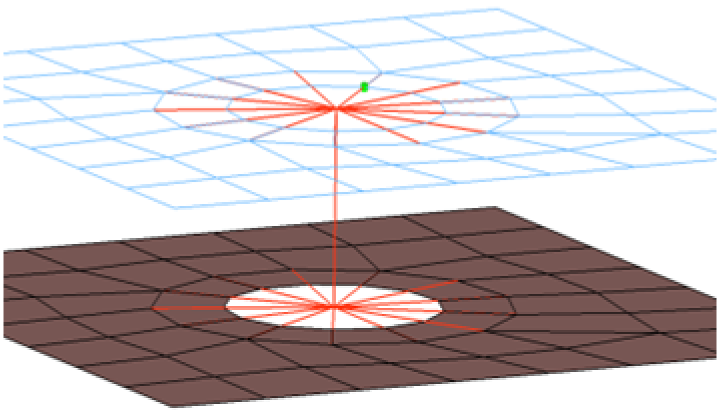


Figure 1225:

```
CFG abaqus 55 bolt (washer 2)
*filter bolt
*style bolt 0
*head
rigidlink 1 1
rigidlink 1 2
*body 0
rigid 1 1
cfg_abaqus_55_bolt_washer2
```

Abaqus bolt (washer 2 alt)

Creates KINCOUP elements for the head and the body. There are two individual KINCOUP elements at the head of the connection, one to connect to the inner row of nodes, and a second to connect to the washer layer nodes. The KINCOUP head element that connects to the washer layer nodes only connects to every other node on the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes, or on the second row of nodes which form the washer layer.

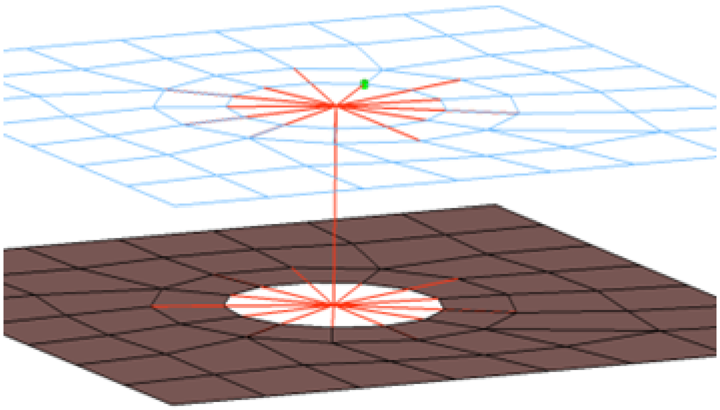


Figure 1226:

```
CFG abaqus 56 bolt (washer 2 alt)
*filter bolt
*style bolt 0
*head
rigidlink 1 1
rigidlink 1 3
*body 0
rigid 1 1
```

Abaqus bolt (washer 1)

Creates KINCOUP elements for the head and body. The head elements project and connect to the nodes of the adjoining elements, forming the hole and also the second row of nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes, or on the second row of nodes which form the washer layer.

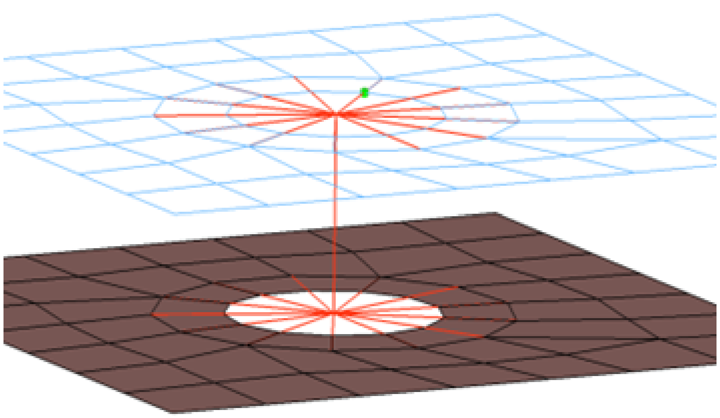


Figure 1227:

```
CFG abaqus 57 bolt (washer 1)
*filter bolt
*style bolt 0
*head
rigidlink 1 12
*body 0
rigid 1 1
```

```
cfg_abaqus_57_bolt_washer1
```

Abaqus bolt (washer 1 alt)

Creates KINCOUP elements for the head and body. The head elements project and connect to the nodes of the adjoining elements, forming the hole and also the second row of nodes which form the washer layer. The head only connects to every other node on the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes, or on the second row of nodes which form the washer layer.

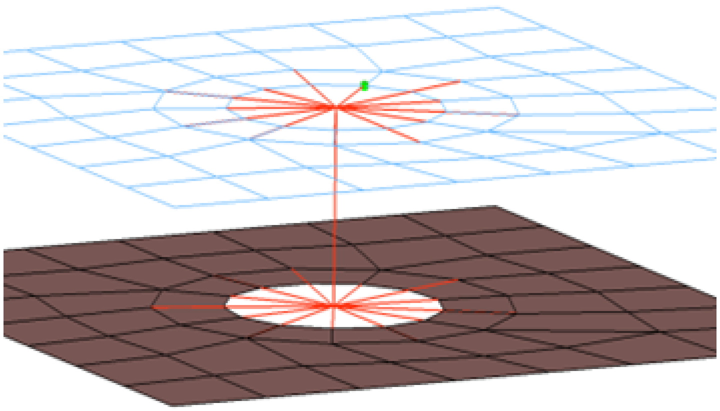


Figure 1228:

```
CFG abaqus 58 bolt (washer 1 alt)
*filter bolt
*style bolt 0
*head
rigidlink 1 13
*body 0
rigid 1 1
```

Abaqus bolt (cylinder bolt)

Creates a KINCOUP element for the body as well as for the head elements.

This realization uses the `prop_cylinder.tcl`[6] property script.

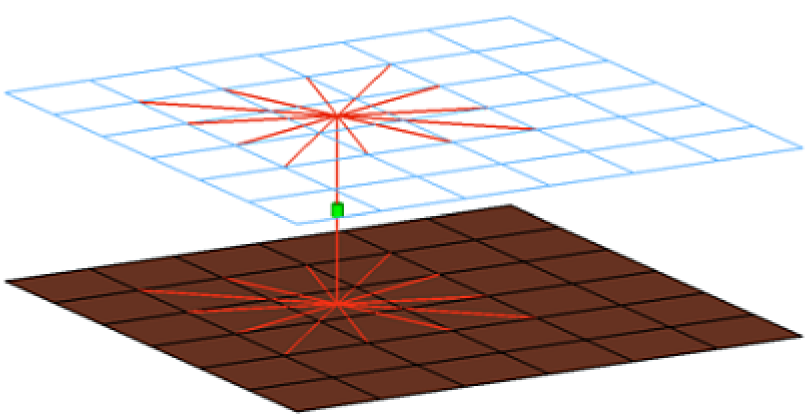


Figure 1229:

```
CFG abaqus 60 bolt (cylinder rigid)
```

```
*filter bolt
*style bolt 4
*head
rigidlink 1 1
*body 0
rigid 1 1
*post prop_cylinder.tcl
cfg_abaqus_60_cylinderbolt
```

Abaqus bolt (cylinder bar)

Creates a B31 element for the body and KINCOUP elements for the head elements.

This realization uses the `prop_cylinder.tcl`[6] property script.

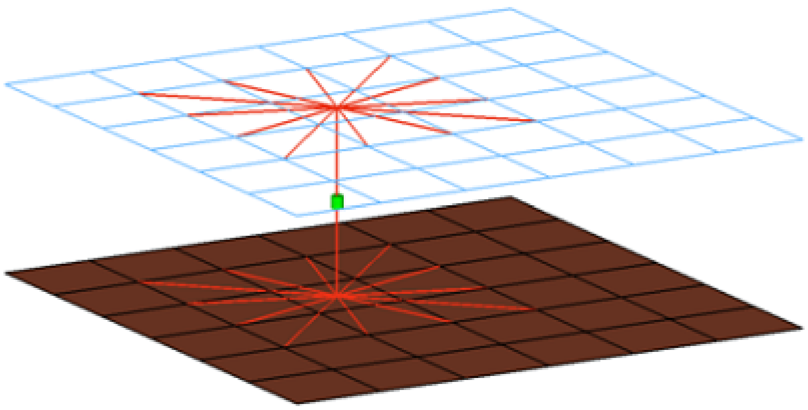


Figure 1230:

```
CFG abaqus 61 bolt (cylinder bar)
*filter bolt
*style bolt 4
*head
rigidlink 1 1
*body 0
bar2 9 1
*post prop_cylinder.tcl
cfg_abaqus_61_bolt_clinderbar
```

Abaqus acm (detached-(T1+T2)/2)

Creates a hexa element with DCOUP3D elements projecting and connecting to the surrounding shell elements. This realization uses the shell thickness to calculate the hexa offset from the shell elements. In the case where the model is a 3T connection, the acm (detached-(T1+T2)/2) realization will not join the hexa elements.

This realization uses the `prop_abaqus_acm.tcl`[4] property script.

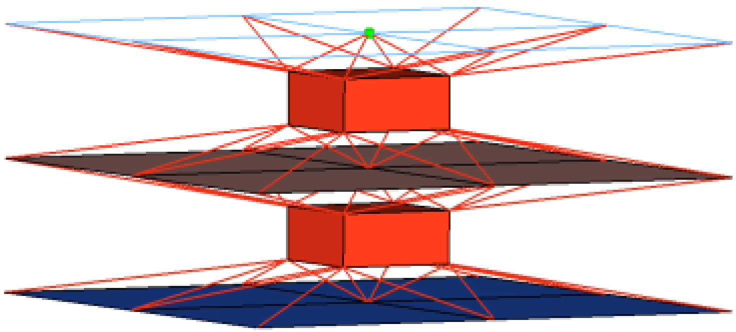


Figure 1231:

```
CFG abaqus 70 acm (detached-(T1+T2)/2)
*filter spot
*style acm 2
*head
rbe3 1 0
*body 1
hex8 1 1
*post prop_abaqus_acm.tcl
cfg_abaqus_70_acm
```

Abaqus acm (shell gap)

Creates a hexa element with DCOUP3D elements projecting and connecting to the surrounding shell elements. This realization does not use the shell thickness to calculate the hexa offset, therefore the hexa will project and be touching the shell elements.

This realization uses the `prop_abaqus_acm.tcl`[4] property script.

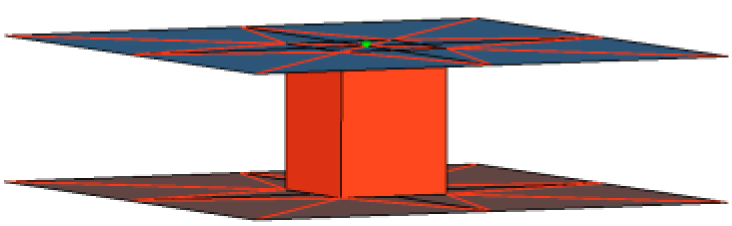


Figure 1232:

```
CFG abaqus 71 acm (shell gap)
*filter spot
*style acm 3
*head
rbe3 1 0
*body 0
```

```
hex8 1 1
*post prop_abaqus_acm.tcl
cfg_abaqus_71_acm
```

Abaqus acm (shell gap + coating)

Creates one hexa cluster per connector and realizes a node to node connection to the linked shell meshes by adjusting it (shell coating). Different patterns are available. This is driven by the number of hexas. The appearance can be influenced via the diameter and the washer layer activation.

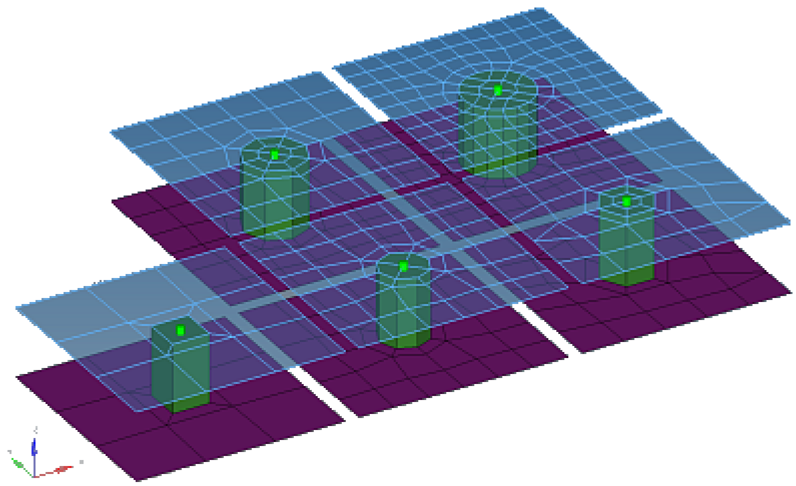


Figure 1233:

```
CFG abaqus 72 acm (shell gap + coating)
*filter spot
*style acm 4
*body 0
hex8 1 1
acm_shellgap_coating_2
```

Abaqus acm (general)

Consolidates several ACM definitions into one general, flexible ACM definition. Besides mid thickness, constant thickness, and maintain gaps, the definition of several coats with different hexa patterns is available.

This realization uses the `prop_abaqus_acm.tcl`[4] property script.

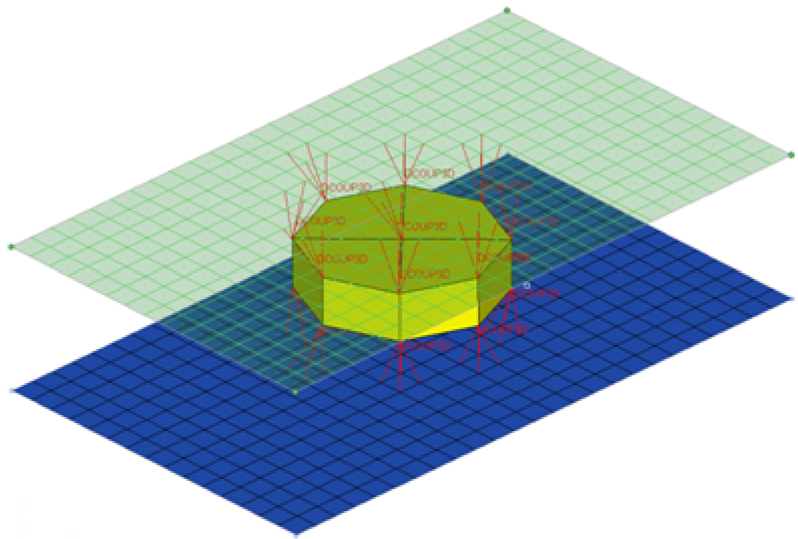


Figure 1234:

```
CFG abaqus 73 acm (general
*filter spot
*style acm 3
*head
rbe3 1 0 dofs=123
*body 0
hex8 1 1
*post prop_abaqus_acm.tcl
cfg_abaqus_73_acm_general
```

Abaqus seam-quad (angled+capped+L)

Creates a quad row with tria caps at the seam ends. In addition, a certain pure quad element pattern is created around the seam elements, shown here in red. These elements normally get imprinted into the shell links. The exact geometry of the seam can be influenced by certain attributes in the Seam panel.

This realization is mainly intended to be used for lap welds.

> **Note:** You can revert the direction of quad seam connectors during the next realization by activating the reverse direction checkbox in the Seam panel.

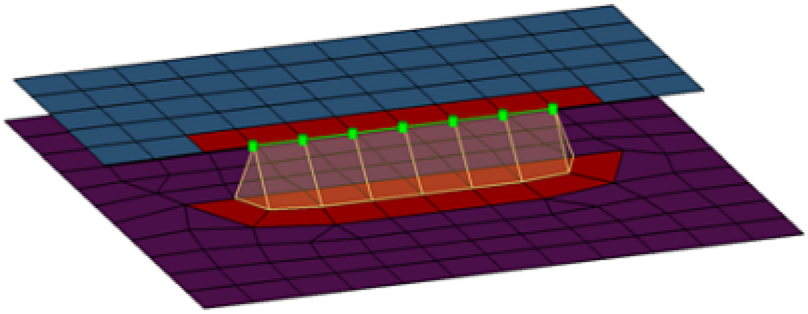


Figure 1235:

```
CFG abaqus 101 seam-quad (angled+capped+L)
*filter seam
*style quad 4
*head
*body 0
quad4 1 1
```

Abaqus seam-quad (angled+capped+T)

Creates a quad row with tria caps at the seam ends. In addition, a certain pure quad element pattern is created around the seam elements, shown here in red. These elements normally get imprinted into the shell links. The exact geometry of the seam can be influenced by certain attributes in the Seam panel.

This realization is mainly intended to be used for lap welds.

> **Note:** You can revert the direction of quad seam connectors during the next realization by activating the reverse direction checkbox in the Seam panel.

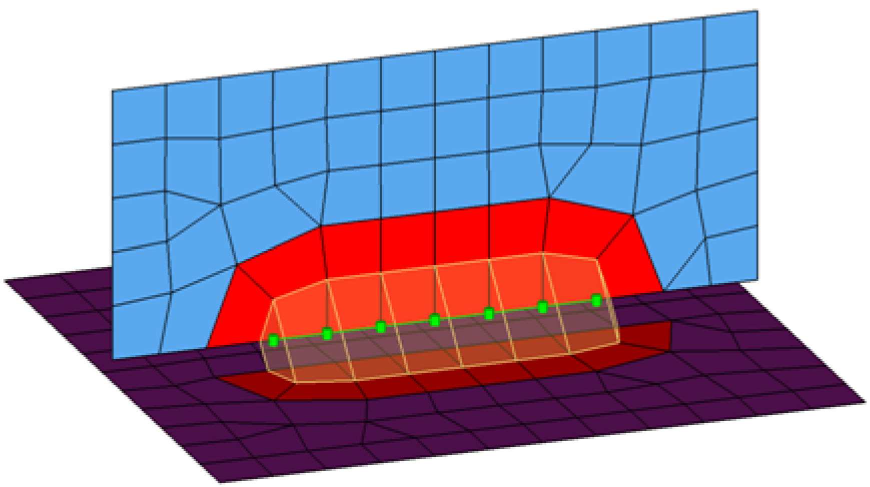


Figure 1236:

```
CFG abaqus 102 seam-quad (angled+capped+T)
*filter seam
*style quad 5
*head
*body 0
```

```
quad4 1 1
```

Abaqus seam-quad (vertical+angled)

Description: Creates two quad rows-the first one perpendicular to the opposite shell link, and the second one with a certain angle to the first one. In addition, a certain pure quad element pattern is created around the seam elements, shown here in red. These elements normally get imprinted into the shell links. The exact geometry of the seam can be influenced by the angle value.
This realization is can be used for both lap- and T-welds.

> **Note:** You can revert the direction of quad seam connectors during the next realization by activating the reverse direction checkbox in the Seam panel.

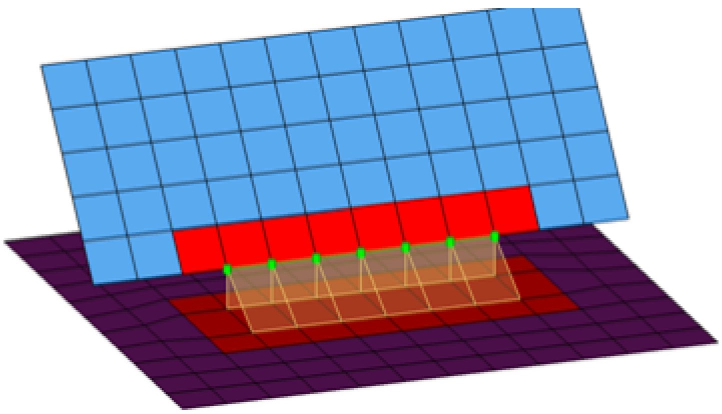


Figure 1237:

```
CFG abaqus 103 seam-quad (vertical+angled)
*filter seam
*style quad 1
*head
*body 0
quad4 1 1
```

Abaqus seam-quad (angled)

Creates one quad row under a certain angle. The angle is measured between the quad row and the perpendicular projection from the free edge to the opposite shell link. In addition, a certain pure quad element pattern is created around the seam elements, shown here in red. These elements normally get imprinted into the shell links. The exact geometry of the seam can be influenced by the angle value.
This realization is can be used for both, lap- and T-welds.

> **Note:** You can revert the direction of quad seam connectors during the next realization by activating the reverse direction checkbox in the Seam panel.

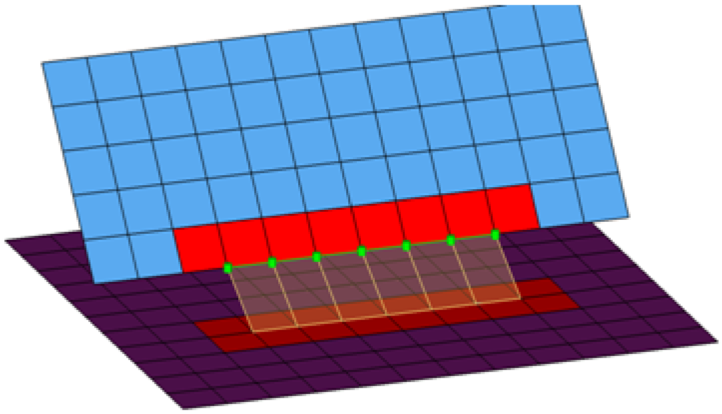


Figure 1238:

```
CFG abaqus 104 seam-quad (angled)
*filter seam
*style quad 2
*head
*body 0
quad4 1 1
```

Abaqus Mastic

Creates SPRING elements for the body, and projects and connects to the adjoining shell/solid elements with DCOUP3D elements.

The realization uses the `prop_mastic.tcl`[2] property script.

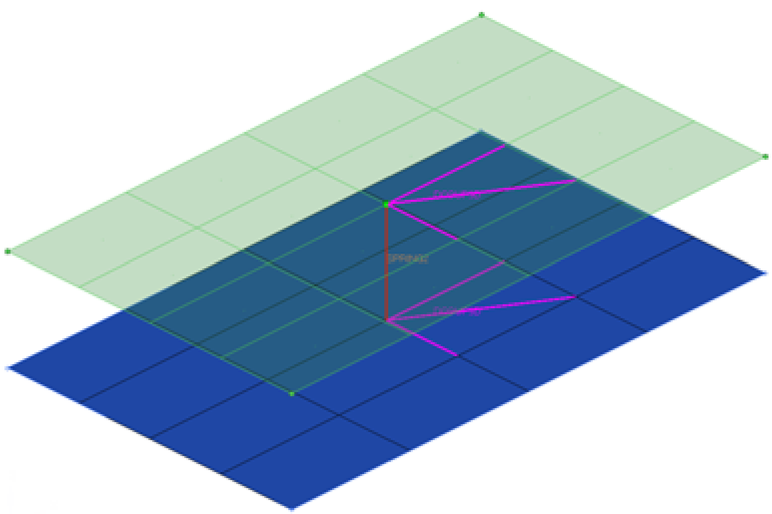


Figure 1239:

```
CFG abaqus 105 Mastic
*filter spot
*head
rbe3 1 0
*body 0
spring 1 1
spring 1 1
```

```
spring 1 1
*post prop_mastic.tcl
```

Abaqus hexa (adhesive)

Creates a row of hexa elements for the body, and numerous DCOUP3D elements for the head. The head elements project and connect to the nodes of the adjoining shell/solid elements. The hexa elements are projected so that they touch the shell/solid elements of the connecting components.

This realization also uses the `prop_abaqus_acm.tcl`[4] property script.

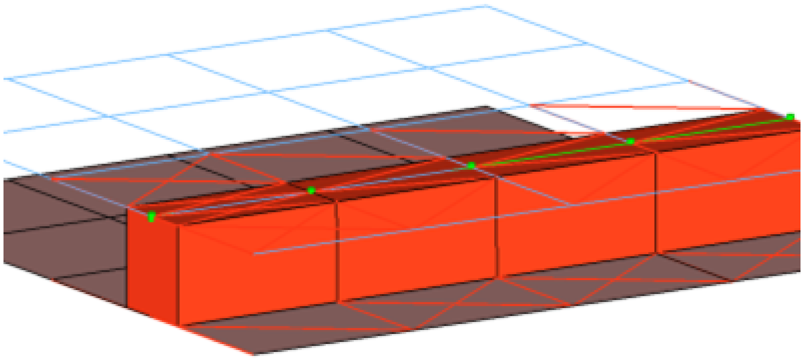


Figure 1240:

```
CFG abaqus 106 hexa (adhesive)
*filter seam
*style continuous 3
*head
rbe3 1 0
rigid 1 0
*body 0
hex8 1 1
*post prop_abaqus_acm.tcl
cfg_abaqus_106_hexa_adhesive
```

Abaqus seam (vectors)

Creates perpendicular and parallel vectors to the surface along a line/nodelist. On exporting the connector file, a vector file (`.asc` format) containing the vector information is also exported for this realization.

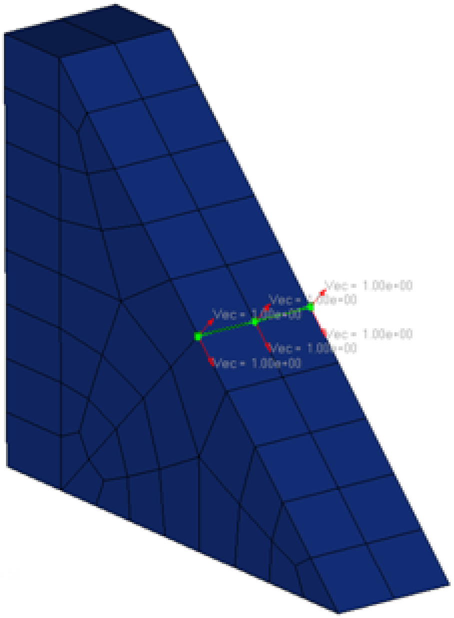


Figure 1241:

```
CFG abaqus 107 seam (vectors)
*filter seam
*style continuous_vec 1
*head
*body 0
quad4 1 1
```

Abaqus hexa (tapered T)

Intended to be used for t-cases. The size and exact position can be defined thickness dependent, or the exact dimension and position parameters can be given.

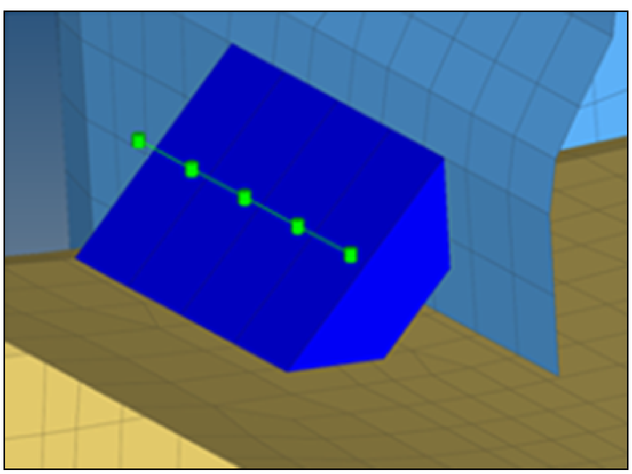


Figure 1242:

```
CFG abaqus 108 hexa (tapered T)
*filter seam
*style continuous 6
```

```
*head
rbe3 1 0
rigid 1 0
*body 0
hex8 1 1
hexa_tapered_t
```

Abaqus fastener-nodes

Creates a Node Set that contains the nodes that are selected to create the connector element, and creates an empty Element Set. Connector elements are not created for this realization type. Abaqus creates the required connector elements on its own.

This realization uses the `prop_fastener_nodes.tcl`[1] property script.

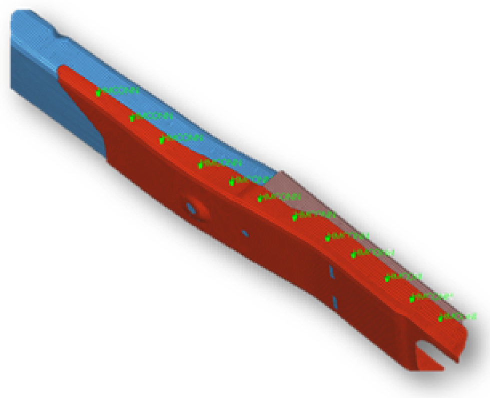


Figure 1243:

```
CFG abaqus 110 fastener-nodes
*filter spot
*head
*body 0
mass 99 2
*post prop_fastener_nodes.tcl
cfg_abaqus_110_fastener_nodes
```

Abaqus bolt (step hole)

Figure 1244:

This realization creates a B31 element for the bolt shaft and connects to the solids' nodes with numerous KINCOUP rigid elements based on the given bolt/hole parameters. It connects two solids through holes, or it connects one solid through a hole with a solid blind hole.

```
CFG abaqus 114 bolt (step hole)
*filter bolt
*style bolt 6
*head
rigidlink 1 1
*body 0
bar2 9 1
```

Abaqus bolt (threaded step hole)

Connects two solids through holes, or connects one solid through a hole with a solid blind hole. A thread length can be defined to define the dimensions of the rigid elements connecting the bolt shaft models as a bar.

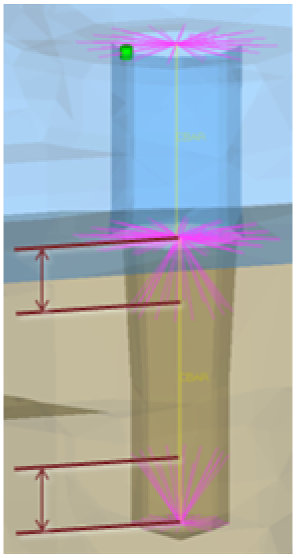


Figure 1245:

```
CFG abaqus 115 bolt (threaded step hole)
*filter bolt
*style bolt 7
*head
rigidlink 1 1
*body 0
bar2 9 1
```

Abaqus hexa (spot tie)

Creates hexa (C3D8) elements between shell and/or solid elements in order to connect them using a tie contact definition. The hexa element nodes will project and touch the shell and/or solid element faces. During the realization, a default tie contact and referencing master and slave surfaces are created; unless defined differently, the hexas are assigned a default property and material, and are organized into a component with the same name base as the property.

The default tie contact and material parameters can be changed in the files below this path: ..\Altair\2019\hm\scripts\connectors\Hexa_Tie\abaqus\.

> **Note:** IDs, names, and card type cannot be changed.

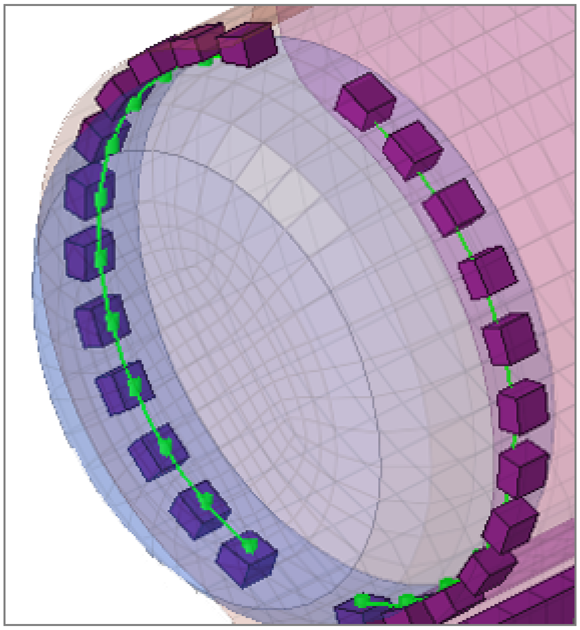


Figure 1246:

```
CFG abaqus 152 hexa (spot tie)
*filter spot
*style spot_tie 1
*head
*body 0
hex8 1 1
```

Abaqus rod (spot tie)

Creates rod (CONM3D2) elements between shell and/or solid elements in order to connect them using a tie contact definition. The rod element nodes will project and touch the shell and/or solid element faces. During the realization, a default tie contact and referencing master and slave surfaces are created; unless defined differently, each rod is assigned a property, which references a default material (CONNECTOR BEHAVIOR) and an individual coordinate system. By default, the property is directly assigned to the element so that all rods can be hosted in one component. The default tie contact and material parameters can be changed in the files below this path: ..\Altair\2019\hm\scripts\connectors\Rod_Tie\abaqus\.

> **Note:** IDs, names, and card type cannot be changed.

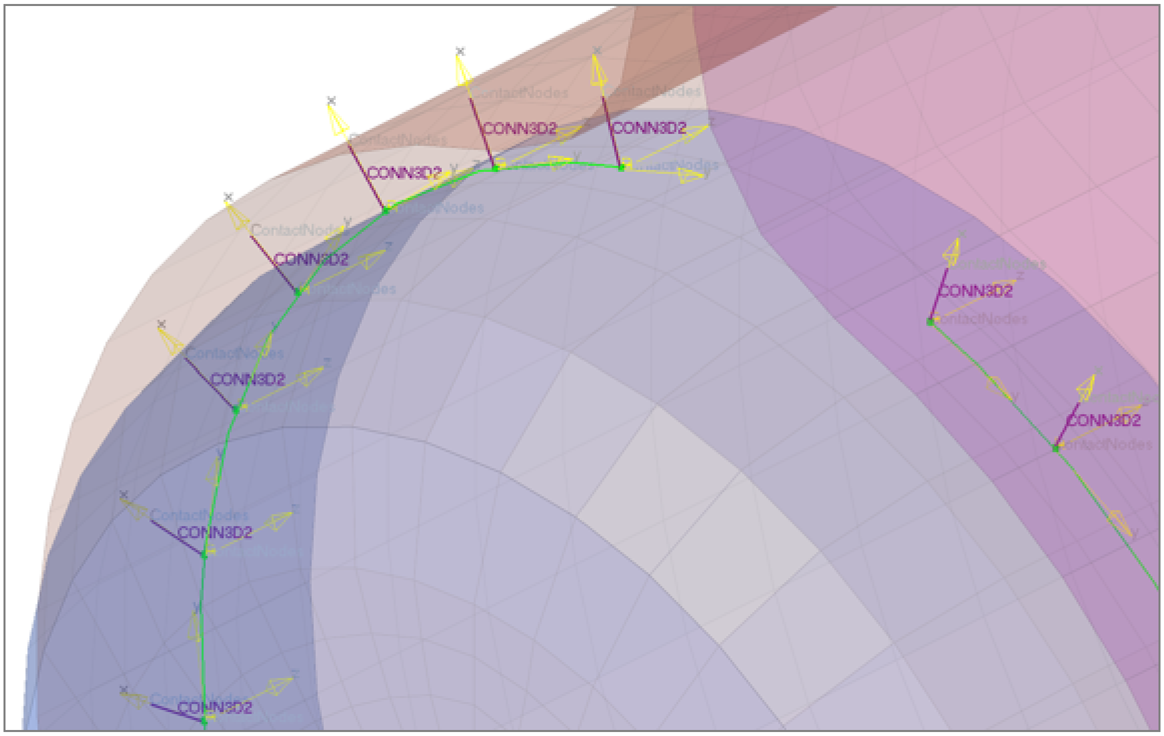


Figure 1247:

```
CFG abaqus 153 rod (spot tie)
*filter spot
*style spot_tie 3
*head
*body 0
rod 13 1
```

Abaqus hexa (seam tie)

Creates hexa (C3D8) elements between shell and/or solid elements in order to connect them using a tie contact definition. The hexa element nodes will project and touch the shell and/or solid element faces. During the realization, a default tie contact and referencing master and slave surfaces are created; unless defined differently, the hexas are assigned a default property and material, and are organized into a component with the same name base as the property.

The default tie contact and material parameters can be changed in the files below this path: ..`\Altair\2019\hm\scripts\connectors\Hexa_Tie\abaqus\`.

> **Note:** IDs, names, and card type cannot be changed.

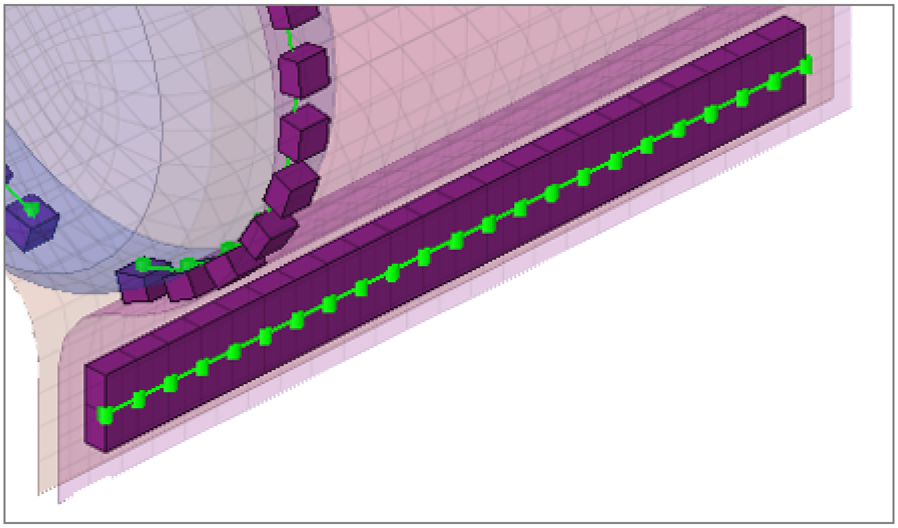


Figure 1248:

```
CFG abaqus 154 hexa (seam tie)
*filter seam
*style seam_tie 1
*head
*body 0
hex8 1 1
os_hexa_seam_tie
```

Abaqus hexa (area tie)

Creates hexa (C3D8) elements between shell and/or solid elements in order to connect them using a tie contact definition. The hexa element nodes will project and touch the shell and/or solid element faces. During the realization, a default tie contact and referencing master and slave surfaces are created; unless defined differently, the hexas are assigned a default property and material, and are organized into a component with the same name base as the property.

The default tie contact and material parameters can be changed in the files below this path: ..\Altair\2019\hm\scripts\connectors\Hexa_Tie\abaqus\.

> **Note:** IDs, names, and card type cannot be changed.

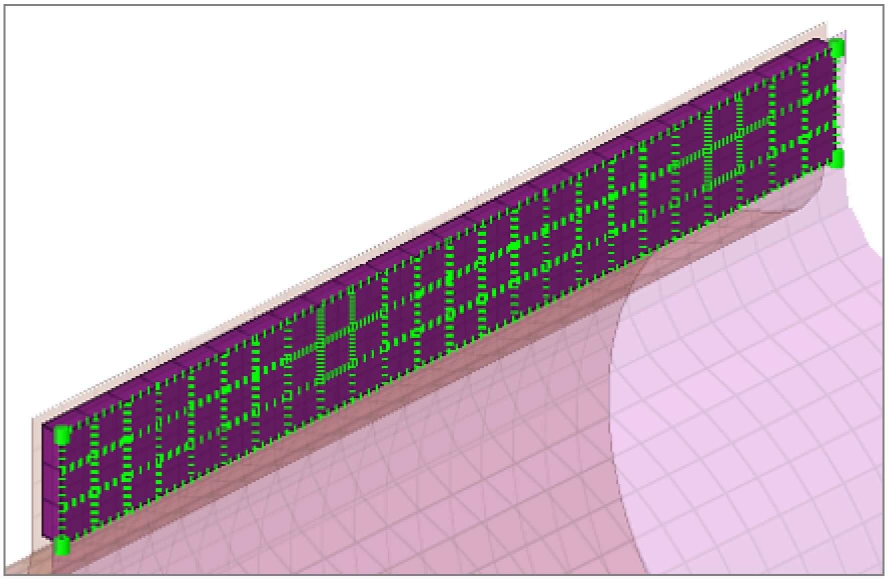


Figure 1249:

```
CFG abaqus 155 hexa (area tie)
*filter area
*style area_tie 1
*head
*body 0
hex8 1 1
```

Property Scripts

1. `prop_fastener_nodes.tcl`

   Used while creating Abaqus Fasteners-Nodes in the Spot panel. It performs the following tasks:

   - Organizes the realized weld elements into their respective components, based upon the link they are connected to. Thus, if a weld is created between comp_1(1) and comp_2(2), the script creates a component collector with the name HM_HMCONN_<id> and organizes all the welds (Dummy element) created as links between these two components into this collector. This collector is later referenced as the element set while creating Groups (Interfaces).
   - Creates groups with the card image *FASTENER, and assigns them the name HM_FastenerInteraction_<id>. The fastener connects two component collectors and refers to the fastener property card. The Automatic_Surface_from_components option is used to show the elements to which the weld elements are linked to.
   - Creates the following properties/material collectors:

     HM_ConnectorBehavior<id>
     This material collector is created with the *CONNECTOR BEHAVIOR card assigned to it.

     HM_FastenerProperty_r_<radius in property>
     This property collector is created with the *FASTENER PROPERTY card assigned to it.

HM_ConnectionSection_<id>
This property collector is created once per model (card image *CONNECTOR SECTION). The property is assigned to each HM_CONN3D2 component collector and carries the material HM_ConnectorBehavior.

- Creates the following sets:

  HM_FastenerNodes_Node_Set1
  Contains the selected nodes for the Fastener.

  HM_HMCONN_<id>
  This is the dummy elset. The connector element is created and collected based on the nodeset HM_FastenerNodes_Node_Set1 assigned to this elset. You can refer the elset to outputblock.

2. `prop_mastic.tcl`

   Performs the following tasks:

   - Organizes the SPRING elements into components with the names SPRING_X, SPRING_Y and SPRING_Z.
   - Organizes the DCOUP3D elements into the component DCOUP_3D_no_prop.
   - Creates properties with the SPRING card image, and names the properties spring_prop_K1_ElemId-##, spring_prop_K2_ElemId-##, and spring_prop_K3_ElemId-## (where ## is the element ID of the SPRING element).

   > **Note:** New components will only be created if their are not any components with the same names that already exist; otherwise the existing components are used.

3. `prop_abaqus_b31.tcl`

   Updates the direction nodes of a group of bar elements created during realization to use the y axis. The `*bardirectionupdate` command is called to update the orientation node of bar element along Y-axis.

4. `prop_abaqus_acm.tcl`

   Used while creating acm (equivalenced-(T1+T2)/2) / (detached-(T1+T2)/2) /shell gap in the Spot panel and adhesives in the Area panel. It performs the following tasks:

   - Organizes the realized weld elements [acm Equivalence-(T1+T2/2)] into the respective components based upon the *HEAD and the *BODY information of the weld. During realization of this configuration type a solid hexa element [C3D8] is connected to the shell elements by the rbe3 elements [DCOUP3D].
     - A collector with the name C3D8_comp_<id> is created with the SOLIDSECTION card image associated with it. This component contains all of the solid C3D8 elements which are created during realization.
     - A collector with the name DCOUP3D_comp_<id> is created, containing all of the DCOUP3D elements created as the heads to the weld element.
   - If this script is called during the realization of adhesives in the Area panel, this script creates the above two components by different names:

hexa_comp_<id>
: For the Hexa elements

rbe2_comp_<id>
: For the rbe elements

- The script also creates a property collector named prop_<id>, with the SOLIDSECTION card image associated to it. This property collector is referenced to the component containing the Hexa elements created during realization process (i.e. C3D8_comp_<id> in the case of spots, or hexa_comp_<id> in the case of adhesives).

5. `prop_fastener.tcl`

Used while creating Abaqus Fasteners in the Spot panel. It performs the following tasks.

- Organizes the realized weld elements into their respective components, based upon the link they are connected to. Thus, if a weld is created between comp_1(1) and comp_2(2), the script creates a component collector with the name HM_CONN3D2<id> and organizes all the welds created as links between these two components into this collector. This collector is later referenced as the element set while creating the Groups (Interfaces).
- Creates the following properties/materials collectors:

HM_ConnectorBehavior<id>
: This material collector is created with the *CONNECTOR BEHAVIOR card associated with it.

HM_ConnectionSection_<id>
: This property collector is created once per model (card image CONNECTOR SECTION). The property is assigned to each HM_CONN3D2 component collector and carries the material HM_ConnectorBehavior.

HM_FastenerProperty_r_<radius in property>
: This property collector is created with the *FASTENER PROPERTY card associated with it. It defines the RADIUS and the degree of freedom definition of the fastener.

- Creates Groups (HM Interfaces) with the name HM_FastenerInteraction<id> and with the *FASTENER card associated with it. The fastener connects two component collectors and refers to the fastener property card mentioned above. It can also show the link elements to which the weld elements are linked via the Automatic_Surface_from_components option.
  - If any system option (Single System ,1- System per layer or 2- Systems per layer) is used in the Spot panel during realization, this script creates ORIENTATION systems in the current collector with the name HM_ORI<weld_id>_n<node_id>. A property HM_ConnectorSection_<CONN3D2 element id> is created and assigned per element. Depending on whether one or two systems per layer are created the property points to one or both systems.

6. `prop_cylinder.tcl`

Used while creating bolt (cylinder rigid) and bolt (cylinder bar) in the Bolt panel (Abaqus, Nastran, OptiStruct). It organizes the realized bolt elements into the respective components based upon the*HEAD and the *BODYinformation of the bolt:

- A collector with the name Rigid_M<diameter> is created. This component contains all of the rigid head elements and the rigid body elements, if available.

- A collector with the name Beam_M<diameter> is created. This component contains all of the bar2 head elements, if available. This component then gets a property Beam_M assigned (*BEAMSECTION or PBEAM).

ANSYS Connector Types

Supported ANSYS connector types.

hexa (tapered T)

Intended to be used for t-cases. The size and exact position can be defined thickness dependent, or the exact dimension and position parameters can be given.

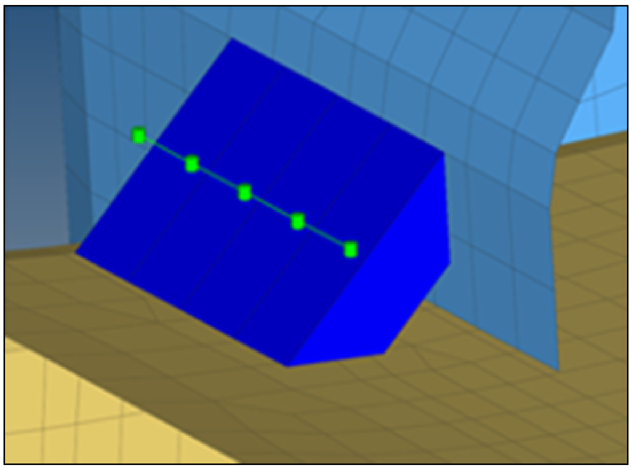


Figure 1250:

```
CFG ansys 105 hexa (tapered T)
*filter seam
*style continuous 6
*head
*body 0
hex8 1 1
```

Bolt (link10)

Creates a body with element type ‘Link10’ element. CERIG elements will be created at the head. Head elements project and connect to the adjoining elements that form the hole. Properties and materials for the Link10 element are also created.

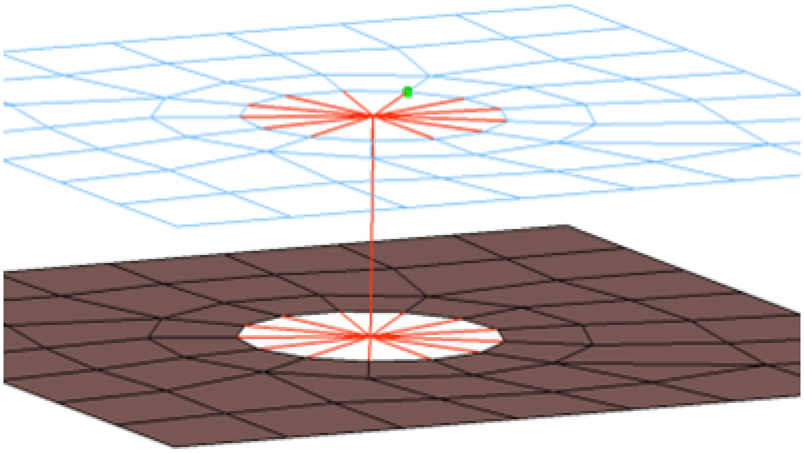


Figure 1251:

```
CFG ansys 112 bolt
*filter bolt
*style bolt 0
*head
rigid 1 1
*body 0
rod 2 1
*post prop_ansys.tcl
cfg_ansys_boltlink10
```

Bolt(BEAM44)

Creates a body with element type 'Beam44' element. CERIG elements will be created at the head. Head elements project and connect to the nodes of adjoining elements that form the hole. Properties and materials for the Beam44 element are also created.

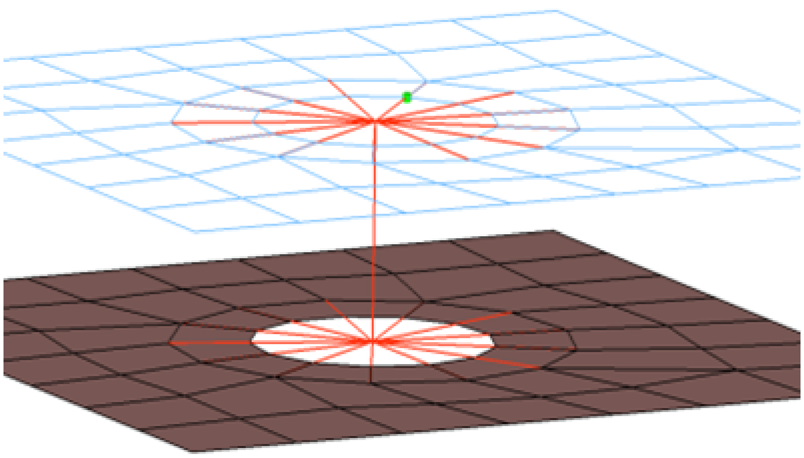


Figure 1252:

```
CFG ansys 113 bolt (BEAM44)
*filter bolt
*style bolt 0
*head
rigid 1 1
```

```
*body 0
bar2 7 1
*post prop_ansys.tcl
cfg_ansys_boltlink10
```

Clip

Creates CERIG and Mass21 elements. Mass21 element is created at the center location of bolt body. CERIG elements connect Mass21 to the nodes of adjoining elements of the shell that form hole. CERIG elements, from Mass21 also connect to the nodes of the element that represent the washer of the bolt. Properties and materials are also created for the Mass21 element type. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes, or on the second row of nodes which form the washer layer.

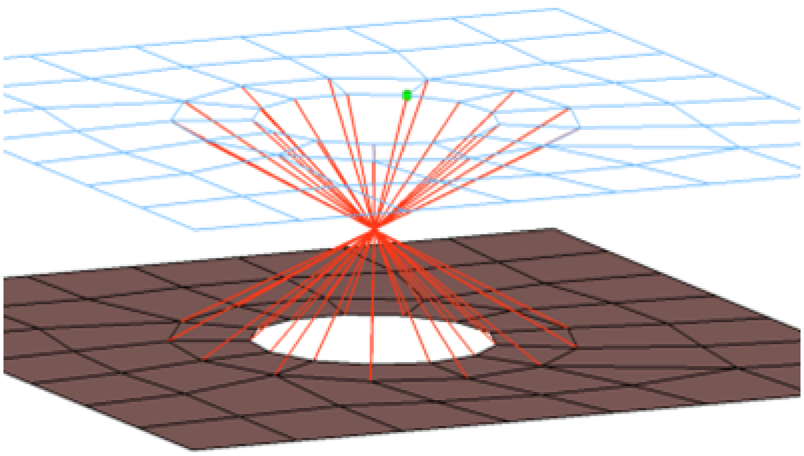


Figure 1253:

```
CFG ansys 114 clip
*filter bolt
*style bolt 1
*head
*body 0
rigidlink 1 2
*post prop_ansys.tcl
```

Bolt (spider)

Creates many individual CERIG elements. The element projects and connects to the nodes of the adjoining shell elements which form the hole, the CERIG elements are joined at the midpoint of the bolted connection. A MASS21 element is created at this location. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer. Properties and materials for the MASS21 element is also created.

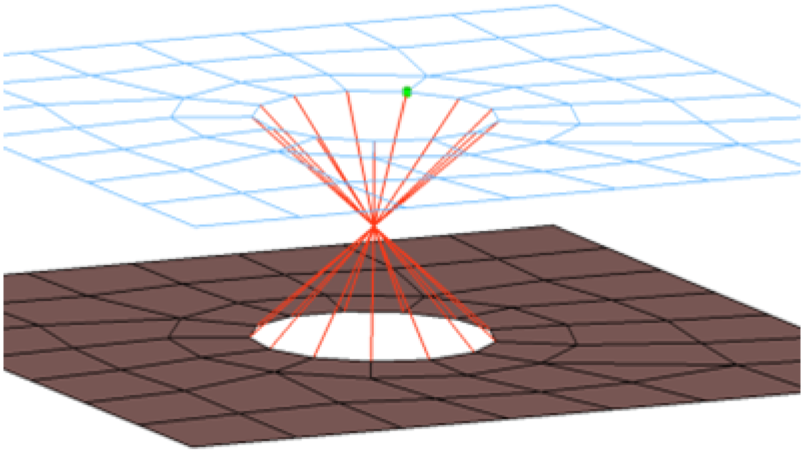


Figure 1254:

```
CFG ansys 115 bolt (spider)
*filter bolt
*style bolt 1
*head
*body 0
rigid 1 1
*post prop_ansys.tcl
```

Bolt (Washer1) link10

Creates CERIG element for the head and LINK10 for body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole and also the second row of nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer. Properties and materials are also created for the LINK10 element.

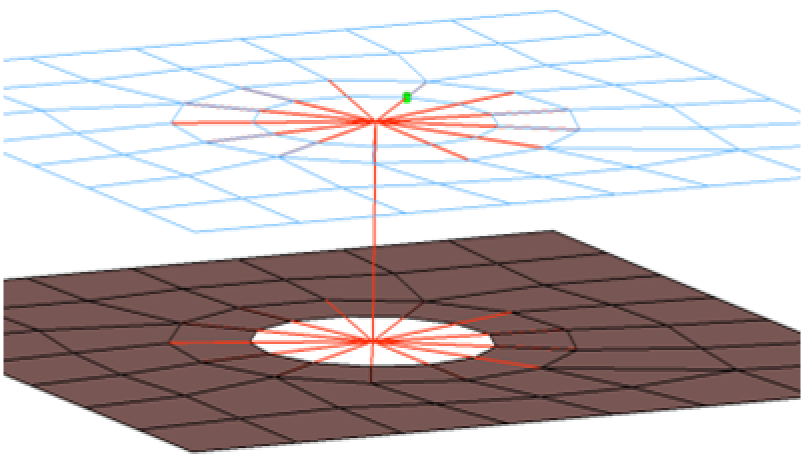


Figure 1255:

```
CFG ansys 116 bolt (washer 1)
*filter bolt
*style bolt 0
*head
rigidlink 1 12
```

```
*body 0
rod 2 1
*post prop_ansys.tcl
cfg_ansys_boltwasher1link10
```

Bolt (Washer1 alternate) link10

Creates CERIG element for the head and LINK10 element for body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole and also the second row of nodes which form the washer layer. The head only connects to every other node on the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer. Properties and materials are also created for the LINK10 element.

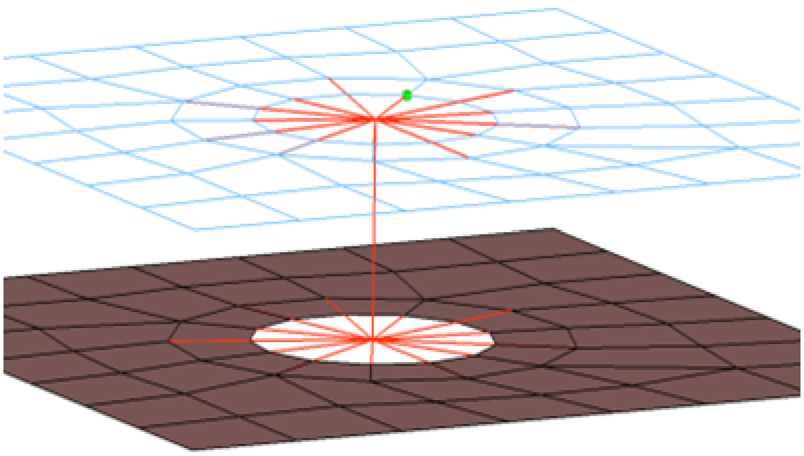


Figure 1256:

```
CFG ansys 117 bolt (washer 1 alt) LINK10
*filter bolt
*style bolt 0
*head
rigidlink 1 13
*body 0
rod 2 1
*post prop_ansys.tcl
cfg_ansys_boltwasher1alternatelink10
```

Bolt (Washer 1) BEAM44

Creates CERIG elements for the head and BEAM44 element for the body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole and also the second row of nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer. Property and material cards for the BEAM44 element are created.

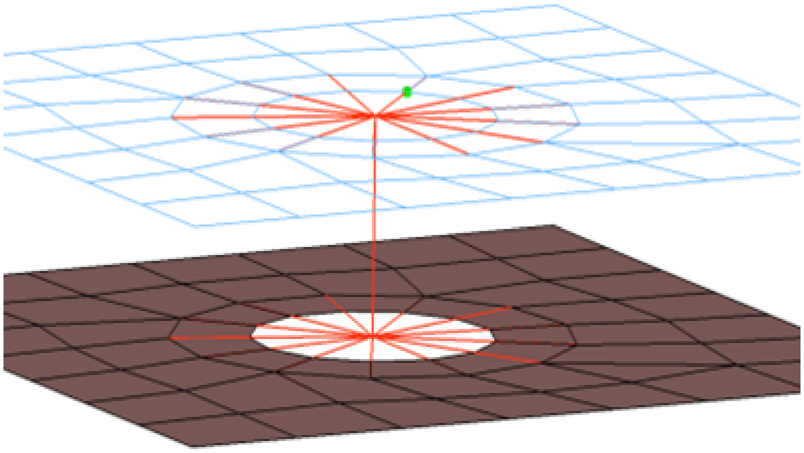


Figure 1257:

```
CFG ansys 118 bolt (washer 1) BEAM44
*filter bolt
*style bolt 0
*head
rigidlink 1 12
*body 0
bar2 7 1
*post prop_ansys.tcl
cfg_ansys_boltwasher1beam44
```

Bolt (Washer 2) LINK10

Creates CERIG elements for the head and the LINK10 element for body. There are two individual CERIG elements at the head of the connection, one to connect to the inner row of nodes, the other to connect to the washer layer nodes. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer. Property and material cards are created for the LINK10 element.

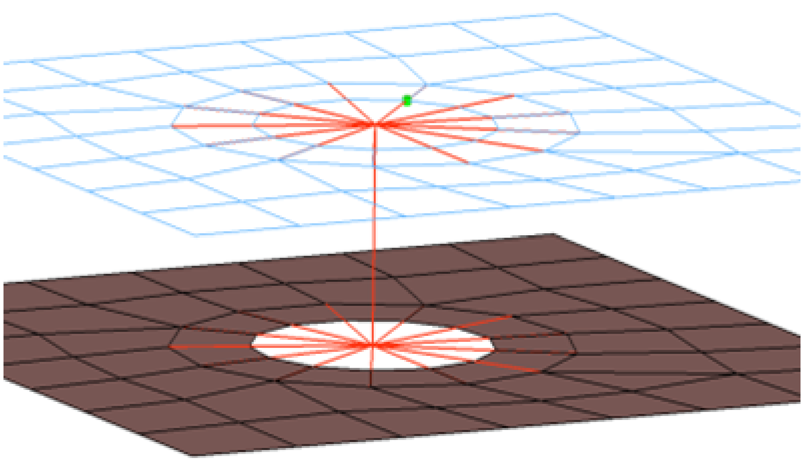


Figure 1258:

```
CFG ansys 119 bolt (washer 2) LINK10
*filter bolt
*style bolt 0
```

```
*head
rigidlink 1 1
rigidlink 1 2
*body 0
rod 2 1
*post prop_ansys.tcl
cfg_ansys_boltwasher2link10
```

Bolt (Washer 2 alt) LINK10

Creates CERIG elements for the head and the LINK10 element for body. There are two individual CERIG elements at the head of the connection, one to connect to the inner row of nodes, the other to connect to the washer layer nodes. The CERIG head element that connects to the washer layer nodes only connects to every other node on the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer. Property and material cards are created for the LINK10 element.

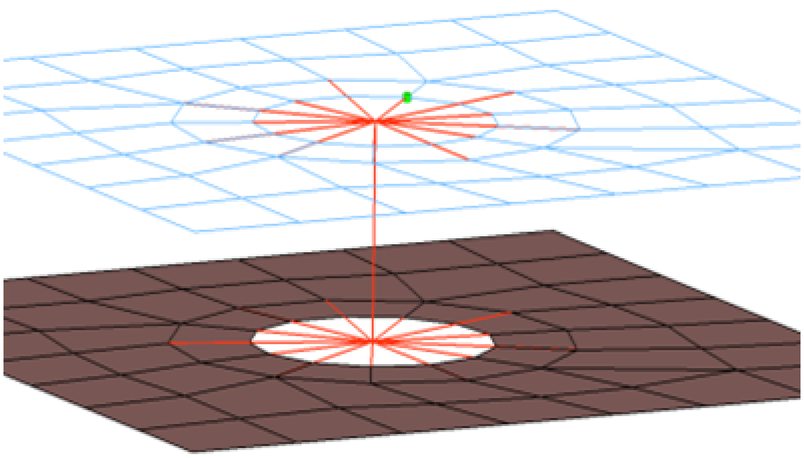


Figure 1259:

```
CFG ansys 120 bolt (washer 2 alt) LINK10
*filter bolt
*style bolt 0
*head
rigidlink 1 1
rigidlink 1 3
*body 0
rod 2 1
*post prop_ansys.tcl
cfg_ansys_boltwasher2altlink10
```

LS-DYNA Connector Types

Supported LS-DYNA connector types and property/post scripts.

Connector Types

dyna rigid (crbody)

Creates a RgdBody element for the body.

The dyna rigid (crbody) exists in exactly the same configuration with the following additional names:

- HC mig weld
- HC laser

This is in order to keep the realization names from HyperCrash.

This realization also uses the `prop_rigid_crbody.tcl`[1] property script.

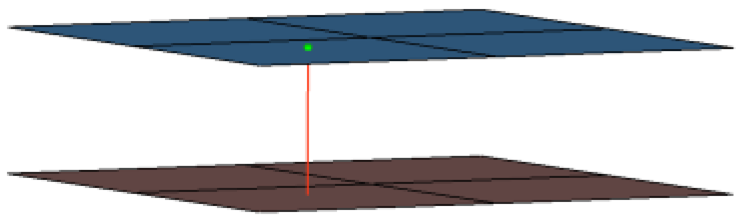


Figure 1260:

```
CFG dyna 5 rigid (crbody)
*filter spot
*head
*body 0
rigid 2 1
*post prop_rigid_crbody.tcl
```

dyna ConNode (spider)

Creates a ConNode rigidlink element for the body. It connects to the nearest node to the connector position and then projects to the nearest nodes on the adjoining elements of the connected components.
The dyna ConNode (spider) exists in exactly the same configuration with the following additional names:

- HC cylinder rigid bolt
- HC cylinder rigid clip

This is in order to keep the realization names from HyperCrash.

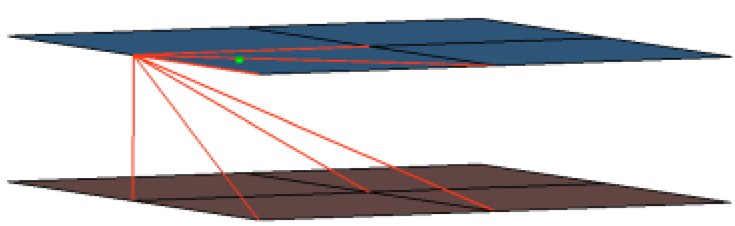


Figure 1261:

```
CFG dyna 56 ConNode (spider)
*filter bolt
*style bolt 2
```

```
*head
*body 0
rigidlink 1 1
```

dyna RgdBody (spider)

Creates a RgdBody rigidlink element for the body. It connects to the nearest node to the connector position and then projects to the nearest nodes on the adjoining elements of the connected components.

If holes are detected the nodes on the edges are connected.

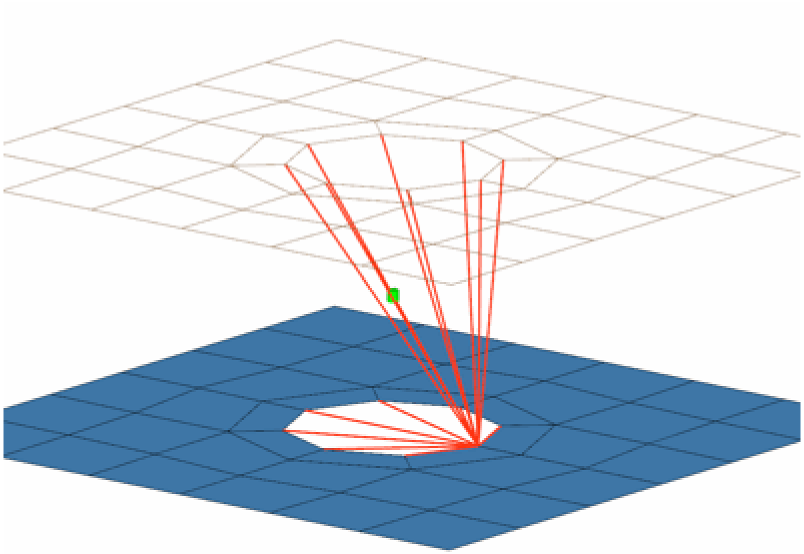


Figure 1262:

```
CFG dyna 57 RgdBody (spider)
*filter bolt
*style bolt 2
*head
*body 0
rigidlink 2 1
```

dyna RgdBody (spider+washer)

Creates a RgdBody rigidlink element for the body. It connects to the nearest node to the connector position and then projects to the nearest nodes on the adjoining elements of the connected components.

If holes are detected the nodes on the edges and the washer nodes are connected.

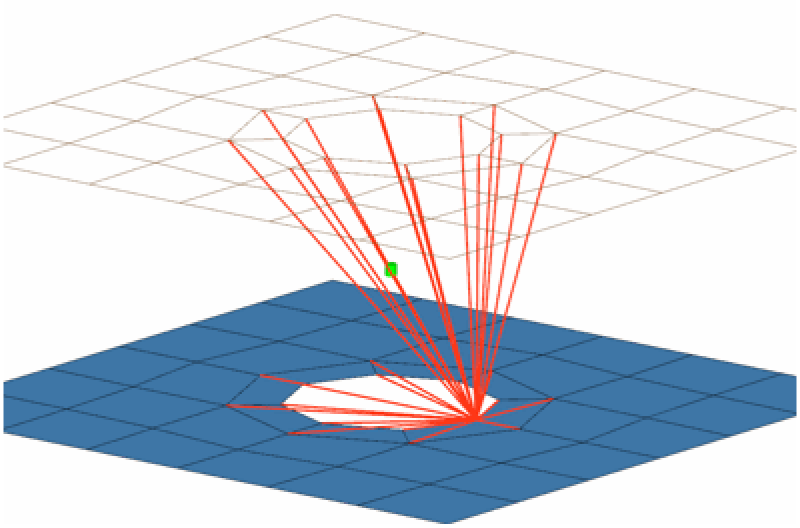


Figure 1263:

```
CFG dyna 58 RgdBody (spider+washer)
*filter bolt
*style bolt 21
*head
*body 0
rigidlink 2 1
```

dyna acm (shell gap + coating)

This realization creates one hexa cluster per connector, and realizes a node to node connection to the linked shell meshes by adjusting it (shell coating). Different patterns are available. This is driven by the number of hexas. The appearance can be influenced via the diameter and the washer layer activation.

This realization uses the `prop_acm_coating.tcl`[2] property script.

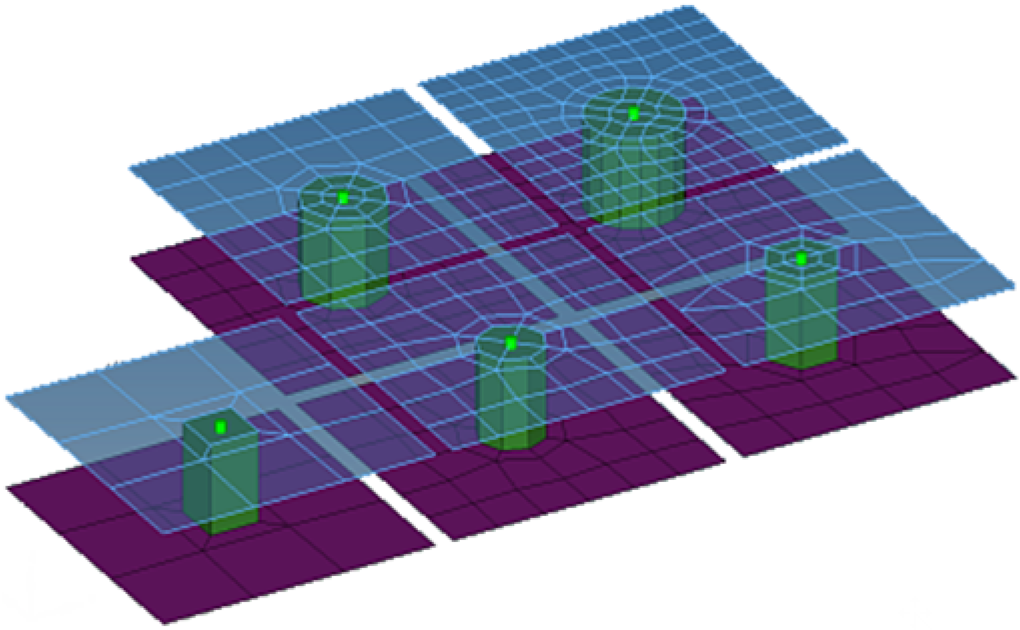


Figure 1264:

```
CFG dyna 72 acm (shell gap + coating)
*filter spot
*style acm 4
*body 0
hex8 1 1
*post prop_acm_coating.tcl
```

dyna mat100

Creates a BEAM element for the body and plot elements for the head, the plot elements are created for visualization purposes and find operations.

The dyna mat100 exists in exactly the same configuration with the following additional names:

- HC beam spotweld
- HC glue
- HC glue structural adhesive
- HC welding line
- HC hemming

This is in order to keep the realization names from HyperCrash.

This realization also uses the `prop_dyna_matnum.tcl`[3] property script.

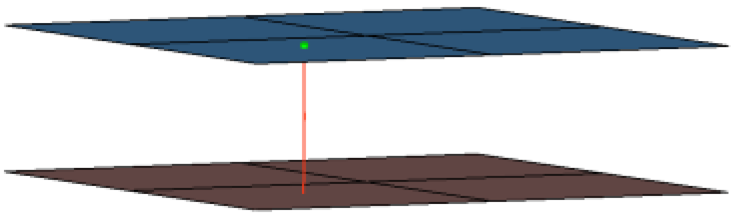


Figure 1265:

```
CFG dyna 100 mat100
*filter spot
*head
*body 0
bar2 1 1
*post prop_dyna_matnum.tcl
```

dyna mat100 (hexa)

Creates hexa element with plot elements projecting and connecting to the surrounding shell elements. This realization does not use the shell thickness to calculate the hexa offset, therefore the hexa will project and be touching the shell elements.
The dyna mat100 (hexa) exists in exactly the same configuration with a HC hexa spotweld. This is in order to keep the realization names from HyperCrash.

This realization also uses the `prop_dyna_matnum.tcl`[3] property script.

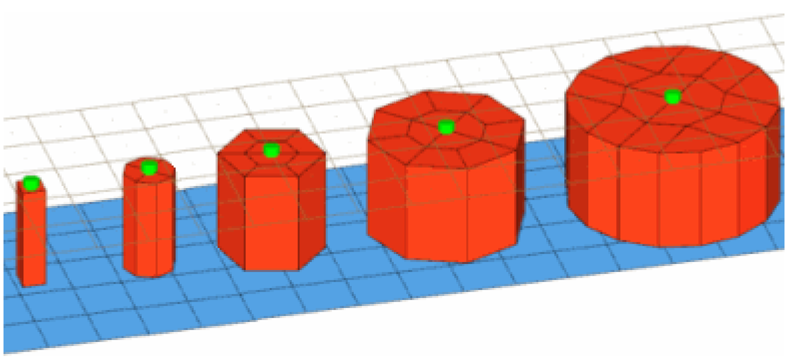


Figure 1266:

```
CFG dyna 101 mat100 (hexa)
*filter spot
*head
*body 0
hex8 1 1
*post prop_dyna_matnum.tcl
```

dyna mat196

Creates a BEAM element for the body and plot elements for the head, the plot elements are created for visualization purposes and find operations. This realization is the same as the “CFG dyna 100 mat100” realization except it uses Mat196 as opposed to Mat100.

This realization also uses the `prop_dyna_matnum.tcl`[3] property script.

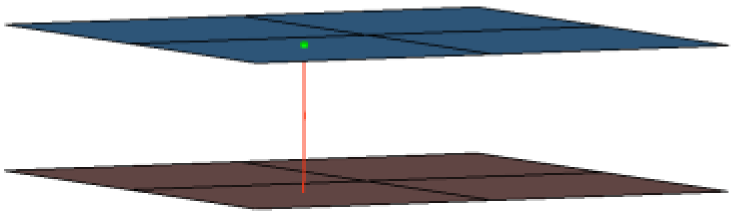


Figure 1267:

```
CFG dyna 102 mat196
*filter spot
*head
*body 0
bar2 1 1
*post prop_dyna_matnum.tcl
```

dyna hexa (tapered T)

Intended to be used for t-cases. The size and exact position can be defined thickness dependent, or the exact dimension and position parameters can be given.

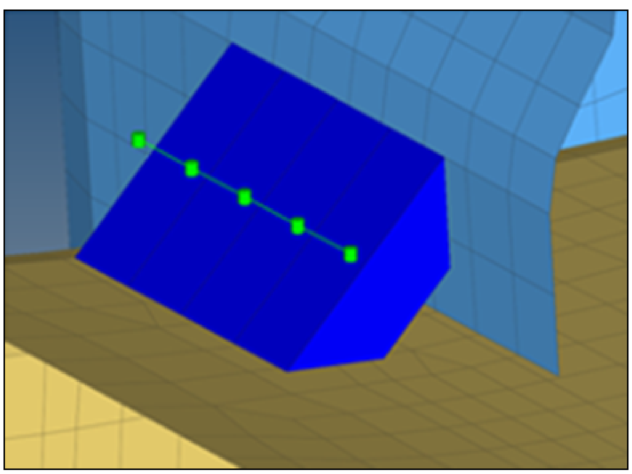


Figure 1268:

```
CFG dyna 105 hexa (tapered T)
*filter seam
*style continuous 6
*head
*body 0
hex8 1 1
```

dyna hexa (adhesive - shell gap)

This realization creates rows of HEXA elements for the body. The HEXA elements project and connect to the adjoining shell/solid elements by touching them.

> **Note:** Uses the default parameters, material thickness lookup table, and material strength lookup table from the `weld_config.ini` config file to set the default attributes for the the created MATL100 material.

The dyna hexa (adhesive - shell gap) exists in exactly the same configuration with the following additional names:

- HC glue mastic sealer
- HC glue spot sealer
- HC glue glass adhesive

This is in order to keep the realization names from HyperCrash.

The realization uses the `prop_dyna_matnum_seamarea.tcl`[4] post script.

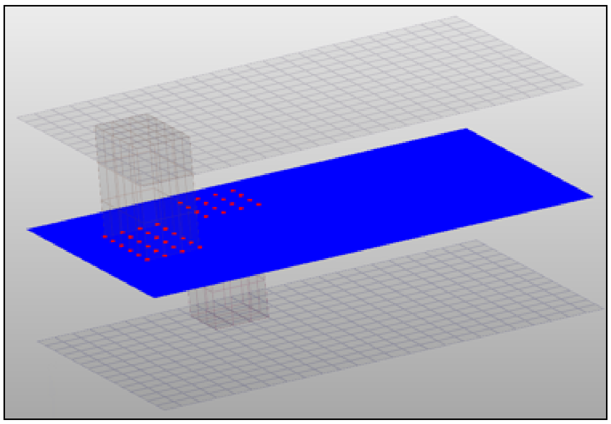


Figure 1269:

```
CFG dyna 106 hexa (adhesive - shell gap)
*filter seam
*style continuous 2
*head
*body 0
hex8 1 1
*post prop_dyna_matnum_seamarea.tcl
```

dyna HC cylinder rigid bolt

This realization creates a single ConNode rigidlink element for the body, and projects and connects to nodes of the adjoining shell/solid within the prescribed cylinder diameter, L1 (cylinder height along the vector from the connector location) and L2 (cylinder height in the opposite direction of the vector from the connector location).

The realization uses the `prop_dyna_rigidbolts.tcl`[5] property script.

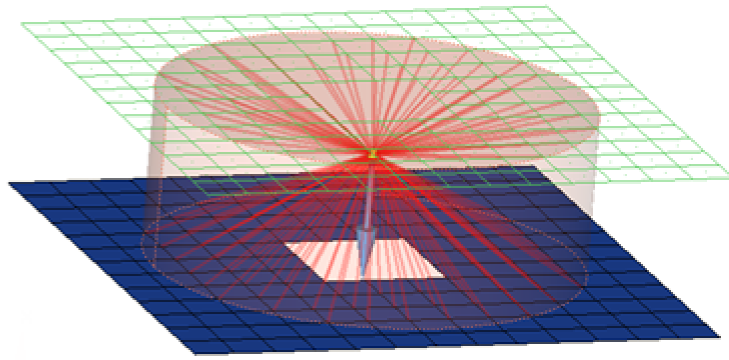


Figure 1270:

```
CFG dyna 115 HC cylinder rigid bolt
*filter bolt
*style bolt 4
*head
*body 0
rigidlink 1 1
*post prop_dyna_rigidbolts.tcl
```

dyna adhesive (shell gap)

This realization creates a DISCRETE element for the body, and projects and connects to nodes of the adjoining shell/solid elements with ConNode rigidlink elements within the prescribed cylinder diameter, L1 (cylinder height along the vector from the connector location) and L2 (cylinder height in the opposite direction of the vector from the connector location).

The realization uses the `prop_dyna_rigidbolts.tcl`[5] property script.

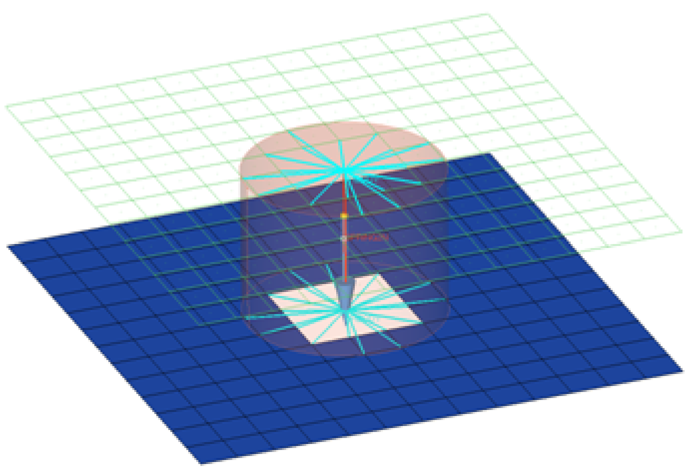


Figure 1271:

```
CFG dyna 116 HC cylinder spring bolt
*filter bolt
*style bolt 4
*head
rigidlink 1 1
*body 0
spring 1 1
```

```
*post prop_dyna_rigidbolts.tcl
```

dyna HC cylinder spring bolt

Creates a row of hexa/penta elements. The hexa/penta elements are projected so that they touch the shell elements of the connecting components.

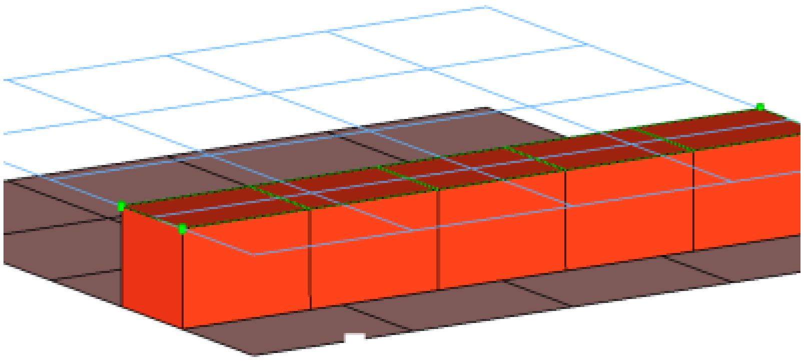


Figure 1272:

```
CFG dyna 121 adhesive (shell gap)
*filter area
*style adhesive 2
*head
*body 1
hex8 1 1
penta6 1 1
```

dyna - adhesive (shell gap)

This realization creates rows of HEXA and PENTA elements for the body. The HEXA and PENTA elements project and connect to the adjoining shell/solid elements by touching them.

> **Note:** Uses the default parameters, material thickness lookup table, and material strength lookup table from the weld_config.ini config file to set the default attributes for the the MATL100 material that was created.

This realization uses the `prop_dyna_matnum_seamarea.tcl`[4] post script.

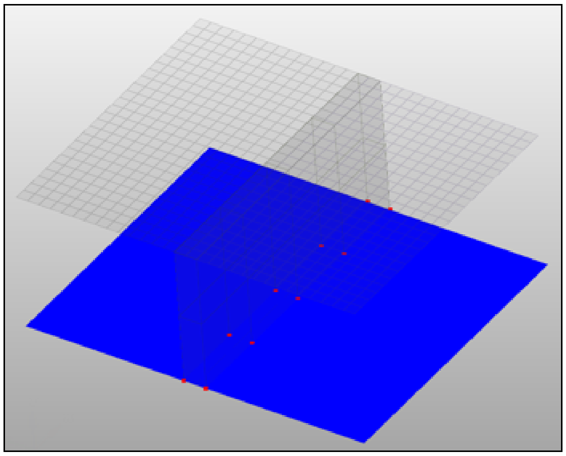


Figure 1273:

```
CFG dyna 121 adhesive (shell gap)
*filter area
*style adhesive 1
*head
*body 1
hex8 1 1
penta6 1 1
*post prop_dyna_matnum_seamarea.tcl
```

dyna - mat196 (single row)

This realization creates a single row of BEAM elements for the body.

> **Note:** Use the `init_dyna_mat196.ini` config file to set the default attributes for the created MATL196 material.

This realization uses the `prop_dyna_matnum_seamarea.tcl`[4] post script and the `weld_config.ini` and `init_dyna_mat196.ini` config file.

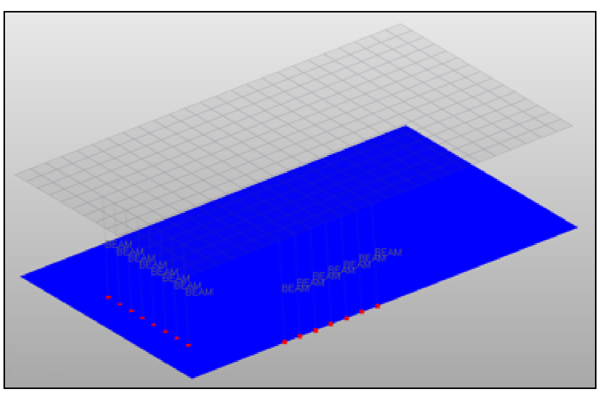


Figure 1274:

```
CFG dyna 122 mat196 (single row)
*filter seam
*style continuous 4
```

```
*head
*body 0
bar2 1 0
*post prop_dyna_matnum_seamarea.tcl
```

Automatic Exclusion of Special Nodes During Rigid Bolt Realization

HyperMesh automatically excludes special nodes as potential slave nodes for any rigid bodies created during bolt realization, even though they fall with the virtual Bolt Cylinder diameter. Nodes that are shared by the following entities are considered special nodes:

- rigid links
- Rigids
- Rbe3 nodes
- boundaries condition
- IMPDISP
- IMPVEL
- IBVEL
- IMPACC
- FXBODY
- Nodes inside the Interfaces with Type2

Property and Post Scripts

1. `prop_rigid_crbody.tcl`

   Updates mesh-dependent rigid welds > 2T into rigidlinks sharing a node. This is a requirement for the LS-DYNA *CONSTRAINED_NODAL_RIGID_BODY definition.
2. `prop_acm_coating.tcl`

   Performs the following task:

   - Organizes the HEXA elements into a set with the name CE_HEXSW_# (# starts with 1 and increments as 1, 2, 3, and so on for every new connector).
3. `prop_dyna_matnum.tcl`

   Used during the creation of mat100/mat100 (hexa)/mat196 custom config welds in the Spot panel.

   This script performs the following tasks:

   - Organizes the realized weld elements to the respective components based upon the link they are connected to and based upon the realization used.

     If a weld is created as mat100 between comp_1(id1) and comp_2(id2), it will create a component collector with the name C_^_<id1_id2>_BEAM_100 and organize all of the welds created as links between these two components into this collector.

     If a weld is created as mat100 between the three components comp_1(id1), comp_2(id2) and comp_3(id3), it will create two component collectors, C_^_1W_<id1_id2>_BEAM_100 and C_^_1W_<id2_id3>_BEAM_100. The suffix is based on the realization type:

Mat100

_BEAM_100

Mat100 (hexa)

_SOLID

Mat196

_BEAM_196

This collector is later referenced as the weld element set while creating the groups (contact) definition.

- This script will create the following properties/materials collectors:
    - M_^<id1_id2>_< MAT100 or MAT196> or M_^_1W_<id1_id2_id3>_<MAT100 or MAT196>: These material collectors are created upon the selection of the configuration type by you with the MAT100 or the MAT196 card. The values for the cards are read from the `*.ini` file. These material collectors are referenced in the above created appropriate components.

```
#MATERIAL STRENGTH LOOKUP TABLE
# FIRST NUMBER INDICATES NUMBER OF LEVELS
# SECOND NUMBER INDICATES MULTIPLIER FOR SIGY OF NUGGET
#MIN         MAX         k           n           a           b
#LAST LINE:
#MIN         MAX         k           n           a           b
 NUMBER
*SIGY        4           1.85
0            0.20        4.0000      1.9000      10.5000     -4.000
0.20         0.40        4.2000      1.9500      12.000      -3.000
0.40         0.80        4.5000      1.9500      14.200      -2.000
0.80         0.90        4.7000      1.9900      16.500      -1.0000
 9.00
```

If $Bound_{lower_j} < SIGY_{base_k} < Bound_{upper_j}$, then $NRR_k = k_j t_{min}^{n_j}$

$$NRS_k = NRT_k = a_j t_{min} + b_j$$

If $SIGY_{base_l} > Bound_n$, then $NRR_k = NRS_k = NRT_k = number$

- P_^<id1_id2>_<BEAM or SOLID> or P_^_1W<id1_id2_id3>_<BEAM or SOLID>: These property collectors are created with the *SECTION_BEAM or *SECTION_SOLID card associated with them. These property collectors are referenced in the above created appropriate components.

```
# SAMPLE MATERIAL THICKNESS LOOKUP TABLE:
# NUMBER INDICATES NUMBER OF LEVELS
#LAST LINE:
*THICKNESS      5
 0              0.1              0.125
 0.1            0.5               .25
 0.5            1.0               .75
 1.0            1.5              1.25
 1.5            2.0             17.75              2.0
```

- This script also creates the necessary group (contact) definition upon the selection of the configuration type. For mat100 and mat196 a group named C_Contact_Spotweld_<group id> and/or for mat100 (hexa) a group named C_Contact_Tied_Shell_Edge_To_Surface_<group id> is/are created. These interfaces are defined with the appropriate solver cards and reference the following master (MSID) and slave (SSID) sets:
    - For the *CONTACT_SPOTWELD_ID card the following FS and FD are set to 0.1.
    - For the *CONTACT_TIED_SHELL_EDGE_TO_SURFACE_ID card FS and FD are set to 0.1 as well. Additionally the values for variable SST and MST in the card image are set to 0.010. These values will override the thickness and establish the tied connection.
- The script creates sets by the name C_S_^Part_<set id>_Contact_<group id> and C_S_^Weld_<set id>_Contact_<group id>. The configuration types mat100 and mat196 share the same sets, while the configuration type mat100 (hexa) gets a different pair of sets. The former set contains the parts to which the welds are connected and the latter contains the weld components created during realization process. The part sets is defined as master in the appropriate contact definition, the weld set as slave.

  In addition, for the mat100 (hexa) realization a set for each hexa cluster is created and named CE_HEXSW_<set id>.

  All these sets get a card *DEFINE_HEX_SPOTWELD_ASSEMBLY_<# hexa> associated with them.

  > **Note:** This script is called if the CFG type is mat100/mat100 (hexa)/mat196

4. `prop_dyna_matnum_seamarea.tcl`

   This script performs the following tasks:

   - Creates NodesToSurface(Tied interfaces) and names them ADHESIVE_SEAM_CONTACT_PID _# , which references the independent/dependent links' master sets and the nodes' slave sets (# is the ID of the link components).
   - Organizes the link components into a set with name ADHESIVE_SEAM_MASTER_PART_SET_PID_#, which is referenced by the above interface group (# is the ID of the link component).
   - Organizes the solids�much nodes on the links into a set with the name ADHESIVE_SEAM_SLAVE_NODE_SET_PID _#, which in turn is referenced by the above interface group (# is the ID of the link component).
   - Creates a component with the name ADHESIVE_SEAM_MAT100_COMP_PID_#_# for the connector SOLID elements (# is the ID of the link component).
   - Creates a property with the name ADHESIVE_SEAM_MAT100_PROP_PID _#_# and SectSld card image, and assigns it to the SOLID component (# is the ID of the link component).
   - Creates a material with the name ADHESIVE_SEAM_MAT100_MAT_PID_#_# and MATL100 card image, and assigns it to the SOLID component (# is the ID of the link component).

   > **Note:** Uses the default parameters, material thickness lookup table, and material strength lookup table from the `weld_config.ini` config file to set the default attributes for the created MATL100 material.

5. `prop_dyna_rigidbolts.tcl`

   The script performs the following tasks:

   - Organizes the ConNode rigidlink elements into a component with the name Realize#001 (# starts with 2 and increments as 2, 4, 6, and so on for each new connector).
   - Creates a view with the name dyna_rigid_bolts.

Nastran Connector Types

Supported Nastran connector types and property scripts.

Nastran sealing

Creates RBE3 elements for the head and CBUSH element for the body. The head elements project and connect to the nodes of the adjoining shell elements.

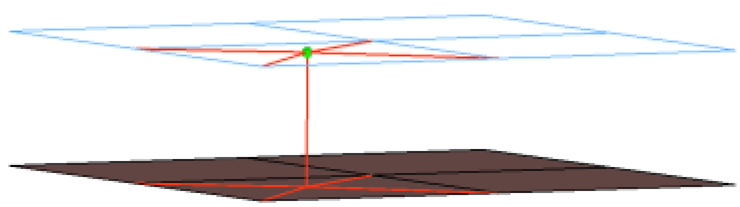


Figure 1275:

```
CFG nastran 5 sealing
*filter spot
*head
rbe3 1 0
*body 0
spring 6 1
```

Nastran bush

Creates RBE2 elements for the head and CBUSH element for the body. The head elements project and connect to the nodes of the adjoining shell elements.

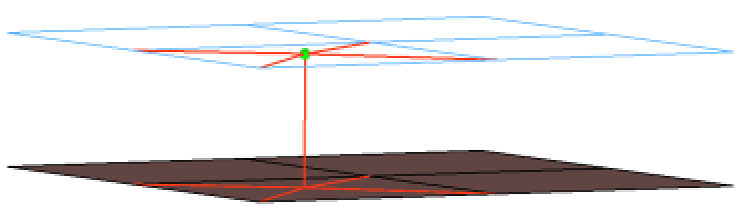


Figure 1276:

```
CFG nastran 6 bush
```

```
*filter spot
*head
rigidlink 1 1
*body 0
spring 6 1
```

Nastran rbe3 (load transfer)

Creates RBE3 elements for the body. The degrees of freedom are constrained in the x, y, z for the dependant nodes.

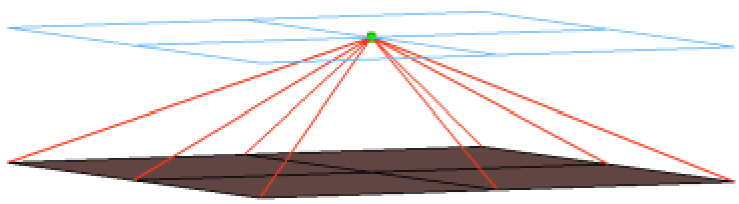


Figure 1277:

```
CFG nastran 31 rbe3 (load transfer)
*filter spot
*style mpc 1
*head
*body 0
rbe3 1 1 dofs=123
```

Nastran clip

Creates a single RBE2 element for the body. The element projects and connect to the nodes of the adjoining shell elements which form the hole and also the nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

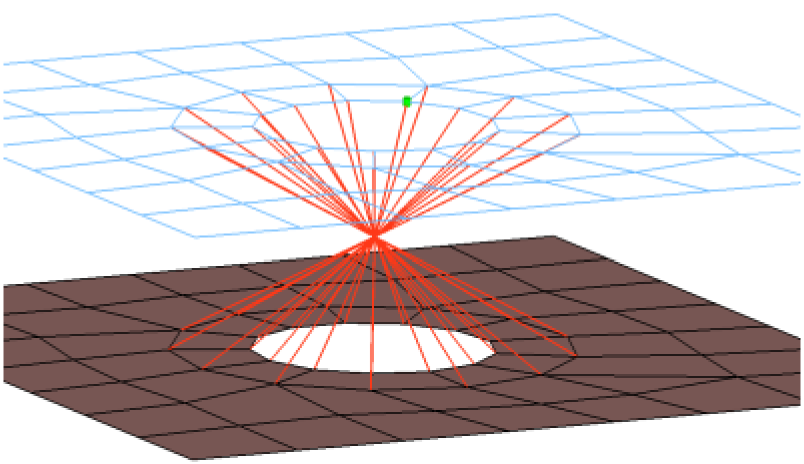


Figure 1278:

```
CFG nastran 50 clip
```

```
*filter bolt
*style bolt 1
*head
*body 0
rigidlink 1 2
```

Nastran bolt (washer 1) cbar

Creates RBE2 elements for the head and CBAR element for the body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole and also the second row of nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

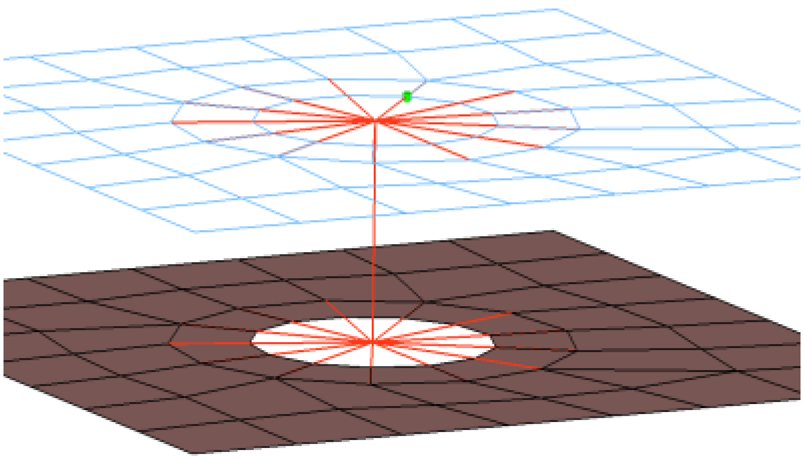


Figure 1279:

```
CFG nastran 51 bolt (washer 1) cbar
*filter bolt
*style bolt 0
*head
rigidlink 1 12
*body 0
bar2 1
```

Nastran bolt (general)

Creates RBE2 elements for the head and the body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

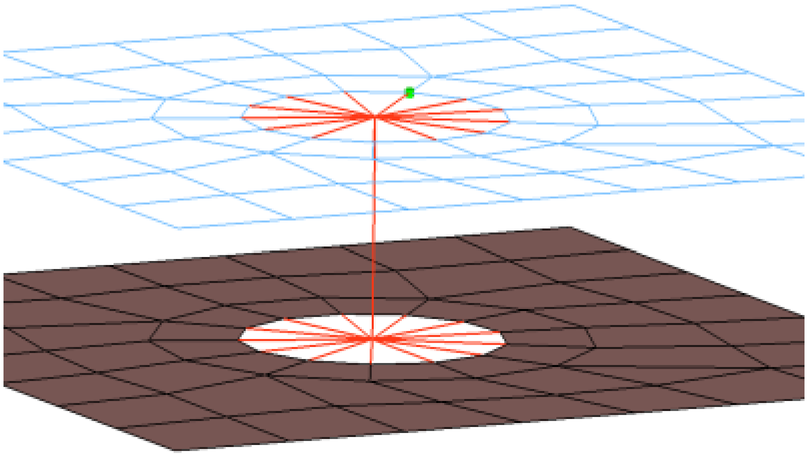


Figure 1280:

```
CFG nastran 52 bolt (general)
*filter bolt
*style bolt 0
*head
rigidlink 1 1
*body 0
rigid 1 1
```

Nastran bolt (CBAR)

Creates RBE2 elements for the head and CBAR element for the body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

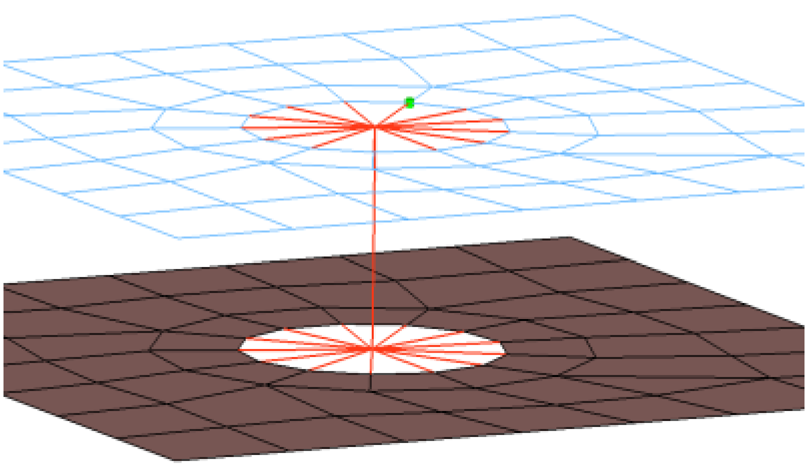


Figure 1281:

```
CFG nastran 53 bolt (CBAR)
*filter bolt
```

```
*style bolt 0
*head
rigid 1 1
*body 0
bar2 1 1
```

Nastran bolt (spider)

Creates a many individual RBE2 elements. The element projects and connect to the nodes of the adjoining shell elements which form the hole, the RBE2 elements are joined at the midpoint of the bolted connection. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

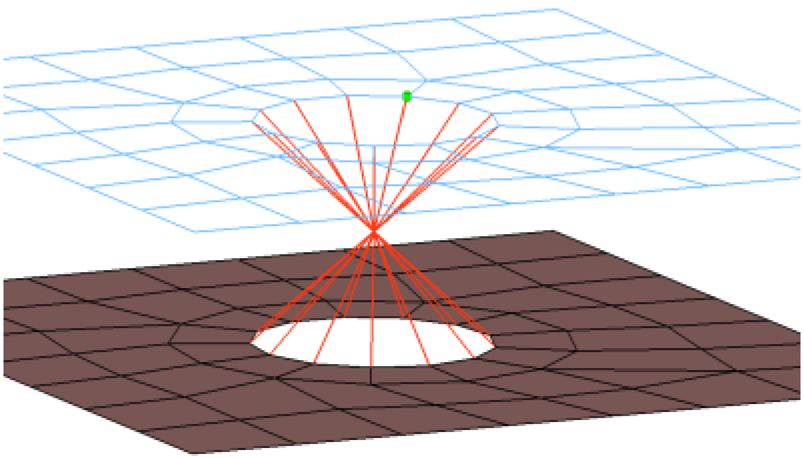


Figure 1282:

```
CFG nastran 54 bolt (spider)
*filter bolt
*style bolt 1
*head
*body 0
rigid 1 1
```

Nastran bolt (washer 2)

Creates RBE2 elements for the head and the body. There are two individual RBE2 elements at the head of the connection, one to connect to the inner row of nodes, the other to connect to the washer layer nodes. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

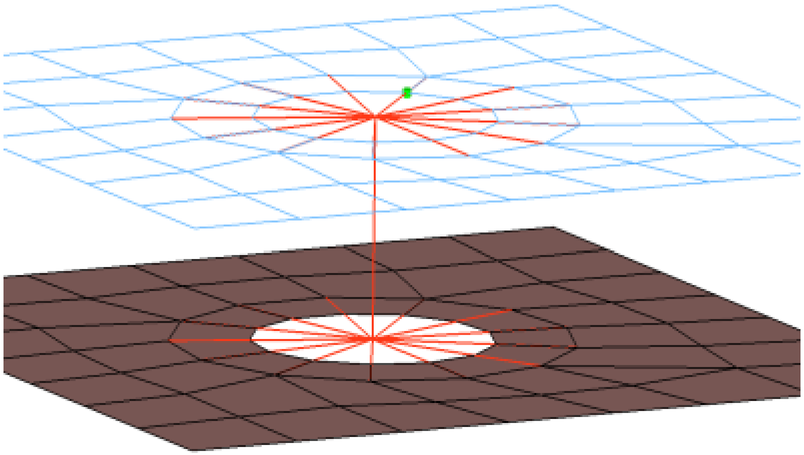


Figure 1283:

```
CFG nastran 55 bolt (washer 2)
*filter bolt
*style bolt 0
*head
rigidlink 1 1
rigidlink 1 2
*body 0
rigid 1 1
```

Nastran bolt (washer 2 alt)

Creates RBE2 elements for the head and the body. There are two individual RBE2 elements at the head of the connection, one to connect to the inner row of nodes, the other to connect to the washer layer nodes. The RBE2 head element that connects to the washer layer nodes only connects to every other node on the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

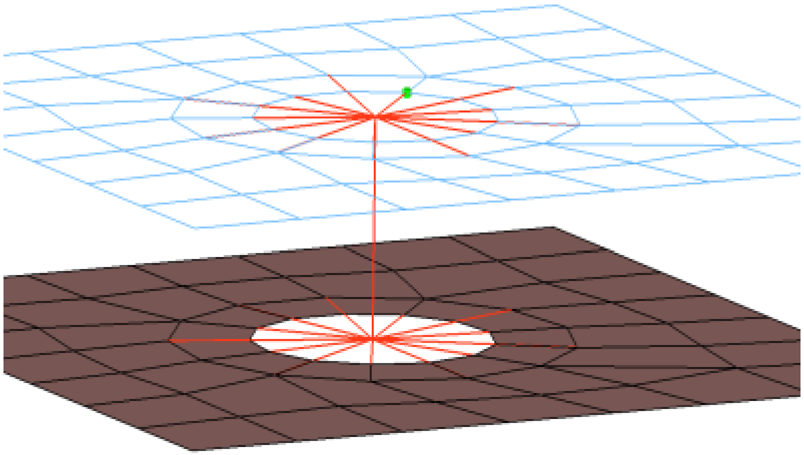


Figure 1284:

```
CFG nastran 56 bolt (washer 2 alt)
*filter bolt
*style bolt 0
*head
rigidlink 1 1
rigidlink 1 3
*body 0
rigid 1 1
cfg_nastran_56_bolt
```

Nastran bolt (washer 1)

Creates RBE2 elements for the head and body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole and also the second row of nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

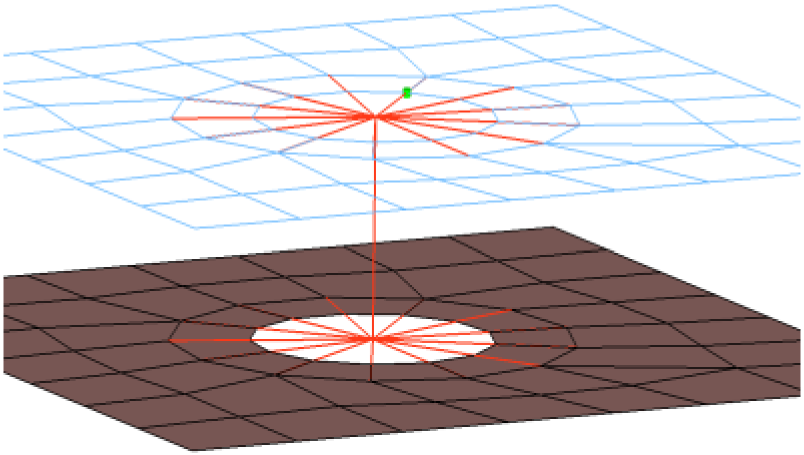


Figure 1285:

```
CFG nastran 57 bolt (washer 1)
*filter bolt
*style bolt 0
*head
rigidlink 1 12
*body 0
rigid 1 1
cfg_nastran_57_bolt
```

Nastran bolt (washer 1 alt)

Creates RBE2 elements for the head and body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole and also the second row of nodes which form the washer layer. The head only connects to every other node on the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

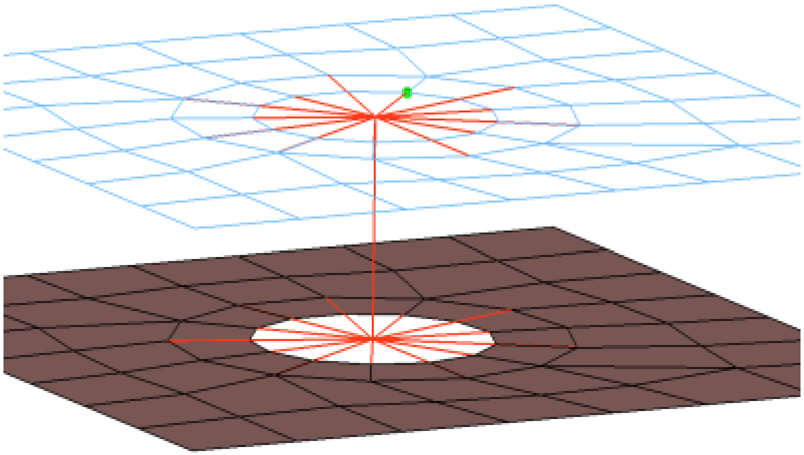


Figure 1286:

```
CFG nastran 58 bolt (washer 1 alt)
*filter bolt
*style bolt 0
*head
rigidlink 1 13
*body 0
rigid 1 1
cfg_nastran_58_bolt
```

Nastran hinge

Creates RBE2 elements for the head and the body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer. The degrees of freedom are constrained in the x, y, z, rot x, rot z for the dependant nodes.

This realization also uses the `prop_hinge.tcl`[1] property script.

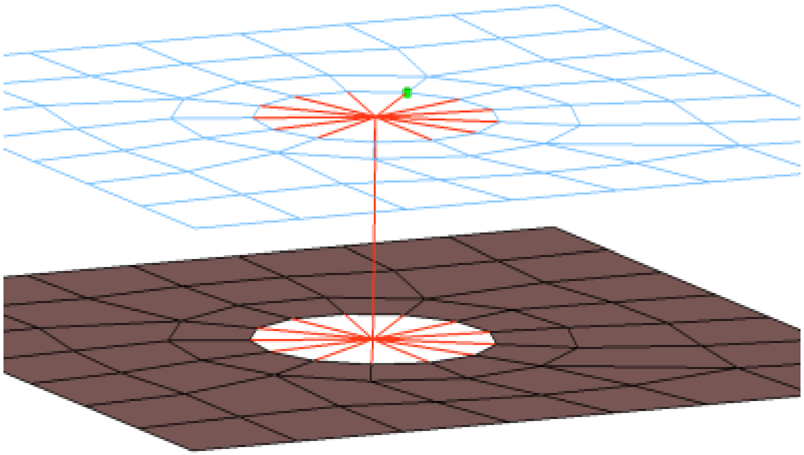


Figure 1287:

```
CFG nastran 59 hinge
*filter bolt
*style bolt 0
*head
rigidlink 1 1
*body 0
rigid 1 1 dofs=12356
*post prop_hinge.tcl
cfg_nastran_59_hinge
```

Nastran bolt (cylinder rigid)

Creates an RBE2 element for the body as well as for the head elements.

See the mesh independent realization methods in the Bolt panel for further information on cylinder-type bolts.

This realization uses the `prop_cylinder.tcl`[2] property script.

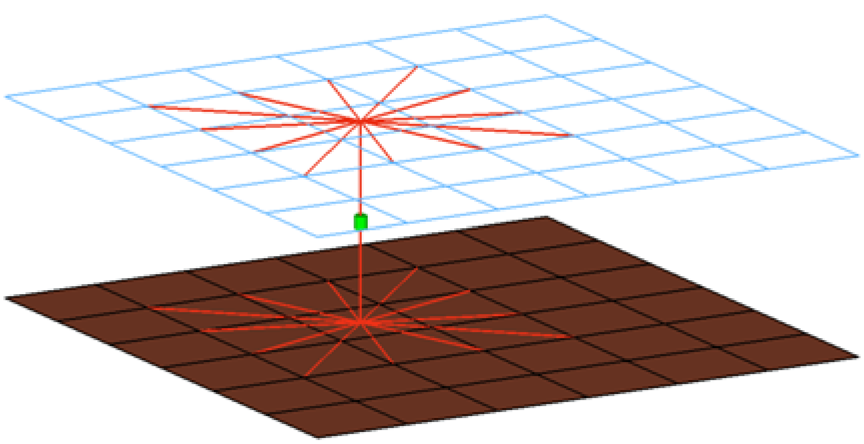


Figure 1288:

```
CFG nastran 60 bolt (cylinder rigid)
*filter bolt
*style bolt 4
```

```
*head
rigidlink 1 1
*body 0
rigid 1 1
*post prop_cylinder.tcl
cfg_nastran_60_bolt_cylinder_rigid
```

Nastran bolt (cylinder bar)

Creates a CBEAM element for the body and RBE2 elements for the head elements.
See the mesh independent realization methods in the Bolt panel for further information on cylinder-type bolts.

This realization uses the `prop_cylinder.tcl`[2] property script.

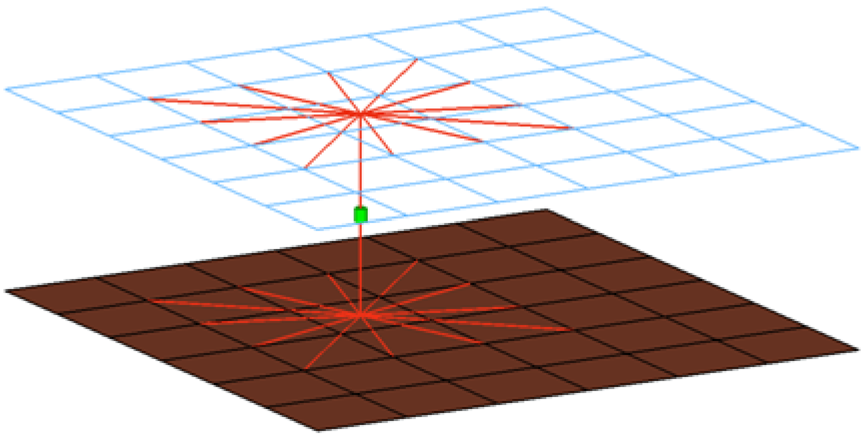


Figure 1289:

```
CFG nastran 61 bolt (cylinder bar)
*filter bolt
*style bolt 4
*head
rigidlink 1 1
*body 0
bar2 2 1
*post prop_cylinder.tcl
cfg_nastran_61_bolt_cylinder_bar
```

Nastran acm (equivalenced-(T1+T2)/2)

Creates hexa element with RBE3 elements projecting and connecting to the surrounding shell elements. This realization uses the shell thickness to calculate the hexa offset from the shell elements. In the case where the model is a 3T connection, the acm (equivalenced-(T1+T2)/2) realization will join the hexa elements.

This realization also uses the `prop_nastran_acm.tcl`[3] property script.

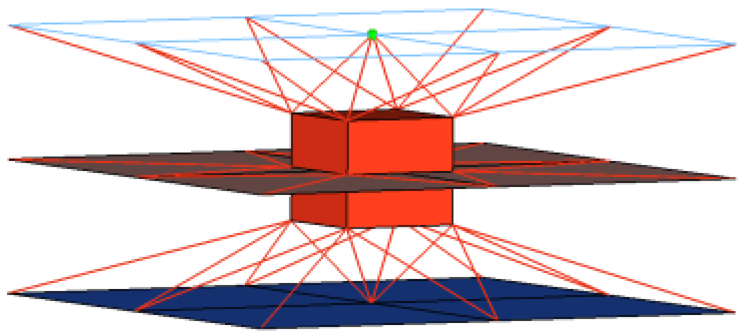


Figure 1290:

```
CFG nastran 69 acm (equivalenced-(T1+T2)/2)
*filter spot
*style acm 1
*head
rbe3 1 0
*body 0
hex8 1 1
*post prop_nastran_acm.tcl
cfg_nastran_69_acm
```

Nastran acm (detached-(T1+T2)/2)

Creates hexa element with RBE3 elements projecting and connecting to the surrounding shell elements. This realization uses the shell thickness to calculate the hexa offset from the shell elements. In the case where the model is a 3T connection, the acm (detached-(T1+T2)/2) realization will not join the hexa elements.

This realization also uses the `prop_nastran_acm.tcl`[3] property script.

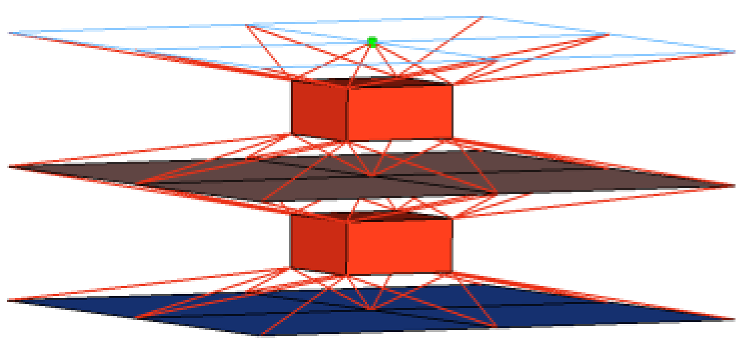


Figure 1291:

```
CFG nastran 70 acm (detached-(T1+T2)/2)
*filter spot
*style acm 2
*head
rbe3 1 0
*body 1
```

```
hex8 1 1
*post prop_nastran_acm.tcl
cfg_nastran_70_acm
```

Nastran acm (shell gap)

Creates hexa element with RBE3 elements projecting and connecting to the surrounding shell elements. This realization does not use the shell thickness to calculate the hexa offset, therefore the hexa will project and be touching the shell elements.

This realization also uses the `prop_nastran_acm.tcl`[3] property script.

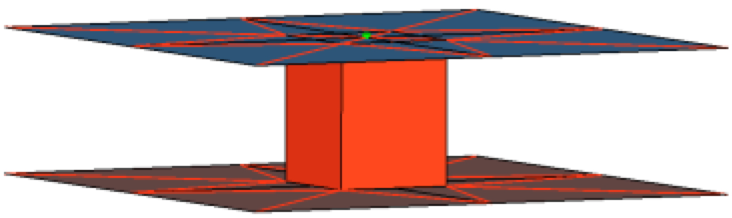


Figure 1292:

```
CFG nastran 71 acm (shell gap)
*filter spot
*style acm 3
*head
rbe3 1 0
*body 0
hex8 1 1
*post prop_nastran_acm.tcl
cfg_nastran_71_acm
```

Nastran acm (shell gap + coating)

This realization creates one hexa cluster per connector and realizes a node to node connection to the linked shell meshes by adjusting it (shell coating). Different patterns are available. This is driven by the number of hexas. The appearance can be influenced via the diameter and the washer layer activation.

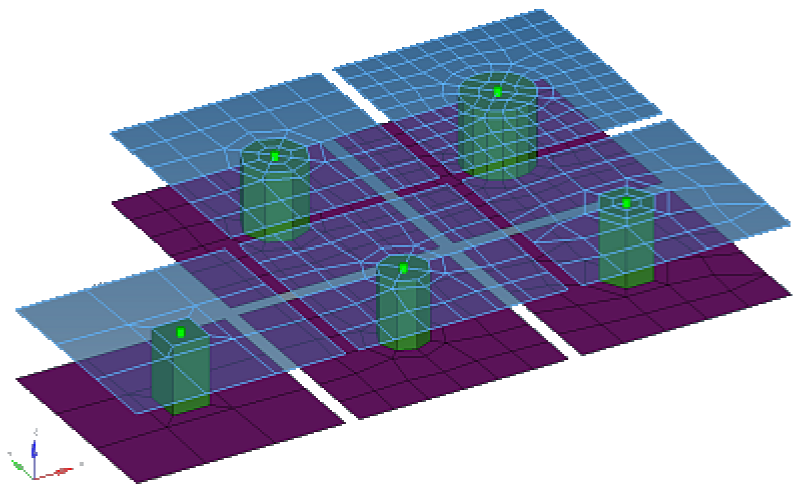


Figure 1293:

```
CFG nastran 72 acm (shell gap + coating)
*filter spot
*style acm 4
*body 0
hex8 1 1
acm_shellgap_coating_2
```

Nastran pie (rigid spider)

This realization prepares a circled shell mesh from a certain number of segments for each link, so the mesh is adjusted to a rigid element created with the independent node centered in the circular arranged dependent nodes. The independent nodes themselves are connected by an additional rigid element.

Different numbers of elements lead to a different pattern. In addition, the appearance can be influenced via the diameter and the washer layer activation.

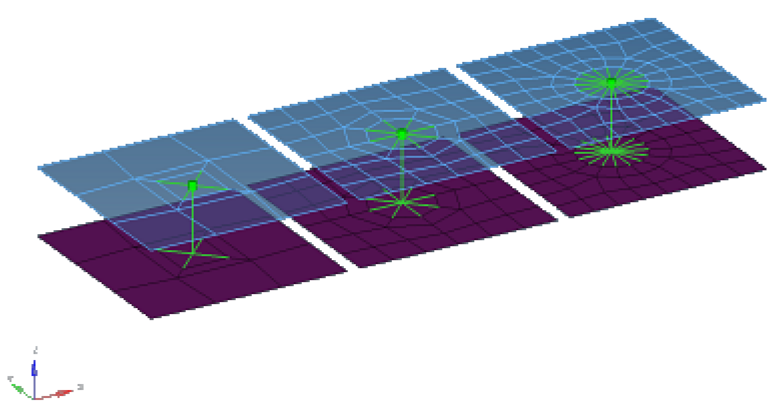


Figure 1294:

```
CFG nastran 73 pie (rigid spider)
```

```
*filter spot
*head
rigidlink 1 4
*body 0
rigid 1 1
pie_rigid_spider_processed
```

Nastran acm (general)

This realization type consolidates several ACM definitions into one general, flexible ACM definition. Besides mid thickness, constant thickness, and maintain gaps, the definition of several coats with different hexa pattern is available. Each hexa node pointing towards the link elements is connected to the links by using RBE3 elements. The independent nodes get different weighting factors. See the formulas in the `prop_nastran_acm.tcl`[3] property script.

The realization also uses the `prop_nastran_acm.tcl`[3] property script.

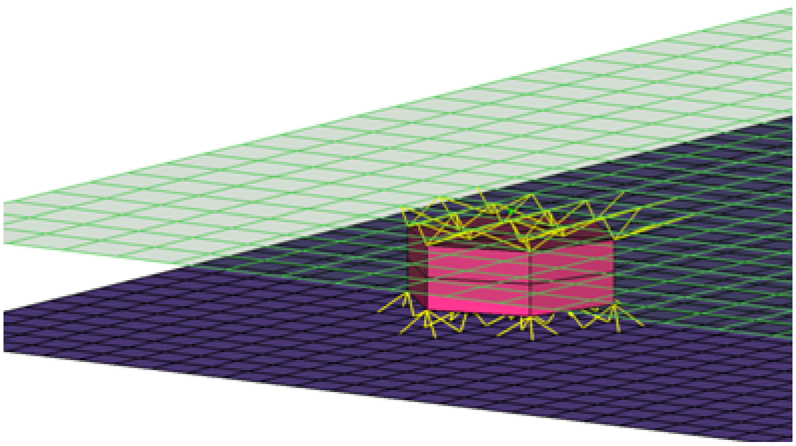


Figure 1295:

```
CFG nastran 74 acm (general)
*filter spot
*style acm 3
*head
rbe3 1 0 dofs=123
*body 0
hex8 1 1
*post prop_nastran_acm.tcl
acm_general2
```

Nastran penta (mig+L)

This realization supports Lap-joints and creates PENTA element for the body. Surrounding shell/solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint. A fitted/equilateral option is also provided for the PENTA creation.

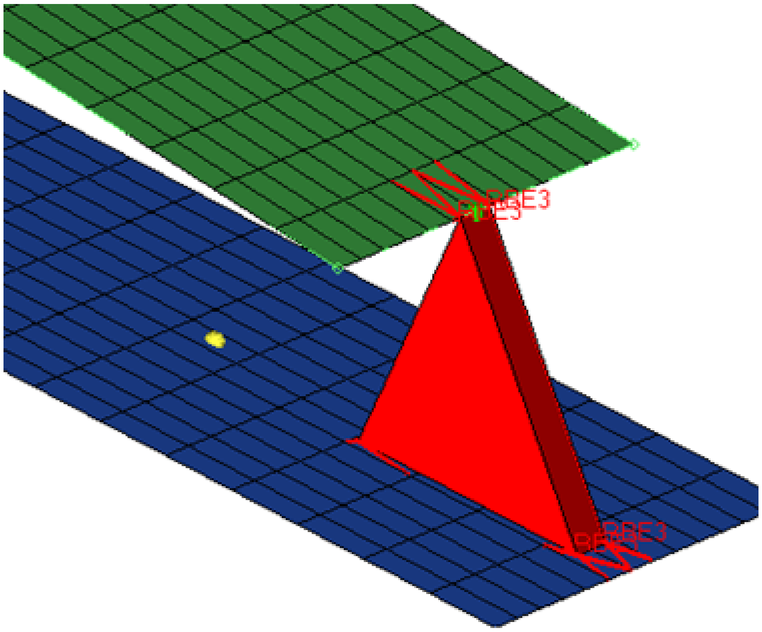


Figure 1296:

```
CFG nastran 76 penta (mig + L)
*filter spot
*style mig 1
*head
rbe3 1 0
*body 0
penta6 1 1
```

Nastran penta (mig+T)

This realization supports T-joints and creates PENTA element for the body. Surrounding shell/solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint, and provides a right-angled option.

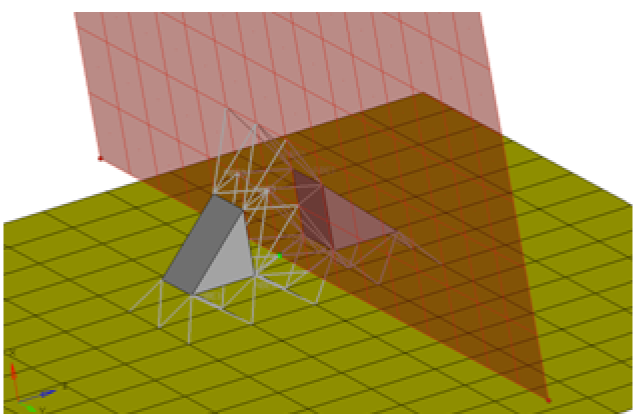


Figure 1297:

```
CFG nastran 77 penta (mig + T)
*filter spot
*style mig 2
```

```
*head
rbe3 1 0
*body 0
penta6 1 1
```

Nastran penta (mig+B)

This realization supports Butt-joints and creates PENTA element for the body. Surrounding shell/ solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint. The non-normal option needs to be ON/Active for this realization.

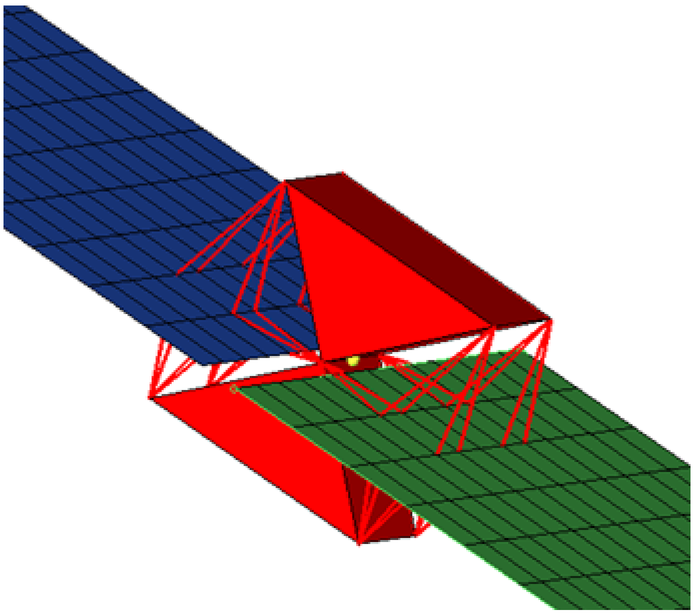


Figure 1298:

```
CFG nastran 78 penta (mig + B)
*filter spot
*style mig 3
*head
rbe3 1 0
*body 0
penta6 1 1
cfg_nastran_78_penta_mig_b
```

Nastran cweld (GA-GB PARTPAT)

Creates 1D CWELD element via GA-GB PARTPAT.

This realization also uses the `prop_cweld.tcl`[4] property script.

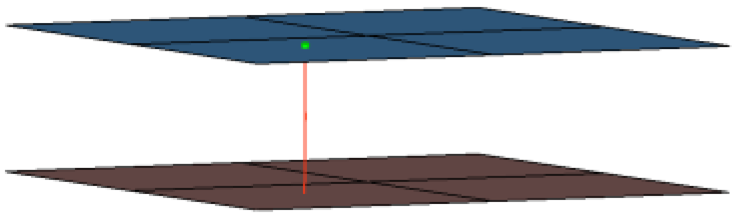


Figure 1299:

```
CFG nastran 80 cweld (GA-GB PARTPAT)
*filter spot
*head
*body 0
rod 4 1
*post prop_cweld.tcl
cfg_nastran_80_cweld
```

Nastran cweld (GA-GB ELPAT)

Creates 1D CWELD element via GA-GB ELPAT.

This realization also uses the `prop_cweld.tcl`[4] property script.

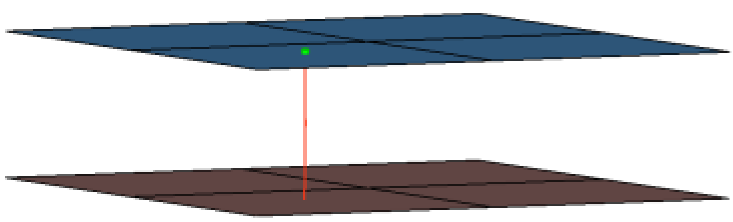


Figure 1300:

```
CFG nastran 82 cweld (GA-GB ELPAT)
*filter spot
*head
*body 0
rod 4 1
*post prop_cweld.tcl
cfg_nastran_82_cweld
```

Nastran cweld (GS ELPAT)

Creates 0D CWELD element via GS ELPAT.

This realization also uses the `prop_cweld.tcl`[4] property script.

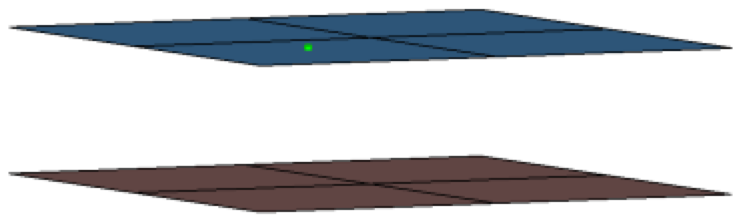


Figure 1301:

```
CFG nastran 83 cweld (GS ELPAT)
*filter spot
*head
*body 0
mass 11 0
*post prop_cweld.tcl
cfg_nastran_83_cweld
```

Nastran cweld (GA-GB ELEMID)

Creates 1D CWELD element via GA-GB ELEMID.

This realization also uses the `prop_cweld.tcl`[4] property script.

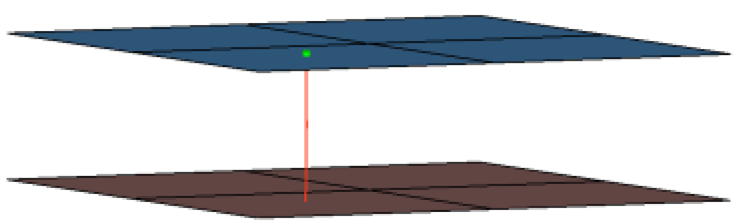


Figure 1302:

```
CFG nastran 84 cweld (GA-GB ELEMID)
*filter spot
*head
*body 0
rod 4 1
*post prop_cweld.tcl
cfg_nastran_84_cweld
```

Nastran cweld (GS ELEMID)

Creates 0D CWELD element via GS ELEMID.

This realization also uses the `prop_cweld.tcl`[4] property script.

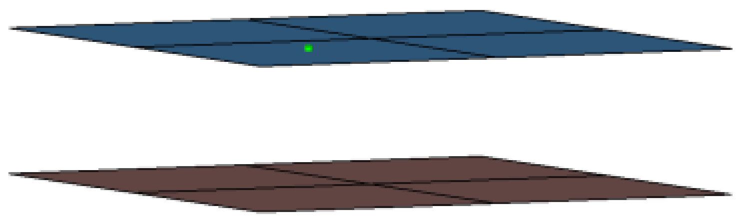


Figure 1303:

```
CFG nastran 85 cweld (GS ELEMID)
*filter spot
*head
*body 0
mass 11 0
*post prop_cweld.tcl
cfg_nastran_85_cweld
```

Nastran cweld (GA-GB GRIDID)

Creates 1D CWELD element via GA-GB GRIDID.

This realization also uses the `prop_cweld.tcl`[4] property script.

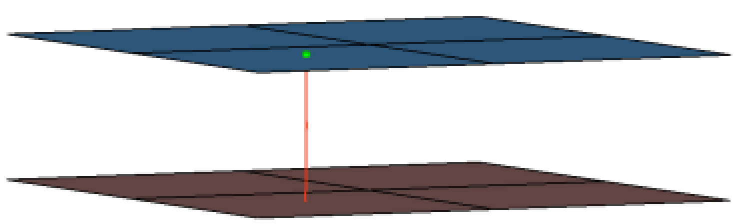


Figure 1304:

```
CFG nastran 86 cweld (GA-GB GRIDID)
*filter spot
*head
*body 0
rod 4 1
*post prop_cweld.tcl
cfg_nastran_86_cweld
```

Nastran cweld (GS GRIDID)

Creates 0D CWELD element via GS GRIDID.

This realization also uses the `prop_cweld.tcl`[4] property script.

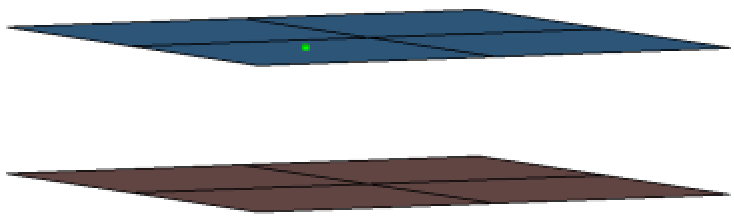


Figure 1305:

```
CFG nastran 87 cweld (GS GRIDID)
*filter spot
*head
*body 0
mass 11 0
*post prop_cweld.tcl
cfg_nastran_87_cweld
```

Nastran cweld (GA-GB ALIGN)

Creates 1D CWELD element via GA-GB ALIGN.

This realization also uses the `prop_cweld.tcl`[4] property script.

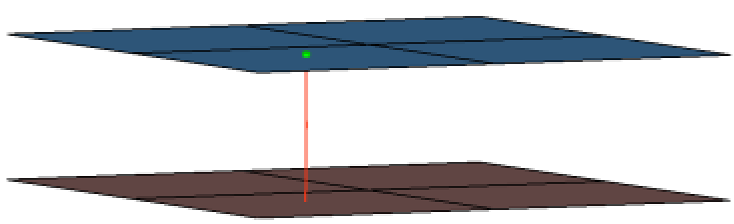


Figure 1306:

```
CFG nastran 88 cweld (GA-GB ALIGN)
*filter spot
*head
*body 0
rod 4 1
*post prop_cweld.tcl
cfg_nastran_88_cweld
```

Nastran rbe3-celas1-rbe3

Creates RBE3 element for the head and zero length CELAS1 element for the body. The head elements project and connect to the nodes of the adjoining shell elements. The degrees of freedom are constrained in the x, y, z, rot x, rot y, rot z for the dependant nodes.

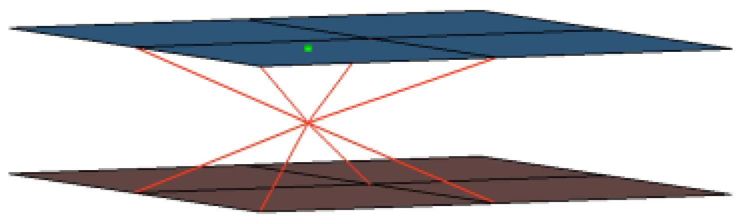


Figure 1307:

```
CFG nastran 89 rbe3-celas1-rbe3
*filter spot
*head
rbe3 1 0 dofs=123456
*body 0
spring 1 0
cfg_nastran_89_rbe3_celas1_rbe3
```

Nastran seam-quad (angled+capped+L)

Creates a quad row with tria caps at the seam ends. In addition, a certain pure quad element pattern is created around the seam elements, shown here in red. These elements normally get imprinted into the shell links. The exact geometry of the seam can be influenced by certain attributes in the Seam panel.

This realization is mainly intended to be used for lap welds. You can revert the direction of quad seam connectors during the next realization by activating the reverse direction check box in the Seam panel.

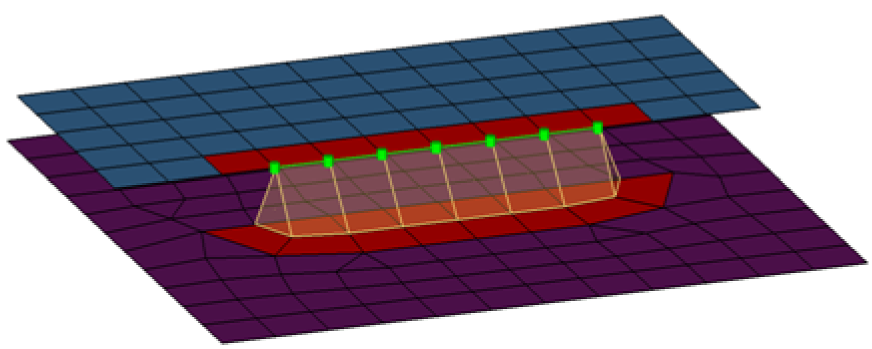


Figure 1308:

```
CFG nastran 101 seam-quad (angled+capped+L)
*filter seam
*style quad 4
*head
*body 0
quad4 1 1
```

Nastran seam-quad (angled+capped+T)

Creates a quad row with tria caps at the seam ends. In addition, a certain pure quad element pattern is created around the seam elements, shown here in red. These elements normally get

imprinted into the shell links. The exact geometry of the seam can be influenced by certain attributes in the Seam panel.
This realization is mainly intended to be used for lap welds. You can revert the direction of quad seam connectors during the next realization by activating the reverse direction check box in the Seam panel.

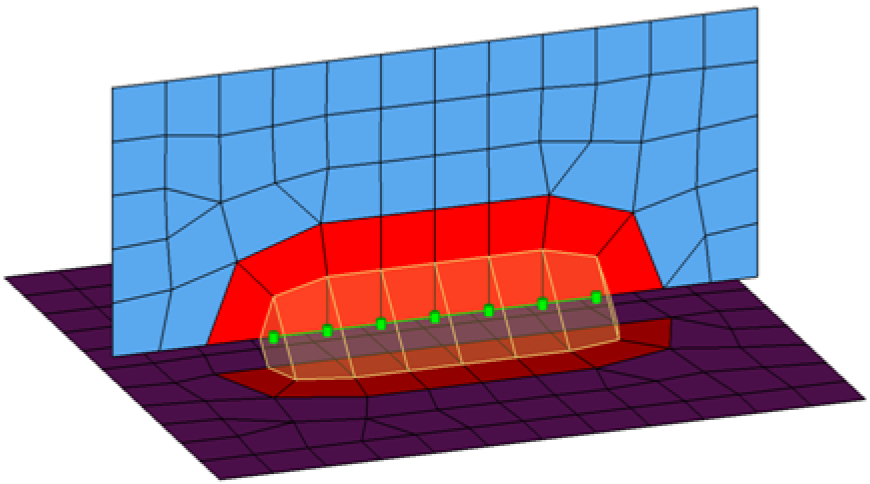


Figure 1309:

```
CFG nastran 102 seam-quad (angled+capped+T)
*filter seam
*style quad 5
*head
*body 0
quad4 1 1
```

Nastran seam-quad (vertical+angled)

Creates two quad rows-the first one perpendicular to the opposite shell link, and the second one with a certain angle to the first one. In addition, a certain pure quad element pattern is created around the seam elements, shown here in red. These elements normally get imprinted into the shell links. The exact geometry of the seam can be influenced by the angle value.
This realization is can be used for both lap- and T-welds.

> **Note:** You can revert the direction of quad seam connectors during the next realization by activating the reverse direction check box in the Seam panel.

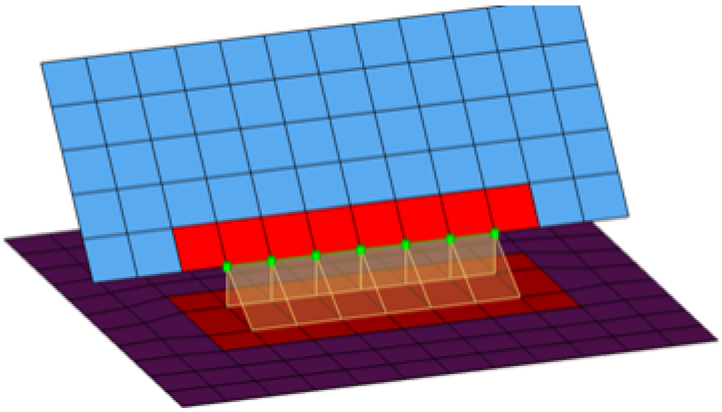


Figure 1310:

```
CFG nastran 103 seam-quad (vertical+angled)
*filter seam
*style quad 1
*head
*body 0
quad4 1 1
```

Nastran seam-quad (angled)

Creates one quad row under a certain angle. The angle is measured between the quad row and the perpendicular projection from the free edge to the opposite shell link. In addition, a certain pure quad element pattern is created around the seam elements, shown here in red. These elements normally get imprinted into the shell links. The exact geometry of the seam can be influenced by the angle value.

This realization is can be used for both lap- and T-welds.

> **Note:** You can revert the direction of quad seam connectors during the next realization by activating the reverse direction checkbox in the Seam panel.

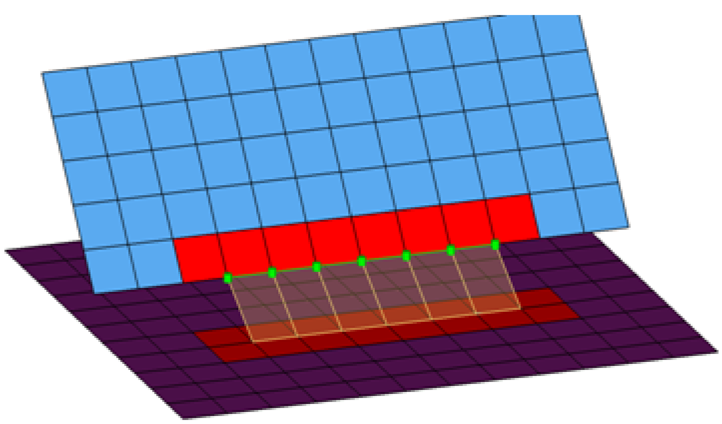


Figure 1311:

```
CFG nastran 104 seam-quad (angled)
*filter seam
```

```
*style quad 2
*head
*body 0
quad4 1 1
```

Nastran penta (mig)

Creates penta elements with RBE3 elements projecting and connecting to the surrounding shell elements. This realization supports many different use cases, including T-joint, angled T-joint, lap joint and butt joint.

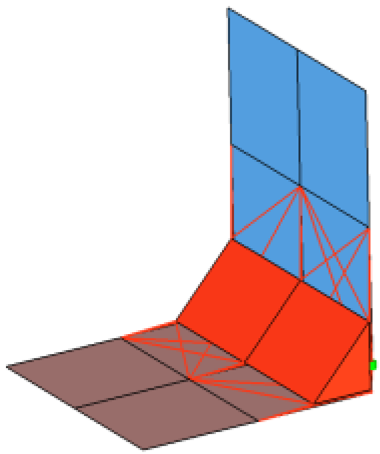


Figure 1312:

```
CFG nastran 105 penta (mig)
*filter seam
*style continuous 3
*head
rbe3 1 0
*body 0
penta6 1 1
cfg_nastran_105_penta
```

Nastran hexa (adhesive)

Creates a row of hexa elements for the body and numerous RBE2/RBE3 elements for the head. The head elements project and connect to the nodes of the adjoining shell elements. If there is a direct normal project then an RBE2 elements will be used, if there are only non-normal projections then RBE3 elements will be created. The hexa elements are projected so that they touch the shell elements of the connecting components.

This realization also uses the `prop_nastran_acm.tcl`[3] property script.

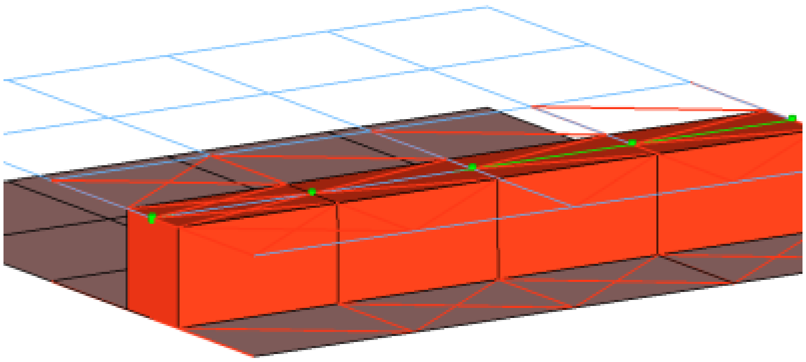


Figure 1313:

```
CFG nastran 106 hexa (adhesive)
*filter seam
*style continuous 3
*head
rbe3 1 0
rigid 1 0
*body 0
hex8 1 1
```

Nastran cfast_elem (GA-GB)

Creates 1D CFAST element of type ELEM.

This realization also uses the `prop_opt_nas_cfast.tcl`[5] property script.

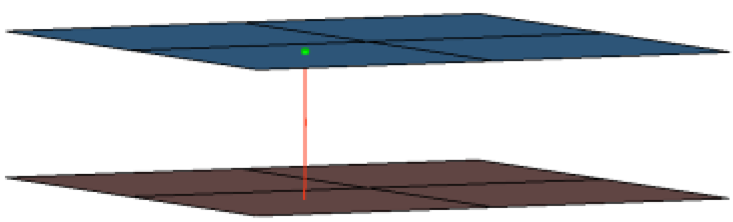


Figure 1314:

```
CFG nastran 107 cfast_elem (GA-GB)
*filter spot
*head
*body 0
rod 7 1
*post prop_opt_nas_cfast.tcl
cfg_nastran_107_cfast
```

Nastran cfast_elem (GS)

Creates 0D CFAST element of type ELEM.

This realization also uses the `prop_opt_nas_cfast.tcl`[5] property script.

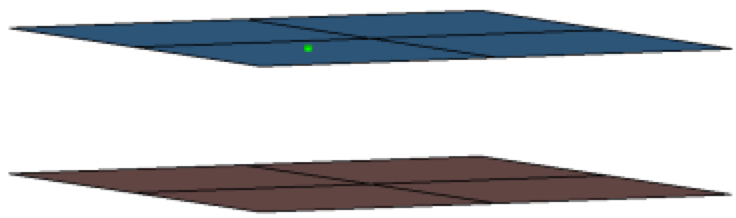


Figure 1315:

```
CFG nastran 108 cfast_elem (GS)
*filter spot
*head
*body 0
mass 23 0
*post prop_opt_nas_cfast.tcl
```

Nastran cfast_prop (GA-GB)

Creates 1D CFAST element of type PROP.

This realization also uses the `prop_opt_nas_cfast.tcl`[5] property script.

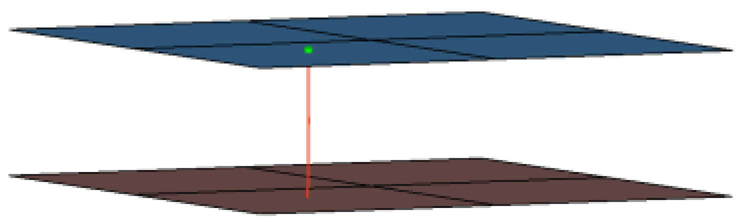


Figure 1316:

```
CFG nastran 109 cfast_prop (GA-GB)
*filter spot
*head
*body 0
rod 7 1
*post prop_opt_nas_cfast.tcl
cfg_nastran_109_cfast
```

Nastran cfast_prop (GS)

Creates 0D CFAST element of type PROP.

This realization also uses the `prop_opt_nas_cfast.tcl`[5] property script.

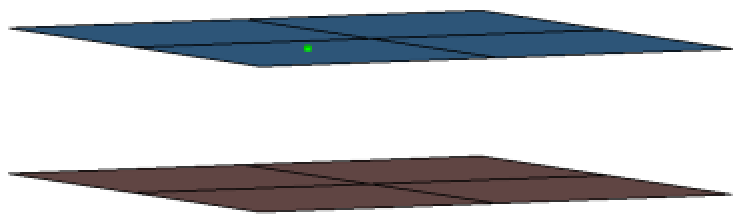


Figure 1317:

```
CFG nastran 110 cfast_prop (GS)
*filter spot
*head
*body 0
mass 23 0
*post prop_opt_nas_cfast.tcl
cfg_nastran_110_cfast
```

Nastran HILOCK

Creates 1D element construct existing out of RBAR, CBAR and CBUSH elements. The outer extensions represent the thicknesses of the outer shell elements. The inner nodes of the RBAR element are connected to the shell elements whereas the inner nodes of the CBAR elements are coincident to the shell nodes only. Between the appropriate connected and coincident nodes CBUSHes are created. Each outer node connects one CBAR and one RBAR. Each HILOCK connection gets an own coordinate system which z-axis is collinear to the HILOCK direction. All affected nodes are assigned to this coordinate system. This coordinate system is taken into account for the DOF definition of the CBAR elements, for the stiffness calculation of the CBUSH elements and for the DOF of the node constraint.
This realization uses the shell properties and materials (PSHELL or PCOMP) and the HILOCK material you select to calculate the exact position of the outer nodes and the stiffness of the PBUSH elements.

This realization also uses the `prop_opt_nas_hilock.tcl`[6] property script.

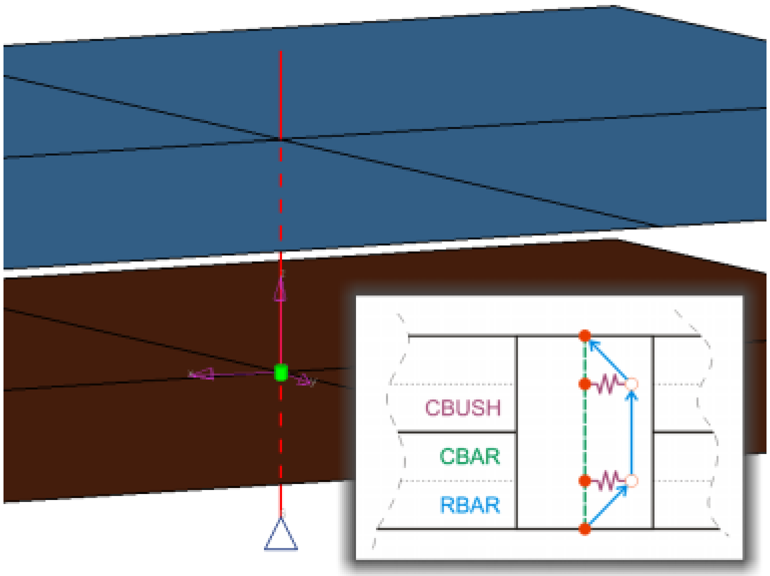


Figure 1318:

```
CFG nastran 111 HILOCK
*filter spot
*style fastener 1
*head
*bodyext 0
bar2 1 1
weld 1 1 dofs=1456
*body 0
spring 6 0 dofs=2356
bar2 1 1
weld 1 1 dofs=156
spring 6 0 dofs=2356
*post prop_opt_nas_hilock.tcl
cfg_nastran_111_HiLock
```

Nastran clip (washer nodes)

Creates a single RBE2 element for the body. The element projects and connects to the nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

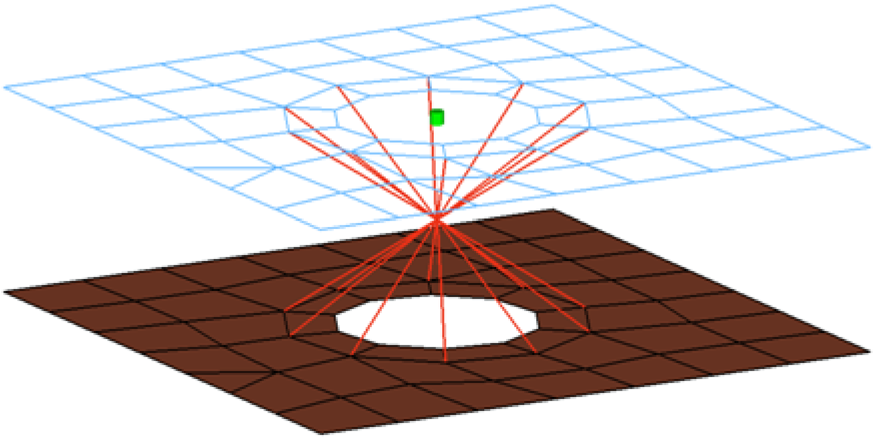


Figure 1319:

```
CFG nastran 112 clip (washer nodes)
*filter bolt
*style bolt 12
*head
*body 0
rigidlink 1 1
```

Nastran bolt (step hole)

This realization creates a CBAR element for the bolt shaft, and connects to the solids' nodes with numerous RBE2 based on the given bolt/hole parameters. It also, connects two solids through holes, or it connects one solid through a hole with a solid blind hole.

This realization uses the `prop_stepboltholes.tcl`[7] property script.

Figure 1320:

```
CFG nastran 114 bolt (step hole)
*filter bolt
*style bolt 6
*head
rigidlink 1 1
*body 0
```

```
bar2 1 1
*post prop_stepboltholes.tcl
```

Nastran bolt (threaded step hole)

Connects two solids through holes or connects one solid through a hole with a solid blind hole. A thread length can be defined to define the dimensions of the rigid elements connecting the bolt shaft models as a bar.

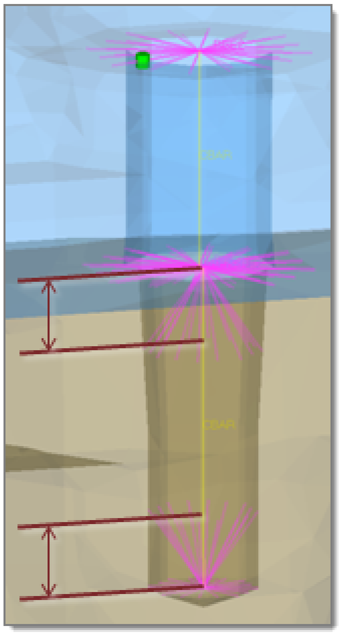


Figure 1321:

```
CFG nastran 115 bolt (threaded step hole)
*filter bolt
*style bolt 7
*head
rigidlink 1 1
*body 0
bar2 1 1
*post prop_stepboltholes.tcl
```

Nastran adhesive-hemmings

This realization type is used for modeling roll hemmings, where the outer shell is bent around the inner shell. The inner shell is connected to the outer shell on one side with simple hexa adhesive, and the other side is connected with RBE2 elements. A definable orientation node decides which side the hexa adhesive should be used. This seam realization type is capable of connecting three layers that contains two components.

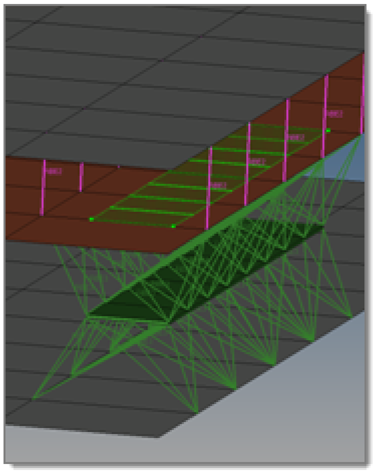


Figure 1322:

```
CFG nastran 116 adhesive-hemmings
*filter area
*style adhesive 3
*head
rbe3 1 0
*body 1
hex8 1 1
rigid 1 0
*post prop_nastran_acm.tcl
```

Nastran penta continuous (mig+L)

This realization supports Lap-joints and creates PENTA element for the body. Surrounding shell/solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint, and provides a fitted/equilateral option for the PENTA creation.

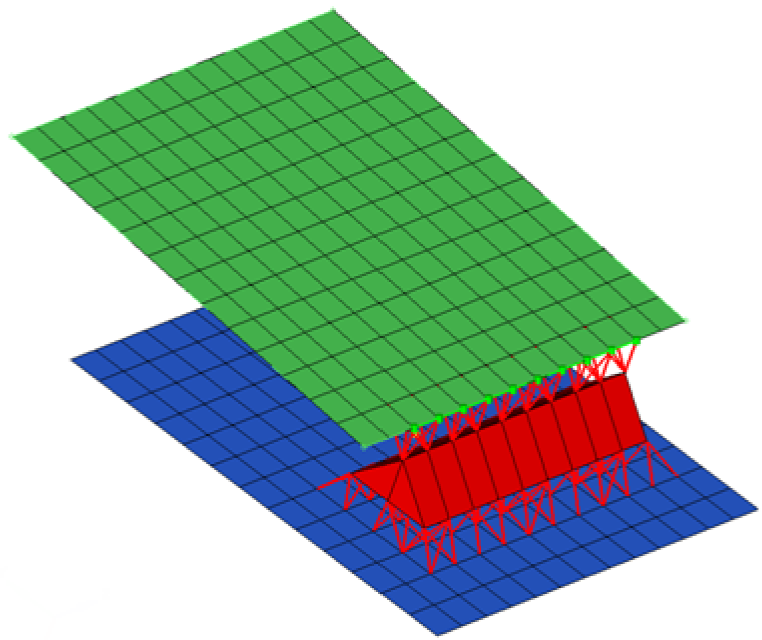


Figure 1323:

```
CFG nastran 117 penta (mig + L)
*filter seam
*style continuous_mig 1
*head
rbe3 1 0
*body 0
penta6 1 1
```

Nastran penta continuous (mig+T)

This realization supports T-joints and creates PENTA element for the body. Surrounding shell/solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint, and provides a right-angled option.

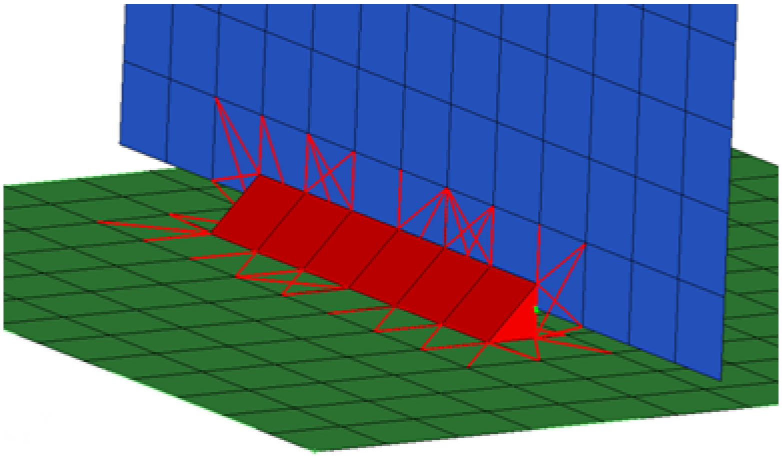


Figure 1324:

```
CFG nastran 118 penta (mig + T)
*filter seam
*style continuous_mig 2
*head
rbe3 1 0
*body 0
penta6 1 1
```

Nastran penta continuous (mig+B)

This realization supports Butt-joints and creates PENTA element for the body. Surrounding shell/solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint.

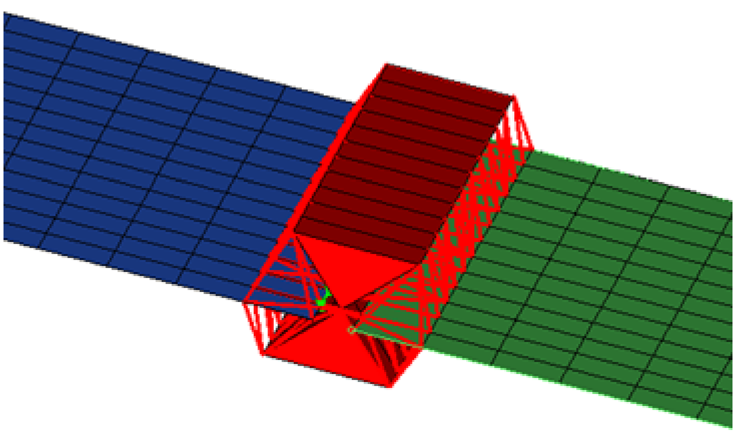


Figure 1325:

```
CFG nastran 119 penta (mig + B)
*filter seam
*style continuous_mig 3
*head
rbe3 1 0
```

```
*body 0
penta6 1 1
```

Nastran wagonwheel

This realization creates RBE2 elements for the body, and projects and connects to the hole edge nodes with RBE3 elements.

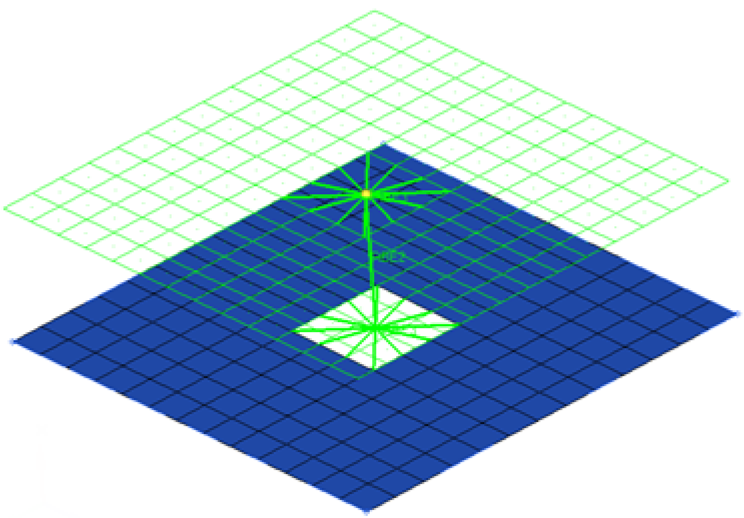


Figure 1326:

```
CFG nastran 120 wagonwheel
*filter bolt
*style bolt 0
*head
rbe3 1 0
*body 0
rigid 1 1
cfg_nastran_120_wagonwheel
```

Nastran adhesives

This realization also uses the `prop_nastran_acm.tcl`[3] property script.

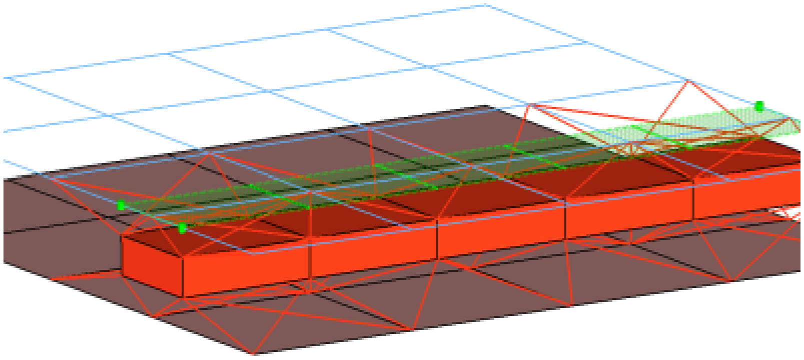


Figure 1327:

```
CFG nastran 121 adhesives
*filter area
*style adhesive 1
*head
rbe3 1 0
rigid 1 0
*body 1
hex8 1 1
penta6 1 1
*post prop_nastran_acm.tcl
cfg_nastran_121_adhesives
```

Nastran hemming

Creates RBE3 elements for the body, the head elements project and connect to the nodes of the adjoining shell elements.

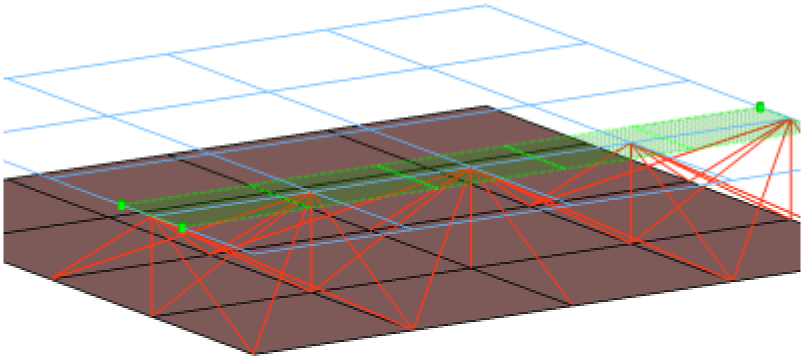


Figure 1328:

```
CFG nastran 122 hemming
*filter area
*style adhesive 1
*head
*body 0
rbe3 1 1
```

Nastran bolt (collapse Rigid)

This realization creates a single RBE2 element for the body. The element projects and connect to the nodes of the adjoining shell/solid elements which form the hole.

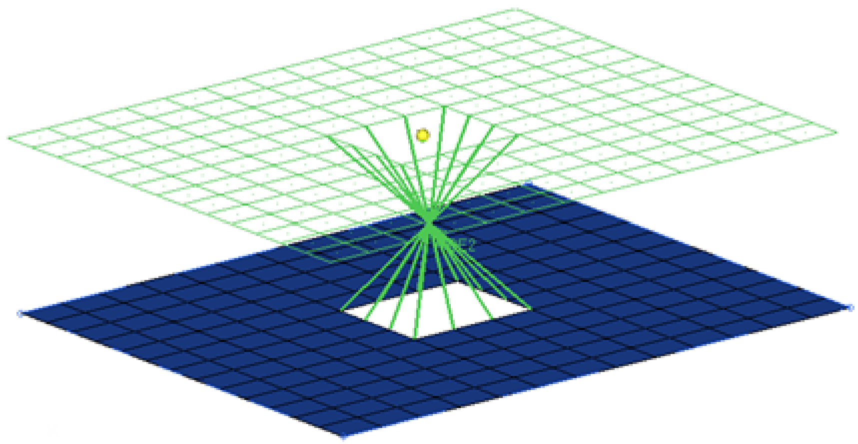


Figure 1329:

```
CFG nastran 123 bolt (collapse Rigid)
*filter bolt
*style bolt 14
*head
*body 0
rigidlink 1 12
```

Nastran seam-quad LTB

Serves and realizes t-welds, lap-welds and butt-welds simultaneously. The weld type is identified automatically based on the orientation of the links to each other.
The dimensions and property for all heat affected zones (HAZ) can be defined separately.
Normal directions of quad weld elements and HAZ elements can be controlled.
An edge treatment can be defined for t-welds and butt-welds to move the edge a precise distance from the opposite link.

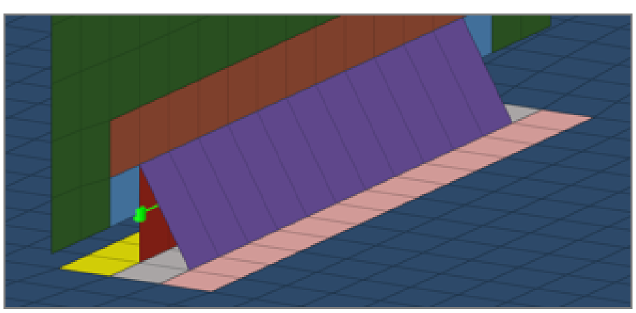


Figure 1330: T-weld

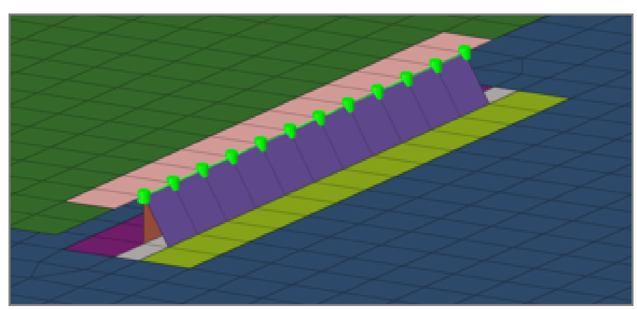


Figure 1331: Lap-weld

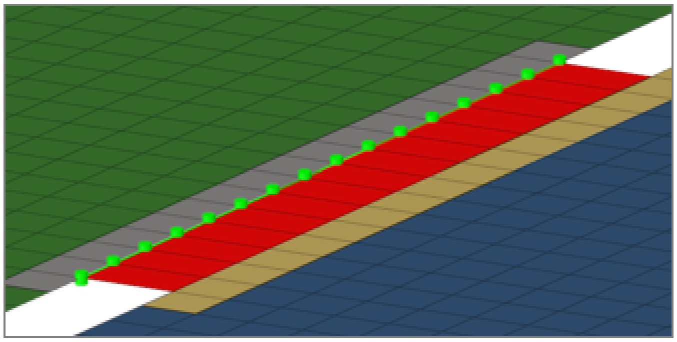


Figure 1332: Butt-weld

```
CFG nastran 128 seam-quad LTB
*filter seam
*style quad 7
*head
*body 0
quad4 1 1
```

Nastran seam-rigid LTB

Serves and realizes t-welds, lap-welds and butt-welds at the same time. The weld type is identified automatically based on the orientation of the links to each other.
The dimensions and property for all heat affected zones (HAZ) can be defined separately.
An edge treatment can be defined for t-welds and butt-welds to move the edge a precise distance from the opposite link.

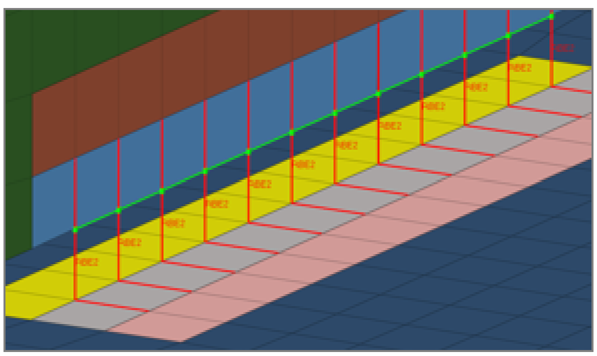


Figure 1333: T-weld

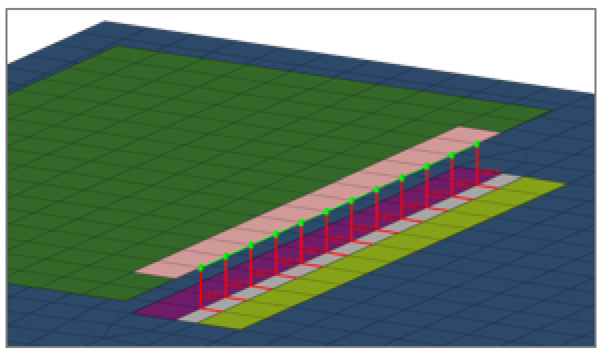


Figure 1334: Lap-weld

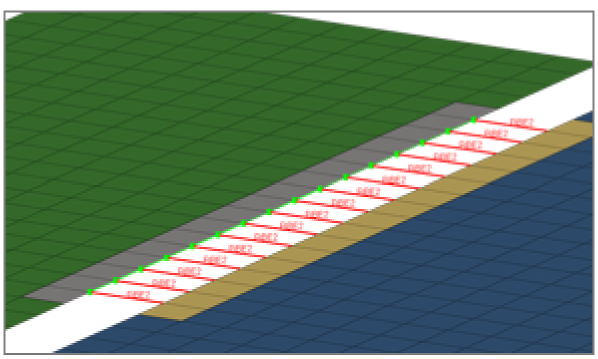


Figure 1335: Butt-weld

```
CFG nastran 129 seam-rigid LTB
*filter seam
*style rigid 1
*head
*body 0
rigid 1 1
```

Nastran cbush (rigid)

Creates bush (CBUSH) elements between shell and/or solid elements in order to connect them using rigid (RBE3) elements. The bush element nodes will project and touch the shell and/or solid element faces. Unless defined differently, the bushs are assigned a default property, and are organized into a component with the same name base as the property. If no specific bush coordinate system is defined, the bush elements are defined with a vector x1, x2, and x3 normal to it.

The default property parameters can be changed in the files below this path: `..\Altair\2019\hm\scripts\connectors\Bush_Rigid\nastran\`.

Note: IDs, names, and card type cannot be changed.

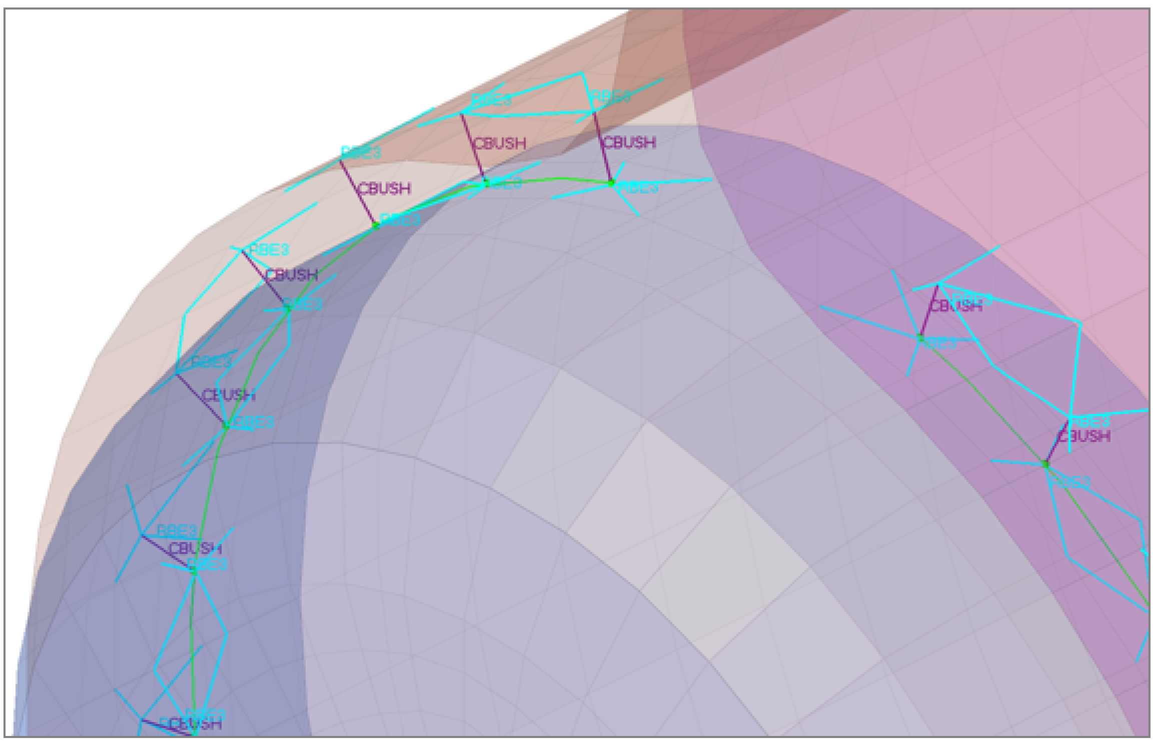


Figure 1336:

```
CFG nastran 156 cbush (rigid)
*filter spot
*style bush_rigid 1
*head
rbe3 1 0
*body 0
spring 6 1
```

Automatic Exclusion of Special Nodes During Rigid Bolt Realization

HyperMesh automatically excludes special nodes as potential slave nodes for any rigid bodies created during bolt realization, even though they fall with the virtual Bolt Cylinder diameter. Nodes that are referred in the constraints are considered special nodes.

Property Scripts

1. `prop_hinge.tcl`

   This script is called while creation of HINGE– custom config welds in the connector bolts panel. This script performs the tasks when the Systems option is active in the Connector Bolt panel, such as "Single System","1- System per layer" or 2- Systems per layer.

   This Script Assigns both reference and analysis systems ID to weld element nodes of each Bolt (Hinge) created during realization process.

2. `prop_cylinder.tcl`

   Used while creating bolt (cylinder rigid) and bolt (cylinder bar) in the Bolt panel (Abaqus, Nastran, OptiStruct). It organizes the realized bolt elements into the respective components based upon the*HEAD and the *BODYinformation of the bolt:

   - A collector with the name Rigid_M<diameter> is created. This component contains all of the rigid head elements and the rigid body elements, if available.
   - A collector with the name Beam_M<diameter> is created. This component contains all of the bar2 head elements, if available. This component then gets a property Beam_M assigned (*BEAMSECTION or PBEAM).

3. `prop_nastran_acm.tcl`

   This script is used in the Nastran and OptiStruct user profiles during the creation of the following configurations:

   - acm – equivalence/detached –(T1+T2)/2, and shell gap custom config welds in the Spot panel,
   - seam hexa adhesive and seam hexa (RBE2-RBE3) in the Seam panel, and
   - Area adhesives in the Area panel.

The script performs the following tasks:

- Organizes the realized Solid Hexa weld elements created during realization process into components with names based on the realization, such as solid_spot_acm_detached, solid_seam_hexa_adhesive_shell_gap, or solid_area_hexa_adhesive_shell_gap. Components and the connected RBE's created as the *HEAD type are organized into components using similar naming, such as rbe3_spot_acm_detached, rbe3_seam_hexa_adhesive, or rbe3_area_hexa_adhesive.
- This script creates property collectors, again using names based on the realization such as solid_spot_acm_detached, solid_seam_hexa_adhesive_shell_gap, or solid_area_hexa_adhesive_shell_gap.
    - These property collectors are created with the PSOLID card associated with them, and are referenced in the above created components containing the Solid Hexa weld elements.
- In addition, this script also updates the weights of any RBE3 that is almost zero, because weight factors close to 0.0 cause Nastran and OptiStruct solvers to generate incorrect results.

> **Note:** New components and properties will only be created if they do not already exist; otherwise the existing components and properties are used. For this reason, comps/props will not always follow the naming conventions given here, because preexisting ones might already have different names.
>
> Also, when creating realizations with a mid-thickness option, the naming conventions include the presence of the mid-thickness. For example, when creating a hexa (RBE2-RBE3) configuration using a mid-thickness option:
>
> - Solid elements will be organized into a Component named solid_seam_ hexa_RBE2_RBE3_mid_thick
> - RBE3 elements will be organized into a Component named rbe3_seam_ hexa_RBE2_RBE3_mid_thick
> - RBE2 elements will be organized into a Component named rbe2_seam_ hexa_RBE2_RBE3_mid_thick
> - Properties will be created with the name solid_seam_ hexa_RBE2_RBE3_mid_thick
> - Materials will be created with the name solid_seam_ hexa_RBE2_RBE3_mid_thick

4. `prop_cweld.tcl`

   This script is called while creation of all the CWELD GA-GB and GS– custom config welds in the Spot panel. Theses include PARTPAT, ELPAT, ELEMID, GRIDID, ALIGN. It performs the following tasks:

   - Assigns the attributes to the CWELD weld element created during the realization process, which is either a rod element [GA-GB] or mass Element [GS] of the types PARTPAT, ELPAT, ELEMID, GRIDID or ALIGN.
   - Creates the property collector with the name prop_<id> with the PWELD card associated with it. This property is referenced to the CWELD element created during realization.
   - This script also updates the weld radius value in the CWELD card. The diameter value is either defined by you on the Spot panel, or is taken from the dvst (diameter versus thickness) file.

   > **Note:** This script is called if the CWELD GA-GB and GS– custom config welds and shell gap custom config welds across Nastran and OptiStruct user profiles.

5. `prop_opt_nas_cfast.tcl`

   This script is called while creation of all the CFAST GA-GB and GS– custom config welds in the Spot panel. Theses include ELEM, and PROP. It performs the following tasks:

   - Assigns the attributes to the CFAST weld element created during the realization process, which is either a rod element [GA-GB] or mass Element [GS] of the types ELEM or PROP.
   - Creates the property collector with the name PFAST_<diameter> with the PFAST card associated with it. This property is referenced to the CFAST element created during realization.
   - This script also updates the weld diameter value in the CFAST card. The diameter value is either defined by you on the Spot panel, or is taken from the dvst (diameter versus thickness) file.

   > **Note:** This script is called for the CFAST GA-GB and GS– custom config welds across Nastran and OptiStruct user profiles.

6. `prop_opt_nas_hilock.tcl`

This script is used while creation of HILOCK custom config welds in the Spot panel from the Nastran and OptiStruct user profile.

This script does the following tasks:

- Organizes the realized 1D weld elements (RBAR, CBAR, CBUSH) created during realization process into a component named HiLock components.
- This script will create the following property collectors:
    - HiLock_PBAR_<diameter>: This property collector is created with the PBAR card associated with it. The RBAR elements reference to this property. The attributes are calculated depending on the used diameter in the Spot panel during realization.
    - HiLock_PBUSH_<translational stiffness>_<rotational stiffness>: These property collectors are created with the PBUSH card associated with them. The CBUSH elements reference to this property. The attributes are calculated depending on the HILOCK material you select and the properties and materials of the connected shells (PSHELL and/or PCOMP).
- This script will create the following load collector:

  HiLock_SPC6
  This load collector is created and the SPCs, which are created for each HiLock will be moved into this collector.
- This script will create the following system collector:

  HiLock
  This system collector is created and the systems created during the realizations will be moved into this collector. If the system collector exists already the new created systems will be moved into the same collector.
- If a HiLock material is not chosen, a default material is created:

  HiLock_MAT1
  This material will be assigned to PBAR cards, and can be found in the following folder of the installation directory: `[hm_scripts_dir]/connectors/HiLock_Mats`.
  The predefined values are:

  set E 1.8e+07

  set G 4.7e+04

  set NU 0.330

  set RHO 8.9e-09

  set A 1.7e-05.

> **Note:** This script is called if the realization CFG Nastran 111 HILOCK or CFG OptiStruct 111 HILOCK is used.

7. `prop_stepboltholes.tcl`

The script performs the following tasks:

- Organizes the CBAR elements into a component with the name HM_Bolt_CBAR.
- Organizes the RBE2 elements into a component with the name HM_Bolt_RBE2.
- Creates a property with the name HM_PBAR and assigns it the PBAR card image.

> **Note:** New components and properties will only be created if their are not any components and properties with the same names that already exist; otherwise the existing components and properties are used.

Nastran Non-Structural Mass Connector (NSM)

HyperMesh handles non-structural masses for Nastran as group entities with a card image NSM1 or NSML1 assigned to them. Whereas normally a connector creates a specific element construct during realization, the NSM connector does not create any element. Instead, each NSM connector receives one group with the appropriate NSM1 or NSML1 card assigned.

Creation and Realization

The non-structural mass connectors can be created and realized in the Apply Mass panel in the Connectors module. The connector location is arbitrary and does not have any influence.

The created NSM connectors are listed in the Connector Browser in a folder named app_mass_ns. The Mass column lists the lumped mass values corresponding to the appropriate NSM solver card the connector is referenced to.

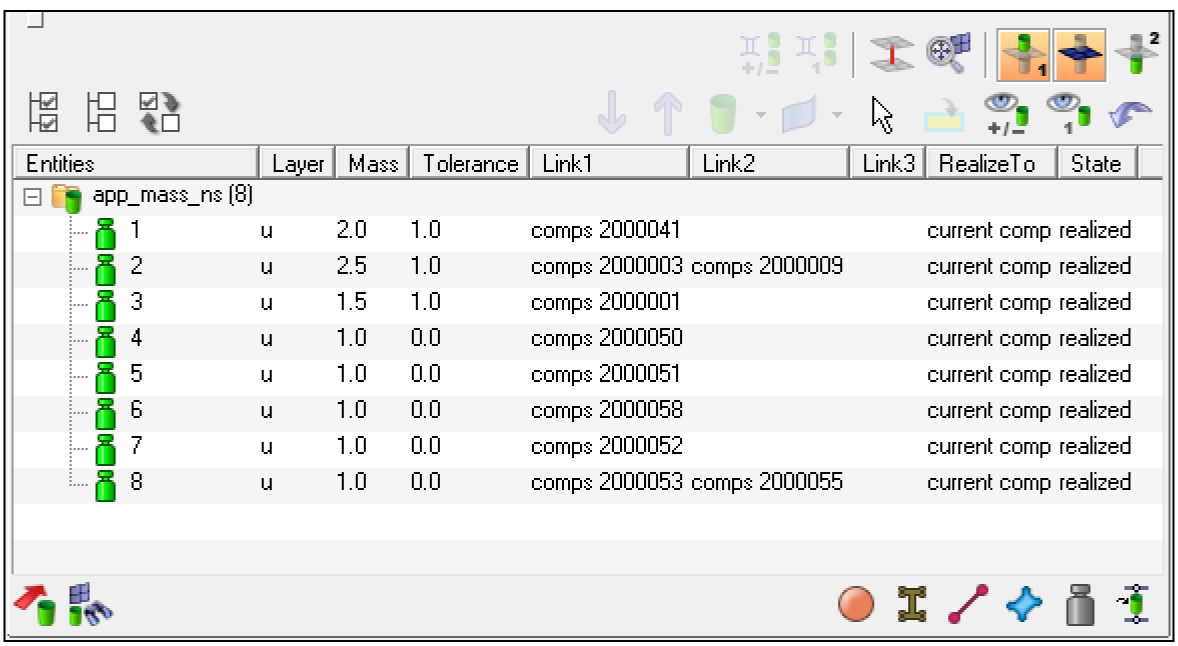


Figure 1337:

Even though the connector is referenced to one specific group, it does not recognize whether this group is manually modified. This means that adding elements to the group will not automatically lead to updated link definitions on the connector. In addition, editing the lumped mass value on the NSM solver card is not synchronized with the connector mass. Deleting the group causes the connector to become unrealized.

Updating the connector links would unrealize the connector. Then the appropriate group is deleted.

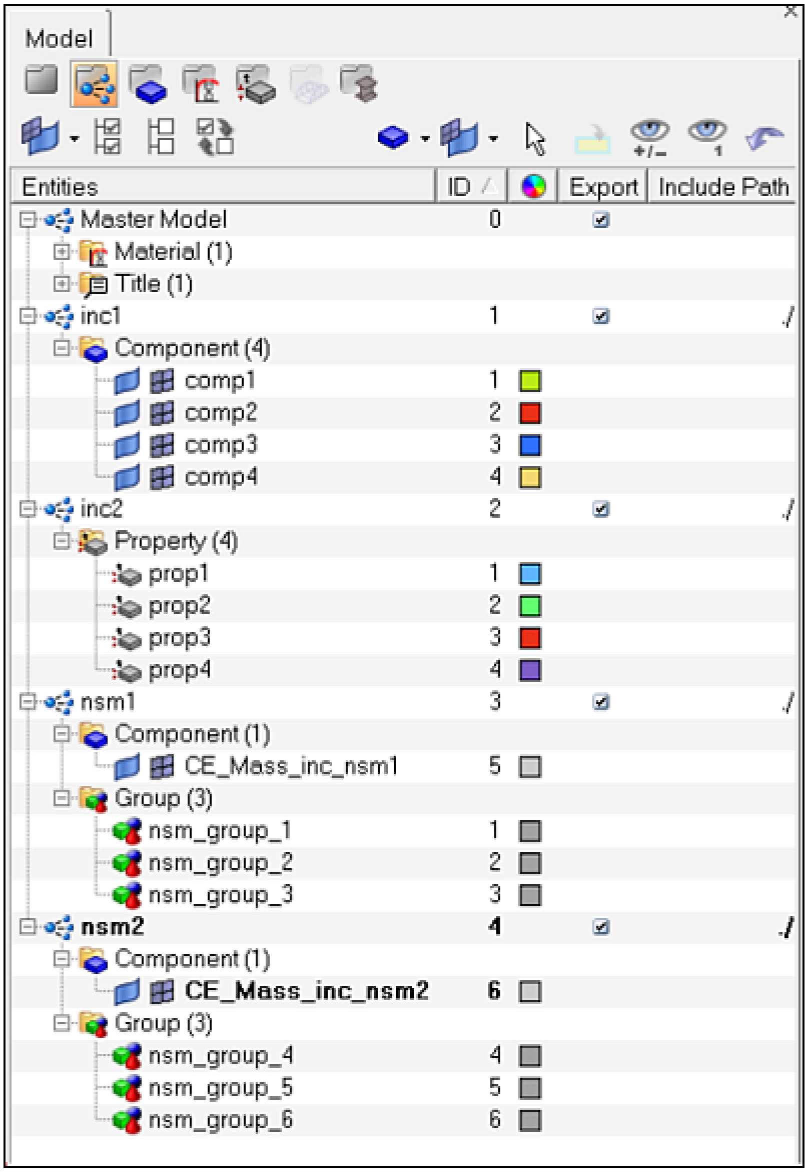


Figure 1338:

The groups created along the NSM connector realization are named as nsm_group_<CE_ID>.nsm_group_<CE_ID>, and placed in the current include. A new component named CE_Mass_inc_<include name> is created which contains all the connector information for that include. Each include is meant to host its own CE_Mass_inc_<include name> component, and HM will produce a warning if the component name already exists in a different include. This helps ensure that unrealizing and rerealizing nsm connectors will keep the FE data in its original include.

The assigned NSM solver card can be of either the PROPERTY or ELEMENT type. This strongly depends on the NSM entity type attribute, which is defined in the panel. The attribute used during creation is written to the connector and is reused for a realization.

With these attributes you can define whether you want to create property- or element-based NSM groups during connector realization. In certain cases, if the defined links cannot be referenced by exclusive properties, a realization as property-based NSM group is not possible. In such cases the connector fails, or can optionally be realized as an element-based NSM group.

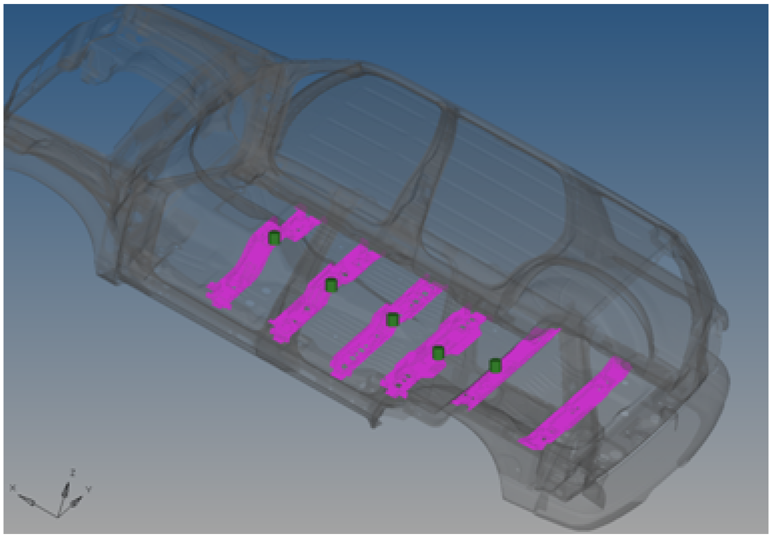


Figure 1339: Groups belonging to Connectors 4-8 in Review Mode

Absorption

The absorption works on all different types like PSHELL, PCOMP, PBAR, PBARL, PBEAM, PBEAML, PBCOMP and PROD.

During absorption the group definition is not modified. The connector is created in the center of a virtual box bounding all referenced elements. Upon NSM absorption, connectors are created inside newly created components (CE_Mass_inc_<include name>), which are created in each include that holds nsm entities. The absorbed connector contains the following information:

- Reference to a certain group
- Lumped mass value in the appropriate NSM solver card
- All elements listed in the NSM solver card. These elements are all defined somehow as connector links. If possible, the single elements are condensed in component links.

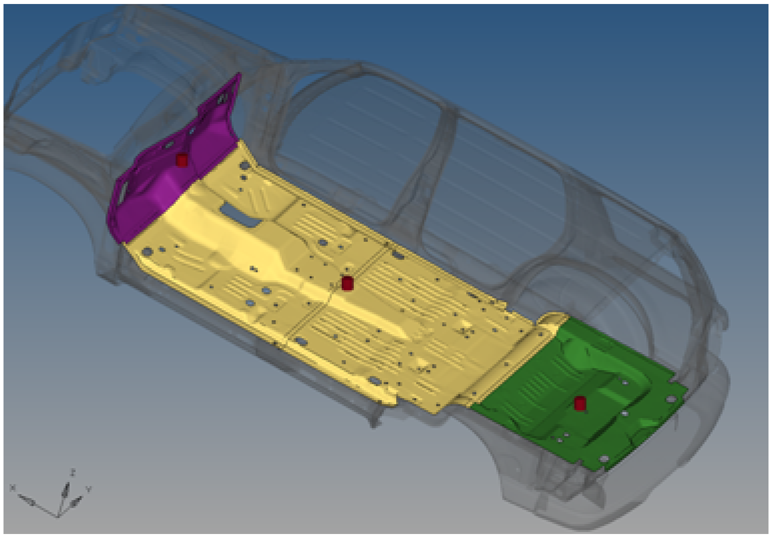


Figure 1340: Dashboard, Front Floor and Trunk Groups Absorbed into Connectors 1-3

OptiStruct Connector Types

Supported OptiStruct connector types and property scripts.

OptiStruct Sealing

Creates RBE3 elements for the head and CBUSH element for the body. The head elements project and connect to the nodes of the adjoining shell elements.

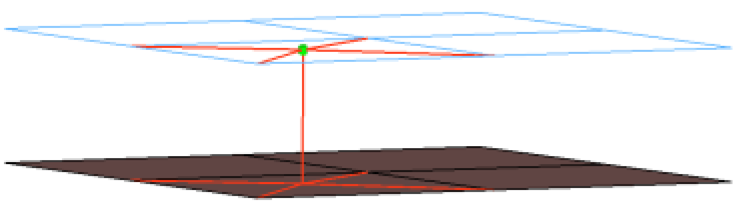


Figure 1341:

```
CFG optistruct 5 sealing
*filter spot
*head
rbe3 1 0
*body 0
spring 6 1
cfg_optistruct_5_sealing
```

OptiStruct bush

Creates RBE2 elements for the head and CBUSH element for the body. The head elements project and connect to the nodes of the adjoining shell elements.

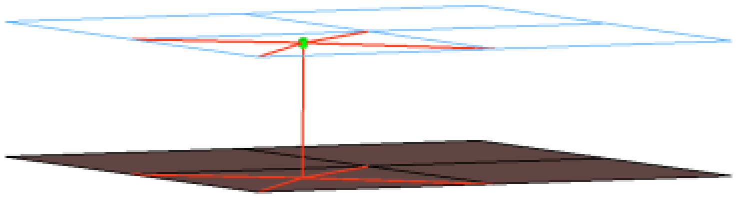


Figure 1342:

```
CFG optistruct 6 bush
*filter spot
*head
rigidlink 1 1
*body 0
spring 6 1
cfg_optistruct_6_bush
```

OptiStruct rbe3 (load transfer)

Creates RBE3 elements for the body. The degrees of freedom are constrained in the x, y, z for the dependant nodes.

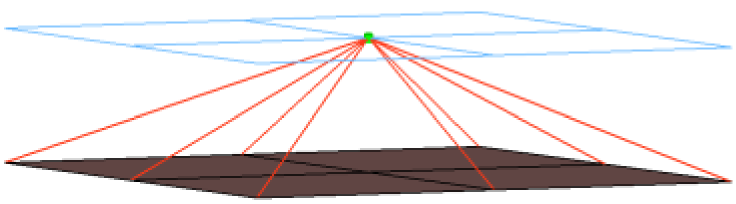


Figure 1343:

```
CFG optistruct 31 rbe3 (load transfer)
*filter spot
*style mpc 1
*head
*body 0
rbe3 1 1 dofs=123
```

OptiStruct clip

Creates a single RBE2 element for the body. The element projects and connects to the nodes of the adjoining shell elements which form the hole and also the nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

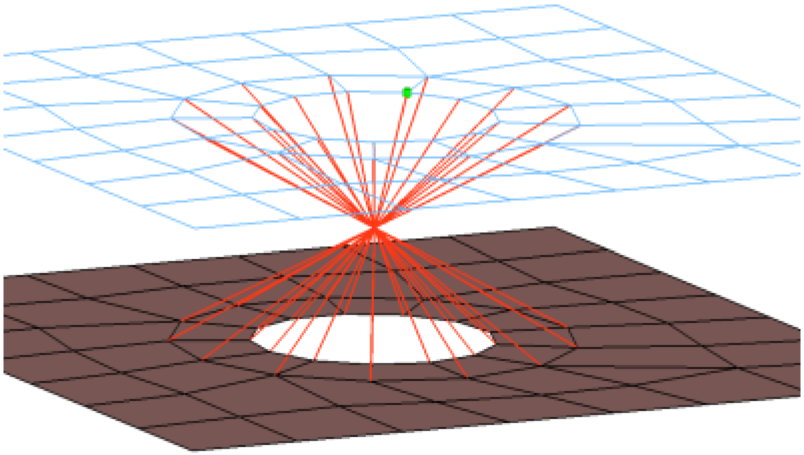


Figure 1344:

```
CFG optistruct 50 clip
*filter bolt
*style bolt 1
*head
*body 0
rigidlink 1 2
```

OptiStruct bolt (washer 1) cbar

Creates RBE2 elements for the head and CBAR element for the body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole and also the second row of nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

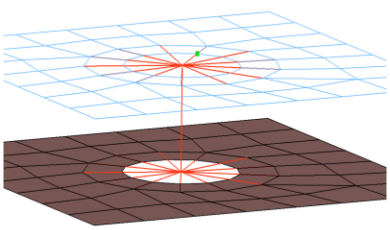


Figure 1345:

```
CFG optistruct 51 bolt (washer 1) cbar
*filter bolt
*style bolt 0
*head
rigidlink 1 12
*body 0
bar2 1 1
```

OptiStruct bolt (general)

Creates RBE2 elements for the head and the body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

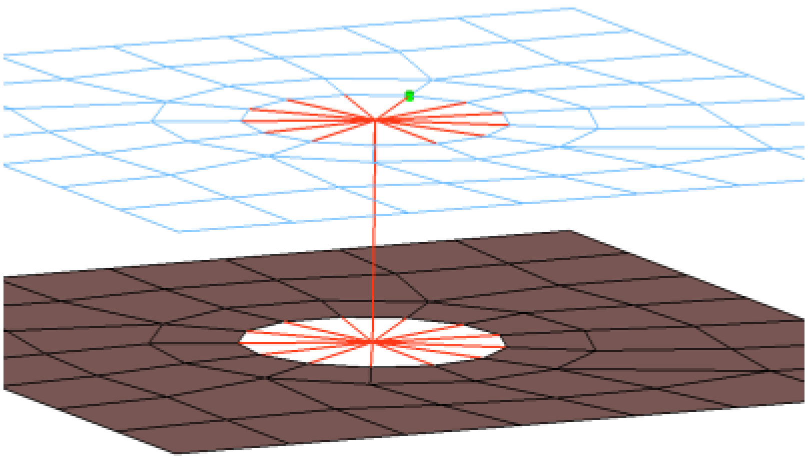


Figure 1346:

```
CFG optistruct 52 bolt (general)
*filter bolt
*style bolt 0
*head
rigidlink 1 1
*body 0
rigid 1 1
```

OptiStruct bolt (CBAR)

Creates RBE2 elements for the head and CBAR element for the body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

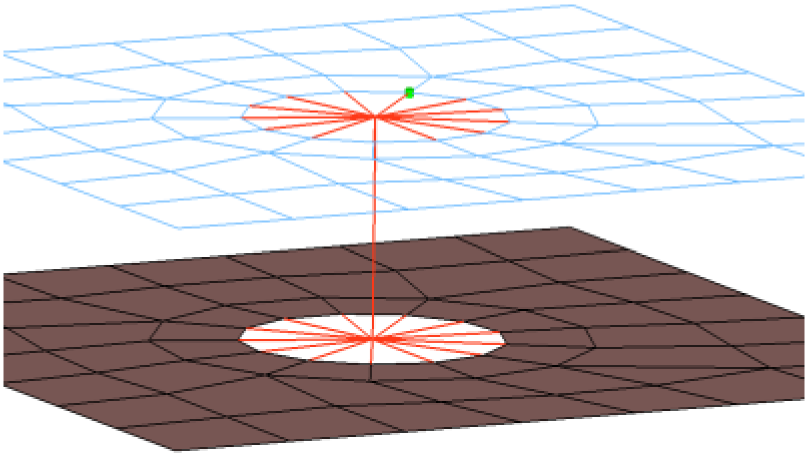


Figure 1347:

```
CFG optistruct 53 bolt (CBAR)
*filter bolt
*style bolt 0
*head
rigid 1 1
*body 0
bar2 1 1
cfg_optistruct_53_bolt
```

OptiStruct bolt (spider)

Creates a many individual RBE2 elements. The element projects and connect to the nodes of the adjoining shell elements which form the hole, the RBE2 elements are joined at the midpoint of the bolted connection. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

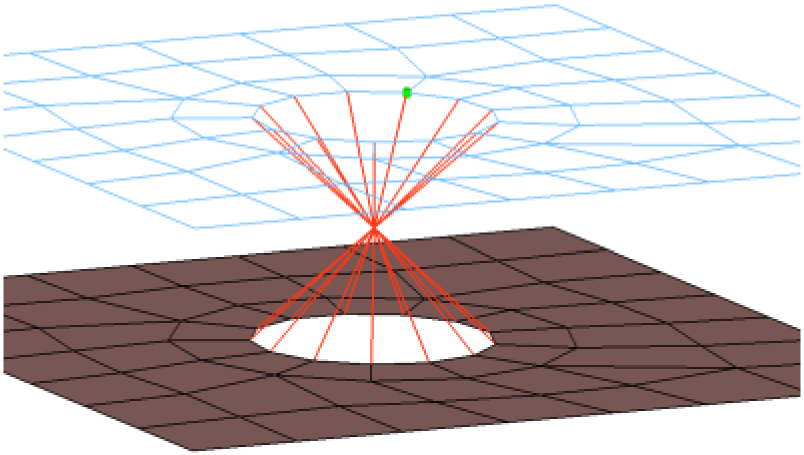


Figure 1348:

```
CFG optistruct 54 bolt (spider)
*filter bolt
*style bolt 1
*head
*body 0
rigid 1 1
cfg_optistruct_54_bolt
```

OptiStruct bolt (washer 2)

Creates RBE2 elements for the head and the body. There are two individual RBE2 elements at the head of the connection, one to connect to the inner row of nodes, the other to connect to the washer layer nodes. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

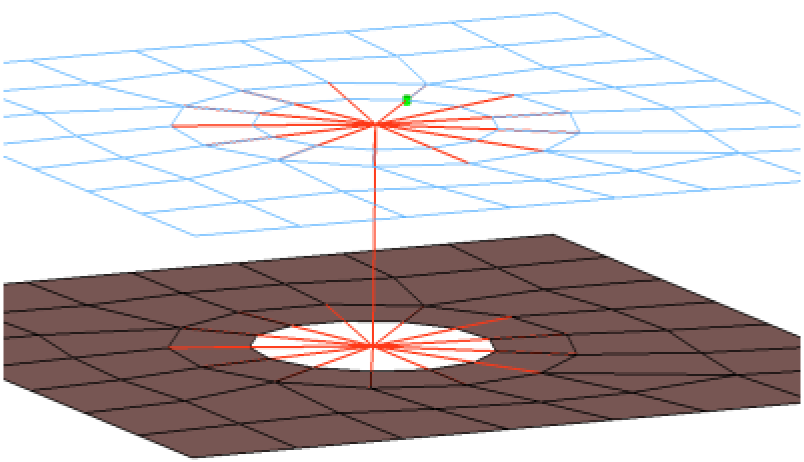


Figure 1349:

```
CFG optistruct 55 bolt (washer 2)
```

```
*filter bolt
*style bolt 0
*head
rigidlink 1 1
rigidlink 1 2
*body 0
rigid 1 1
cfg_optistruct_55_bolt
```

OptiStruct bolt (washer 2 alt)

Creates RBE2 elements for the head and the body. There are two individual RBE2 elements at the head of the connection, one to connect to the inner row of nodes, the other to connect to the washer layer nodes. The RBE2 head element that connects to the washer layer nodes only connects to every other node on the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

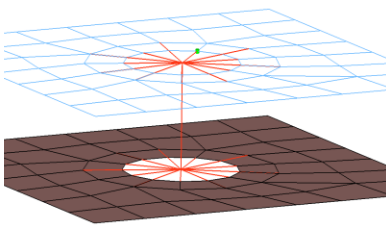


Figure 1350:

```
CFG optistruct 56 bolt (washer 2 alt)
*filter bolt
*style bolt 0
*head
rigidlink 1 1
rigidlink 1 3
*body 0
rigid 1 1
```

OptiStruct bolt (washer 1)

Creates RBE2 elements for the head and body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole and also the second row of nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

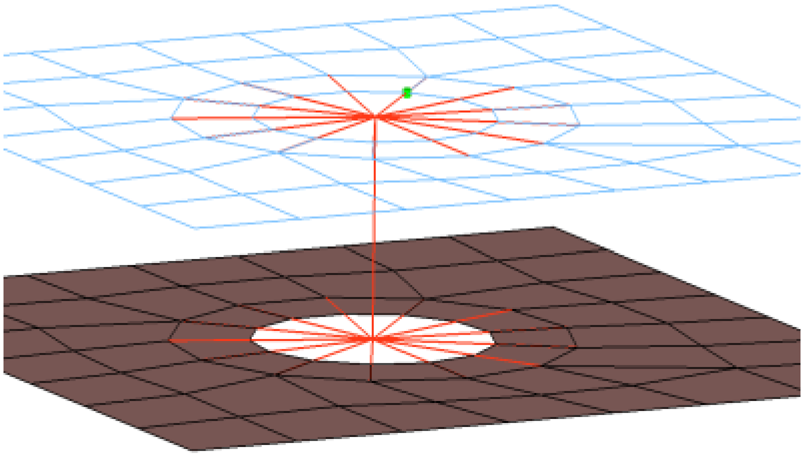


Figure 1351:

```
CFG optistruct 57 bolt (washer 1)
*filter bolt
*style bolt 0
*head
rigidlink 1 12
*body 0
rigid 1 1
cfg_optistruct_57_bolt
```

OptiStruct bolt (washer 1 alt)

Creates RBE2 elements for the head and body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole and also the second row of nodes which form the washer layer. The head only connects to every other node on the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

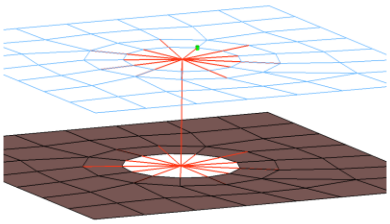


Figure 1352:

```
CFG optistruct 58 bolt (washer 1 alt)
*filter bolt
*style bolt 0
*head
rigidlink 1 13
*body 0
rigid 1 1
cfg_optistruct_58_bolt_washer1_alt
```

OptiStruct hinge

Creates RBE2 elements for the head and the body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer. The degrees of freedom are constrained in the x, y, z, rot x, rot z for the dependant nodes.

This realization also uses the `prop_hinge.tcl`[1] property script.

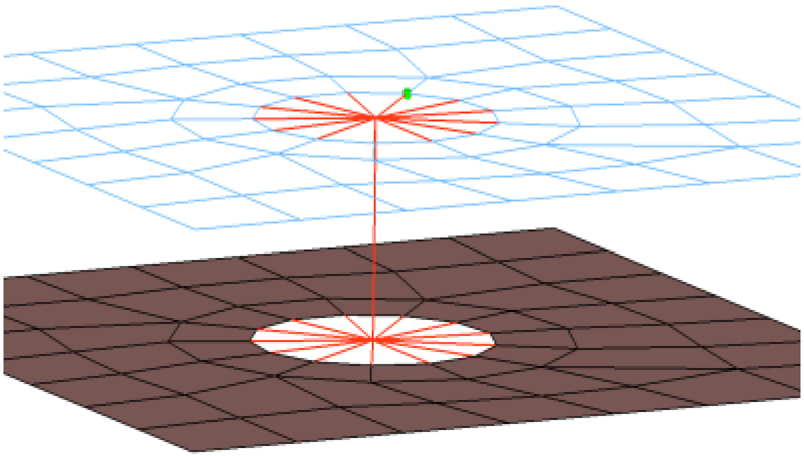


Figure 1353:

```
CFG optistruct 59 hinge
*filter bolt
*style bolt 0
*head
rigidlink 1 1
*body 0
rigid 1 1 dofs=12356
*post prop_hinge.tcl
cfg_optistruct_59_hinge
```

OptiStruct bolt (cylinder rigid)

Creates an RBE2 element for the body as well as for the head elements.
See the mesh independent realization methods in the Bolt panel for further information on cylinder-type bolts.

This realization uses the `prop_cylinder.tcl`[2] property script.

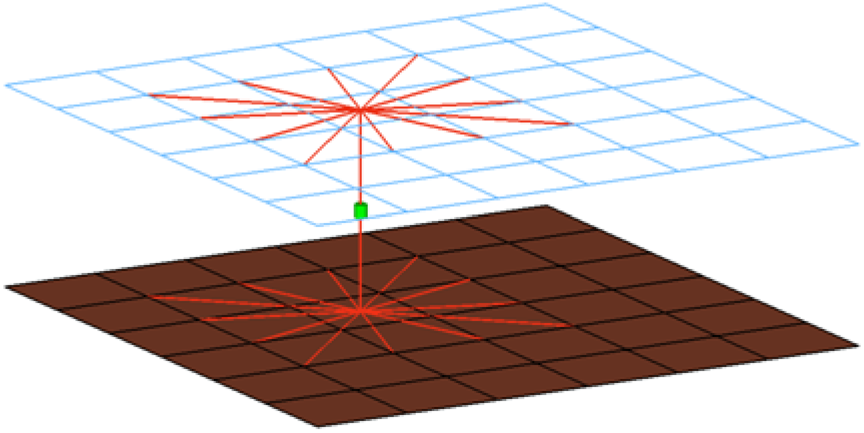


Figure 1354:

```
CFG optistruct 60 bolt (cylinder rigid)
*filter bolt
*style bolt 4
*head
rigidlink 1 1
*body 0
rigid 1 1
*post prop_cylinder.tcl
cfg_nastran_60_bolt_cylinder_rigid
```

OptiStruct (cylinder bar)

Creates a CBEAM element for the body and RBE2 elements for the head elements.
See the mesh independent realization methods in the Bolt panel for further information on cylinder-type bolts.

This realization uses the `prop_cylinder.tcl`[2] property script.

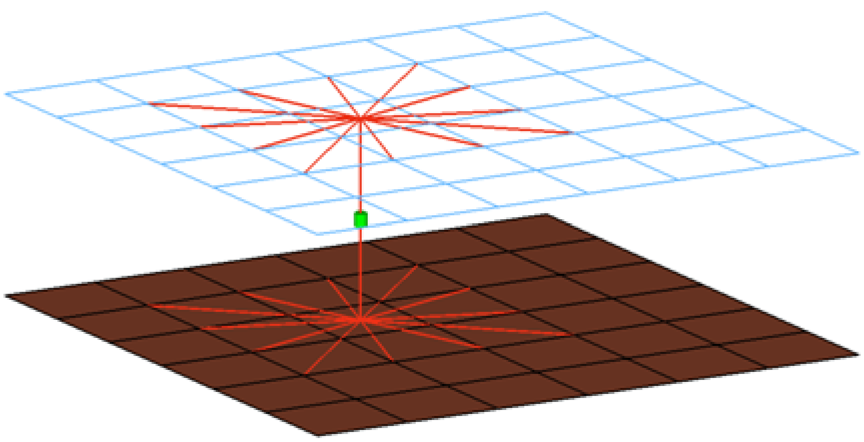


Figure 1355:

```
CFG optistruct 61 bolt (cylinder bar)
*filter bolt
*style bolt 4
*head
rigidlink 1 1
```

```
*body 0
bar2 2 1
*post prop_cylinder.tcl
cfg_nastran_61_bolt_cylinder_bar
```

OptiStruct acm (equivalenced-(T1+T2)/2)

Creates hexa element with RBE3 elements projecting and connecting to the surrounding shell elements. This realization uses the shell thickness to calculate the hexa offset from the shell elements. In the case where the model is a 3T connection, the acm (equivalenced-(T1+T2)/2) realization will join the hexa elements.

This realization also uses the `prop_nastran_acm.tcl`[3] property script.

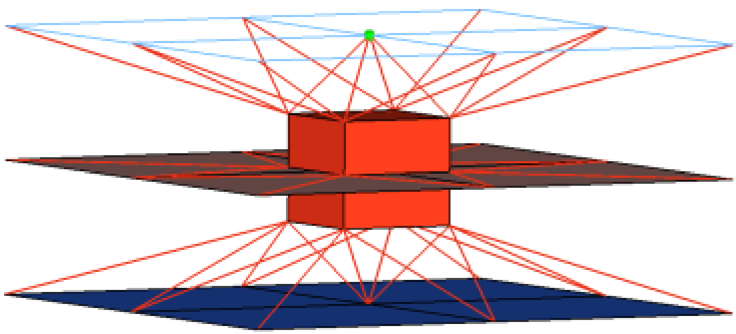


Figure 1356:

```
CFG optistruct 69 acm (equivalenced-(T1+T2)/2)
*filter spot
*style acm 1
*head
rbe3 1 0
*body 0
hex8 1 1
*post prop_nastran_acm.tcl
cfg_optistruct_69_acm
```

OptiStruct acm (detached-(T1+T2)/2)

Creates hexa element with RBE3 elements projecting and connecting to the surrounding shell elements. This realization uses the shell thickness to calculate the hexa offset from the shell elements. In the case where the model is a 3T connection, the acm (detached-(T1+T2)/2) realization will not join the hexa elements.

This realization also uses the `prop_nastran_acm.tcl`[3] property script.

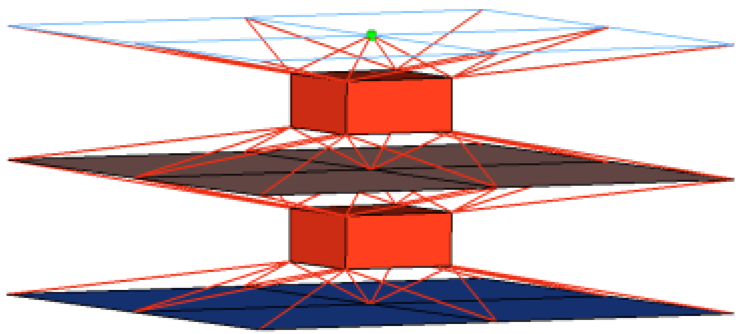


Figure 1357:

```
CFG optistruct 70 acm (detached-(T1+T2)/2)
*filter spot
*style acm 2
*head
rbe3 1 0
*body 1
hex8 1 1
*post prop_nastran_acm.tcl
```

OptiStruct acm (shell gap)

Creates hexa element with RBE3 elements projecting and connecting to the surrounding shell elements. This realization does not use the shell thickness to calculate the hexa offset, therefore the hexa will project and be touching the shell elements.

This realization also uses the `prop_nastran_acm.tcl`[3] property script.

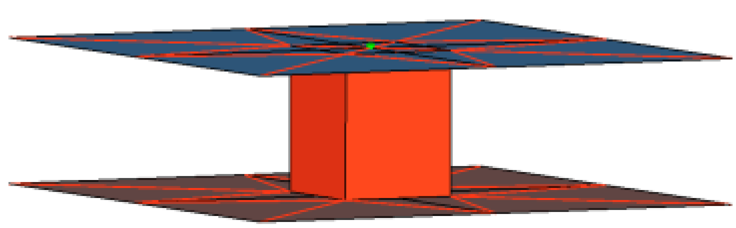


Figure 1358:

```
CFG optistruct 71 acm (shell gap)
*filter spot
*style acm 3
*head
rbe3 1 0
*body 0
hex8 1 1
*post prop_nastran_acm.tcl
cfg_optistruct_71_acm
```

OptiStruct acm (shell gap + coating)

This realization creates one hexa cluster per connector and realizes a node to node connection to the linked shell meshes by adjusting it (shell coating). Different patterns are available. This is driven by the number of hexas. The appearance can be influenced via the diameter and the washer layer activation.

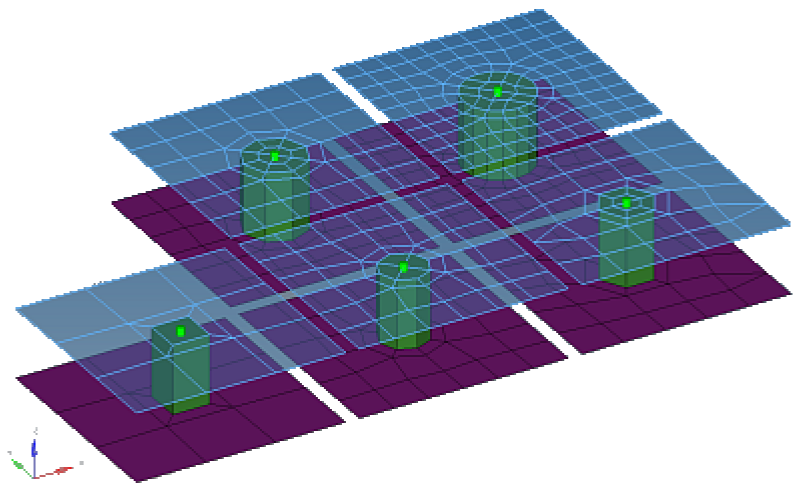


Figure 1359:

```
CFG optistruct 72 acm (shell gap + coating)
*filter spot
*style acm 4
*body 0
hex8 1 1
```

OptiStruct pie (rigid spider)

This realization prepares a circled shell mesh from a certain number of segments for each link, so the mesh is adjusted to a rigid element created with the independent node centered in the circular arranged dependent nodes. The independent nodes themselves are connected by an additional rigid element.

Different numbers of elements lead to a different pattern. In addition, the appearance can be influenced via the diameter and the washer layer activation.

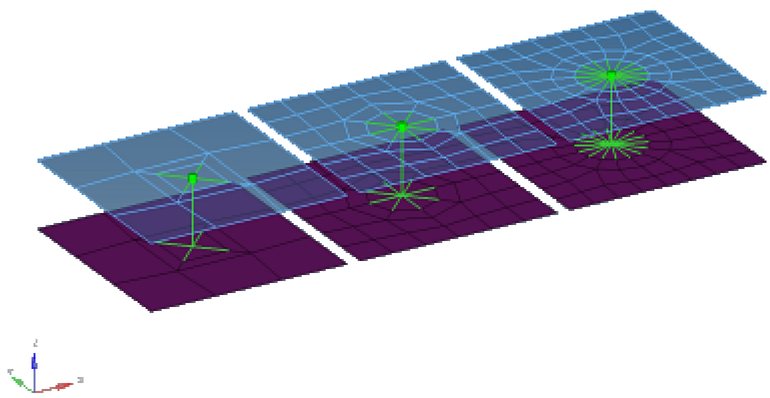


Figure 1360:

```
CFG optistruct 73 pie (rigid spider)
*filter spot
*head
rigidlink 1 4
*body 0
rigid 1 1
```

OptiStruct acm (general)

This realization type consolidates several ACM definitions into one general, flexible ACM definition. Besides mid thickness, constant thickness and maintain gaps, the definition of several coats with different hexa pattern is available.

The realization also uses the `prop_nastran_acm.tcl`[3] property script.

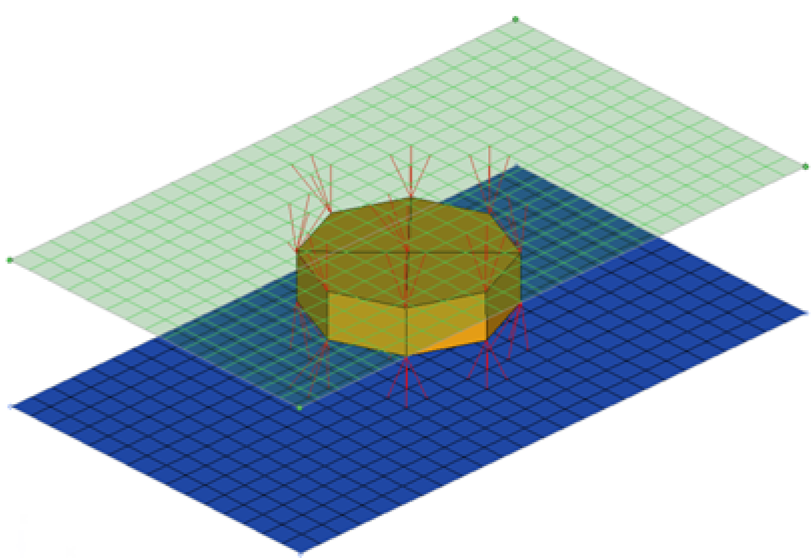


Figure 1361:

```
CFG optistruct 74 acm (general)
```

```
*filter spot
*style acm 3
*head
rbe3 1 0 dofs=123
*body 0
hex8 1 1
*post prop_nastran_acm.tcl
```

OptiStruct penta (mig+L)

This realization supports Lap-joints and creates PENTA element for the body. Surrounding shell/solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint, and provides a fitted/equilateral option for the PENTA creation.

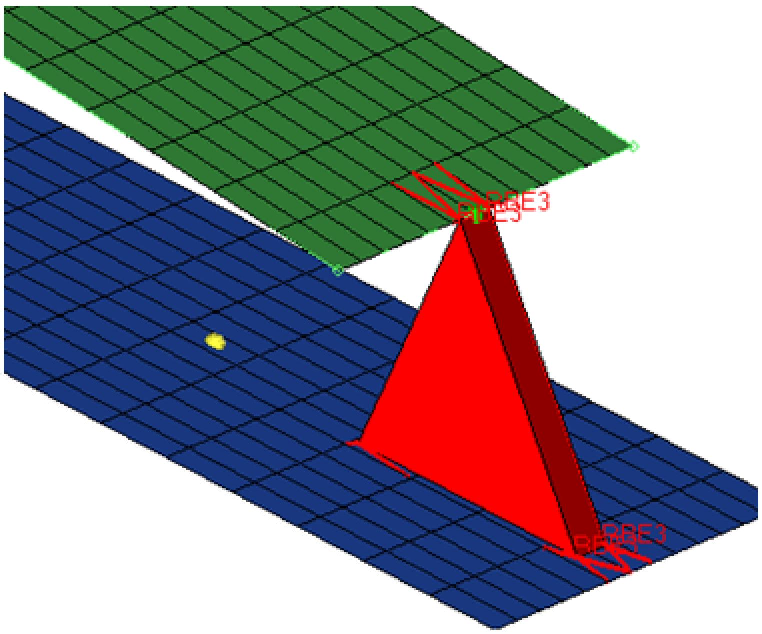


Figure 1362:

```
CFG optistruct 76 penta (mig + L)
*filter spot
*style mig 1
*head
rbe3 1 0
*body 0
penta6 1 1
```

OptiStruct penta (mig+T)

This realization supports T-joints and creates PENTA element for the body. Surrounding shell/solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint, and provides a right-angled option.

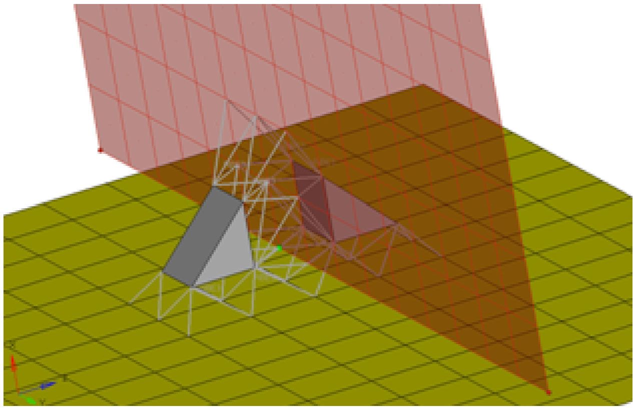


Figure 1363:

```
CFG optistruct 77 penta (mig + T)
*filter spot
*style mig 2
*head
rbe3 1 0
*body 0
penta6 1 1
```

OptiStruct penta (mig+B)

This realization supports Butt-joints and creates PENTA element for the body. Surrounding shell/ solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint. The non-normal option needs to be ON/Active for this realization.

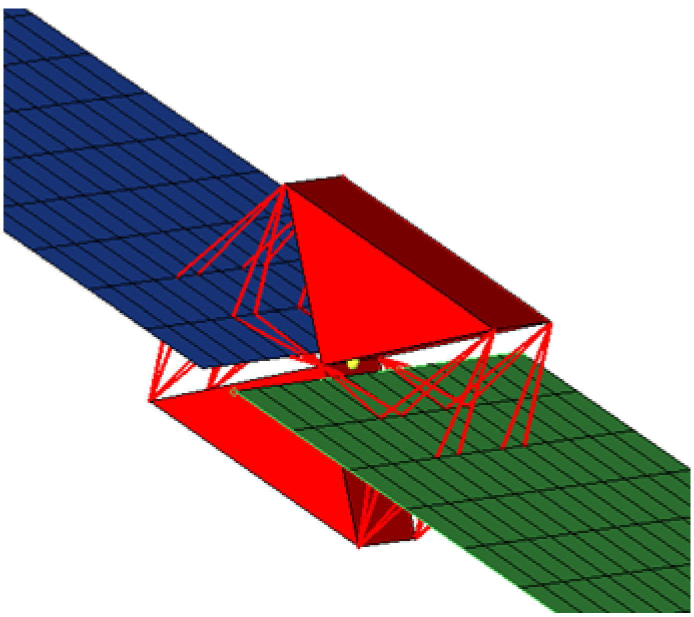


Figure 1364:

```
CFG optistruct 78 penta (mig + B)
*filter spot
```

```
*style mig 3
*head
rbe3 1 0
*body 0
penta6 1 1
```

OptiStruct cweld (GA-GB PARTPAT)

Creates 1d CWELD element via GA-GB PARTPAT.

This realization also uses the `prop_cweld.tcl`[4] property script.

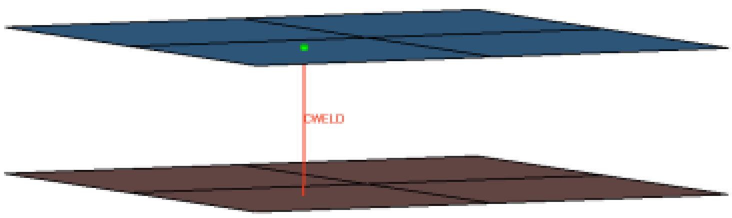


Figure 1365:

```
CFG optistruct 80 cweld (GA-GB PARTPAT)
*filter spot
*head
*body 0
rod 4 1
*post prop_cweld.tcl
cfg_optistruct_80_cweld
```

OptiStruct cweld (GS PARTPAT)

Creates 0D CWELD element via GS PARTPAT.

This realization also uses the `prop_cweld.tcl`[4] property script.

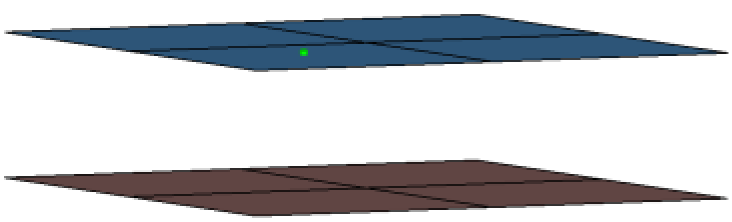


Figure 1366:

```
CFG optistruct 81 cweld (GS PARTPAT)
*filter spot
*head
*body 0
mass 11 0
*post prop_cweld.tcl
cfg_optistruct_81_cweld
```

OptiStruct cweld (GA-GB ELPAT)

Creates 1D CWELD element via GA-GB ELPAT.

This realization also uses the `prop_cweld.tcl`[4] property script.

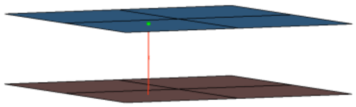


Figure 1367:

```
CFG optistruct 82 cweld (GA-GB ELPAT)
*filter spot
*head
*body 0
rod 4 1
*post prop_cweld.tcl
cfg_optistruct_82_cweld
```

OptiStruct cweld (GS ELPAT)

Creates 0D CWELD element via GS ELPAT.

This realization also uses the `prop_cweld.tcl`[4] property script.

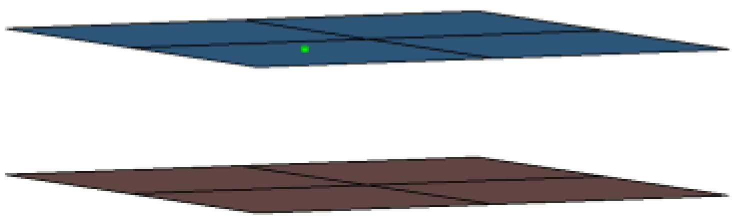


Figure 1368:

```
CFG optistruct 83 cweld (GS ELPAT)
*filter spot
*head
*body 0
mass 11 0
*post prop_cweld.tcl
cfg_optistruct_83_cweld
```

OptiStruct cweld (GA-GB ELEMID)

Creates 1D CWELD element via GA-GB ELEMID.

This realization also uses the `prop_cweld.tcl`[4] property script.

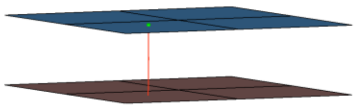


Figure 1369:

```
CFG optistruct 84 cweld (GA-GB ELEMID)
*filter spot
*head
*body 0
rod 4 1
*post prop_cweld.tcl
cfg_optistruct_84_cweld
```

OptiStruct cweld (GS ELEMID)

Creates 0D CWELD element via GS ELEMID.

This realization also uses the `prop_cweld.tcl`[4] property script.

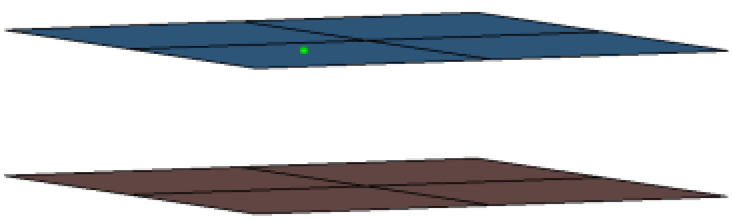


Figure 1370:

```
CFG optistruct 85 cweld (GS ELEMID)
*filter spot
*head
*body 0
mass 11 0
*post prop_cweld.tcl
cfg_optistruct_85_cweld
```

OptiStruct cweld (GA-GB GRIDID)

Creates 1D CWELD element via GA-GB GRIDID.

This realization also uses the `prop_cweld.tcl`[4] property script.

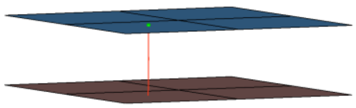


Figure 1371:

```
CFG optistruct 86 cweld (GA-GB GRIDID)
*filter spot
*head
*body 0
rod 4 1
*post prop_cweld.tcl
```

OptiStruct cweld (GS GRIDID)

Creates 0D CWELD element via GS GRIDID.

This realization also uses the `prop_cweld.tcl`[4] property script.

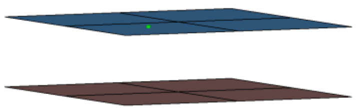


Figure 1372:

```
CFG optistruct 87 cweld (GS GRIDID)
*filter spot
*head
*body 0
mass 11 0
*post prop_cweld.tcl
cfg_optistruct_87_cweld
```

OptiStruct cweld (GA-GB ALIGN)

Creates 1D CWELD element via GA-GB ALIGN.

This realization also uses a property script, please see prop_cweld.tcl for further details.

This realization also uses the `prop_cweld.tcl`[4] property script.

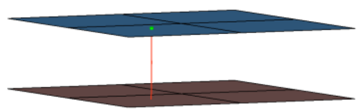


Figure 1373:

```
CFG optistruct 88 cweld (GA-GB ALIGN)
*filter spot
*head
*body 0
rod 4 1
*post prop_cweld.tcl
```

OptiStruct rbe3-celas1-rbe3

Creates RBE3 element for the head and zero length CELAS1 element for the body. The head elements project and connect to the nodes of the adjoining shell elements. The degrees of freedom are constrained in the x, y, z, rot x, rot y, rot z for the dependant nodes.

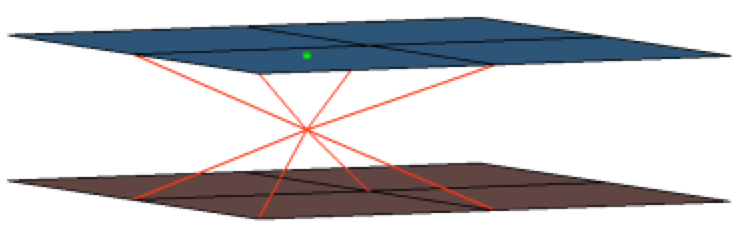


Figure 1374:

```
CFG optistruct 89 rbe3-celas1-rbe3
*filter spot
*head
rbe3 1 0 dofs=123456
*body 0
spring 1 0
```

OptiStruct seam-quad (angled+capped+L)

Creates a quad row with tria caps at the seam ends. In addition, a certain pure quad element pattern is created around the seam elements (shown here in red). These elements normally get imprinted into the shell links. The exact geometry of the seam can be influenced by certain attributes in the Seam panel.

This realization is mainly intended to be used for lap welds.

> **Note:** You can revert the direction of quad seam connectors during the next realization by activating the reverse direction check box in the Seam panel.

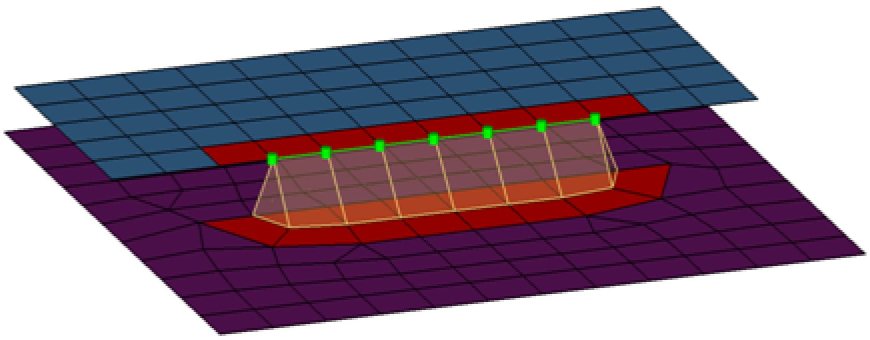


Figure 1375:

```
CFG optistruct 101 seam-quad (angled+capped+L)
*filter seam
*style quad 4
*head
*body 0
quad4 1 1
```

OptiStruct seam-quad (angled+capped+T)

Creates a quad row with tria caps at the seam ends. In addition, a certain pure quad element pattern is created around the seam elements, shown here in red. These elements normally get imprinted into the shell links. The exact geometry of the seam can be influenced by certain attributes in the Seam panel.
This realization is mainly intended to be used for lap welds.

> **Note:** You can revert the direction of quad seam connectors during the next realization by activating the reverse direction check box in the Seam panel.

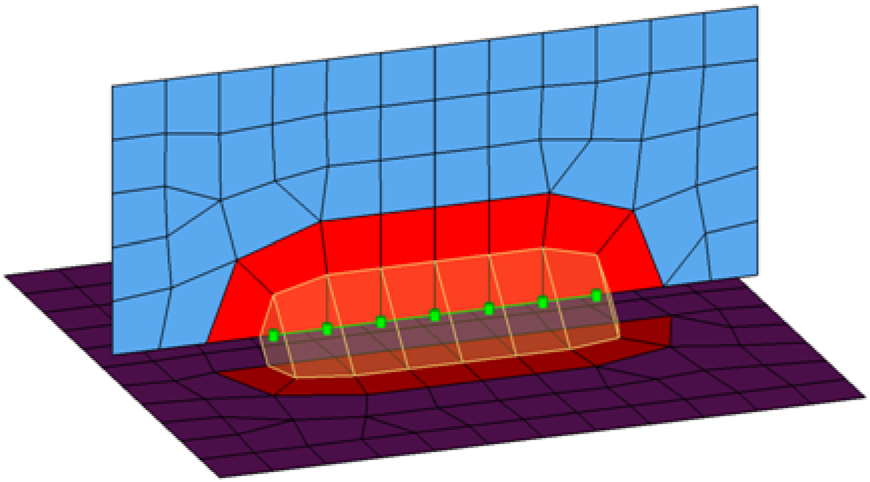


Figure 1376:

```
CFG optistruct 102 seam-quad (angled+capped+T)
*filter seam
*style quad 5
*head
*body 0
quad4 1 1
```

OptiStruct seam-quad (vertical+angled)

Creates two quad rows-the first one perpendicular to the opposite shell link, and the second one with a certain angle to the first one. In addition, a certain pure quad element pattern is created around the seam elements, shown here in red. These elements normally get imprinted into the shell links. The exact geometry of the seam can be influenced by the angle value.
This realization is can be used for both lap- and T-welds.

> **Note:** You can revert the direction of quad seam connectors during the next realization by activating the reverse direction check box in the Seam panel.

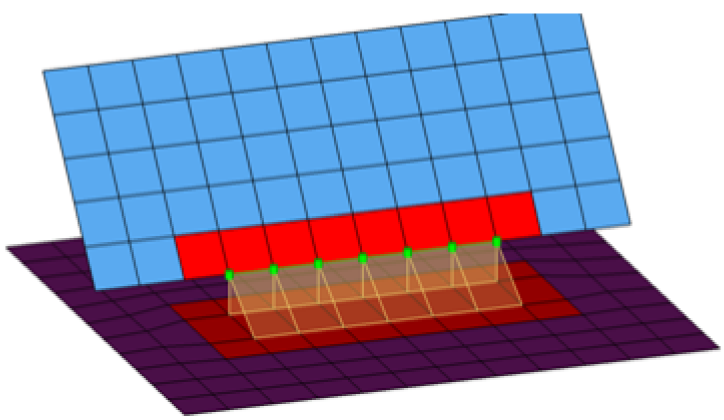


Figure 1377:

```
CFG OptiStruct 103 seam-quad (vertical+angled)
*filter seam
```

```
*style quad 1
*head
*body 0
quad4 1 1
```

OptiStruct seam-quad (angled)

Creates one quad row under a certain angle. The angle is measured between the quad row and the perpendicular projection from the free edge to the opposite shell link. In addition, a certain pure quad element pattern is created around the seam elements, shown here in red. These elements normally get imprinted into the shell links. The exact geometry of the seam can be influenced by the angle value.
This realization is can be used for both, lap- and T-welds.

> **Note:** You can revert the direction of quad seam connectors during the next realization by activating the reverse direction check box in the Seam panel.

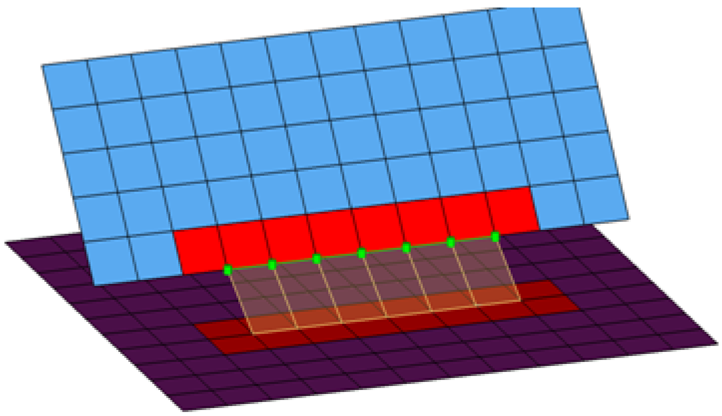


Figure 1378:

```
CFG OptiStruct 104 seam-quad (angled)
*filter seam
*style quad 2
*head
*body 0
quad4 1 1
cfg_optistruct_104_seam_quad_angled
```

OptiStruct penta (mig)

Creates penta elements with RBE3 elements projecting and connecting to the surrounding shell elements. This realization supports many different use cases, including T-joint, angled T-joint, lap joint and butt joint.

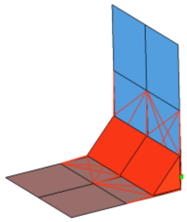


Figure 1379:

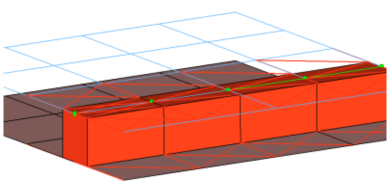
```
CFG OptiStruct 105 penta (mig)
*filter seam
*style continuous 3
*head
rbe3 1 0
*body 0
penta6 1 1
cfg_optistruct_105_penta
```

OptiStruct hexa (adhesive)

Creates a row of hexa elements for the body and numerous RBE2/RBE3 elements for the head. The head elements project and connect to the nodes of the adjoining shell elements. If there is a direct normal project then an RBE2 elements will be used, if there are only non-normal projections then RBE3 elements will be created. The hexa elements are projected so that they touch the shell elements of the connecting components.

This realization also uses the `prop_nastran_acm.tcl`[3] property script.

Figure 1380:



```
CFG OptiStruct 106 hexa (adhesive)
*filter seam
*style continuous 3
*head
rbe3 1 0
rigid 1 0
*body 0
hex8 1 1
```

OptiStruct cfast_elem (GA-GB)

Creates 1D CFAST element of type ELEM.

This realization also uses the `prop_opt_nas_cfast.tcl`[5] property script.

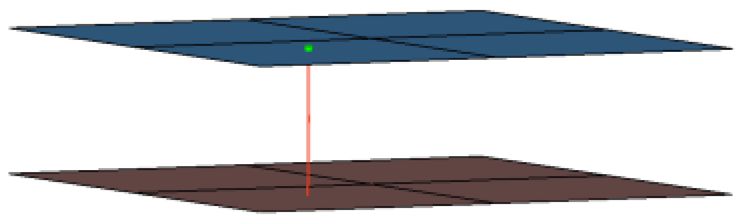


Figure 1381:

```
CFG optistruct 107 cfast_elem (GA-GB)
*filter spot
*head
*body 0
rod 7 1
*post prop_opt_nas_cfast.tcl
cfg_nastran_107_cfast
```

OptiStruct cfast_elem (GS)

Creates 0D CFAST element of type ELEM.

This realization also uses the `prop_opt_nas_cfast.tcl`[5] property script.

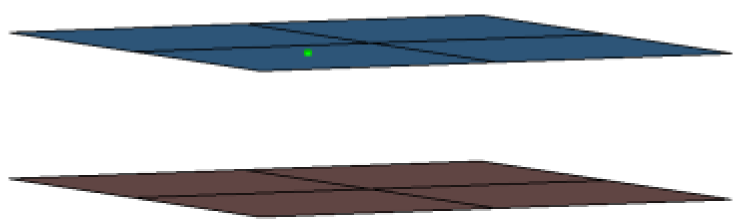


Figure 1382:

```
CFG optistruct 108 cfast_elem (GS)
*filter spot
*head
*body 0
mass 23 0
*post prop_opt_nas_cfast.tcl
cfg_nastran_108_cfast
```

OptiStruct cfast_prop (GA-GB)

Creates 1D CFAST element of type PROP.

This realization also uses the `prop_opt_nas_cfast.tcl`[5] property script.

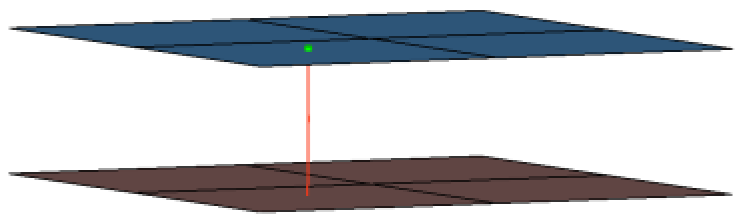


Figure 1383:

```
CFG optistruct 109 cfast_prop (GA-GB)
*filter spot
*head
*body 0
rod 7 1
*post prop_opt_nas_cfast.tcl
```

OptiStruct cfast_prop (GS)

Creates 0D CFAST element of type PROP.

This realization also uses the `prop_opt_nas_cfast.tcl`[5] property script.

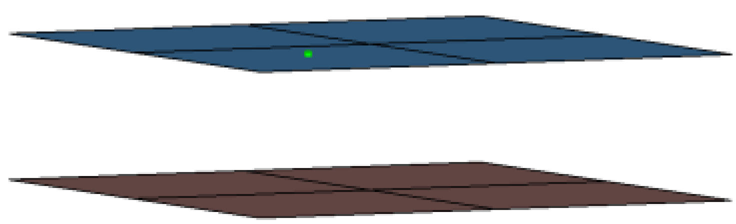


Figure 1384:

```
CFG optistruct 110 cfast_prop (GS)
*filter spot
*head
*body 0
mass 23 0
*post prop_opt_nas_cfast.tcl
cfg_nastran_110_cfast
```

OptiStruct HILOCK

Creates 1D elements that are constructed out of existing RBAR, CBAR, and CBUSH elements. The outer extensions represent the thicknesses of the outer shell elements. The inner nodes of the RBAR element are connected to the shell elements, whereas the inner nodes of the CBAR elements are coincident to the shell nodes only. CBUSHes are created between the appropriate connected and coincident nodes. Each outer node connects one CBAR and one RBAR. Each HLOCK connection gets its own coordinate system which has a z-axis that is collinear to the HILOCK direction. All affected nodes are assigned to the coordinate system. The coordinate system is

taken into account for the DOF definition of the CBAR elements, for the stiffness calculation of the CBUSH elements, and for the DOF of the node constraint.
This realization uses shell properties and materials (PSHELL or PCOMP) and a HILOCK material that you select to calculate the exact position of the outer nodes and the stiffness of the PBUSH elements.

This realization also uses the `prop_opt_nas_hilock.tcl`[6] property script.
For a more detailed examination of the HiLock realization, refer to Special Realization Types.

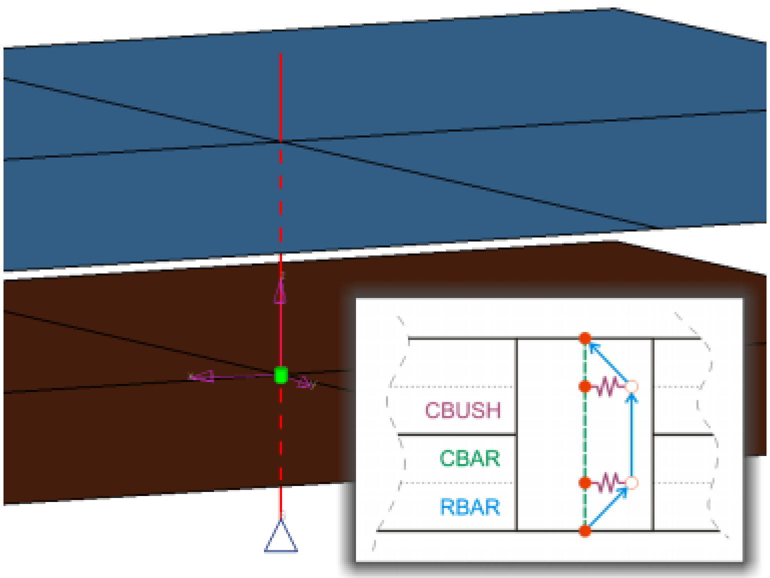


Figure 1385:

```
CFG optistruct 111 HILOCK
*filter spot
*style fastener 1
*head
*bodyext 0
bar2 1 1
weld 1 1 dofs=1456
*body 0
spring 6 0 dofs=2356
bar2 1 1
weld 1 1 dofs=156
spring 6 0 dofs=2356
*post prop_opt_nas_hilock.tcl
cfg_optistruct_111_HiLock
```

OptiStruct clip (washer nodes)

Creates a single RBE2 element for the body. The element projects and connects to the nodes which form the washer layer. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

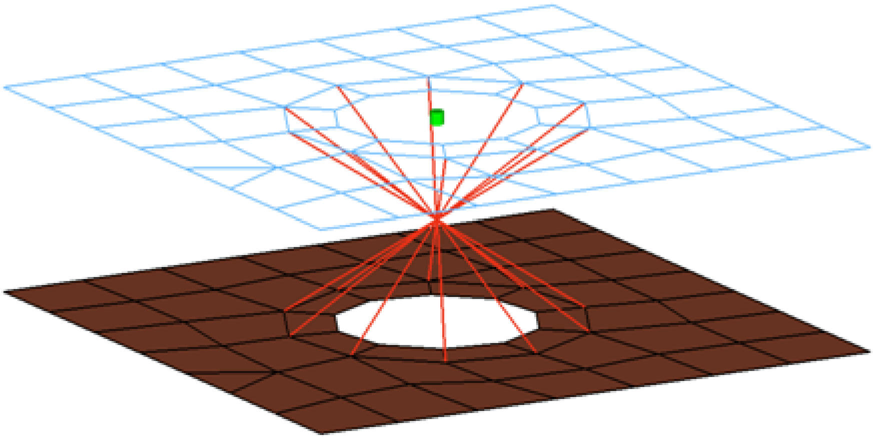


Figure 1386:

```
CFG optistruct 112 clip (washer nodes)
*filter bolt
*style bolt 12
*head
*body 0
rigidlink 1 1
```

OptiStruct bolt (step hole)

This realization creates a CBAR element for the bolt shaft, and connects to the solids' nodes with numerous RBE2 based on the given bolt/hole parameters. It also, connects two solids through holes, or it connects one solid through a hole with a solid blind hole.

This realization uses the `prop_stepboltholes.tcl`[7] property script.

Figure 1387:

```
CFG optistruct 114 bolt (step hole)
*filter bolt
*style bolt 6
*head
rigidlink 1 1
*body 0
```

```
bar2 1 1
*post prop_stepboltholes.tcl
```

OptiStruct bolt (threaded step hole)

Connects two solids through holes or connects one solid through a hole with a solid blind hole. A thread length can be defined to define the dimensions of the rigid elements connecting the bolt shaft models as a bar.

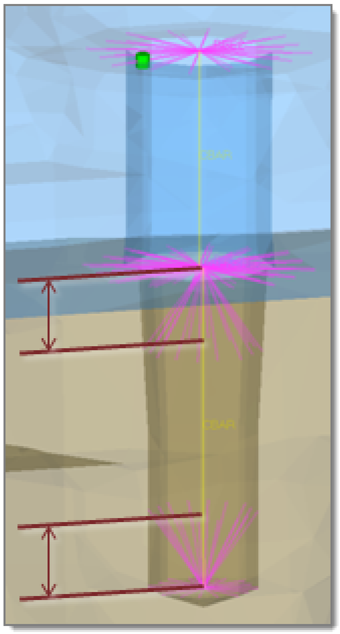


Figure 1388:

```
CFG optistruct 115 bolt (threaded step hole)
*filter bolt
*style bolt 7
*head
rigidlink 1 1
*body 0
bar2 1 1
*post prop_stepboltholes.tcl
```

OptiStruct adhesive-hemmings

This realization type is used for modeling roll hemmings, where the outer shell is bent around the inner shell. The inner shell is connected to the outer shell on one side with simple hexa adhesive, and the other side is connected with RBE2 elements. A definable orientation node decides which side the hexa adhesive should be used. This seam realization type is capable of connecting three layers that contains two components.

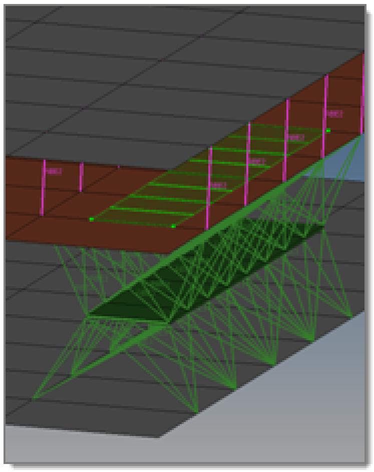


Figure 1389:

```
CFG optistruct 116 adhesive-hemmings
*filter area
*style adhesive 3
*head
rbe3 1 0
*body 1
hex8 1 1
rigid 1 0
*post prop_nastran_acm.tcl
```

OptiStruct penta continuous (mig+L)

This realization supports Lap-joints and creates PENTA element for the body. Surrounding shell/solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint, and provides a fitted/equilateral option for the PENTA creation.

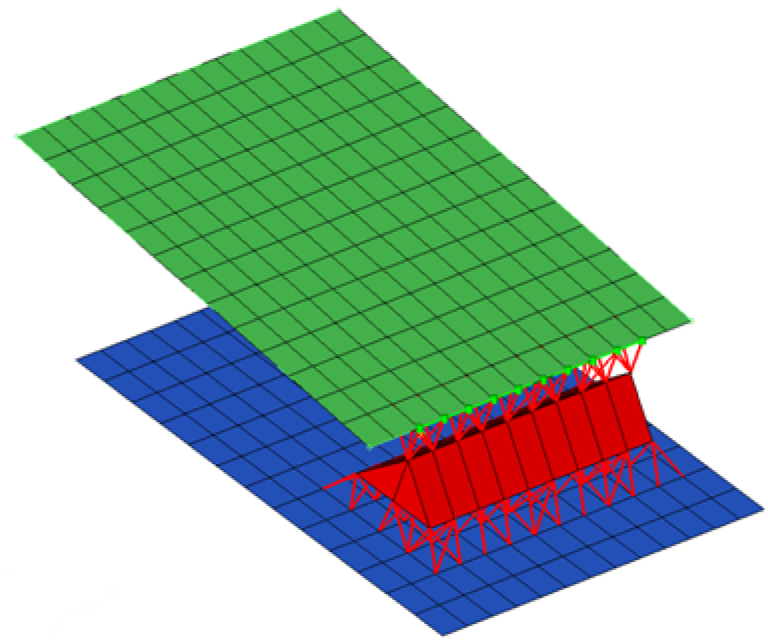


Figure 1390:

```
CFG optistruct 117 penta (mig + L)
*filter seam
*style continuous_mig 1
*head
rbe3 1 0
*body 0
penta6 1 1
```

OptiStruct penta continuous (mig+T)

This realization supports T-joints and creates PENTA element for the body. Surrounding shell/solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint, and provides a right-angled option.

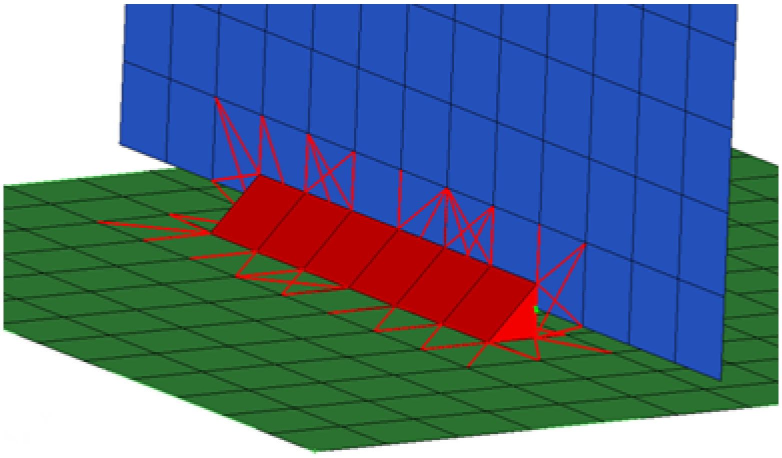


Figure 1391:

```
CFG optistruct 118 penta (mig + T)
*filter seam
*style continuous_mig 2
*head
rbe3 1 0
*body 0
penta6 1 1
```

OptiStruct penta continuous (mig+B)

This realization supports Butt-joints and creates PENTA element for the body. Surrounding shell/solid elements are projected and connected with RBE3 elements. This realization supports the creation of PENTA elements on one side or on both sides of the joint.

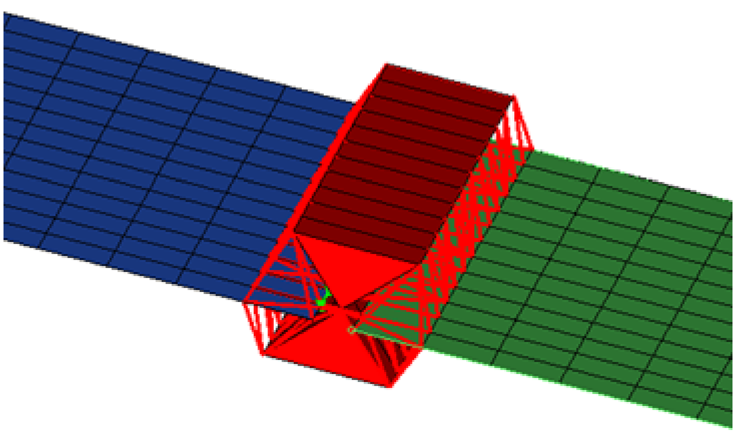


Figure 1392:

```
CFG optistruct 119 penta (mig + B)
*filter seam
*style continuous_mig 3
*head
rbe3 1 0
```

```
*body 0
penta6 1 1
```

OptiStruct wagonwheel

This realization creates RBE2 elements for the body, and projects and connects to the hole edge nodes with RBE3 elements.

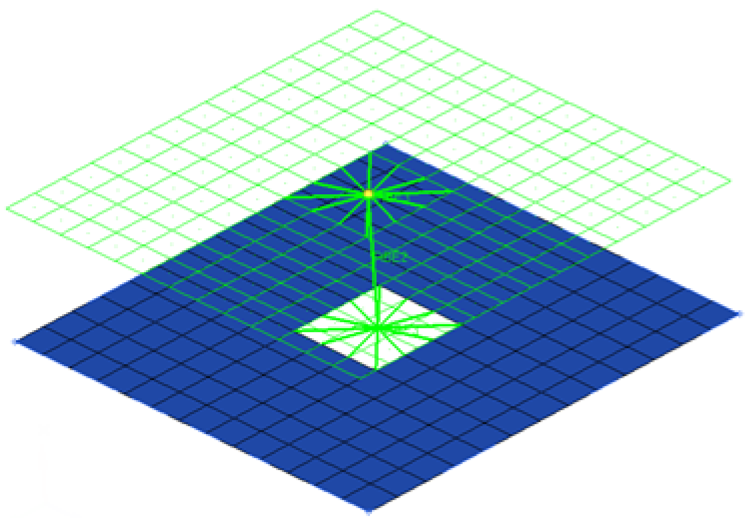


Figure 1393:

```
CFG optistruct 120 wagonwheel
*filter bolt
*style bolt 0
*head
rbe3 1 0
*body 0
rigid 1 1
cfg_nastran_120_wagonwheel
```

OptiStruct adhesives

Creates a row of hexa/penta elements for the body and numerous RBE2/RBE3 elements for the head. The head elements project and connect to the nodes of the adjoining shell elements. If there is significant curvature in the area connector then penta elements will be created, otherwise hexa elements will normally be created. If there is a direct normal project then an RBE2 elements will be used, if there are only non-normal projections then RBE3 elements will be created.

This realization also uses the `prop_nastran_acm.tcl`[3] property script.

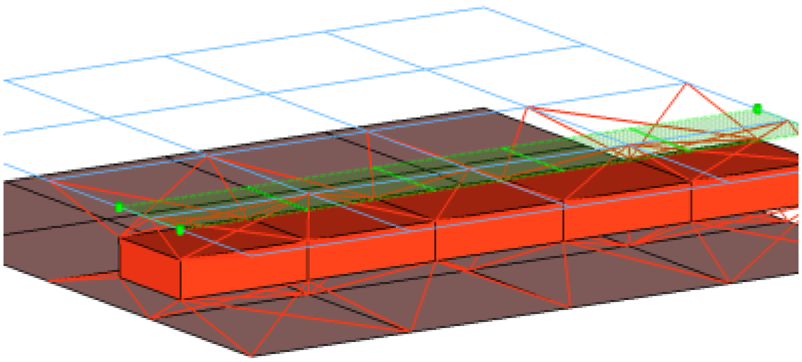


Figure 1394:

```
CFG optistruct 121 adhesives
*filter area
*style adhesive 1
*head
rbe3 1 0
rigid 1 0
*body 1
hex8 1 1
penta6 1 1
*post prop_nastran_acm.tcl
cfg_optistruct_121_adhesives_t1t22
```

OptiStruct hemming

Creates RBE3 elements for the body, the head elements project and connect to the nodes of the adjoining shell elements.

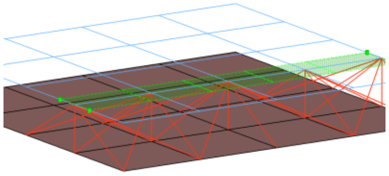


Figure 1395:

```
CFG OptiStruct 122 hemming
*filter area
*style adhesive 1
*head
*body 0
rbe3 1 1
cfg_optistruct_122_hemming
```

OptiStruct CGAP(G)

This realizations creates a CGAP element for the body, which projects and connects to the adjoining shell/solid nodes.

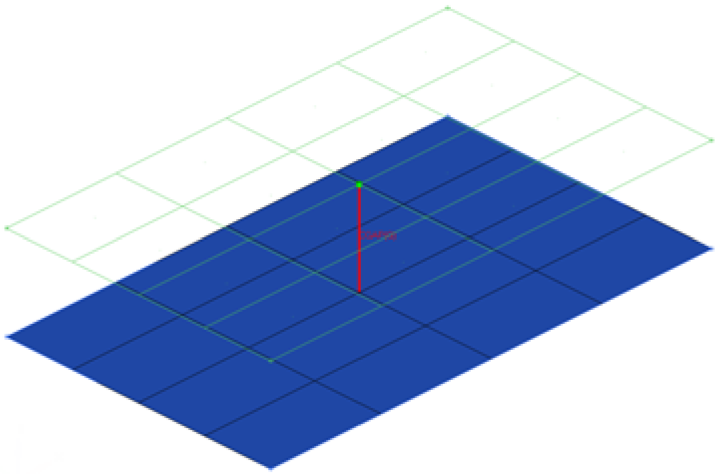


Figure 1396:

```
CFG optistruct 123 CGAP(G)
*filter spot
*head
*body 0
gap 1 1
*post prop_opti_gap.tcl
```

This realization uses the prop_opti_gap.tcl property script.

OptiStruct bolt (collapse rigid)

This realization creates a single RBE2 element for the body. The element projects and connects to the nodes of the adjoining shell/solid elements which form the hole.

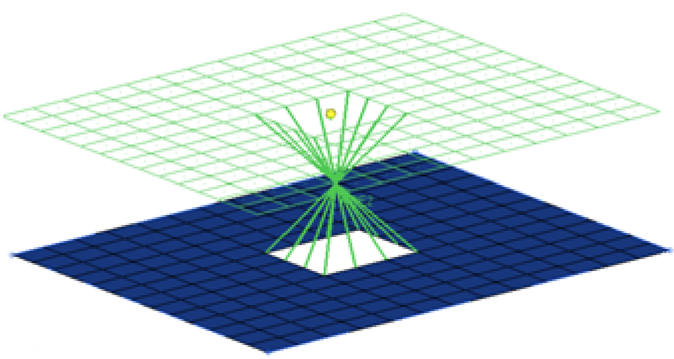


Figure 1397:

```
CFG optistruct 124 bolt (collapse Rigid)
*filter bolt
*style bolt 14
*head
*body 0
rigidlink 1 12
```

OptiStruct seam-quad LTB

Serves and realizes t-welds, lap-welds and butt-welds simultaneously. The weld type is identified automatically based on the orientation of the links to each other.
The dimensions and property for all heat affected zones (HAZ) can be defined separately.
Normal directions of quad weld elements and HAZ elements can be controlled.
An edge treatment can be defined for t-welds and butt-welds to move the edge a precise distance from the opposite link.
For more information, see seam-quad LTB.

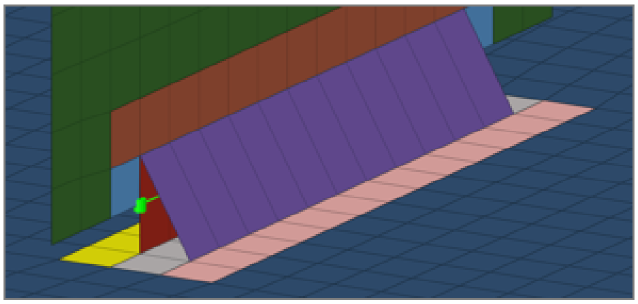


Figure 1398: T-Weld

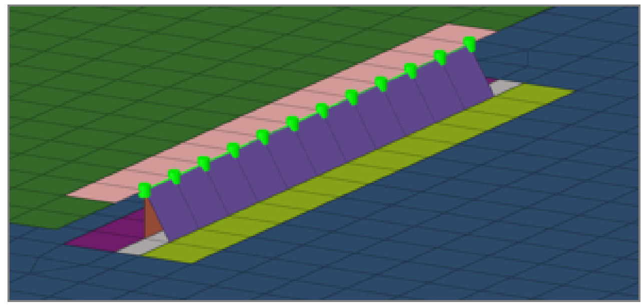


Figure 1399: Lap-Weld

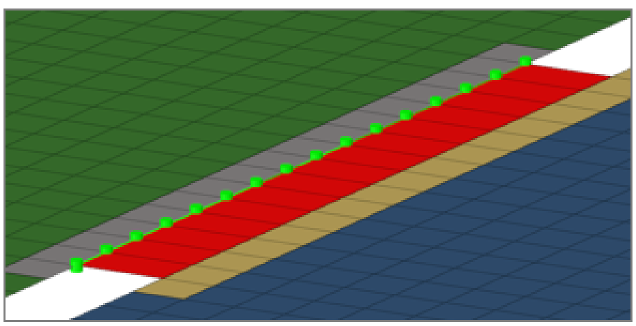


Figure 1400: Butt-Weld

```
CFG optistruct 128 seam-quad LTB
*filter seam
*style quad 7
*head
*body 0
quad4 1 1
```

OptiStruct seam-rigid LTB

Serves and realizes t-welds, lap-welds and butt-welds at the same time. The weld type is identified automatically based on the orientation of the links to each other.
The dimensions and property for all heat affected zones (HAZ) can be defined separately.

An edge treatment can be defined for t-welds and butt-welds to move the edge a precise distance from the opposite link.
For more information, see seam-rigid LTB.

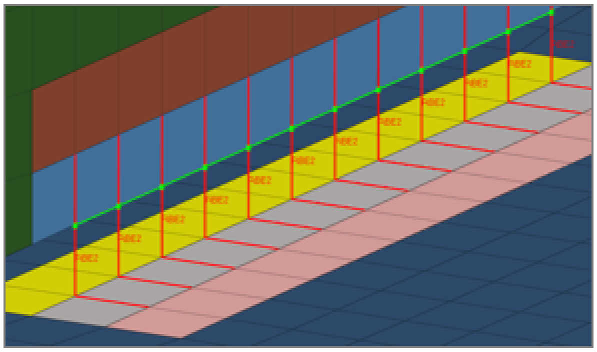


Figure 1401: T-Weld

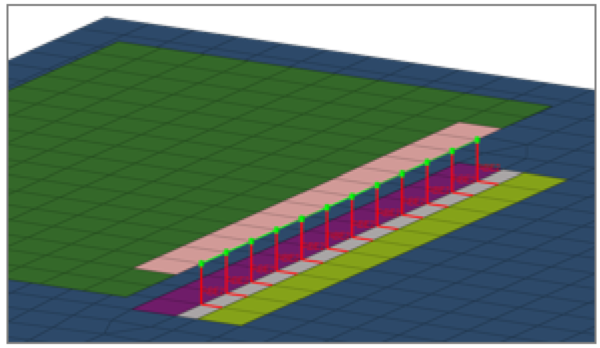


Figure 1402: Lap-Weld

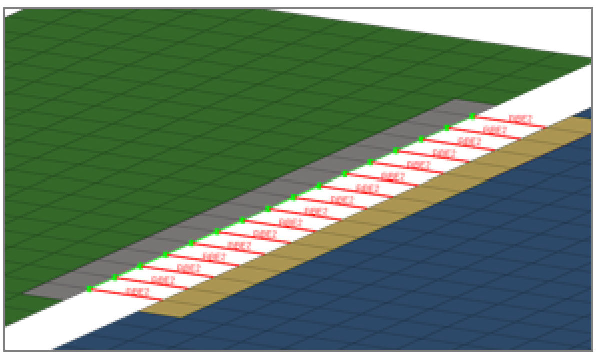


Figure 1403: Butt-Weld

```
CFG optistruct 129 seam-rigid LTB
*filter seam
*style rigid 1
*head
*body 0
rigid 1 1
```

OptiStruct hexa (adhesive - shell gap)

This realization creates rows of HEXA elements for the body. The HEXA elements project and connect to the adjoining shell/solid elements by touching them.

This realization uses the `prop_opt_tie_contacts.tcl`[9] post script.

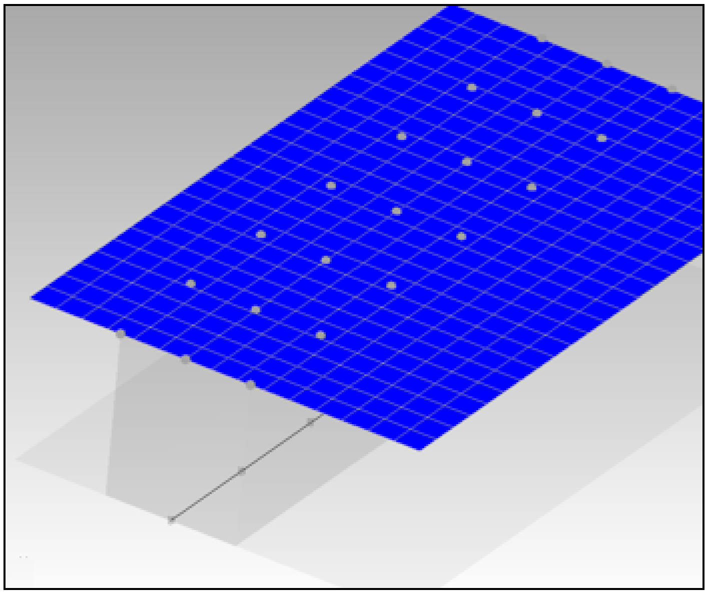


Figure 1404:

```
CFG optistruct 138 hexa (adhesive - shell gap)
*filter seam
*style continuous 2
*head
*body 0
hex8 1 1
*post prop_opt_tie_contacts.tcl
```

OptiStruct hexa (spot tie)

Creates hexa elements between shell and/or solid elements in order to connect them using a tie contact definition. The hexa element nodes will project and touch the shell and/or solid element faces. During the realization, a default tie contact and referencing master and slave sets are created; unless defined differently, the hexas are assigned a default property and material, and are organized into a component with the same name base as the property.

The default tie contact and material parameters can be changed in the files below this path: ..`\Altair\2019\hm\scripts\connectors\Hexa_Tie\optistruct\`.

> **Note:** IDs, names, and card type cannot be changed.

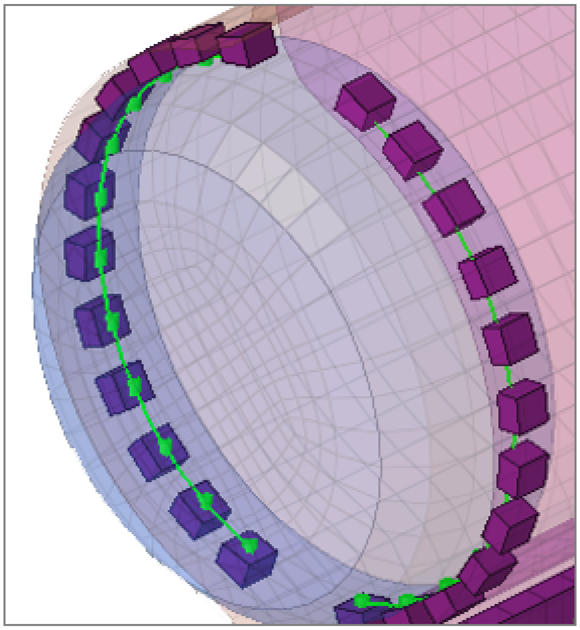


Figure 1405:

```
CFG optistruct 152 hexa (spot tie)
*filter spot
*style spot_tie 1
*head
*body 0
hex8 1 1
```

OptiStruct cbush (spot tie)

Creates bush (CBUSH) elements between shell and/or solid elements in order to connect them using a tie contact definition. The bush element nodes will project and touch the shell and/or solid element faces. During the realization, a default tie contact and referencing master and slave sets are created; unless defined differently, the bushs are assigned a default property, and are organized into a component with the same name base as the property. If no specific bush coordinate system is defined, the bush elements are defined with a vector x1, x2, and x3 normal to it.

The default tie contact and material parameters can be changed in the files below this path: ..`\Altair\2019\hm\scripts\connectors\Bush_Tie\optistruct\`.

> **Note:** IDs, names, and card type cannot be changed.

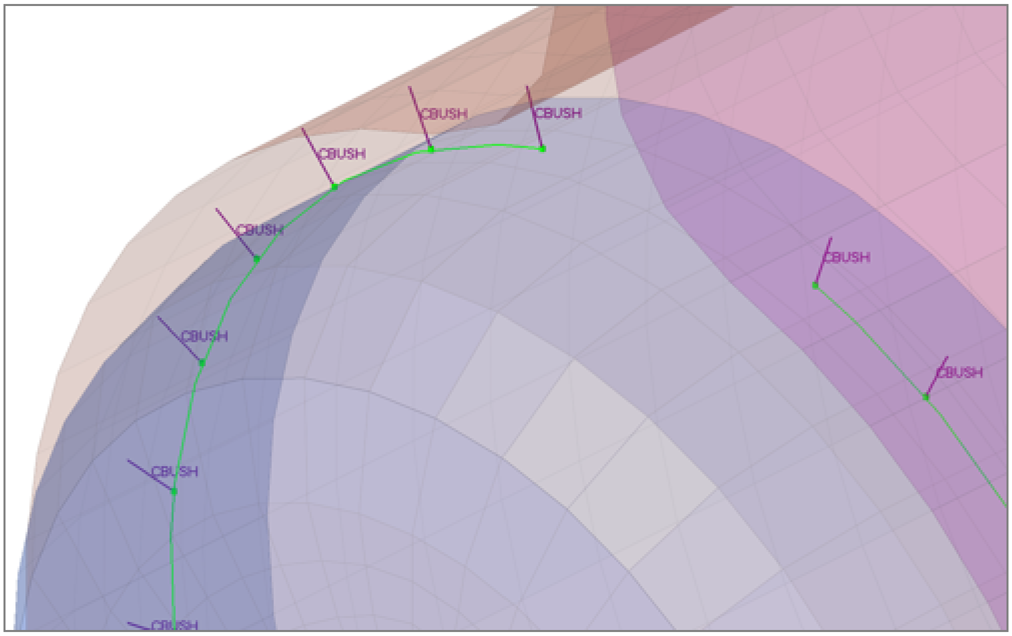


Figure 1406:

```
CFG optistruct 153 cbush (spot tie)
*filter spot
*style spot_tie 2
*head
*body 0
spring 6 1
```

OptiStruct hexa (seam tie)

Creates hexa elements between shell and/or solid elements in order to connect them using a tie contact definition. The hexa element nodes will project and touch the shell and/or solid element faces. During the realization, a default tie contact and referencing master and slave sets are created; unless defined differently, the hexas are assigned a default property and material, and are organized into a component with the same name base as the property.

The default tie contact and material parameters can be changed in the files below this path: ..\Altair\2019\hm\scripts\connectors\Hexa_Tie\optistruct\.

Note: IDs, names, and card type cannot be changed.

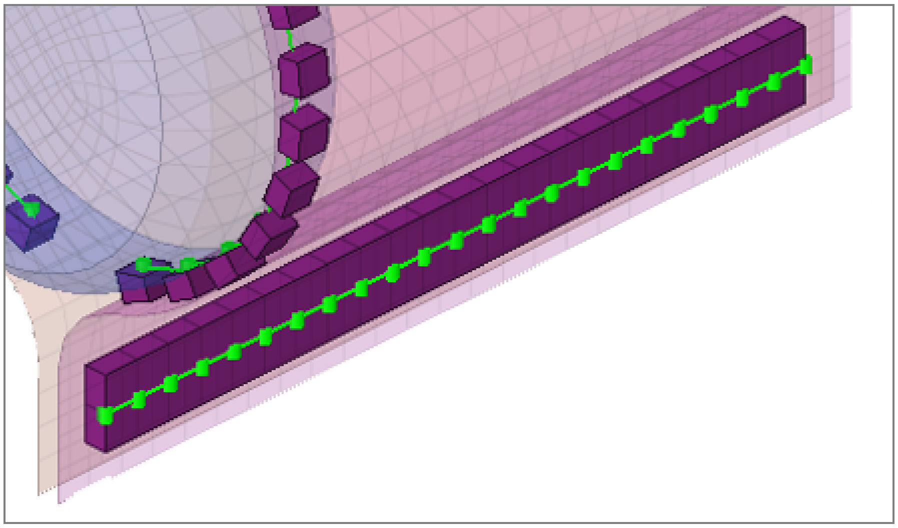


Figure 1407:

```
CFG optistruct 154 hexa (seam tie)
*filter seam
*style seam_tie 1
*head
*body 0
hex8 1 1
```

OptiStruct hexa (area tie)

Creates hexa elements between shell and/or solid elements in order to connect them using a tie contact definition. The hexa element nodes will project and touch the shell and/or solid element faces. During the realization, a default tie contact and referencing master and slave sets are created; unless defined differently, the hexas are assigned a default property and material, and are organized into a component with the same name base as the property.

The default tie contact and material parameters can be changed in the files below this path: `..\Altair\2019\hm\scripts\connectors\Hexa_Tie\optistruct\`.

Note: IDs, names, and card type cannot be changed.

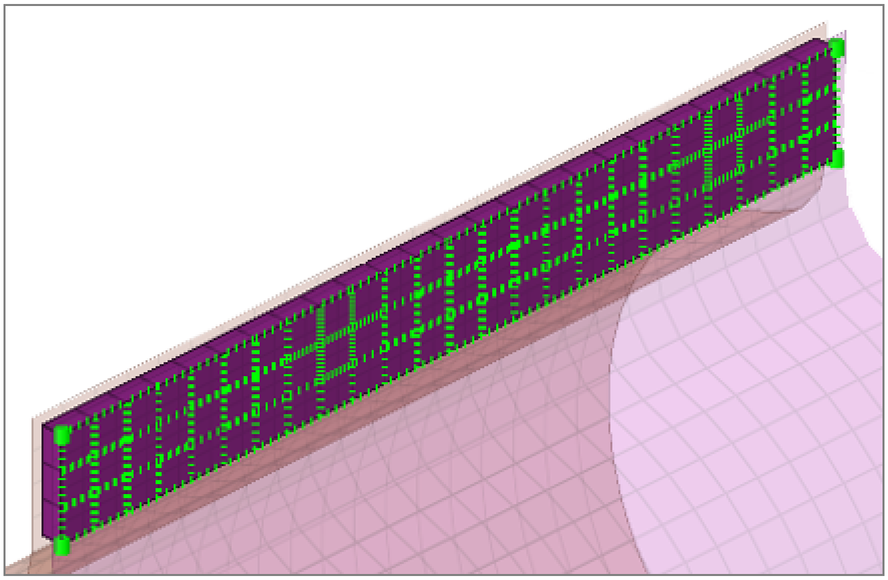


Figure 1408:

```
CFG optistruct 155 hexa (area tie)
*filter area
*style area_tie 1
*head
*body 0
hex8 1 1
```

OptiStruct cbush (rigid)

Creates bush (CBUSH) elements between shell and/or solid elements in order to connect them using rigid (RBE3) elements. The bush element nodes will project and touch the shell and/or solid element faces. Unless defined differently, the bushs are assigned a default property, and are organized into a component with the same name base as the property. If no specific bush coordinate system is defined, the bush elements are defined with a vector x1, x2, and x3 normal to it.

The default property parameters can be changed in the files below this path: `..\Altair\2019\hm\scripts\connectors\Bush_Rigid\optistruct\`.

> **Note:** IDs, names, and card type cannot be changed.

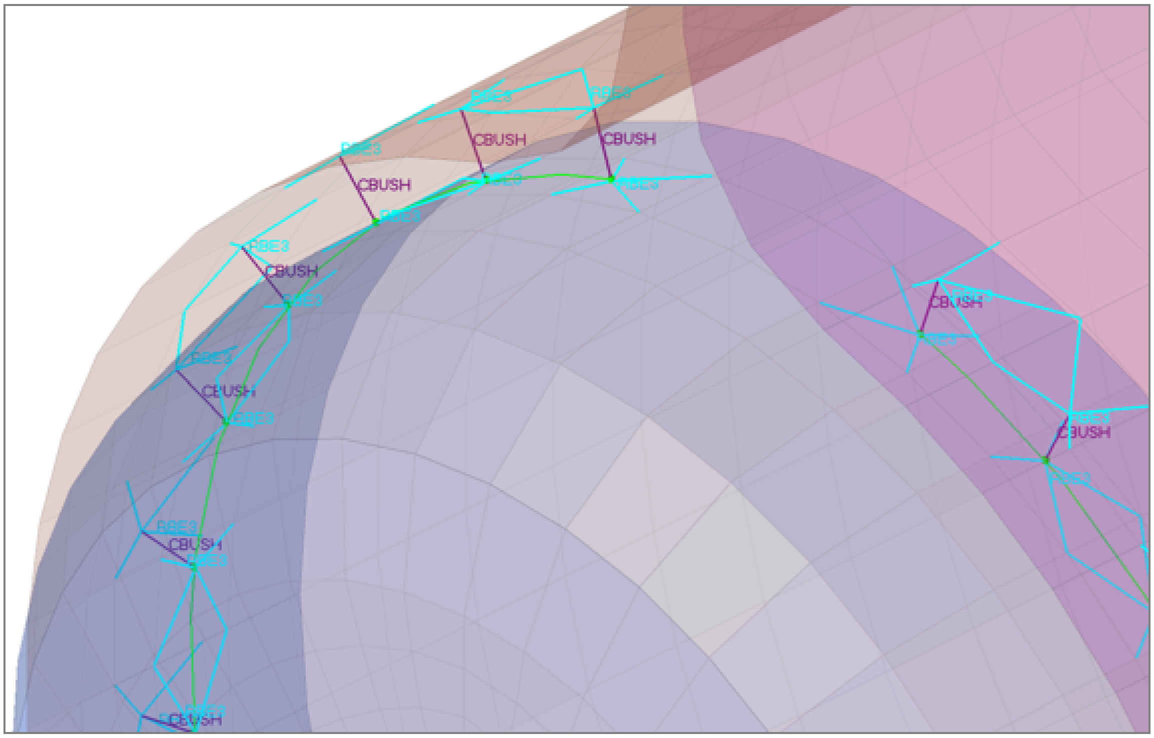


Figure 1409:

```
CFG optistruct 156 cbush (rigid)
*filter spot
*style bush_rigid 1
*head
rbe3 1 0
*body 0
spring 6 1
os_ns_bush_rigid
```

Automatic Exclusion of Special Nodes During Rigid Bolt Realization

HyperMesh automatically excludes special nodes as potential slave nodes for any rigid bodies created during bolt realization, even though they fall with the virtual Bolt Cylinder diameter. Nodes that are referred in the constraints are considered special nodes.

Property Scripts

1. `prop_hinge.tcl`

   This script is called while creation of HINGE– custom config welds in the connector bolts panel. This script performs the tasks when the Systems option is active in the connector Bolt panel (such as "Single System","1- System per layer" or 2- Systems per layer).

   This Script Assigns both reference and analysis systems ID to weld element nodes of each Bolt (Hinge) created during realization process.

2. `prop_cylinder.tcl`

Used while creating bolt (cylinder rigid) and bolt (cylinder bar) in the Bolt panel (Abaqus, Nastran, OptiStruct). It organizes the realized bolt elements into the respective components based upon the*HEAD and the *BODYinformation of the bolt:

- A collector with the name Rigid_M<diameter> is created. This component contains all of the rigid head elements and the rigid body elements, if available.
- A collector with the name Beam_M<diameter> is created. This component contains all of the bar2 head elements, if available. This component then gets a property Beam_M assigned (*BEAMSECTION or PBEAM).

3. `prop_nastran_acm.tcl`

This script is used in the Nastran and OptiStruct user profiles during the creation of the following configurations:

- acm – equivalence/detached –(T1+T2)/2, and shell gap custom config welds in the Spot panel,
- seam hexa adhesive and seam hexa (RBE2-RBE3) in the Seam panel, and
- Area adhesives in the Area panel.

The script performs the following tasks:

- Organizes the realized Solid Hexa weld elements created during realization process into components with names based on the realization, such as solid_spot_acm_detached, solid_seam_hexa_adhesive_shell_gap, or solid_area_hexa_adhesive_shell_gap. Components and the connected RBE's created as the *HEAD type are organized into components using similar naming, such as rbe3_spot_acm_detached, rbe3_seam_hexa_adhesive, or rbe3_area_hexa_adhesive.
- This script creates property collectors, again using names based on the realization such as solid_spot_acm_detached, solid_seam_hexa_adhesive_shell_gap, or solid_area_hexa_adhesive_shell_gap.
    - These property collectors are created with the PSOLID card associated with them, and are referenced in the above created components containing the Solid Hexa weld elements.
- In addition, this script also updates the weights of any RBE3 that is almost zero, because weight factors close to 0.0 cause Nastran and OptiStruct solvers to generate incorrect results.

> **Note:** New components and properties will only be created if they do not already exist; otherwise the existing components and properties are used. For this reason, comps/props will not always follow the naming conventions given here, because preexisting ones might already have different names.
>
> Also, when creating realizations with a mid-thickness option, the naming conventions include the presence of the mid-thickness. For example, when creating a hexa (RBE2-RBE3) configuration using a mid-thickness option:
>
> - Solid elements will be organized into a Component named solid_seam_ hexa_RBE2_RBE3_mid_thick
> - RBE3 elements will be organized into a Component named rbe3_seam_ hexa_RBE2_RBE3_mid_thick
> - RBE2 elements will be organized into a Component named rbe2_seam_ hexa_RBE2_RBE3_mid_thick
> - Properties will be created with the name solid_seam_ hexa_RBE2_RBE3_mid_thick
> - Materials will be created with the name solid_seam_ hexa_RBE2_RBE3_mid_thick

4. `prop_cweld.tcl`

   This script is called while creation of all the CWELD GA-GB and GS– custom config welds in the Spot panel. Theses include PARTPAT, ELPAT, ELEMID, GRIDID, ALIGN. It performs the following tasks:

   - Assigns the attributes to the CWELD weld element created during the realization process, which is either a rod element [GA-GB] or mass Element [GS] of the types PARTPAT, ELPAT, ELEMID, GRIDID or ALIGN.
   - Creates the property collector with the name prop_<id> with the PWELD card associated with it. This property is referenced to the CWELD element created during realization.
   - This script also updates the weld radius value in the CWELD card. The diameter value is either defined by you on the Spot panel, or is taken from the dvst (diameter versus thickness) file.

   > **Note:** This script is called if the CWELD GA-GB and GS– custom config welds and shell gap custom config welds across Nastran and OptiStruct user profiles.

5. `prop_opt_nas_cfast.tcl`

   This script is called while creation of all the CFAST GA-GB and GS– custom config welds in the Spot panel. Theses include ELEM, and PROP. It performs the following tasks:

   - Assigns the attributes to the CFAST weld element created during the realization process, which is either a rod element [GA-GB] or mass Element [GS] of the types ELEM or PROP.
   - Creates the property collector with the name PFAST_<diameter> with the PFAST card associated with it. This property is referenced to the CFAST element created during realization.
   - This script also updates the weld diameter value in the CFAST card. The diameter value is either defined by you on the Spot panel, or is taken from the dvst (diameter versus thickness) file.

> **Note:** This script is called for the CFAST GA-GB and GS– custom config welds across Nastran and OptiStruct user profiles.

6. `prop_opt_nas_hilock.tcl`

   This script is used while creation of HILOCK custom config welds in the Spot panel from the Nastran and OptiStruct user profile.

   This script does the following tasks:

   - Organizes the realized 1D weld elements (RBAR, CBAR, CBUSH) created during realization process into a component named HiLock components.
   - This script will create the following property collectors:
     - HiLock_PBAR_<diameter>: This property collector is created with the PBAR card associated with it. The RBAR elements reference to this property. The attributes are calculated depending on the used diameter in the Spot panel during realization.
     - HiLock_PBUSH_<translational stiffness>_<rotational stiffness>: These property collectors are created with the PBUSH card associated with them. The CBUSH elements reference to this property. The attributes are calculated depending on the HILOCK material you select and the properties and materials of the connected shells (PSHELL and/or PCOMP).
   - This script will create the following load collector:

     HiLock_SPC6
     This load collector is created and the SPCs, which are created for each HiLock will be moved into this collector.
   - This script will create the following system collector:

     HiLock
     This system collector is created and the systems created during the realizations will be moved into this collector. If the system collector exists already the new created systems will be moved into the same collector.
   - If a HiLock material is not chosen, a default material is created:

     HiLock_MAT1
     This material will be assigned to PBAR cards, and can be found in the following folder of the installation directory: `[hm_scripts_dir]/connectors/HiLock_Mats`.
     The predefined values are:

     set E 1.8e+07

     set G 4.7e+04

     set NU 0.330

     set RHO 8.9e-09

     set A 1.7e-05.

> **Note:** This script is called if the realization CFG Nastran 111 HILOCK or CFG OptiStruct 111 HILOCK is used.

7. `prop_stepboltholes.tcl`

The script performs the following tasks:

- Organizes the CBAR elements into a component with the name HM_Bolt_CBAR.
- Organizes the RBE2 elements into a component with the name HM_Bolt_RBE2.
- Creates a property with the name HM_PBAR and assigns it the PBAR card image.

> **Note:** New components and properties will only be created if their are not any components and properties with the same names that already exist; otherwise the existing components and properties are used.

8. `2019`

The script performs the following tasks:

- Organizes the CGAP elements into a component with the name Realize2001.
- Creates a property with the name PGapProp, and assigns the card image PGAP.

> **Note:** New components and properties will only be created if their are not any components and properties with the same names that already exist; otherwise the existing components and properties are used.

9. `prop_opt_tie_contacts.tcl`

The script performs the following tasks:

- Creates TIE interfaces (groups) with the name ADHESIVE_HEXA_TIE_CONTACT_PID _# and the card image TIE. The groups reference the independent/dependent links' master sets and the nodes' slave sets (# is the ID of the link components).
- Organizes the links' component elements into sets with the name ADHESIVE_HEXA_MASTER_PART_SET_PID_# and the card image SET_ELEM. The sets are referenced by the above interface groups (# is the ID of the link component).
- Organizes the solids' nodes on the links into sets with the name ADHESIVE_HEXA_SLAVE_NODE_SET_PID _# and the card image SET_GRID. The sets are referenced by the above interface groups (# is the ID of the link component).
- Creates components with the name ADHESIVE_HEXA_COMP_PID _#_# for the connector SOLID elements (# is the ID of the link component).
- Creates properties with the name ADHESIVE_HEXA_PROP_PID_ #_# and the PSOLID cardimage, and assigns them to respective SOLID components (# is the ID of the link component).

- Creates a material with the name ADHESIVE_HEXA_MAT_PID _#_# and the card image MAT1, and assigns it to respective SOLID properties (# is the ID of the link component).

OptiStruct Non-Structural Mass Connector (NSM)

Description, Creation, and Realization:

HyperWorks Desktop handles non-structural masses for OptiStruct as group entities with a card image NSM1 or NSML1 assigned to them. Whereas normally a connector creates a specific element construct during realization, the NSM connector does not create any element. Instead, each NSM connector receives one group with the appropriate NSM1 or NSML1 card assigned.

The non-structural mass connectors can be created and realized in the Apply Mass panel in the Connectors module. The connector location is arbitrary and does not have any influence.

The created NSM connectors are listed in the Connector Browser in a folder named app_mass_ns. The Mass column lists the lumped mass values corresponding to the appropriate NSM solver card the connector is referenced to.

> **Note:** Even though the connector is referenced to one specific group, it does not recognize whether this group is manually modified. This means that adding elements to the group will not automatically lead to updated link definitions on the connector. In addition, editing the lumped mass value on the NSM solver card is not synchronized with the connector mass. Deleting the group causes the connector to become unrealized.

Updating the connector links would unrealize the connector. Then the appropriate group is deleted.

The groups created along the NSM connector realization are named as nsm_group_<CE_ID>.nsm_group_<CE_ID>, and placed in the current include. A new component named CE_Mass_inc_<include name> is created which contains all the connector information for that include. Each include is meant to host its own CE_Mass_inc_<include name> component, and HyperMesh will produce a warning if the component name already exists in a different include. This helps ensure that unrealizing and rerealizing nsm connectors will keep the FE data in its original include.

The assigned NSM solver card can be of either the PROPERTY or ELEMENT type. This strongly depends on the NSM entity type attribute, which is defined in the panel. The attribute used during creation is written to the connector and is reused for a realization.

With these attributes you can define whether you want to create property- or element-based NSM groups during connector realization. In certain cases, if the defined links cannot be referenced by exclusive properties, a realization as property-based NSM group is not possible. In such cases the connector fails, or can optionally be realized as an element-based NSM group.

Absorption

The absorption works on all different types like PSHELL, PCOMP, PBAR, PBARL, PBEAM, PBEAML, PBCOMP and PROD.

During absorption the group definition is not modified. The connector is created in the center of a virtual box bounding all referenced elements. Upon NSM absorption, connectors are created inside newly created components (CE_Mass_inc_<include name>), which are created in each include that holds nsm entities.The absorbed connector contains the following information:

- Reference to a certain group
- Lumped mass value in the appropriate NSM solver card
- All elements listed in the NSM solver card. These elements are all defined somehow as connector links. If possible, the single elements are condensed in component links.

PAM-CRASH 2G Connector Types

Supported PAM-CRASH 2G connector types and property scripts.

Connector Types

PAM-CRASH 2G plink (connector position)

Creates a PLINK element. The PLINK is created at the connector location.

This realization also uses the `prop_plink.tcl`[1] property script.

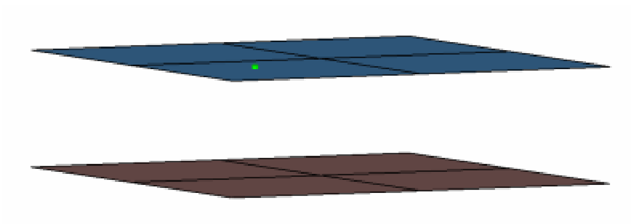


Figure 1415:

```
CFG pamcrash2g 1 plink (connector position)
*filter spot
*head
*body 0
mass 5 2
*post prop_plink.tcl
```

PAM-CRASH 2G plink (middle of the gap)

Creates a PLINK element. The PLINK is created at the center location between the two components and is offset from the connector location.

This realization also uses the `prop_plink.tcl`[1] property script.

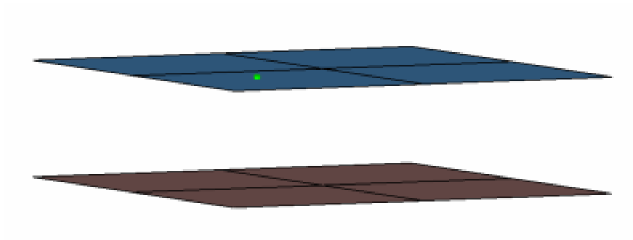


Figure 1416:

```
CFG pamcrash2g 2 plink (middle of the gap)
*filter spot
*head
*body 0
mass 5 1
*post prop_plink.tcl
```

PAM-CRASH 2G bolt (spider)

Creates an RBODY element. The body element projects and connect to the nodes of the adjoining shell elements.

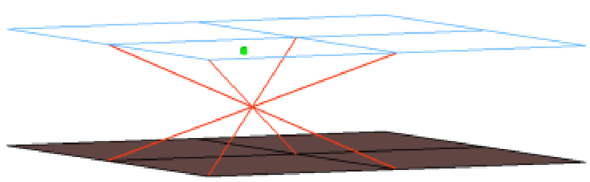


Figure 1417:

```
CFG pamcrash2g 54 bolt (spider)
*filter bolt
*style bolt 1
*head
*body 0
rigidlink 1 1
```

PAM-CRASH 2G link

Creates LLINK elements. The LLINK elements are created along the line connector.

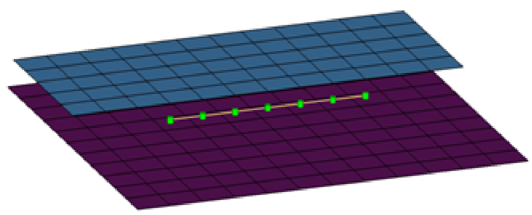


Figure 1418:

```
CFG pamcrash2g 55 llink
*filter seam
*style parallel 1
*head
*body 0
rod 5 1
*post prop_llink.tcl
```

PAM-CRASH 2G adhesives(contact)

This realization creates rows of HEXA and PENTA elements for the body. The HEXA and PENTA elements project and connect to the adjoining shell/solid elements by touching them.

The realization uses the `prop_pam_rad_adhesives.tcl`[2] property script.

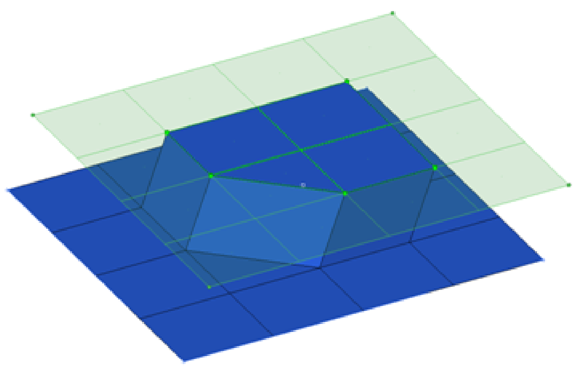


Figure 1419:

```
CFG pamcrash2g 56 adhesives (contact)
*filter area
*style adhesive 1
*head
*body 1
hex8 1 1
penta6 1 1
*post prop_pam_rad_adhesives.tcl
```

PAM-CRASH 2G hexa (adhesive-shell gap)

Creates a row of hexa elements for the body. The hexa elements are projected so that they touch the elements of the connecting components.

This realization uses the `prop_acm_adhesives.tcl`[3] property script.

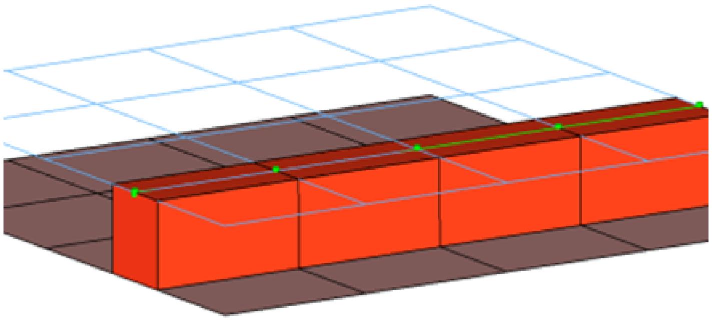


Figure 1420:

```
CFG pamcrash2g 57 hexa (adhesive - shell gap)
*filter seam
*style continuous 2
*head
*body 0
hex8 1 1
*post prop_pam_adhesives.tcl
```

PAM-CRASH 2G hexa (adhesive)

Creates a row of hexa elements for the body and numerous RBE2/RBE3 elements for the head. The head elements project and connect to the nodes of the adjoining elements. If there is a direct normal project then RBE2 element will be created, if there are only non-normal projections then RBE3 elements will be created. The hexa elements are projected so that they touch the elements of the connecting components.

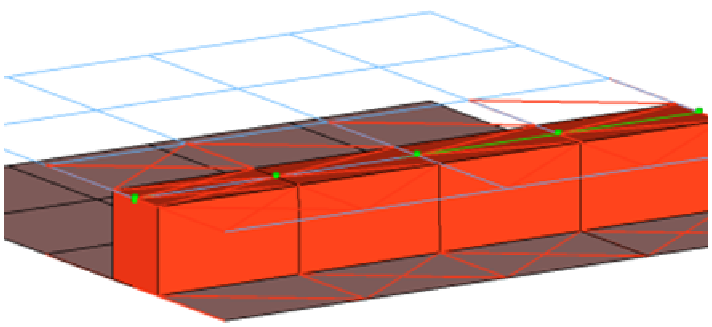


Figure 1421:

```
CFG pamcrash2g 58 hexa (adhesive)
*filter seam
*style continuous 3
*head
rbe3 1 0
rigid 1 0
*body 0
hex8 1 1
```

PAM-CRASH 2G hexa (tapered T)

Intended to be used for t-cases. The size and exact position can be defined thickness dependent, or the exact dimension and position parameters can be given.

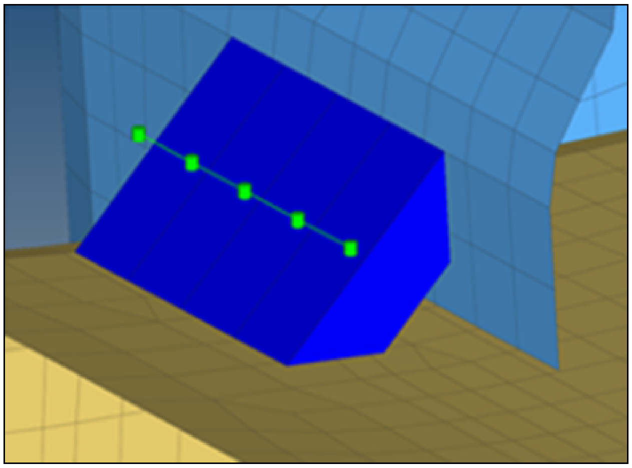


Figure 1422:

```
CFG pamcrash2g 105 hexa (tapered T)
*filter seam
*style continuous 6
*head
*body 0
hex8 1 1
```

Property Scripts

1. `prop_plink.tcl`

   This script is called while creation of PLINK– custom config welds in the Spot panel inside PAM-CRASH 2G user interface.

   This script does the following tasks:

   - Organizes the PLINK weld elements created during realization process into C_PLINK_PSCRIPT_<id1_id2> component with the PART_LINK card image associated to it. The id1 and id2 shown above refers to the ids of the link components with which the connector is connected to.
   - Creates the M_PLINK_PSCRIPT_<id1_id2> material collector, with the MAT_LINK card image associated with it. This material collector is referenced in the above created component containing the PLINK weld elements.
   - Updates the various attributes to the above created material/ property cards.
2. `prop_pam_rad_adhesives.tcl`

   The script performs the following tasks:

   - Creates TYPE2 interfaces (groups) with the names ADHESIVES_CONTACTS_PID_=_#, which reference the independent/dependent links' master sets, and the nodes' slave sets (# is the ID of the links).
   - Organizes the link entities (components, and so on) into sets with the names MASTER_PART_SET_PID, which in turn are referenced by the above interface groups (# is the ID of the link entity).

- Organizes the solids' nodes on links into sets with the names SLAVE_NODE_SET_PID_=_#, which in turn are referenced by the above interface groups (# is the ID of the link entity).
- Creates and assigns a property with the name Adhesive_Solid_Property and the card image P43_CONNECT to the solid component.
- Creates and assigns a material with the name Adhesive_Solid_Material and card image M59_CONNECT to the solid component.
- Creates a Failure Model with the name Failure_CONNECT_# and card image FAIL_CONNECT. The curves Adhesive_Solid_Material_YsvsNormalElong and Adhesive_Solid_Material_YsvsTangentialElong are required for the material definition.

> **Note:** For Radioss versions less than Block100 (Block51 and Block90), HyperMesh creates a property definition with the P14_SOLID card image, and a material definition with the M1_ELAS card image.

3. `prop_acm_adhesives.tcl`

The script performs the following tasks:

- Organizes the HEXA elements and nodes into a component with the name ADHESIVES_#_@ (# is the entity ID of the independent link and @ is the entity ID of the dependent link).
- Creates groups with the names ADHESIVES_CONTACTS_PID_=_#, which reference the independent/dependent links' master sets and the nodes' slave sets (# is the ID of the links).
- Organizes the link entities (components, and so on) into a set with the name MASTER_PART_SET_PID_=_#_parts, which is referenced by set MASTER_PART_SET_PID_=_#, which in turn is referenced by above link group (# is the ID of the link entity).
- Organizes the solids' nodes on links into a set with the name SLAVE_NODE_SET_PID_=_#_nodes, which is referenced by the set SLAVE_NODE_SET_PID_=_#, which in turn is referenced by the above link group (# is the ID of the link entity).

> **Note:** The names of the dependent links node sets are SLAVE NODE SET PID_=_#_nodes and SLAVE NODE SET PID_=_#.

Radioss Connector Types

Supported Radioss connector types and property scripts.

Connector Types

Radioss type2 (spring)

Creates a SPRING2N element for the body and plot elements for the head, the plot elements are created for visualization purposes and find operations.

This realization also uses the `prop_type2.tcl`[1] property script.

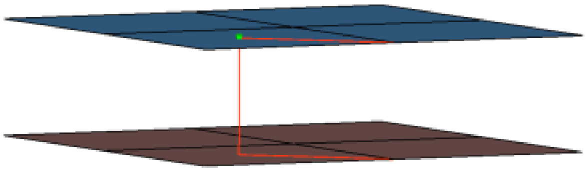


Figure 1423:

```
CFG radioss 2 type2 (spring)
*filter spot
*head
plot 1 0
*body 0
spring 1 1
*post prop_type2.tcl
```

Radioss bolt (general)

Creates RBODY elements for the head and SPRING2N body. The head elements project and connect to the nodes of the adjoining shell elements which form the hole. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

This realization also the `prop_radioss_rigidupdate.tcl`[2] property script.

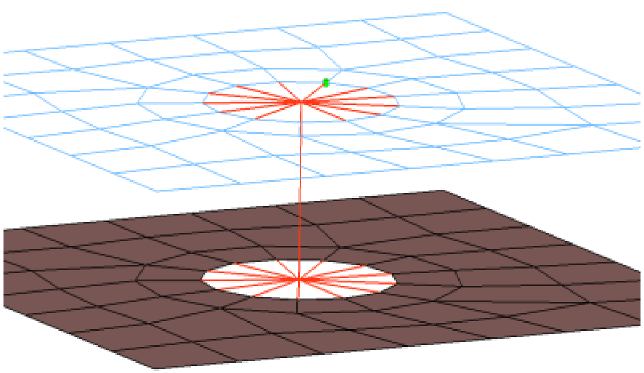


Figure 1424:

```
CFG radioss 52 bolt (general)
*filter bolt
*style bolt 0
*head
rigidlink 1 10
*body 0
spring 1 1
*post prop_radioss_rigidupdate.tcl
```

Radioss hinge

Creates RBODY elements for the head and SPRING2N body. The rot x degree of freedom is released so that the RBODY can rotate. The head elements project and connect to the nodes of the adjoining shell elements which form the hole. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

This realization also the `prop_radioss_rigidupdate.tcl`[2] property script.

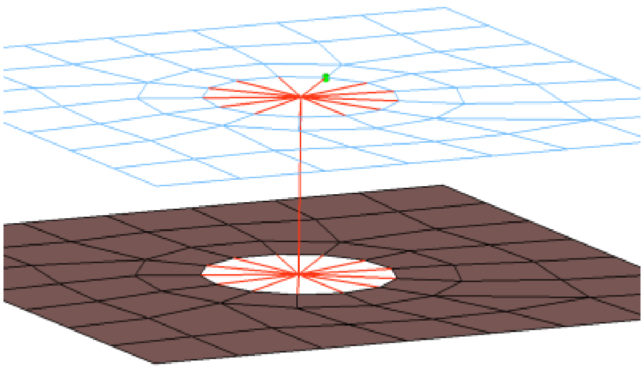


Figure 1425:

```
CFG radioss 53 hinge
*filter bolt
*style bolt 0
*head
rigidlink 1 10
*body 0
spring 1 1 dofs=4
*post prop_radioss_rigidupdate.tcl
```

Radioss bolt (spider)

Creates an RBODY element. The element projects and connect to the nodes of the adjoining shell elements which form the hole, the RBODY element is joined at the midpoint of the bolted connection. The connector location can either be on the edge of the hole, center of the hole, midpoint in between the two holes or on the second row of nodes which form the washer layer.

This realization also the `prop_radioss_rigidupdate.tcl`[2] property script.

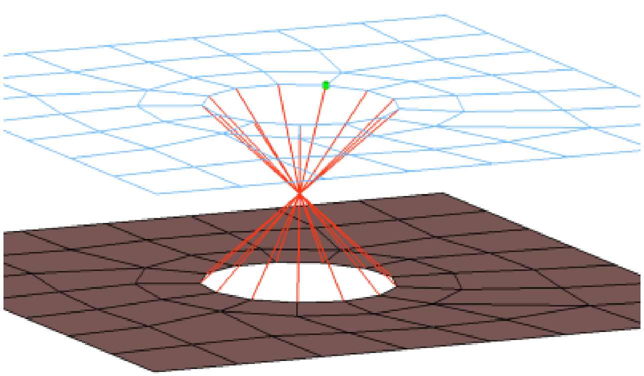


Figure 1426:

```
CFG radioss 54 bolt (spider)
*filter bolt
*style bolt 1
*head
*body 0
```

```
rigidlink 1 1
*post prop_radioss_rigidupdate.tcl
```

Radioss bolt (cylinder rigid)

Creates an RBODY element. Please reference "Cylinder Bolt" help for further details.

This realization also the `prop_radioss_rigidupdate.tcl`[2] property script.

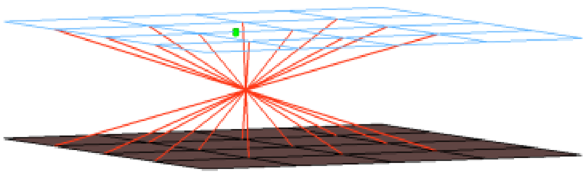


Figure 1427:

```
CFG radioss 60 bolt (cylinder rigid)
*filter bolt
*style bolt 3
*head
*body 0
rigidlink 1 1
*post prop_radioss_rigidupdate.tcl
```

Radioss bolt (cylinder spring)

Creates an RBODY elements and a zero length SPRING2N element. Please reference "Cylinder Bolt" help for further details.
The Radioss bolt (cylinder spring) exists in exactly the same configuration with the following additional names:

- HC deformable cylinder bolt
- HC deformable cylinder clip

This is in order to keep the realization names from HyperCrash.

This realization also the `prop_radioss_rigidupdate.tcl`[2] property script.

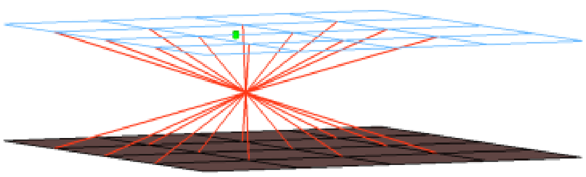


Figure 1428:

```
CFG radioss 61 bolt (cylinder spring)
*filter bolt
*style bolt 3
*head
rigidlink 1 1
*body 0
spring 1 1
```

```
*post prop_radioss_rigidupdate.tcl
```

Radioss type2 (adhesive-spring)

Creates multiple SPRING2N elements for the body and plot elements for the head, the plot elements are created for visualization purposes and find operations.

This realization also uses the `prop_type2.tcl`[1] property script.

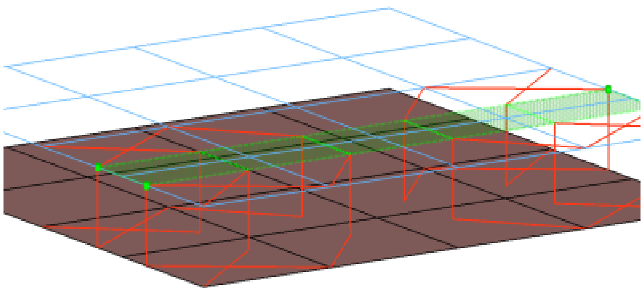


Figure 1429:

```
CFG radioss 62 type2 (adhesive-spring)
*filter area
*head
plot 1 0
*body 0
spring 1 1
*post prop_type2.tcl
```

Radioss rigidlnk (midnode)

Creates an RBODY with the independent node located in the middle between the two parts. Both parts are connected to the RBODY via independent nodes. The needed sets are created automatically.

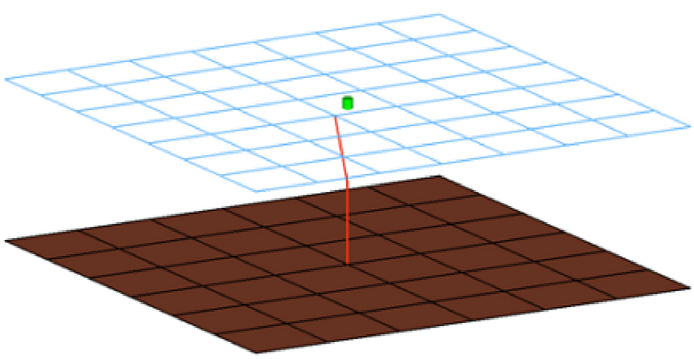


Figure 1430:

```
CFG radioss 63 rigidlnk (midnode)
*filter spot
*style mpc 2
*head
*body 0
rigid 1 1
```

Radioss HC cylinder rigid bolt

This realization creates a single RBE2 for the body, and projects and connects the element to nodes of the adjoining shell/solid within the prescribed cylinder diameter, L1 (cylinder height along the vector from the connector location) and L2 (cylinder height in the opposite direction of the vector from the connector location).

The realization uses the `prop_radioss_rigidbolts.tcl`[3] property script.

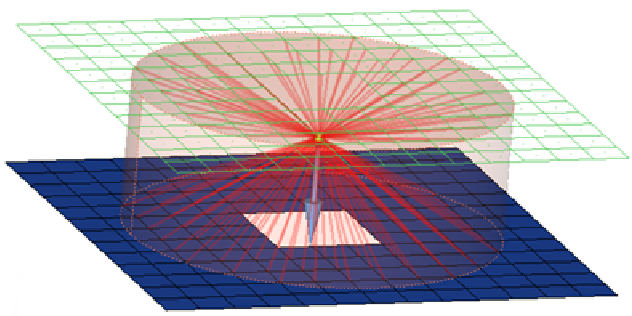


Figure 1431:

```
CFG radioss 64 HC cylinder rigid bolt
*filter bolt
*style bolt 4
*head
*body 0
rigidlink 1 1
*post prop_radioss_rigidbolts.tcl
```

Radioss HC cylinder spring bolt

This realization creates a SPRING element for the body, and projects and connects the element to nodes of the adjoining shell/solid elements with RBE2 elements within the prescribed cylinder diameter, L1 (cylinder height along the vector from the connector location) and L2 (cylinder height in the opposite direction of the vector from the connector location).

The realization uses the `prop_radioss_rigidbolts.tcl`[3] property script.

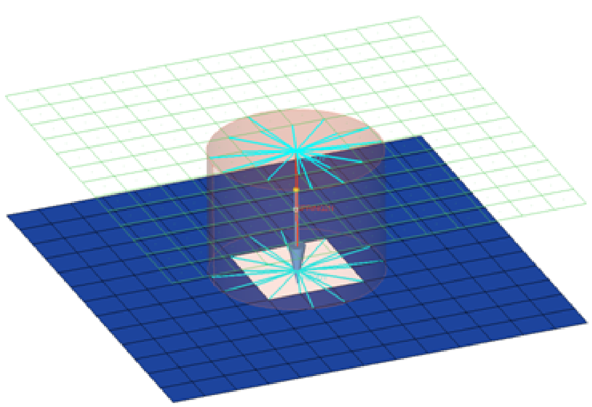


Figure 1432:

```
CFG radioss 65 HC cylinder spring bolt
*filter bolt
```

```
*style bolt 4
*head
rigidlink 1 1
*body 0
spring 1 1
*post prop_radioss_rigidbolts.tcl
```

Radioss type2 (spring multiple row)

Creates a certain pattern of SPRING2N elements.

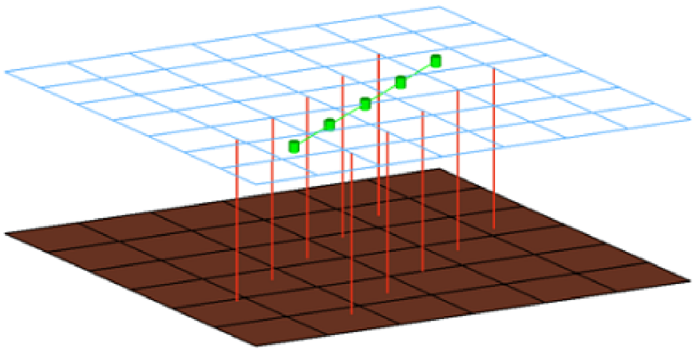


Figure 1433:

```
CFG radioss 66 type2(spring multiple row)
*filter seam
*style continuous 4
*head
*body 0
spring 1 0
*post prop_type2radioss.tcl
```

Radioss type2 (spring single row)

Creates a certain pattern of SPRING2N elements.

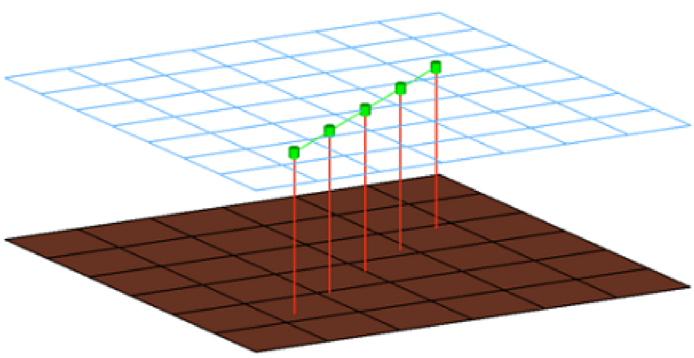


Figure 1434:

```
CFG radioss 67 type2(spring single row)
*filter seam
*style continuous 4
*head
*body 0
spring 1 0
*post prop_type2radioss.tcl
```

Radioss bolt (2 cylinder rigid)

Creates a RBODY per part with the center nodes connected to the SPRING2N element. Which nodes belong to the RBODY is determined with a defined cylinder volume. If a hole per part can be found in the defined cylinder, only the nodes of the holes are connected. Otherwise all nodes in the cylinder belong to the RBODY element (per part).

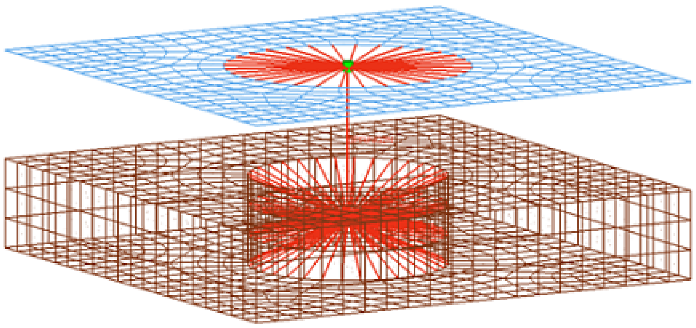


Figure 1435:

```
CFG radioss 68 bolt (2 cylinder rigid)
*filter bolt
*style bolt 5
*head
rigidlink 1 1
*body 0
spring 1 1
*post prop_boltsradioss.tcl
```

Radioss HC hexa spotweld

This realization creates various configurations of hexas for the body, and the hexas project and connect to the adjoining shell/solid elements by touching the shell/solid elements.

This realization uses the `prop_radiosshexa.tcl`[4] property script.

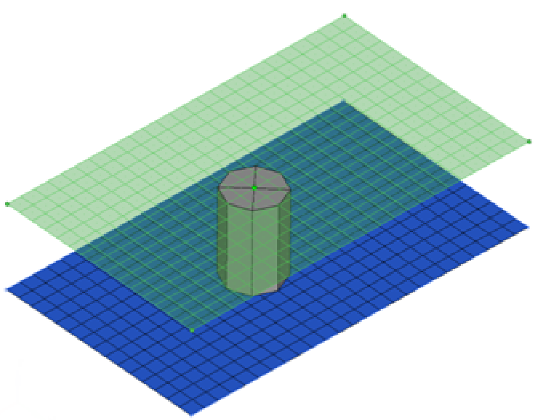


Figure 1436:

```
CFG radioss 69 HC hexa spotweld
*filter spot
*head
*body 0
hex8 1 1
```

```
*post prop_radiosshexa.tcl
```

Radioss adhesive(contacts)

This realization creates rows of HEXA and PENTA elements for the body. The HEXA and PENTA elements project and connect to the adjoining shell/solid elements by touching them.

This realization uses the `prop_pam_rad_adhesives.tcl`[5] property script the HC_HexaAdhesive.rad config file.

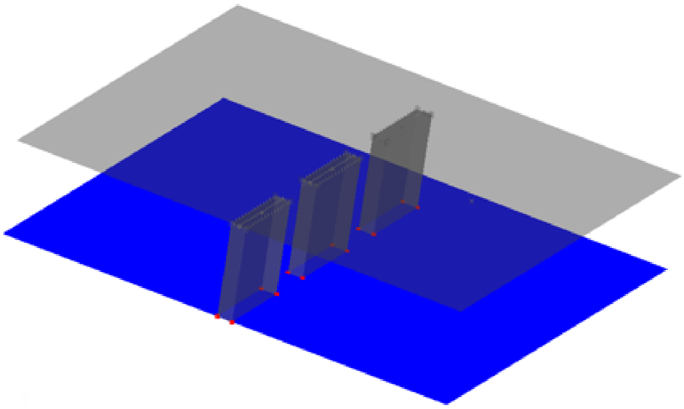


Figure 1437:

```
CFG radioss 70 adhesive(contact)
*filter area
*style adhesive 1
*head
*body 1
hex8 1 1
penta6 1 1
*post prop_pam_rad_adhesives.tcl
```

Radioss acm(shell gap contact and coating)

This realization creates hexa clusters between shell components. Contacts get defined between the shell components and the appropriate hexa nodes. A heat affected zone for the shells from ultra high strength steel material is created.

This realization uses the `prop_rad_acm_shellgapcoating.tcl`[6] property script and the `uhss_washersolid_matprop.rad config` file.

```
CFG radioss 71 acm (shell gap contact + coating)
*filter spot
*style acm 5
*body 0
hex8 1 1
*post prop_rad_acm_shellgapcoating.tcl
```

Radioss hexa(adhesive - shell gap)

This realization creates rows of HEXA elements for the body. The HEXA elements project and connect to the adjoining shell/solid elements by touching them.

This realization uses the `prop_pam_rad_adhesives.tcl`[5] property script and the `HC_HexaAdhesive.rad` config file.

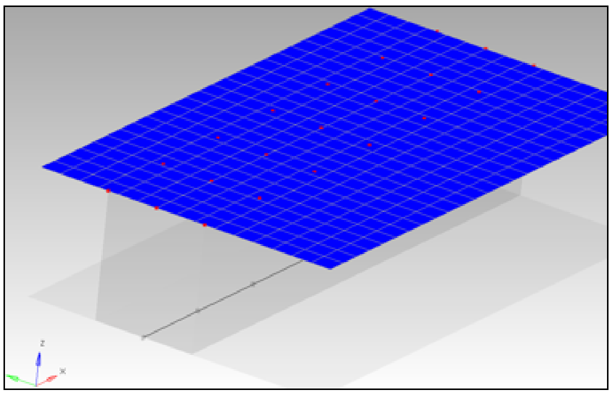


Figure 1439:

```
CFG radioss 72 hexa (adhesive - shell gap)
*filter seam
*style continuous 2
*head
*body 0
hex8 1 1
*post prop_pam_rad_adhesives.tcl
```

Radioss hexa (tapered T)

Intended to be used for t-cases. The size and exact position can be defined thickness dependent, or the exact dimension and position parameters can be given.

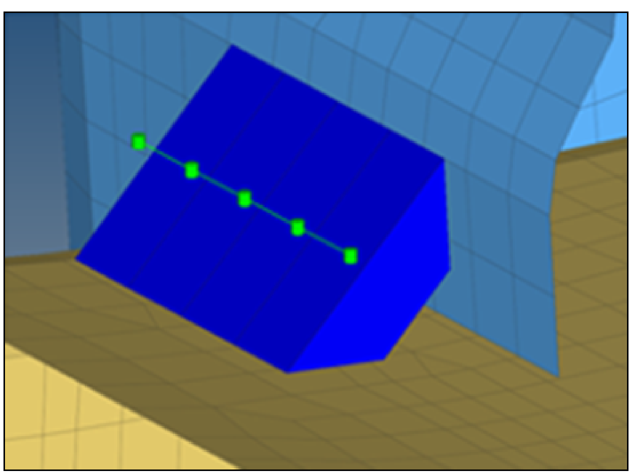


Figure 1440:

```
CFG radioss 105 hexa (tapered T)
*filter seam
*style continuous 6
*head
*body 0
hex8 1 1
```

Automatic Exclusion of Special Nodes During Rigid Bolt Realization

HyperMesh automatically excludes special nodes as potential slave nodes for any rigid bodies created during bolt realization, even though they fall with the virtual Bolt Cylinder diameter. Nodes that are shared by the following entities are considered special nodes:

- rigid links
- Rigids
- Rbe3 nodes
- boundaries condition
- IMPDISP
- IMPVEL
- IBVEL
- IMPACC
- FXBODY
- Nodes inside the Interfaces with Type2

Property Scripts

1. `prop_type2.tcl`

   This script is used while creation of RADIOSS [Type2 Spring] in the Spot panel.

   This script does the following tasks:

   - Organizes the realized weld elements to the respective components based upon the link they are connected to. For example, if a weld is created between comp_1(1) and comp_2(2) it creates a component collector with the name RW^^_<id1_id2> and organizes all the welds created as links between these two components into this collector.
   - Creates the following properties collectors:

     RW^^_<id1_id2>
     This property collector (with the P13_SPR_BEAM in case of R-BLOCK and SectBemSpr in case of R-FIX solver subtype) is associated with it as the card image.
   - Creates sets in the following order:

     I1_M_<id1_id2>
     This contains the master as the FIRST link component ID to which the weld is connected to.

I1_S_<id1_id2>
: This set contains the node ID of the projected spring element to the above component link as the slave node N1.

I2_M_<id1_id2>
: This contains the master as the SECOND link component ID to which the weld is connected to.

I2_S_<id1_id2>
: This set contains the node ID of the projected spring element to the above component link as the slave node N2.

- Creates two interfaces Groups (interfaces) for the spring weld elements created between the same component links by the name

  RW^^1_<id1_id2>
  : This references the above created sets that contain the ids of first node NI and first component Link C1.

  RW^^1_<id1_id2>
  : This references the above created sets that contain the ids of second node N2 and second component Link C2.

- Creates a plot named Shear_Normal_Force_Plot with two curves from the Normal Force Function [named RW^^FN_1.0] and Shear Force Function [named RW^^FS_2.5], the values of which are read from the `Radiossweld_config.ini` file.

2. `prop_radioss_rigidupdate.tcl`

   This script is run for all the rigid/rigidlink weld configurations in the Radioss user profile. It creates the sets of all the slave node ids of the rbodies created during the realization process, and assigns the GRNOD card image to them. It also updates some attributes of these cards.

3. `prop_radioss_rigidbolts.tcl`

   The script performs the following tasks:

   - Organizes the SPRING element into a component with the name HM_Bolt_SPRING.
   - Organizes the RBE2 elements into a component with the name Realize#001 (# is a number starting with 2 and increments as 2, 4, 6, and so on).
   - Creates a view with the name radioss_rigid_bolts.

   > **Note:** A new component HM_Bolt_SPRING will only be created if their is not a component with the same name that already exists; otherwise the existing component will be used.

4. `prop_radiosshexa.tcl`

   The script performs the following tasks:

   - Organizes the hexas into a component with the name Realize_#001 (# is a number starting with 2 and incrementing as 2, 4, 6 for every new component).

4. `prop_pam_rad_adhesives.tcl`

   The script performs the following tasks:

   - Creates TYPE2 interfaces (groups) with the names ADHESIVES_CONTACTS_PID_=_#, which reference the independent/dependent links' master sets, and the nodes' slave sets (# is the ID of the links).
   - Organizes the link entities (components, and so on) into sets with the names MASTER_PART_SET_PID, which in turn are referenced by the above interface groups (# is the ID of the link entity).
   - Organizes the solids' nodes on links into sets with the names SLAVE_NODE_SET_PID_=_#, which in turn are referenced by the above interface groups (# is the ID of the link entity).
   - Creates and assigns a property with the name Adhesive_Solid_Property and the card image P43_CONNECT to the solid component.
   - Creates and assigns a material with the name Adhesive_Solid_Material and card image M59_CONNECT to the solid component.
   - Creates a Failure Model with the name Failure_CONNECT_# and card image FAIL_CONNECT. The curves Adhesive_Solid_Material_YsvsNormalElong and Adhesive_Solid_Material_YsvsTangentialElong are required for the material definition.

   > **Note:** For Radioss versions less than Block100 (Block51 and Block90), HyperMesh creates a property definition with the P14_SOLID card image, and a material definition with the M1_ELAS card image.

6. `prop_rad_acm_shellgapcoating.tcl`

   This script performs the following tasks:

   - For each connected link the contact /inter/TYPE2/ gets created and is named TYPE2_CONTACT_PID_<link ID>. The following sets are created and referenced.

     MASTERPART_SET_PID_<link ID>
     In this set, which is referenced as the master by the above mentioned contact, the link entities like the component get organized.

     SLAVENODE_SET_PID_<link ID>
     In this set, which is referenced as the slave by the above mentioned contact, the hexa nodes projecting onto the master entities get organized.

   - For each link combination the hexa clusters are organized into separate components and named RAD_SOLID_SPOTWELD_PID_<link1 ID>_<link2 ID>. All components are assigned the following material and property:

     RAD_SOLID_SPOTWELD_DEFAULT_MAT.
     This material is defined as /MAT/LAW59/.

     RAD_SOLID_SPOTWELD_DEFAULT_PROP
     This property is defined as /PROP/CONNECT/.

     The default values are read from `uhss_washersolid_matprop.rad` in the installation.

- The heat affected zone elements (washer) are organized into one separate component for each link from the ultra high strength steel material and named RAD_WASHER_PID_<link ID>. All components are assigned the following material and property:

  RAD_WASHER_MAT
  This material is defined as /MAT/PLAS_JOHNS/.

  RAD_WASHER_PROP
  This property is defined as /PROP/SHELL/.

  The material and property values are read from `uhss_washersolid_matprop.rad` in the installation.

Autopitch

Create weld points at a predefined pitch distance so that the model build process can continue without the need to wait for the published weld data from CAD.

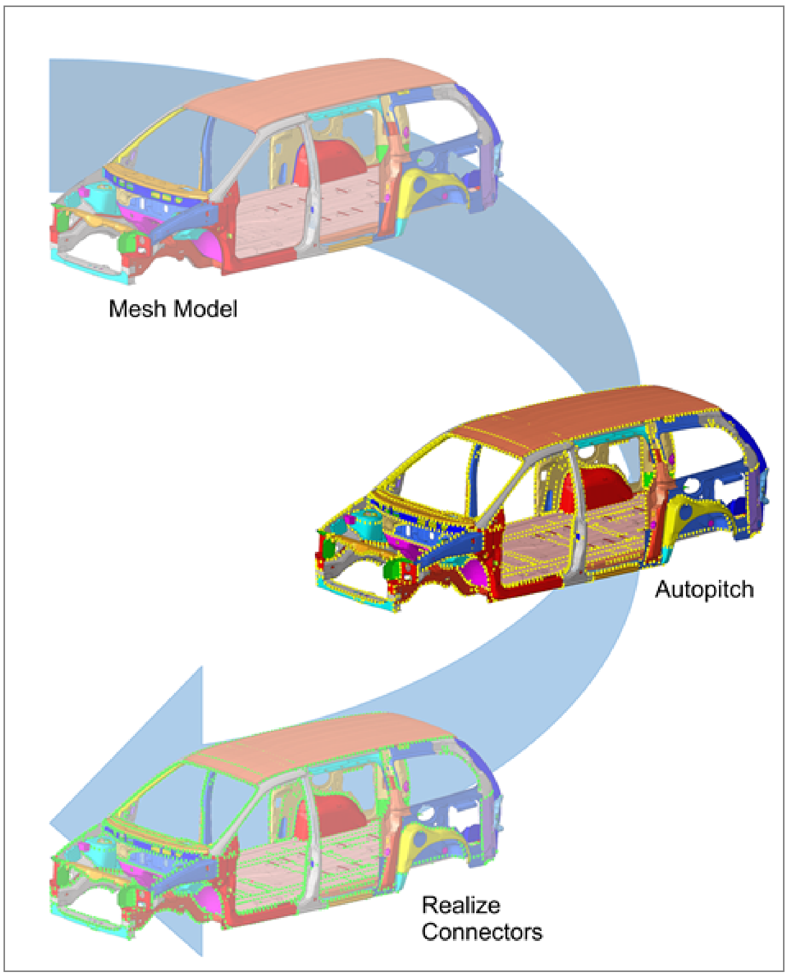


Figure 1441:
Starting with a CAD model, the Autopitch tool creates unrealized connectors (yellow) which are then realized via the Spot panel.

1. From the menu bar, click **Connectors** > **Create** > **Autopitch**.
   The **Autopitch** dialog opens.
2. Using the Components selector, select the components to automatically add connectors to.

   All selected components receive connectors with the same qualities to help you perform a blanket application of connectors-using the same pitch-to all components that need them as a single operation. Be wary of simply selecting the whole model, however, as this could result in undesirable actions, such as adding welds to a cars tires.
3. To use shell meshes that enclose a volume (some small gaps are allowed) as input, enable the **Consider closed shell thin solids** checkbox.

   For example, the outer skin of a solid part can be shell meshed and used as input to create connectors. Standard mid-plane meshes are also still considered when this option is used.
4. To create connectors in the middle of the found flanges, enable the **Create in middle** checkbox.

   By default, connectors are created on either one of the flanges. This applies to both mid-plane and closed shell thin solid inputs, when appropriate.
5. Define settings.
   a) In the **Search** distance field, enter the distance to consider between components.
   b) In the **Spot pitch distance** field, enter the distance between each connector.
   c) In the **Spot pitch end offset** field, enter the distance from the end of an edge/flange to the connector.
   d) In the **Distance from free edge** field, enter the distance from the free edge to the connector.
   e) In the **Feature angle** field, enter the angle used to segregate the model into faces that are planar within its specified value.
   f) In the **Max deviation from avg dist (%)** field, enter the average distance that can be calculated based on the estimation that the distance between two flanges does not change too much in the areas where connectors should be placed.

      If the distance at the position where a connector is planned exceeds the given deviating value, no connector will be created at that position.

      This segregation is used to identify where autopitch connectors are placed. For example, faces found to have significant topological complexity are not used to create autopitch connectors.

      > **Note:** By reducing its value, the complexity of some of these faces is generally reduced. Of course, flat regions are unaffected by the parameter.
6. Click **create**.

The created output is a series of connectors with the appropriate pitch distance and other associated parameters. These connectors are in the unrealized state, thereby allowing the connector to be realized to a configured state.

The output is organized in the current component collector. If there is no current component collector, then a new collector called ^autopitch is created.

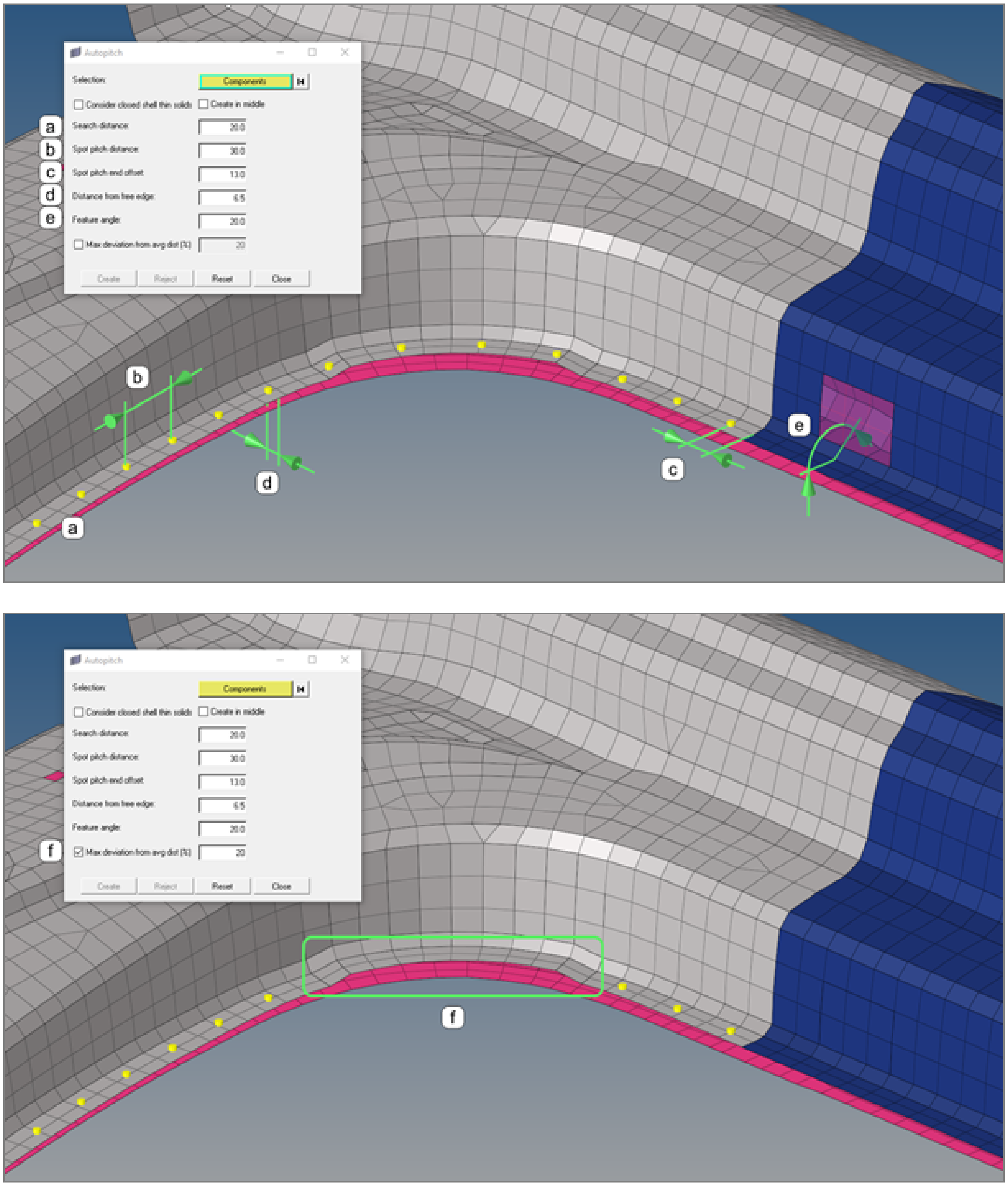

Autopitch
Selection:
Components
Consider closed shell thin solids
Create in middle
a
Search distance:
20.0
b
Spot pitch distance:
30.0
c
Spot pitch end offset:
13.0
d
Distance from free edge:
6.5
e
Feature angle:
20.0
f
Max deviation from avg dist (%)
20
Create
Reject
Reset
Close
a
b
c
d
e
f

Figure 1442:

Create Connector Realizations using the FEMSITE Utility

Create specialized connector realization types which can be used in fatigue calculations with the third party FEMSITE tool.

In HyperMesh, use the FEMSITE utility to preprocess models and submit connection information to the FEMSITE generator.

Setup the FEMSITE Utility

Use the third party FEMSITE utility to preprocess models and submit connection information to the FEMSITE generator.

The FEMSITE options will only appear if you have an installation of FEMSITE on your computer.

> **Restriction:** The FEMSITE utility is only available in the Nastran user profile for Windows and Linux platforms.

Add the `hmfemsite.ini` initialization file to one of the following locations:

HyperMesh searches all locations for the `hmfemsite.ini` file in the order listed above, and uses the last found existing file to parse the variables pointing to your FEMSITE installation.

- `$INSTALL_HOME/hm/bin/win64/hmfemsite.ini` (win64 or linux64)
- `HW_CONFIG_PATH/hmfemsite.ini`
- `$HOME/hmfemsite.ini`
- `$WORKING_DIR/hmfemsite.ini`

Example hmfemsite.ini File

Lines that do not contain the strings FEMSITE_DATA_DIR=, FEMSITE_LINUX_DATA_DIR=, FEMSITE_EXE=, or FEMSITE_LINUX_EXE= will be ignored, and can be treated as a comment. If HyperMesh does not locate the FEMSITE exe or the data directory, the FEMSITE options will not load.

Welds need to have a minimum amount of metadata in order to be realized within FEMSITE. The Define Spotweld, Define Robscan, and Define Rivet options in the FEMSITE utility allow the assignment of this metadata from within HyperMesh (instead of via CONN file) for spotwelds, Robscans and Rivets.

```
This file can be edited to point Hypermesh to the FEMSITE installation:
This is the path to the FEMSITE data directory:
FEMSITE_DATA_DIR=W:\FEMSITE\installfemsite31\src\BOTH\english
FEMSITE_LINUX_DATA_DIR=W:\FEMSITE\installfemsite31\src\BOTH\english

This is the path to the executable:
FEMSITE_EXE=W:\FEMSITE\installfemsite31\src\BOTH\english\femsite_swv31.exe
This is the path to the executable:
```

```
FEMSITE_LINUX_EXE=/apps/FEMSITE/femsite_swv31
```

Define Spotwelds

Define a spotweld with the FEMSITE utility ensures that you will be able to create a FEMSITE spotweld within HyperMesh, and assign the proper metadata to the connector for successful realization.

Before you begin, Setup the FEMSITE Utility.

1. From the menu bar, click **Connectors** > **FEMSITE** > **Define Spotweld**.
   The **Create Spotweld Connectors Interactive** dialog opens.
2. Using the Select Connectors selector, select input.
   - Choose **Nodes** to create new connectors with nodes.
   - Choose **Connectors** to change existing connectors to FEMSITE connectors.
3. In the Element Label field, enter a label to be used in FEMSITE.
4. In the Property ID field, enter an ID to be sent to the FEMSITE executable.
5. In the Nugget Diameter field, enter a diameter to be sent to the FEMSITE executable.
6. Define part search parameters.
   a) For RefMode, choose to use the **Property** or **Part** ID as the reference mode for connector assignment.
   b) Using the Select Components selector, select the components that contain the parts to search.
      The RefList field lists the assigned IDs that are found after components are selected.
   c) In the Max number of interactions field, enter the maximum number of connector links to create.
   d) In the Maximum distance field, enter the search tolerance the FEMSITE executable should use for realization.
7. Click **Define**.

Define Robscan Weld

Defining a robscan weld with the FEMSITE utility.

Before you begin, Setup the FEMSITE Utility.

1. From the menu bar, click **Connectors** > **FEMSITE** > **Define Robscan**.
   The **Rob Scan Data** dialog opens.
2. Using the Select Connectors selector, select input.
   - Choose **Nodes** to select a node location as input.
   - Choose **Connectors** to select an exiting connector, for example a connector imported from a connector file.
3. In the Property ID field, enter an ID to be sent to the FEMSITE executable.

4. Select a weld pattern.
   The values in the Width, Length, Seam Width, and Gap Size fields are populated after a Weld Pattern has been selected.
5. Define part search parameters.
   a) For RefMode, choose to use the **Property** or **Part** ID as the reference mode for connector assignment.
   b) Using the Select Components selector, select the components that contain the parts to search.
      The RefList field lists the assigned IDs that are found after components are selected.
   c) In the Max number of interactions field, enter the maximum number of connector links to create.
   d) In the Maximum distance field, enter the search tolerance the FEMSITE executable should use for realization.
6. Define the orientation of the weld.
   a) In the Pattern Orientation Vector (X) fields, enter the triad (x,y,x) and orientation angle.
   b) To reverse both direction inputs, enable the **Invert robscan direction (Z)** checkbox.
   c) To reverse the pattern orientation, enable the **Invert pattern Orientation** checkbox.
   d) To preview the orientation of the weld, click **?**.
7. Click **Define**.

Define Rivets

Define a rivet configuration with the FEMSITE utility.

Before you begin, Setup the FEMSITE Utility.

1. From the menu bar, click **Connectors** > **FEMSITE** > **Define Rivet**.
   The **Self Piercing Rivet Data** dialog opens.
2. Using the Select Connectors selector, select input.
3. Define the rivet and die.
   a) For Rivet, select a Label.
      The values in the Shaft Diameter, Head Diameter, Length, Hardness, and Geometry fields are dependent on the Label selected.
   b) For Die, select a Label.
      The values in the Diameter and Depth field are dependent on the Label selected.
4. Define part search parameters.
   a) For RefMode, choose to use the **Property** or **Part** ID as the reference mode for connector assignment.
   b) Using the Select Components selector, select the components that contain the parts to search.
      The RefList field lists the assigned IDs that are found after components are selected.
   c) In the Max number of interactions field, enter the maximum number of connector links to create.

d) In the Maximum distance field, enter the search tolerance the FEMSITE executable should use for realization.

5. To reverse the default direction input, enable the **Invert Rivet Direction** checkbox.

   The rivet direction is defined by the normal of the shells.
6. Click **Define**.

Import Metadata and Connectors from a *.conn File

Import metadata and connectors from a `*.conn` file with the FEMSITE utility.

Before you begin:

- Setup the FEMSITE Utility
- Properly format and define connectors and metadata in the `*.conn` file. For more information, refer to Required *.conn File Format and Guidelines.

, .

You do not need to import a FE model into HyperMesh prior to importing a `*.conn` file.

1. From the menu bar, click **Connectors** > **FEMSITE** > **Import CONN**.
2. In the **Select conn-File** dialog, open the `*.conn` file.
   The connectors and metadata that are correctly and completely defined are imported and translated to a XML connector format.
3. If there were unsupported or incorrectly defined connectors, the **Editor dialog** opens, from which you can save these connectors in a separate file by clicking **Save**.

   > **Note:** You can add this information again during export.

Required *.conn File Format and Guidelines

In order for `*.conn` files to be successfully imported, they must be properly formatted, and connectors and metadata must be properly defined.

- The separator must be a semicolon.
- The line containing the definition and description of the Format (content of the columns) must start with the type of connector, and must have 25 entries.
- Lines with a "&" can only be used within a connector definition.
- Connector IDs must be higher than 10,000, and can only have a maximum of 5 digits.
- Supported connector types: spotwelds, weldlines, and bondlines.
- Supported methods: RS1, SN, LASER, FILLER, MIGMAG, BL_ST, and BL_SR.
- The following options must be specified for the supported connectors: first dimension of tangent, second dimension of tangent, and dimension of normal vector.

- A reference mode must be specified: Part-ID or property-ID. The reference mode defines which components or property IDs are connected.
- The x, y, and z position, and the tangent and normal vector must be defined.
- Only 2 parts are allowed for spotwelds and weldlines.
- Parts must exist in the model.
- MAX_NUMB_CONN_PARTS must be defined for spotweld connectors that have a reference mode of AUTO.
- SELF_PIERCING* must be defined for spotweld connectors that have PROC=SN.
- ROB_SCAN_SEAM_WIDTH, ORIENTATION_ANGLE, and GAPSIZE must be defined for spotweld connectors that have PROC=ROBSCAN.
- MAXIMUM_DISTANCE must be defined for spotweld, weldline, and bondline connectors.
- ADJUST_LIMIT must be defined for spotweld, weldline, and bondline connectors.
- POSITION_IN_PART_TREE must be defined.
- WELD_POSITION_ID and WELD_ANGLE must be defined for weldline connectors.

If `*.conn` files are missing important parameters, the following attributes are set:

- TYPE is set to spotweld (for spotwelds) or bondlines (for seamwelds).
- JEID is set to the ID of the connector in HyperMesh.
- PROC is set to 21 (for spotwelds) or bl (for seamwelds).
- REFMODE is set to PROPERTY.
- TAN_DIM1 is set to 5.0.

The internal HyperMesh parameter ce_diameter is synchronized with TAN_DIM1 during import. This parameter is used for realization in HyperMesh (diameter).

Import PART/PID Mapping Files

Import PART/PID mapping `*.csv` files generated by the BOM tool into the FEMSITE utility,

Before you begin, Setup the FEMSITE Utility.

This function is used with tailored blanks, where one part consists of several property IDs. REFMODE is set to PROPERTY when referencing two properties, or it is set to PART when referencing parts.

For every PID, there is a corresponding component which is used to reference a connector. For every PART, there is a corresponding assembly which is used to reference a connector.

The mapping of components/assemblies (PID/PART) can be done using an external file. The external file specifies the assembly structure in HyperMesh.

```
A2126120114;QUERTRAEGER UNTER FAHRERSITZ VO LI;26120114;Shell  ;1700200;HT700T;Stahl
 EN 10336-  HT700T +ZE+A+O;1,50;;;;;;;;;;;;Stahl EN 10336-  HT700T +ZE+A
+O;1,5;;;;;;;;;;;;
A2126160516;VERSTAERKUNG LI VO HPT.BODEN;26160516;Shell  ;1100500;HC340LA;Stahl
 HC340LA EN10268-1.0548+ZE+A+O / DBL:4062.51;1,00;;;;;;;;;;;;Stahl HC340LA
 EN10268-1.0548+ZE+A+O / DBL:4062.51;1;;;;;;;;;;;;
```

```
A2126160716;VERSTAERKUNG HI LI / HAUPTBODEN;26160716;Shell  ;1100500;HC340LA;Stahl
 HC340LA EN10268-1.0548+ZE+A+O / DBL:4062.51;1,00;;;;;;;;;;;;Stahl HC340LA
 EN10268-1.0548+ZE+A+O / DBL:4062.51;1;;;;;;;;;;;;
```

1. From the menu bar, click **Connectors** > **FEMSITE** > **Import PART/PID mapping**.
2. In the **Select Part/Property File** dialog, open the `*.csv` file.
   Only the parts that are correctly and completely defined are imported.
3. If there were parts that could not be converted, the **Editor dialog** opens, from which you can save these parts in a separate file by clicking **Save**.

   > **Note:** You can add this information again during export.

Export Connector Data

Connector data that has been imported or defined in HyperMesh can be exported to `*.conn` files.

Before you begin, Setup the FEMSITE Utility.

1. From the menu bar, click **Connectors** > **FEMSITE** > **Export CONN**.
2. In the **Select conn-File** dialog, save the `*.conn` file to your working directory.

The supported connector information and the unsupported connector information that you saved during import is converted to CONN format and exported to a `*.conn` file.

Filter Connectors using Metadata

Filter connectors in the modeling window based on metadata using the FEMSITE Filter tool.

Before you begin, Setup the FEMSITE Utility.

1. From the menu bar, click **Connectors** > **FEMSITE** > **Filter**.
2. In the **Filter connectors** dialog, define connector criteria accordingly.
3. Click **Submit**.

Connectors that do not meet the filter criteria are masked in the modeling window.

Check Connector Connections

Check if the connection partners of connectors are available in the HyperMesh database.

Before you begin, Setup the FEMSITE Utility.

From the menu bar, click **Connectors** > **FEMSITE** > **Check Connected**.

After a check is performed, the number of connectors that have no existing connector partners in the HyperMesh database is reported, and failed connectors are displayed.

There is no geometrical check done if a projection is possible, only the availability is checked.

When REFMODE is set to Part, only assemblies are checked. PIDS are not checked.

Review and Edit Connector Metadata

Review and edit connector metadata that was created during import using the FEMSITE Review Conn tool.

Before you begin, Setup the FEMSITE Utility.

All of the supported connector metadata that can be imported is listed in the `parameter.cfg` file, which can be found in the installation: `/config/parameter.cfg`.

> **Tip:** If certain metadata is not applicable to you, create comments for them using "#". HyperMesh will ignore metadata commented with "#".

1. From the menu bar, click **Connectors** > **FEMSITE** > **Review CONN**.
2. In the panel area, use the connectors selector to select one or more connectors.
3. Click **proceed**.
4. In the **Edit params** dialog, review and edit the corresponding metadata of the selected connector(s).

   > **Note:** If you selected multiple connectors, only identical metadata is displayed. Blank fields indicate that metadata was not identical for the selected connectors. If metadata was not defined, field will read "not defined".

5. Click **Submit**.

Assign Metadata to Existing Connectors Imported from a *.conn File

Resolve undefined custom type connectors imported from a `*.conn` by assigning them metadata.

Before you begin, Setup the FEMSITE Utility.

If you are working with an HyperMesh model that contains existing connectors that were imported from a `*.conn` file, they will appear in the Connector browser as undefined custom types. In order for these connectors to appear with the correct connector type, they must be assigned metadata.

From the menu bar, click **Connectors** > **FEMSITE** > **Assign Metadata to assign metadata**.

Model Build and Assembly 9

Create, organize, and manage the CAE parts.

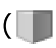
This chapter covers the following:

Manage Parts

About Parts, Part Assemblies, Part Instances

A part is an engineering representation of a physical part, and a part assembly is a group of part assemblies and/or parts. Part Instances, which are recognized from PDM, are automatically converted to Part Instances on import into HyperMesh.

Parts and Part Assemblies

HyperMesh supports CAE parts () and part assemblies () and PDM parts () and part assemblies ().

Parts assemblies and parts facilitate the one-to-one mapping of a CAD hierarchy into the HyperMesh environment as a CAE hierarchy. The CAE hierarchy can be created manually or can be imported from a PDM system via a neutral file format such as PLMXML.

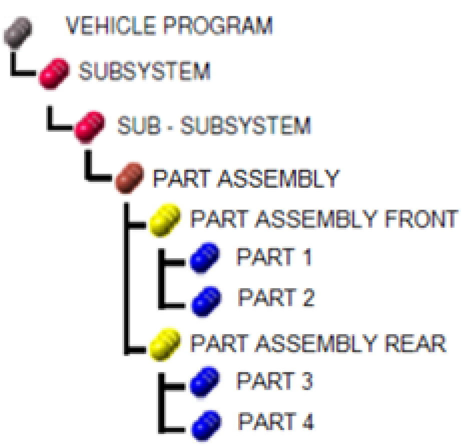


Figure 1443: PDM BOM

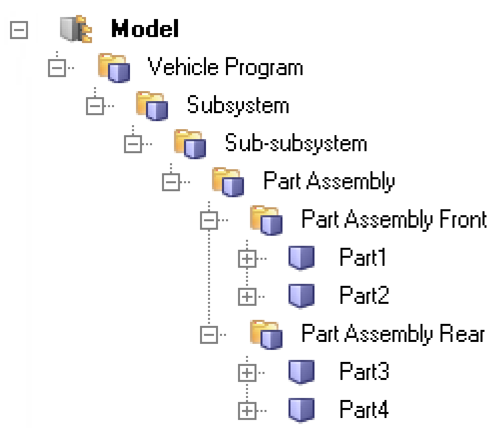


Figure 1444: HyperMesh BOM

If a part is comprised of multiple components, such as cast parts or tailor-welded blanks, you can perform actions such as visualization view modes, model management, and connector link definition at the part level.

Parts can be used to organize a physical part that is made up of multiple components in a CAE model as a single part.

When using a part, you can organize multiple components that represent physical parts into a single part.

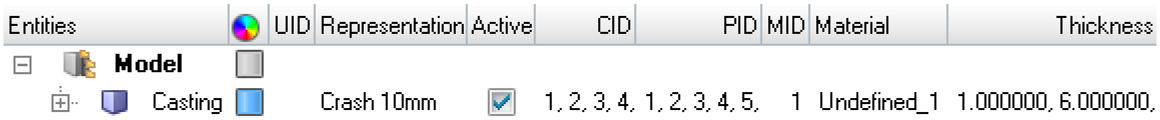


Figure 1445: Cast Part in Part Browser

In the Part Browser, collapsing all folders at the part level enables you to see all owned and referenced entities. Expanding folders at the part level displays a nested list of all referenced components and their entity specific attribute values which are shown in the respective columns.

Part Instances

The Part browser provides limited support for part instances of externally sourced BOMs.

Part Instances ( ) share UID and Part names. The name assigned to child instances is incremented by `*.ixx`, where `xx` indicates the number of the child instance. The positioning of Part Instances is dictated by the transformation matrix applied to the Part in the PDM system. Part Instances can also be created interactively in a session. Upon creation, you will automatically be prompted to transform the part via the Transform tool.

In circular, symmetric, or rectangular patterns, certain parts may appear in a BOM multiple times. Typical examples include bolts, hinges, and symmetric parts.

Creating or saving part representations for instantiated parts saves a single representation in the repository. Importing instantiated parts positions them according to their respective 4 x 3 transformation matrices.

Create Parts, Part Assemblies, and Part Instances

Only CAE parts and part assemblies can be manually created in HyperMesh. PDM parts and part assemblies must be imported imported into HyperMesh.

In the Part Browser, right-click and select **Create** > **<entity type>** from the context menu.

The new part, part assembly, or part instance is created and displayed in the browser. Part assembly or part level entities cannot be modified or deleted.

Upon the creation of a part instance, you will automatically be prompted to transform the part via the Transform tool.

Import Parts, Part Assemblies, and Part Instances

PDM parts and part assemblies can be imported into HyperMesh via a PDM generated PLMXML file.

1. From the menu bar, click **File** > **Import** > **BOM**.
   The Import browser opens.
2. In the File field, navigate to the file that contains the parts, part assemblies, and part instances to import.

3. Define additional options as needed.
4. Click **Import**.

Organize Part Assemblies and Parts

Organize part assemblies, parts, and components in your model.

Entity organization rules are as follows:

- The Model is the root of the hierarchy in the browser. It represents the contents of the HyperMesh binary file and can contain part assemblies, parts, and components.
- A part assembly can contain part assemblies and parts. These can be both assemblies and parts.
- A part can only contain components.

- Reorganize part assemblies, parts, and components in the Part Browser.
    a) From the Part Browser, left-click on an entity and drag and drop it to a new location.

  In Figure 1446, Component B is being reorganized into Part B.

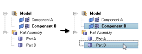
  

  Figure 1446:

- Reorganize components into a new part using the Organize panel.
    a) From the Tool page, click the **Organize** panel, **Parts** subpanel.
    b) Use the comps selector to select the component(s) to move.
    c) In the dest part field, select the part to move the selected components into.
    d) Click **move**.

Save Assemblies as HyperMesh Binary Files

The part assembly binary file is self-contained and includes the part assembly based hierarchy and attributes such as components, properties, and materials.

You can perform all necessary tasks such as geometry updates and meshing in the distributed HyperMesh binary file. The completed file can then be imported into the master HyperMesh session by importing a model.

1. In the Part Browser, right-click on a part assembly and select **Save As** from the context menu.
2. In the **Save As** dialog, save the binary in the representations directory for the current binary file.

In Figure 1447, the LeftRail_A_000433_Safety part assembly in the Frame_Assembly_000495 part assembly is being saved as an HyperMesh binary file.

> **Tip:** Alternatively, you can select multiple Parts in different assemblies and select **Save As** from context menu.

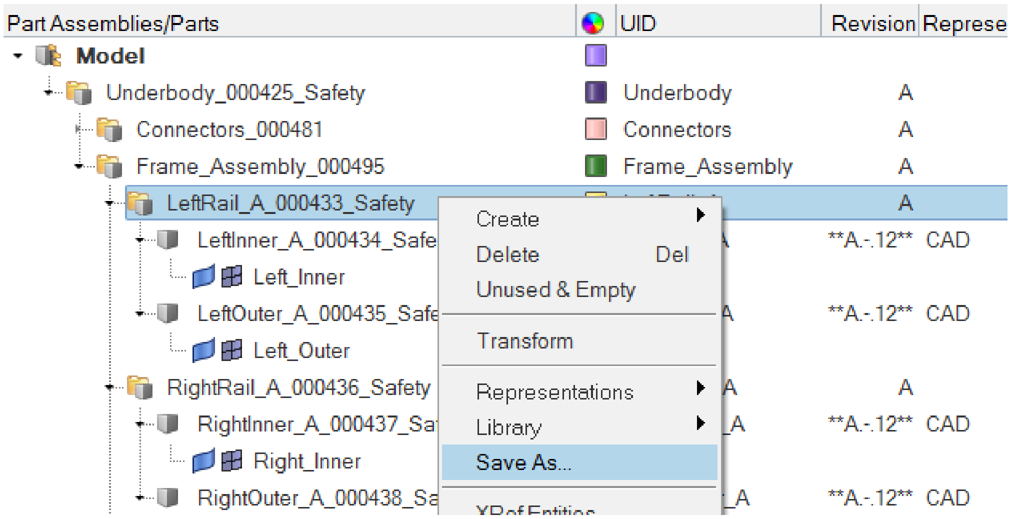


Figure 1447:

Because the saved part assembly is self-contained, all HyperMesh entities shown in the Part Browser are saved to the binary file. Opening the saved binary file in a new HyperMesh session results in the Part Browser view shown in Figure 1448.

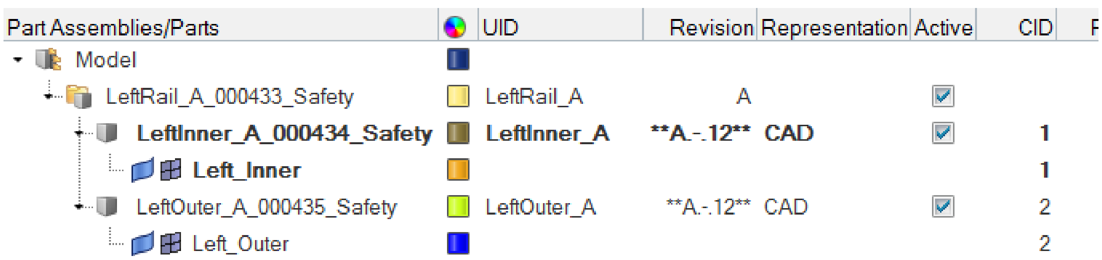


Figure 1448:

Manage Representations

About Representations

The CAD representation forms the basis of the Common representation, which in turn is the basis of all subsequent discipline specific mesh representations. A part can contain multiple representations.

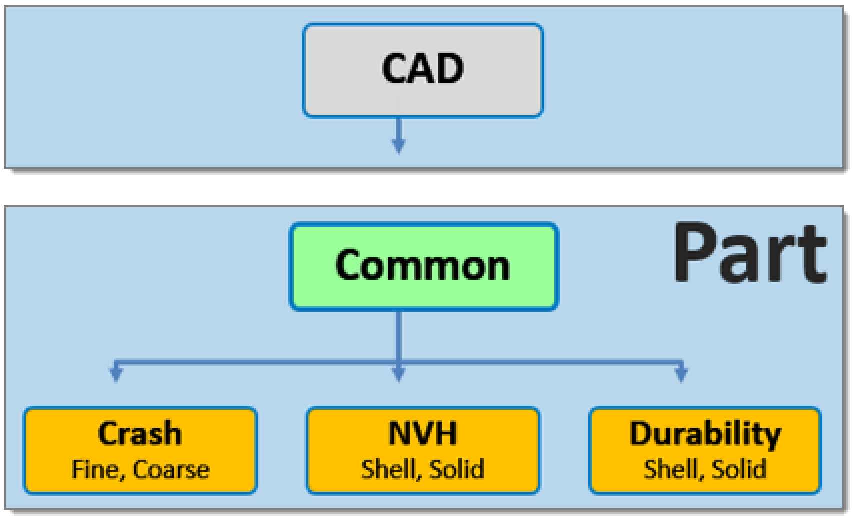


Figure 1449: Hierarchical Relationship of Representations

The Common representation component typically contains geometry, though it can also consist of FE mesh.

A folder based Representation repository stores all CAE data that is required during the model build and assembly process. CAE data stored in the Representation repository includes the geometric and FE representation of the parts that comprise the subsystem specific model hierarchy.

You can import a BOM via a neutral file format such as PLMXML or manually create a part structure using the Part Browser context menu.

Common Representation

The Common representation is derived from the CAD representation and forms the basis of all subsequent discipline specific mesh representations.

For sheet metal parts, the Common representation consists of midsurfaced geometry or FE. The CAD is sent to the Batchmesher for midsurface extraction; upon completion it is saved into the repository and you can elect to immediately import the representation into the session.

HyperMesh entities are generated from the PDM metadata, if available, in the post-run procedure of the BatchMesh operation. The PDM PID is assigned to the component and property, the PDM MID and PDM material is assigned to the material and the PDM Thickness is assigned to the Thickness attribute of the property. If the PDM Thickness is blank, the CAD Thickness calculated during the midsurface operation is automatically assigned to the Thickness attribute of the property.

For parts such as castings and tailor-welded blanks, you can save CAD representations as Common models.

Alternatively, you can send solid parts to the Batchmesher. If **thin-solid detection** is enabled in the Common representation parameter file, then solids will be detected and saved as the Common representation without processing. By default, the midsurface algorithm skin is used. CAD representation does not need to be loaded into the session in order to generate the Common representation since it is sent directly to the Batchmesher for processing.

When you select a discipline specific mesh representation from the **Change Representation** dialog, Create tab, the Common representation residing in the repository is automatically sent to the Batchmesher for processing. If the Common representation does not exist it will be automatically generated.

Discipline Specific Mesh Representations

Common representations form the basis of all discipline specific representations. When you select a discipline specific mesh representation from the **Create Representation** dialog, the Common representation residing in the repository is automatically sent to the Batchmesher for processing. The Common representation does not need to be loaded in the session to generate discipline specific mesh representations.

Figure 1450: PDM Column Data in Part Browser

Entities	PDM Material	PDM MID	PDM PID	PDM Thickness
Model				
Frame_Assembly_F01				
Center_Rail_F02				
Center_Outer_F04_descr	Steel	1000	202	3
Center_Inner_F03	Steel	1000	201	3
Left_Rail_F05				
Left_Inner_F06	Steel	1000	301	2.5
Left_Outer_F07	Steel	1000	302	2.5
Right_Rail_F08				
Right_Inner_F09	Steel	1000	401	2.5
Right_Outer_F10	Steel	1000	402	2.5

Create Representations

Representations utilize the mesh parameter and criteria files that are included with HyperMesh. You can find these files in the Batchmesher. The Param File and Criteria File fields display the representation specific mesh parameter and criteria file.

Common representations form the basis of all subsequent discipline specific mesh representations. When you select a discipline specific mesh representation from the **Change Representation** dialog, Create tab, the Common representation residing in the repository is automatically sent to the

Batchmesher for processing. If the Common representation does not exist it will be automatically generated.

1. In the Part Browser, right-click on part assemblies or parts and select **Representations** > **Create** from the context menu.
2. In the **Change Representation** dialog, Create tab, select a desired representation.
   The availability of the selected representation in the repository is displayed next to the representation type within parentheses.

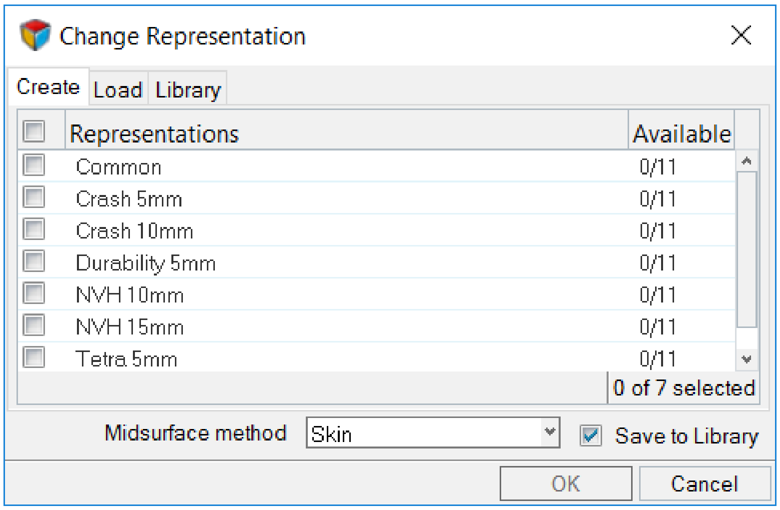


Figure 1451: Available Representations
(0/11) common representations available in the repository.

3. Optional: To save the newly created representations to the Part Library select the **Save to Library** checkbox.
4. Select the **Midsurface method** (default is Skin).
5. Click **OK**.
   All representations are sent to the Batchmesher for processing in parallel. Upon completion, all representations are automatically saved to the repository.

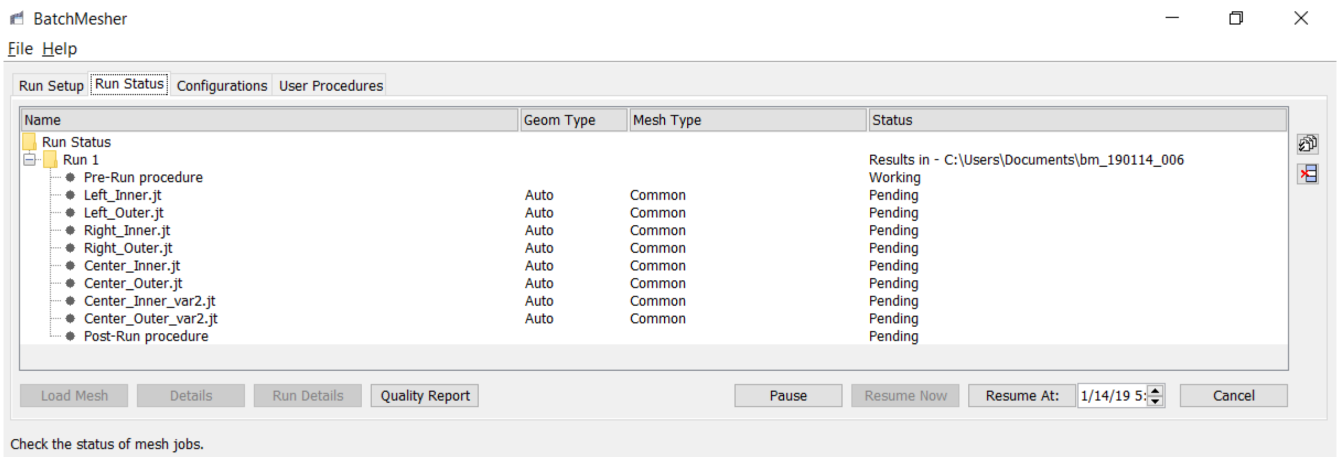


Figure 1452: BatchMesher Processing

6. In the **BatchMesh** dialog, specify how representations are imported.
   - Click **Yes** to import all representations into the session.
   - Click **No** to not import the representations into the session.

   > **Note:** The representations will be available in the **Load Representation** dialog.

Define Representations

Create your own user defined representation. User defined representations are saved to the settings and will be available in subsequent HyperMesh sessions.

1. In the Part Browser, right-click on part assemblies or parts and select **Representations** > **Settings** > **User Representations** from the context menu.
2. In the **User Representation** dialog, click **+** to add a new representation.
3. In the Representation field, enter a name for the representation.
4. In the Param File and Criteria File fields, select the appropriate mesh parameter and criteria file.
5. If the representation is for solids, select the **Solid** checkbox.
6. Click **OK**.

Add Representations

Part representations can be added from external sources such as solver decks and HyperMesh binary files or from the Part Library.

Add Representations from External Sources

Add representations from external sources for a single part or for multiple parts and part assemblies.

When selecting a part assembly or multiple parts, the **Add Representations** dialog enables you to map and associate representation files to multiple parts simultaneously.

If appropriate metadata is available in the BOM, then the alias values will be preselected. After choosing a Representation Folder, representation files will be mapped according to the HyperMesh naming convention. You can then manually select or update any of the alias and representation file values.

Indicators are displayed for each part to indicate the status of that row.

Skip.
: Any Part which does not have a mapped or selected representation.

OK.
: Alias and representation are chosen for the part.

Overwrite.
: Part already has a representation which will be overwritten by the selected mapping.

Duplicated.
: The same representation file is chosen for more than one part, or the chosen representation file is already used by another part in the same BOM. You cannot click OK to add the representations when representations are duplicated.

1. In the Part Browser, right-click on the part assemblies or part(s) and select **Representations** > **Add** > **from Files** the context menu.
2. In the **Add Representation(s)** dialog, define options accordingly.

To add representations for	Do this
Single part	1. From the Representation field, specify a desired representation type to which a representation file will be added. • Choose a default representation type. • Enter your own user defined representation into the field. 2. If you entered your own representation, select the appropriate mesh parameter and criteria files. User defined representations with defined mesh parameter and criteria files created in the **Add Representation** dialog will be available

To add representations for	Do this
	in the **Change Representation** dialog, Create tab. User defined representations with undefined mesh parameter and criteria files created in the **Add Representation** dialog will not be available in the **Change Representation** dialog, Create tab, but will appear in the Load tab if a representation is found in the repository. The availability of the selected representation in the repository is displayed next to the representation type within parentheses.

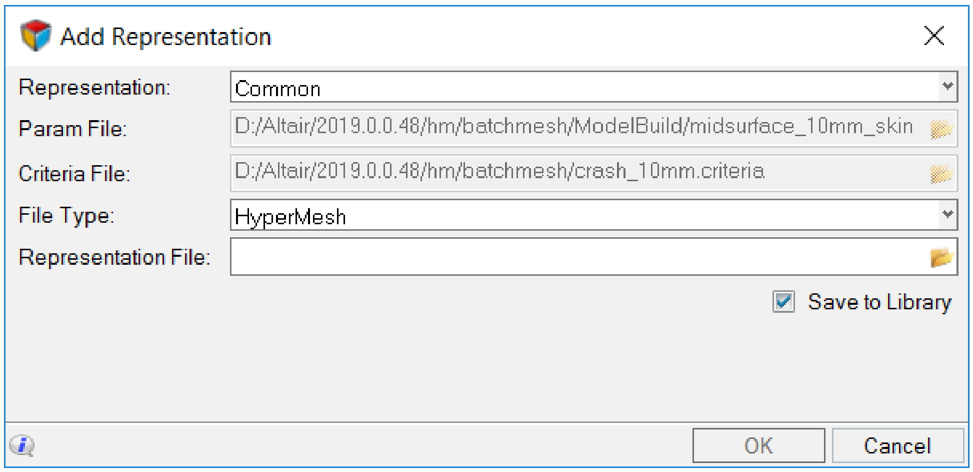


Figure 1453:

Multiple part or part assemblies

1. In the Representation Folder field, navigate to the representation file.
2. Select the parts to map and associate with the representation file.

> **Note:**
> - You cannot add CAD or connectors.
> - You can select Part or Part Assembly for adding the representation.

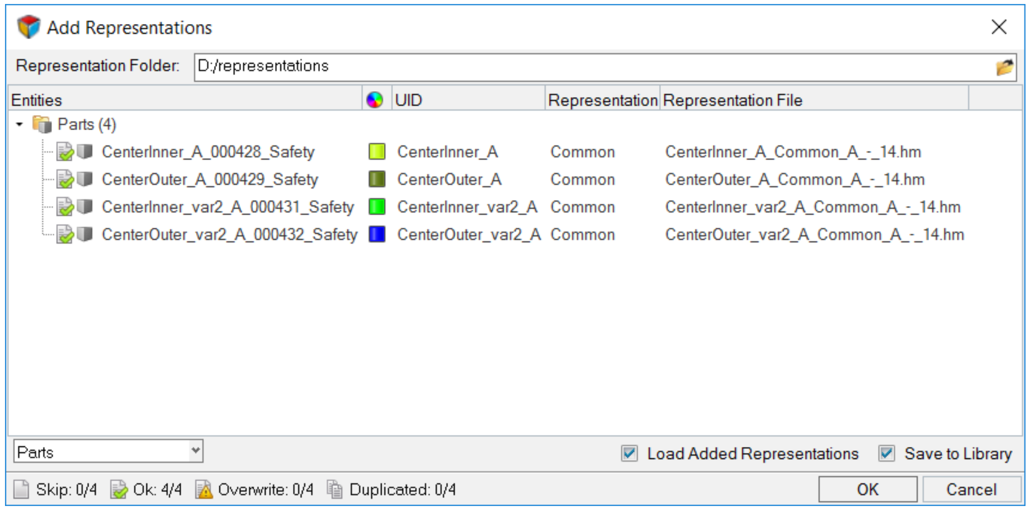
To add representations for	Do this
	

Figure 1454:

3. To save the representations to the Part Library select the **Save to Library** checkbox.
4. Click **OK**.

Add Representations from the Part Library

1. In the Part Browser, right-click on a part/part assembly and select **Representations** > **Add** > **from Library** from the context menu.
2. In the **Add Representations from Library** dialog, select representations to add.
3. Optional: To simultaneously add and load representations, select representations to load from the Load Added Representations list.
4. Click **OK**.

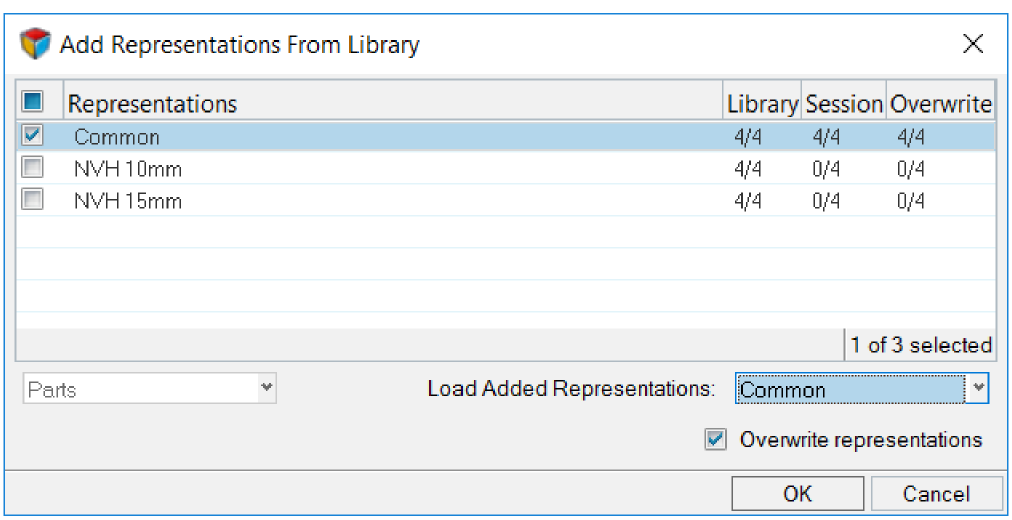


Figure 1455:

Add Representations to the Part Library

After importing a BOM, you can add CAD representations to the Part Library.

1. In the Part Browser, right-click on a part/part assembly and select **Representations** > **Add** > **to Library** from the context menu.
2. In the **Add Representation to Library** dialog, select representations and click **OK**.

The **Add Representation to Library** dialog displays information regarding the availability of representations in library, session, and overwrite details.

In Figure 1456, CAD representations are being added to the Part Library. The Library column displays 0/8, which indicates there are no CAD representations available in the Part Library. The Session column displays 6/8, which indicate 6 out of 8 representations are currently available in the active HyperMesh session. The Overwrite column displays 0/8, which indicates none of the representations will need to be overwritten.

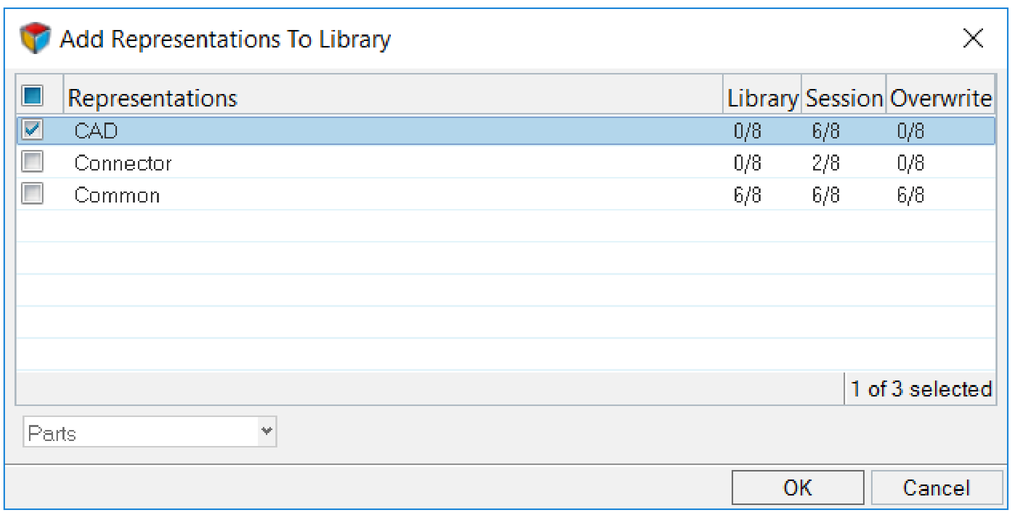


Figure 1456:

Browse Library Content

Review library content using the **Library Viewer**.

1. In the Part Browser, right-click on the part assemblies or part(s) and select one of the following:
   - **Representations** > **Add** > **browse Library** the context menu.
   - **Library** > **Library Viewer** the context menu.

   The **Library Viewer** opens.
2. Select the Part that contains the representations you would like to review.
   A list of properties associated with the Part are displayed in the Properties pane.

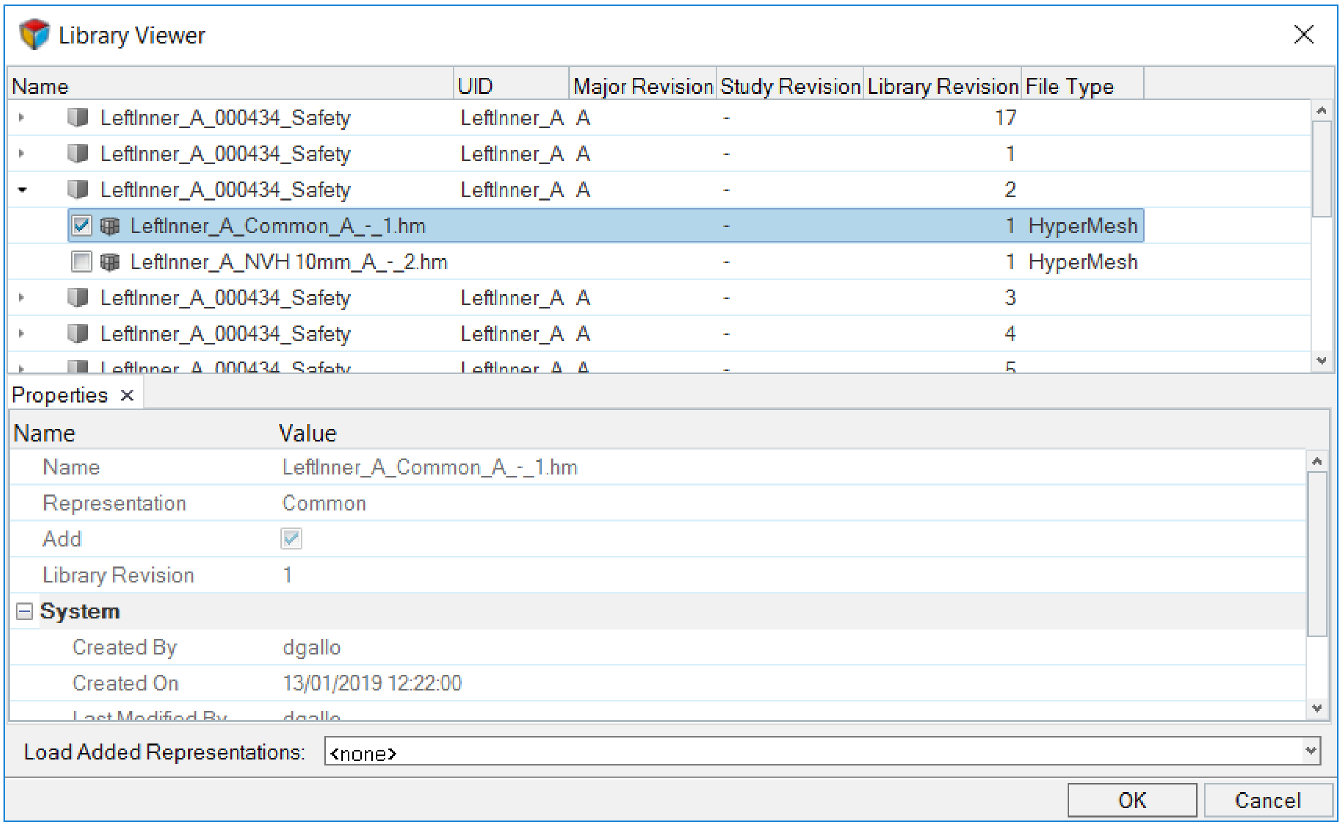


Figure 1457: Library Viewer

Load Representations

Part representations can be loaded from your current HyperMesh session or from the Part Library.

Load Representations from Current HyperMesh Session

1. In the Part Browser, right-click on part assemblies or parts and select **Representations** > **Load** > **from Session** from the context menu.
2. In the **Change Representation** dialog, Load tab, select a type of representation to load. Representations that exist in the repository are shown in the Representations column, and their availability is indicated in the Available column.

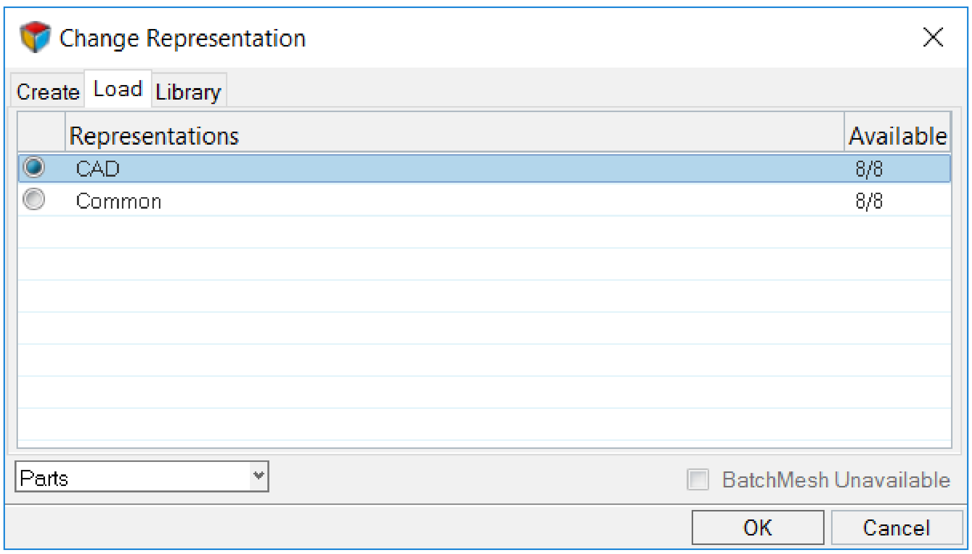


Figure 1458:

3. To send unavailable representations to the Batchmesher directly from the **Change Representation** dialog, select the **Batch Mesh Unavailable** checkbox.

> **Note:** Available representations will also be loaded when you click **OK** if the **Batch Mesh Unavailable** checkbox is selected.

4. To save the representations to the Part Library select the **Save to Library** checkbox.
5. Click **OK**.
6. In the **Confirm Load Representation** dialog, specify how representations are loaded.
   - Click **Load All** to load all available representations into the current session for selected parts.
   - Click **Skip Loaded** to ignore representations that are already loaded for selected parts.

As representations are created and saved to the repository they are displayed in the Available Representations pane.

Load Representations from the Part Library

After importing a BOM that has representations saved in the Part library, you can load the saved representations.

1. In the Part Browser, right-click on a part/part assembly and select **Representations** > **Load** > **from Library** from the context menu.
2. In the **Change Representation** dialog, Library tab, select representations and click **OK**.

The selected representations are loaded to the Representation folder on the file system. Similarly, every time a BOM is imported, files are loaded to the Representations folder, if you retrieve them from the Part Library.

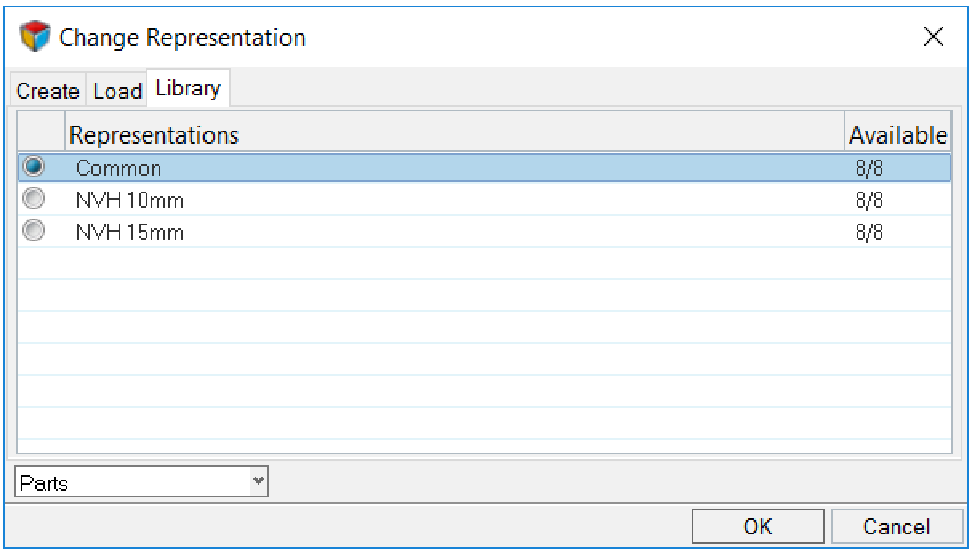


Figure 1459:

Reload Representations

Restore part representations to their original state.

In the Part Browser, right-click on parts or part assemblies and select **Representation** > **Reload** from the context menu.

Representations are reloaded from the repository.

Unload Representations

Unload part representations from a session.

1. In the Part Browser, right-click on the model, part assemblies, or parts and select **Representation** > **Unload** from the context menu.
2. In the **Confirm Unload Representation** dialog, click **Yes** to unload the selected representations.

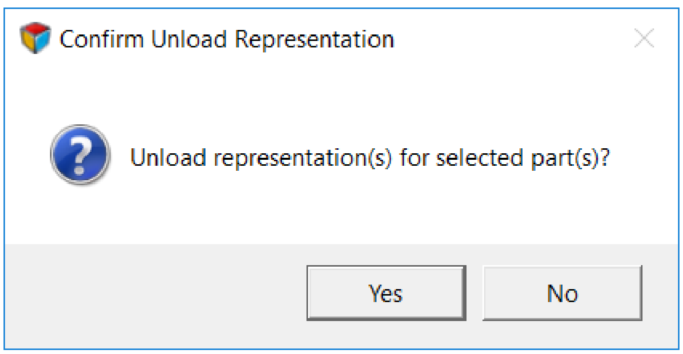


Figure 1460:

Update Metadata From PDM

1. In the Part Browser, right-click on a part or parts to update and select **Representations** > **Update** from the context menu.
2. Update metadata.
    - Rename the selected parts or part's component, or create if it does not already exist.
    - Create a material based on PDM Material and PDM MID.

      If a material of the given ID already exists, then it will not be overwritten or recreated in order to avoid overwriting any existing material properties.

      Only linear attributes are updated. Default steel attributes are used with the following unit system: millimeter, second, tonne, and Newton.
    - Create a property based on PDM PID and PDM Thickness*.

      If a property of the given PID already exists, its thickness will be updated based on PDM Thickness; however, the existing property will not be recreated. Only relevant metadata will be updated in order to avoid overwriting any existing property card values.

> **Note:** *If the PDM MeshFlag attribute is set to SMT (Solid Mesh Tetra) or SMH (Solid Mesh Hexa), than a solid card image will be assigned to the property.

Sync Metadata To PDM

Sync PDM metadata (PDM PID, PDM Thickness, PDM Material, and PDM MID) based on a selected part or part's metadata (PID, Thickness, Material, MID).

In the Part Browser, right-click on a part or parts and select **Representations** > **Sync Metadata** from the context menu.

*When a part has multiple properties, only the first property will display in the PID PDM field when you select **Sync Metadata**.

Save Representations

After importing a BOM and creating respective representations, you can save the representations locally or in the Part Library.

Parts without a UID cannot be saved to the Library. Considering this, the UID field in the **Save Representation** dialog is user editable.

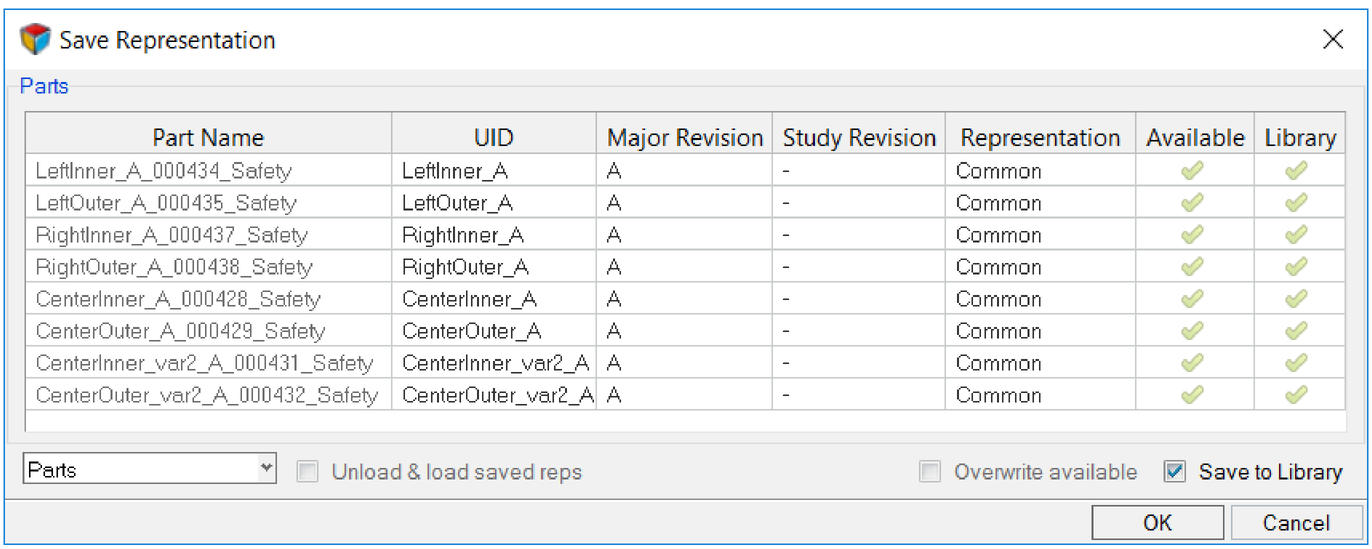
1. In the Part Browser, right-click on part assemblies or parts and select **Representations** > **Save** from the context menu.

   A checkmark in the Available column indicates that the representation already exists in the repository. A checkmark in the Library column indicates that a revision is already available in the Part Library.

   The **Save Representation** dialog opens.
2. Edit the Major Revision and Study Revision columns to reflect the intended revisions.
3. To overwrite representations that already exist in the repository or Part Library, select the **Overwrite available** checkbox.

   > **Note:** Original CAD representations will not be overwritten because the original file location is stored as a link on the part. Enabling **Overwrite available** saves an HyperMesh binary file of the modified CAD representation in the repository.
4. To only save parts locally, disable the **Save to Library** checkbox.
5. Click **OK**.

Unavailable parts are saved, and available parts are overwritten and saved if Overwrite available was selected.



Part Name	UID	Major Revision	Study Revision	Representation	Available	Library
LeftInner_A_000434_Safety	LeftInner_A	A	-	Common	✓	✓
LeftOuter_A_000435_Safety	LeftOuter_A	A	-	Common	✓	✓
RightInner_A_000437_Safety	RightInner_A	A	-	Common	✓	✓
RightOuter_A_000438_Safety	RightOuter_A	A	-	Common	✓	✓
CenterInner_A_000428_Safety	CenterInner_A	A	-	Common	✓	✓
CenterOuter_A_000429_Safety	CenterOuter_A	A	-	Common	✓	✓
CenterInner_var2_A_000431_Safety	CenterInner_var2_A	A	-	Common	✓	✓
CenterOuter_var2_A_000432_Safety	CenterOuter_var2_A	A	-	Common	✓	✓

Figure 1461:

Delete Representations

Part representations can be deleted from the current HyperMesh session and from the Part Library.

Delete Representations from the Current HyperMesh Session

1. In the Part Browser, right-click on part assemblies or parts and select **Representations** > **Delete** > **from Session** from the context menu.
2. In the **Delete Representations** dialog, select representations to be deleted.
3. To remove representations from the Part Library, select the **Delete representation from library** checkbox.

   > **Note:** The Study version will be incremented if a representation is deleted from the Library.

4. Click **OK**.

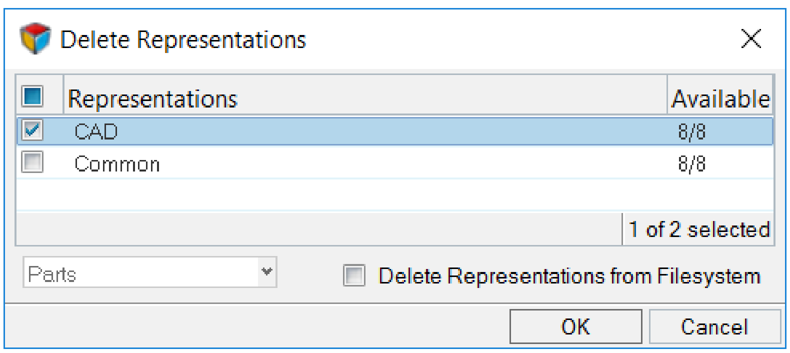


Figure 1462:

Delete Representations from the Part Library

1. In the Part Browser, right-click on part assemblies or parts and select **Representations** > **Delete** > **from Library** from the context menu.
2. In the **Delete Representations From Library** dialog, select representations to be deleted.
3. Click **OK**.

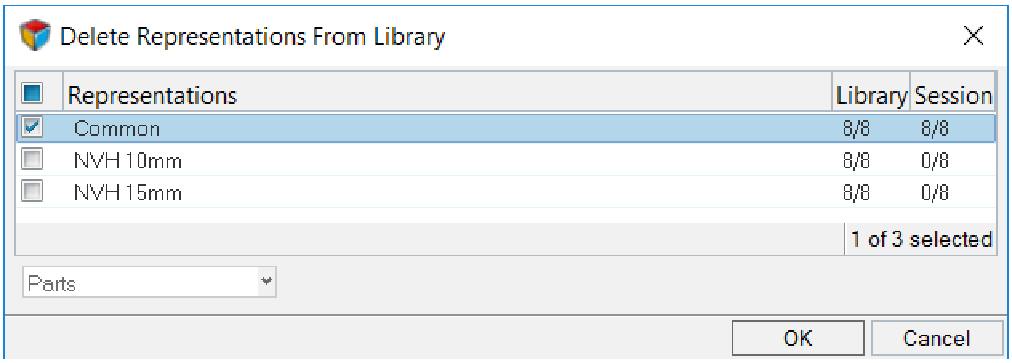


Figure 1463:

Representation Load Settings

Settings to configure the representation entity management behavior.

Open the **Representation Load Settings** dialog by right-clicking in the Part Browser and selecting **Representations** > **Settings** > **Load Settings**.

The representation entity management settings work independent of the import binary and import deck entity management settings.

The following options are available for components, properties, and materials.

Offset ID.
: Merges incoming attributes with conflicting IDs into the session, and offsets their IDs.

Keep Existing Attributes.
: Maintains existing in-session entity attributes and incoming conflicting entity IDs.

Keep Incoming Attributes (default).
: Maps incoming entity attributes to the in-session entities.

When **Offset ID** is selected for components, both incoming and existing geometry and FE residing in the component with conflicting IDs are kept. When **Keeping Existing Attributes** or **Keep Incoming Attributes** are selected for components, incoming geometry and FE that resides in the component with conflicting IDs are kept.

Manage Part Revisions

Manage, control and update Major, Study and Library Part revisions using the Part Library.

About Part Library and Revisions

The Part Library is an integral part of data management and revision control within the Part Browser.

Part Library

By default, a Part Library is generated when the Part Browser is enabled. There are two modes:

Unmanaged
Local file management

Managed
Uses the Part Library

The default Part Library is located at `<user home>\AltairLibraries\2019`.

The Part Library serves as a centralized library of HyperMesh parts, which in turn facilitates the collaboration between simulation teams both locally and abroad.

You must be connected to a Part Library to access all Part Library related options.

Parts without a UID cannot be saved to the Library. Considering this, the UID field in the **Save Representation** dialog is user editable.

Revisions

In the Part Browser, the Revision column displays the Major revision, Study revision, and Library Part revision. All of the appropriate workflows have entry or access points to the part library, enabling quick and easy access to previous revisions or simply generating new revisions.

Major Revision
Depicts a change or set of changes that have been finalized and released for further review. In most cases the Major Revision is mapped to the PDM Revision, but you can modify it.

Study Revision
Created locally to track experimental changes or prototyping changes that may or may not be published. The purpose of the Study Revision is to manage design and prototype exploratory concepts. You can modify the Study Revision.

Library Part Revision
Locally, published revision within the Part Library. You cannot modify the Library Part Revision.

You can review and edit a part's Major Revision, Study Revision, and Library Part Revision in Entity Editor.

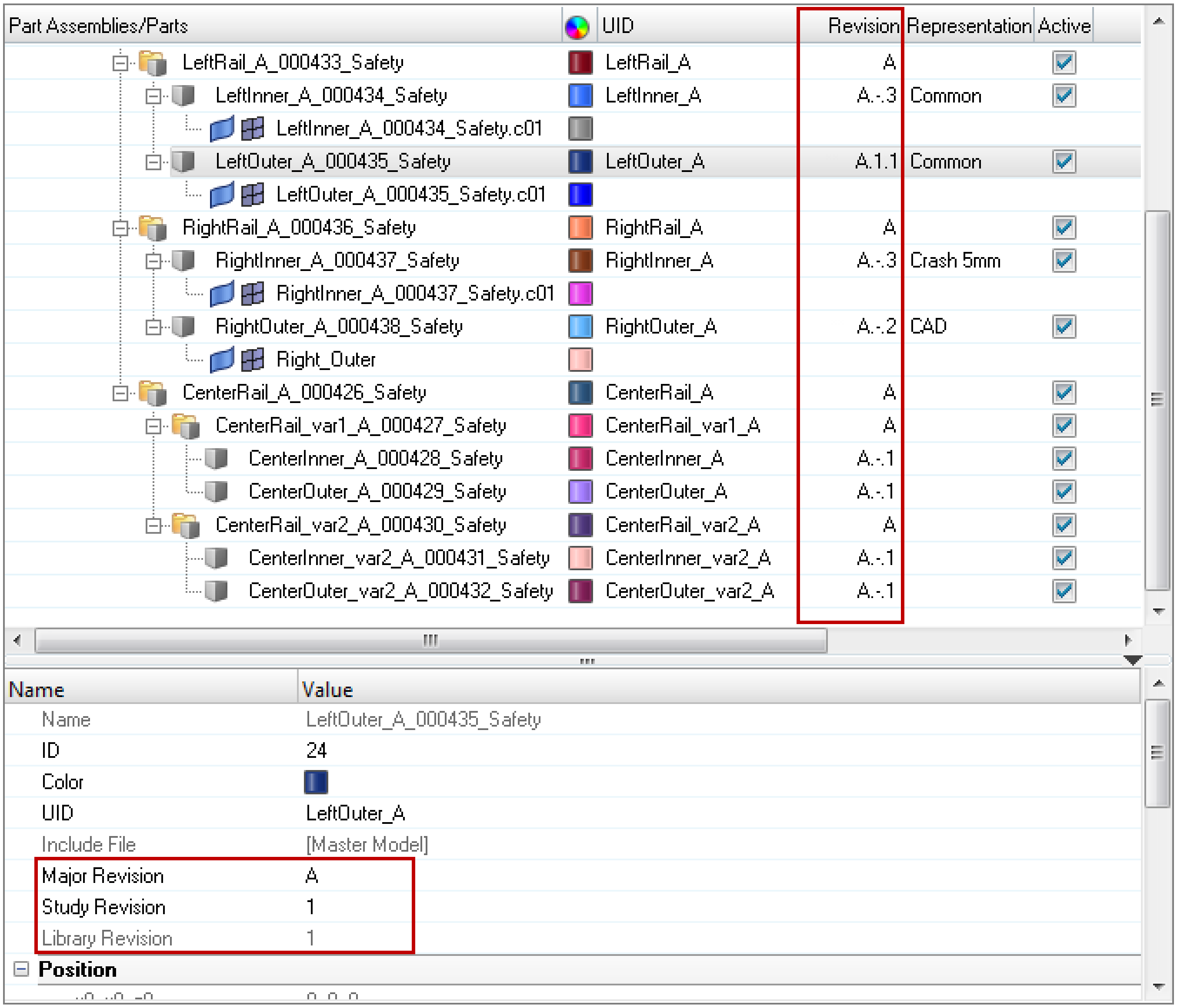


Figure 1464: Revisions for the LeftOuter_A_000435_Safety.c01 Part

Major Revision = A (mapped to the PDM revision, user editable), Study Revision = 1 (can be any letter or numerical value, user editable), Library Part Revision = 1 (library version, not editable)

Register and Connect Libraries

Register and connect to new libraries of your choice.

A default Part Library is generated when the Part Browser is enabled, and is located at `<user home>\AltairLibraries\2019`.

1. In the Part Browser, right-click and select **Library** > **Libraries** from the context menu.
   The **Libraries** dialog opens.
2. Register new library.

   a) Click **+**.
   b) Select Library type.
      - Local
      - Shared (see PostGres installation for a shared Library)
   c) Enter Library host/port.

      If you are working with a shared Library, enter a Library port.
   d) Enter a Library Name and Library Path.
   e) Click **Add**.

3. Disconnect currently connected library.

   Before you can connect to a new library, you must disconnect from the currently connected library.
   a) Select the currently connected library.
   b) Click **Disconnect**.

4. Connect library.
   a) Select a disconnected library.
   b) Click **Connect**.

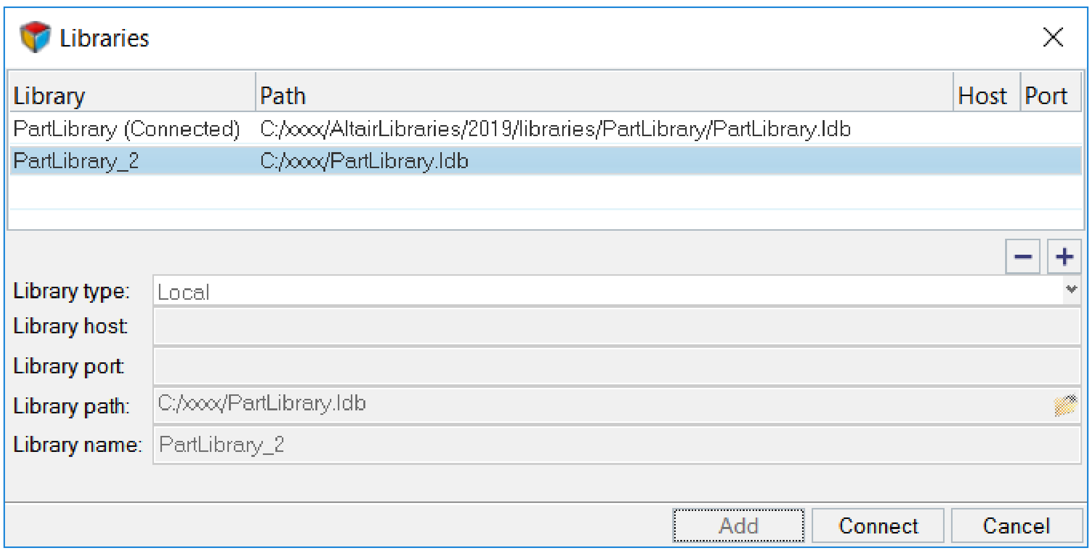


Figure 1465:

Sync Library Revisions

Sync the in-session version with the latest, available version in the Part Library.

If the version available in the current session is out of sync with the Library reversion, use Sync to update the session version with the Part Library version.

Before you can sync library revisions, you must Register and Connect Libraries.

In the Part Browser, right-click on a part/part assembly and select **Library** > **Sync** from the context menu.

Edit Study Revisions

Study Revisions are a way to explore design space, and can be promoted to a Major Revision as required.

Assign and edit Study Revisions in the Entity Editor.

1. In the Part Browser, select a part/part assembly.
2. In the Entity Editor, edit the Study Revisions field.

Manage Configurations

Group common and unique parts/part assemblies together in part sets, and organize parts and part sets that are unique to a configuration.

In the traditional model build and assembly workflow, subsystems that contain multiple variants are stored in multiple HyperMesh binary files. This complicates the model build and update process as part updates and revisions will need to be performed on each binary file.

An example of a subsystem that may contain multiple configurations is a vehicle body-in-white (BIW). A typical sedan BIW may have the following configurations:

- Left-hand drive (LHD)
- Right-hand drive (RHD)
- Fixed roof (Fixed)
- Panoramic roof (Pano)

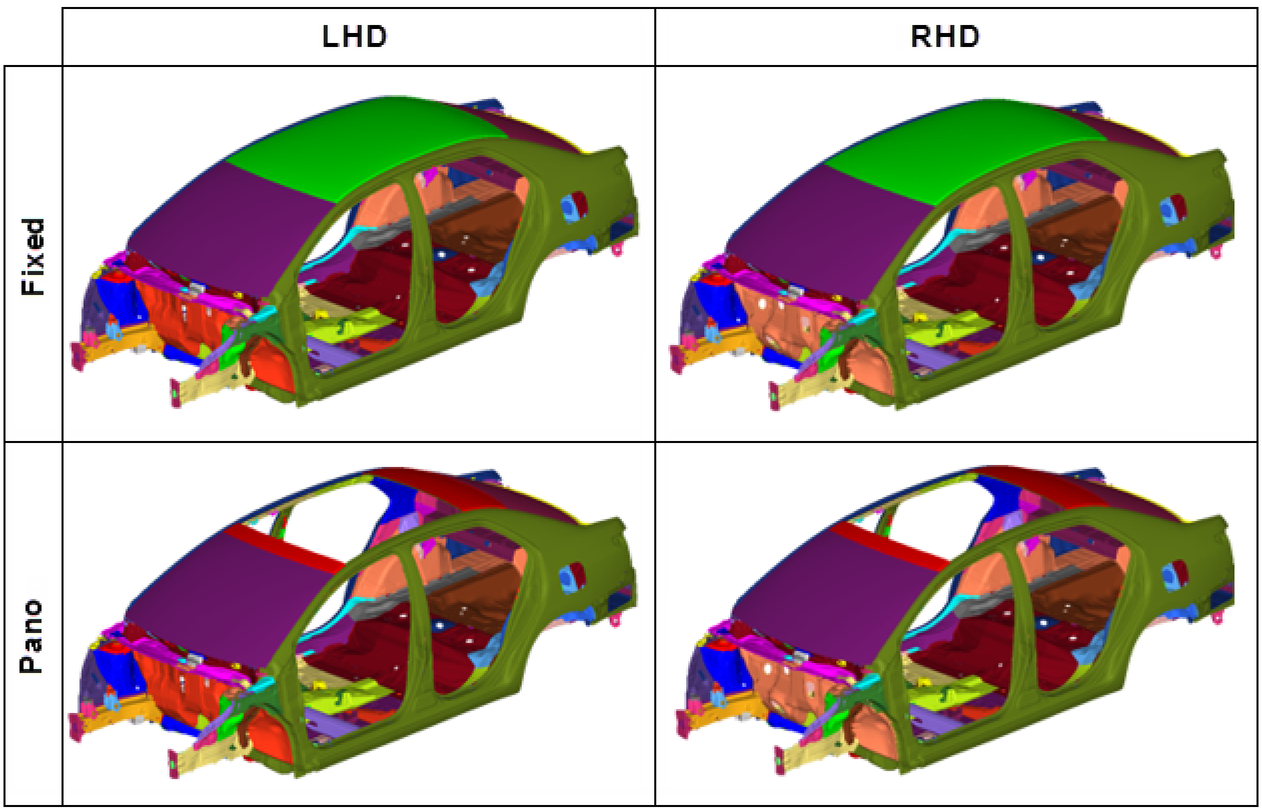


Figure 1466:

All common and unique parts that appear in all configurations for a given subsystem are stored in a single HyperMesh binary file, known as a Layered Model. In a Layered Model, common parts are active in all configurations. Unique parts are active only in a specific configuration. You must deactivate unique parts not appearing in a configuration.

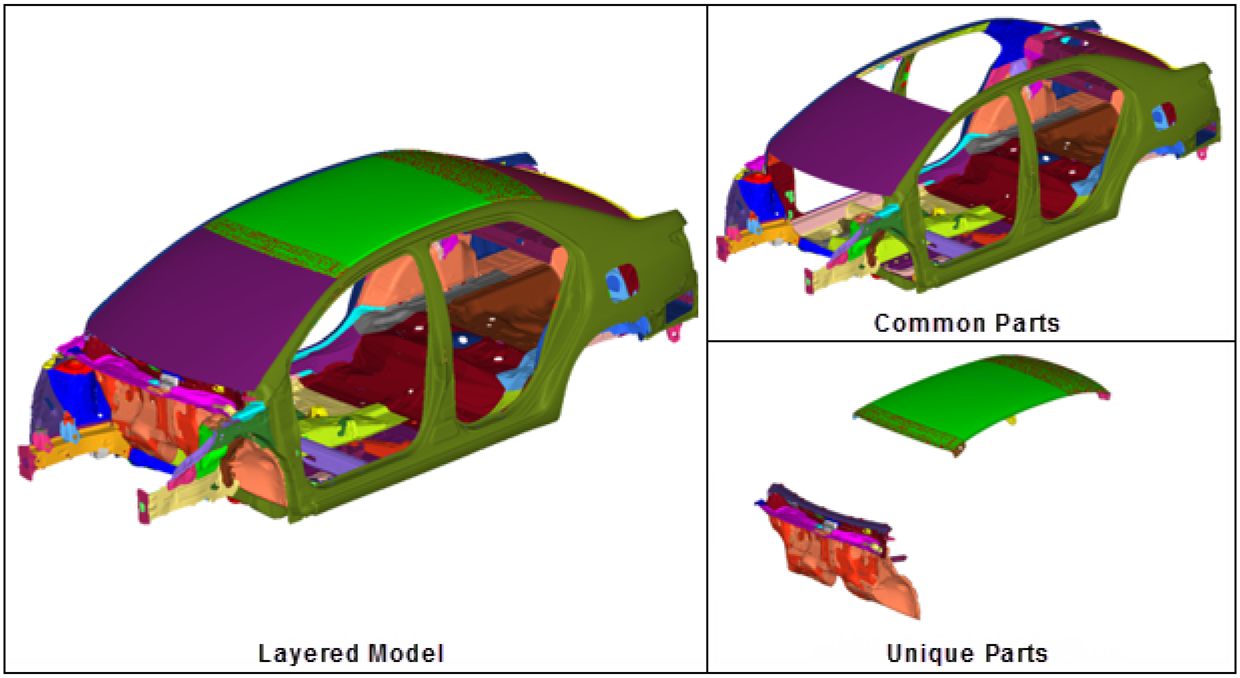


Figure 1467:

About Part Sets and Configurations

Part sets group common or unique parts, and configurations organize parts and part sets that are mutually exclusive to a configuration.

Example: Configuration Management Workflow

Configuration management workflow for a Dash and Cowl subsystem.

The configuration management workflow for a Dash and Cowl subsystem is shown in Figure 1468, along with the Left-hand drive (LHD) and Right-hand drive (RHD) configurations contained in the Layered HyperMesh binary file.

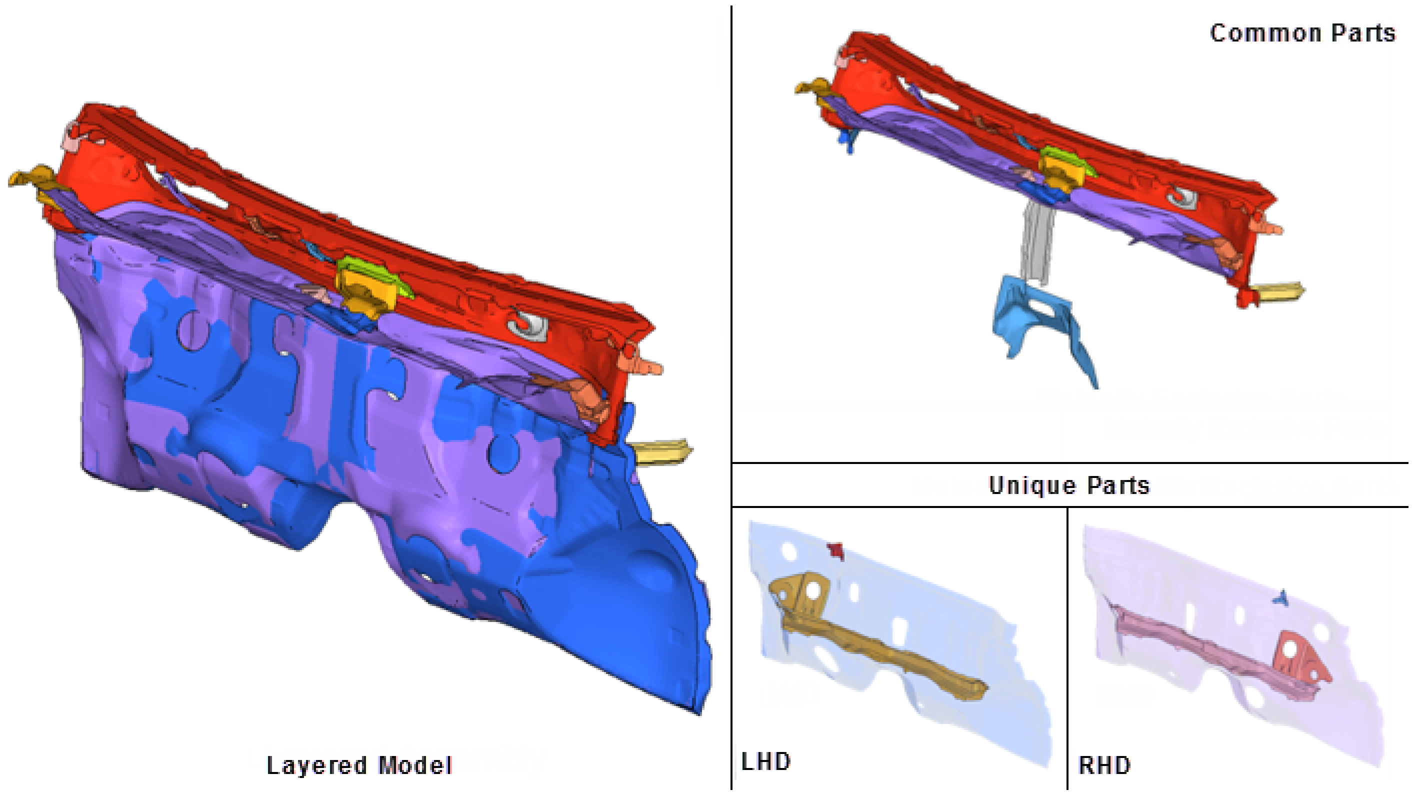


Figure 1468:

Common parts are active in all configurations and unique parts are active only in a specific configuration. The model hierarchy is shown in Figure 1469.

Part Assemblies/Parts		UID	Representation	Active
Model				
Dash and Cowl Assembly				☑
Dash RHD Assembly				☑
Dash Panel RHD			Common	☑
Dash Inner Bracket RHD			Common	☑
Dash Cross Member RHD			Common	☑
Dash Bracket Large RHD			Common	☑
Dash LHD Assembly				☑
Dash Panel LHD			Common	☑
Dash Inner Bracket LHD			Common	☑
Dash Cross Member LHD			Common	☑
Dash Bracket Large LHD			Common	☑
Dash and Cowl Common				☑

Figure 1469:

To activate the LHD configuration of the Dash and Cowl assembly, perform one of the following:

- Clear the Active column for the following individual, unique parts that belong to the RHD configuration.
  - Dash Panel RHD
  - Dash Bracket Large RHD
  - Dash Inner Bracket RHD
  - Dash Cross Member RHD
- If all unique parts are nested in a single part assembly, clear the Active column for Dash RHD Assembly.
- If all parts and part sets that are unique are organized in a configuration, enable the LHD configuration's associated checkbox in the Active column of the Configuration view.

All RHD unique parts are removed from the modeling window and are automatically set to do not export. Export the solver deck for the LHD configuration using the Export Solver Deck Browser.

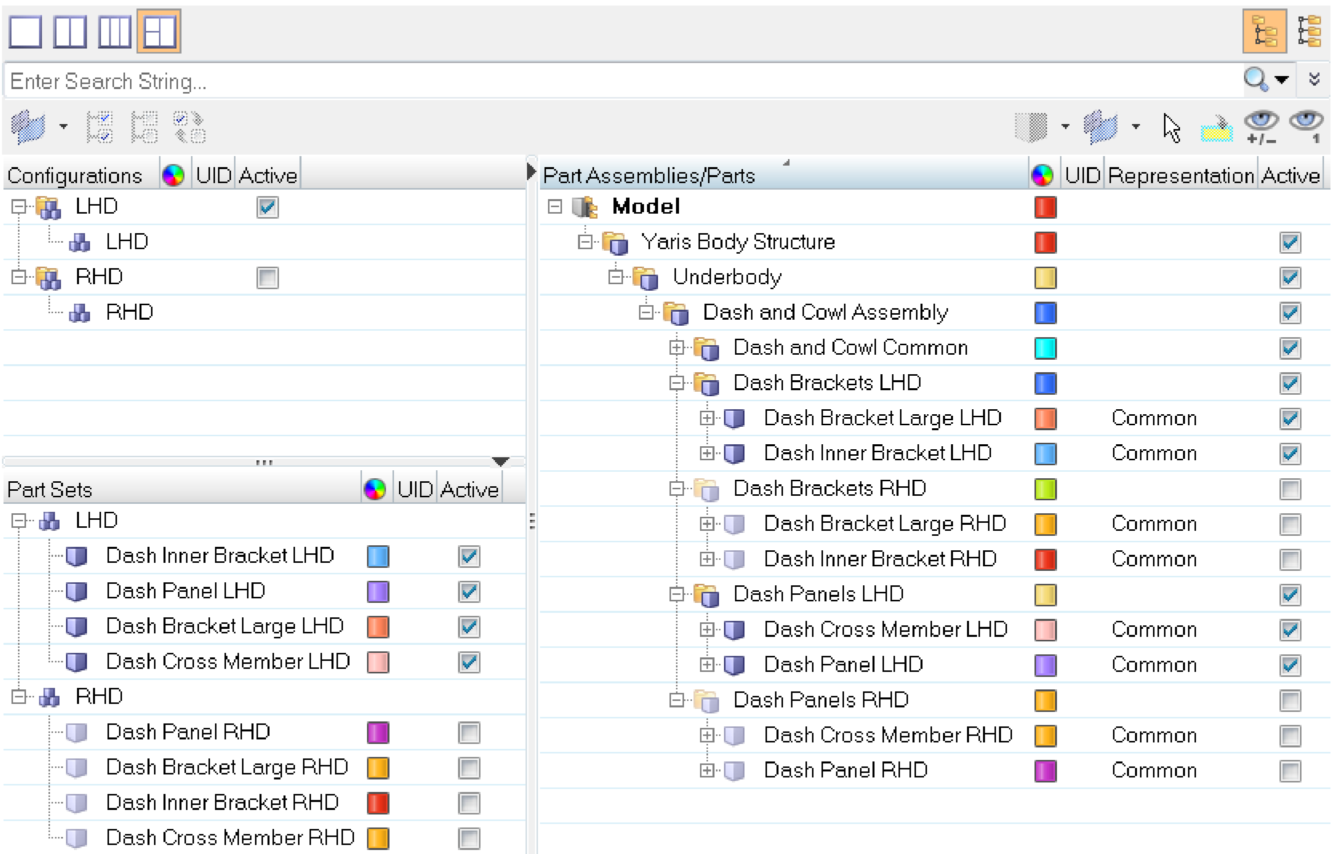


Figure 1470:

Create Part Sets

1. Open the Part Browser.
2. In the Part Set view, right-click and select **Create** > **Part Set** from the context menu.
   A new part set is created.

3. From the Part view, click-and-drag parts and part assemblies that are common and/or unique onto the part set.

> **Tip:** Dragging-and-dropping selected parts/part assemblies into the white space of the Part Set view automatically creates a new part set for the selected.

Create Configurations

1. Open the Part Browser.
2. In the Configuration view, right-click and select **Create** > **Configuration** from the context menu.
3. From the Part and/or Part Set views, click-and-drag parts, part assemblies, and part sets that are unique to a configuration onto the configuration.

> **Tip:** Dragging-and-dropping selected parts, part assemblies, and part sets into the white space of the Configuration view automatically creates a new Configuration for the selected.

Remove Contents of Part Sets and Configurations

In the Part Browser, remove the contents of part sets and configurations in the following:

- Right-click on the entity and select **Remove** from the context menu.
- Drag the entity into the white space of the respective browser view.

Activate/Deactivate Configurations

Control the display and export state of parts and part assemblies by changing the active/inactive state of configurations.

Entities set to inactive are still visible in the Part Browser. Deactivated components will not be visible in the Model Browser, Display panel, and panel entity collectors.

- Set configuration to active.
  a) Go to the Part Browser, Configuration view.
  b) In the Active column, select the configuration's checkbox.

  > **Note:** Only one configuration can be activated.

  > **Tip:** For simple models you can activate a configuration in the Part view by setting all unique parts not appearing in the configuration to inactive.

  - All of the parts, part assemblies, components, and part sets organized in the active configuration are isolated in the modeling window.

    - All of the parts, part assemblies, components, and part sets not associated with the active configuration become inactive and their display is turned off in the modeling window.
    - Inactive components are set to do not export.
- Set part assembly to inactive.
    - Removes all nested parts and owned components from the modeling window.
    - Sets all owned components to do not export.
- Set part to inactive.
    - Removes the part and its owned components from the modeling window.
    - Sets all owned components to do not export.

Create and Organize Part Sets from PDM Variants

1. In the Part Browser, Part Set view, right-click on a part set entity (if available) or in the white space and select **Create Variants** from the context menu.
2. In the **Confirm Create Variants** dialog, click **Yes**.

The created part sets are nested under a Variants Part Set.

> **Note:** Part sets are recreated on each invoke of the Create Variants operation.

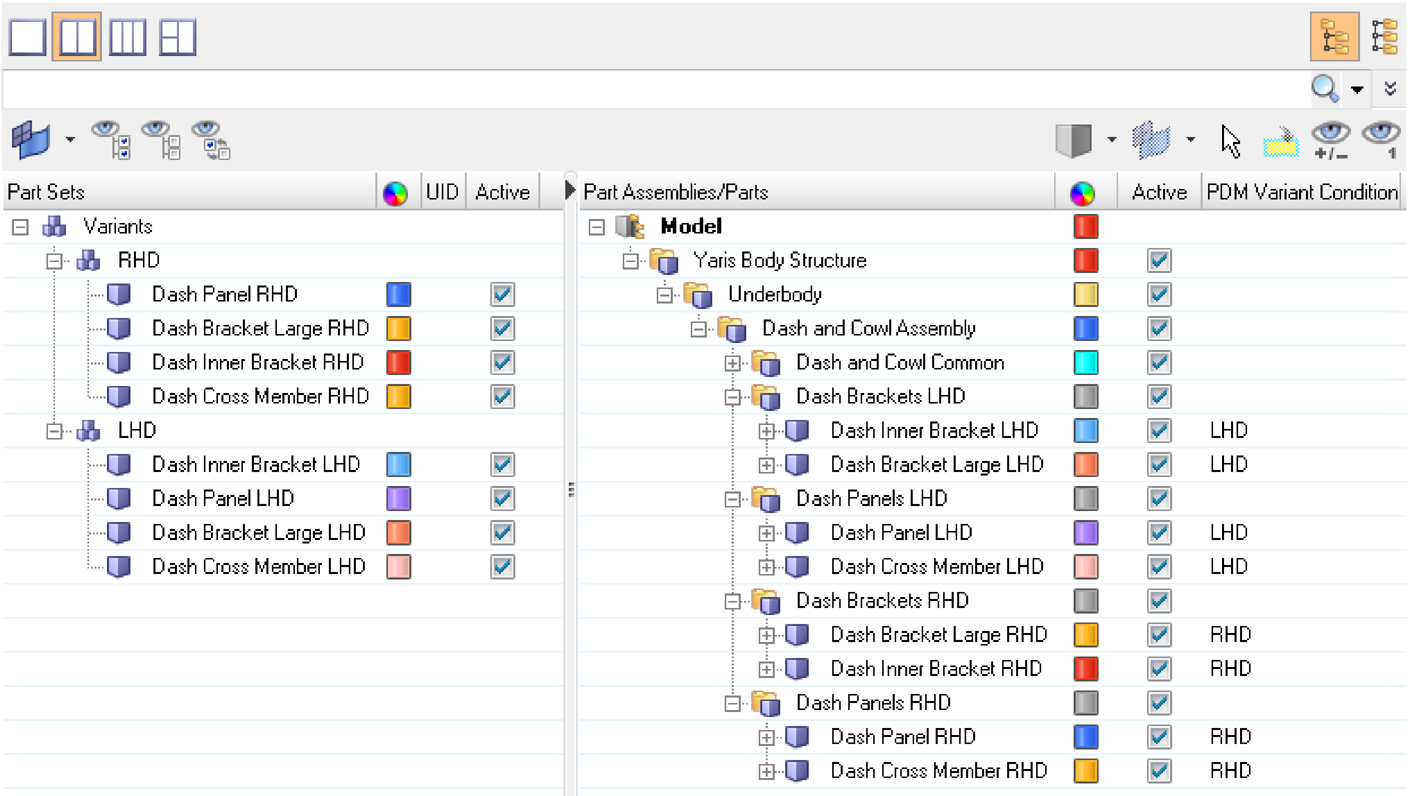


Figure 1471:

PDM Variant Conditions

PDM Variant Conditions are utilized in PDM systems, such as Teamcenter, to mark similar parts in BOMs that may contain multiple variants, such as an automotive BIW.

Attributes supported in the Part Browser via BOM Import and user editing via the Entity Editor include:

PDM Variant Condition

If non-empty, the part is used as a variant in one or more part configurations.

PDM Variant Scope

Along with the Variant Condition attribute, it describes which part configurations the part belongs to as a variant.

Part sets are created per PDM Variant Condition attribute found in the global part assembly/part hierarchy. The operation can be invoked at model, part assembly, or part level.

You can organize part sets from PDM Variants in any view of the Part Browser.

Teamcenter - HyperMesh Integration

All BOM-related information, including part/assembly hierarchy, part attributes, and representation information is organized and displayed in the Part Browser.

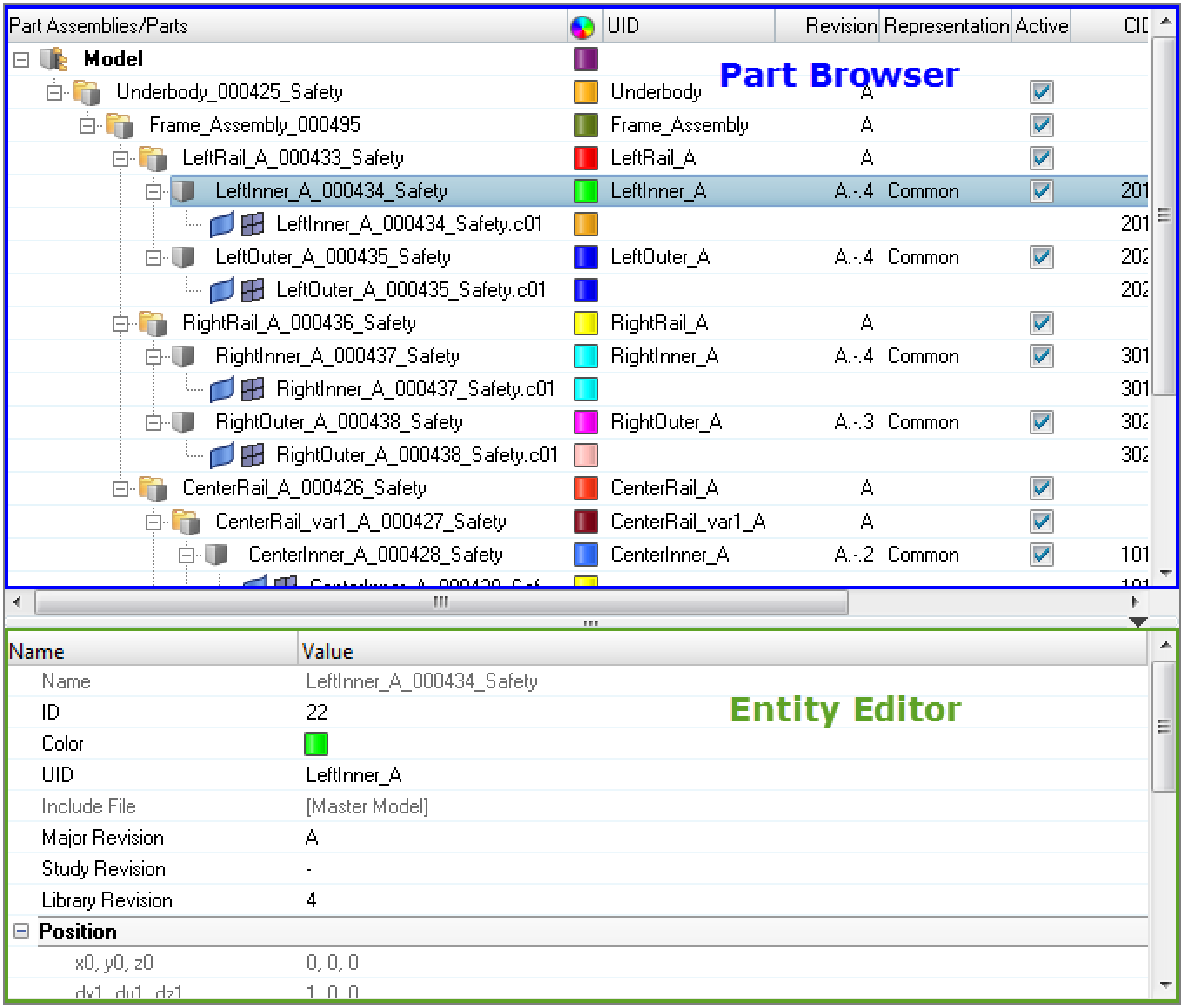


Figure 1472:

Representation (HyperMesh session) attributes are now displayed independent of the PDM attributes. Attribute columns can be turned on/off in the **Column Visibility** dialog, which can be accessed by right-clicking in the Part Browser and selecting **Column Visibility** from the context menu.

InFigure 1472, the session attributes (PID, MID, Material, Thickness) are blank because representations have yet to be created or loaded. The corresponding PDM attributes, from the imported PLMXML BOM file, show their values from Teamcenter.

Representation Options

<table>
<tr><th>Option</th><th>Description</th></tr>
<tr><td>Create</td><td>Creates HyperMesh representation(s) via Batchmesher.</td></tr>
<tr><td>Save</td><td>Saves representations for selected part(s) based on the chosen alias and file type.</td></tr>
<tr><td>Add</td><td>

Adds representations for selected part(s) based on the chosen alias and file type or from the Part Library.

When adding representations for multiple parts or part assemblies from external sources, the **Add Representations** dialog enables you to map and associate representation files to multiple parts simultaneously.

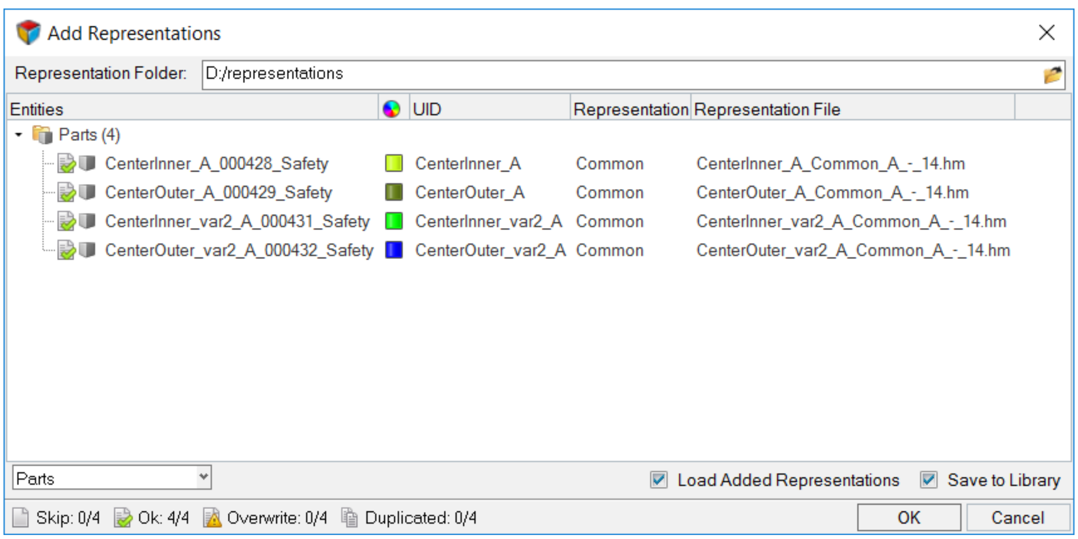


Figure 1473:

If appropriate metadata is available in the BOM, then the alias values will be preselected. After choosing a Representation Folder, representation files will be mapped according to the Teamcenter naming convention. You can then manually select or update any of the alias and representation file values.

Indicators are displayed for each part to indicate the status of that row.

Skip.
Any Part which does not have a mapped or selected representation.

</td></tr>
</table>

Option	Description
	OK. Alias and representation are chosen for the part. **Overwrite.** Part already has a representation which will be overwritten by the selected mapping. **Duplicated.** The same representation file is chosen for more than one part, or the chosen representation file is already used by another part in the same BOM. You cannot click OK to add the representations when representations are duplicated.
Load	Load part representations from your current Teamcenter-HyperMesh session or from the Part Library. If any of the selected parts currently have a representation loaded in the session, you will be prompted whether or not to unload them before loading new representations. If representations are not unloaded first, incoming representations will be imported on top of the existing representations, and Load Settings will be used.
Unload	Unloads representations for selected part(s).
Delete	Deletes representations for selected part(s) from your current Teamcenter-HyperMesh session or from the Part Library.
Reload	Restores part representations to their original state.
Settings	Displays the current load settings, which are used when representations are imported.

Setup the Teamcenter - HyperMesh Environment

The Teamcenter integration environment requires an additional layer, on top of the out-of-the-box Part Browser behavior.

When launching HyperMesh via Teamcenter, the environment will be set automatically. However, all of the integration features may be accessed by manually setting the environment.

To manually set the Teamcenter-HyperMesh environment:

1. Locate the Teamcenter-HyperMesh custom layer in your installation.

By default, the custom layer is located in `<ALTAIR_HOME>\hm\scripts\br\views\modules\custom\TC`, where `<ALTAIR_HOME>` refers to the base installation folder where HyperWorks is installed, for example `C:\Program Files\Altair\HW2019`.

2. Set the environment variable HW_CONFIG_PATH=`<ALTAIR_HOME>\hm\scripts\br\views\modules\custom\TC\properties`.
3. Set the environment variable HM_CAT_CUSTOM=`<ALTAIR_HOME>\hm\scripts\br\views\modules\custom\initTC.tcl`.

> **Note:** For customer-specific customization that are above and beyond the standard Teamcenter-HyperMesh integration, set HM_CAT_CUSTOM to the `init<XYZ>.tcl` file, where XYZ refers to customer “XYZ’s” individual custom layer.

Update Teamcenter

Update Teamcenter via Teamcenter-HyperMesh Integration

If HyperMesh was launched via the Teamcenter - HyperMesh Integration, you will be prompted to import updated data back to Teamcenter.

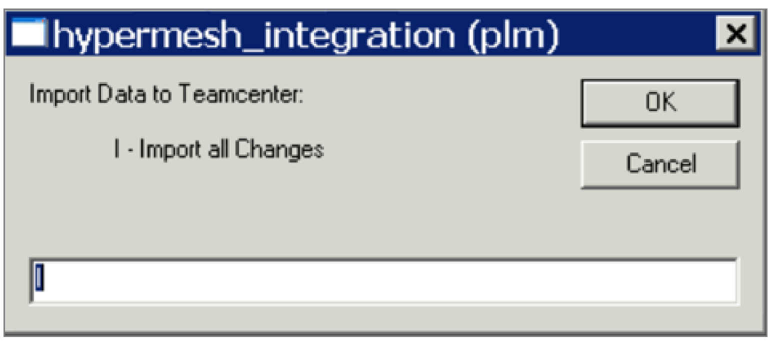


Figure 1474:

Update Teamcenter via Manual PLMXML Import in Teamcenter

If the BOM was loaded manually, for example HyperMesh was not launched via the Teamcenter-HyperMesh Integration, then the updated PLMXML package can be manually imported into Teamcenter.

1. Copy the exported BOM PLMXML, along with the folder(s) containing the monolithic file, if saved, and any saved/updated representations to a convenient folder accessible by Teamcenter.
2. Select **Import PLMXML** to import the updates.

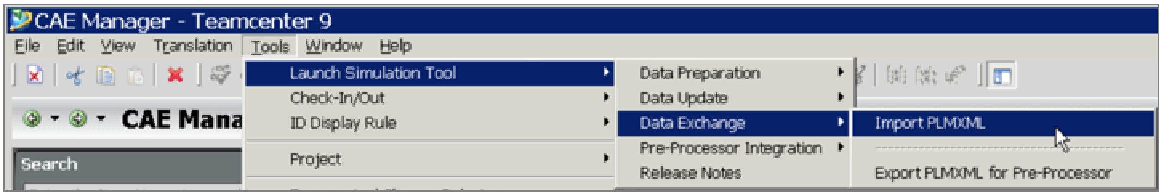


Figure 1475:

Import BOMs

Import BOMs in HyperMesh via Teamcenter - HyperMesh Integration

Directly launch HyperMesh and import the selected BOM.

The selected BOM and associated files will be exported from Teamcenter and automatically loaded in the HyperMesh Part Browser, which replaces the Assembly Browser used in previous versions.

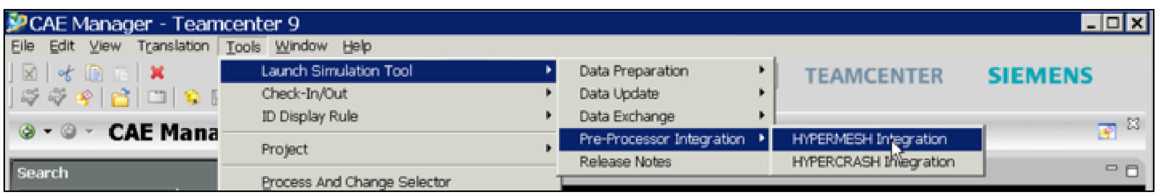


Figure 1476:

Import BOMs in HyperMesh via the HyperMesh Import - BOM Tab

1. In Teamcenter, select the **Pre-Processor Integration** option to export the PLMXML package to the last run folder, which Teamcenter continues to interact with until the HyperMesh session is closed.

   Once closed, Teamcenter prompts you to accept or reject updates. The last run folder is then automatically cleaned up.
2. Export the PLMXML package so that you can work on it over time by selecting the **Export PLMXML for Pre-Processor** option.

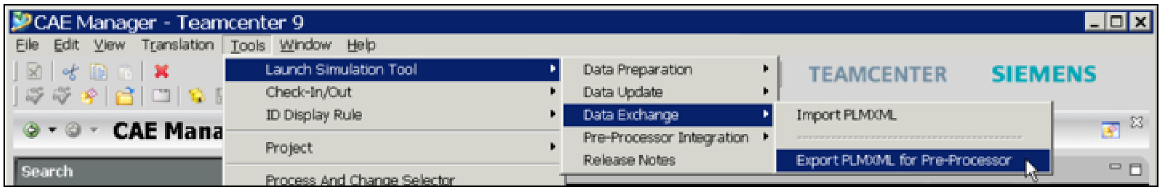


Figure 1477:

3. The resulting PLMXML BOM file, along with its associated data files, will be exported to the last data folder and can be imported directly into HyperMesh via the Import - BOM tab.

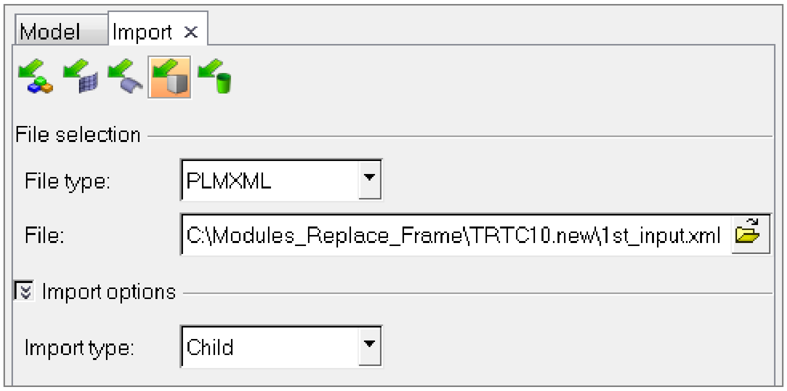


Figure 1478:

The BOM is displayed in the Part Browser.

Import BOMs in HyperMesh via Teamcenter - HyperMesh Integration

Directly launch HyperMesh and import the selected BOM.

The selected BOM and associated files will be exported from Teamcenter and automatically loaded in the HyperMesh Part Browser, which replaces the Assembly Browser used in previous versions.

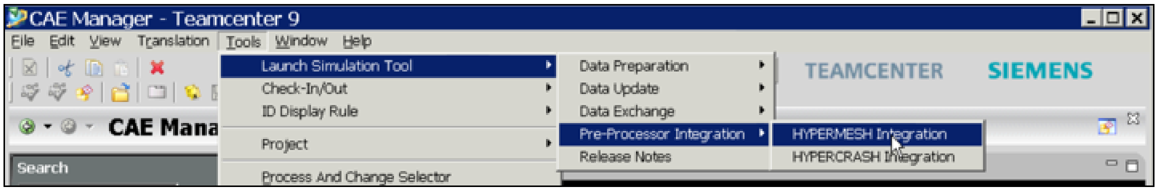


Figure 1479:

Import BOMs in HyperMesh via the HyperMesh Import - BOM Tab

1. In Teamcenter, select the **Pre-Processor Integration** option to export the PLMXML package to the last run folder, which Teamcenter continues to interact with until the HyperMesh session is closed.

   Once closed, Teamcenter prompts you to accept or reject updates. The last run folder is then automatically cleaned up.
2. Export the PLMXML package so that you can work on it over time by selecting the **Export PLMXML for Pre-Processor** option.

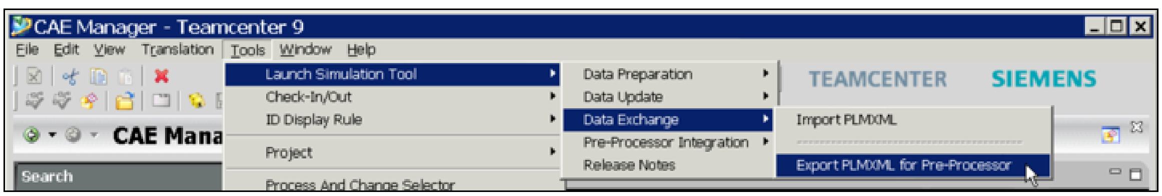


Figure 1480:

3. The resulting PLMXML BOM file, along with its associated data files, will be exported to the last data folder and can be imported directly into HyperMesh via the Import - BOM tab.

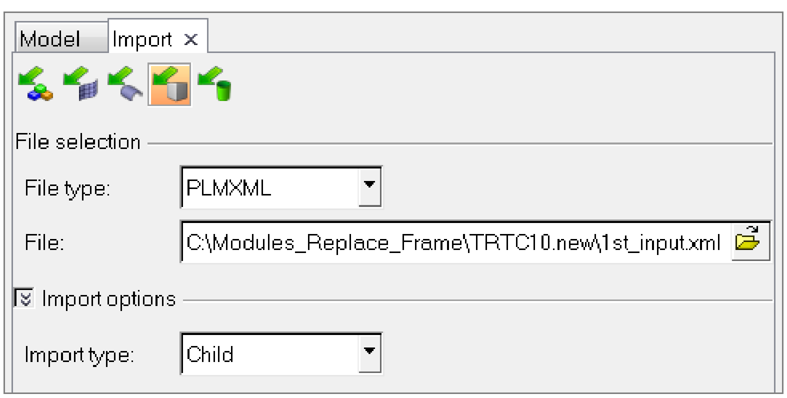


Figure 1481:

The BOM is displayed in the Part Browser.

Export BOMs

1. In the Part Browser, right-click on the Model part assembly and select **Export PLMXML** from the context menu.
   You will automatically be directed to the Export - BOM tab, where the correct name and path will be set, using the `lst_output.xml` file name expected by the Teamcenter-HyperMesh integration.
2. For File Type, choose to export BOM files as either PLMXML or UDMXML.

   For the Teamcenter-HyperMesh process, PLMXML will be used.
3. In the File field, enter the path to exported BOM file.

   For the Teamcenter-HyperMesh process, the file should be named `lst_output.xml`, and the file should be saved to the same location as the original BOM file, `lst_input.xml`.
4. To warn you before saving a BOM file over an existing file, select **Prompt before overwrite**.
5. To save the assembly monolithic file, according to the PLMXML Master Format attribute value, while exporting the BOM, select **Save Monolithic file**.
6. To update PDM attributes based on part attributes prior to writing the exported BOM file, select **Update Attributes to PDM**.

   Using this option assures that attribute updates made in the session will be sent back to Teamcenter.

Load CAD Representations

Load part representations from your current Teamcenter-HyperMesh or from the Part Library.

When CAD representations are loaded, components are created for the loaded CAD representations. Materials and properties, if available in the CAD file would also be created, and their values would be displayed in the Part Browser columns. Component naming conventions are not yet imposed, nor are the materials and properties created automatically at this time, therefore you can review information within the CAD files themselves prior to mesh representation creation. According to the Teamcenter-HyperMesh naming conventions, component naming and material and property handling will occur during the meshing process.

Load Representations from Current HyperMesh Session

1. In the Part Browser, right-click on part assemblies or parts and select **Representations** > **Load** > **from Session** from the context menu.
2. In the **Change Representation** dialog, Load tab, select a type of representation to load. Representations that exist in the repository are shown in the Representations column, and their availability is indicated in the Available column.

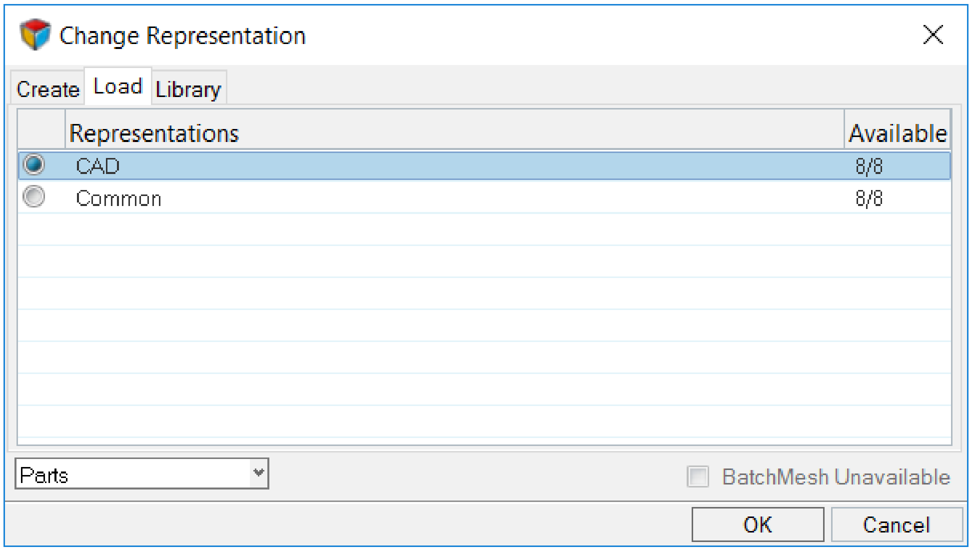


Figure 1482:

3. To send unavailable representations to the Batchmesher directly from the **Change Representation** dialog, select **Batch Mesh Unavailable**.

> **Note:** Available representations will also be loaded when you click **OK** if **Batch Mesh Unavailable** is selected.

4. To save the representations to the Part Library, select the **Save to Library** checkbox.
5. Click **OK**.
6. In the **Confirm Load Representation** dialog, specify how representations are loaded.
    - Click **Load All** to load all available representations into the current session for selected parts.
    - Click **Skip Loaded** to ignore representations that are already loaded for selected parts.

As representations are created and saved to the repository they are displayed in the Available Representations pane.

Load Representations from the Part Library

After importing a BOM that has representations saved in the Part library, you can load the saved representations.

1. In the Part Browser, right-click on a part/part assembly and select **Representations** > **Load** > **from Library** from the context menu.
2. In the **Change Representation** dialog, Library tab, select representations and click **OK**.

The selected representations are loaded to the Representation folder on the file system. Similarly, every time a BOM is imported, files are loaded to the Representations folder, if you retrieve them from the Part Library.

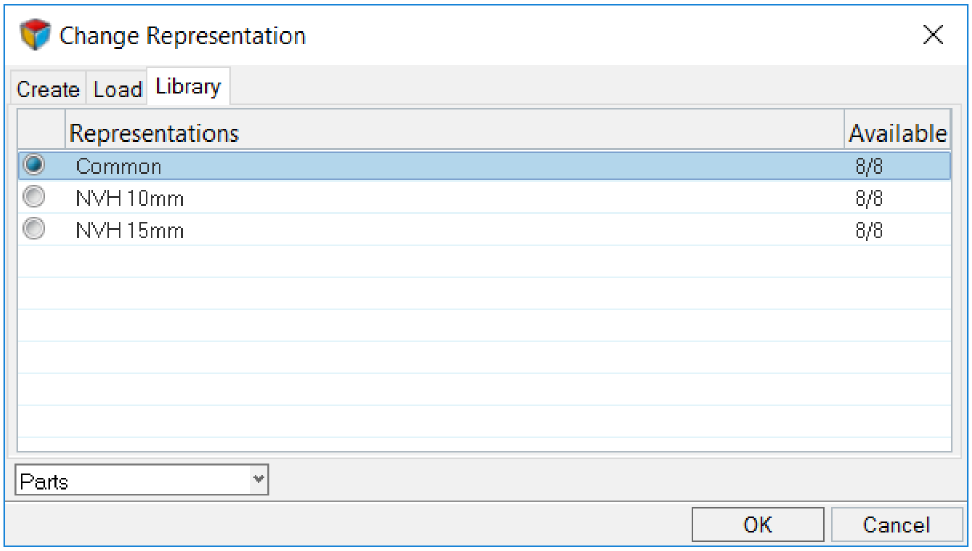


Figure 1483:

Create Mesh Representations

1. In the Part Browser, right-click and select **Representation** > **Create** from the context menu.
2. In the **Change Representation** dialog, Create tab, select the **NVH_Standard_8** representation.

   Each representation is uniquely identified by its alias and file type. For example, "NVH_Standard_8/HyperMesh" and "NVH_Standard_8/Nastran" representations.

   The appropriate representation will automatically be selected, based on the available PLMXML metadata: discipline, mesh representation, and mesh density. If the appropriate metadata is unavailable, then you can select the desired representation type to create. In the **Change Representation** dialog, Create tab, only HyperMesh representations can be created.

   Per the current Teamcenter-HyperMesh integration rules, only one representation per file type is allowed per part. This means that you can only have a single HyperMesh file type representation, Nastran file type representation, CAD representation, and so on for each part. For example, if you create a NVH_Standard_8/HyperMesh representation and then subsequently create a NVH_Coarse_15/HyperMesh representation, the NVH_Coarse_15/HyperMesh representation will become the only HyperMesh representation for the given part.

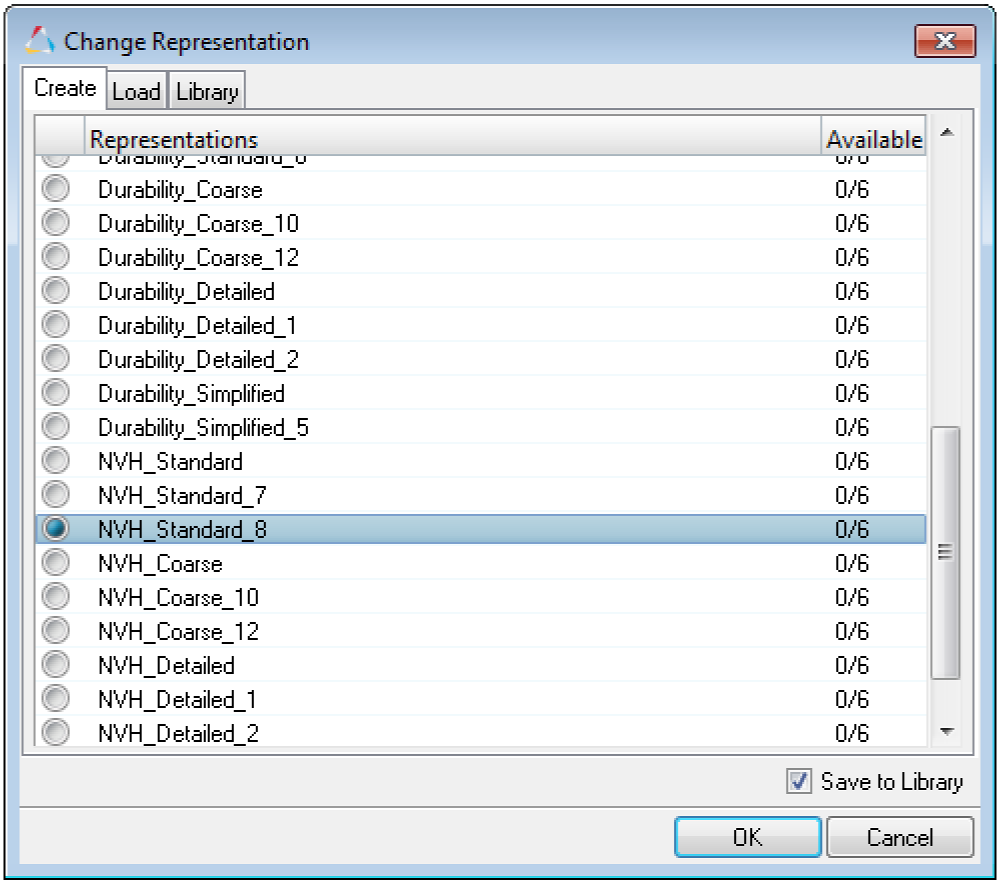


Figure 1484:

3. To save the newly created representations to the Part Library select the **Save to Library** checkbox.
4. Click **OK**.

Batchmesher is invoked, and meshes are created using the appropriate batch meshing parameter and criteria files.

The parameter and criteria file mappings are configurable based on the `batchmesher_config.cfg` file available in the installation.

As part of the post-batchmeshing process, each part's components are created, using the Teamcenter-HyperMesh naming convention. In addition, the loaded representation (NVH_Standard_8 in this case) is displayed, and the component ID (CID), property ID (PID), Material, and Thickness are created, per the values available in the PLMXML BOM.

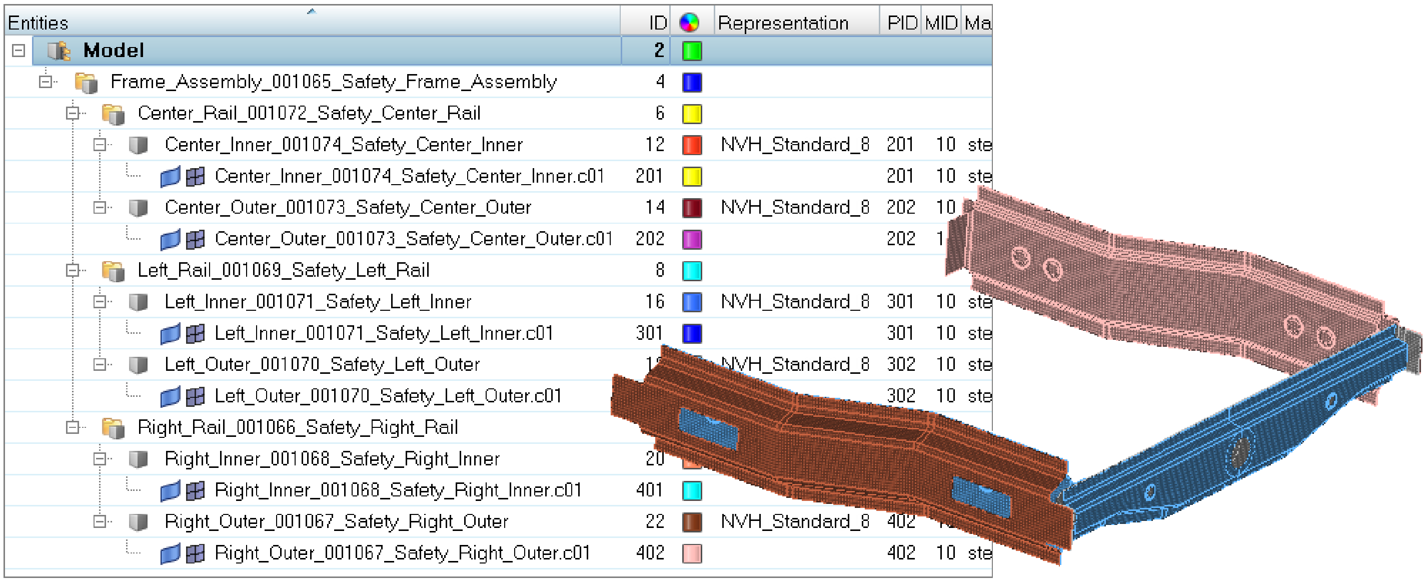


Figure 1485:

Update Metadata from PDM

1. In the Part Browser, right-click on the part(s) to update and select **Representations** > **Update** from the context menu.
2. Update metadata accordingly.
   - Rename the selected parts or part's component, or create if it does not already exist.
   - Create a material based on PDM Material and PDM MID.

     If a material of the given ID already exists, then it will not be overwritten or recreated in order to avoid overwriting any existing material properties.

     Only linear attributes are updated. Default steel attributes are used with the following unit system: millimeter, second, tonne, and Newton.
   - Create a property based on PDM PID and PDM Thickness*.

     If a property of the given PID already exists, its thickness will be updated based on PDM Thickness; however, the existing property will not be recreated. Only relevant metadata will be updated in order to avoid overwriting any existing property card values.

> **Note:**
> *If the PDM MeshFlag attribute is set to **SMT (Solid Mesh Tetra)** or **SMH (Solid Mesh Hexa)**, than a solid card image will be assigned to the property.

Sync Metadata to PDM

Sync PDM metadata (PDM PID, PDM Thickness, PDM Material, and PDM MID) based on a selected part or part's metadata (PID, Thickness, Material, MID).

In the Part Browser, right-click on the part(s) to sync and select **Representations** > **Sync Metadata** from the context menu.

*When a part has multiple properties, only the first property will display in the PID PDM field when you select **Sync Metadata**.

Save and Open HyperMesh Models

You can save the HyperMesh file at any time. The HyperMesh session stores all of the BOM-related information, including hierarchy, metadata, and representations. The original PLMXML BOM file is no longer required to load, view, or modify any of this information.

Save the Current HyperMesh Models

1. From the menu bar, click **File** > **Save** > **Model**.
2. In the **Save Model As** dialog, save the model to your working directory.

Open HyperMesh Models

1. From the menu bar, click **File** > **Open** > **Model**.
2. In the **Open Model** dialog, open the recently saved HyperMesh model.

All of the BOM-related information is retained with the most recently loaded representations loaded in the session.

Entities	ID		Representation	PID	MID	Material	Thickness	PDM PID	PDM MID	PDM Material	PDM Thickness
Model	**2**										
Frame_Assembly_001065_Safety_Frame_Assembly	4										
Center_Rail_001072_Safety_Center_Rail	6										
Center_Inner_001074_Safety_Center_Inner	12		NVH_Standard_8	201	10	steel	3.000000	201	10	steel	3
Center_Inner_001074_Safety_Center_Inner.c01	201			201	10	steel	3.000000				
Center_Outer_001073_Safety_Center_Outer	14		NVH_Standard_8	202	10	steel	3.000000	202	10	steel	3
Center_Outer_001073_Safety_Center_Outer.c01	202			202	10	steel	3.000000				
Left_Rail_001069_Safety_Left_Rail	8										
Left_Inner_001071_Safety_Left_Inner	16		NVH_Standard_8	301	10	steel	2.500000	301	10	steel	2.5
Left_Inner_001071_Safety_Left_Inner.c01	301			301	10	steel	2.500000				
Left_Outer_001070_Safety_Left_Outer	18		NVH_Standard_8	302	10	steel	2.500000	302	10	steel	2.5
Left_Outer_001070_Safety_Left_Outer.c01	302			302	10	steel	2.500000				
Right_Rail_001066_Safety_Right_Rail	10										
Right_Inner_001068_Safety_Right_Inner	20		NVH_Standard_8	401	10	steel	2.500000	401	10	steel	2.5
Right_Inner_001068_Safety_Right_Inner.c01	401			401	10	steel	2.500000				
Right_Outer_001067_Safety_Right_Outer	22		NVH_Standard_8	402	10	steel	2.500000	402	10	steel	2.5
Right_Outer_001067_Safety_Right_Outer.c01	402			402	10	steel	2.500000				

Figure 1486:

Model Verification

10

Perform automatic checks on CAD models, and identify potential issues with geometry that may slow down the meshing process using the Verification and Comparison tools.

This chapter covers the following:

The Verification and Comparison tools performs intersection, spot weld, connection and part comparison checks on the CAD models, and exports a clear report that highlights the issues in MS-Excel spreadsheet and PowerPoint slides, with embedded HyperView Player.

This solution can drastically decrease the meshing process by enabling you to identify and correct problems with CAD data at early stages before the meshing process begins. Current users of the Model Verification and Comparison tools have reported time savings up to 90% compared to the traditional process.

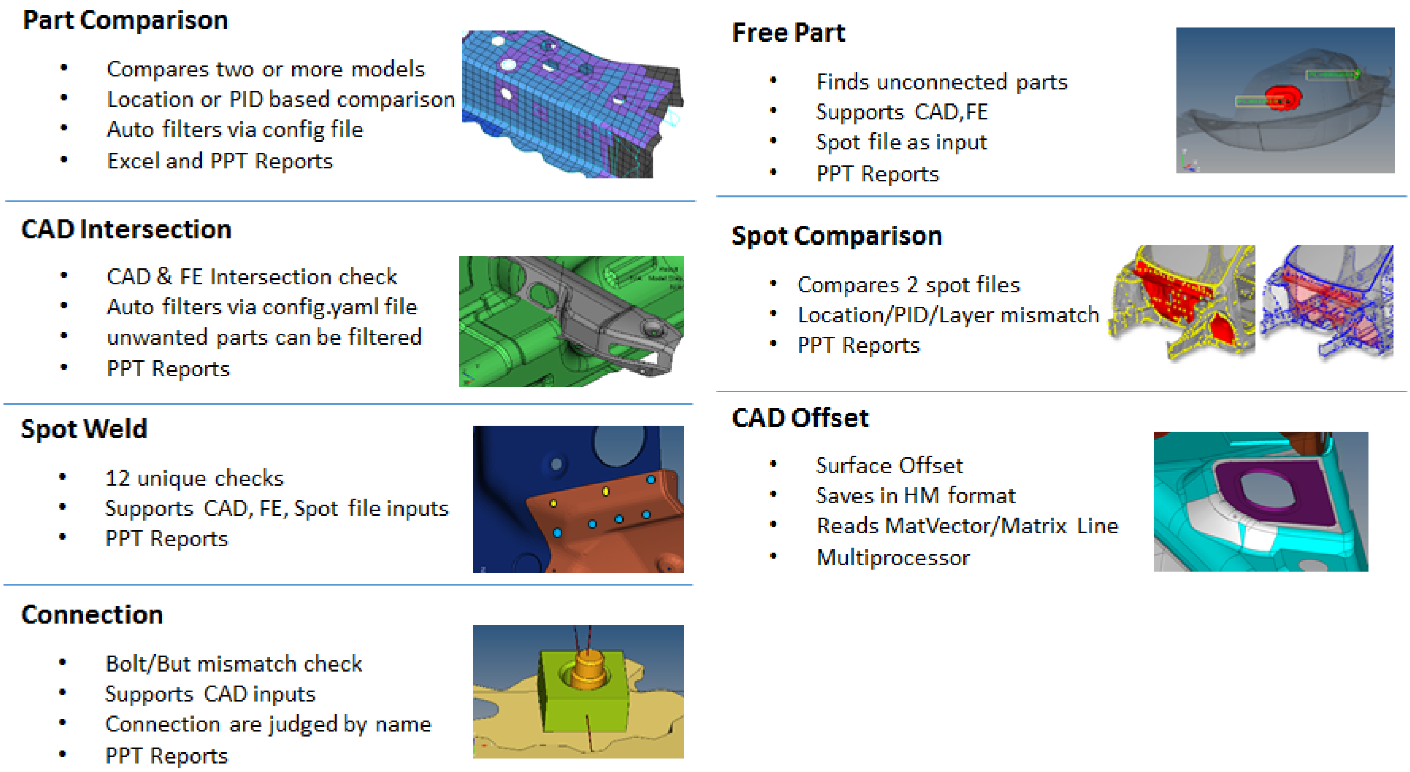


Figure 1487: Supported Checks

Launch Verification Browser

Use the Verification browser to inspect CAD data supplied from your design or CAD team.

The **Verification** browser supports Offset, Intersection, Spot Weld, Connection, Spot Comparison, and Free Part checks. After performing the verification, results are exported to reports.

From the menu bar, select **View** > **Browsers** > **HyperMesh** > **Verification**.

The **Verification** browser opens.

Browser User Interface

Overview of the Verification and Comparison browser user interface.

The **Verification** and **Comparison** browser consists of a search, filter, column data, and Entity Editor.

You can switch to the Part browser any time to perform detailed operations on a part.

Column Details

By default, the following attributes are listed as columns in the first pane.

Name

Description

ID
: Part assembly and model IDs.

Color
: Part entity colors.

MVD Part Name
: Part entity name.

MVD PID
: Entity specific property ID's. At the part level, the IDs for referenced properties are displayed. At Component level, IDs of the referenced properties are displayed.

UID
: Part assembly and part Unique IDs.

MVD Revision
: Major revision, Study revision, and Library Part revision.

MVD Material
: Material name of the part

MVD Thickness
: Entity specific thickness. At the part level it displays the thicknesses of the referenced properties. At component level it shows the thickness of the referenced property.

MVD Action
: Type of operation for the part entity.

MVD Modeling Type
Type of modeling either mid-plan or skin mesh for the part entity.

MVD Mesh Size
Mesh size for the part entity.

MVD Error
Errors accrued in respective operation on the part.

MVD Function Name
Sub assembly name of the part.

MVD Assembly Name
Assembly name of the part.

MVD Counter Parts
Matching or intersecting part's UDMID.

MVD Bounding Box
Bounding box of the part.

MVD Oriented Bounding Box
Oriented bounding box of the part.

MVD Weight
Weight of the part.

MVD Delta Weight
Weight difference between part and counter part.

MVD Load Collectors
Load collector card name of the part.

MVD Contacts
Contact card names of the part.

MVD Boundary Conditions
BCs card names of the part.

MVD Representation File
Default representation file path of the part.

Context Menu

Overview of the Verification and Comparison browser context menu.

Name

Description

Import
Import CAD or FEM as a BOM file into the Verification browser for manual cleanup, and then run verifications. It is recommended to use a respective check in case no data cleanup in necessary, as this Import function is integrated into each verification function.

Export

Export an `.xml` file to a specified location. You can use this `.xml` file to run Model verification using the **XML** option.

Check

Check functions. For more information, refer to Perform Model Checks.
"Check (Comparison, Representation Comparison)", "Multiple Checks".

Parts

Consists of part functions.
"Count","Rename", "Renumber" "Export", "Delete" , "Load", "Load in other", "Load in HyperView Player", "Unload".

Representations

Consist of representation import options "Load", "Load in other HM", "Load in Hyper View Player", "Unload", and "Save".

Review Results

Comparison and Intersection results are directly displayed in HyperMesh graphics. For Comparison function, you can right-click on **Base Part** and select **review** to display the base part and matched variant parts in the graphics area along with mismatched area with temp nodes.

CSV

Compare

Compare and identify unmatched information between a CSV file and the imported BOM file. Results will be populated in the Error column. The CSV file header details can be configured through the Configuration option, under the CSV comparison section.

Export

Export selected parts and children parts in to a CSV file.

Update

Read a CSV file exported with the Export function, and update the information displayed in the Verification browser. The **Entities** column is the reference column considered for updating the BOM.

Tools

Consist of "Divide Slides", "Open In Explorer", "Reset Results", "Get Revision menus", details are available in the functional detail sections.

Configuration

Open the configuration dialog.

Launch Comparison Browser

Use the Comparison browser to compare and identify the topology difference between two model/part assembly/parts.

The **Comparison** browser supports CAD Offset and Comparison check. After the comparison check, the results are reported in PowerPoint, Excel and CSV file, in addition the results can be reviewed and modified directly in HyperMesh.

From the menu bar, select **View** > **Browsers** > **HyperMesh** > **Comparison**.

The **Comparison** browser opens.

Browser User Interface

Overview of the Verification and Comparison browser user interface.

The **Verification** and **Comparison** browser consists of a search, filter, column data, and Entity Editor.

You can switch to the Part browser any time to perform detailed operations on a part.

Column Details

By default, the following attributes are listed as columns in the first pane.

Name

Description

ID
: Part assembly and model IDs.

Color
: Part entity colors.

MVD Part Name
: Part entity name.

MVD PID
: Entity specific property ID's. At the part level, the IDs for referenced properties are displayed. At Component level, IDs of the referenced properties are displayed.

UID
: Part assembly and part Unique IDs.

MVD Revision
: Major revision, Study revision, and Library Part revision.

MVD Material
: Material name of the part

MVD Thickness
: Entity specific thickness. At the part level it displays the thicknesses of the referenced properties. At component level it shows the thickness of the referenced property.

MVD Action
: Type of operation for the part entity.

MVD Modeling Type
: Type of modeling either mid-plan or skin mesh for the part entity.

MVD Mesh Size
: Mesh size for the part entity.

MVD Error
: Errors accrued in respective operation on the part.

MVD Function Name
: Sub assembly name of the part.

MVD Assembly Name
: Assembly name of the part.

MVD Counter Parts
: Matching or intersecting part's UDMID.

MVD Bounding Box
: Bounding box of the part.

MVD Oriented Bounding Box
: Oriented bounding box of the part.

MVD Weight
: Weight of the part.

MVD Delta Weight
: Weight difference between part and counter part.

MVD Load Collectors
: Load collector card name of the part.

MVD Contacts
: Contact card names of the part.

MVD Boundary Conditions
: BCs card names of the part.

MVD Representation File
Default representation file path of the part.

Context Menu

Overview of the Verification and Comparison browser context menu.

Name

Description

Import
Import CAD or FEM as a BOM file into the Verification browser for manual cleanup, and then run verifications. It is recommended to use a respective check in case no data cleanup in necessary, as this Import function is integrated into each verification function.

Export
Export an `.xml` file to a specified location. You can use this `.xml` file to run Model verification using the **XML** option.

Check
Check functions. For more information, refer to Perform Model Checks.
“Check (Comparison, Representation Comparison)”, “Multiple Checks".

Parts
Consists of part functions.
“Count”,”Rename”, “Renumber” “Export”, “Delete” , “Load”, “Load in other”, “Load in HyperView Player”, “Unload".

Representations
Consist of representation import options “Load”, “Load in other HM”, “Load in Hyper View Player”, “Unload”, and “Save”.

Review Results
Comparison and Intersection results are directly displayed in HyperMesh graphics. For Comparison function, you can right-click on **Base Part** and select **review** to display the base part and matched variant parts in the graphics area along with mismatched area with temp nodes.

CSV

Compare
Compare and identify unmatched information between a CSV file and the imported BOM file. Results will be populated in the Error column. The CSV file header details can be configured through the Configuration option, under the CSV comparison section.

Export
Export selected parts and children parts in to a CSV file.

Update
Read a CSV file exported with the Export function, and update the information displayed in the Verification browser. The **Entities** column is the reference column considered for updating the BOM.

Tools

Consist of “Divide Slides”, “Open In Explorer”, “Reset Results”, “Get Revision menus”, details are available in the functional detail sections.

Configuration

Open the configuration dialog.

Import Models

Import a model using Model Verification.

1. In the Comparison/Verification browser, right-click and select **Import** from the context menu.
   The **Import - Model Verification** dialog opens.
2. Select **Import Type** for Base Model.
   - XML (PLMXML or UDMXML)
   - Assembly (CAD assembly files: UG Assembly, CATProduct, JT Assembly…)
   - CAD (Refer Config / CAD section)
   - FE (Refer Config / FE section)
   - Multiple Assembly (Multiple CAD assembly files)
   - Spot Files (Refer Config / FE / Connector section)
3. Select a datatype.

   Respective data types will be listed.
4. Click the folder icon and navigate to the model to import (Folder/File selections).

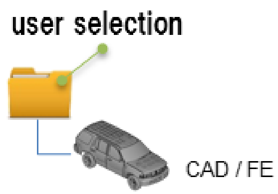
   

   Figure 1488:
5. Choose a method for running the model.
   - Choose **Interactive** to run the model in the same HyperMesh session.
   - Choose **Background** to launch a new HyperMesh session in batch mode.
6. The **Processes** drop-down list becomes active when you select **Background**, and it also depends on the processor. The Model Verification tool launches as many hmbatch processes selected.
   - Choose **Check** to execute only the comparison without the report generation.
   - Choose **Report** to generate only the Reports (Check executed previously).
   - Choose **Both** to perform the comparison report generation sequentially.
7. Click **Run** to execute the function.
8. Click **Stop** to stop all the operation.
   This will close front and background HyperMesh sessions. The Comparison/Intersection browser is populated with the parts structure.

   > **Note:** This function can be used to create XML or to review the model. This function is included in all Verification and Comparison function.

Perform Model Checks

Perform various model checks in the Comparison Browser.

Offset

Offset is mainly used to offset surface to mid plane location and to convert the CAD files to `.hm` format.

Before you begin, it is expected that the BOM or CAD data contains thickness information in order to perform offset. If a thickness value does not exist, you can input a constant thickness for offset.

1. In the Comparison browser, right-click and select **Comparison/Verification** > **Offset** from the context menu.
   The **Offset** dialog opens.
2. Select an **Import Type** for Base Model.
   - XML (PLMXML or UDMXML)
   - Assembly (CAD assembly files: UG Assembly, CATProduct, JT Assembly…)
   - CAD (Refer Config / CAD section)
   - FE (Not Applicable)
   - Multiple Assembly (Multiple CAD assembly files)
   - Spot Files (Refer Config / FE / Connector section)
3. Select a datatype.

   Respective data types will be listed.
4. Click the folder icon and navigate to the model to import (Folder/File selections).

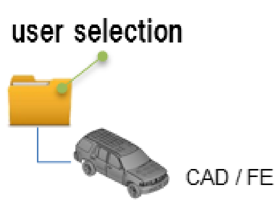


Figure 1489:

5. In the **Thickness[%]** field, enter the thickness percentage used to offset/move the geometry.
6. In the **Thickness** field, enter the constant component thickness in case thickness value does not exist in the CAD data or BOM file.

   Offset distance = 50%*Thickness
7. Choose a method for running the model.
   - Choose **Interactive** to run the model in the same HyperMesh session.
   - Choose **Background** to launch a new HyperMesh session in batch mode.
8. The **Processes** drop-down list becomes active when you select the **Background** option and it also depends on the processor.

   The Model Verification tool launches as many hmbatch processes selected.

- Choose **Check** to execute only the comparison without the report generation.
- Choose **Report** to generate only the Reports (Check executed previously).
- Choose **Both** to perform the comparison report generation sequentially.

9. Click **Run** to execute the function.
10. Click **Stop** to stop all the operation.
    This will close front and background HyperMesh sessions. The browser is populated with offset data in `.hm` format. The remainder of the check will use these `.hm` files.

Comparison

Comparison is used to compare two models using the Model Verification tool.

1. In the Comparison browser, right-click and select **Check** from the context menu.
2. Select **Import Type** for Base Model.
    - XML (PLMXML or UDMXML)
    - Assembly (CAD assembly files: UG Assembly, CATProduct, JT Assembly…)
    - CAD (Refer Config / CAD files)
    - FE (Solver deck file)
    - Multiple Assembly (Multiple CAD assembly files)
    - Spot Files (Not applicable)
3. Select a datatype (Respective data types will be listed).
4. Click the folder icon and navigate to the model to import (Folder/File selections).

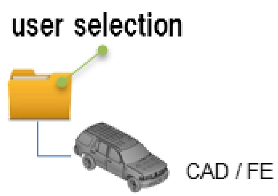


Figure 1490:

5. Select **Import Type** for Variant Model.
    - XML (PLMXML or UDMXML)
    - Assembly (CAD assembly files: UG Assembly, CATProduct, JT Assembly…)
    - CAD (Refer Config / CAD section)
    - FE (nastran, pam, radioss, abaqus, optistruct, hyperMesh)
    - Multiple Assembly (Multiple CAD assembly files)
    - Spot Files (Not applicable)
6. Select a datatype.
    Respective data types will be listed.
7. Click the folder icon and navigate to the model to import (Folder/File selections).

Figure 1491:

8. In the **Tolerance** field, enter a value between 0.1 to 5mm to display unmatched entities.
9. Enter the **Threshold[%]** value.

   Its min match percentage value, parts that are matching less than this value is treated as Mismatching and parts that are matching above this value will be treated as Matching parts.

   If the match percentage value is greater than the threshold value, then the two entities are shown in report as overlaid image.
10. In the **Report Path**, field navigate to the directory where the reports generated by the Model Verification tool will be stored.
11. In the **Project Name** field, enter a name for the project.
12. In the **Slide Number** field, enter a starting number of the slides to use in the PowerPoint report.
13. Select a method for running the model.
    - Choose **Interactive** to run the model in the same HyperMesh session.
    - Choose **Background** to launch a new HyperMesh session in batch mode.
14. The **Processes** drop-down list becomes active when the user selects the **Background** option and it also depends on the processor.

    The Model Verification tool launches as many hmbatch processes selected.
    - Choose **Check** to execute only the comparison without the report generation.
    - Choose **Report** to generate only the Reports (Check executed previously).
    - Choose **Both** to perform the comparison report generation sequentially.
15. Click **Run** to execute the function.
16. Click **Stop** to stop all the operation.
    This will close front and background HyperMesh sessions.
17. Click **View Report** to display the Summary PowerPoint report.

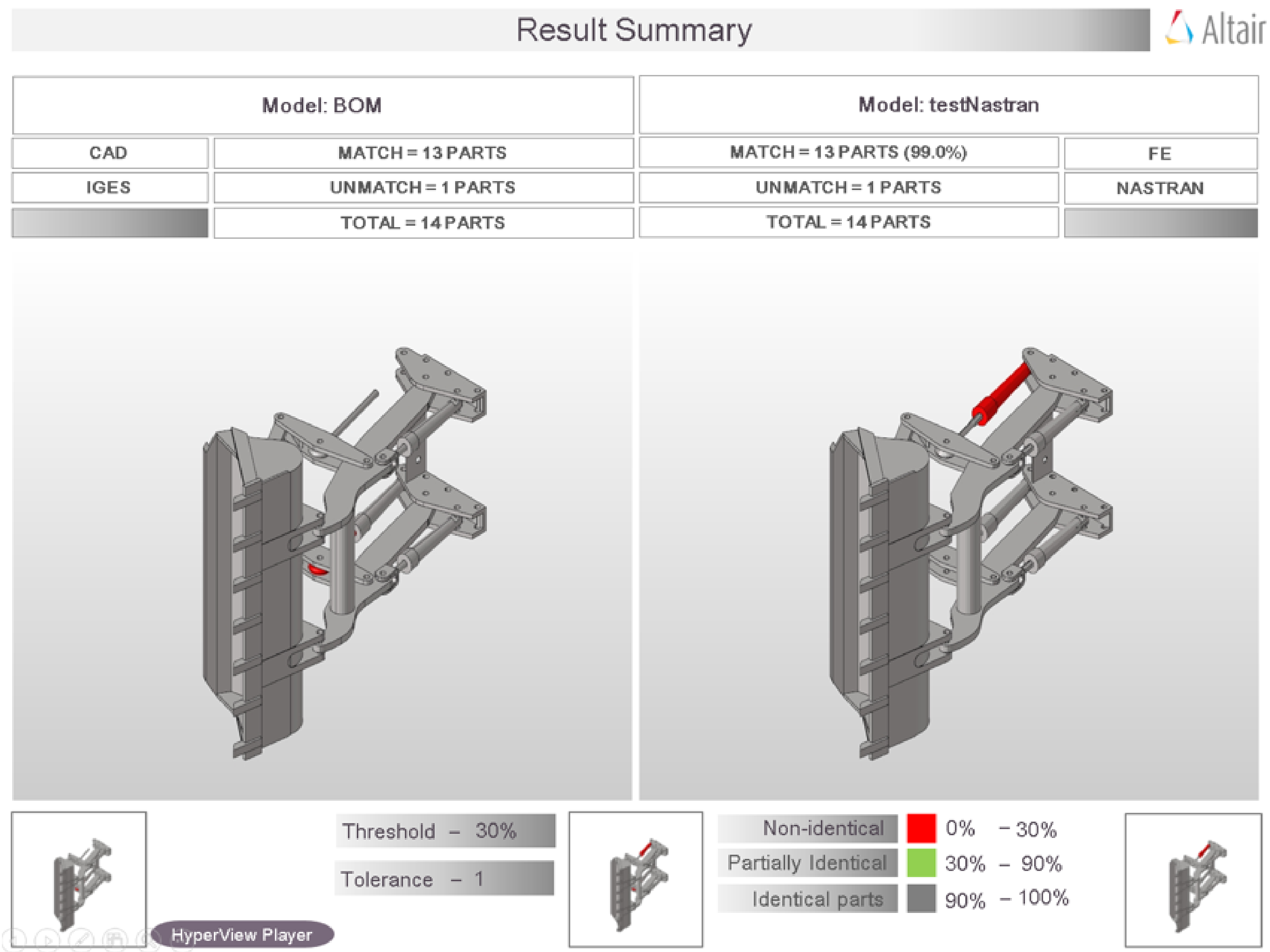


Figure 1492: PowerPoint Summary
(Stored in Report Path)

The slide shows the comparison between a source assembly and the target assembly. The left half of the image shows the source assembly and the right half of the image shows the target assembly. In the example illustrated above, the source assembly has a total of 39 parts. The target assembly has a total of 38 parts. The matching parts (between the source and target assemblies) are highlighted in black and the unmatched parts are shown in red.

The second slide shows the matched and unmatched results for individual parts, number of parts are limited to 40 which matches Excel report. This limitation can edited in configuration.

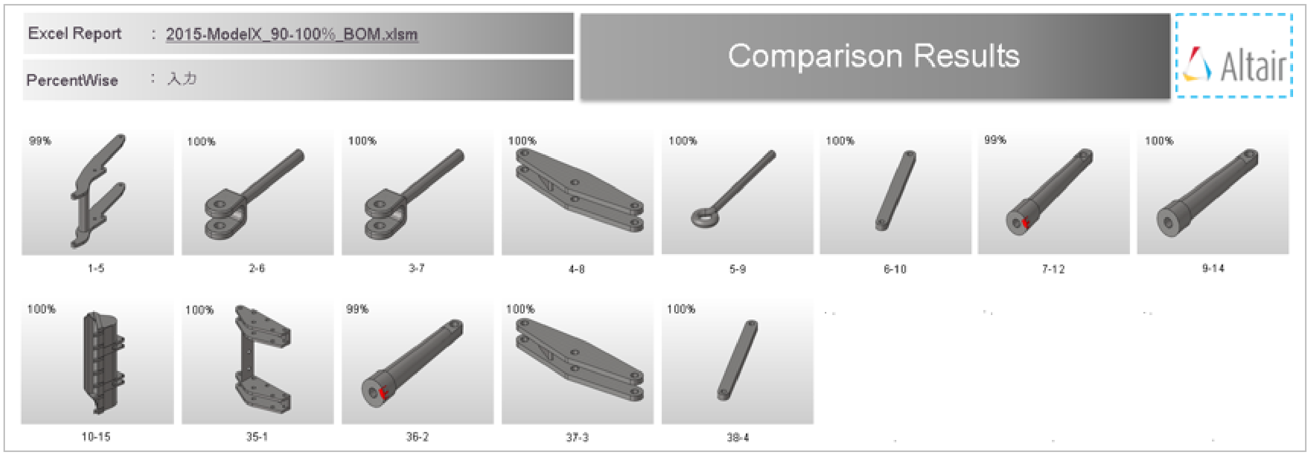


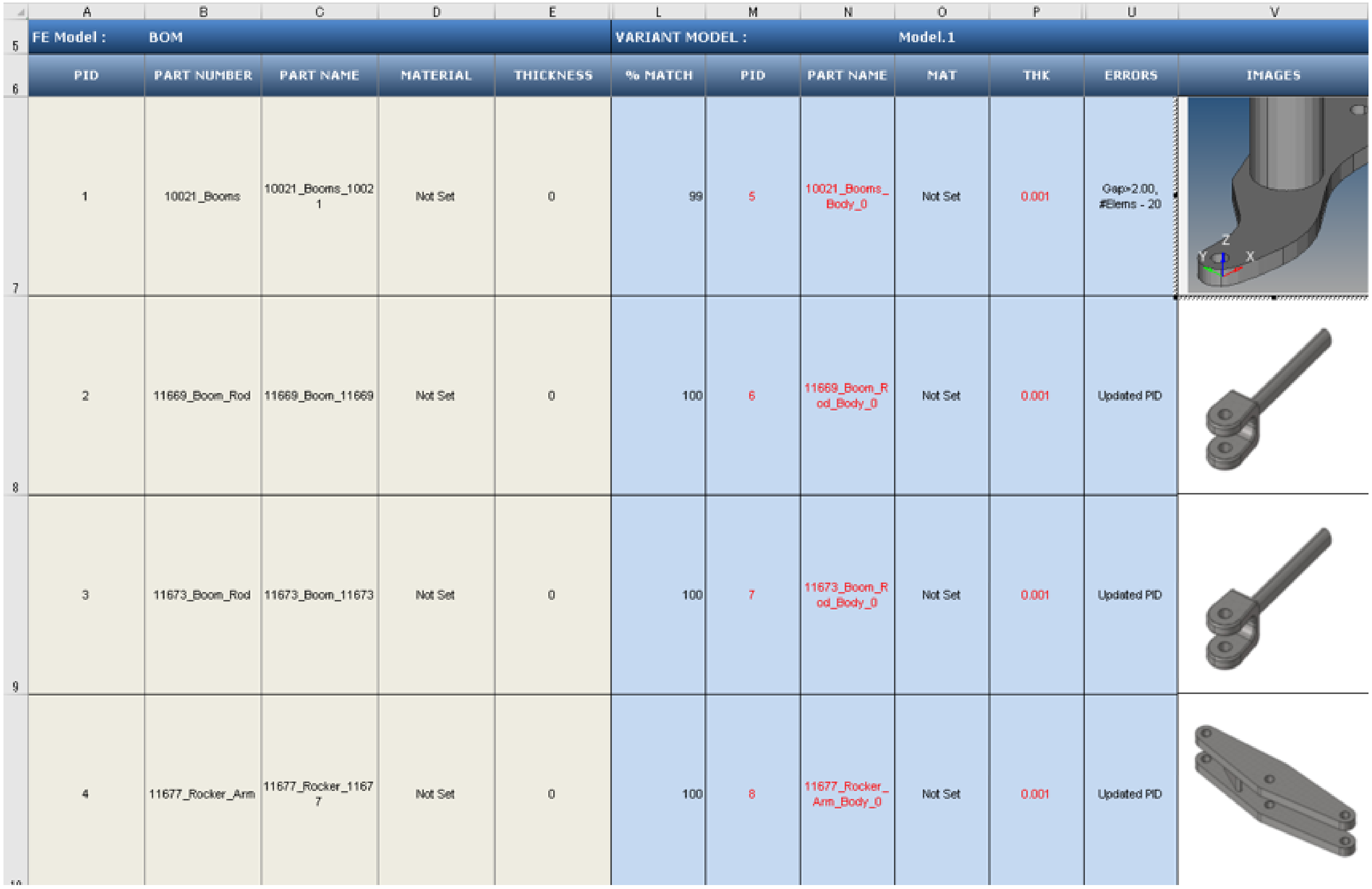
Figure 1493: Matched and Unmatched Results
(from 2nd slide of Summary PowerPoint Report)

Output

Overview of the output generated from a Comparison check.

Excel

The contents of the Excel report depend on the number of parts/models that are being compared. The report consists of the details of matched and unmatched parts from the assemblies selected for comparison.



FE Model :	BOM				VARIANT MODEL :		Model.1				
PID	PART NUMBER	PART NAME	MATERIAL	THICKNESS	% MATCH	PID	PART NAME	MAT	THK	ERRORS	IMAGES
1	10021_Booms	10021_Booms_10021	Not Set	0	99	5	10021_Booms_Body_0	Not Set	0.001	Gap>2.00, #Elems - 20	
2	11669_Boom_Rod	11669_Boom_11669	Not Set	0	100	6	11669_Boom_Rod_Body_0	Not Set	0.001	Updated PID	
3	11673_Boom_Rod	11673_Boom_11673	Not Set	0	100	7	11673_Boom_Rod_Body_0	Not Set	0.001	Updated PID	
4	11677_Rocker_Arm	11677_Rocker_11677	Not Set	0	100	8	11677_Rocker_Arm_Body_0	Not Set	0.001	Updated PID	

Figure 1494:

The information in the Excel reports can be classified into different sections.

Base Model Input

Input information about each of the parts of the CAD model that the tool uses for performing the a comparison check. The Base Model Input is the first assembly shown in the Part Browser, and can be cross checked at row 5.

Variant Model Input

Results of the comparison check. The match percentage column shows a part vise match percentage. If the value of match percentage for a part is 0 or less than the threshold percentage, then the part is said to be unmatched with the corresponding part of the other assembly. If the match percentage is above the threshold value, then the parts are considered matching. The error is reported in the error column. Variant model's Material name or Thickness values are compared against Base model, mismatch values are shown in red text. Variant Model Input increases depending on the number of Variant model found in the Variant Input folder. Variant model name can be cross checked at row 5.

Image Type

Shortcut to the image the part.

- If the image type is JPG, then this shortcut opens the full sized image for viewing.

- If the image type is H3D, then the shortcut opens in the part in H3D format in a new HyperMesh window.

Comparison with Browser Selection

Compare CAD vs. CAD models, FE vs. FE models, and CAD vs. FE models with a Comparison check.

In the Comparison browser, you can select two modules that are required to be compared from the Assembly browser. Use the Import Models function to import data into the browser, and then perform the comparison.

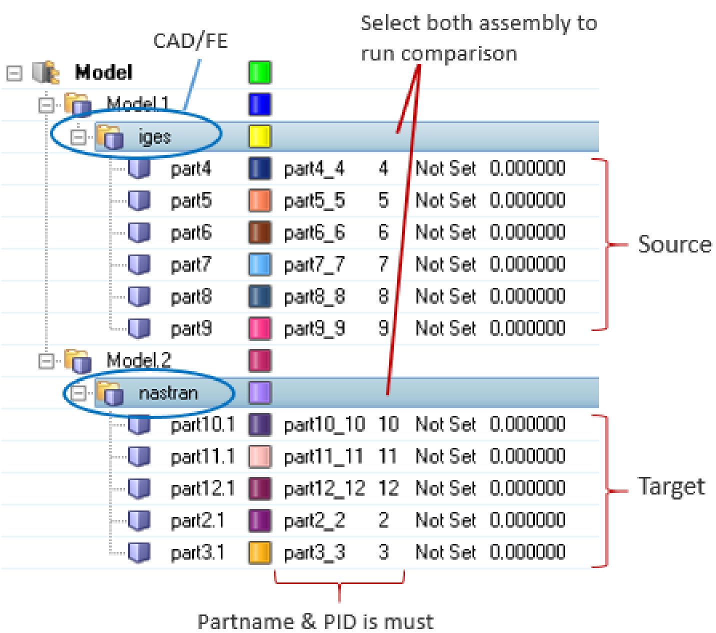


Figure 1495: Import Data to Compare

Comments

- During a Comparison check, the part files may get reloaded several times. Converting all models to the HyperMesh format can improve performance. It is recommended that you perform a No-Offset operation before starting the comparison process by selecting **Config** > **Offset** > **<action>**.
- During a Comparison check, the part files may get reloaded several times. It is recommended that all models be in HyperMesh format to improve performance. Use the Offset operation before performing a comparison.
- When generating reports, Excel or PowerPoint reports may open several times. It is recommended that you do not to launch or close any Microsoft Office application or use the keyboard during this operation.
- Translation or Smetry comparison runs using 1 CPU. A single CPU is automatically utilized, even when you manually set multiple CPUs.

- If more than one input model is found in the variant folder, a multiple model comparison is activated.
- The folder structure is to be maintained when organizing variant model files. CAD files must be managed under variant folders, and you must select the variant root folder. If the variant root contains more than one sub folder (variants), a multiple model (variant) comparison is activated.

Troubleshoot Model Verification Comparison Check Problems

Approaches used to troubleshoot Model Verification comparison check problems.

Comparison check results depend on the following Model Verification settings:

1. The type of CAD files and the corresponding license required to import the CAD files into Model Verification.
2. The tolerance value specified in the GUI (Thickness/2.0).
3. Filters specified, such as those in the following list:
   - Connection Name Filter
   - Area Filter
   - CG Filter
   - Match % Filter
4. Availability of license when using the multiple CPU option. The unit requirement can be calculated using this formula: `40 + ((No. of CPUs * 21) - 21)`

Find which filter parts that are excluded from during comparison.

1. Turn on the debug log in the Configuration panel by selecting **Tools** > **Configuration** > **Log**.
2. Select **log** > **add**.

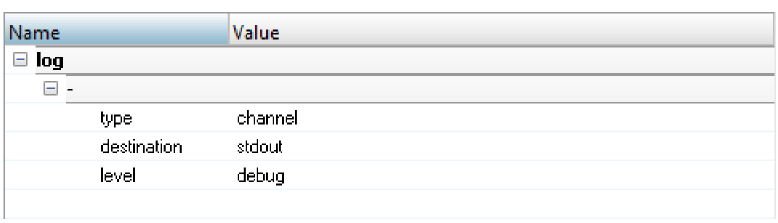


Figure 1496:

3. Restart HyperMesh.
4. Run the check again.
5. Check the Tk Console of the stdout files.

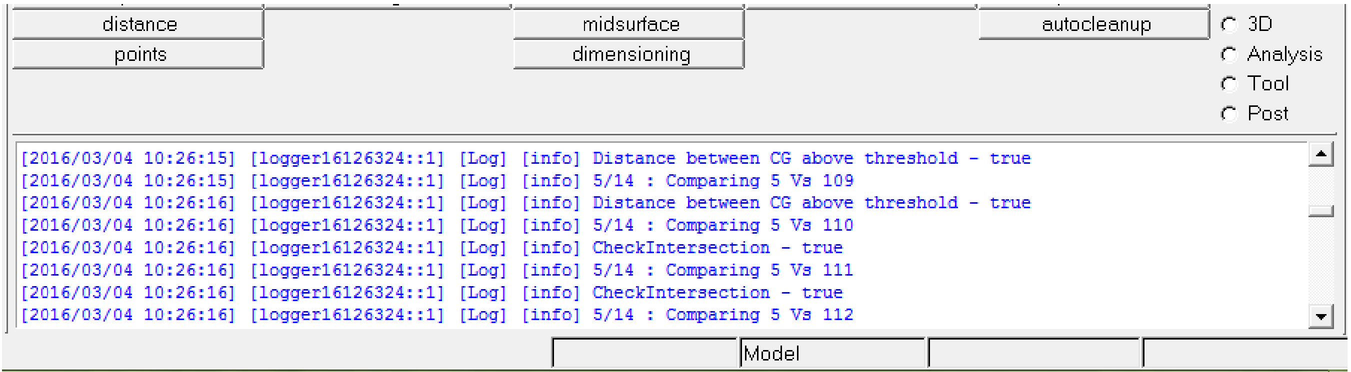


Figure 1497:

6. Based on the information in the log file, identify the filters applied on the concern parts/PID.

   Example: 5 vs 109 comparison is ignored due to the centroid distance between these parts are more then value set in the config “cgtolerable-distance” value. In case these two parts need to be compared, increase the “cgtolerable-distance” from 200 to desired value.

Intersect

Intersect checks provide information in Power Point reports on all the inter-part intersection/overlapping in the CAD parts.

Once the model is done with the Intersection check, the tool locates the point of intersection as a cross-hair and summarizes all the intersections found in the PowerPoint report.

1. In the Comparison browser, right-click and select **Check** > **Intersect** from the context menu. The **Intersection** dialog opens.
2. Select **Import Type** for Base Model.
   - XML (PLMXML or UDMXML)
   - Assembly (CAD assembly files: UG Assembly, CATProduct, JT Assembly…)
   - CAD (Refer Config / CAD files)
   - FE (nastran, pam, radioss, abaqus, optistruct, hyperMesh)
   - Multiple Assembly (Multiple CAD assembly files)
   - Spot Files (Not applicable)
3. Select a datatype.

   Respective data types will be listed.
4. Click the folder icon and navigate to the model to import (Folder/File selections).

   The folder or the selected location must contain the spot weld file. The spot weld file type must be set in the config file.

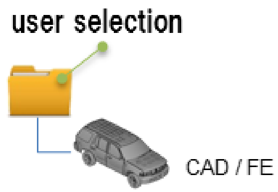


Figure 1498:

5. In the **Allowance** field, enter a value between 0.1 to 5mm to display unmatched entities.
6. In the **Report Path** field, navigate to the directory where the reports generated by the Model Verification tool will be stored.
7. In the **Project Name** field, enter a name for the project.
8. In the **Slide Number** field, enter a starting number of the slides to use in the PowerPoint report.
9. Choose a method for running the model.
    - Choose **Interactive** to run the model in the same HyperMesh session.
    - Choose **Background** to launch a new HyperMesh session in batch mode.
10. The **Processes** drop-down list becomes active when the user selects the **Background** option and it also depends on the processor.

    The Model Verification tool launches as many hmbatch processes selected.
    - Choose **Check** to execute only the comparison without the report generation.
    - Choose **Report** to generate only the Reports (Check executed previously).
    - Choose **Both** to perform the comparison report generation sequentially.
11. Click **Run** to execute the function.
12. Click **Stop** to stop all the operation.
    This will close front and background HyperMesh sessions.
13. Click **View Report** to display the Summary PowerPoint report.

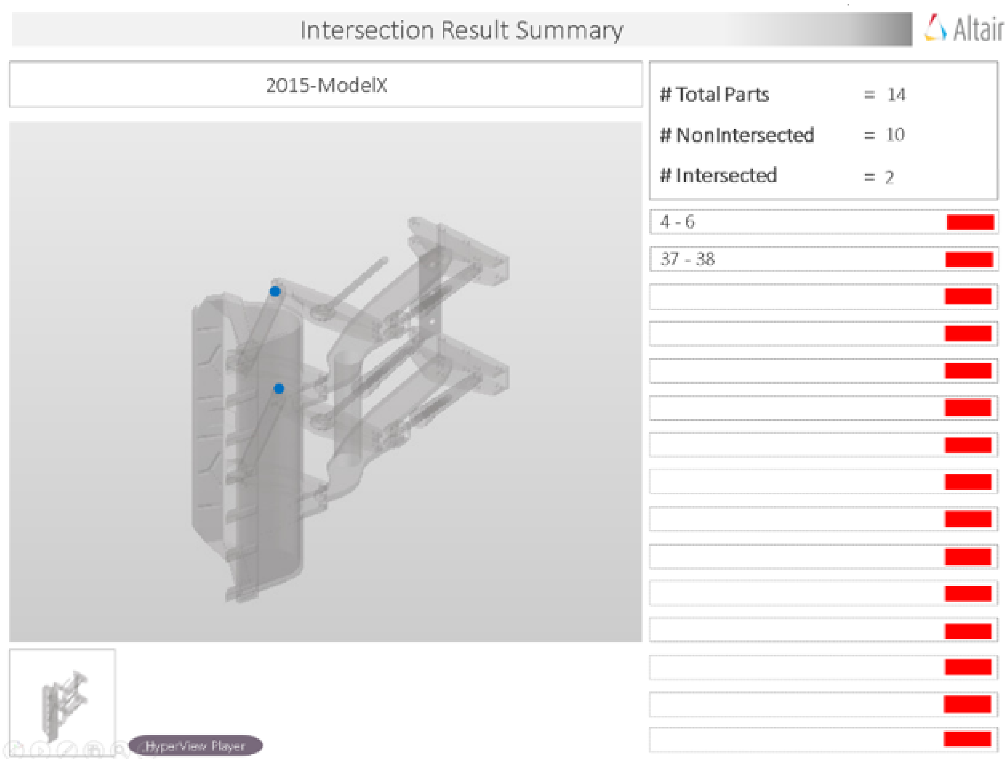


Figure 1499: PowerPoint Summary
(Stored in Report Path)

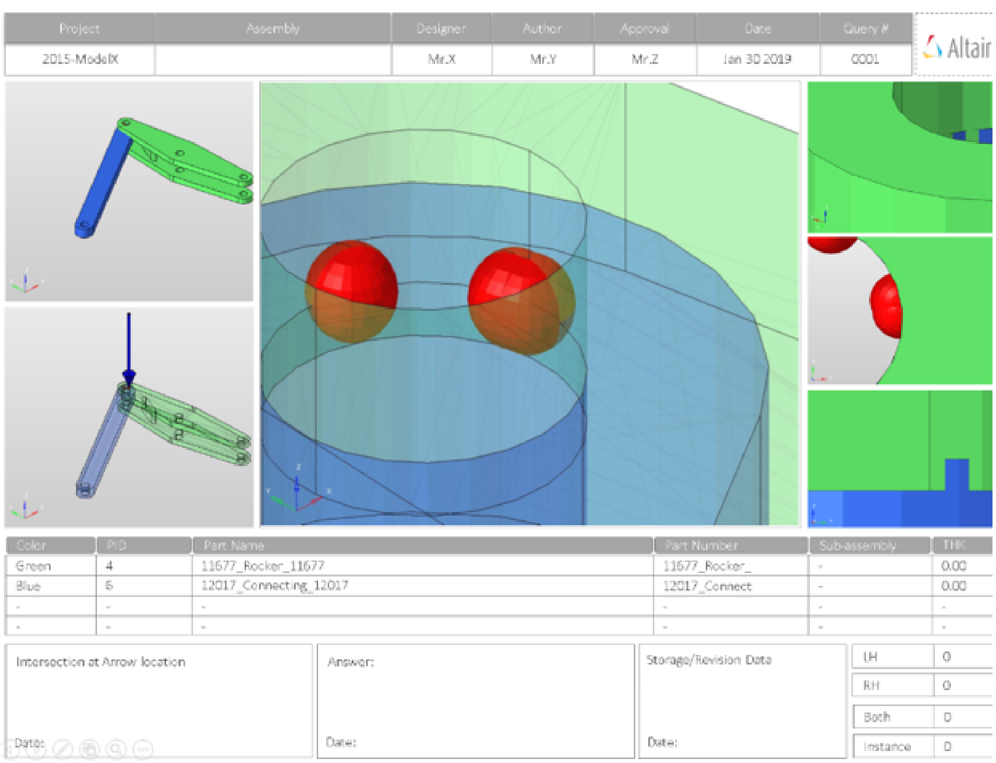


Figure 1500: PowerPoint Detailed Report (Stored in Report Path)

Output

Overview of the PowerPoint report generated from the Intersection check.

A PowerPoint report is separated into several sections.

Report Header
: Project name, assembly name, and slide number.

Assembly Picture
: Image of the components that have intersection issues.

H3D Image
: Section can be viewed in the HyperView Player, and shows the CAD model with the intersection marked with Cross Hairs.

Issue Details
: Details of components which have intersection issues.

User Review
: Add additional information when exchanging information with designers/engineers.

Additional Info
: Occurrences of the issue shown in the current slide. The number of instances show the number of occurrences of the issue.

Spotweld

Spot Weld checks identify problems with spot welds in the model and shows them in the report.

The identified problems in spot welds are highlighted using a cross-hair in the images.

1. In the Comparison Browser, right-click and select **Check** > **Spotweld** from the context menu.
   The **Spotweld** dialog opens.
2. Select **Import Type** for Base Model.
   - XML (PLMXML or UDMXML)
   - Assembly (CAD assembly files: UG Assembly, CATProduct, JT Assembly…)
   - CAD (Refer Config / CAD files)
   - FE (nastran, pam, radioss, abaqus, optistruct, hyperMesh)
   - Multiple Assembly (Multiple CAD assembly files)
   - Spot Files (Not applicable)
3. Select a datatype.

   Respective data types will be listed.
4. Click the folder icon and navigate to the model to import (Folder/File selections).

The folder or the selected location must contain the spot weld file. The spot weld file type must be set in the config file.

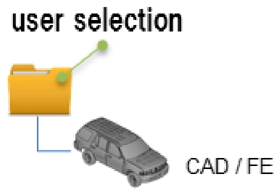


Figure 1501:

5. In the **Angle** field, enter a value between 15 to 30 degrees for feature recognition.
6. In the **Report Path** field, navigate to the directory where the reports generated by the Model Verification tool will be stored.
7. In the **Project Name** field, enter a name for the project.
8. In the **Slide Number** field, enter a starting number of the slides to use in the PowerPoint report.
9. Choose a method for running the model.
   - Choose **Interactive** to run the model in the same HyperMesh session.
   - Choose **Background** to launch a new HyperMesh session in batch mode.
10. The **Processes** drop-down list becomes active when the user selects the Background option and it also depends on the processor.

    The Model Verification tool launches as many hmbatch processes selected.
    - Choose **Check** to execute only the comparison without the report generation.
    - Choose **Report** to generate only the Reports (Check executed previously).
    - Choose **Both** to perform the comparison report generation sequentially.
11. Click **Run** to execute the function.
12. Click **Stop** to stop all the operation.
    This will close front and background HyperMesh sessions.
13. Click **View Report** to display the Summary PowerPoint report.

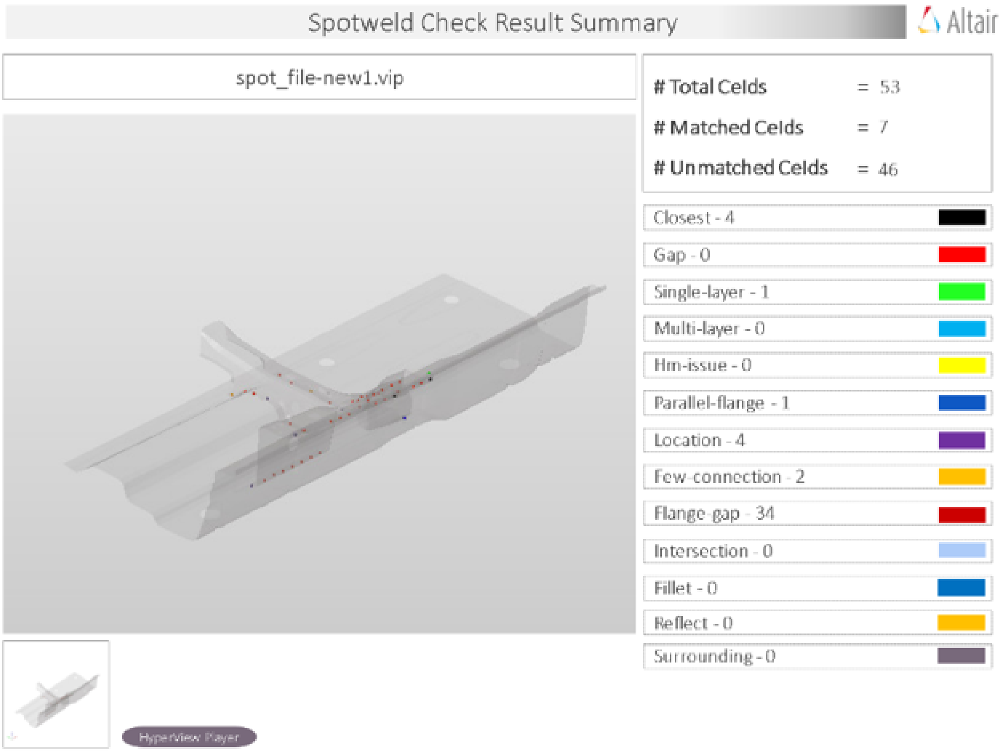


Figure 1502: PowerPoint Summary

(Stored in Report Path)

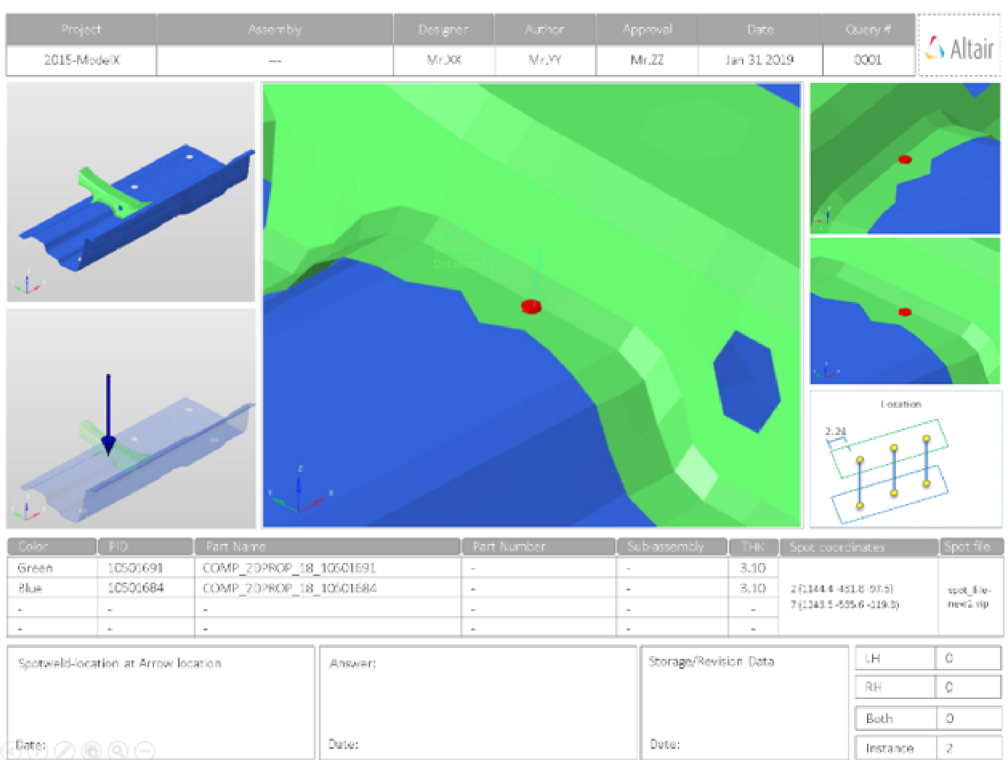


Figure 1503: PowerPoint Detailed Report (Stored in Report Path)

Output

Overview of the PowerPoint report generated from the Spot Weld check.

A PowerPoint report is separated into several sections.

Report Header
: Project name, assembly name, and slide number.

Assembly Picture
: Image of the components that have spot weld issues.

H3D Image
: Section can be viewed in the HyperView Player, and shows an H3D image shows the FE model with the spot weld marked with Cross Hairs.

Issue Details
: Details of components which have spot weld issues.

User Review Section
: Used to add additional information when exchanging information with designers/engineers.

Additional Info
: Occurrences of the issue shown in the current slide. The number of instances show the number of occurrences of the issue.

Supported Spotweld Checks

Overview of the spotweld checks available to perform in the model.

Closest
: Detects the spot welds which are too close to each other. The closeness tolerance is used as the criteria for deciding if two spot welds are considered as too close.

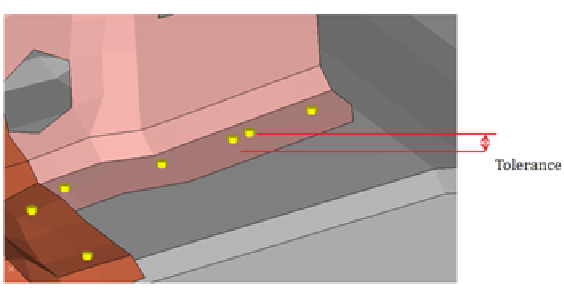


Figure 1504:

Gap
: Detects the spot welds in which the distance between the spot weld location and the connected components lie outside the given maximum and minimum gap limits.

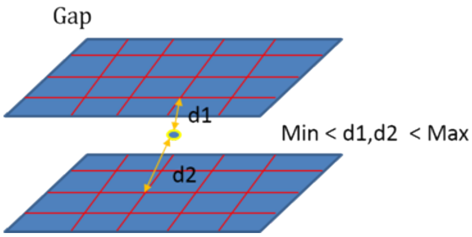


Figure 1505:

Single Layer

Detects the spot welds which are connected to only one part. The second part is missing from the spot weld definition.

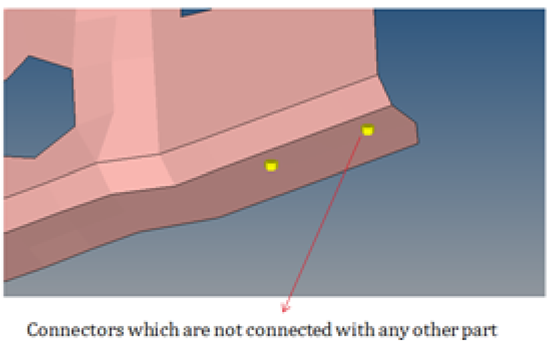


Figure 1506:

Multiple Layers

Detects the spot welds which are connected to more parts than what is specified as the threshold number for multiple parts.

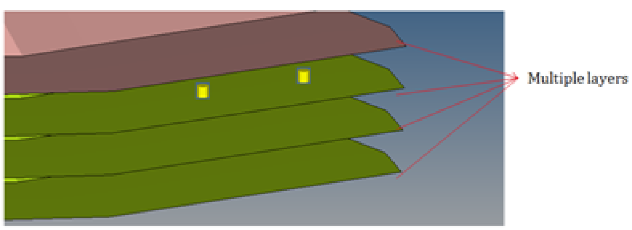


Figure 1507:

HyperMesh Issues

Detects the spot welds that failed during realization.

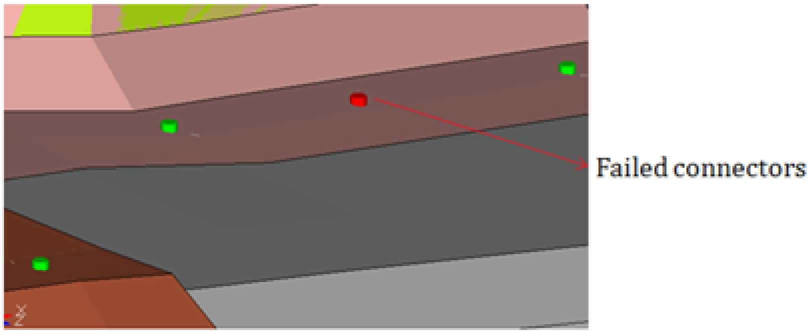


Figure 1508:

Non-Parallel Flanges

Detects the spot welds that are connecting the non-parallel flanges. Flanges are said to be non-parallel if there is a gap or step of the flanges diverge.

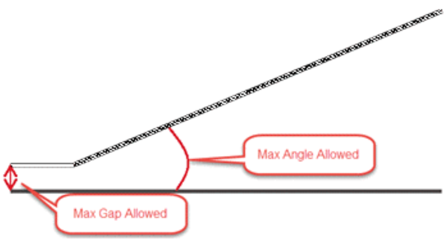


Figure 1509:

Incorrect Location

The Model Verifications creates feature lines internally to check the distance between the connector and the feature line. When the distance between the connector and feature line is less that the specified tolerance, the location of the connector is said to be incorrect. The feature angle used to create feature lines is specified in the **Feature Angle** field.

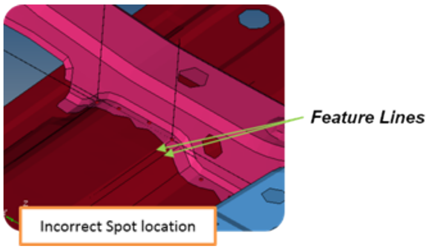


Figure 1510:

Few Connections

Detects the possible locations where the spot welds may be missing. This detection is done based on the adjacent spot welds in that area.

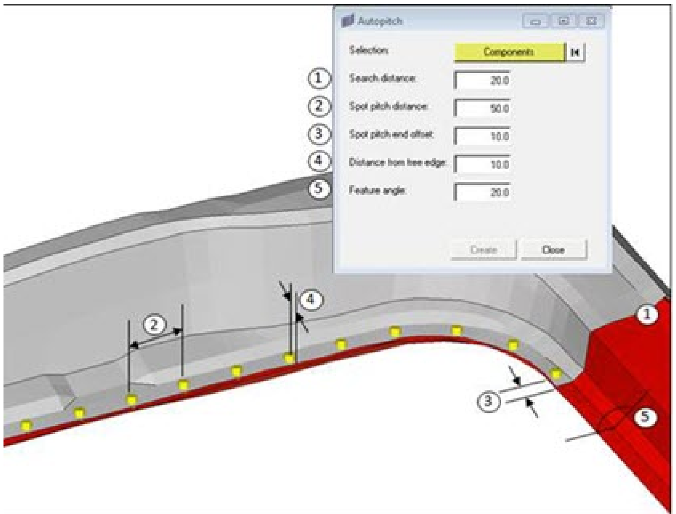


Figure 1511:

Flange Gap

Detects if the gap between the flanges is within the minimum and maximum limit of the average of the component thicknesses being connected by the weld.

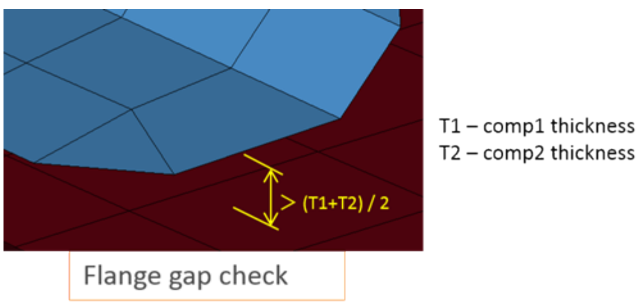


Figure 1512:

Spotweld Intersection

Detects if a spot weld has an intersection with an unconnected component.

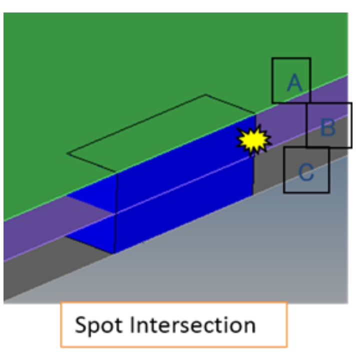


Figure 1513:

Spotweld on Fillet

Detects if any spot weld is located on fillet features.

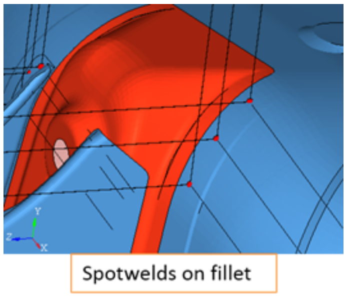


Figure 1514:

Reflect Spotweld

Detects if any spot weld can be realized on reflecting its location about a plane defined by the user.

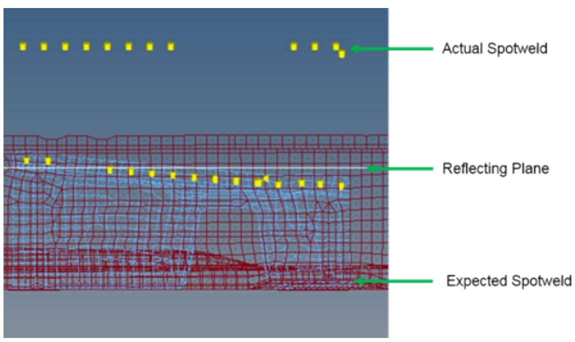


Figure 1515:

Connection

Connection check detects problems with parts/components such as bolts, nuts, clips, plates and reports the problems in a PowerPoint file.

There are multiple checks that can be performed, depending on the selection, the tool perform the checks and results of the checks are written out to the report files.

1. In the Comparison Browser, right-click and select **Check** > **Connection** from the context menu. The **Connection** dialog opens.
2. Select **Import Type** for Base Model.
    - XML (PLMXML or UDMXML)
    - Assembly (CAD assembly files: UG Assembly, CATProduct, JT Assembly...)
    - CAD (Refer Config / CAD files)
    - FE (nastran, pam, radioss, abaqus, optistruct, hyperMesh)
    - Multiple Assembly (Multiple CAD assembly files)
    - Spot Files (Not applicable)

3. Select a datatype.

   Respective data types will be listed.
4. Click the folder icon and navigate to the model to import (Folder/File selections).

   The folder or the selected location must contain the spot weld file. The spot weld file type must be set in the config file.

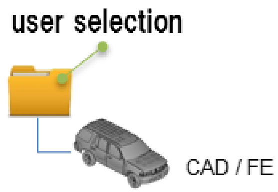
   

   Figure 1516:

5. In the **Report Path** field, navigate to the directory where the reports generated by the Model Verification tool will be stored.
6. In the **Project Name** field, enter a name for the project.
7. In the **Slide Number** field, enter a starting number of the slides to use in the PowerPoint report.
8. Choose a method for running the model.
   - Choose **Interactive** to run the model in the same HyperMesh session.
   - Choose **Background** to launch a new HyperMesh session in batch mode.
9. The **Processes** drop-down list becomes active when the user selects the **Background** option and it also depends on the processor.

   The Model Verification tool launches as many hmbatch processes selected.
   - Choose **Check** to execute only the comparison without the report generation.
   - Choose **Report** to generate only the Reports (Check executed previously).
   - Choose **Both** to perform the comparison report generation sequentially.
10. Click **Run** to execute the function.
11. Click **Stop** to stop all the operation.
    This will close front and background HyperMesh sessions.
12. Click **View Report** to display the Summary PowerPoint report.

Figure 1517: PowerPoint Summary

(Stored in Report Path)

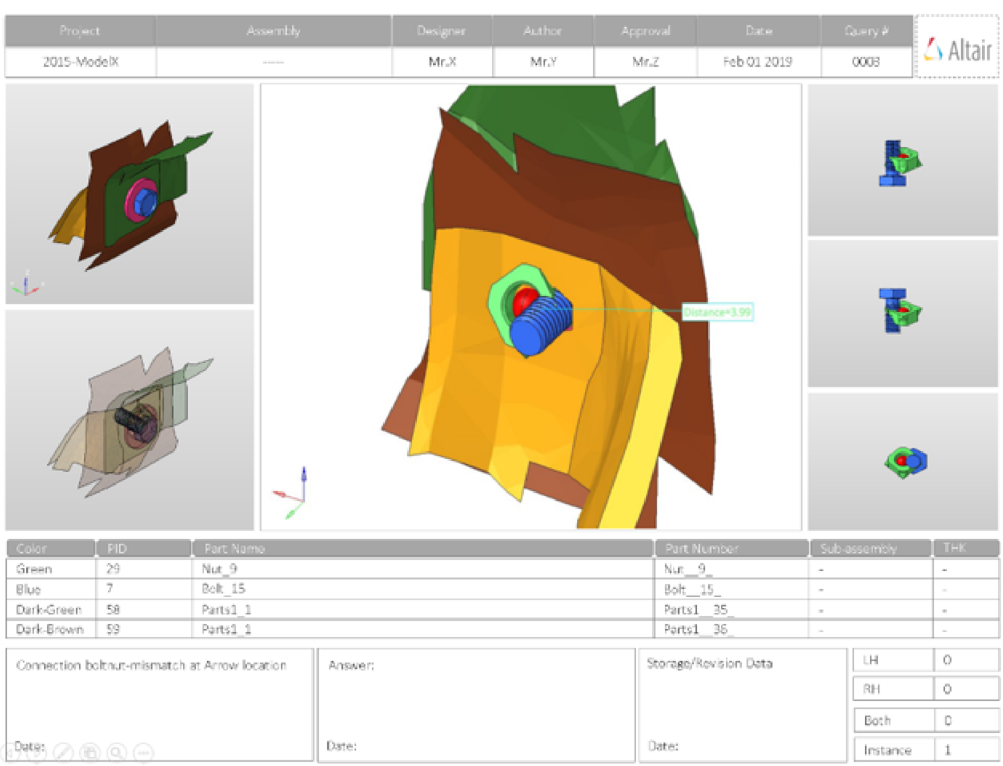


Figure 1518: PowerPoint Detailed Report

(Stored in Report Path)

Output

Overview of the PowerPoint report generated from the Connection check.

A PowerPoint report is separated into several sections.

Report Header
: Project name, assembly name and slide number.

Full Assembly Picture
: Image of the components that have connection issues.

H3D Image
: Section can be viewed in the HyperView Player, and shows an H3D image shows the FE model with the connections marked with Cross Hairs.

Issue Details
: Details of components which have connection issues.

User Review Section
: Used to add additional information when exchanging information with designers/engineers.

Additional Info
: Occurrences of the issue shown in the current slide. The number of instances show the number of occurrences of the issue.

Supported Connection Checks

Types of checks to perform on connections in the model.

Free Hole
: Detects the holes in the model which do not have connectors around them.

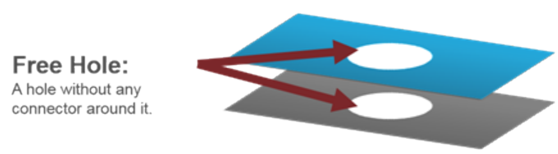


Figure 1519:

Free Bolt-Nut
: Detects the bolts and nuts which do not have corresponding holes to fit in.

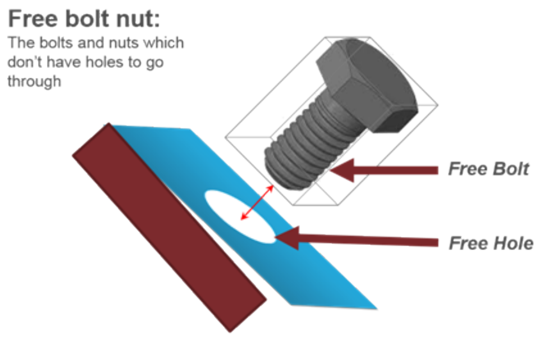


Figure 1520:

Bolt Hole Mismatch

Detects the bolts which have a hole around it, but the distance between the bolt center and hole center is more than the threshold value.

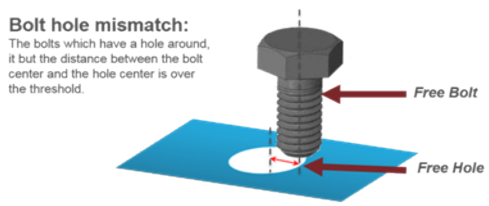


Figure 1521:

Nut Hole Mismatch

Detects the nuts which have a hole around it, but the distance between the nut center and the hole center is over the threshold.

Bolt-Nut Mismatch

Detects the bolt nut pairs whole location or alignment direction do not match properly.

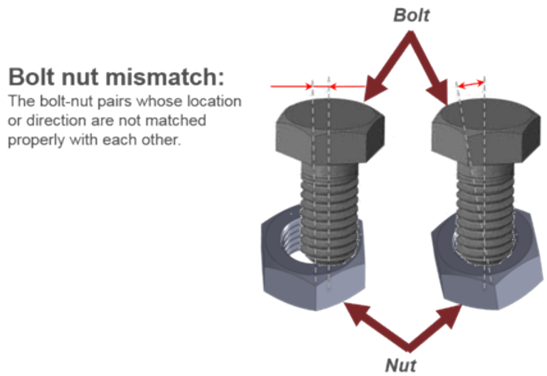


Figure 1522:

Bolt-Nut Size Mismatch

Detects the bolt nut pairs whose sizes are different from each other.

> **Note:** The diameters of bolt, nut and clip are found through a CSV file.

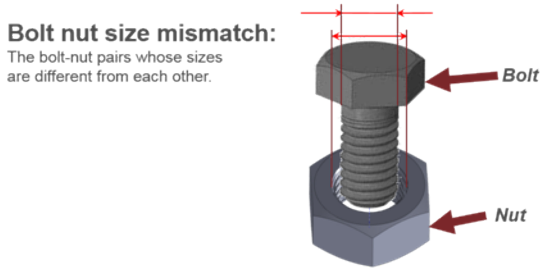


Figure 1523:

Holes Mismatch

Detects pairs of holes whose center axes do not match.

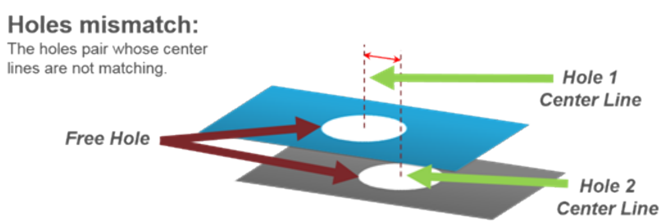


Figure 1524:

Holes Pair Missing

Detects the holes that do not have a corresponding pairing hole within the specified tolerance.

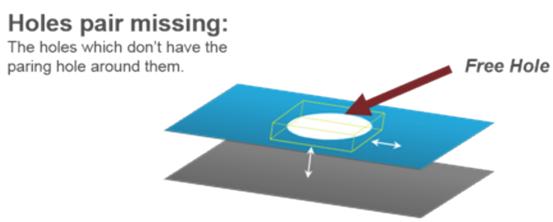


Figure 1525:

Clip Mismatch

Detects the clips which have a hole around it but the distance between the clip center and the hole center is over the threshold.

FreePart

FreePart check detects the components in a model that are not connected to any other component through ID connectors or shared nodes.

1. In the Comparison Browser, right-click and select **Check** > **FreePart** from the context menu.
   The **Freepart** dialog opens.
2. Select **Import Type** for Base Model (Supported types are below).
   - XML (PLMXML or UDMXML)
   - Assembly (CAD assembly files: UG Assembly, CATProduct, JT Assembly…)
   - CAD (Refer Config / CAD files)
   - FE (nastran, pam, radioss, abaqus, optistruct, hyperMesh)
   - Multiple Assembly (Multiple CAD assembly files)
   - Spot Files (Not applicable)
3. Select a datatype.
   Respective data types will be listed.
4. Click the folder icon and navigate to the model to import (Folder/File selections).
   The folder or the selected location must contain the spot weld file. The spot weld file type must be set in the config file.

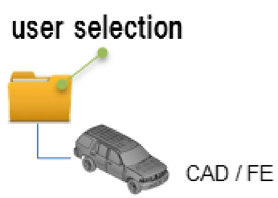


Figure 1526:

5. In the **Gap** field, enter the gap to use between two components to check if they are connected or disconnected.
6. In the **Report Path** field, navigate to the directory where the reports generated by the Model Verification tool will be stored.
7. In the **Project Name** field, enter a name for the project.
8. In the **Slide Number** field, enter a starting number of the slides to use in the PowerPoint report.
9. Choose a method for running the model.
   - Choose **Interactive** to run the model in the same HyperMesh session.
   - Choose **Background** to launch a new HyperMesh session in batch mode.
10. The **Processes** drop-down list becomes active when the user selects the Background option and it also depends on the processor.

The Model Verification tool launches as many hmbatch processes selected.

- Choose **Check** to execute only the comparison without the report generation.
- Choose **Report** to generate only the Reports (Check executed previously).
- Choose **Both** to perform the comparison report generation sequentially.

11. Click **Run** to execute the function.

12. Click **Stop** to stop all the operation.
This will close front and background HyperMesh sessions.

13. Click **View Report** to display the Summary PowerPoint report.

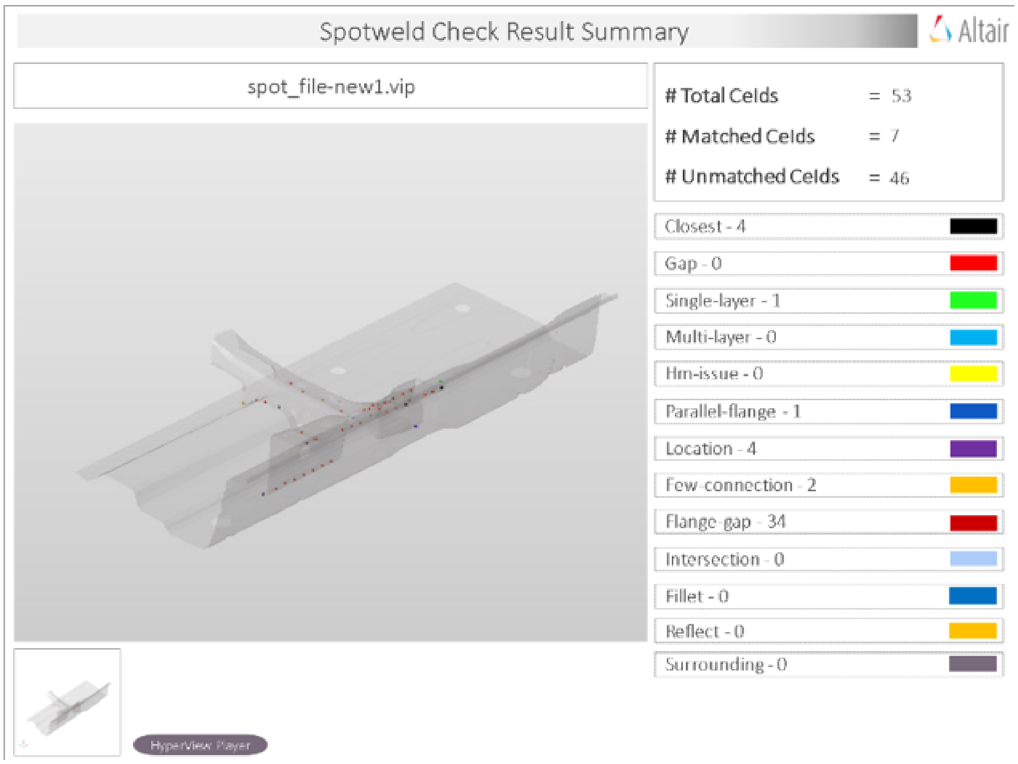


Figure 1527: PowerPoint Summary and Detailed Report
(Stored in Report Path)

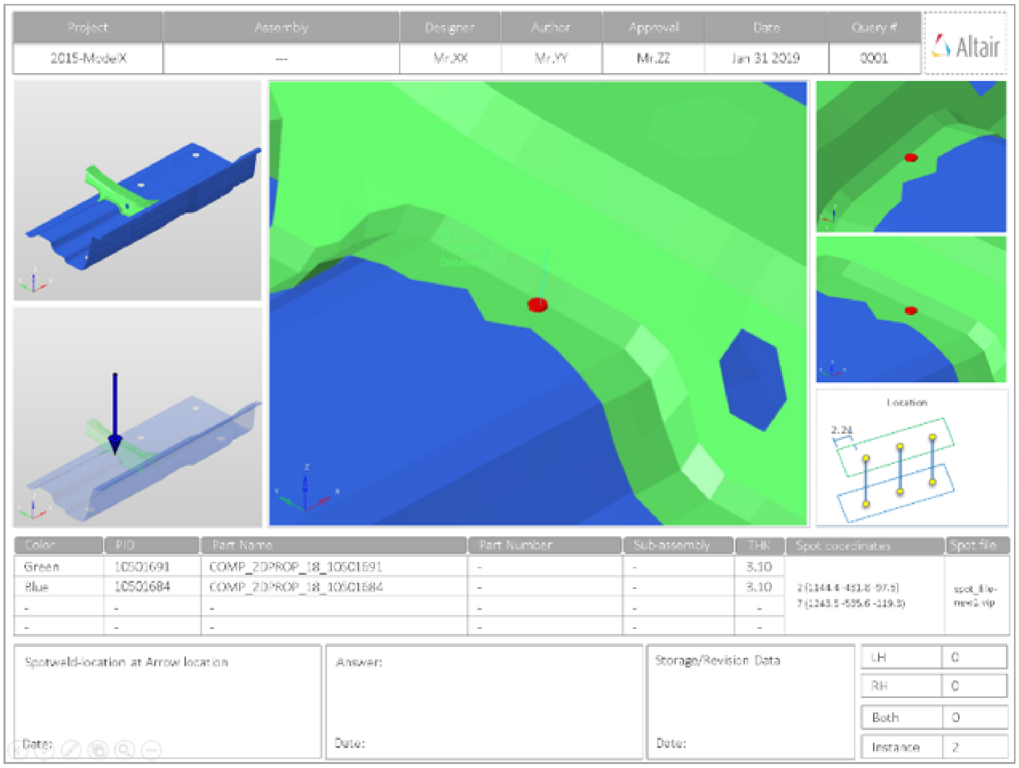


Figure 1528: PowerPoint Detailed Report

(Stored in Report Path)

Output

Overview of the PowerPoint report generated from the FreePart check.

A PowerPoint report is separated into several sections.

Report Header

Project name, assembly name, and slide number.

Assembly Picture

Image of the components that have free part issues.

H3D Image

Section can be viewed in the HyperView Player, and shows an H3D image shows the FE model with the free parts marked with Cross Hairs.

Free Part Details

Details of components which have free part issues.

User Review Section

Used to add additional information when exchanging information with designers/engineers.

Spot Comparison

Spot Comparison check compares two spot connector files using the Model Verification tool.

Supported Model Verification formats include:

1. In the Comparison Browser, right-click **Check** > **Spot Comparison** from the context menu.
   The **Spotcompare** dialog opens.
2. Select **Import Type** for Base Model.
   - XML (PLMXML or UDMXML)
   - Assembly (CAD assembly files: UG Assembly, CATProduct, JT Assembly…)
   - CAD (Refer Config / CAD files)
   - FE (nastran, pam, radioss, abaqus, optistruct, hyperMesh)
   - Spot Files (`.vip`, `.mwf`, `.mcf`, and `.xml`)
3. Select a datatype.

   Respective data types will be listed.
4. Click on the folder icon and navigate to the model to import (Folder/File selections).

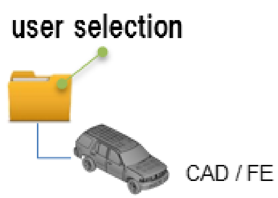


Figure 1529:

5. Select Import Type for Variant Model.
   - XML (PLMXML or UDMXML)
   - Assembly (CAD assembly files: UG Assembly, CATProduct, JT Assembly…)
   - CAD (Refer Config / CAD files)
   - FE (nastran, pam, radioss, abaqus, optistruct, hyperMesh)
   - Spot Files (`.vip`, `.mwf`, `.mcf`, and `.xml`)
6. Select a datatype.

   Respective data types will be listed.
7. Click on the folder icon and navigate to the model to import (Folder/File selections).

8. In the **Report Path**, navigate to the directory where the reports generated by the Model Verification tool will be stored.
9. In the **Project Name** field, enter a name for the project, if applicable.

10. In the **Slide Number** field, enter a starting number of the slides to use in the PowerPoint report, if applicable.

11. Choose a method for running the model.

- Choose **Interactive** to run the model in the same HyperMesh session.
- Choose **Background** to launch a new HyperMesh session in batch mode.

12. The **Processes** drop-down list becomes active when the user selects the Background option and it also depends on the processor.

The Model Verification tool launches as many hmbatch processes selected.

- Choose **Check** to execute only the comparison without the report generation.
- Choose **Report** to generate only the Reports (Check executed previously).
- Choose **Both** to perform the comparison report generation sequentially.

13. Click **Run** to execute the function.

14. Click **Stop** to stop all the operation.
This will close front and background HyperMesh sessions.

15. Click **View Report** to display the Summary PowerPoint report.

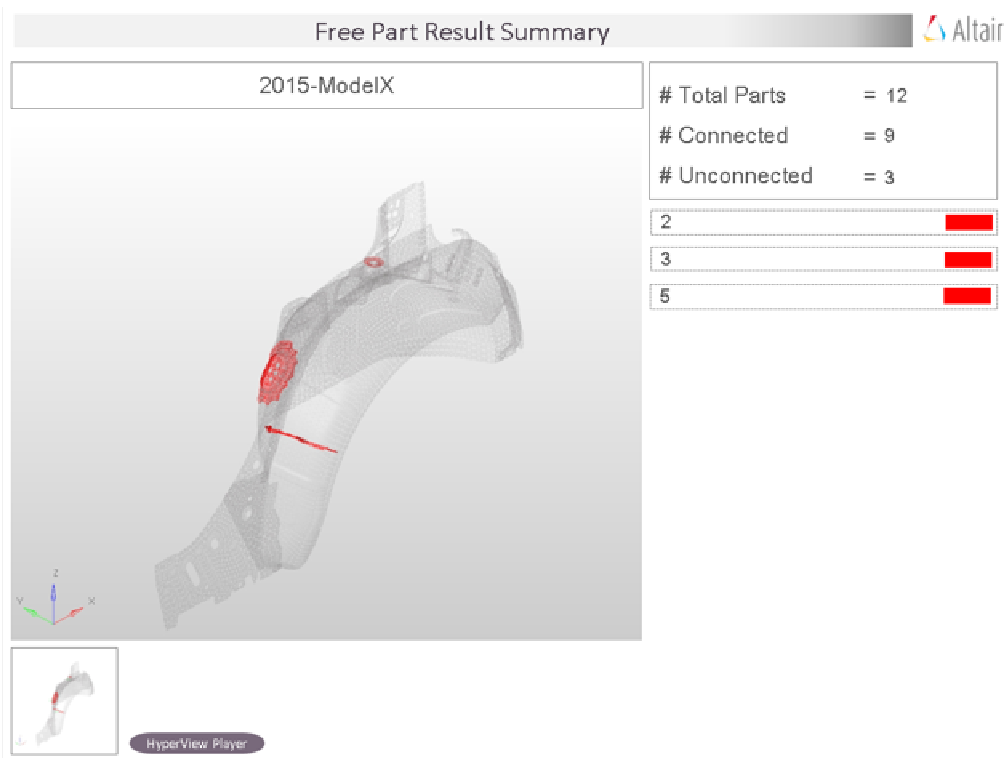


Figure 1530: PowerPoint Summary Report

(Stored in Report Path)

CSV-Comparison

CSV comparison tool compares CAD/FE attributes against CSV file column.

Supported attributes are Part name, PID, Part Number, Material Name, Thickness, File name. It has 3 options, "compare", "compare-update" and create "new XML".

1. In the Comparison Browser, right-click on **CSV Comparison** from the context menu.
2. Click on **Compare**.
   The **File Selection** window opens.
3. CSV file must be selected.
   Check will proceed.

   It generates reports showing the updated information in Excel report same as comparison.

Multiple Checks

Multiple Checks perform multiple checks using single interface.

If Offset check is ON, CAD data will be offset using thickness value. Offset CAD data will be used to check all other checks. Check and Reports are sequentially executed for each check. Offset, Intersection, Spotweld, Connection, Free Part check uses Base Model selection alone and ignore Variant model selections. Comparison function uses both Base Model and Variant Model selections. Check items, input rules are as per individual checks explained in respective check functions.

1. In the Comparison Browser, right-click on **Check** > **Multiple_Checks** from the context menu.
   The **Model Verification** dialog opens.
2. Upload the base model using the Base Model options.
   a) Select an import type.
   b) Select a data type.
   c) Navigate to the model to import.
3. Upload the variant model using the Variant Model(s) options.
   a) Select an import type.
   b) Select a data type.
   c) Navigate to the model to import.

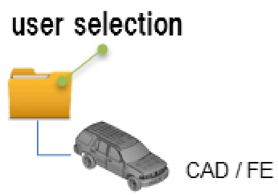


Figure 1531:

4. Select the type of checks to perform, and define additional settings as needed.
5. In the **Report Path**, navigate to the directory where the reports generated by the Model Verification tool will be stored.
6. In the **Project Name** field, enter a name for the project.
7. In the **Slide Number** field, enter a starting number of the slides to use in the PowerPoint report.

   For example, if you enter a value of `1` for the slide number, then the slide numbers in the report starts from 0001.
8. Choose a method for running the model.
   - Choose **Interactive** to run the model in the same HyperMesh session.
   - Choose **Background** to launch a new HyperMesh session in batch mode and perform the operation. The Background run mode is faster than the Interactive mode and also allows you to utilize multiple processors that are available on the machine to complete the operation faster. Each pair of models are divided and handled in multiple HyperMesh sessions depending on the number of processes selected.
9. For Action, choose to an operation to perform.
   - Choose **Check** to only perform a check.
   - Choose **Report** to generate a PowerPoint report.
   - Choose **Both** to perform a check and generate a report.
10. Click **Run**.

Configure Model Verification Settings

Overview of various settings that can be configured for Model Verification.

1. In the Comparison/Intersection browser, right-click and select **Configuration** from the context menu.
   The **Configuration** dialog opens.
2. Configure settings as needed.
3. Click **OK** to save and load the new settings in current session.

log

Specify default values for Model Verification log file settings.

Settings

type
: Type of log information output.
  Default: file or channel

destination
: Log information file name.
  Default: stdout, path , stderr

level
: Level of messages to be written in the log file.
  Extension: debug, info, notice, warn, error, critical, alert or emergency

Remove Log Information

Right-click on **log** and select **-** > **Cut Item**

Display Log Information

Right-click on **log** and select **Add Item**

cad

Specify defaults for CAD model import - file extensions.

Settings

type
: Supported CAD file format.
  Default allowed: catia, iges, jt, parasolid, proe, step, ug or hypermesh

catia
: File extension of CATIA children file, user, can add many extensions for this type.

extension

CATPart

iges

File extension of IGES children file.

extension

`.igs` or `.iges`

jt

File extension of JT children file.

extension

`.jt`

parasolid

File extension of Parasolid children file.

extension

`.x_t`, `.x_b` or `.xml_txt`

proe

File Extension of Creo children file.

extension

`.prt`

ug

File extension of UG children file

extension

`.prt`

step

File extension of STEP children file.

extension

`.step` or `.stp`

hypermeshcad

File extension of HyperMesh children file.

extension

`.hm`

Import Settings

readMultipleParts

Single CAD file will be the input and each part inside the file will be physically separated and exported to create part structure.
Default/Allowed: ON or OFF

readmetadata

Reads metadata from the IGS file and displays those information like PID, Name in the Parts attribute column.
Default/Allowed: ON or OFF

readFileHeader

Reads information stored in the IGS file header and displays the information in the Part attribute column.
Default/Allowed: ON or OFF

readTransformation

Takes care of reading and managing transformation or part position information. It also takes care of the parts position while CAD offset. Parts will not be in the correct position, if this option is OFF, while executing surface offset.
Default/Allowed: ON or OFF

thickness-update

If this option is ON, thickness value from Offset function's GUI will be used as standard thickness for all parts during surface offset. This is done to assign clearance between parts while checking Intersection.
Default/Allowed: ON or OFF

fe

Specify defaults for FE model solver types, version and file extensions.

Settings

type

Supported CAD file format.
Default/Allowed: dyna, nastran, pam, radioss, abaqus, optistruct, connector or hypermesh.

dyna

File extension of LS-DYNA input file, user can add many extensions for this type.

version

971

extension

`.k`, `.key`, `.dat`, `.dyn*`, or `.dynain`.

nastran

File extension of Nastran input file, user can add many extensions for this type.

extension

`.bdf`, `.bulk`, `.blk`, `.dat`, `.nas` or `.nastran`

pam

File extension of PAM-CRASH input file, user can add many extensions for this type.

version

2015

extension

`.pc` or `.dat`

radioss

File extension of Radioss input file, user can add many extensions for this type.

version

120

extension

`.d00`, `.d01`, `.rad` or `._000.d01`

abaqus

File extension of Abaqus input file, user can add many extensions for this type.

version

Explicit, Standard.2d or Standard.3d

extension

`.imp` or `.pes`

optistruct

File extension of OptiStruct input file, user can add many extensions for this type.

extension

`.fem`

connector

File extension of Connector input file used for Spot weld check and Spot comparison function. If the type specified here and the actual file extension are not matching, the Check will not be executed.

version

971

extension

`.vip`, `.mcf` (HM format), `.mwf`, `.xml` (HM format)

hypermeshcad

File extension of HyperMesh file contains only Mesh.

extension

`.hm`

Import Settings

fe-attached

Parts with node sharing are organized to one part if the thickness of these parts are not same. The organization will not be executed if the thickness of these parts are same.
Default/Allowed: ON or OFF

fe-dettach

LH and RH parts will be separated two separated physical part incase these two parts are organized in to one part.
Default/Allowed: ON or OFF

importby

While importing fe data in to HyperMesh creating components by reading HyperMesh Comments or by reading HyperMesh information managed in HyperMesh file.
Default/Allowed: hmcomments or property

Include-stamping

Reads stamping results information like Element shell thickness from the solver deck. This information will be used for mass calculation. Property thickness will be ignoredm if this option is ON.
Default/Allowed: ON or OFF

dummy-thickness

Assigns a dummy thickness for the FE data in case the thickness no thickness or property assigned to the part. This dummy thickness is needed for CAD solid vs FE comparison in case no thickness exist from the original FE.
Default/Allowed: ON or OFF.

general

General defaults for the Model Verification instance.

GUI Settings

project

Project name displayed in PowerPoint report
Default/Allowed: 2015-ModelX

action

Type or Run to be selected.
Default/Allowed: check, report or both

auto-close

After the check and report is complete the GUI will be closed automatically if this is ON.

Default/Allowed: 0 or 1

number-of-processes

Number of parallel processing to be executed. It is advised to keep 2 CPUs for System purpose.
Default/Allowed: 1 to 32

number-of-processes-max

Maximum number of parallel processor allowed (Total # CPUs – 2).
Default/Allowed: 2 to 32

assembly-level

This value indicates the number of level above the part to be considered as Assembly Name. Assembly name is used in Intersection, Comparison, Reports. Parts will be skipped if the base, variant assembly names are not same during Comparison.
Default/Allowed: 0 - 10

metalTrim-level

This value indicates the number of level above the part to be considered as Sheet Metal or TRIM Name.

METAL-METAL

All the parts inside the METAL folder will be checked.

TRIM-TRIM

All the parts inside the TRIM folder will be checked.

METAL-TRIM

Parts across METAL and TRIM folder will be checked. No check will be executed inside the METAL or TRIM folders.

Default/Allowed: 0 - 10

port

This value to be edited incase of specific port is needed for Thread communication.
Default/Allowed: 0

timezone

Not valid.

deleteLogFiles

Deletes all of the scratch files after the report generation. All of the XML, HM, Image files will remain if this is OFF.
Default/Allowed: ON or OFF

maxloopcount

Maximum number of recovery limits in case of crash or application hanging.
Default/Allowed: 2 - 50

wait-time

Maximum wait time for a process to wait for a check per part or pair of part.
Default/Allowed: 400 - 3200 (seconds)

progress-time-scale

Progress bar interval, do not reduce but can increase the value.
Default/Allowed: 2000

configa-application

File extension of Parasolid children file.
Default/Allowed: `C:/windows/System32/notpad.exe`

Model / Preview / Import-type

When preview huge CAD data, the CAD option allow to read the CAD as it is. The Facet option reads CAD by converting to Facets, this is done for the performance and visualization purpose.
Default/Allowed: CAD or Facets

Representation Settings

reptype-name / CAD

Allowable representation type for original CAD data read. Do not change.
Default/Allowed: common

reptype-name / Common Midsurface

Allowable representation type for Midsurface data. Do not change.
Default/Allowed: common Midsurface

reptype-name / FE

Allowable representation type for FEM data. Do not change.
Default/Allowed: Mesh FE

Solid Recognition

model / close-shell / free-edge-allowance

Allowable number of free edges, if actual number of free edges in the CAD is less than this value is solid CAD data.
Default/Allowed: 1 to 4

model / close-shell / mesh / free-edge-allowance

Allowable distance between two nodes to be automatically equivalenced or stitched.
Default/Allowed: 0.01 to 0.1

auto-rename

tool / auto-rename

CAD Name will be assign to Part Name of the browser by considering below string pattern.
Default/Allowed: ON or OFF

auto-renumber

tool / auto-renumber

CAD Name will be assign to PID of the browser by considering below string pattern. By default the IDs will be renumbered from 1. This must be turned OFF if the PID and Part Names are managed in CAD attributes.
Default/Allowed: ON or OFF

type

Part name subexpression along with PID value will be assigned.
Default/Allowed: $sub() $pid

start-with

Part name subexpression along with PID value will be assigned.
Default/Allowed: $sub() $pid

increment

Part name subexpression along with PID value will be assigned.
Default/Allowed: $sub() $pid

rename / part-id / pattern

Decimal value, except excluding first 0 will be read from file name and assigned as PID.
Default/Allowed: 0*(\d+)

rename / part-id / format

Regular subexpression will be used.
Default/Allowed: $sub(0)

rename / part-name / pattern

String value will be read from file name and assigned as PartName.
Default/Allowed: (\w+)

rename / part-name / format

Part name subexpression along with PID value will be assigned.
Default/Allowed: $sub(0) $pid

pName To PID map

Below Letter to Integer table is used while renumbering the PID if ON. The normal auto-renumber will be applied if OFF.
Default/Allowed: ON or OFF

Letter to integer map

Mapping table to change string value to decimal value for the PID Column.
Default/Allowed: Alphabet(A-Z): Decimal value (1-1000)

String-length

Number of digits to be considered for the Renumber using mapping table.
Default/Allowed: 5 to 8

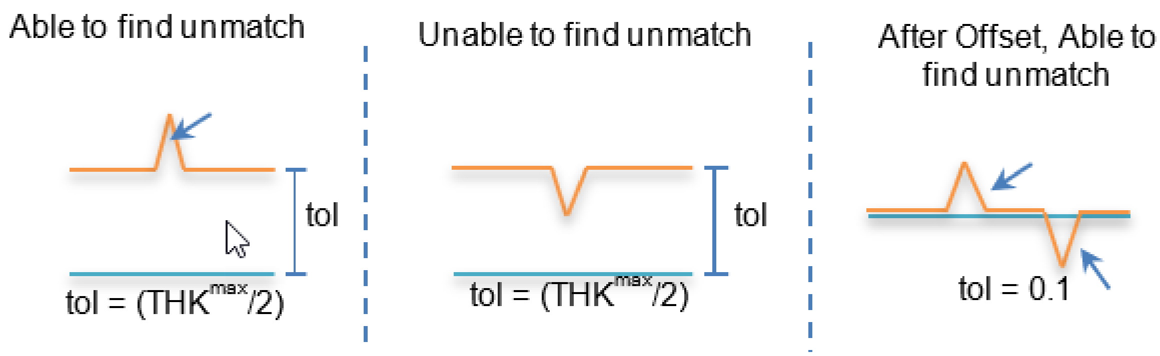
comparisonunified

Defaults for the offset functionality of Model Verification.

GUI Default Settings

tolerance

Search tolerance for searching nearest parts as well as search tolerance for finding matching entities.



In case of Solid CAD vs FE, FE data must have correct thickness on property and the tolerance can be from 0.1 to 0.5mm. In case of sheet metal parts, the recommended tolerance is 1.0mm.
Default/Allowed: 0.1 to 5.0 mm

threshold

Cut off percentage for judging matched or unmatched parts. Too low value like 5% are not recommended while checking huge number of parts.
Default/Allowed: 15 to 30%

base-directory

Initial director for Base model.
Default/Allowed: `C:/temp`

base-filetype

Input model types.

XML

BOM file or UDMXML file saved from HyperMesh.

CAD

CAD geometry,1 part/file (folder contains multiple CAD).

FE

Solver Deck file (single file).

Assembly

CAD Assembly file (CatProduct/JT Assembly).

Multiple Assembly

Multiple Assembly files.

Spot File

Spot weld files (`.vip`, `.mwf`, `.mcf`, or `.xml`).

Default/Allowed: XML/CAD/FE/Assembly/../Spot File

variant-directory

Initial director for Variant model.
Default/Allowed: `C:\temp`

variant-filetype

Input model types.

XML

BOM file or UDMXML file saved from HyperMesh.

CAD

CAD geometry,1 part/file (folder contains multiple CAD).

FE

Solver Deck file (single file).

Assembly

CAD Assembly file (CatProduct/JT Assembly).

Multiple Assembly

Multiple Assembly files.

Spot File

Spot weld files (`.vip`, `.mwf`, `.mcf`, or `.xml`).

Default/Allowed: XML/CAD/FE/Assembly/../Spot File

report

Initial director for report out path.

Default/Allowed: `C:/temp`

report option

Output report formats. After check is completed, user can change this option to generate report any number of times. Excel report is faster than PowerPoint report.

Default/Allowed: Excel, PowerPoint or both

scope

Output result types.

Matched

30.0 (threshold) to 100% matched parts

Unmatched

0.0 to 30% matched parts

Both

0.0 to 100% parts

Default/Allowed: matched, unmatched or both

imageType

Output image types.

.jpg

Light weight 2D

Default/Allowed: `.h3d` or `.jpg`

mode

Default radio button options for Run type.

interactive

Checks will be run in the front ground HyperMesh session.

background

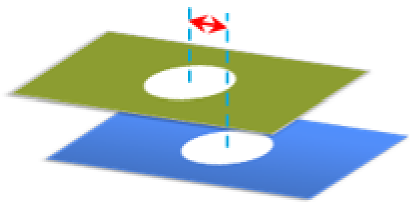
Checks will be executed in the background HyperMesh sessions, if error occurs recovery will be executed, the errors will be displayed in the browser as "Crash" key word.

Default/Allowed: interactive or background

Comparison Settings

Check elems in Surf Hole

CAD holes vs FE holes are compared, displays if any mismatches in the holes via comparison report.



Check

Compares FE holes vs surface holes or surface holes vs FE holes.

Default/Allowed: ON or OFF

Min Dia of Hole

The minimum hole diameter for checking the holes mismatch. Big diameter holes are taken care by default.

Default/Allowed: 0.5 to 5.0

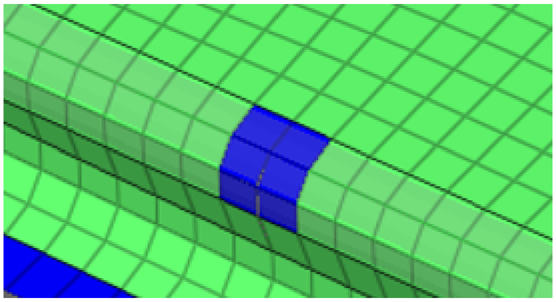
Max Dia of Hole

The maximum hole diameter for checking the holes mismatch.

Default/Allowed: 5 to 20.0

free-edge-check

Compares free edge of FE data against free edges of CAD surface boundary lines, mismatch will be displayed in the comparison report.



Default/Allowed: ON or OFF

free-edge-tolerance

Free edges in FE data less than this tolerance will be ignored.

Default/Allowed: 0.0. to 0.1mm

find-cad-thickness

Calculates thickness from cad by using Volume and Area of the CAD, the Thickness column will display the results. This calculated thickness value will be given preference over GUI tolerance for judging the match percentage.
Default/Allowed: ON or OFF

use-fe-thickness

Thickness from FE data be given preference over GUI tolerance for judging the match percentage.
Default/Allowed: ON or OFF

delete-fefiles

Deletes the temporary FE folder created my MVD during the CAD-FE comparison. Once this folder is deleted, Result review may not be possible.
Default/Allowed: ON or OFF

bomcomparison

Compares PID, MATERIAL, THICKNESS, WEIGHT values, except the shape comparison.
With bomcomparison=ON, and same-id-only=ON options, BOM comparison is executed, PowerPoint and Excel Reports are generated.
Default/Allowed: ON or OFF

contactcomparison

Comparison of contact cards for the parts, it checks for card names and number of contact cards. Limited to Radioss, Nastran, PAM-CRASH, LS-DYNA.
Default/Allowed: ON or OFF

loadcomparison

Comparison of Load collector cards for the parts, it checks for card names and number of contact cards. Limited to Radioss, Nastran, PAM-CRASH, LS-DYNA.
Default/Allowed: ON or OFF

remove-pinhold

Removes small diameter in holes on CAD surface, applicable only on CAD-FE comparison.
Default/Allowed: ON or OFF

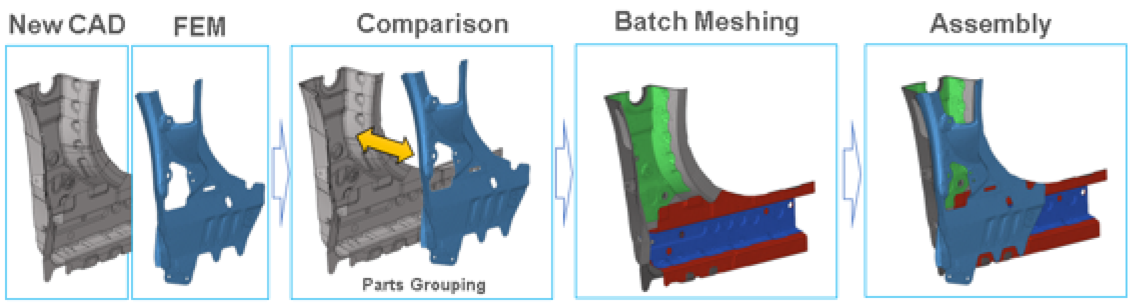
pinhold-radius

Pin hole radius, holes less than this value will be physically removed while comparison.
Default/Allowed: 0.1 to 5.0

modeassembly

After comparison for unmatched parts below, process is automatically executed.

- Group parts based on match% and report threshold%
- Execute Batchmesher
- Replace CAD with Batch Meshed FEs
- Assign material and property
- Load assembly (old + new mesh) import



- Parts Group (Part Browser / Part Set View)

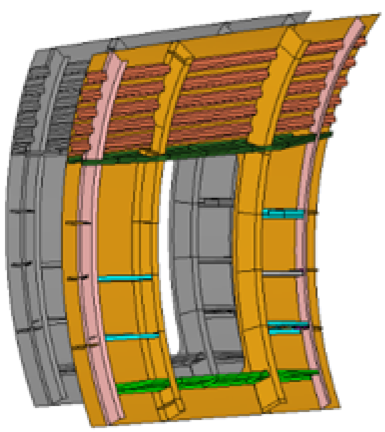
Default/Allowed: ON or OFF

loopLimit

Each time two parts will be compared if the value is 0. # of parts are loaded at once in one HyperMesh session and comparison will be executed, if the value is more than 0.
Default/Allowed: 0 to 1000

samelocation

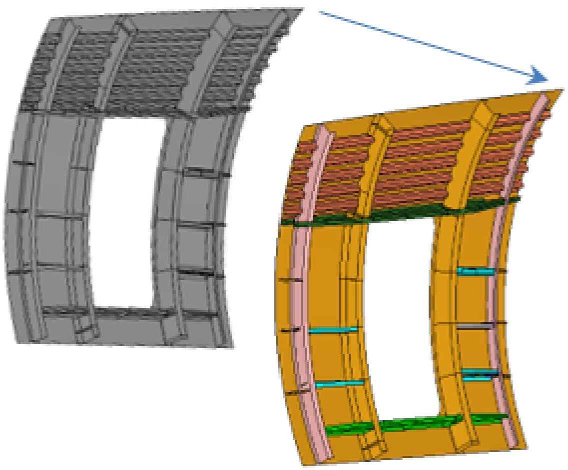
Parts will be searched at same location of base part, along with the tolerance value. Parts far from each other will be ignored. Works fine with small offset between both parts.



Default/Allowed: ON or OFF

translation

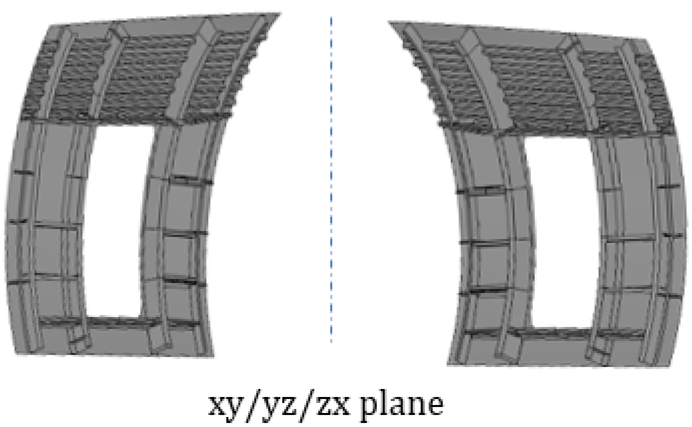
Parts will be searched at same location of base part, if the same shape part is not found, the variant part is translated, then translated comparison will be executed. Translation distance will be automatically calculated.



Default/Allowed: ON or OFF

symmetry

Parts will be searched at same location of base part, if the same shape part is not found variant part is reflected, then symmetry comparison will be executed. Symmetry plane will be calculated by parameter below.



Default/Allowed: ON or OFF

symmetryPlane

Plane to be considered for symmetryPlane comparison.
Default/Allowed: xy, yz or zx

Filters

cgtolerance-distance

Distance between centroid of base mode part and variant model part. Parts will not be compared, if the parts centroid are more than this value. Recommended value is around 200mm.
Default/Allowed: 10 to 1000

min-areapercent

Minimum area ratio of base and variant parts. Parts will not be compared, if the parts ratio is less than this value. Recommended value is around 60%.
Default/Allowed: 1 to 60%

max-areapercent

Maximum area ratio of base and variant parts. Parts will not be compared, if the parts ratio is more than this value. Recommended value is around 140%.
Default/Allowed: 101 to 140%

same-pid-only

Parts with same PID will be compared and Parts with different PIDs will be ignored, if the value is ON. There is not much gain in the performance, but 1% match will be reported, in case this option is ON. This must be turned OFF in case not all the parts PID are NOT matching.
Default/Allowed: ON or OFF

same-partname-only

Parts with same PartName will be compared and Parts with different PartName will be ignored, if the value is ON. This must be turned OFF in case not all the parts Name are NOT matching.
Default/Allowed: ON or OFF

compare-midsurf

Solid CAD vs Mid Mesh comparison will happen if this value is ON. This will affect only CAD-FE only when CAD is solid or closed volume surface.
Default/Allowed: ON or OFF

seamtreatment

Seam nodes are excluded from the unmatched percentage calculation. This must be turned OFF in case Seam welds are not used in FE model.
Default/Allowed: ON or OFF

bbox-view

Bounding boxes for each part are displayed in the HyperMesh graphics for reference after the check and report is done. User can check if both of CAD or FE is at same place or same size.
Default/Allowed: ON or OFF

Representation Comparison

Check

Single BOM comparison will be executed, if this parameter is ON, inside single part multiple representations will be compared and reports generated. Two model comparisons will be executed, if this parameter is OFF.
Default/Allowed: ON or OFF

base-reptypes

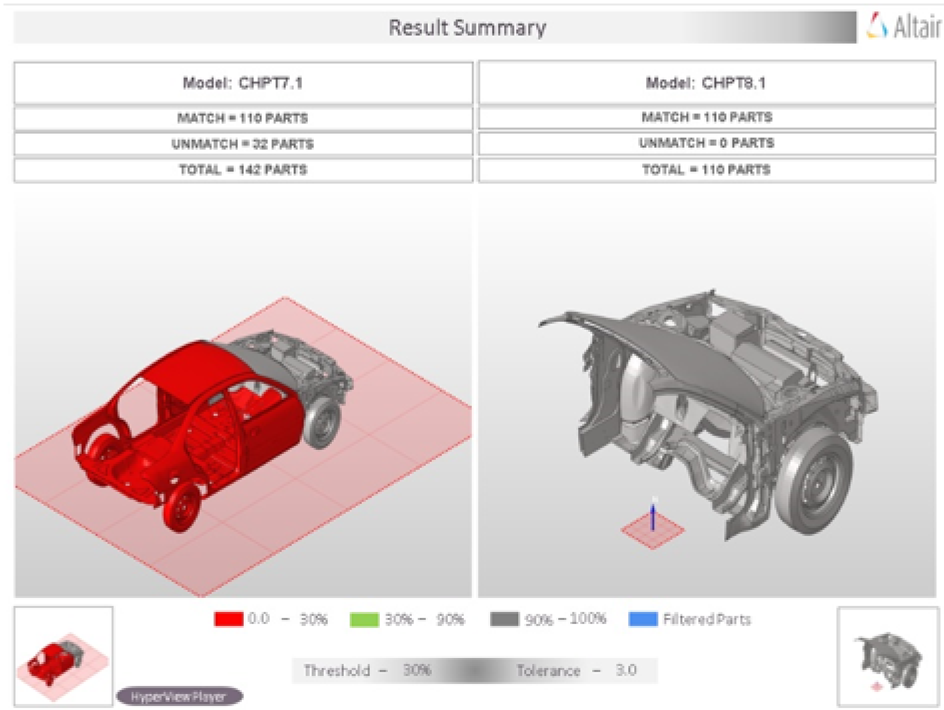
Representations available in the Part Browser. CAD represents the Original CAD data, Mesh represents the FE data created by Batchmesher.
Default/Allowed: CAD, Mesh or Common

Variant-reptypes

Representations available in the Part Browser. CAD represents the Original CAD data, Mesh represents the FE data created by Batchmesher. Usually this can be Mesh or Common.

Default/Allowed: CAD, Mesh or Common

Rep Comparison Output (stored in Report folder



Match-area

cad-fe, cad-cad, fe-fe

Matched

Matched results will be calculated only if the entities are at exact location.

matched-overlapped

Matched results will be the sum of matched and Overlapped entities.

matched-overlap-intersected

Matched results will be the sum of matched, Overlapped and Intersected entities.

Default/Allowed: matched, matched-overlapped, or matched-overlapped-intersected

match-value

Match % output type.

Default/Allowed: source, target or average

round-off

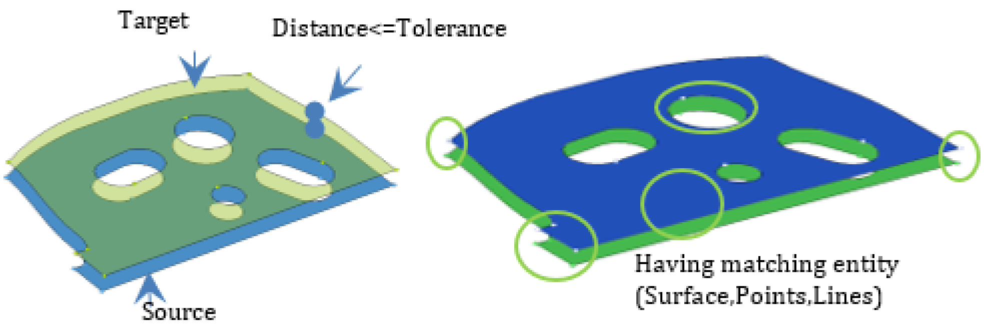
Match % display format.

Default/Allowed: up, down, nearest, or %.2f

Details on the Calculation Method

Matched

Occurs when a source or target surface is within the given tolerance of a compared surface using a direct surface to surface comparison. All points and lines comprising each surface must match between the surfaces. Each matched surface is placed in a separate match type group with the surface it matches.

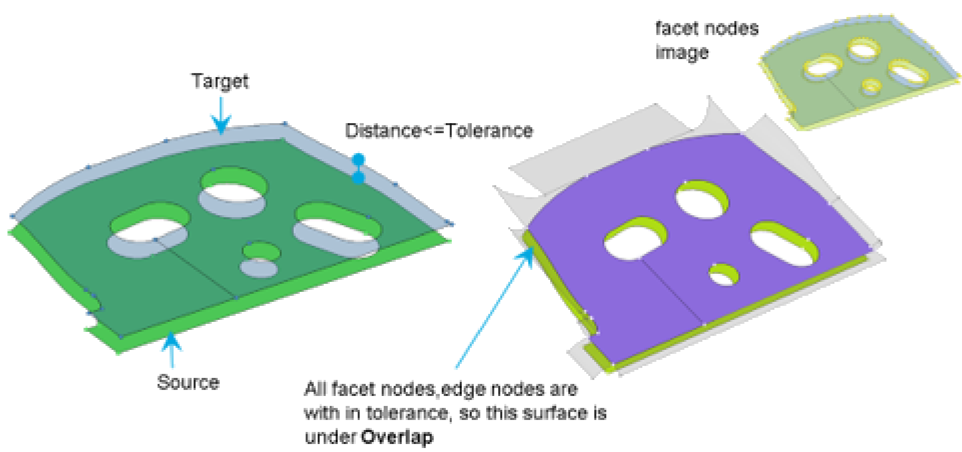


Unmatched

Occurs for surfaces and elements when the matched criteria are not met.

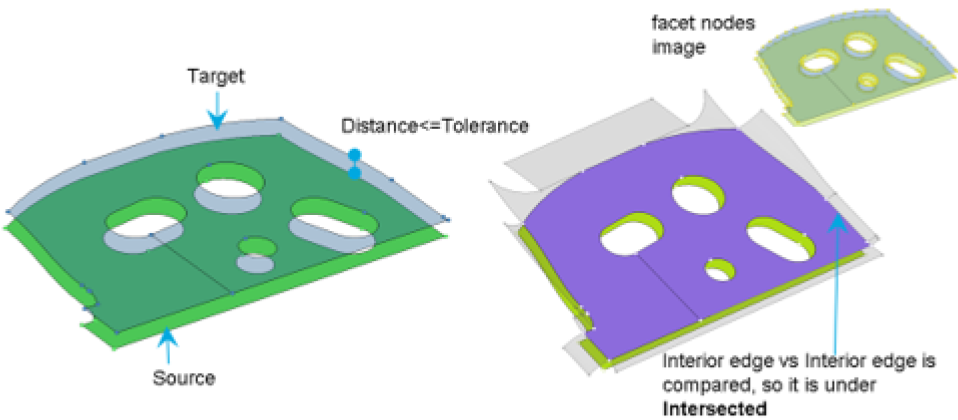
Overlapped

Occurs when all facet nodes of a source or target surface are within the given tolerance of the compared surfaces and all of the facet nodes on the nearest compared surfaces' exterior edges are within the given tolerance of the source or target surface's exterior edges.



Intersected

Occurs when at least one but not all facet nodes on a source or target surface are within the given tolerance of the compared surfaces or at least one of the facet nodes on the nearest compared surfaces' exterior edges is outside the given tolerance of the source or target surface's exterior edges.



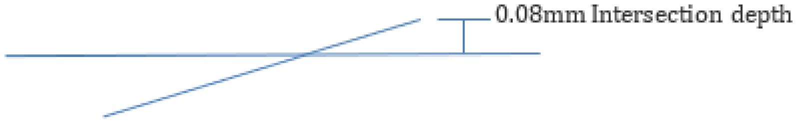
intersection

Configurations for the intersection check functionality.

GUI Default Settings

tolerance

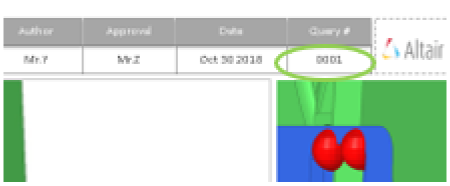
Depth of the intersection to be filtered. Intersection less than this value will not be reported in the PowerPoint.



Default/Allowed: 0.1 to 1.0 mm

slide-number

The starting number is shown in right top corner of PowerPoint file.



Default/Allowed: 1 to 10000

report

Initial directory for report output path.
Default/Allowed: `C:/temp`

action

User action type to be executed.

Check

Only intersection check will be executed, no reports.

Report

Only Reports will be generated from the previous check

Both

Check and Report generation will be executed in a sequence.

Default/Allowed: Check, Report or Both

mode

Default opitons for Run type.

interactive

Check executed in the front ground HyperMesh session

background

Check executed in the background HyperMesh sessions, automatic restart executed, if errors occur, the errors will be displayed in the browser as "Crash" keyword.

Default/Allowed: interactive or background

logic

Algorithm used for the checks.

allcomp

All parts will be imported, and check will be executed. Faster than “loop” logic but it requires good PC hardware.

loop

Two part will be imported at once and check will be executed, very slow but with less memory full model can be checked.

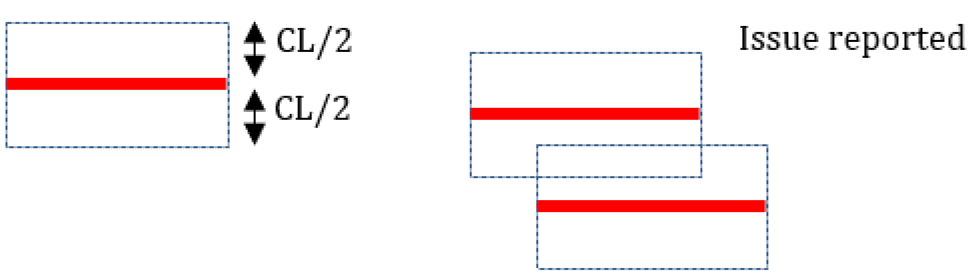
subsystem

Checks are executed across two subsystems, if two or more subsystems are selected from the tree or selected via assembly-level option. All parts will be checked, if one sub-assembly is selected from tree or assembly-level is 0.

Default/Allowed: allcomp, loop or subsystem

Clearance_Zone (CL)

Clearance value will be assigned on the CAD data during the intersection check. Half of the value will be assigned on each side of the CAD data. Increasing the value might increase the execution time.



-

Default/Allowed: 0.0. to 5.0 mm

Filter Settings

revision-check

New revisioned CAD data and its surrounding CAD data will be checked for intersection, if this option is ON. All the CAD parts irrespective of revision will be checked, if this option is turned OFF.
Default/Allowed: ON or OFF

revision-folder

Initial directory for selecting the revision CAD data. File name is compared against files in the selected folder and CAD files will be replaced and revision flag is ON for that part.
Default/Allowed: `C:/temp`

body-type

Special treatment based on the name.

METAL-METAL

All the parts inside the METAL folder will be checked.

TRIM-TRIM

All the parts inside the TRIM folder will be checked.

METAL-TRIM

Parts across METAL and TRIM folder will be checked. No check will be executed inside the METAL or TRIM folders.

Default/Allowed: METAL-METAL, TRIM-TRIM or METAL-TRIM

Max-number-solids

Parts will not be considered for the check, if the number of solid CAD inside a part exceeds this number. This is done to avoid check on solid spot welds or harness cables.
Default/Allowed: 2 to 150

loopLimit

Each time two parts will be compared if the value is 0. If more than 0, parts (specified # of parts) are loaded at once in one HyperMesh session and comparison will be executed.
Default/Allowed: 0 to 1000

min-areapercent

Minimum area of part that must be considered for the check. Area of the part less than this will be ignored in the check. This is implemented to avoid small parts that do not have proper Name or Part Number.
Default/Allowed: 1.0 to 1000.0 mm square

min-Diagonal-BBox

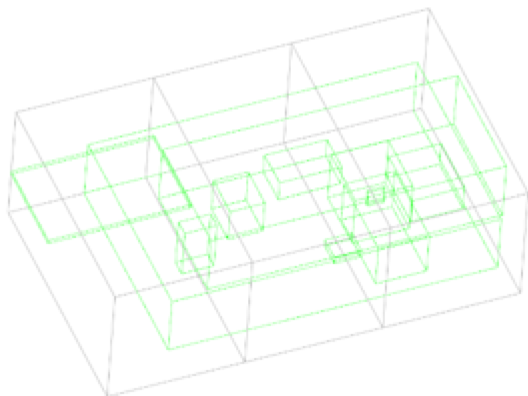
Minimum bounding box diagonal length of part that must be considered for the check. Diagonal length of the part less than this will be ignored in the check. This is implemented to avoid small parts that do not have proper Name or Part Number.
Default/Allowed: 1.0 to 10.0 mm

bbox-view

Bounding boxes for each part are displayed in the HyperMesh graphics for reference after the check and report is done. You can check if both of CAD or FE is at same place or same size.

Grey box
Indicates # CPUs

Green box
Indicates bounding box of the parts.



Default/Allowed: ON or OFF

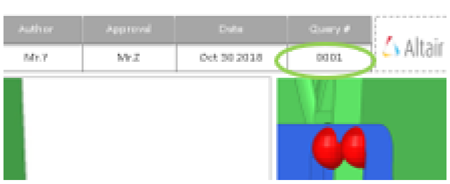
spotweld

Specify defaults for the spotweld check functionality of Model Verification.

General Settings

angle
Feature angle of the faces of FE.
Default/Allowed: 25 to 30 degrees

Slide-number
The starting number shown in right top corner of PowerPoint file.



Default/Allowed: 1 to 10000

input
Initial directory for the Input file path.
Default/Allowed: `C:/temp`

output
Initial directory for report output path.
Default/Allowed: `C:/temp`

action

User action type to be executed.

Check

Only intersection check will be executed, no reports.

Report

Only Reports will be generated from the previous check.

Both

Check and Report generation will be executed in a sequence.

Default/Allowed: Check, Report or Both

mode

Default options for Run type.

interactive

Check executed in the front ground HyperMesh session

background

Check executed in the background HyperMesh sessions, automatic restart executed, if errors occur, the errors will be displayed in the browser as "Crash" keyword.

Default/Allowed: interactive or background

vip-format

Space or 8-digit separated format.

Default/Allowed: space or 8-digit

resolveconflictingpid

5 digits or 7 digits format correction.

Default/Allowed: ON or OFF

realize-check

Ignores failed spot welds during the check.

Default/Allowed: ON or OFF

allow-multiple-issues

All check on all spot welds, there may be many issues on same welds.

Default/Allowed: ON or OFF

extract-spot-cad

Converts CAD points / solids to spot weld file. A copy of the spot weld vip file will be stored in CAD folder.

Default/Allowed: ON or OFF

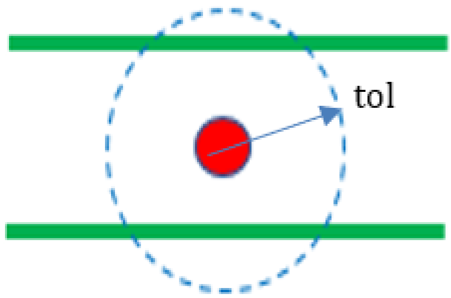
extract-spot-cad-option

Converts point/solid entities to spot weld file.

Default/Allowed: points or solids

realize-tolerance (tol)

Allow searching nearest spot on same Layer or different layer or bot / all duplications.



Default/Allowed: 1.0 to 10.0

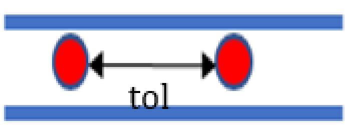
Closest

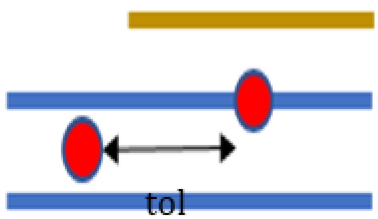
do-check

Turn OFF or ON the check
Default/Allowed: ON or OFF

Tolerance (tol)

Distance between two spot welds less than/equal to this tolerance are reported as duplicates. Distance are calculated after projecting to normal direction.



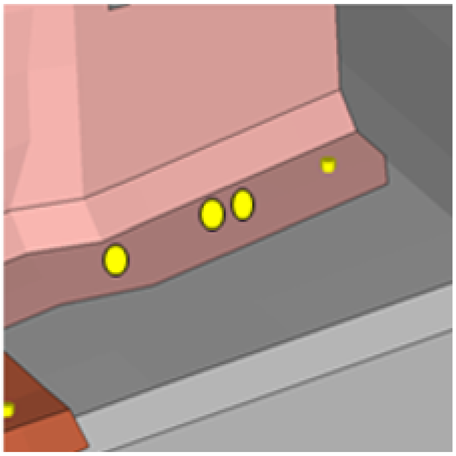


Default/Allowed: 0.1 to 20.0

checktypes

Allow searching duplicate spot welds on same layer (Same weld plates) or different layer or bot / all duplications.
Default/Allowed: sameStack, DifferentStack or Both

Spot weld found with the search radius of tolerance is identified.



Gap

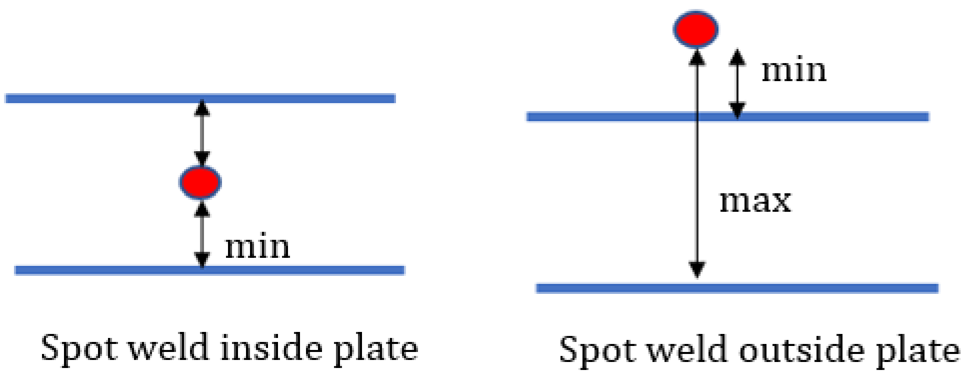
do-check

Turn OFF or ON the check

Default/Allowed: ON or OFF

min

Minimum gap between Spot location and the weld plates.



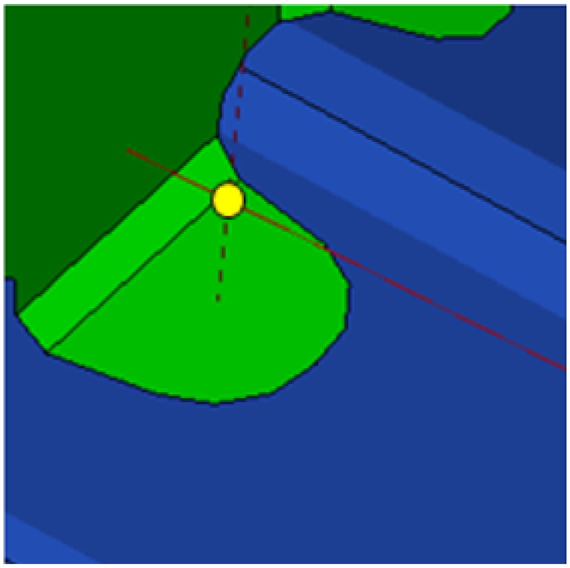
Default/Allowed: 1.0 to 10.0

max

Maximum gap to be considered between Spot location and the weld plates. This value is added to filter some spotwelds which are too far from the model.

Default/Allowed: 10.0 to 100.0

Spot weld between min and max distance from the nearest flange is identified, perpendicular distance is checked.



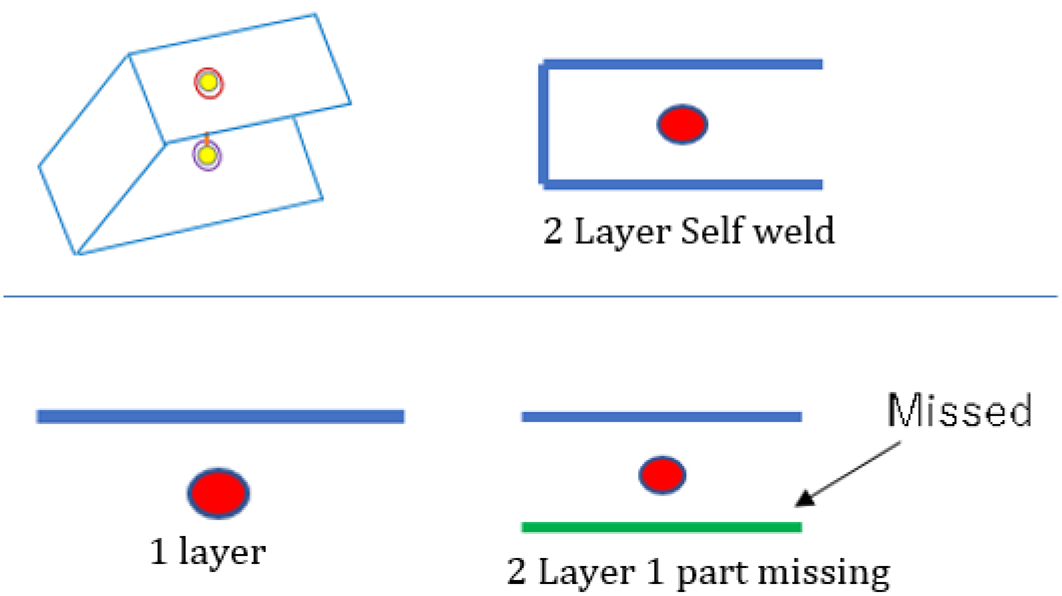
Single-Layer

do-check

Turn OFF or ON the check

Default/Allowed: ON or OFF

Self-welded spot welds are identified for reference. The link component 1 and link component 2 is same.



Multi-Layer

do-check

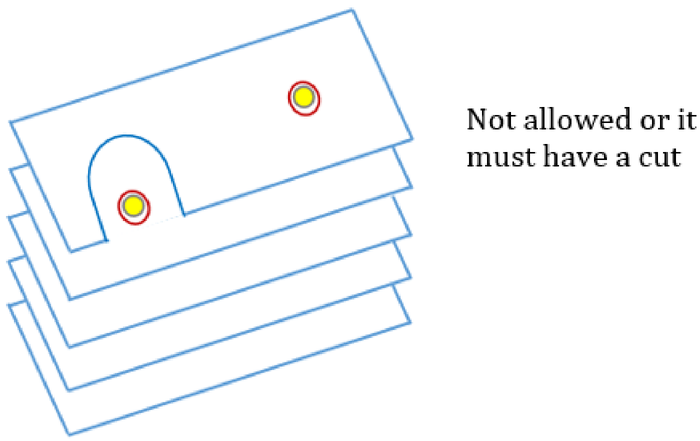
Turn OFF or ON the check

Default/Allowed: ON or OFF

threshold

Maximum allowed spot weld layers.
Default/Allowed: 2 to 4

Spot welds that has layers more than threshold value are identified for reference. It is expected to have a notch to allow Spot gun.



hm-issue

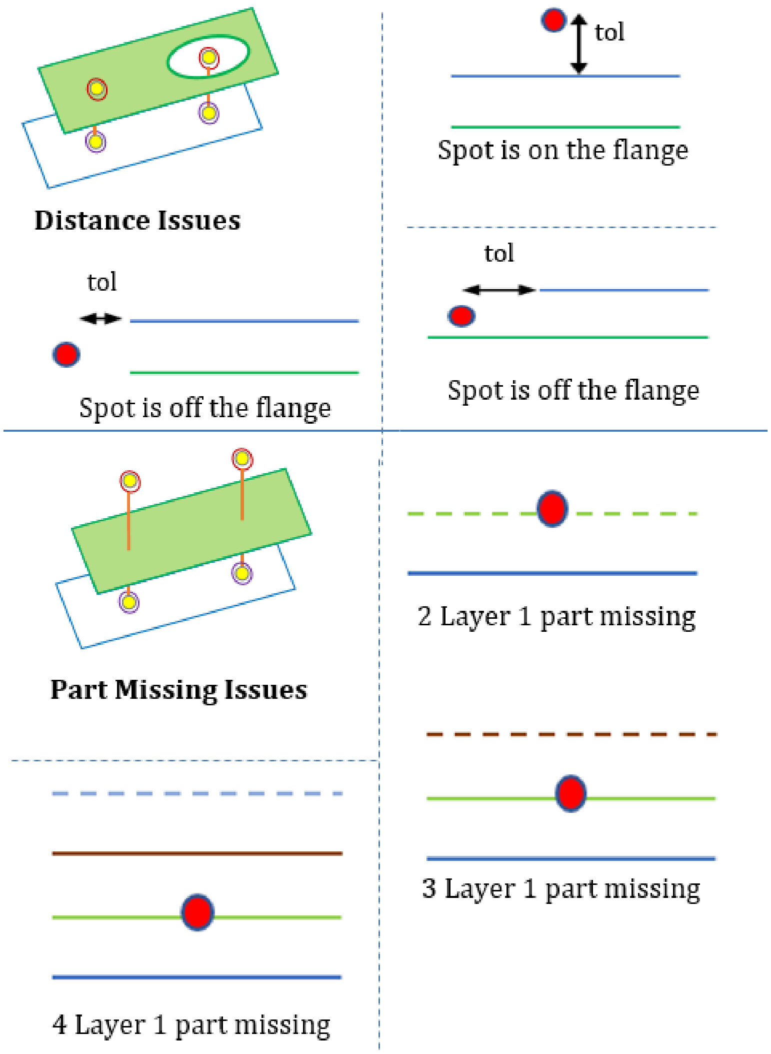
do-check

Turn OFF or ON the check
Default/Allowed: ON or OFF

Spot welds that are far from the Sheet metal part by "tolerance" search are identified and Part missing issues are directly identified by HyperMesh.

Tol

realize-tolerance (tol)



Parallel-flanges

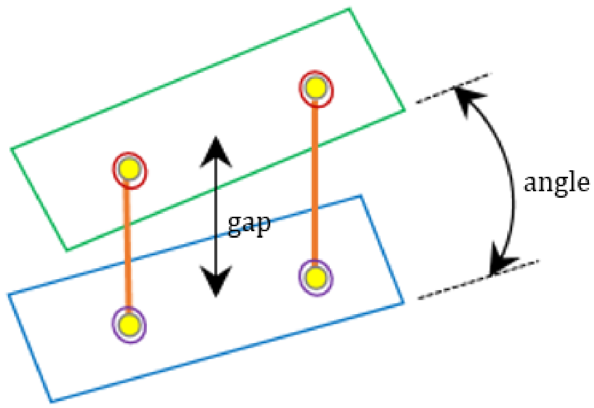
do-check

Turn OFF or ON the check

Default/Allowed: ON or OFF

angle

Maximum angle between two plates at spot location.

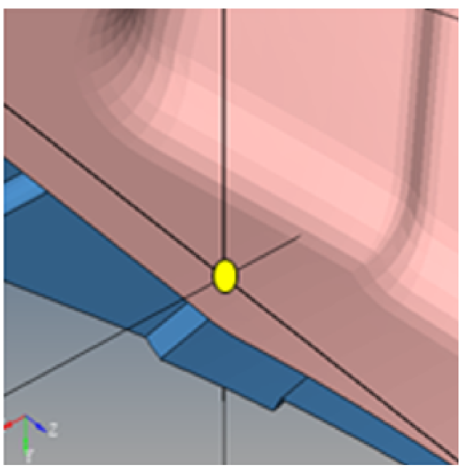


Default/Allowed: 2.0 to 15.0

max-gap

Maximum physical gap between two plates at Spot location.
Default/Allowed: 1.0 to 5.0

Non-Parallel flanges are identified at Spot weld location, if the angle between two sheet metal parts is more than angle value and more than gap value are identified.



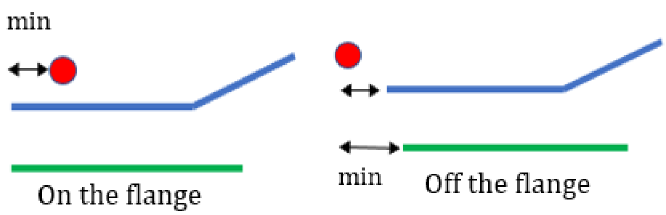
Incorrect-Location

do-check

Turn OFF or ON the check
Default/Allowed: ON or OFF

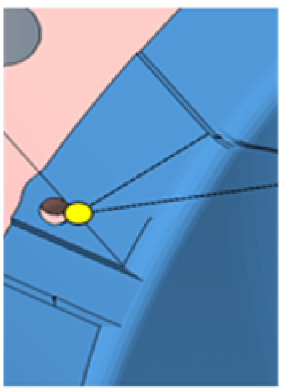
min-distance (min)

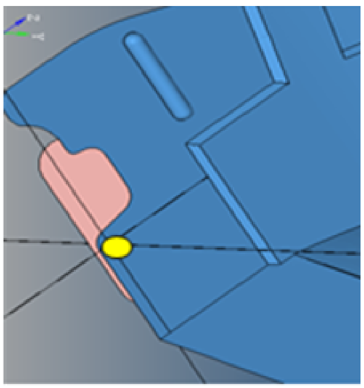
Minimum distance allowed from feature-line to the spot location.

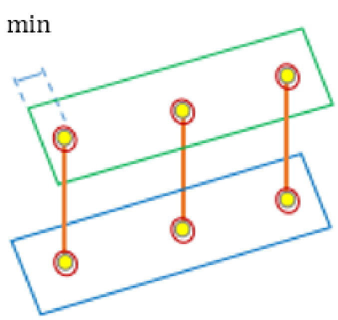


Default/Allowed: 1.0 to 5.0

Issue is Identified, if the distance between Spot weld and the nearest feature edge is more than the tolerance value. The tolerance refers to Spot gun "radius+gap".







Few-Connections

do-check

Turn OFF or ON the check
Default/Allowed: ON or OFF

Proximity (a)

Search tolerance for realizing spot weld on the flanges, it is global pitch.
Default/Allowed: 1.0 to 5.0 mm

pitch-spacing (b)

Average pitch distance for creating spot weld on flange.
Default/Allowed: 15.0 to 30.0

pitch-offset (c)

End offset distance on both side of the flange.

Default/Allowed: 5.0 to 8.0

pitch-edge-distance (d)

Offset distance from the edge of the flange.

Default/Allowed: 5.0 to 8.0

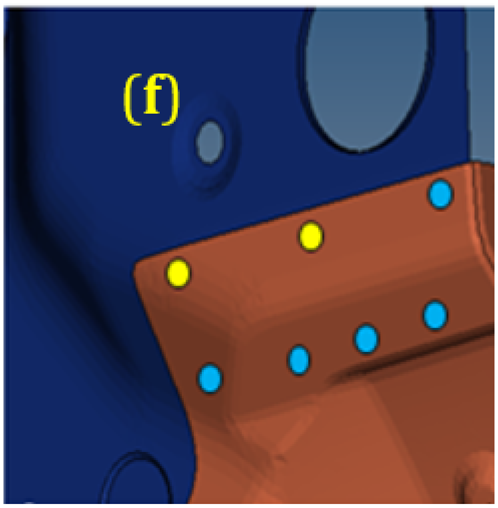
search tolerance (e)

Tolerance for searching Virtua connectors from existing connectors created from above parameter.

Default/Allowed: 30.0

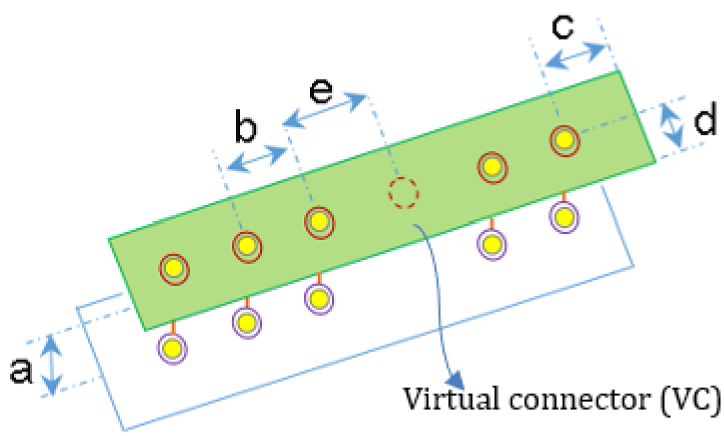
hole-diameter (f)

Search tolerance for finding nearest Bolt holes to avoid finding the issue.



Default/Allowed: 4.0 to 40.0

Creating virtual spot (VS) welds with above parameter. For every virtual connector (VC) search nearest existing spot weld (blue) with "search tolerance" value, identify location (yellow), if blue spot weld not found.



Flange-gap

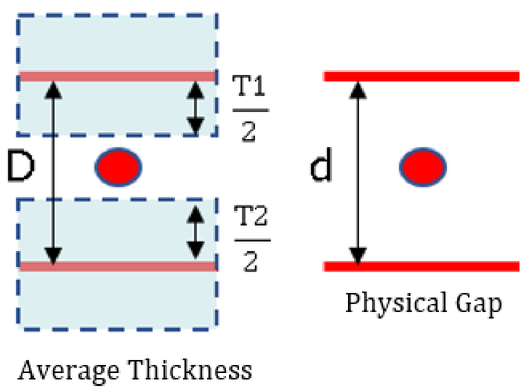
do-check

Turn OFF or ON the check

Default/Allowed: ON or OFF

limit

+/- ratio of shortest gap between flanges at connector to average component thickness value. It is also % of deviation.

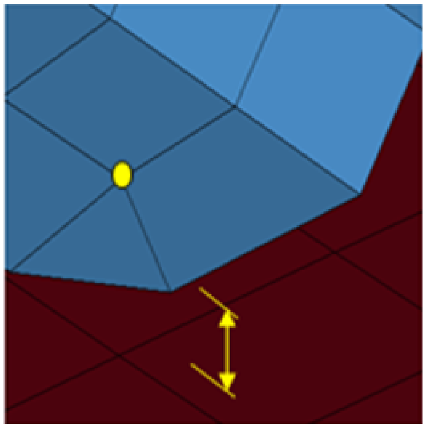


Calculation : abs(D - d) ≤ limit

Default/Allowed: 0.1 to 0.5 mm

This ratio of flange gap to average thickness of the component is compared with tolerance value. Issue is identified if the difference is same or less than the limit value.

Note: No Error will be reported if thickness does not exist or offset is not done.



Intersection

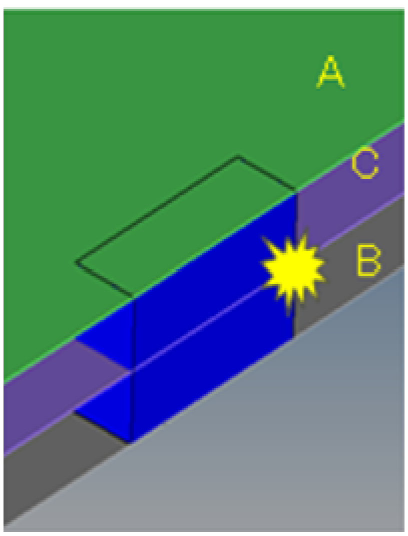
do-check

Turn OFF or ON the check

Default/Allowed: ON or OFF

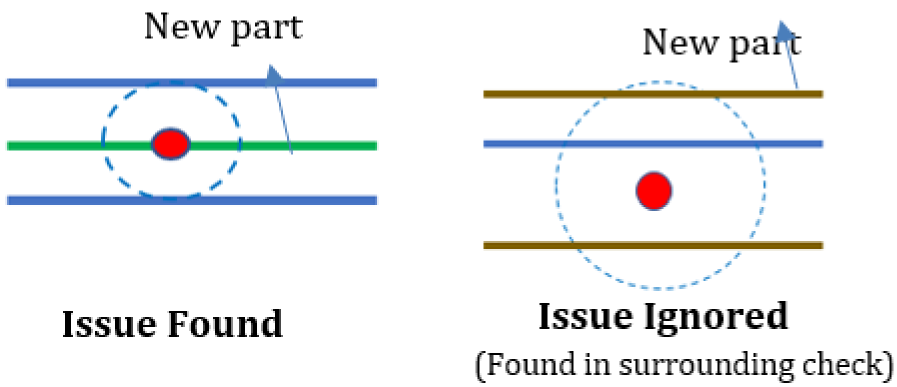
spot-layers

Maximum number of layers considered for this check.



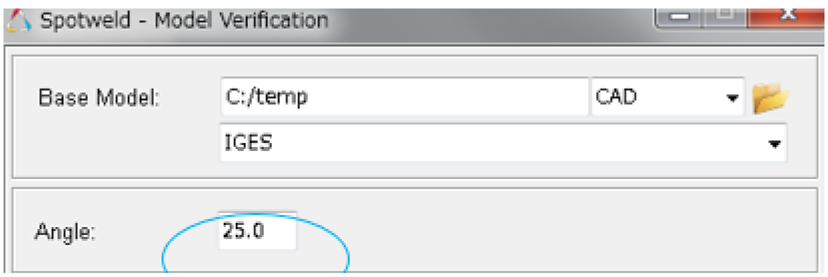
Default/Allowed: 2 to 3

Intersection on Spot realized FE data is found here or 2 layer to 3 layer or 3 layer to 4 layer conversion possible welds are identified.



Fillets

Fillet identification angle



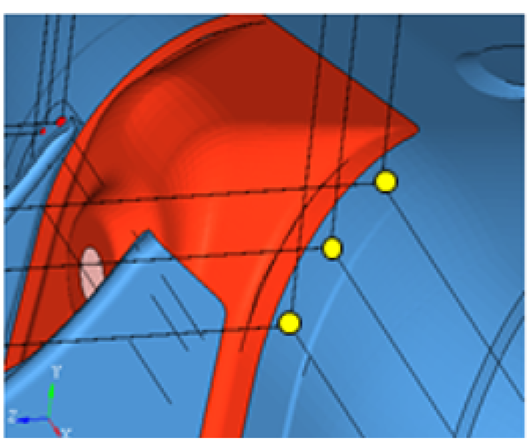
do-check

Turn OFF or ON the check

Default/Allowed: ON or OFF

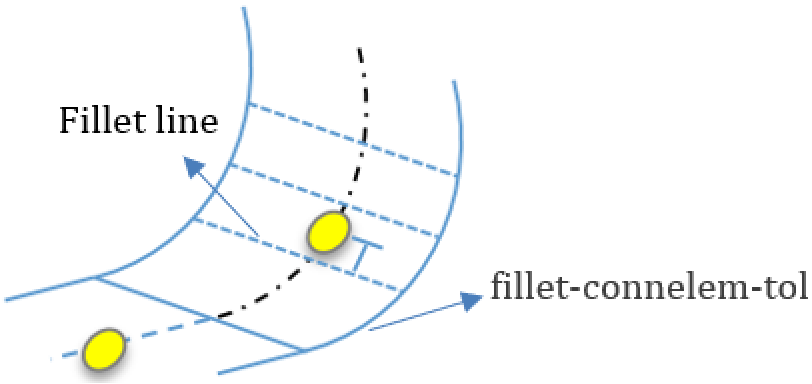
fillet-connelem-tol

Search tolerance for searching nearest fillet edges to the connector.



Default/Allowed: 10.0

If Spot welds location less than the fillet-connelem-tol distance from the nearest feature, line is identified here.



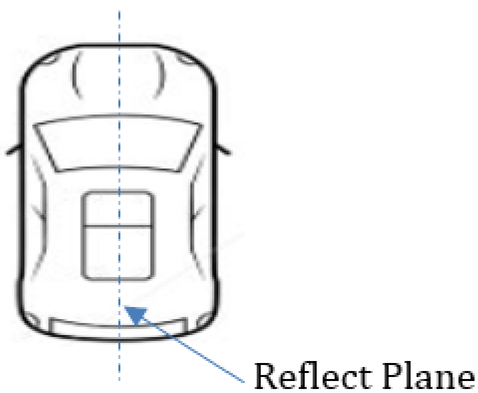
Reflect

do-check

Turn OFF or ON the check
Default/Allowed: ON or OFF

reflect-plane

Reflecting plane for the model which the spot welds are reflected and checked again for realization.



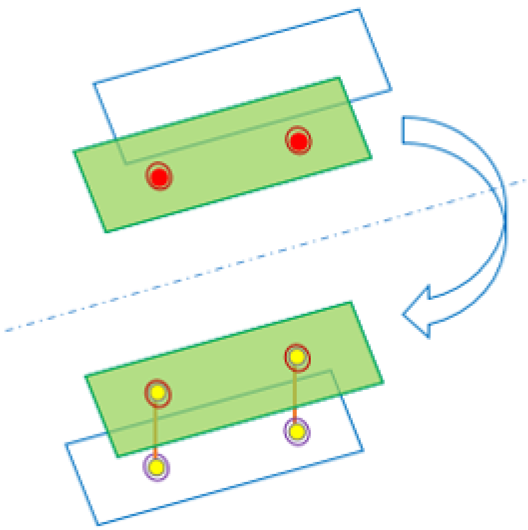
Default/Allowed: zx

origin

Origin value. (for XY plane, origin is constant Z value)
Default/Allowed: 0 to 1000.0

Failed spot welds are reflected at reflect-plane and re-realized, if successful in realization, it is identified as issue.

Issues will be ignored if surface offset is not executed and another spotweld exist at reflected location.



surrounding

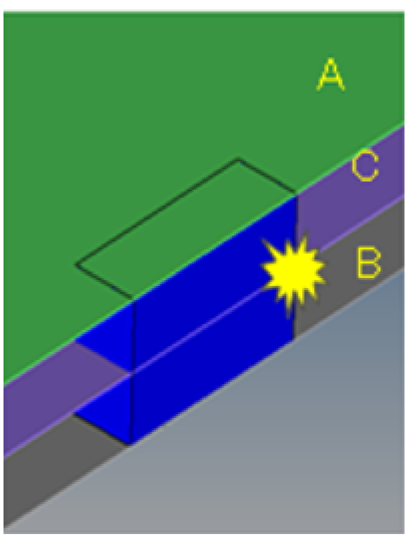
do-check

Turn OFF or ON the check

Default/Allowed: ON or OFF

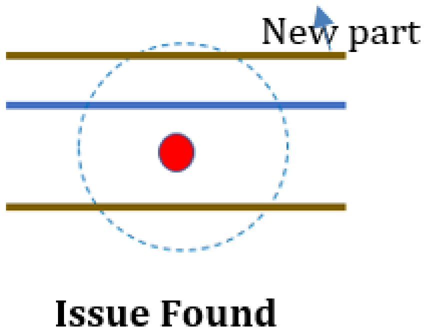
spot-layers

Maximum number of layers considered for this check.



Default/Allowed: 2 to 3

Intersection on Spot realized FE data is found here or 2 layer to 3 layer or 3 layer to 4 layer conversion possible welds are identified.



connection

Connection check configurations.

GUI Default Settings

slide-number
Initial slide # displayed on the report.
Default/Allowed: 1

action
Action items.
Default/Allowed: check, report or both (default)

mode
Execution mode, interactive mode will be slower than background mode.
Default/Allowed: interactive or background

input
Initial director for CAD Input.
Default/Allowed: `C:/temp`

output
Initial director for results output.
Default/Allowed: `C:/temp`

Hole Detection

min-diameter
Minimum diameter for surface hole recognition. Holes below this diameter will be ignored.
Default/Allowed: 1.0 to 10.0

max-diameter
Maximum diameter for surface hole recognition. Holes above this diameter will be ignored.
Default/Allowed: 1.0 to 10.0

min-nodes

Minimum number of nodes to recognize FE hole. Hole with nodes less than this number will be ignored.
Default/Allowed: 4 to 10

max-nodes

Maximum number of nodes to recognize FE hole. Hole with nodes greater than this number will be ignored.
Default/Allowed: 30 to 50

adjacent-tolerance

Distance to judge duplicate holes for the same plate. If the Holes to Holes distance is less than this value is considered as duplicate hole.
Default/Allowed: 3.0 to 5.0

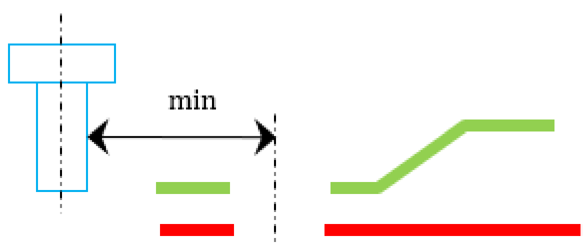
Free Hole

do-check

Turn OFF or ON the check.
Default/Allowed: ON or OFF

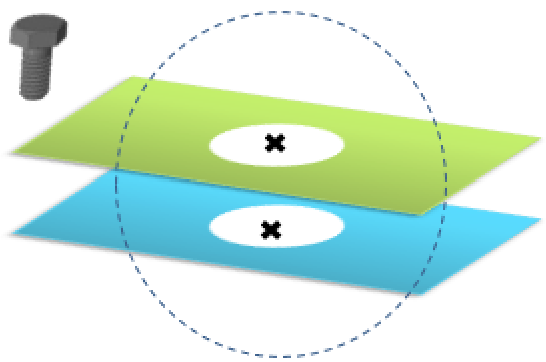
elem-adj-bbox-tolerance

Search tolerance for searching nearest bolts/nuts from the hole center.



Default/Allowed: 10.0

If there is no bolt, nut or screw nearby, the holes are considered as free holes.



freeboltnut

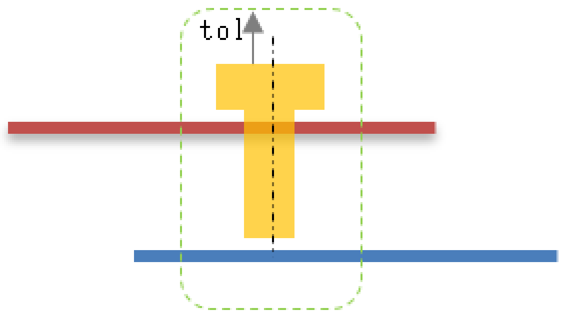
do-check

Turn OFF or ON the check.

Default/Allowed: ON or OFF

bolt-box-tolerance

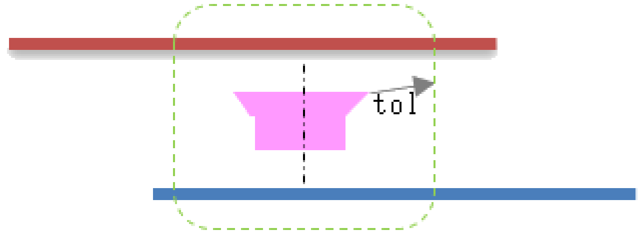
Search tolerance for searching nearest plates from the bolt outer surfaces.



Default/Allowed: 1.0 to 5.0

nut-box-tolerance

Search tolerance for searching nearest plates from the nut outer surface.



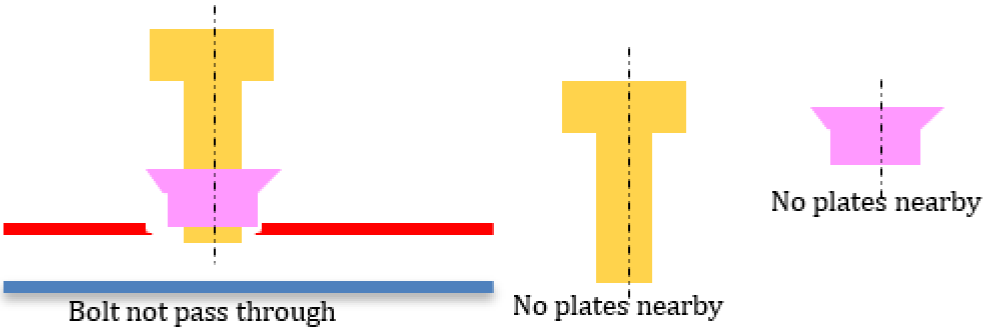
Default/Allowed: 5.0 to 10.0

allowed-plate-comps

These named parts will not be considered as plates.

Default/Allowed: string names of special parts

Bolt or Nut CAD data that are not passing through the plates or floating in the space without the plates are found.



bolthole-mismatch

do-check

Turn OFF or ON the check.

Default/Allowed: ON or OFF

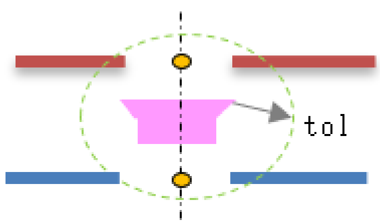
input-type

Continuous CAD lines or CAD solids.

Default/Allowed: lines or solids

centernode-search-tolerance

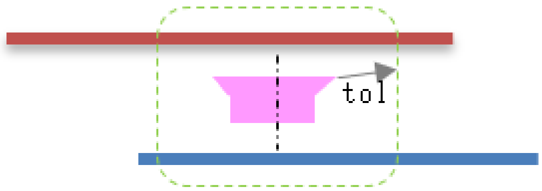
Search tolerance for searching holes from bolt CAD surface.



Default/Allowed: 1.0 to 2.0

elem-adj-bbox-tolerance

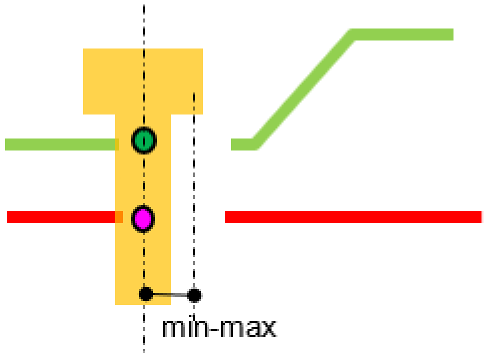
Tolerance for searching nearest plates from the bolt outer surface.



Default/Allowed: 5.0 to 10.0

min-dist

Allowable mismatch distance between bolt axis and hole axis.

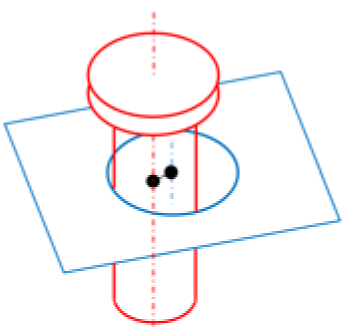


Default/Allowed: 0.1 to 1.0

max-dist

Maximum search distance between bolt axis and the hole axis.
Default/Allowed: 5.0 to 10.0

Mismatch issues is found if the mismatch is between min-dist to max-dist value.



Issue =
> (min-dist) and < (max-dist)

nuthole-mismatch

do-check

Turn OFF or ON the check.
Default/Allowed: ON or OFF

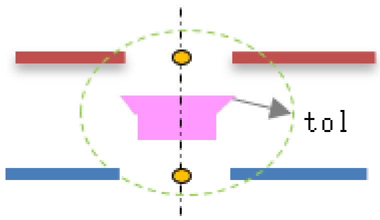
input-type

Continuous CAD lines or CAD solids.

Default/Allowed: lines or solids

centernode-search-tolerance

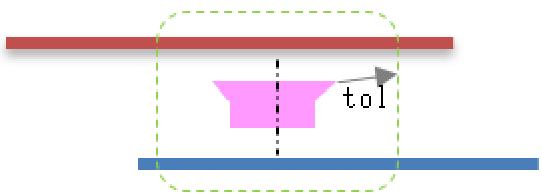
Search tolerance for searching holes from bolt/nut CAD surface.



Default/Allowed: 1.0 to 2.0

elem-adj-bbox-tolerance

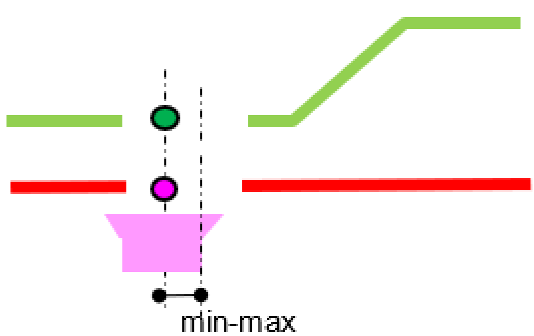
Tolerance for searching nearest plates from the nut outer surface.



Default/Allowed: 5.0 to 10.0

min-dist

Allowable mismatch distance between nut axis and hole axis. If the mismatch is less than this value issues will not be reported.



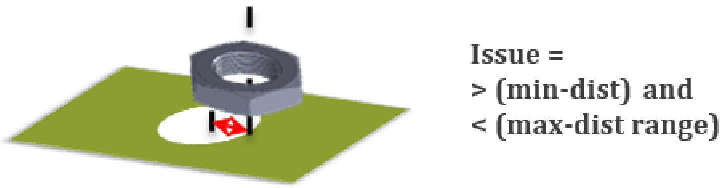
Default/Allowed: 0.1 to 1.0

max-dist

Maximum search distance between nut axis and the hole axis. If the mismatch is greater than this value, issues will not be reported.

Default/Allowed: 5.0 to 10.0

Issues are found if the mismatch distance is between min-dist to max-dist range.



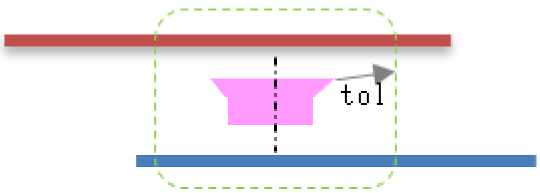
boltnut-mismatch

do-check

Turn OFF or ON the check.
Default/Allowed: ON or OFF

elem-adj-bbox-tolerance

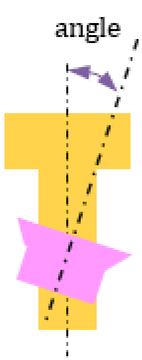
Tolerance for searching nearest plates from the nut outer surface.



Default/Allowed: 5.0 to 10.0

threshold-angle

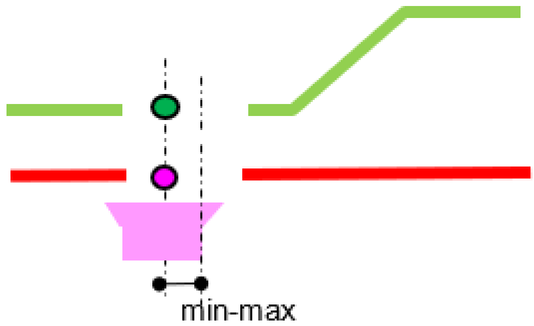
Allowable angle between center axis of bolt and the nut. More than this angle, the check will be ignored for that pair of connection parts.



Default/Allowed: 10.0 to 30 deg

min-dist

Allowable mismatch distance between nut axis and hole axis. If the mismatch is less than this value, issues will not be reported.

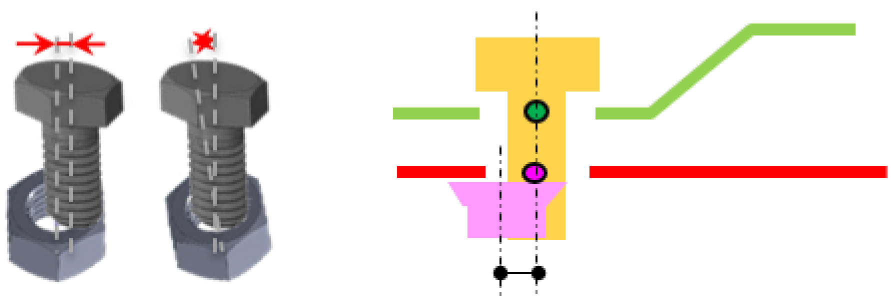


Default/Allowed: 0.1 to 1.0

max-dist

Maximum search distance between nut axis and the hole axis. If the mismatch is greater than this value, issues will not be reported.
Default/Allowed: 5.0 to 10.0

Bolt-nut mismatch issues will be reported if the Bolt and Nut sizes are not within min-dist and max-dist range.



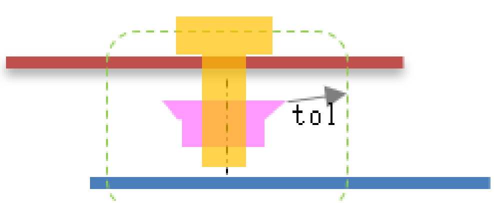
boltnutsize-mismatch

do-check

Turn OFF or ON the check.
Default/Allowed: ON or OFF

elem-adj-bbox-tolerance

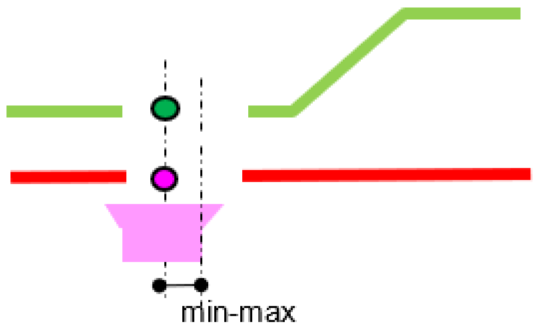
Tolerance for searching nearest bolt, plates from the nut outer surface.



Default/Allowed: 5.0 to 10.0

min-dist

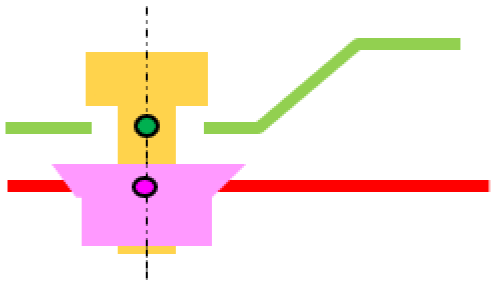
Allowable mismatch distance between nut axis and hole axis. If the mismatch is less than this value, issues will not be reported.



Default/Allowed: 0.1 to 1.0

min-nut-tol-bolt-ratio

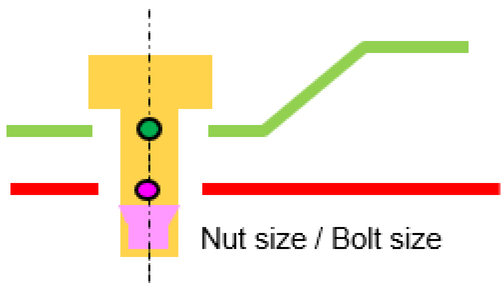
Allowable minimum ratio between bolt and nut size. If the ratio is less than this value, issue is reported.



Default/Allowed: 0.8 to 0.9

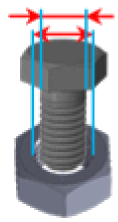
max-nut-tol-bolt-ratio

Allowable maximum ratio between bolt and the nut size. If the ratio is greater than this value, issue is reported.



Default/Allowed: 1.1 to 1.3

Issues will be reported if the Bolt size and Nut size are not in within the range of min-nut-tol-bolt-ratio and the max-nut-tol-bolt-ratio.



holes-mismatch

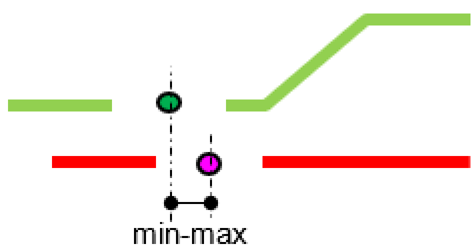
do-check

Turn OFF or ON the check.

Default/Allowed: ON or OFF

min-dist

Allowable mismatch distance between nut axis and hole axis. If the mismatch is less than this value, issues will not be reported.



Default/Allowed: 0.1 to 1.0

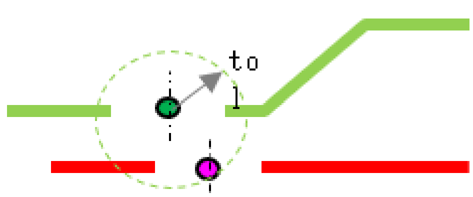
max-dist

Maximum search distance between hole center node axis.

Default/Allowed: 5.0 to 10.0

centernode-search-tolerance

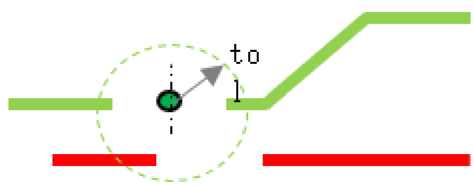
Tolerance for searching nearest plates from the hole center.



Default/Allowed: 1.0 to 2.0

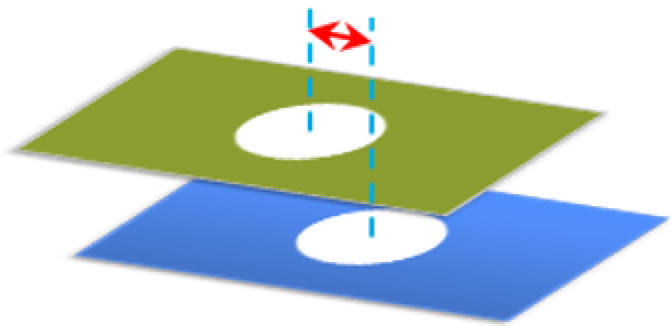
elem-adj-bbox-tolerance

Tolerance for searching nearest plates from the hole center.



Default/Allowed: 5.0 to 10.0

Issues will be reported if the center axis distance between two holes are not with the min-dist and max-dist.



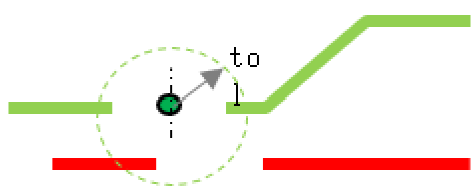
holespair-mismatch

do-check

Turn OFF or ON the check.
Default/Allowed: ON or OFF

elem-adj-bbox-tolerance

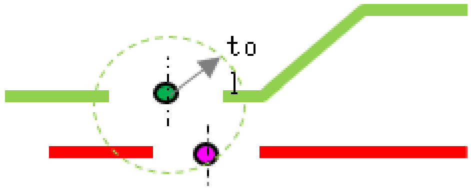
Tolerance for searching nearest plates from the hole center.



Default/Allowed: 5.0 to 10.0

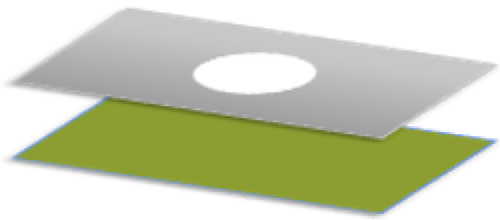
centernode-search-tolerance

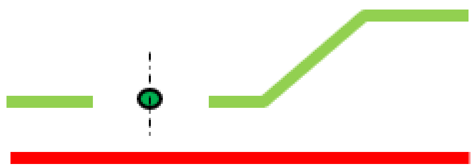
Tolerance for searching nearest plates from the hole center.



Default/Allowed: 1.0 to 2.0

Holes not found in the nearest plates are reported as an issue nearest plates. Plate holes are searched with centernode-search-tolerance tolerance.





Clip-mismatch

do-check

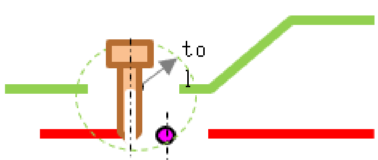
Turn OFF or ON the check.
Default/Allowed: ON or OFF

input-type

Continuous CAD lines or CAD solids.
Default/Allowed: lines or solids

centernode-search-tolerance

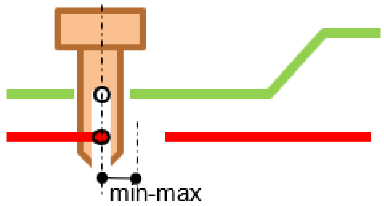
Search tolerance from center clip to holes center.



Default/Allowed: 1.0 to 2.0

min-dist

Allowable mismatch distance between clip axis and hole axis.

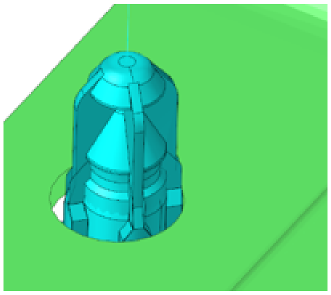


Default/Allowed: 0.5 to 1.0

max-dist

Maximum distance between clip axis and the hole axis. Greater than this value the clip parts are ignored in the check.
Default/Allowed: 5.0 to 15.0

Issues will be reported if the center axis of clip data and the center axis of the hole node is greater than the min-dist and less than max-dist.



Arc/Seam/Adhesive/Seal/Hemming -mismatch

do-check

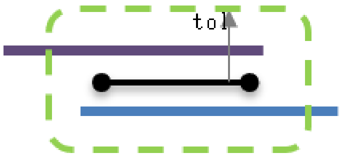
Turn OFF or ON the check.
Default/Allowed: ON or OFF

input-type

Continuous CAD lines or CAD solids.
Default/Allowed: lines or solids

elem-adj-bbox-tolerance (tol)

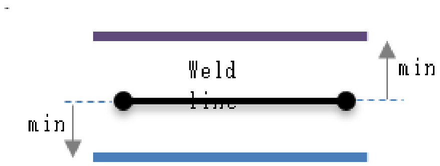
Search tolerance for finding nearest weldable plates. If less than 2 parts found, then issue is reported.



Default/Allowed: 10.0

min-dist

Minimum distance between arc weld to the nearest parts. If less than 2 parts found, then issue is reported.



Default/Allowed: 0.1 to 5.0

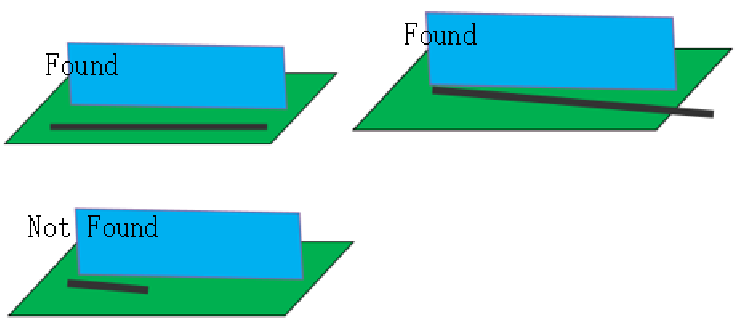
max-dist

Maximum distance between weld lines to the nearest parts.

Default/Allowed: 5.0 to 10.0

Finds shape mismatch between the weld lines against the plates with the given min and max distance.

The mismatches are measured from the end points of the line.



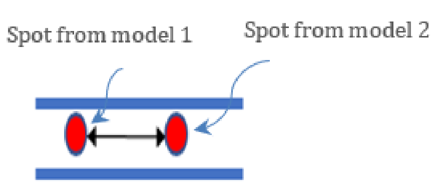
spot comparison

Spot comparison check configurations.

GUI Default Settings

tolerance

Distance between two spot welds less/equal to this tolerance are considered matching. Distance more than this tolerance is reported as positional difference.



Default/Allowed: 1.0 to 10.0

slide-number

The start number displayed on the top right corner of the PowerPoint slide.
Default/Allowed: 1 to 10000

input-1

Spot weld file 1.
Default/Allowed: `C:/temp`

input-2

Spot weld file 2 (same file cannot be selected).
Default/Allowed: `C:/temp`

report

Initial directory for report out path.
Default/Allowed: `C:/temp`

action

User action type to be executed.

Check

Only intersection check will be executed, no reports.

Report

Only Reports will be generated from the previous check.

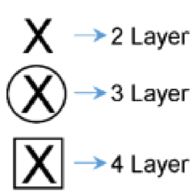
Both

Check and Report generation will be executed in a sequence.

Default/Allowed: Check, Report or Both

extract-spot-cad-option

CAD lines, points or solids are converted to spot weld file. A copy of the spot weld file is stored in the input folder.



In case of CatProduct file

Default/Allowed: lines, points or solids

extract-spot-method

Logic to convert CAD to spot file.

compname

CAD names are used to recognize CAD spot data. These names are recognized via configuration file, name, connection and spot inputs.

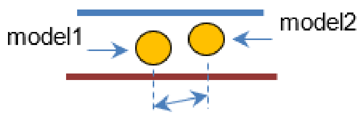
bbox

Based on the location and surrounding parts, CAD is converted to spot weld file.

Default/Allowed: compname or bbox

position

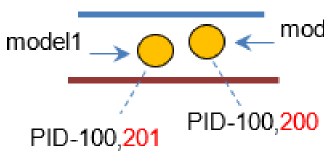
Position (Coordinates) difference between spot weld 1 and spot weld 2 will be reported if ON. Allowable positional difference is less than tolerance value.



Default/Allowed: ON or OFF

part-id

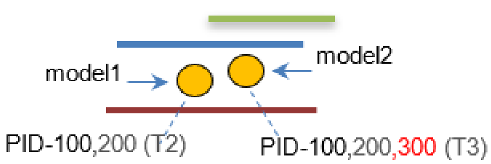
Part ID differences between spot weld 1 and spot weld 2 will be reported if ON.



Default/Allowed: ON or OFF

layer

#Layer, #Stack or #Link Components differences between spot weld 1 and spot weld 2 will be reported if ON.



Default/Allowed: ON or OFF

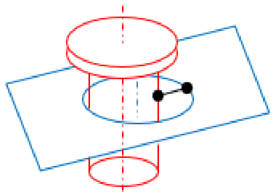
free part

Free park check configurations.

GUI Default Settings

gap

Allowable distance between two parts. Example, a bolt and the bolt hole clearance distance.



Default/Allowed: 0.1 to 5.0 mm

slide-number

The start number displayed on the top right corner of the PowerPoint slide.
Default/Allowed: 1 to 10000

inputdir

Initial director for the input data.
Default/Allowed: `C:/temp`

outputdir

Initial director for report output path.
Default/Allowed: `C:/temp`

action

User action type to be executed.

Check

Only intersection check will be executed, no reports.

Report

Only Reports will be generated from the previous check.

Both

Check and Report generation will be executed in a sequence.

Default/Allowed: Check, Report or Both

mode

Default options for Run type.

interactive

Check executed in the front ground session.

background

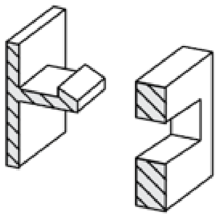
Check executed in the background HyperMesh sessions, automatic restart executed if error occurs, the errors will be displayed in the browser as “Crash” keyword.

Default/Allowed: interactive or background

Settings

contact-tolerance

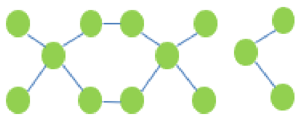
Allowable contact thickness between two parts. Example, for exhaust pipe press fit clearance distance.



Default/Allowed: 0.01 to 1.0mm

unconnected_count_min

Allowable unconnected parts. Issue will be found if few parts interconnected but the # of interconnected parts are less than this value will be reported.



3 parts unconnected

Default/Allowed: 1 to 5

min-area

Minimum area of part that must be considered for the check. Area of the part less than this will be ignored in the check. This is implemented to avoid small parts that do not have proper Name or Part Number.
Default/Allowed: 1.0 to 1000.0 mm square

color

Color of the unconnected parts in the PowerPoint report.
Default/Allowed: Red

csv-comparison

CSV comparison tool compares CAD/FE attributes against CSV file column.

Settings

mapattributes

displayname

Displays the Part Number.

PID

Displays the entity specific Property ID's. At the Part level it displays IDs for referenced properties. At the Component level it shows the ID of the referenced property.

Part Name

Displays the Part entity name.

Material
Displays the material name of the Part.

Thickness
Displays the entity specific thickness. At the Part level it displays the thicknesses of the referenced properties. At the Component level it shows the thickness of the referenced property.

Function Name
Displays sub assembly name of the Part.

Assembly Name
Displays assembly name of the Part.

Weight
Displays weight of the Part.

Configuration Options

csvcomparison/compare
Compares only the attributes between CSV and MVD attributes and results will be displayed in Error column.

csvcomparison/compare-update
Compares and updates the attributes if mismatch exist in the attributes.

csvcomparison/NewXML
CSV file will be converted to BOM and added as Variant BOM for future topology comparison.

reference-item
It is Common attribute name between FE/CAD model and CSV file header.

string-match-format

Generic
Full string vs string will be compared.

Prefix
Pre-fix, base-name, post-fix will be compared in the CSV file part name.

report
Excel report will be generated with model image in JPEG/H3D.

report-dir

Output path for report.

names

Naming conventions used in various Model Verification Modules.

Settings

module

Name settings for the modules within Model Verification.

comparison-base

Identify FE Model in CAD-FE Comparison.
Supported values: "*base_model*"

spotweld

Naming pattern for component to be identified as spot weld component.
Supported values: "*-w-*"

part

Naming conventions for components.

connection

Naming convention for classifying the component as various connecting components.

bolt

Supported values: "*bolt*", "!*plate*"

nut

Supported values: "*nut*", "!*plate*"

clip

Supported values: "*clip*"

rivet

Supported values: "*rivet*"

stud

Supported values: "*stud*"

screw

Supported values: "*screw*", "*scr*"

pin

Supported values: "*pin*"

arc

Supported values: "*arc*", "*weld*", "!*bolt*", "!*nut*"

spot

Supported values: *spot*

other

Supported values: "*", "!101-*", "!102-*", "!103-*", "!104-*"

unwanted

Supported values: "f/j_boot", "f/j_b/band", "s/j_boot", "s/j_b/band", "*bane*", "*cable*", "*fulcrum*", "*spring*", "*surf*", "*valve*""*wstrip*", "*zone*"

Add and Remove List Items

Steps on how to add and remove list items.

- Add new list item.
  a) Right-click on a Category with the list item and select **Add item** from the context menu.
  b) Enter the values.
  c) Click **Save** to save the addition.

  Note:
  Add is not allowed for Categories with no list items or sub-items.

- Remove list item.
  a) Right-click on the item and select **Cut item** from the context menu.
  b) Click **Save** to save the deletion.

Examples:

- Category - bolt, nut, unwanted or others.
- Property - *bolt, *nut or *clip.

report

Report outputs and visualization configuration.

Settings

advance

true = Advance style report with MultiCPU, false = Normal style report with 1CPU.
Supported values: true

avoid_duplicate_slide

true = duplicate slides avoided, #duplicate issues shown under instance, false = all of the issues for single pairs will be reported.
Supported values: true

visualization settings define configurations related to visualization.

bg

Background colors to be used for creating H3D files and images.

h3d

The background color to be used when creating H3D files.
Supported values: 255, 255, 255, 255, 255 255

jpg

The background color to be used when creating JPG image files.
Supported values: 49, 111, 255, 95, 181 266

color

Default colors for various check results. Click here for more information about color numbers.

comparison

Default colors for comparison check.

master

Color number for Master component.
Supported values: Color Number, ex - 12

slave

Color number for Master component.
Supported values: Color Number, ex - 6

unmatched

Color number for unmatched entities.
Supported values: Color Number, ex - 33

WT-Calculation

Option to display the weight in the reports. true - will enable the display and false - will disable the display.
Supported values: true or false

WT-Comparison

Option to display the delta weight in the reports. true - will enable the display and false - will disable the display.
Supported values: true or false

WT-Graph

Option to display the weight graph in the reports. true - will enable the display and false - will disable the display.
Supported values: true or false

WT-Graphtype

Option to choose the waterfall graph type. The values 0 / 1 / 2 / 3 represent the available graph types.
Supported values: 0, 1, 2, 3

WT-Scalefactor

Multiplies the weight with the specified multiplication factor.
Supported values: A desired multiplication factor. Default is 1.

intersection

Default colors for intersection test. Click here for more information about color numbers.

master

Color number for Master component.
Supported values: Color Number, ex - 55

slave

Color number for Master component.
Supported values: Color Number, ex - 21

master-intersecting

Color number for intersecting elements on the master component.
Supported values: Color Number, ex - 62

slave-intersecting

Color number for intersecting elements on the slave component.
Supported values: Color Number, ex - 47

transparency

Transparency settings.

intersection

Transparency settings for intersection check.

master

The choice to turn on transparency for the master component. True enables transparency and false makes the component opaque.
Supported values: true or false

slave

The choice to turn on transparency for the master component. True enables transparency and false makes the component opaque.
Supported values: true or false

level

The level of transparency on a scale of 0 to 10. 0 being no transparency and 10 being completely transparent.
Supported values: Number, ex - 3

intersection

Visualization related settings for intersection reporting.

cross

Settings related to Cross.

length

The length of the cross as a percentage of the model length.
Supported values: Percentage, ex - 8

width

The width of the cross as a percentage of the model length.
Supported values: Percentage, ex - 0.15

sphere

Sphere dimension for the sphere.

radius

The radius of the sphere as a percentage of the cross length.
Supported values: Percentage, ex - 2

color

The color to be used for the sphere by specifying the corresponding color number from the palette.
Supported values: Percentage, ex - 3

reportgroup

multiple = Multiple slides in single PPT.
Supported values: single / multiple

spotweld

Setting related to spot weld check reporting.

cross

Settings related to cross.

length

The length of the cross as a percentage of the model length.
Supported values: Percentage, ex - 8

width

The width of the cross as a percentage of the model length.
Supported values: Percentage, ex - 0.15

sphere

Sphere dimension for the sphere.

radius

The radius of the sphere as a value in model units.
Supported values: Percentage, ex - 2

color

The color to be used for the sphere by specifying the corresponding color number from the palette.
Supported values: Percentage, ex - 3

clip

Clip dimensions.

radius

Attribute - TBD.
Supported values: Value in model units, ex - 150

hvp

DO NOT CHANGE.
Supported values: Text Value, ex – HvpLeft, HvpRight

bordercolor

Border color of the HVP title note.
Supported values: Color Number, ex - 2

titlefontsize

Title font size.

Supported values: Color Number, ex - 2

titlecolor

Title Color.

Supported values: Color Number, ex - 4

titlefont

Title Font.

Supported values: Color Number, ex - 1

comparisonunified settings:

max-record

Supported values: 50

template

The file name of the MS PowerPoint template to be used for reporting.

Supported values: File name, ex – "`Template_compare.potx`"

template-sheet

Name given to template.

skip-assembly-image

Parameter to skip assembly image in Comparison PowerPoint (check box).

template-ppt

Path of the power point template.

percentage-sequence

This is parameter for arranging the order of thumb nail images of components in the PowerPoint.

Supported values:

0 – Dfault order.

1 – Match percent in decreasing order.

2 – Part IDs in decreasing order.

resultslide

This is parameter to get the default multiple slides for multi-variant comparison or to get single slide which shows best match result.

Supported values:

Default – Multiple slides are shown.

Best match – Best matches for each components are shown from the multiple variants.

threshold_report

This is an intermediate percentage value to be given between threshold value and maximum value. The components having match percent value between threshold and threshold_report value is shown in green color and the components with value between threshold_report and maximum value in grey color.

intersection settings are related to reporting for intersection check.

text-static

User can enter the information.

Supported values:

```
object: "Text left"
description: "Intersection Check"
object: "Text right"
description: "Answer:"
object: "Info3"
description: "_"
object: "Info4"
description: "_"
object: "Info5"
description: "_"
object: "Info6"
description: "_"
```

spotweld settings:

max-slide

Maximum number of slides to be included in the PPT file.

Supported values: Number,ex-50

text-static

User can enter the information.

Supported values:

```
object: "Text left"
description: "Spotweld Check"
object: "Text right"
description: "Answer:"
object: "Info3"
description: "_"
object: "Info4"
description: "_"
object: "Info5"
description: "_"
object: "Info6"
description: "_"
```

spot-comparison settings:

max-slide

Maximum number of slides to be included in the PPT file.

Supported values: Number,ex-50

template

The file name of the MS PowerPoint template to be used for reporting.

Supported values: File name, ex – "`Template.potx`"

text-static

User can enter the information.

Supported values:

```
object: "Text left"
description: "Spotweld Comparison"
object: "Text right"
description: "Answer:"
```

```
object: "Info3"
description: "_"
object: "Info4"
description: "_"
object: "Info5"
description: "_"
object: "Info6"
description: "_"
```

freepart settings:

max-slide
Maximum number of slides to be included in the PPT file.
Supported values: Number,ex-50

template
The file name of the MS PowerPoint template to be used for reporting.
Supported values: File name, ex – "`Template.potx`"

layout
DO NOT CHANGE.
Supported values: Number, ex – 2

text

description
DO NOT CHANGE.
Supported values: Text Value, ex – "Text1"

project
DO NOT CHANGE.
Supported values: Text Value, ex – "Info1"

id
DO NOT CHANGE.
Supported values: Text Value, ex – "Info7"

date
DO NOT CHANGE.

text-static
User can enter the information.
Supported values:

```
object: "Text left"
description: "Connection Check"
object: "Text right"
description: "Answer:"
object: "Info3"
description: "_"
object: "Info4"
description: "_"
object: "Info5"
description: "_"
object: "Info6"
description: "_"
```

Save Reps

Save Representations in the input folder.

In the Comparison Browser, right-click on **Model** and select **Representation** > **Save** from the context menu.

Configure Parts

Configure any part.

1. From the Verification browser, right-click and select **Tools** > **Part Configuration** from the context menu.
2. Select the **Data Type** from the drop-down list.
3. Enter or browse for the Representation path.
4. Select the FE or CAD type from the drop-down list.
5. Select the new path for Representation folder.
6. Select the base configuration
7. Select the **Part Name** or **Part ID** from the drop-down list.
8. Click **Close** to exit the configuration.
9. Click **Check** to check and update the Representation.
10. Click **Update** to update the Representation with new settings.
11. Click **Import** to import the Representation.

Review in HyperView Player

Review the selected Part Assembly/Part in HyperView Player.

This review function uses parallel processing and is used in case of huge number of selected parts and it takes time in normal import operation.

From the Verification browser, right-click and select **Representations** > **Load in HyperView Player** from the context menu.

Once the model is displayed in the HyperView Player, user can perform the following:

- Review all the displayed parts.
- Right-click to cross check the details.

  A copy of the H3D file is stored in working folder.
- Drag and drop this file on PowerPoint / Excel / Word to review the model in business meetings.

> **Note:** The number CPU set in the configuration general section is used for parallel processing.

Import happens by filtering connection parts set in config GUI / Names. After review the parts in HyperView Player, it is expected to close the HyperView Player to proceed with other operation in HyperMesh.

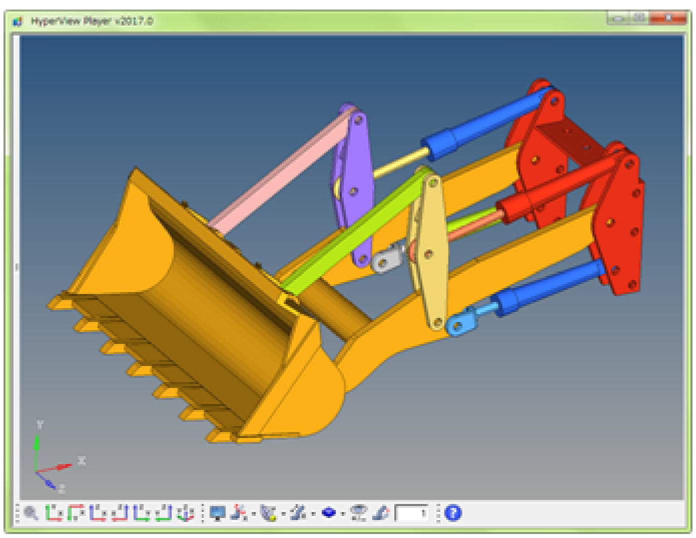


Figure 1532:

Review in HyperMesh

Review the selected Part Assembly/Part in HyperMesh graphics.

This review function uses parallel processing and is used in case of huge number of selected part and it takes time in normal import operation.

Once the model is displayed in the graphics, user can select unwanted parts in the graphics and delete them, simultaneously the Part Browser gets updated.

Note: The number CPU set in the configuration general section is used for parallel processing.

It is recommended to use this function before CAD Intersection or Comparison is executed for full car model where it is not sure the parts are in right position.

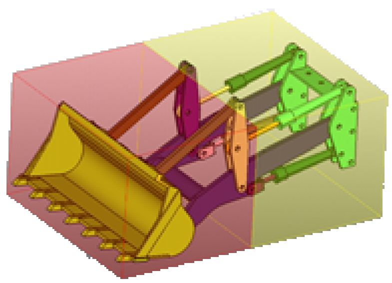


Figure 1533: Intersection Check on Each CPU

From the Verification/Comparison browser right-click context menu, select **Review Results**.

Export Parts

Export part representations.

1. From the Verification browser, right-click and select **Parts** > **Export** from the context menu.
2. In the **Browse for Folder** dialog, select the folder to export the Part Representation and click **OK**.

Selected parts are exported along with the XML file.

Rename Parts

Rename and set PIDs for all parts.

Part name and PIDs are extracted from file name displayed in Representation File column. For CAD inputs it is automatically executed during Intersection or Comparison.

From the Verification browser, right-click and select **Parts** > **Rename** from the context menu.

Renumber Parts

Renumber PIDs for all parts.

PIDs are renumbered based on configuration setting under **GUI** > **general** > **tool** > **Renumber**. For CAD inputs, it is automatically executed during Intersection or Comparison. You can turn off the auto-renumber options in case the PIDs to be preserved.

From the Verification browser, right-click and select **Parts** > **Renumber** from the context menu.

Count Parts

Get the count of the selected Part Representations.

Multiple Parts and Part Assemblies can be selected.

From the Verification browser, right-click and select **Parts** > **Count** from the context menu.

Batch Mode

Run Model Verification operations in batch mode.

Batch Command

Batch command to load all the functionalities of Model Verification into the batch mode of HyperMesh.

```
<altair_home>/hm/bin/<platform>/hmbatch.exe -tcl <mvd/src\main>/batch.tcl
```

Batch Options

Syntax and the description of the available options when using Model Verification in batch mode.

Intersection Check

Supported Batch options for Intersection checks.

Mandatory Arguments

"%hmbatch%"
: File path of `hmbatch.exe`
: Value: hmbatch.exe

-tcl
: File path of `batch.tcl`
: Value: batch.tcl

-mvd-feature
: Function name
: Value: intersection

-mvd-input
: Folder / File name
: Value: "%input%"

-mvd-input-type
: Cad/Fe/XML file type of %input%
: Value: " iges"

-mvd-output
: Result xml file name
: Value: "intresult.xml"

-mvd-report
: Report output path

Value: "%report%"

Optional Arguments

-nocommand
No input needed
Value: Blank

-nouserprofiledialog
No input needed
Value: Blank

-mvd-action
Action type
Value: check/report/both

-mvd-bg
No input needed
Value: Blank

-mvd-ncpu
# of cores in current machine
Value: 2

-mvd-log
File name / File name
Value: mvdLog.log

-mvd-project-name
Any string value display in report
Value: “Name”

-mvd-slide-number
Any integer value display in report
Value: “1”

-mvd-progress
File name / File name
Value: mvdprogress.log

Example

Example of a batch command.

```
"%hmbatch%" -nocommand -nouserprofiledialog -tcl "%tcl_file%" -mvd-feature
 intersection
-mvd-input "%input%" -mvd-input-type "iges" -mvd-output "%output%" -mvd-report
 "%report%"
-mvd-action both -mvd-report-scope both -mvd-bg -mvd-ncpu 1 -mvd-log mvdLog.log -mvd-
progress
```

```
mvdprogress.log
```

Comparison Check

Supported Batch options for Comparison checks.

Mandatory Arguments

"%hmbatch%"
: File path of `hmbatch.exe`
: Value: hmbatch.exe

-tcl
: File path of `batch.tcl`
: Value: batch.tcl

-mvd-feature
: Function Name
: Value: intersection

-mvd-base
: Folder path / File name
: Value: "%base%"

-mvd-base-type
: Cad/Fe/XML file type of %input%
: Value: “iges”

-mvd-variant
: Folder path / File name
: Value: "%variant%"

-mvd-variant-type
: Cad/Fe/XML file type of %input%
: Value: “radioss”

-mvd-output
: Result xml file name
: Value: "compresult.xml"

-mvd-report
: Report output path
: Value: "%report%"

Optional Arguments

-nocommand
: No input needed
: Value: Blank

-nouserprofiledialog
: No input needed
: Value: Blank

-mvd-action
Action type
Value: check/report/both

-mvd-report-scope
Report Option
Value: Excel / PPT / both

-mvd-bg
No input needed
Value: Blank

-mvd-ncpu
# of cores in current machine
Value: 2

-mvd-log
File name / File name
Value: mvdLog.log

-mvd-progress
File name / File name
Value: mvdprogress.log

Example

Example of a batch command.

```
"%hmbatch%" -nocommand -nouserprofiledialog -tcl "%tcl_file%"
-mvd-feature comparison -mvd-base "%base%" -mvd-base-type iges -mvd-variant
"%variant%" -mvd-variant-type dyna -mvd-output "%output%" -mvd-report "%report%"
-mvd-action both -mvd-report-scope both -mvd-bg -mvd-ncpu 1 -mvd-log mvdLog.log
-mvd-progress mvdprogress.log
```

Spotweld Check (Fe)

Supported Batch options for Spotweld checks (Fe).

Mandatory Arguments

"%hmbatch%"
File path of `hmbatch.exe`
Value: hmbatch.exe

-tcl
File path `batch.tcl`
Value: batch.tcl

-mvd-feature
Function name
Value: spotweld

-mvd-input

Folder path of spot file & Fe file

Value: "%input%"

-mvd-input-type

Fe file type of %input%

Value: "optistruct/radioss/pam"

-mvd-output

Report output path

Value: "%output%"

> **Note:** The Spot file type is read from the configuration file.

Optional Arguments

-nocommand

No input needed

Value: Blank

-nouserprofiledialog

No input needed

Value: Blank

-mvd-action

Action needed

Value: check/report/both

-mvd-bg

No input needed

Value: Blank

-mvd-ncpu

# of cores in current machine

Value: 2

-mvd-log

File name / File name

Value: mvdLog.log

-mvd-project-name

Any string value display in report

Value: “Name”

-mvd-slide-number

Any integer value display in report

Value: “1”

-mvd-progress

File name / File name

Value: mvdprogress.log

Example

Example of a batch command.

```
"%hmbatch%" -nocommand -nouserprofiledialog -tcl "%tcl_file%"
-mvd-feature spotweld -mvd-input "%input%" -mvd-input-type “pam” -mvd-output
"%output%" -mvd-action both -mvd-report-scope both -mvd-bg -mvd-ncpu 1 -mvd-log
mvdLog.log -mvd-progress mvdprogress.log
```

Spotweld Check (CAD)

Supported Batch options for Spotweld checks (CAD).

Mandatory Arguments

"%hmbatch%"

File path of `hmbatch.exe`
Value: hmbatch.exe

-tcl

File path `batch.tcl`
Value: batch.tcl

-mvd-feature

Function name
Value: spotweld

-mvd-input

Folder path contains spot file
Value: "%input%"

-mvd-spotweld-bom

XML file path
Value: "%bomfile%"

-mvd-output

Report output path
Value: "%output%"

> **Note:** The Spot file type is read from the configuration file.

Optional Arguments

-nocommand

No input needed
Value: Blank

-nouserprofiledialog

No input needed
Value: Blank

-mvd-action
: Action type
: Value: check/report/both

-mvd-bg
: No input needed
: Value: Blank

-mvd-ncpu
: # of cores in current machine
: Value: 2

-mvd-log
: File name / File name
: Value: mvdLog.log

-mvd-project-name
: Any string value display in report
: Value: "Name"

-mvd-slide-number
: Any integer value display in report
: Value: “1”

-mvd-progress
: File name / File name
: Value: mvdprogress.log

Example

Example of a batch command.

```
"%hmbatch%" -nocommand -nouserprofiledialog -tcl "%tcl_file%"
-mvd-feature spotweld -mvd-input "%input%" -mvd-spotweld-bom "%bomfile%"
-mvd-output "%output%" -mvd-action both -mvd-report-scope both -mvd-bg
-mvd-ncpu 1 -mvd-log mvdLog.log -mvd-progress mvdprogress.log
```

Connector Check (CAD)

Supported Batch options for Connection checks (CAD).

Mandatory Arguments

"%hmbatch%"
: File path of `hmbatch.exe`
: Value: hmbatch.exe

-tcl
: File path `batch.tcl`
: Value: batch.tcl

-mvd-feature
: Function Name

Value: connection

-mvd-input
Folder path
Value: "%input%"

-mvd-connection-bom
XML file path
Value: "%bomfile%"

-mvd-output
Report output path
Value: "%output%"

Optional Arguments

-nocommand
No input needed
Value: Blank

-nouserprofiledialog
No input needed
Value: Blank

-mvd-action
Action type
Value: check/report/both

-mvd-bg
No input needed
Value: Blank

-mvd-ncpu
# of cores in current machine
Value: 2

-mvd-log
File name / File name
Value: mvdLog.log

-mvd-project-name
Any string value display in report
Value: “Name”

-mvd-slide-number
Any integer value display in report
Value: “1”

-mvd-progress
File name / File name

Value: mvdprogress.log

Example

Example of a batch command.

```
"%hmbatch%" -nocommand -nouserprofiledialog -tcl "%tcl_file%"
-mvd-feature connection -mvd-input "%input%" -mvd-connection-bom "%bomfile%"
-mvd-output "%output%" -mvd-action both -mvd-report-scope both -mvd-bg -mvd-ncpu
1 -mvd-log mvdLog.log -mvd-progress mvdprogress.log
```

Free-Part Check (CAD)

Supported Batch options for Free-Part checks (CAD).

Mandatory Arguments

"%hmbatch%"

File path of `hmbatch.exe`
Value: hmbatch.exe

-tcl

File path `batch.tcl`
Value: batch.tcl

-mvd-feature

Function name
Value: freepart

-mvd-input

Folder path (contains Spot file)
Value: "%input%"

-mvd- freepart-bom

XML file path
Value: "%bomfile%"

-mvd-output

Report output path
Value: "%output%"

> **Note:** The Spot file type is read from the configuration file.

Optional Arguments

-nocommand

No input needed
Value: Blank

-nouserprofiledialog

No input needed
Value: Blank

-mvd-action

Action type
Value: check/report/both

-mvd-bg

No input needed
Value: Blank

-mvd-ncpu

# of cores in current machine
Value: 2

-mvd-log

File name / File name
Value: mvdLog.log

-mvd-project-name

Any string value display in report
Value: “Name”

-mvd-slide-number

Any integer value display in report
Value: “1”

-mvd-progress

File name / File name
Value: mvdprogress.log

Example

Example of a batch command.

```
"%hmbatch%" -nocommand -nouserprofiledialog -tcl "%tcl_file%"
-mvd-feature freepart -mvd-input "%input%" -mvd-output "%output%" -mvd-freepart-bom
"%bomfile%" -mvd-action both -mvd-report-scope both -mvd-bg -mvd-ncpu 1 -mvd-log
mvdLog.log -mvd-progress mvdprogress.log
```

Spot Comparison Check

Supported Batch options for Spot Comparison checks.

Mandatory Arguments

"%hmbatch%"

File path of `hmbatch.exe`
Value: hmbatch.exe

-tcl

File path `batch.tcl`
Value: batch.tcl

-mvd-feature

Function name

Value: spotweld-comparison

-mvd-base
File path of spot file A
Value: "%base%"

-mvd-variant
File path of spot file B
Value: "%variant%"

-mvd-spotcompare-bom
XML file path
Value: "%bomfile%"

-mvd-report
Report output path
Value: "%report%"

Note: The Spot file type is read from the configuration file.

Optional Arguments

-nocommand
No input needed
Value: Blank

-nouserprofiledialog
No input needed
Value: Blank

-mvd-action
Action type
Value: check/report/both

-mvd-bg
No input needed
Value: Blank

-mvd-ncpu
# of cores in current machine
Value: 2

-mvd-log
File name / File name
Value: mvdLog.log

-mvd-project-name
Any string value display in report
Value: “Name”

-mvd-slide-number
Any integer value display in report
Value: “1”

-mvd-progress

File name / File name

Value: mvdprogress.log

Example

Example of a batch command.

```
"%hmbatch%" -nocommand -nouserprofiledialog -tcl "%tcl_file%"
-mvd-feature spotweld-comparison -mvd-base "%base%" -mvd-variant "%variant%"
-mvd-spotcompare-bom "%bom%" -mvd-report "%report%" -mvd-action both
-mvd-report-scope both -mvd-bg -mvd-ncpu 1 -mvd-log mvdLog.log -mvd-progress
 mvdprogress.log
```

Batch Mode Error Codes

Error messages displayed when you use incorrect input argument.

SL	Error Code Function and Message
1	1001 Import: Missing CAD file to XML converter
2	1002 Import: Failed to import CAD
3	1003 Import: Failed to export the FE part to the file
4	1004 Import: Problem obtaining HyperMesh version
5	1005 Import: Unable to find the result file
6	1006 Import: Unable to find the XML file
7	1007 Import: Failed to export the FE part to the file
8	1008 Import: Unable to find the result file in Import BG
9	1009 Import: Select only one module.
10	1010 Import: Invalid directory name specified
12	2001 Offset: Failed in license checkout for Offset
13	2002 Offset: Unknown action type in Offset
14	2003 Offset: Failed to export the FE part to the file in Offset
15	2004 Offset: No valid modules selected
17	3001 Comparison: Unable to find the result file in BG

SL	Error Code Function and Message
18	3002 Comparison: No valid modules selected
19	3003 Comparison: Select two or more modules for reporting
20	3004 Comparison: No files in module selected. Please import the modules for comparison
21	3005 Comparison: Unable to compare two modules in parental relationship
22	3006 Comparison: Children found for Selected Module - Import cannot be done. For Compare Check - Select two or more modules
23	3007 Comparison Report: No template file specified in the configuration for excel
24	3008 Comparison Report: No template worksheet specified in the configuration for excel
25	3009 Comparison Report: Template file not found in Comparison for excel
26	3010 Comparison Report: Worksheet not found in the template for excel
27	3011 Comparison Report: Failed in license checkout
28	3012 Comparison Report: Record/Result file not found
29	3013 Comparison Report: No template row specified in the configuration
30	3014 Comparison Report: Invalid variant index
31	3015 Comparison Report: Problem obtaining HyperMesh version
32	3016 Comparison Report: No template file specified in the configuration for PPT
33	3017 Comparison Report: No layout specified in the configuration for PPT
34	3018 Comparison Report: Template file not found in PPT
36	4001 Intersection: Failed in license checkout
37	4002 Intersection: No valid modules selected in Intersection
38	4003 Intersection Report: No template file specified in the configuration for PPT
39	4004 Intersection Report: No layout specified in the configuration for PPT
40	4005 Intersection Report: Template file not found in PPT

SL	Error Code Function and Message
42	5001 Spotweld: Failed in license checkout
43	5002 Spotweld: Unable to find connector files
44	5003 Spotweld: Unable to load FE model files
45	5004 Spotweld: Unable to import model, Check Configuration
46	5005 Spotweld: Unable to merge the hm file in HyperMesh
47	5006 Spotweld: Unable to load result file
48	5007 Spotweld: Unable to load model file
49	5008 Spotweld: Unexpected action type
50	5009 Spotweld: No connector files found in the input path
51	5009 Spotweld: No valid modules selected
52	5010 Spotweld Report: Error in sphere creation
53	5011 Spotweld Report: Result Model not saved successfully. Backup in output path
54	5012 Spotweld Report: No template file specified in the configuration for PPT
55	5013 Spotweld Report: No layout specified in the configuration for PPT
56	5014 Spotweld Report: Template file not found in PPT
58	6001 Connection: Failed in license checkout
59	6002 Connection: Unable to calculate the centroid from surfaces of component
60	6003 Connection: Unable to create lines between centroid and surfaces of component
61	6004 Connection: Unable to calculate the center line direction from surface edges of component
62	6005 Connection: Unable to find the appropriate line to determine the center line direction
63	6006 Connection: Unable to read the boltnut.hm file in HyperMesh
64	6007 Connection: Unable to find the result file in BG

SL	Error Code Function and Message
65	6008 Connection: Unable to load result file
66	6009 Connection: Unable to load model file
67	6010 Connection Unexpected action type
68	6010 Connection Report: Check Result does not exist
69	6011 Connection Report: No template file specified in the configuration for PPT
70	6012 Connection Report: No layout specified in the configuration for PPT
71	6013 Connection Report: Template file not found in PPT
73	7001 Freepart: Failed in license checkout
74	7002 Freepart: Unable to load FE model files
75	7003 Freepart: Unable to read the freepartmodel.hm file in HyperMesh
76	7004 Freepart: No Model loaded
77	7005 Freepart: Unable to find the result file in BG
78	7006 Freepart: Unable to find connector files
79	7007 Freepart: Unexpected action type
80	7008 Freepart: Unable to load result file
81	7008 Freepart: Unable to load model file
82	7009 Freepart: No connector files found in the input path
83	7010 Freepart Report: No template file specified in the configuration for PPT
84	7011 Freepart Report: No layout specified in the configuration for PPT

SL	Error Code Function and Message
85	7012 Freepart Report: Template file not found in PPT

Batch File Example

Example of a Model Verification batch file.

Note:

You must edit “ALTAIR_HOME”, “datapath” path in the following example.

```
set ALTAIR_HOME=C:\Program Files\Altair\2019
set dataPath=G:\BVT\2019\data
set hmbatch=%ALTAIR_HOME%\hm\bin\win64\hmbatch.exe
set tcl_file=%ALTAIR_HOME%\hm\scripts\MVD\mvdMain\src\main\batch.tcl
set base=%dataPath%\batch\iges
set variant=%dataPath%\batch\dyna
set output=%dataPath%\comparisonres.xml
set report=%dataPath%

cd /d g:
cd %dataPath%
IF not exist %report% (mkdir %report%)

"%hmbatch%" -nocommand -nouserprofiledialog -tcl "%tcl_file%" -mvd-feature comparison
 -mvd-base "%base%" -mvd-base-type iges -mvd-variant "%variant%" -mvd-variant-type
 dyna -mvd-output "%output%" -mvd-report "%report%" -mvd-action both -mvd-report-
scope both -mvd-bg -mvd-ncpu 1 -mvd-log mvdLog.log -mvd-progress mvdprogress.log
```

Limitations

Limitations of the Model Verification tool.

- Supported only for Windows OS.
- Translation and Symmetry comparison runs in single CPU.
- UDMXML/BOM with No PID, Part name information may lead no result, it is expected to provide clean BOM file.
- Solid CAD vs Midmesh comparison needs thickness value in FE data.
- HyperView Player must be registered for the first time use or if user login changes.
- Office 2010 and above is supported for report generation.

Crash and Safety

11

Tools used for crash and safety analysis.

This chapter covers the following:

Dummy Positioning

Position a dummy model using the Dummy Browser.

Dummy Browser

Overview of the Dummy Browser.

The Dummy Browser can be accessed from the menu bar by clicking **Tools** > **Dummy**.

> **Restriction:** The Dummy Browser is available in the LS-DYNA and Radioss user profiles.

This browser is compatible with all LS-DYNA and Radioss Humanetics dummies (encrypted or not encrypted) and also with LSTC dummies.

You can undo and redo actions made in the browser using the Undo and Redo commands on the Restore toolbar.

The Dummy Browser consists of two panes. The first pane displays the dummy structure which consists of the different bodies defining the dummy model. The second pane displays the Entity Editor in which positioning parameters for each dummy articulations can be defined.

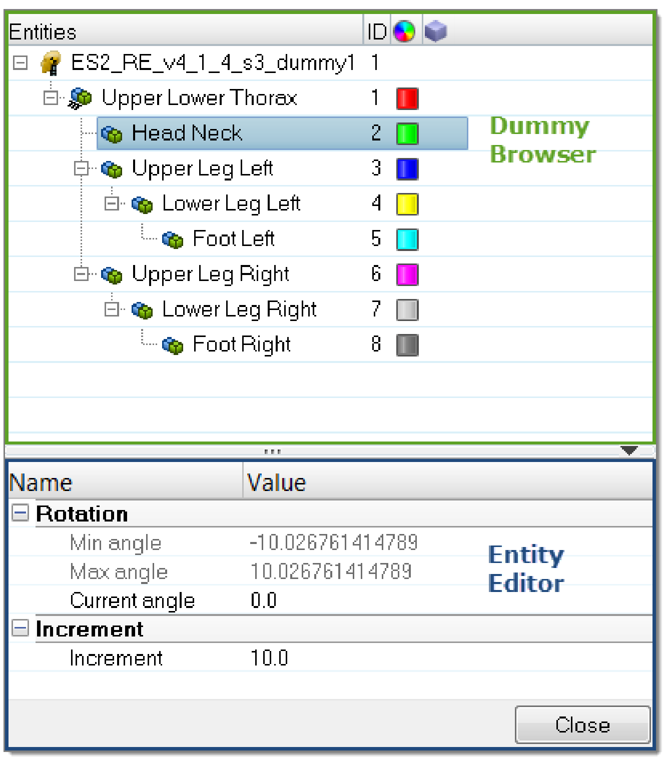


Figure 1534:

The following data is displayed in the Dummy Browser:

Column	Description
Entity	List of the dummies and dummy bodies.
ID	Displays the dummies and bodies IDs.
Color	Displays the body entity colors. Body color is different from the component color defined in the Model Browser. The body color is activated when a body is in Review mode or when display mode is set on "By Body".

Entity Editor

The Entity Editor is used to assign, modify and quickly view the attributes defined inside Dummy Browser entities.

In the Entity Editor you can define the H-Point coordinates or apply Global rotations on the dummy to position the dummy in space. The H-Point coordinates can be directly specified in the fields, or by clicking on the blue arrow, which then enables you to select a target location by picking a node in the graphics area.

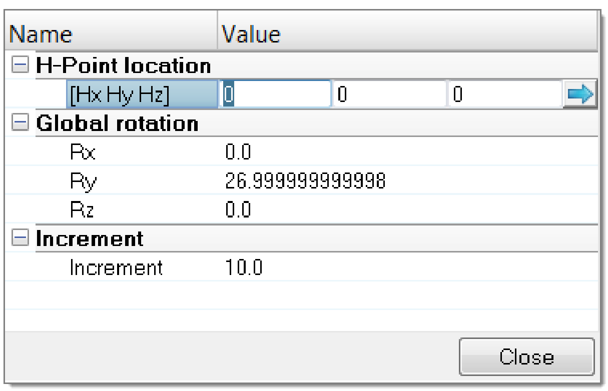


Figure 1535:

Global rotations on the dummy can also be defined. The magnitude of the rotation can be modified, using the up and down arrow buttons. Moreover, you can control the increments of the operations by changing the Increment value.

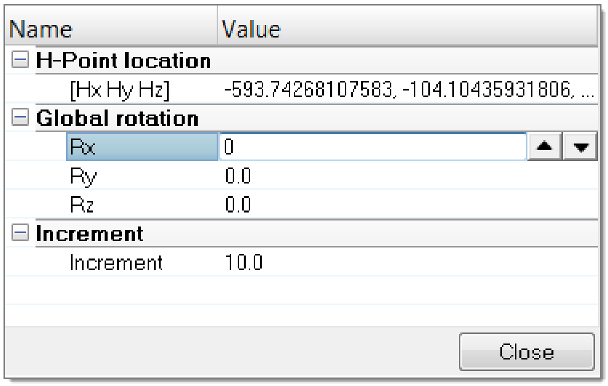


Figure 1536:

The Entity Editor is also used to modify the position of bodies. You can directly define the angle of rotation to apply in each axis of rotations in which the selected body is able to move.

The magnitude of the rotation can be also modified, using the up and down arrow buttons. Moreover, you can control the increments of the operations by changing the Increment value.

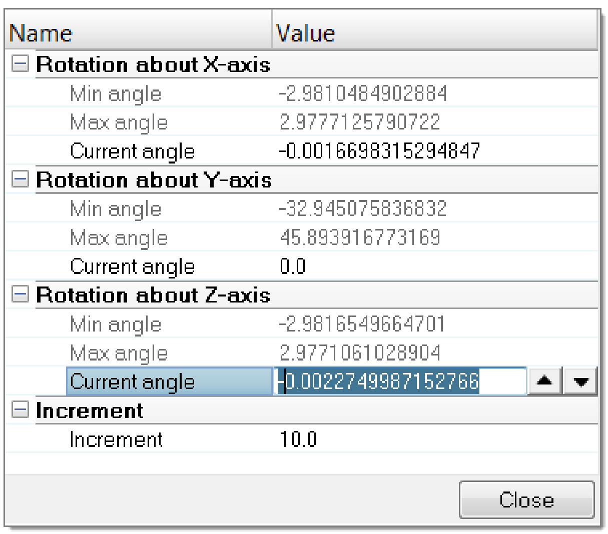


Figure 1537:

Context Menu

Option	Available for	Description
Define Position	Dummy	Save the displayed position of the dummy by creating a New position or by overwriting an existing position.

Option	Available for	Description
Retrieve Position	Dummy	Position the dummy to its initial position or to another position by selecting this in the list of the saved positions.
Move Limbs	Dummy	Open the Entity Editor to define the position of the dummy using the automatic move to target functionality.
Positioning File	Dummy	Import or export a positioning file (`*.daf`) containing the position values of the dummy.
Pre-Simulation	Dummy	Export an input deck, for the appropriate solver, in order to simulate the deformation of the dummy.
Show	Dummy and Bodies	Displays the entity in the graphics area. The entities icon changes to bold indicating that the display state is on.
Hide	Dummy and Bodies	Turns off the entity in the graphics area. This selection affects each entities local display control, that is, will make the icon become ghosted indicating the display state is off.
Isolate	Dummy and Bodies	Displays only the selected entities, and turns off all other entities of the same type.
Review	Dummy	Display the dummy articulations and bodies in a simplified display mode.

Option	Available for	Description
Reset Review	Dummy	Resets the review of the previously selected entity.

Supported Entities

The Dummy ( ) is the root of the hierarchy in the Dummy Browser. A dummy is defined by bodies ( ) representing the different kinematic assemblies of the dummy.

Position Dummies

Position the Body Manually

1. In the Dummy Browser, select a body.
2. In the graphics area, click-and-drag the manipulator to interactively modify the position of the selected body.

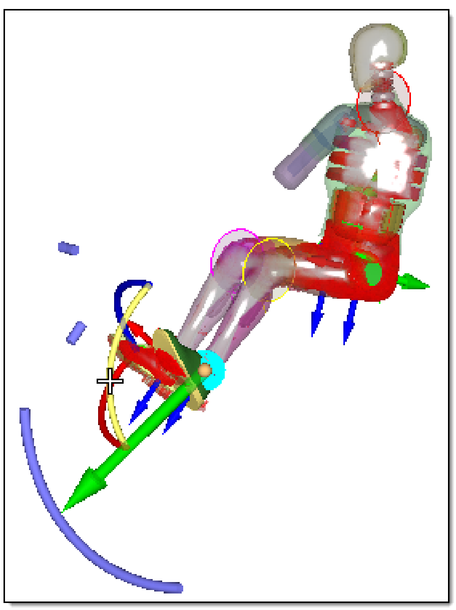


Figure 1538:

Position the Body Automatically

1. In the Dummy Browser, right-click on a body and select **Move Limbs** from the context menu.
   The Entity Editor opens.

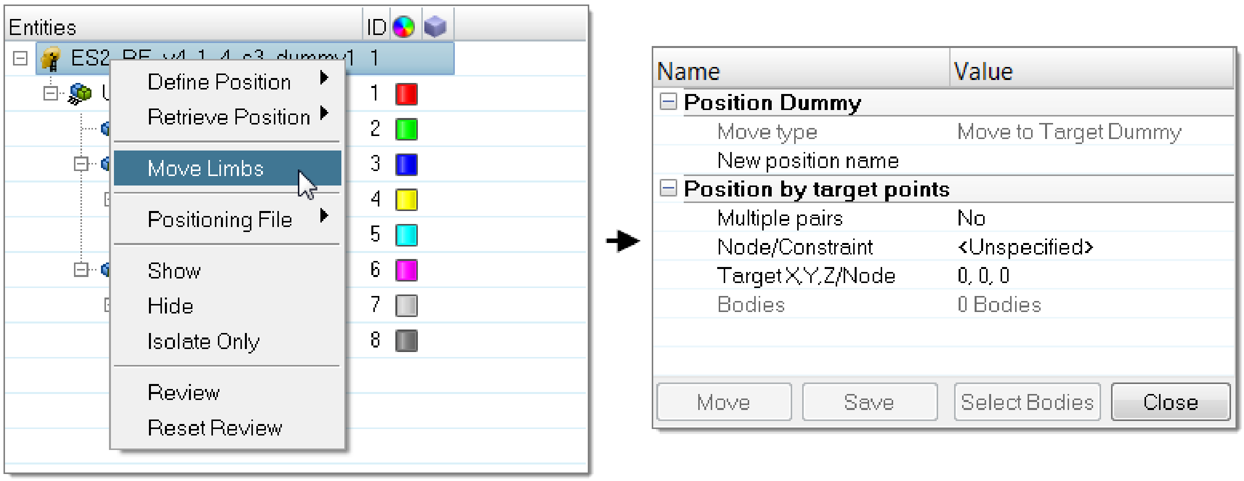


Figure 1539:

2. In the Multiple pairs field, select **No** (apply option to a single node) or **Yes** (apply option to a set of nodes and targets).
3. If Multiple pairs is set to Yes, click **Select pairs** to open the **Select multi nodes** dialog and select nodes and target point pairs.
4. Define a Node of a body.
5. Define the Target location.
6. Click **Select Bodies** to open the **Dummy Bodies DOF** dialog and select the bodies and degrees of freedom of the active bodies that will be able to move during automatic positioning.

   Select bodies by activating their corresponding checkbox, or picking them in the graphics area (right-click or left-click to activate/deactivate a body). Lock/unlock degrees of freedom by clicking the lock icon, or by picking the DOF arrows in the graphics area (right-click or left-click to activate/deactivate a DOF).

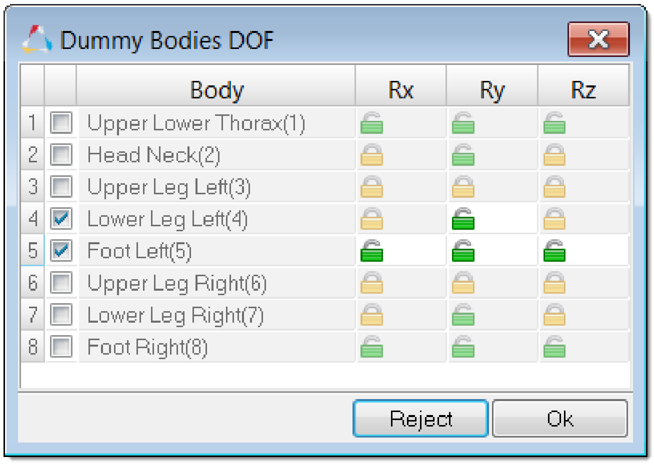


Figure 1540:

7. Click **Move** to start the automatic positioning process.

8. Click **Save** to save the achieved position.
A name will be appended to the New position name field.

Setup Pre-Simulation

Export an input deck to simulate the deformation of the dummy.

Before you begin, Position Dummies.

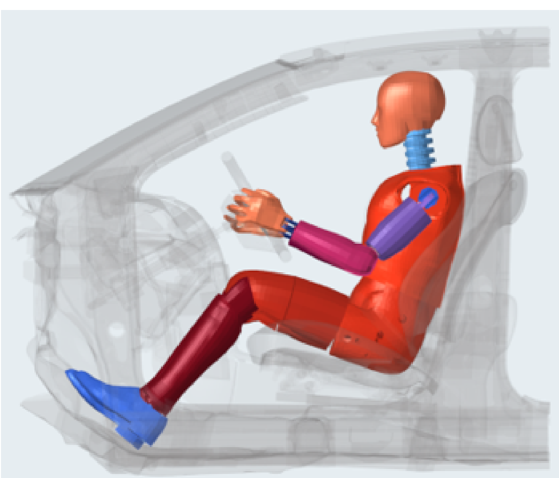


Figure 1541:

The simulation result files can be imported to update the initial FE model and thus remove the intersections and penetrations between the dummy components.

1. In the Dummy Browser, right-click and on the dummy and select **Pre-Simulation** from the context menu.

2. In the **PreSimulation Tool** dialog, define settings accordingly.
   a) In the **PreSimulation type** field, select a unit system.

      The unity system will automatically set the default simulation parameters to the correct unit. In any case, you can still modify these values manually.
   b) In the **Reference Position** field, select the Position that will define the starting position of the dummy in the pre-simulation.

      Per default, the Initial Position of the dummy is selected.
   c) In the **Export File** field, enter the pre-simulation deck name and directory.

      Per default, a deck with name "result" is exported into ~…`/Local/Temp` directory.
3. Define PreSimulation Tool Options accordingly.
4. Optional: Import the simulation result file to update the initial model, which allows the dummy node's coordinates to be updated and element initial stress state to be defined.
   - For LS-DYNA, click **Import dynain File** to find the `.dynain` file.
   - For Radioss, click **Import h3d File** to find the `.h3d` file.
5. Click **Export**.

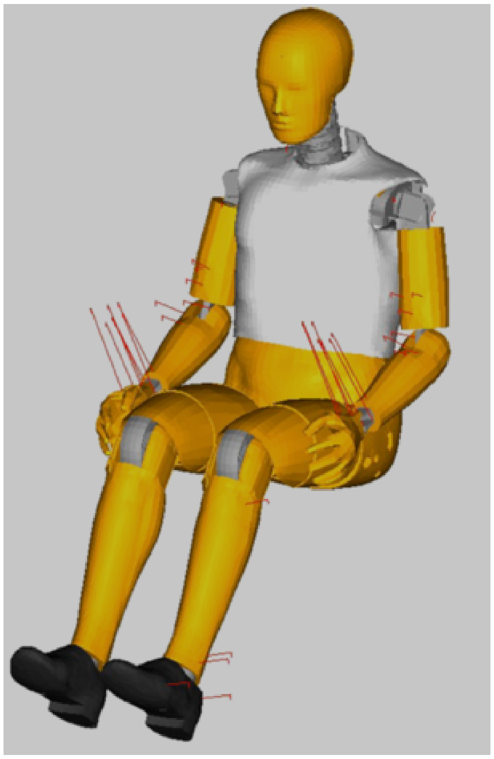
The pre-simulation deck is exported to the specified location.

During the export process of the dummy pre-simulation deck, HyperMesh exports the complete dummy model only, positioned in the selected Reference Position, per default, the Initial Position of the dummy.

The method used for the pre-simulation is known as the "cable" method, which uses 1D elements connected to dummy bodies, in order to pull them from their initial position to their final position.

All "cables" are automatically created on each dummy body and all boundary conditions needed for the pre-simulation.

Figure 1542: Exported Dummy Pre-Simulation Model, with Cable Elements Attached to Dummy Bodies



The attachment nodes of the cables, nodes "N1; N2; N3" in the dummy assembly keyword are used, if they are defined.

Figure 1543: Radioss

```
/ASSEMBLY/11
LOWER ARM RIGHT WRIST
#    subset      grnod      childs     presim
          1          1           1          1
#subset id
   1501067
# grnod id
   1500522
#     child      Joint        node       dofs
         12    1560030     1500107          1-63.999752 116.00021
#        N1         N2          N3
   1582795    1582796     1582745
```

```
*ASSEMBLY
         2Head And Neck
$
$:#S.PARTs     #PARTs      #kids  #S.NODEs      restr       csys     #conts  dyna_pos
         1          0          0         0          0          0          0         1
$
$: List of SET_PARTs
   1502001
$
$:       N1         N2          N3
   1557573    1566335     1556898
```

Figure 1544: LS-DYNA

In case the nodes “N1; N2; N3” in the dummy assembly keyword are not defined, HyperMesh automatically detects the best three nodes to use on each body. In this case, you will receive a message.

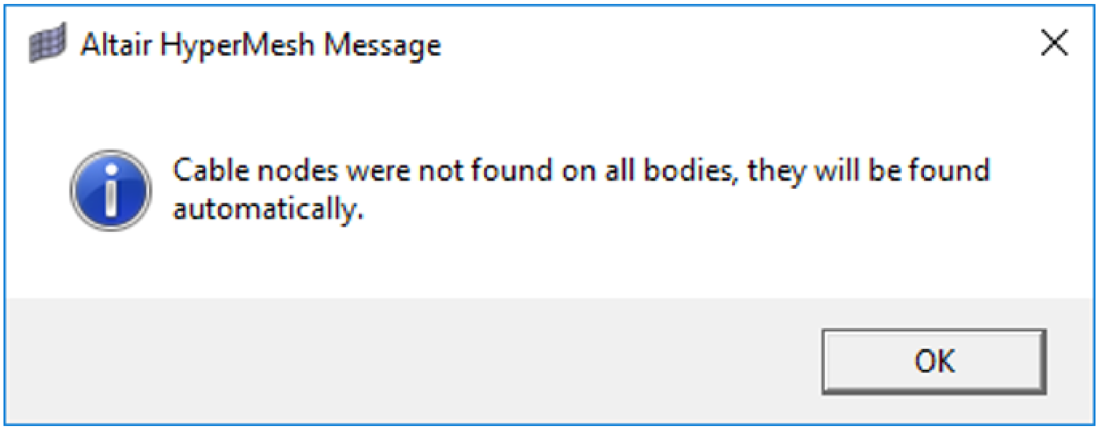


Figure 1545: Altair HyperMesh Message

After the simulation of the dummy positioning, the components are deformed and intersections between components are removed.

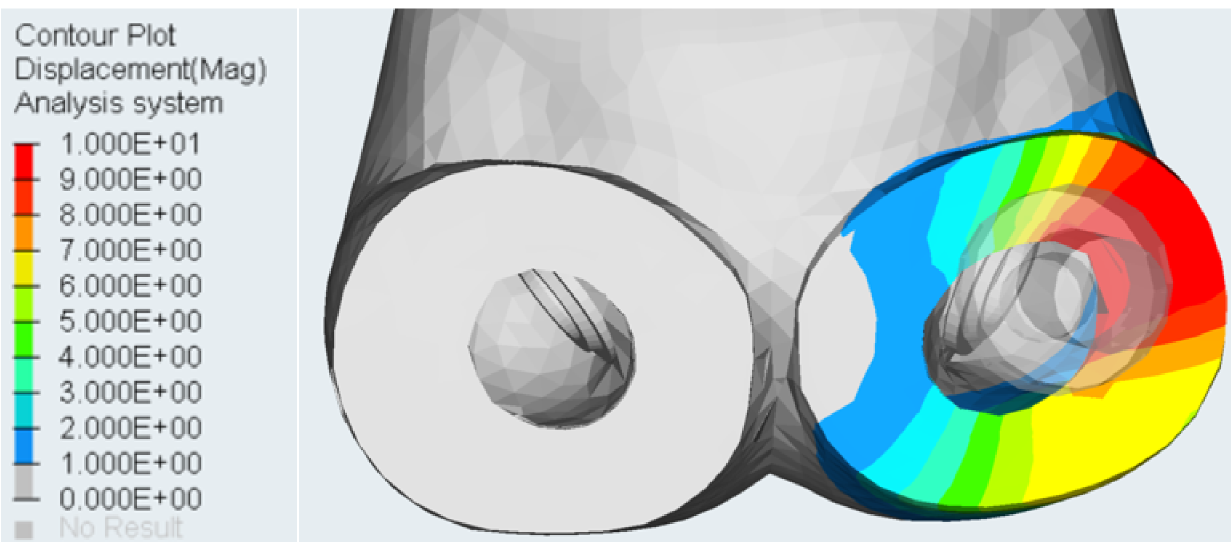


Figure 1546:

The "cable" method guarantees a perfect final position matching with the final position defined in the Dummy Browser.

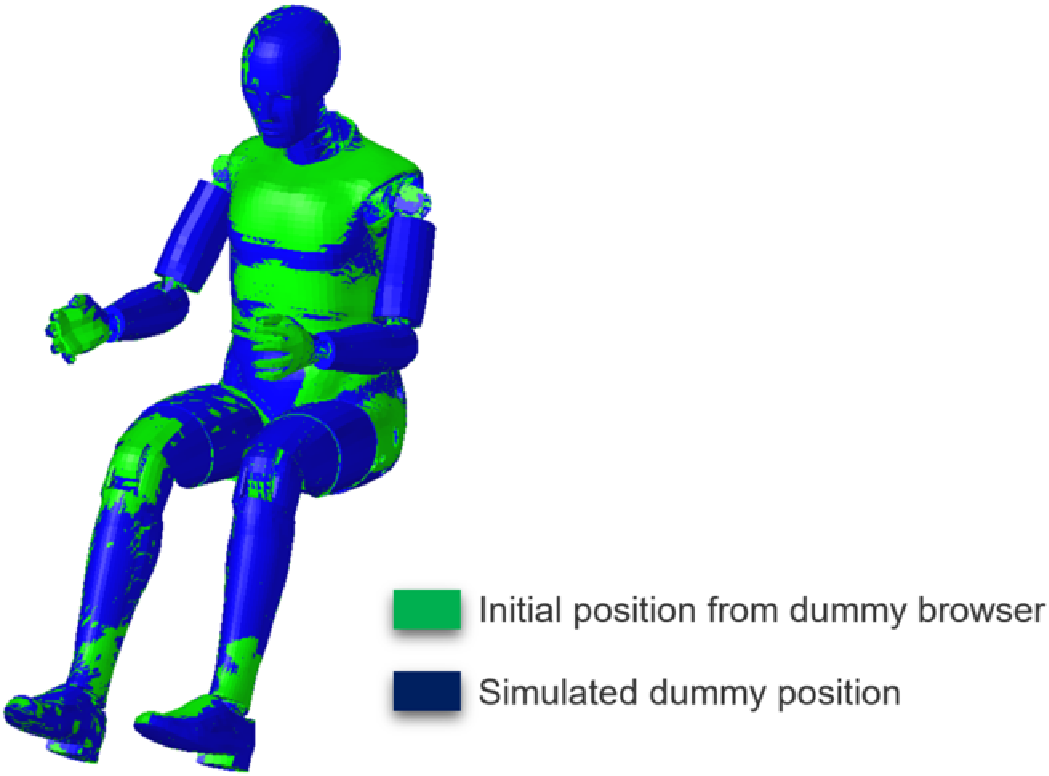


Figure 1547:

PreSimulation Tool Options

Overview of supported options in the PreSimulation Tool.

LS-DYNA

Simulation Parameter

Simulation Time
: Define the total simulation time in *CONTROL_TERMINATION for the pre-simulation.
Default value = 250ms.

Time Step
: Define the control time step value in *CONTROL_TIMESTEP for the pre-simulation.
Default value = 0.001ms.

Force in cables
: Define the pre-tension force to be applied on the cable elements in *MAT_CABLE_DISCRETE_BEAM.
Default value = 1.0 kN.

Force ramp up time
: Define the ramp-up time for the pre-tension force in *MAT_CABLE_DISCRETE_BEAM.
Default value = 10.0ms.

Damping on cables
Define the damping value on the discrete elements in *MAT_DAMPER_VISCOUD.
Default value = 0.5.

Global damping value
Define the system damping constant in *DAMPING_GLOBAL.
Default value = 0.05.

Initial Stress Results

Import *INITIAL_STRESS_SOLID
Import the initial stresses for solid elements from the `.dynain` file.

Import *INITIAL_STRESS_SHELL
Import the initial stresses for shell elements from the `.dynain` file.

Import *INITIAL_STRESS_BEAM
Import the initial stresses for beam elements from the `.dynain` file.

Radioss

Simulation Parameters

Generate XREF for initial stresses
Create /XREF cards for the dummy components which are compatible with this RADIOSS feature in terms of material type and element formulation. The XREF cards are generated within the original session during the export of the pre-simulation deck, and not during import of `.h3d` file.

Simulation time
Define the total simulation time in /RUN card for the pre-simulation.
Default value = 250ms

Time Step
Define the control time step value in /DT/NODA/CST card for the pre-simulation.
Default value = 0.001ms.

Global damping value
Define the system damping constant in /DAMP card.
Default value = 0.05

Create rigids for end bodies
Automatically rigidify the end bodies (feet, hands, head) of the dummy during the pre-simulation.

Read Positioning Files

In the Dummy Browser, right-click on the dummy root name and select **Positioning File** > **Import** from the context menu.
Once the positioning file (`*.daf`) is read, the dummy will automatically be positioned according to the imported position data.

Seat Mechanism

Create and articulate a kinematic mechanism based on FE mesh using the Mechanism Browser.

Mechanism Browser

Overview of the Mechanism Browser.

Access the Mechanism Browser from the menu bar by clicking **Tools** > **Mechanism**.

> **Restriction:** The Mechanism Browser is available in the LS-DYNA and Radioss user profiles.

You can undo and redo actions made in the Mechanism Browser using the Undo and Redo commands on the Restore toolbar.

The Mechanism Browser consists of two panes. The first pane displays the mechanism structure which consists of joints, bodies, and constraints. The second pane displays the Entity Editor.

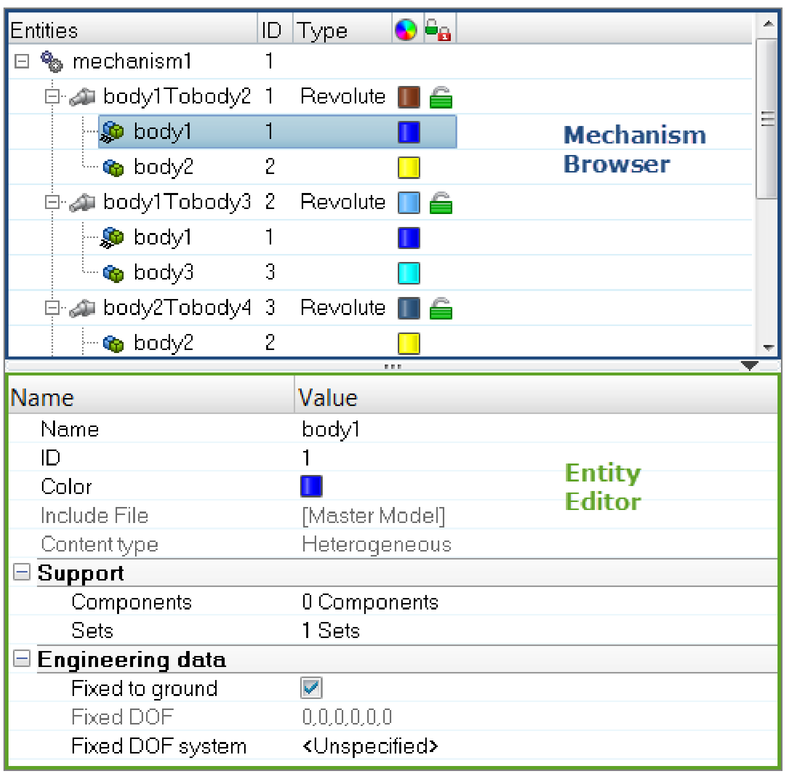


Figure 1548:

The following data is displayed in the Mechanism Browser:

Column	Description
Entity	Lists the mechanism, joints, bodies, and constraints in your model.

Column	Description
ID	Displays the mechanism, joints, bodies, and constraints IDs.
Type	Displays the joint or constraints type.
Color	Displays the joint and body entity colors. Body color is different from the component color set-up in the Model Browser. The body color is activated when a body is in Review mode.
Lock level	Displays the lock level of a joint (green = free joint, yellow = locked joint).

Entity Editor

The Entity Editor is used to assign, modify and quickly view the attributes defined inside Mechanism Browser entities.

For example, once you create a new body or select a body in the browser, the Entity Editor opens and displays the bodies corresponding attributes, which you can view and modify.

From the Entity Editor, you can also create or edit an entity assigned to the selected body by right-clicking on the entity assignment field and selecting **Create** or **Edit**, respectively.

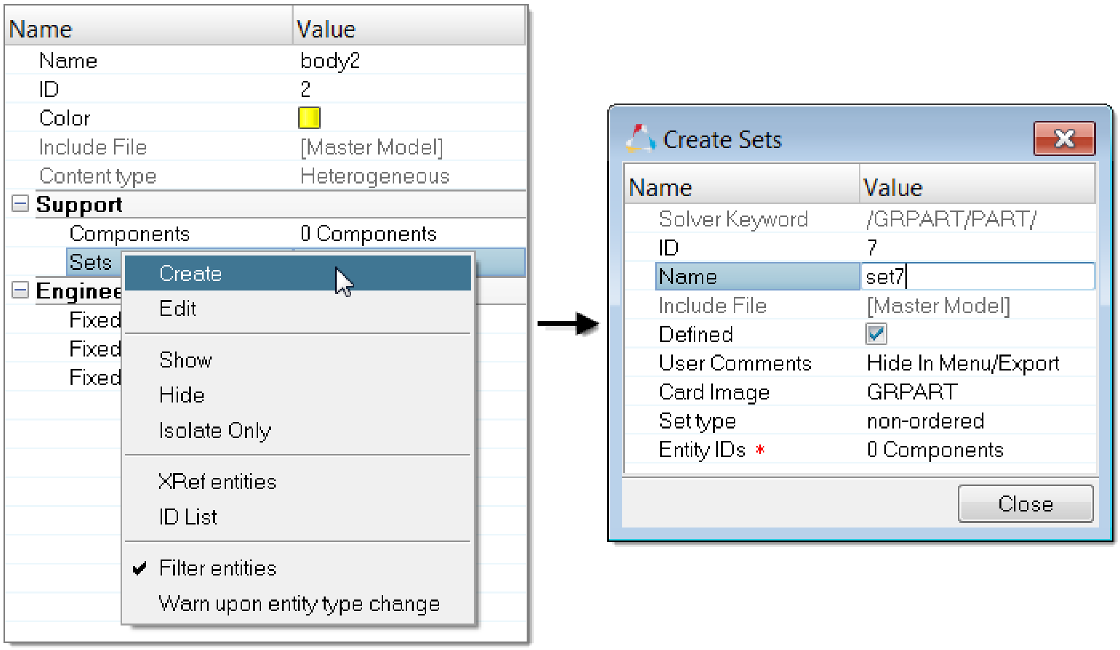


Figure 1549:

Context Menu

Option	**Available for**	**Description**
Actuate Mechanism	Mechanism	Gives the ability to articulate a mechanism by defining one of the following actions: • Translate a node • Rotate a body • Move a node to a target location
Check Mechanism	Mechanism	Enables you to check the validity of a mechanism, looking for common nodes, empty bodies, and redundant joints.
Collapse	All entities	Closes all of the folders for the selected mechanism, so that only the mechanism item displays.
Collapse All	In Mechanism Browser	Closes all of the folders in the browser, so that only the mechanism items display.
Column Visibility	Mechanism and in Mechanism Browser	Enables you to hide or show the ID, Type, Color, Lock, Common nodes, Empty Body

Option	Available for	Description
		and Redundant Joint columns in the browser.
Create	Mechanism and in Mechanism Browser	Create a new mechanism, body, or constraint. You can only create a body or constraint by right-clicking on a mechanism in the browser.
Delete	All entities	Deletes the selected entity. Shortcut: To delete a selected entity in the Mechanism Browser, press Delete.
De-Link Dummy	Mechanism	When a dummy has been linked to a mechanism, this option is activated in order to remove the link between the dummy and the mechanism.
Expand	All entities	Opens the selected folder, exposing every entity nested at every level.
Expand All	In Mechanism Browser	Opens all of the folders in the browser, exposing every entity nested at every level.
Export Position	Mechanism	Opens the **Export Position** dialog, from which you can export a saved position of the selected mechanism into a solver file containing the corresponding solver transformation cards.
Hide	All entities	Turns off the entity in the graphics area. This selection affects each item's local display control, that is, it will make the icon become ghosted indicating the display state is off.
Isolate	All entities	Displays only the selected entities, and turns off all other entities of the same type.
Link to Dummy	Mechanism	In case a dummy is present in the session, this option is activated in order to link the dummy, for example, to a seat mechanism.

Option	Available for	Description
Lock/Unlock Joint	Joint	Block (Lock) all degrees of freedom of the selected joint or make them free again (Unlock).
Move	Joint	Opens the Entity Editor to define the positioning values of the selected joint and activate the joint manipulator in the display that can be used to interactively move the joint.
Move Mechanism To	Mechanism	Moves the selected mechanism to its initial position or other positions, which have been previously saved.
Review	All entities	Invokes Review mode, which displays selected entities irrespective of their display state, masked, active state (Entity State Browser), but not outside of the spherical clipping (if enabled). By selecting a body, it will be displayed with its body color while other entities are grayed out and displayed with transparency. By selecting a joint or a mechanism, the joint marker with the bodies links between will be displayed.
Reset Review	All entities	Resets the review of the previously selected entity.
Rename	All entities	Rename an entity in the Name field. The new name must be unique. All instances of the renamed entity will update automatically. You can cancel the rename operation by pressing Esc.
Save Position as	Mechanism	Save the displayed mechanism position as a new position or replace an already existing position.
Show	All entities	Displays the selected entity in the graphics area. The entities icon changes to bold indicating that the display state is on. Using the Show option on a

Option	Available for	Description
		mechanism or joint will display all nested entities inside.
XRef Entities	All entities	Opens the References Browser and displays the relationship of the selected entities to other entities in the model in a hierarchical tree structure. Any single entity or multiple entities can be selected.

Supported Entities

A Mechanism () is the root of the hierarchy in the Mechanism Browser. To create a Mechanism, right click in the Mechanism Browser and select **Create** > **Mechanism** from the context menu. A mechanism is defined by the following entities:

- Bodies () define a kinematic assembly made of FE parts or nodes, which can be selected in the Entity Editor. To create a body, right-click on the Mechanism and select **Create** > **Body** from the context menu.
- Joints () define the kinematic relationship between two bodies (for the following joint types: Ball, Cylinder, Revolute, Slider) or three bodies (for the DoubleSlider joint). To create a joint, select two bodies to connect in the browser, then right-click and select **Connect** from the context menu. For the DoubleSlider joint, the third body has to be defined in the Entity Editor.
- Constraints () define kinematical constraints on a body at a specified node or point location.

Supported Keywords Exported in Solver Deck

The mechanism information is embedded in the input deck using keywords following the `/END` (for Radioss) or *END (for LS-DYNA).

For Radioss, a mechanism is defined between the `/MECHANISM_START` and `/MECHANISM_END` keywords, and is composed of:

- `/ASSEMBLY` defines a body with the following attributes:
  - Assembly ID
  - Assembly name
  - Number of part sets and related set IDs
  - Number of parts and related part IDs
  - Number of node sets and related set IDs

    - Locked degrees of freedom
    - Optional local coordinate system
- `/CONNECTION` defines a joint between assemblies with the following attributes:
    - Connection name
    - Assembly ID #1
    - Assembly ID #2
    - Connection position (defined by Node IDs or connection position coordinates)
    - Joint limits in + and - directions
    - Current joint distance/angle value
    - Lock level
    - Assembly ID #3 (for the Double Slider only)
    - Scale factors for relative displacement between Assembly #3 and Assembly #1 and #2
- `/POSITION` defines, for a stored position, the position information for the mechanism's assemblies with the following attributes:
    - Position name
    - Assemblies position matrixes

    > **Note:** The Reference and Initial positions should never be deleted inside the mechanism.

- `/CHILD_DUMMY` defines the coupling between a mechanism and a dummy with the following attributes:
    - Name
    - Master assembly ID of the mechanism
    - Dummy ID
    - Number of child assemblies of the dummy
    - Degrees of freedom linking master to child
    - Child assembly IDs of the dummy

The following connection keywords can be defined, depending on the type of joint created in the Mechanism Browser:

- `/CONNECTION_PIN` defines a ball joint
- `/CONNECTION_HINGE` defines a revolute joint
- `/CONNECTION_LINE` defines a cylindrical joint, or a double slider joint if assembly ID #3 is defined or a slider joint has the #HM_SLIDER_JOINT comment defined before the name of the connection

For LS-DYNA, the same keywords are used but their syntax begins with * and not /.

Creating a Mechanism

This task explains how to create a mechanism.

In HyperMesh 2019 there are two ways to create a mechanism: Manual and Auto Generate. In this task, you will create a mechanism using the Auto Generate method.

- Using the Manual approach, you create the mechanism manually by defining the bodies and joints.
- Using the Auto Generate approach, HyperMesh will automatically create bodies and joints based on a selection of components and elements of the FE model.

In the Mechanism Browser, right-click and select **Create > Mechanism > Auto Generate.**

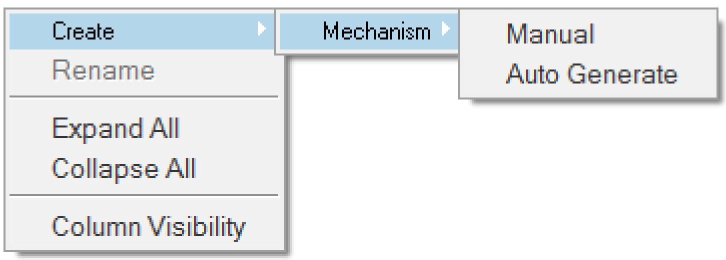


Figure 1550:

When you select Auto Generate, you will see the **Mechanism Builder** dialog box.

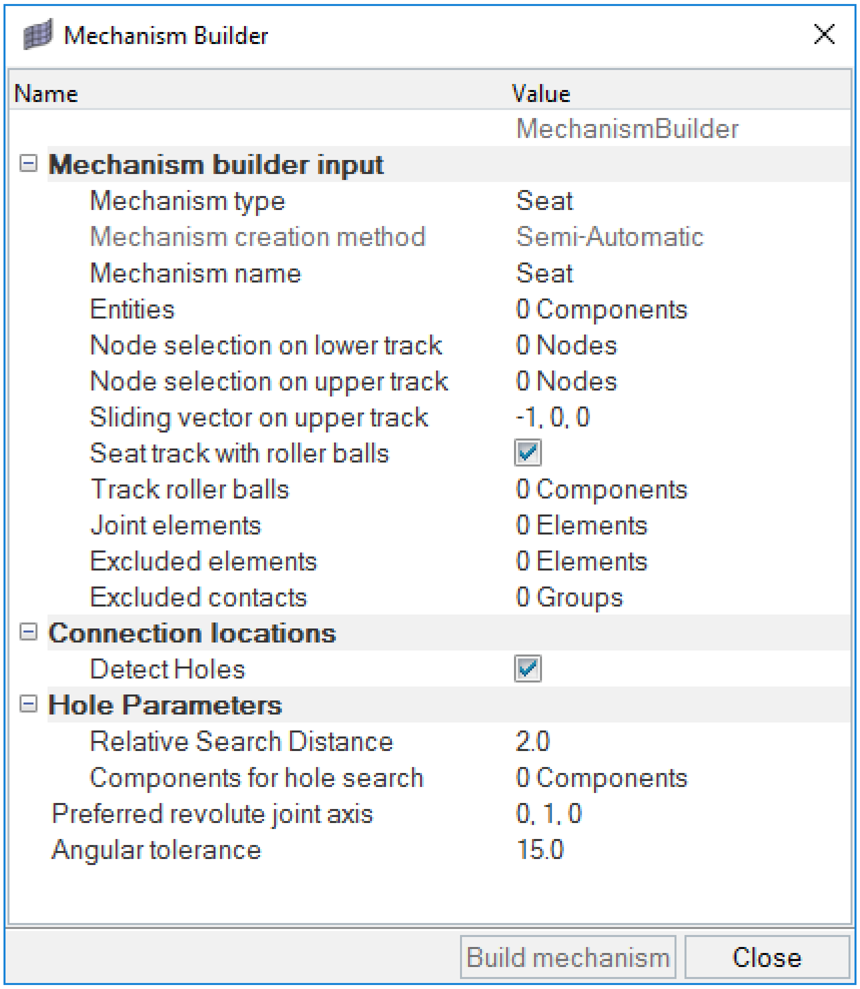


Figure 1551:

To enable an extraction of the mechanism, each field in the dialog must be defined or selected. Set the options according to the Mechanism Builder Options table.

Mechanism Builder Options Table

Use this table to set options for the Mechanism Builder.

Mechanism Builder Options	
Option	**Action**
Mechanism Type	Select the type of mechanism, either **Seat** or **General**. Your selection will affect other options in the dialog box.
Mechanism Name	Type the name you want to give to the mechanism after the extraction is completed.

Mechanism Builder Options	
Option	**Action**
Entities	Select the FE components that define the mechanism. You can choose **Components, Mechanisms, Assemblies**, or **Sets**.
Node Selection on Lower Track	When the Mechanism Type is set to **Seat**, you can select nodes on the lower track components of the seat.
Node Selection on Upper Track	When the Mechanism Type is set to **Seat**, you can select nodes on the upper track components of the seat.
Sliding Vector on Upper Track	When the Mechanism Type is set to **Seat**, you can specify the translation direction of the joint defined for the seat track.
Seat Track with Roller Balls	When the Mechanism Type is set to **Seat**, you can specify whether the seat track contains roller balls. If you check this option, HyperMesh will automatically create a double slider joint for the seat track, and an additional selector for the seat track roller will appear.
Track Roller Balls	When the Mechanism Type is set to **Seat**, and Seat Track with Roller Balls is checked, this option allows you to select the roller ball components that will be defined as 3rd body in the double slider joint.
Joint Elements	Select the FE elements (springs or beams) that have to be created as joints in the mechanism extraction process.
Excluded Elements	Select the FE elements that should be excluded from the extraction process of the mechanism. For example, nodal rigid bodies that are used to link components in the FE model but that are not physical conditions, can be selected here and will not be considered in the extraction process.
Excluded Contacts	Select to exclude tied contacts that may be defined on the FE model but don't represent physical connections. This excludes them from the extraction logic.
Detect Holes	Select to activate the hole detection parameters. This allows you to detect joint positions by looking for concentric holes in components.
Relative Search Distance	When Detect Holes is checked, this option defines the distance of search for concentric holes that define a joint location.
Components for Hole Search	When Detect Holes is checked, this option allows you to select the components on which the hole detection must be performed.

Mechanism Builder Options	
Option	**Action**
Preferred Revolute Joint Axis	Type the vector that defines the revolute joint axis. This will overwrite the direction of the joint elements.
Angular Tolerance	Type the angular tolerance into this field. The joint axis within this angle will be overwritten by the Preferred Revolute Joint Axis value.

Position Mechanisms and Joints

Overview of how to modify the position of mechanism and joints.

Modify the Position of Mechanisms

1. In the Mechanism Browser, right-click on a mechanism and select **Actuate Mechanism** from the context menu, then choose from one of the following:
    - Choose **Translate** to open the Entity Editor and select a Node/Constraint, give a Direction of motion, and define the Magnitude of the translation to apply.

      The Magnitude can also be modified, using the up and down arrow buttons. Moreover, you can control the increments of the operations by changing the Increment value.

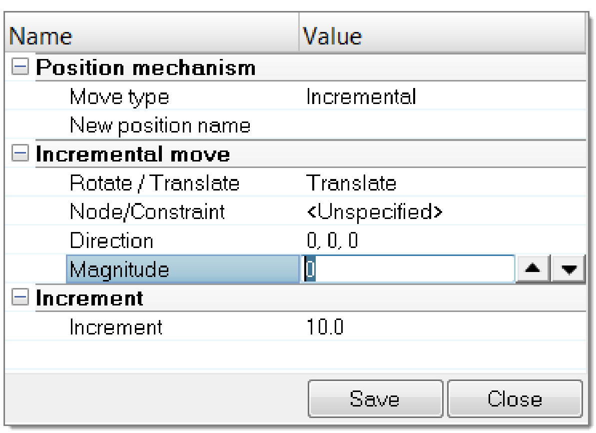
      

      Figure 1552:

    - Choose **Rotate** to open the Entity Editor, select a Body, define a Rotation axis, a rotation center (Base point), and provide the Magnitude of the rotation to apply to the body.

      The Magnitude can also be modified, using the up and down arrow buttons. Moreover, you can control the increments of the operations by changing the Increment value.

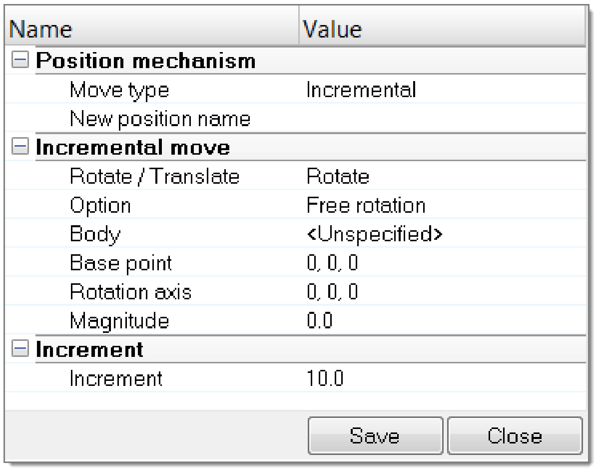


Figure 1553:

- Choose **Target Point** to open the Entity Editor, select a Node and define its Target location. Click **Move** to calculate the position of each body to bring the selected node to the target location or the nearest location, depending on joints limit values.

  This option can be applied on a set of nodes and targets, by setting Multiple pairs to **Yes**.

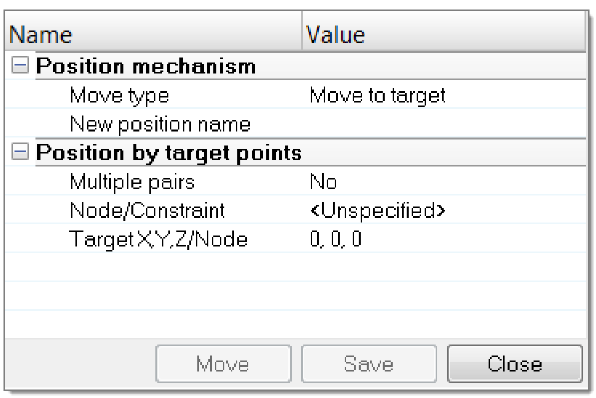


Figure 1554:

2. Click **Save** to save the achieved position.
A name will be appended to the New position name field.

Modify the Position of Joints

1. In the Mechanism Browser, right-click on a joint and select **Move** from the context menu.

2. Modify the position of the joint.

- In the graphics area, click-and-drag the manipulator.

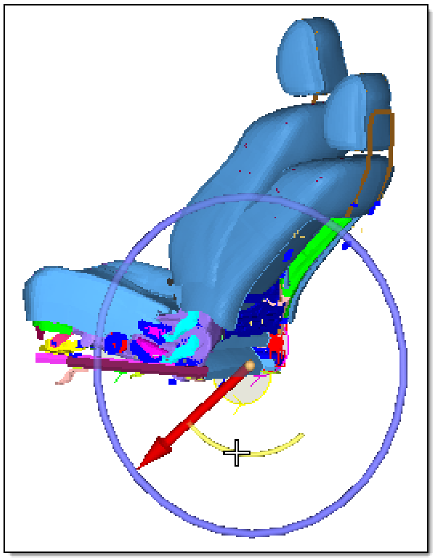


Figure 1555:

- In the Entity Editor, modify the Current angle of the joint.

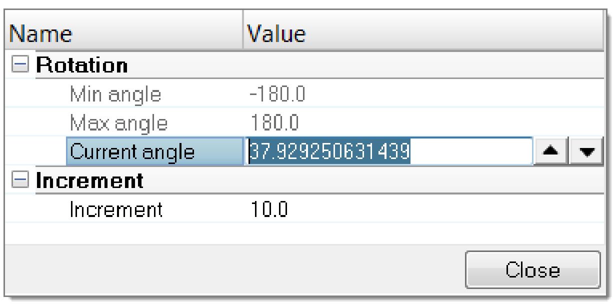


Figure 1556:

3. Change the Increment value to control the increments of the operation.

Link a Dummy to a Mechanism

In a vehicle FE model, quickly and efficiently set kinematic systems such as seats, steering wheels and pedals in different positions using the Mechanism Browser.

A dummy can then be positioned on the seat and may have its hands placed on the steering wheel and feet on the pedals. To couple a dummy to a seat mechanism the degree of freedom of one or more bodies of the dummy ("child" bodies) must be linked to a body of the seat mechanism ("master" body). The motion of the master body will drive the motion of the child bodies; however, the motion of the master body may be limited by the angle limits of the dummy joints.

1. In the Mechanism Browser, right-click on the seat mechanism and select **Link to Dummy** from the context menu.
2. In the **Dummies** dialog, select a dummy to link to the seat mechanism and click **Next**.

3. In the **Mechanism bodies (Master body)** dialog, select the master body from the seat and click **Next**.

   > **Note:** For this use case, it is common to select a seat cushion.

4. In the **Dummy bodies (child bodies)** dialog, select the dummy slave bodies and the linked degrees of freedom and click **Close**.

   > **Note:** For this use case, it is common to link to the lower torso of the dummy in degrees of freedom Tx, Ty, and Tz (selected per default).

5. To fix some of the bodies of the dummy, for example, hands and feet, define constraints on those bodies by selecting **Create** > **Constraint** > **Point Node** or **Point Location** from the context menu.
6. Position the seat mechanism.

   The position of the dummies limbs will be automatically updated.

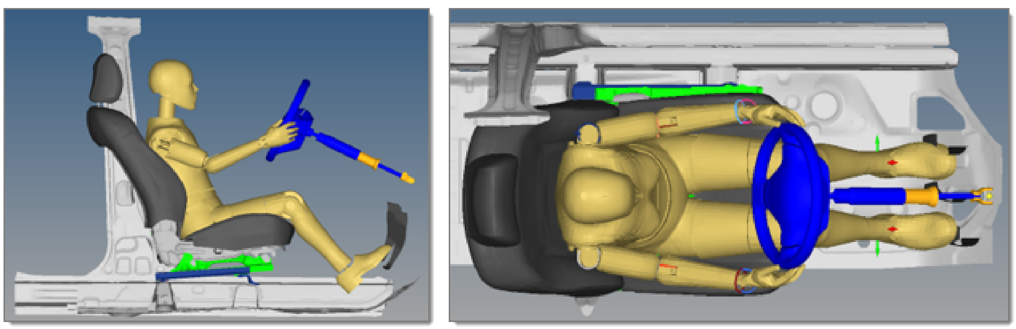


Figure 1557: Initial State

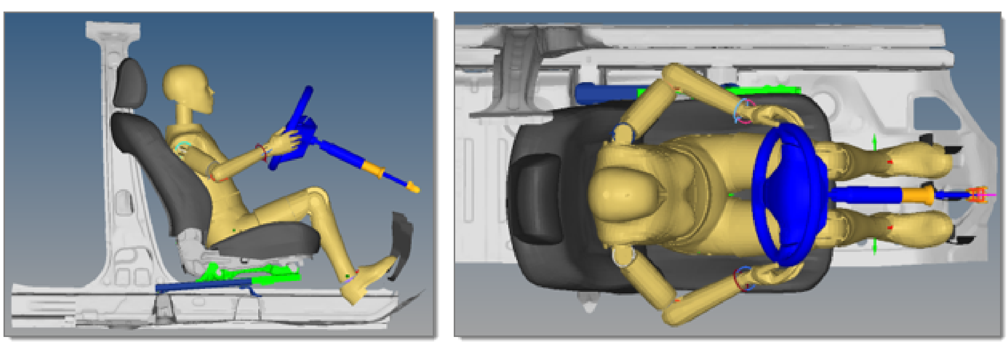


Figure 1558: After Translation of 100mm of the Seat

Pre-Simulation (Seat Deformer) Setup

Export an input deck, for the appropriate solver, in order to simulate the deformation of the seat under the dummy. The simulation result files can be imported to update the initial FE model and remove the intersections and penetrations between the dummy and the seat.

When a dummy is positioned to its H-Point location on a seat, intersections between the dummy and seat foam parts generally occurs.

1. From the menu bar, click **Tools** > **Pre-Simulation (Seat Deformer)**.

   > **Restriction:** Only available in the LS-DYNA and Radioss user profiles.

   The **PreSimulation** tool opens.
2. Define pre-simulation setup settings.
3. Click **Export**.

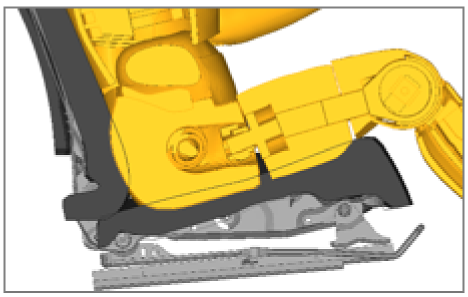


Figure 1559:
Shows the initial model with the dummy positioned at its correct H-Point location, and shows intersections with the seat.

The selected dummy components move following the direction that you move them in until no penetrations between the dummy and seat components are detected. The dummy components are rigidified. A contact between the seat and dummy is automatically created with user defined parameters as well as a fix boundary condition on the selected fixed nodes of the seat. Finally, an imposed displacement is applied on the dummy components to bring them back to the initial position.

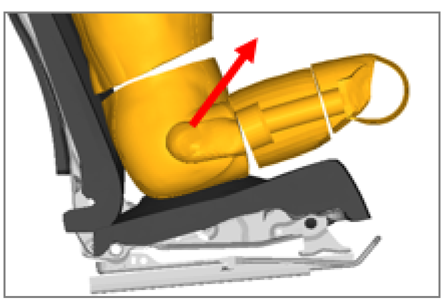


Figure 1560:

The pre-simulation load case is ready and the dedicated input deck is exported to the user specified location, as shown below. When the solver analysis is done, you can import the dedicated solver output file to update the initial model with the deformed geometry and initial stresses of the seat parts.

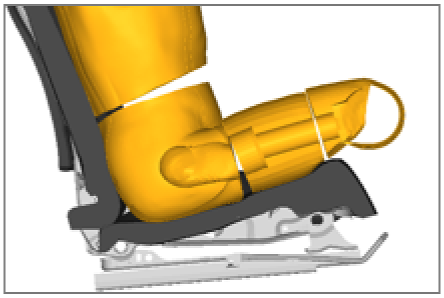


Figure 1561:

PreSimulation Tool

Overview of the PreSimulation Tool user interface.

LS-DYNA

Option	Description
Unit System	Selecting a unit system will automatically set up the default simulation parameters to the correct unit. In any case, you can still modify these values manually.
Dummy	Dummy components that are selected in contact with the seat will be exported for the pre-simulation. The selection can be done via: **Components** **Mechanism** Opens a dialog to select a complete dummy **Assemblies** **Set** Opens a dialog to select a set of type *SET_PART_LIST
Seat	Select the seat components that will be exported for the pre-simulation. The selection can be done via: **Components** **Mechanism** Opens a dialog to select a mechanism defining the seat. Only the bodies that are defined by components or sets of components will be taken into account. **Assemblies** **Set** Opens a dialog to select a set of type *SET_PART_LIST

Option	Description
Fixed Nodes of Seat	Select the nodes that will be fixed during the simulation. The selection can be done via: **Nodes** **Set** Opens a dialog to select a set of type *SET_NODE_LIST
Dummy Displacement Direction	Defines the vector in which the dummy components will be moved to the starting simulation position when penetrations with the seat are no longer detected.
Export Files	Defines the pre-simulation deck name and location.
Dummy Velocity	Defines the imposed velocity value applied on the dummy parts during the simulation. Default value = 0.1mm/ms.
Time Step	Defines the control time step value. Default value = 0.001ms.
Scale Materials Density	Scale factor to scale the material density values in order to increase the critical time step of the model. Default value = 10.
Dummy Displacement Step	Displacement increment used to move the selected dummy components along the prescribed vector. After each increment, the penetration check between the dummy and seat will be performed and the dummy motion will be stopped when penetrations are no longer detected. Default value = 10.
Soft	Defines the Soft parameter value in the generated contact between the dummy and seat. Default value = 2.
Friction	Defines the FS parameter value in the generated contact between the dummy and seat. Default value = 0.0.
Imposed Contact Thickness	Defines the value of parameters SST, MST and SLDTHK in the generated contact between the dummy and seat. Default value = 0.0.
Contact Stiffness Scale Factor	Defines the value of parameter SFS in the generated contact between the dummy and seat. Default value = 0.0.
Import *INITIAL_STRESS_SOLID	If this checkbox is activated, initial stresses for solid elements will be imported from the dynain file. This option is activated by default.
Import *INITIAL_STRESS_SHELL	If this checkbox is activated, initial stresses for shell elements will be imported from the dynain file. This option is deactivated by default.
Import *INITIAL_STRESS_BEAM	If this checkbox is activated, initial stresses for beam elements will be imported from the dynain file. This option is deactivated by default.

Option	Description
Export	Starts the export process of the pre-simulation deck at the specified location.
Import dynain File	Allows the LS-DYNA simulation to update the initial model.
Close	Closes the dialog.

Radioss

Option	Description
Unit System	Selecting a unit system will automatically set up the default simulation parameters to the correct unit. In any case, you can still modify these values manually.
Dummy	Dummy components that are selected in contact with the seat will be exported for the pre-simulation. The selection can be done via: **Components** **Mechanism** Opens a dialog to select a complete dummy. **Assemblies** **Set** Opens a dialog to select a set of type /GRPART.
Seat	Select the seat components that will be exported for the pre-simulation. The selection can be done via: **Components** **Mechanism** Opens a dialog to select a mechanism defining the seat. Only the bodies that are defined by components or sets of components will be taken into account. **Assemblies** **Set** Opens a dialog to select a set of type /GRPART.
Fixed Nodes of Seat	Select the nodes that will be fixed during the simulation. The selection can be done via: **Nodes** **Set** Opens a dialog to select a set of type /GRNOD.

Option	Description
Dummy Displacement Direction	Defines the vector in which the dummy components will be moved to the starting simulation position when penetrations with the seat are no longer detected.
Export File	Defines the pre-simulation deck name and location.
Generate XREF for initial stresses	Enables the creation of /XREF cards for the seat components that are compatible with this Radioss feature in terms of material type and element formulation. This checkbox is activated by default.
Dummy Velocity	Defines the imposed velocity value applied on the dummy parts during the simulation. Default value = 0.1mm/ms.
Time Step	Defines the control time step value. Default value = 0.001ms.
Scale Materials Density	Scale factor to scale the material density values in order to increase the critical time step of the model. Default value = 10.
Dummy Displacement Step	Displacement increment used to move the selected dummy components along the prescribed vector. After each increment, the penetration check between the dummy and seat will be performed and the dummy motion will be stopped when penetrations are no longer detected. Default value = 10.
Friction	Defines the friction value in the generated contact between dummy and seat. Default value = 0.0.
Stmin	Defines the minimum contact stiffness in the generated contact between dummy and seat. Default value = 1.0kN/mm.
Gap_Min	Defines the minimum gap in the generated contact between dummy and seat. Default value = 0.5.
IGAP	Defines the IGAP value for the generated contact between dummy and seat. Default value = 2.
Export	Starts the export process of the pre-simulation deck at the specified location.
Import h3d File	Allows the Radioss simulation to update the initial model.
Close	Closes the dialog.

Create and Route Seatbelts

Create and route seatbelts onto a dummy using the Seatbelt panel.

1. Go to the **Analysis** page, and click the **safety** module.
2. Click the **seatbelt** panel.
3. Select the **Create** subpanel.
4. Use the **node list** selector to select the nodes which define the first endpoint, orientation nodes (used to guide the initial belt position) and the last end point of the seatbelt.

   Multiple nodes can be selected as orientation nodes to guide a smooth belt.
5. Use the **wrap around** selector to select the entities that you want the seatbelt to be wrapped around.
6. Choose which type of elements to use when creating the seatbelt.
   - Choose **1D** to create the seatbelt with no width, and so on.
   - Choose **2D/1D** to create the seatbelt with a combination of 2D and 1D elements.
7. In the **gap** field, enter a gap between the belts and the dummy components to avoid any initial penetration.
8. In the **orient sensitivity** field, enter value can be modified when in orient mode to change the sensitivity of the interactive movement of the belt segments to mouse movement.
9. If you selected **2D/1D**, define the following settings:
   a) For **end type**, select whether the end length of the seatbelt will be **rigid links** (default) or **tria surfaces**.
   b) In the **element size** field, enter the density of the mesh for seatbelts.
   c) For **mesh type**, select whether the mesh of the seatbelt will be **quads only** (default) or **R-trias**.
   d) In the **belt width** field, enter the width (in model units) of the belt.
   e) In the **1d length at start** field, enter a distance the 1D elements must extend from the first endpoint of the seatbelt.
   f) In the **1d length at end** field, enter a distance the 1D elements must extend from the last endpoint of the seatbelt.
   g) Specify when to display the full seatbelt segment.
      - Choose **on release** (default) to display the full seatbelt segment upon release of the mouse.
      - Choose **real time** to display the full seatbelt segment instantaneously.
   h) In the **place 1D elems in** field, select a collector to place 1D elements that are created.
   i) In the **place 2D elems in** field, select a collector to place 2D elements that are created.
10. After you select the desired end list and the wrap-around entities (components, elements, or ellipsoids), click **orient** or **accept**.
    - Click **Orient** to draw a full segment belt representation, which previews the orientation of the final belt. The two red end segments are linear and connect the end points selected to the previous intersection points on the dummy. The middle line segments follow the contour of

the dummy and connect the two end segments. You can select each of the line segments and interactively orient them to a specific location. Once the final position is achieved, release the mouse to create a mesh that follows the specified path. The **Update** subpanel enables you to interactively change/update the element density and biasing.

- Click **accept** to eliminate the belt segment preview and directly create a mesh on the belt. You can then interactively change or update the element density and so on.

11. Click **return** to exit the panel.

Model Setup

12

Overview of how to build a finite element model.

This chapter covers the following:

HyperLaminate

HyperLaminate is an HyperMesh module that facilitates the creation, review and edition of composite laminates.

In support of this process certain materials and design variables are also supported by the HyperLaminate module.

The HyperLaminate Solver (HLS), which is accessed through the HyperLaminate module, uses classical laminated plate theory for simple in-plane analysis of composite laminates.

The current HyperMesh database is only updated with information from the current HyperLaminate session on exit from HyperLaminate, except with Abaqus materials, which are updated simultaneously in HyperMesh and HyperLaminate, so while it is possible to work in HyperMesh while HyperLaminate is running, this is not advisable. Any changes made to those entities which HyperLaminate touches (materials, component collectors and design variables) may result in synchronization problems and loss of data.

HyperLaminate is launched from within HyperMesh either from the HyperLaminate button on the 2D page of the main menu, or by selecting HyperLaminate from the Materials or Properties pull-down menus.

The HyperLaminate module is supported for the OptiStruct, Nastran, ANSYS and Abaqus user profiles.

HyperLaminate Environment

Overview of the HyperLaminate environment.

HyperLaminate Menus

The HyperLaminate menu bar provides access to menus that allow you to manage files, edit materials, laminates, HLS loadcases and design variables, change views, and access online help.

File Menu

New

Generates a new entity, depending on the selected sub-topic in the Laminate Browser.

Export to file

Exports material and laminate information to a text file. This text file can be printed.

Exit

Exit HyperLaminate.

At this point the current HyperMesh database is updated with the information in the current HyperLaminate session.

Edit Menu

Cut

Removes the selected data from an entry field and places it on the clipboard for pasting.
Can also remove rows from a ply lay-up order table and place these on the clipboard for pasting.

Copy

Places selected data from an entry field on the clipboard for pasting.
Can also place rows from a ply lay-up order table on the clipboard for pasting.

Paste

Pastes data from the clipboard in selected entry fields.
Can also paste rows from the clipboard above selected rows on a ply lay-up order table.

Delete

When the cursor is active in the Laminate Browser, this deletes the selected entity from the Laminate Browser. A dialog is displayed to confirm the deletion.
When the cursor is active in the Define/Edit pane, this deletes the selected text from a text box or the selected rows from a ply lay-up order table.

Tools Menu

Laminate Options

Displays the **Laminate Options** dialog, which can be used to select defaults for new Laminates for:

- color
- Convention
- Repetitions
- Ply thickness
- Common thickness

HLS Options

Displays the **HLS Options** dialog, which allows you to retain the input and result files for the HyperLaminate Solver. You can also choose the location these files are written to.
The default behavior is to delete these files once HLS is run.

Help Menu

Toolbar

Display/hide toolbar.

Status Bar

Display/hide status bar.

Help Menu

About HyperLaminate

Displays version, contact, and copyright information.

Help Topics

Activates the HyperLaminate online help.

HyperLaminate Toolbar

The HyperLaminate toolbar contains tools that allow you to generate new materials, laminates, HLS loadcases or design variables, and to cut, copy, paste, and delete entries in text boxes.

The HyperLaminate toolbar is located below the menu bar and its display is controlled by the Toolbar option under the View pull-down menu.

New

Generates a new entity, depending on the selected sub-topic in the Laminate Browser.

Cut

Removes the selected data from an entry field and places it on the clipboard for pasting.
Can also remove rows from a ply lay-up order table and place these on the clipboard for pasting.

Copy

Places selected data from an entry field on the clipboard for pasting.
Can also place rows from a ply lay-up order table on the clipboard for pasting.

Paste

Pastes data from the clipboard in selected entry fields.
Can also paste rows from the clipboard above selected rows on a ply lay-up order table.

Delete

When the cursor is active in the Laminate Browser, this deletes the selected entity from the Laminate Browser. (A dialog is displayed to confirm the deletion).
When the cursor is active in the Define/Edit pane, this deletes the selected text from a text box or the selected rows from a ply lay-up order table.

Laminate Browser

The Laminate Browser provides a vertical tree view of the materials, laminates, and HLS loadcases in your model.

For the OptiStruct and Nastran user profiles the browser also includes size design variables.

The Laminate browser is located on the left side of the HyperLaminate window.

On launching HyperLaminate, the Laminate Browser is populated with all the relevant materials, laminate definitions, HLS loadcases, and size design variables existing in the HyperMesh database, for the active user profile. The data is presented in a slightly different format for the various user profiles.

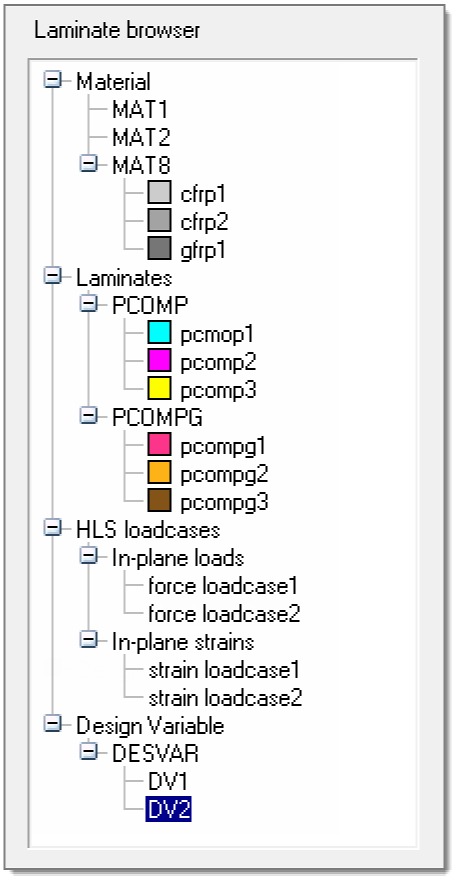


Figure 1562: OptiStruct & Nastran

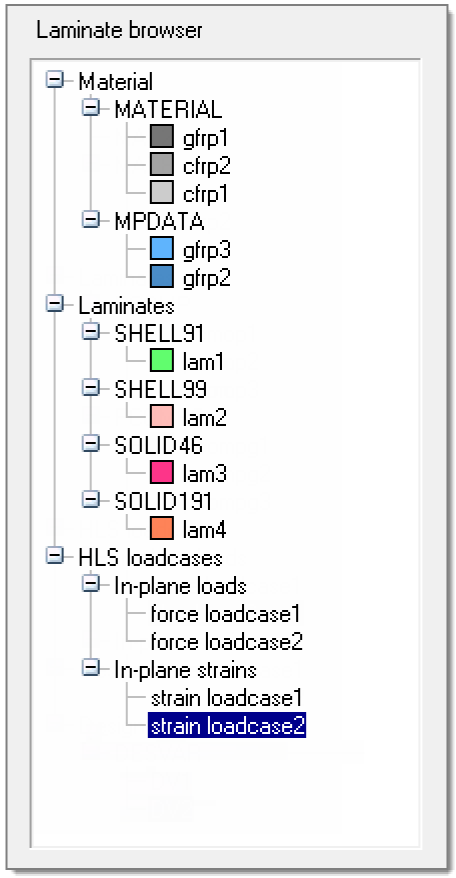


Figure 1563: ANSYS

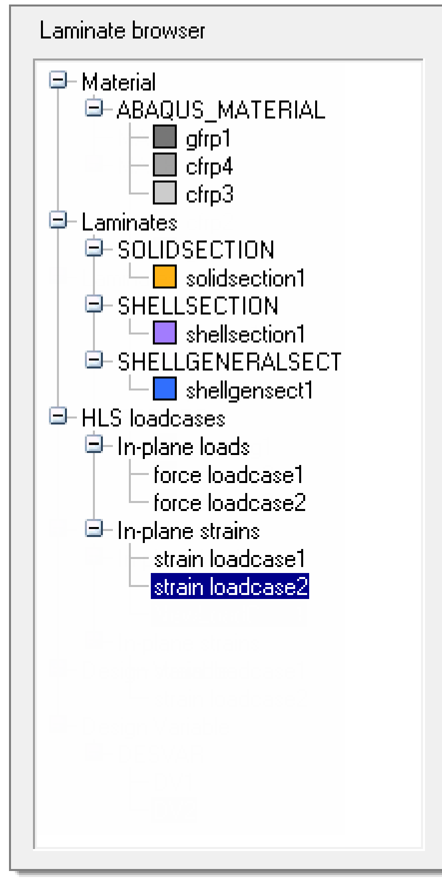


Figure 1564: Abaqus

The Laminate Browser is organized in a three-level hierarchy.

1. At the highest level are the entity types: Materials, Laminates, HLS loadcases, and Design Variables.
2. At the intermediate level are the entity sub-types or card images. These are:
   - for OptiStruct and Nastran:
     - Materials: MAT1, MAT2 and MAT8
     - Laminates: PCOMP and PCOMPG
     - HLS loadcases: In-plane loads and In-plane strains
     - Design variables: DESVAR
   - for ANSYS:
     - Materials: MATERIAL and MPDATA

    - Laminates: SHELL91, SHELL99, SOLID46 and SOLID 191
    - HLS loadcases: In-plane loads and In-plane strains
  - for Abaqus:
    - Materials: ABAQUS_MATERIAL
    - Laminates: SOLIDSECTION, SHELLSECTION and SHELLGENERALSECTION
    - HLS loadcases: In-plane loads and In-plane strains

3. At the lowest level are the entities, displayed with the names as defined by you.

Left- or right-clicking on a branch in the browser selects that branch and it becomes highlighted. When an entity (lowest level branch in the tree hierarchy) is selected, the Define/Edit and Review/Results panes are populated with details related to that entity. It is then possible to alter and update the entity definition.

Right-clicking on an already selected (highlighted) branch offers context-sensitive operations for that branch.

- At the highest level (entity types) no operations are available.
- At the intermediate level (entity sub-types) only one operation is available: New, which will create a new entity of the selected sub-type. For example, if MAT1 is selected and you right-click it and choose New; a new MAT1 entity is created.
- At the lowest level (entities) three operations are available; Rename, which allows the entity to be renamed; Duplicate, which creates a copy of the selected entity; and Delete, which will delete the selected entity.
- For Laminates a fourth operation is available: to export HLS results for the selected laminate to a file.

Create Entities

Methods to create new entities in HyperLaminate.

- In the browser, right-click on an intermediate level branch (an entity sub-type or card image) and select **New** from the context menu.
- Select an intermediate level branch (an entity sub-type or card image) from the browser tree, then select **New** from the File menu.
- Select an intermediate level branch (an entity sub-type or card image), then click  (New) on the toolbar.

A new entity appears under the selected branch.

A default name and ID are assigned to each newly created entity.

Review and Update Entities

1. In the Laminate browser tree, right-or left-click an entity (lowest level in tree hierarchy).
   The selected entity is highlighted.

   The Define/Edit and Review/Results panes are populated with details of that entity.

2. In the Define/Edit pane, make the desired changes to the entity definition.
3. Click **Apply** or **Update Laminate** to update the entity.

Rename Entities

1. In the Laminate Browser tree, right-click an entity (lowest level in tree hierarchy), and select **Rename** from the context menu.
   The name of the selected entity in the Laminate Browser switches to a text box.
2. Enter the desired new name in the text box.

   You can also rename an entity by altering the relevant field in the Define/Edit pane and then clicking **Apply** or **Update Laminate**.

Duplicate Entities

In the Laminate Browser tree, right-click an entity (lowest level in tree hierarchy), and select **Duplicate** from the context menu.

A duplicate of the entity is created and appears in the Laminate Browser.

Delete Entities

1. Delete entity.
   - In the Laminate Browser tree, right-click an entity (lowest level in tree hierarchy), and select **Delete** from the context menu.
   - In the Laminate Browser tree, left-click an entity (lowest level in tree hierarchy), and select **Delete** from the Edit menu.
   - In the Laminate Browser tree, left-click an entity (lowest level in tree hierarchy), and click (Delete) from the toolbar.
2. In the confirmation dialog, click **Yes**.

The entity is deleted and disappears from the Laminate Browser.

> **Note:** Abaqus materials that are created but not defined (they appear in a red font in Laminate Browser) may not be deleted, as they do not really exist. To delete these undefined materials, either complete their definition (by clicking **Edit** – which takes you to the HyperMesh material card previewer) or exit and restart HyperLaminate, in which case the undefined materials are purged.

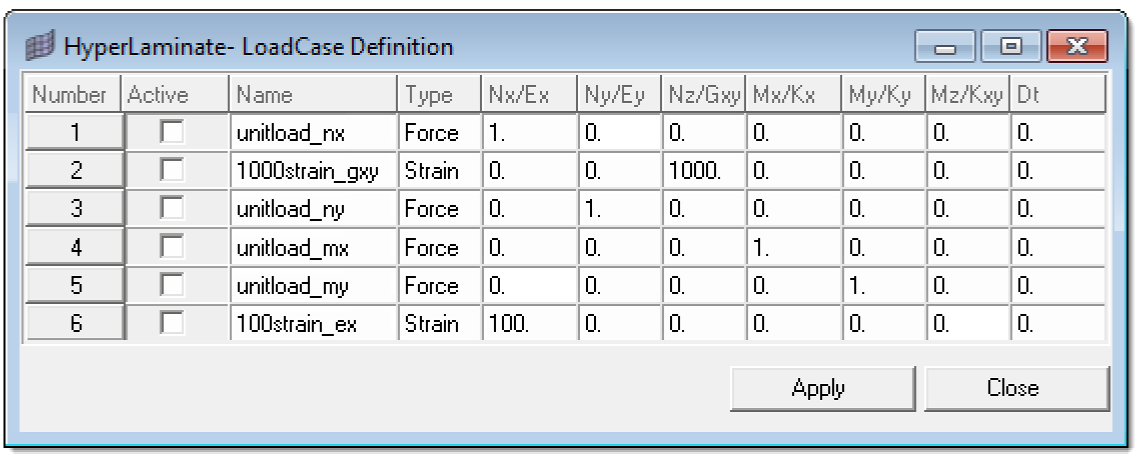
HyperLaminate Solver

The HyperLaminate Solver (HLS) uses classical laminated plate theory to analyze composite laminates subject to various in-plane and thermal loading conditions.

The solver is integrated into the HyperLaminate module of HyperMesh. The following functionalities are provided:

- to define and edit HLS loadcases
- to select a subset of HLS loadcases for analysis for each laminate
- to perform the analysis
- to review the results of the analysis for each laminate
- to export the results to an external file

When a laminate is selected from the Laminate browser, an Assign LoadCases button is present in the lower left corner of the Define/Edit pane. This button launches the LoadCase Definition GUI, allowing you to select which HLS loadcases the current laminate will be analyzed for.



Number	Active	Name	Type	Nx/Ex	Ny/Ey	Nz/Gxy	Mx/Kx	My/Ky	Mz/Kxy	Dt
1	☐	unitload_nx	Force	1.	0.	0.	0.	0.	0.	0.
2	☐	1000strain_gxy	Strain	0.	0.	1000.	0.	0.	0.	0.
3	☐	unitload_ny	Force	0.	1.	0.	0.	0.	0.	0.
4	☐	unitload_mx	Force	0.	0.	0.	1.	0.	0.	0.
5	☐	unitload_my	Force	0.	0.	0.	0.	1.	0.	0.
6	☐	100strain_ex	Strain	100.	0.	0.	0.	0.	0.	0.

Figure 1565:
The LoadCase Definition dialog allows you to select loadcases for the current laminate.

Once the desired loadcases are selected, the analysis can be performed for the current laminate by clicking **Calculate**. Once the analysis is complete several results tabs will appear in the Review/Results pane, namely:

- Stiffness/Material Matrix
- Mid-Plane Results
- Global System Results

- Material System Results
- Principal Results
- Invariant Results

These results will remain so long as the laminate is not updated. Once a laminate is updated, the results will no longer be valid and therefore the results tabs are removed. Clicking **Calculate** will re-launch the HyperLaminate Solver and populate the results tabs for the updated laminate definition.

Select HLS Loadcases for the Current Laminate

1. Click **Assign LoadCases**.
   The **LoadCase Definition** dialog opens.
2. Check the boxes corresponding to the loadcases to be selected.
3. Click **Assign** to save the information.
4. Click **Close** to close the dialog.

Define/Edit Pane

The Define/Edit Pane, the central pane of the HyperLaminate window, allows you to edit the definition of the selected entity.

On selecting an entity in the Laminate Browser the Define/Edit pane is populated with the current definition. The configuration of the Define/Edit pane differs for different user profiles and sub-types (card images).

Materials

For OptiStruct, Nastran and ANSYS materials, all material property information for the selected material may be edited in the Define/Edit pane. Once the desired changes have been made, clicking **Apply** will save those changes for the current HyperLaminate session (it is important to remember that the HyperMesh database is only updated upon exiting HyperLaminate). To reset all material property fields to zero you can click **Clear**. Below are screenshots showing the Define/Edit pane for an OptiStruct MAT8 definition and an ANSYS MATERIAL definition.

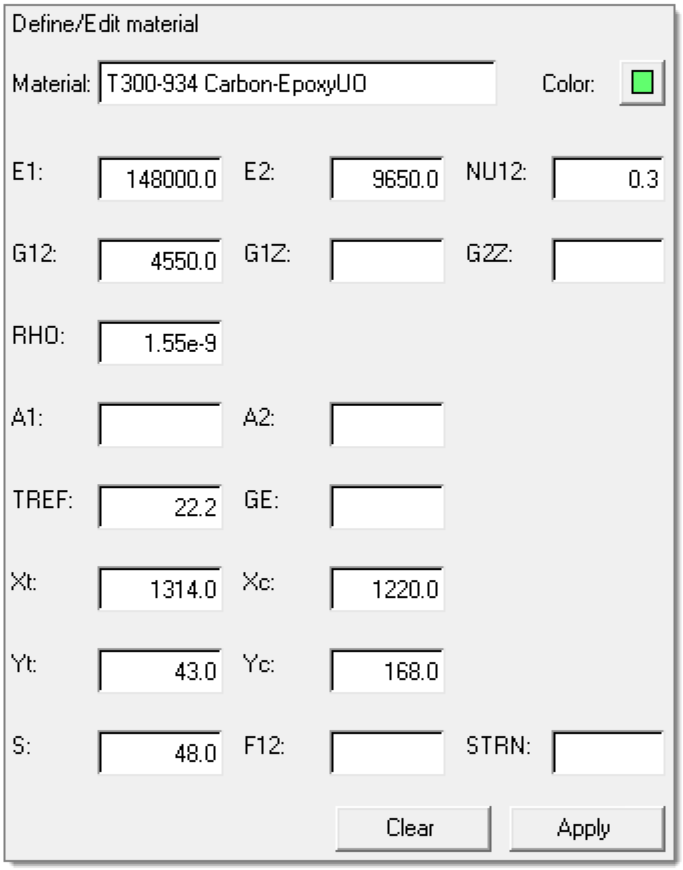


Figure 1566: OptiStruct Materials – MAT8

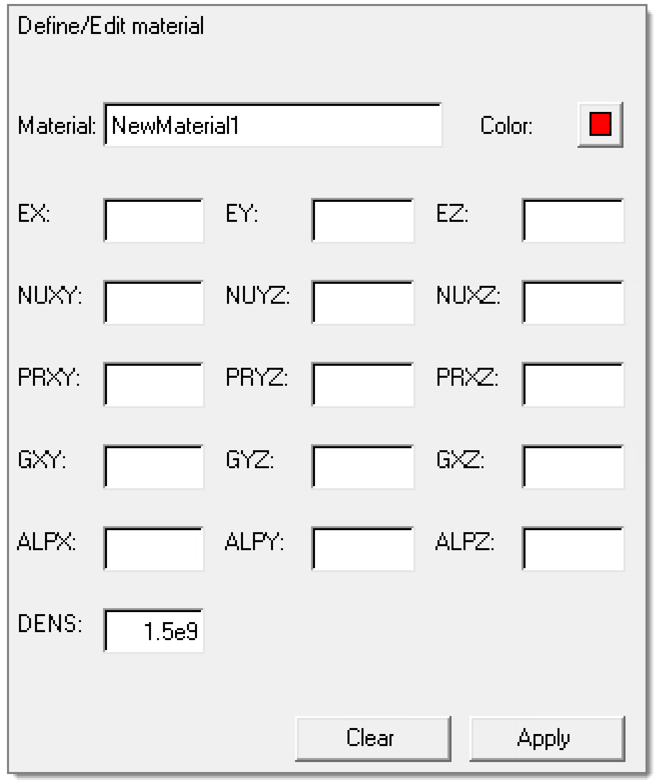


Figure 1567: ANSYS Materials – MATERIAL

For Abaqus materials, you may rename or redefine the color of the material in the Define/Edit pane, but to fully define the material properties you must click **Edit**. Clicking **Edit** takes you to the material card previewer in the HyperMesh GUI, where you can review and alter the definition of the selected material. Once you have finished reviewing/editing the material, clicking **return** will return you to the HyperLaminate GUI. As with the other user profiles, to reset all material property fields to zero you can click **Clear**.

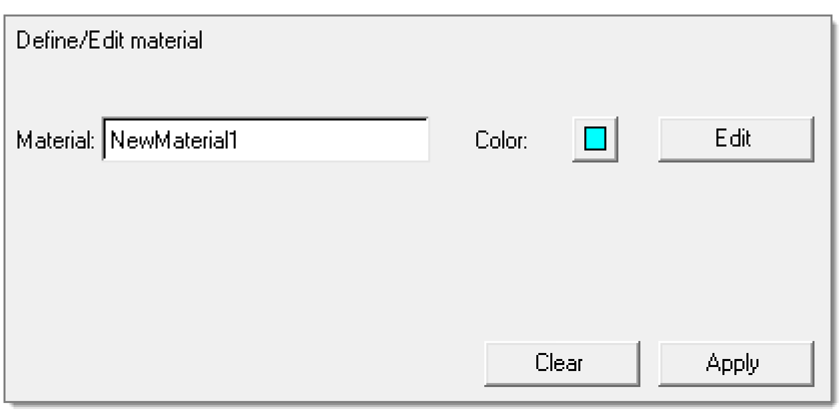


Figure 1568: Abaqus Materials – ABAQUS_MATERIAL

Laminates

For laminates, the Define/Edit pane allows the laminate name, HyperMesh entity color, stacking sequence convention, and the ply lay-up order to be edited. In addition, HLS loadcases may be selected (through the Assign LoadCases button) and solved (through the **Calculate** button) for the current laminate. This is for all supported user profiles and laminate sub-types.

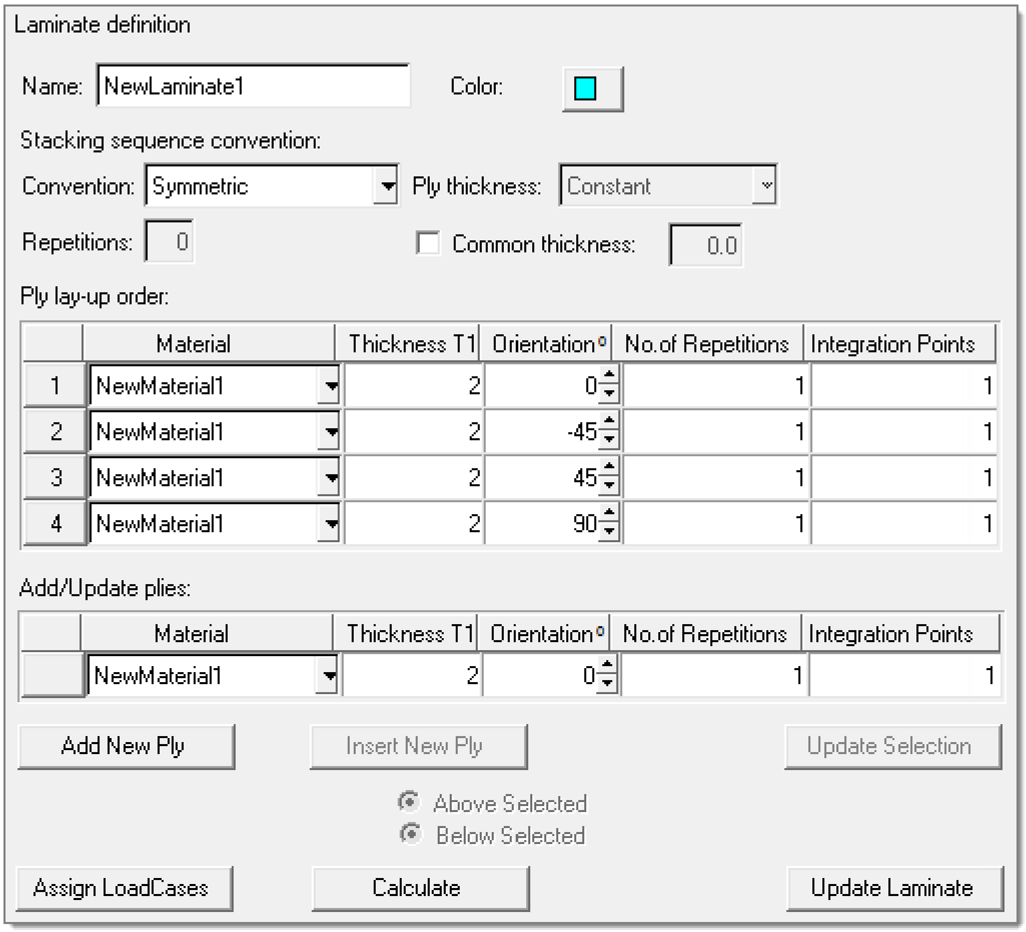


Figure 1569: Abaqus Laminates – SOLIDSECTION

There are a number of options for Convention for the stacking sequence:

Total

The Ply lay-up order table describes the laminate in its entirety.

Symmetric

The Ply lay-up order table describes the bottom half of the laminate. The top half of the laminate is the mirror image of the bottom half. The ply angles used for the top half are the same as the ply angles used in the bottom half.

Antisymmetric

The Ply lay-up order table describes the bottom half of the laminate. The top half of the laminate is the mirror image of the bottom half. The ply angles used for the top half have the opposite sign to the ply angles used in the bottom half (but 0, 90, 180, 270, and 360 remain as 0, 90, 180, 270, and 360, respectively).

Symmetric-Midlayer

The Ply lay-up order table describes the bottom half of the laminate and a midlayer (or core). The midlayer is the last ply defined in the table. The top half of the laminate is the mirror image of the bottom half. The midlayer is not reflected. The ply angles used for the top half are the same as the ply angles used in the bottom half. Due to the midlayer, the total number of plies is always odd.

Antisymmetric-Midlayer

The Ply lay-up order table describes the bottom half of the laminate and a midlayer (or core). The midlayer is the last ply defined in the table. The top half of the laminate is the mirror image of the bottom half. The midlayer is not reflected. The ply angles used for the top half have the opposite sign to the ply angles used in the bottom half (but 0, 90, 180, 270, and 360 remain as 0, 90, 180, 270, and 360, respectively). Due to the midlayer, the total number of plies is always odd.

Repeat

The Ply lay-up order table describes a single sub-laminate which is repeated a number of times. The number of repetitions is determined by the number entered in the Repetitions: field (which is activated when this Convention is chosen).

It is possible to choose between constant and variable ply thickness for certain user profiles; variable ply thickness allows up to four nodal thicknesses to be defined for each ply.

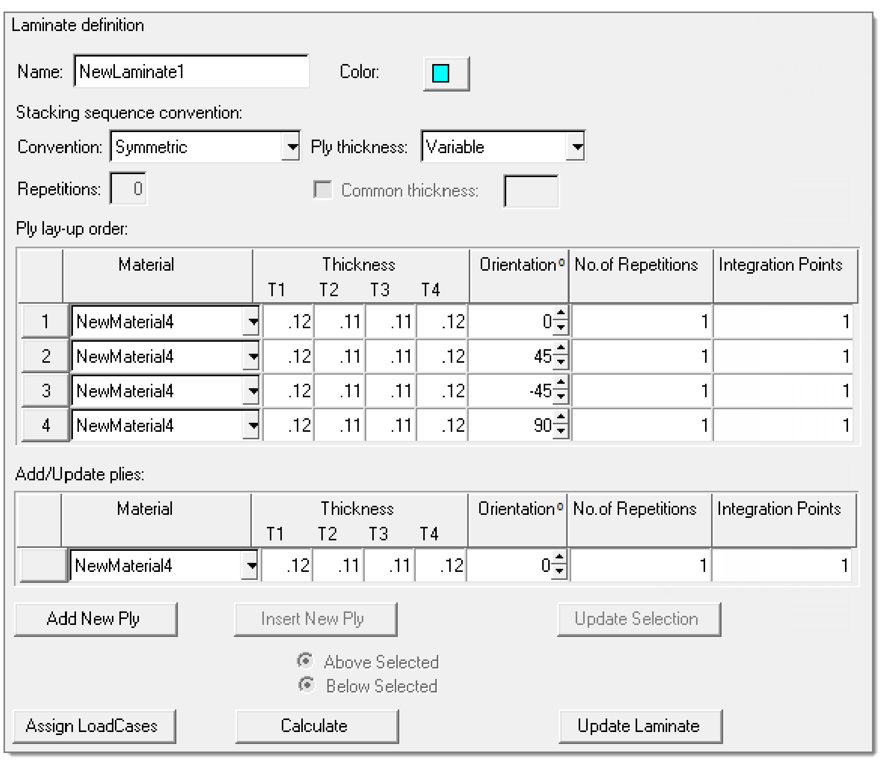


Figure 1570: ANSYS Laminates – SHELL99

It is also possible to choose a common thickness for all plies. Common thickness gives every ply in the laminate the same thickness.

The Ply lay-up order table describes the laminate from the bottom ply (most negative Z) moving upwards (increasing in positive Z direction). Each row of the table defines the material, ply thickness and ply orientation for a number of plies (defined by the No. of repetitions field and based on the selected Convention). The number of integration points for each ply (or group of plies) is also provided in the table. This number is used by the HyperLaminate Solver, as well as the external FEA solver when applicable.

Rows are added to the table by completing the Add/Update plies: entry fields and clicking **Add New Ply**. Rows may be inserted in the table, either above or below selected rows (choose from the **Above Selected** or **Below Selected** radio buttons), by clicking the Insert New Ply button. Rows may be cut, copied from, pasted to, or deleted from the table using the Toolbar, pull-down Edit menu, or Ctrl+X, Ctrl+C, Ctrl+V, and Ctrl+D respectively. Select multiple rows by selecting one row and then, with the Ctrl key held down, selecting other rows (alternatively, multiple rows may be selected with the Shift key held down; this will retain the current selection and add all the rows between the current selection and the newly selected row). Rows are always pasted above the selected rows when multiple rows are selected the clipboard contents are pasted above each selected row.

All fields in the Ply lay-up order table may be edited. It is also possible to edit multiple rows at once. Select multiple rows as described in the previous paragraph. When multiple rows are selected, the Add/Update plies: fields are populated with the information common to the selected rows. Blank fields indicate that not all of the selected rows contain the same values for that field. Changes can be made to the Add/Update plies: fields and Update Selection can be clicked to update the selected rows with the updated information (no changes occur to the selected rows for blank fields).

For the OptiStruct and Nastran user profiles it is possible to request stress and failure theory output. Each row of the Ply lay-up order table has an SOUT field, which when set to YES includes the plies described by that row in the stress output and the failure theory calculation. It is possible to set the SOUT field individually for each row, or for all rows at once through the Output ply stress results: field under the Stress and failure theory output: heading. Once one or more SOUT fields are set to YES it is possible to activate failure theory calculation, by checking the Failure Theory check-box, selecting a theory from the pull-down list and defining an Interlaminar shear allowable: value.

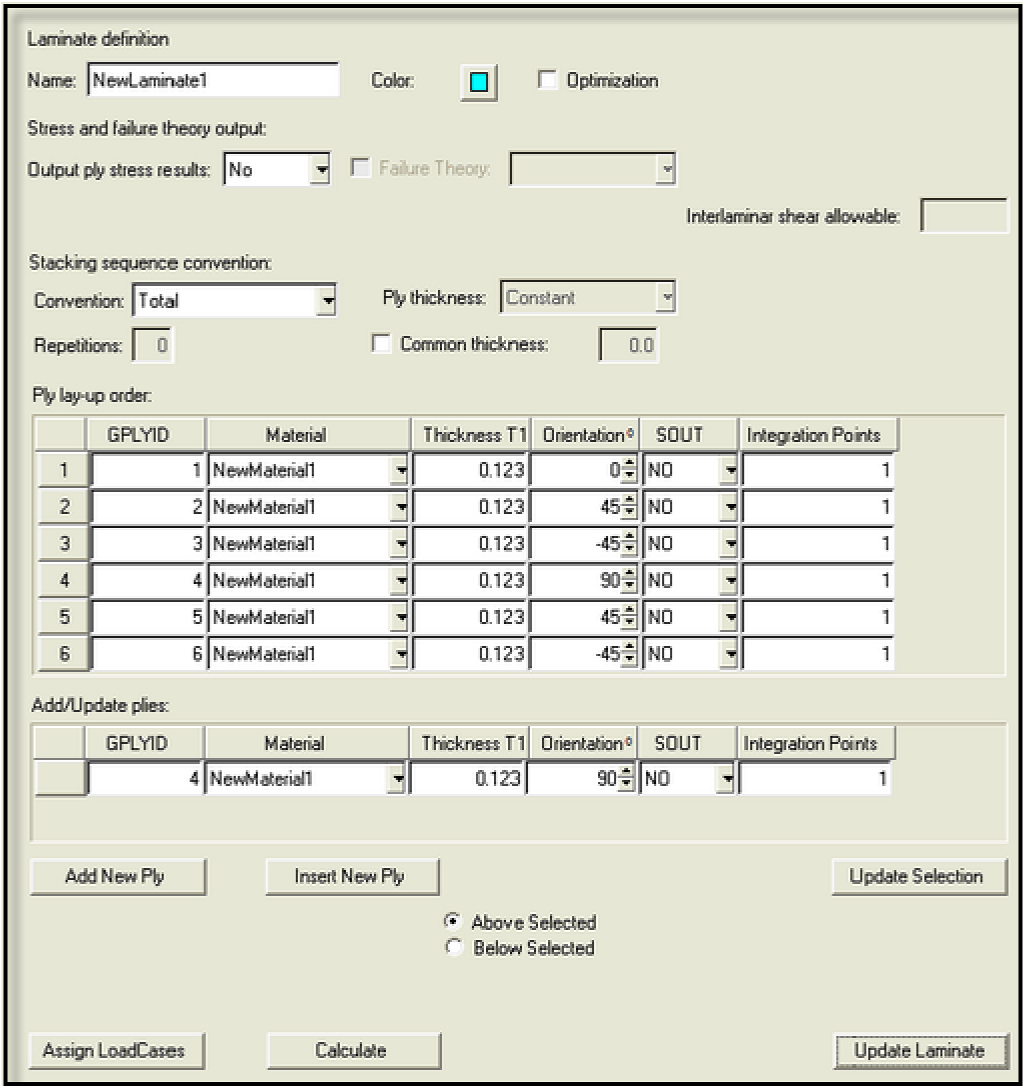


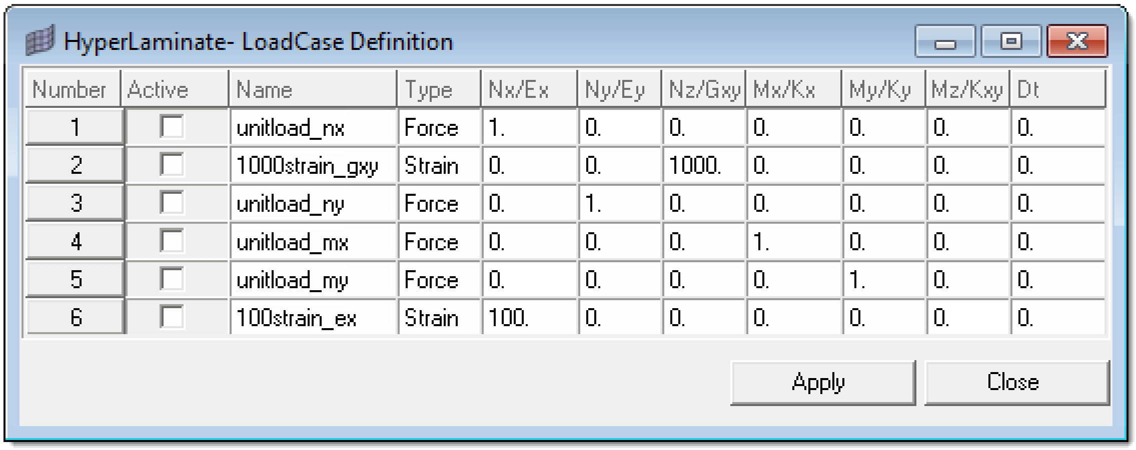
Figure 1571: OptiStruct Laminates – PCOMPG

The Ply lay-up order table for the OptiStruct and Nastran PCOMPG laminate sub-type is different from other laminate sub-types in that it has a GPLYID field. This field is used to assign a global ply ID to a ply definition (the global ply ID is a post-processing aid). As this ID should not be repeated within the same laminate, the No. of repetitions field is not available for PCOMPG. For PCOMPG each row in the Ply lay-up order table should represent a single ply so only the Total stacking convention should be used for PCOMPG, but this is not enforced in the GUI.

For the OptiStruct and Nastran user profiles it is possible to assign a design variable to a thickness or orientation field in the Ply lay-up order table. Checking the **Optimization** checkbox expands the Ply lay-up order table, adding extra fields to the right of the Thickness T1 and Orientation 0 fields. Design variables may be selected in these extra fields. Selecting a design variable to the right of a thickness or orientation assigns the selected design variable to that thickness or orientation.

Click **Update Laminate** to apply all the changes for the current HyperLaminate session (it is important to remember that the HyperMesh database is only updated on exit from HyperLaminate).

The Define/Edit pane for Laminates also provides access to the HyperLaminate Solver. A number of inplane loading scenarios (HLS loadcases) may be solved for a given laminate. The HLS loadcases are selected on the **LoadCase Definition** dialog, which is launched by clicking **Assign LoadCases**.



HyperLaminate- LoadCase Definition

Number	Active	Name	Type	Nx/Ex	Ny/Ey	Nz/Gxy	Mx/Kx	My/Ky	Mz/Kxy	Dt
1		unitload_nx	Force	1.	0.	0.	0.	0.	0.	0.
2		1000strain_gxy	Strain	0.	0.	1000.	0.	0.	0.	0.
3		unitload_ny	Force	0.	1.	0.	0.	0.	0.	0.
4		unitload_mx	Force	0.	0.	0.	1.	0.	0.	0.
5		unitload_my	Force	0.	0.	0.	0.	1.	0.	0.
6		100strain_ex	Strain	100.	0.	0.	0.	0.	0.	0.

Apply Close

Figure 1572: LoadCase Definition GUI

HLS loadcases with a check in the Active column are selected for the current laminate. Different HLS loadcases may be selected for different laminates. Having selected the appropriate loadcases, click **Apply** and then **Close** to exit the dialog.

Once the loadcases are selected, clicking **Calculate** launches the HyperLaminate Solver. The Review/ Results pane is then populated with several tabs containing the HyperLaminate Solver results.

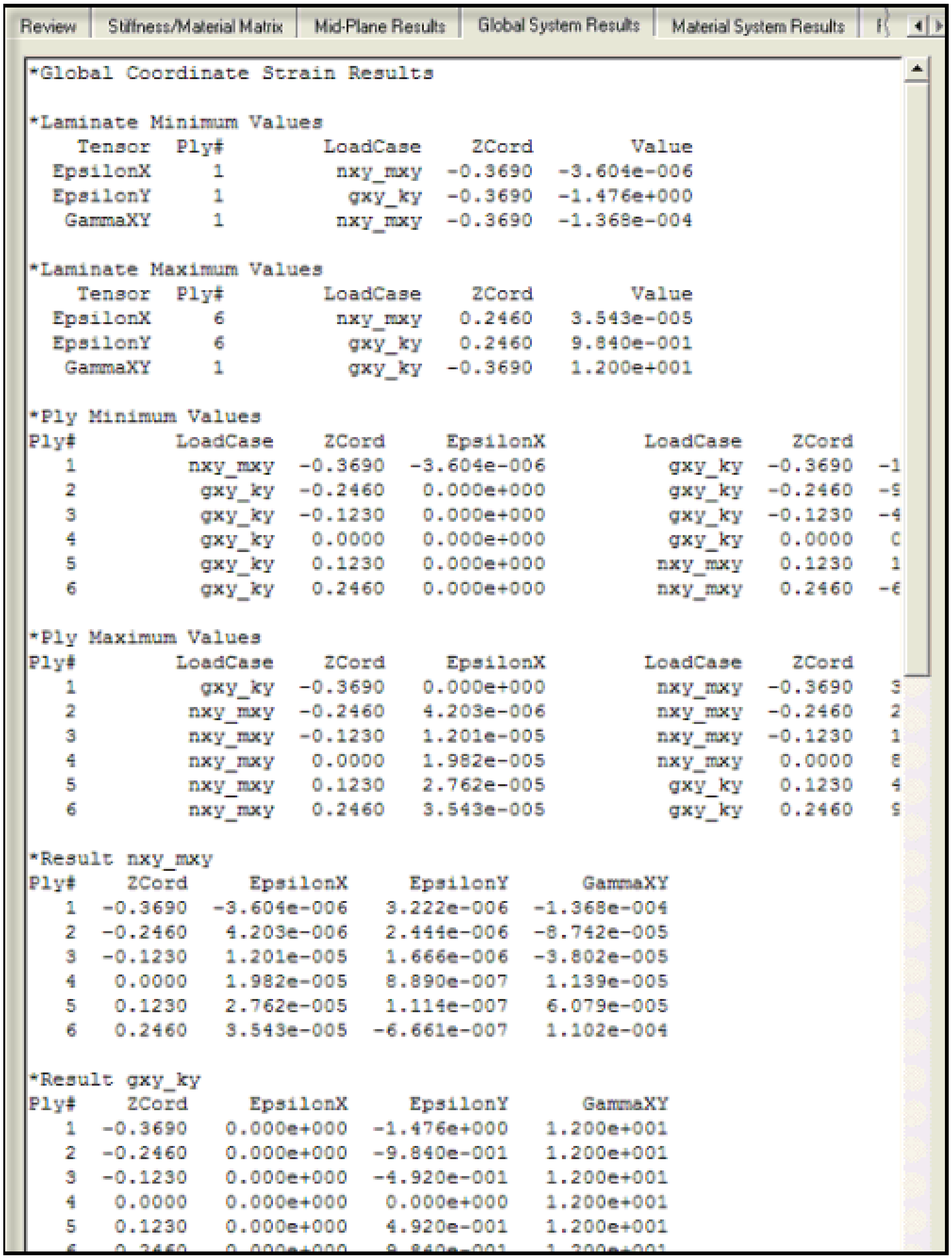


Figure 1573: Hyperlaminte Solver results

HLS Loadcases

For all user profiles the HLS loadcases branch allows various in-plane loading scenarios to be defined and stored. The loading scenarios can be either load based or strain based. All information for the selected HLS loadcase may be edited in the Define/Edit pane. Once the desired changes have been made, clicking **Apply** will save those changes for the current HyperLaminate session. It is important to remember that the HyperMesh database is only updated on exit from HyperLaminate. To reset all fields for the selected HLS loadcase to zero, you can click **Clear**.

Design Variables

For the OptiStruct and Nastran user profiles, the DESVAR design variable card is supported in HyperLaminate. All information for the selected design variable may be edited in the Define/Edit pane. Once the desired changes have been made, clicking **Apply** will save those changes for the current HyperLaminate session. It is important to remember that the HyperMesh database is only updated on exit from HyperLaminate. To reset all fields for the selected design variable to their default values, you can click **Clear**.

Create and Edit HyperLaminate Solver Loadcases

Methods to create and edit HyperLaminate solver loadcases.

Create HyperLaminate Solver Loadcases

1. Create loadcase.
   - In the browser, right-click on a sub-type under the HLS loadcases branch and select **New** from the context menu.
   - In the browser, right-click on a sub-type under the HLS loadcases branch, then select **New** from the File menu.
   - In the browser, right-click on a sub-type under the HLS loadcases branch, then click (New) on the toolbar.

   A new loadcase appears under the selected branch.

   A default name and ID are assigned to newly created HLS loadcases.

   The newly created loadcase is automatically selected in the Laminate Browser and the Define/Edit pane takes on the appropriate configuration.
2. If desired, a new name for the laminate may be entered in the Loadcase field.
3. Provide the loadcase definition by filling in the entry fields in the Define/Edit pane.
4. Click **Apply** to save the changes for the current HyperLaminate session.

   > **Note:** The HyperMesh database is only updated on exit from HyperLaminate.

Edit HyperLaminate Solver Loadcases

1. In the Laminate Browser, select the loadcase to be edited.
   The Define/Edit and Review/Results panes are populated with the selected loadcase definition.
2. Edit the data fields.

   Data may be cut, copied from, pasted to, or deleted from the data fields using the toolbar, pull-down Edit menu, or Ctrl+X, Ctrl+C, Ctrl+V, and Ctrl+D respectively.

Clicking **Clear** will reset all fields to zero.

Each change is reflected in the Review/Results pane.

3. If desired, a new name for the loadcase may be entered in the Loadcase field.
4. Click **Apply** to save the changes for the current HyperLaminate session.

> **Note:** The HyperMesh database is only updated on exit from HyperLaminate.

The final loadcase definition is displayed on the Review tab.

Review/Results Pane

The Review/Results pane allows you to review the information pertaining to the selected entity as well as displaying HyperLaminate Solver results for laminates.

The Review/Results pane is located on the right-hand pane of the HyperLaminate window.

On selecting an entity in the Laminate Browser, the Review/Results pane displays the current definition of that entity.

Materials

For OptiStruct, Nastran and ANSYS materials, all material property information for the selected material is displayed in the Review/Results pane. This information is updated as the definitions are altered in the Define/Edit pane.

For Abaqus materials, no information is displayed in the Review/Results pane.

Laminates

For laminate definitions for all user profiles, the Review/Results pane a Review tab and several results tabs:

- Stiffness/Material Matrix
- Mid-Plane Results
- Global System Results
- Material System Results
- Principal Results
- Invariant Results

Information displayed on these results tabs is for the saved laminate definition, and is removed when you click **Update Laminate**. The results tabs reappear if you run the HyperLaminate Solver for the updated definition by clicking **Calculate**.

The Review tab is headed by the laminate name, the total number of plies in the laminate, and the total thickness of the laminate. This is followed by a description of the laminate, listing the plies in order from the bottom ply (most negative z), showing a graphical representation of each ply's orientation and listing the referenced material, thickness, and orientation.

The Stiffness/Material Matrix tab provides the two sets of matrices. The first set of matrices are the composite shell stiffness matrices, more commonly referred to as the ABD matrices. The second set of matrices are the equivalent material matrices. These are used by many finite element solvers to represent the laminated composite as a homogenized shell.

The remaining results tabs present results of the HyperLaminate Solver.

HLS Loadcases

For all user profiles, information for the selected HLS loadcase is displayed in the Review/Results pane. This information is updated as the definitions are altered in the Define/Edit pane.

Design Variables

For OptiStruct and Nastran user profiles, information for the selected design variable is displayed in the Review pane. This information is updated as the definitions are altered in the Define/Edit pane.

Create and Edit Materials

Methods to create and edit materials in HyperLaminate.

Create Materials

1. Create material.
   - In the browser, right-click on a sub-type under the Material branch and select **New** from the context menu.
   - In the browser, right-click on a sub-type under the Material branch and select **New** from the File menu.
   - In the browser, right-click on a sub-type under the Material branch, then click
     
     (New) on the toolbar.

   A new material appears under the selected branch.

   A default name and ID are assigned to newly created materials.

   The newly created material is automatically selected in the Laminate Browser and the Define/Edit pane takes on the appropriate configuration for the selected material sub-type.
2. Define material.

Option	Description
For the OptiStruct, Nastran and ANSYS user profiles	1. If desired, a new name for the material may be entered in the Material field or the material color may be altered by clicking the color swatch and selecting a new color from the pop-up color palette.

Option	Description
	2. Provide the material definition by filling in the entry fields in the Define/Edit pane. 3. Click **Apply** to save the changes for the current HyperLaminate session. **Note:** The HyperMesh database is only updated on exit from HyperLaminate.
For the Abaqus user profile	1. Click **Edit** and provide the material definition in the HyperMesh card previewer. 2. Click **return**. This returns you to the HyperLaminate . 3. If desired, a new name for the material may be entered in the Material field or the material color may be altered by clicking the color swatch and selecting a new color from the pop-up color palette. 4. Click **Apply** to save the changes. **Note:** It is not possible to rename an Abaqus material until after it has been defined (edited). Also it is not possible to create a new Abaqus material if an undefined material definition already exists (appears in a red font in Laminate Browser).

Edit Materials

1. In the Material Browser, select the material to edit.
   The Define/Edit and Review/Results panes are populated with the selected material definition.
2. Edit material.

Option	Description
For the OptiStruct, Nastran and ANSYS user profiles	1. Edit the data fields in the Define/Edit pane. 2. Data may be cut, copied from, pasted to, or deleted from the data fields using the toolbar, pull-down Edit menu, or Ctrl+X, Ctrl+C, Ctrl+V, and Ctrl+D respectively. 3. Clicking **Clear** will reset all fields to zero.

Option	Description
	Each change is reflected in the Review/Results pane. **4.** If desired, a new name for the material may be entered in the Material: field, or the material color may be altered by clicking the color swatch and selecting a new color from the pop-up color palette. **5.** Click **Apply** to save the changes for the current HyperLaminate session. **Note:** The HyperMesh database is only updated on exit from HyperLaminate. The final material definition is displayed in the Review tab.
For the Abaqus user profile	**1.** Click Edit to see the material definition in the HyperMesh card previewer. **2.** Make all desired changes to the material definition in the card previewer. **3.** Click **return**. This returns you to the HyperLaminate GUI. **4.** If desired, a new name for the material may be entered in the Material: field or the material color may be altered by clicking the color swatch and selecting a new color from the pop-up color palette. **5.** Click **Apply** to save the changes.

Create and Edit Laminates

Methods to create and edit laminates in HyperLaminate.

Create Laminates

1. Create laminate.
    - In the browser, right-click on a sub-type under the Laminate branch and select **New** from the context menu.
    - In the browser, right-click on a sub-type under the Laminaate branch and select **New** from the File menu.
    - In the browser, right-click on a sub-type under the Laminate branch, then click (New) on the toolbar.

A new laminate appears under the selected branch.

A default name and ID are assigned to newly created laminate.

The newly created laminate is automatically selected in the Laminate Browser and the Define/Edit pane takes on the appropriate configuration for the selected material sub-type.

2. If desired, a new name for the laminate may be entered in the Laminate name: field or the component color may be altered by clicking the color swatch and selecting a new color from the pop-up color palette.
3. For OptiStruct and Nastran user profiles, define the Stress and failure theory output: information as desired.
4. For all user profiles, define the Stacking sequence convention: information.
5. For Convention:, select a stacking sequence conventions.
6. If you select **Repeat**, specify how many times you want to repeat the entire block of entry rows.
7. For Ply thickness, select a thickness method.
   - Choose **Constant** to have the Ply lay-up order table include a single thickness column: Thickness T1.
   - Choose **Variable** to have the Ply lay-up order table include multiple thickness columns.

   > **Note:** The option to switch between constant or variable thickness is only available for certain laminate sub-types.

8. For Constantply thickness, you can check the **Common Thickness** checkbox and specify a thickness to be used by all the entry rows.

   Having checked the Common Thickness checkbox and entered a common thickness value, if you now uncheck the box, the thickness fields retain the common thickness value, but are now editable.
9. Complete the Ply lay-up order table.
10. Add/insert rows by completing the Add/Update plies: fields and clicking **Add New Ply** or **Insert New Ply** (for Insert New Ply, it is possible to choose to insert the ply above or below the selected rows).

    > **Note:**
    >
    > The number of rows in the table is not the number of plies. This is governed by the stacking convention and the number of repetitions.

11. Data may be cut, copied from, pasted to, or deleted from selected fields using the toolbar, pull-down Edit menu or Ctrl+X, Ctrl+C, Ctrl+V and Ctrl+D, respectively.
12. Table rows may also be cut, copied from, pasted to, or deleted from, using the toolbar, pull-down Edit menu or Ctrl+X, Ctrl+C, Ctrl+V, and Ctrl+D, respectively.

> **Note:**
>
> Rows are always pasted above selected rows.
>
> When multiple non-sequential rows are copied and then pasted, they will be pasted as sequential rows. For example, if rows 1 and 3 are copied and pasted at row 7, row 1 will be pasted as row 7, row 3 will be pasted as row 8, and what was row 7 will now be row 9.

13. For the OptiStruct and Nastran user profiles it is possible to define thickness and orientation fields in the Ply lay-up order table as designable and to assign design variables to them.
14. Click **Update Laminate** to save the changes for the current HyperLaminate session.

> **Note:** The HyperMesh database is only updated on exit from HyperLaminate.

Edit Laminates

1. In the Laminate Browser, select the laminate to be edited.
   The Define/Edit and Review/Results panes are populated with the selected laminate definition.
2. The laminate definition may be modified in the Define/Edit pane in a manner similar to defining a new laminate.
3. Click **Update Laminate** to save the changes for the current HyperLaminate session.

> **Note:** The HyperMesh database is only updated on exit from HyperLaminate.

Create and Edit Design Variables

Methods to create and edit design variables in HyperLaminate.

Create Design Variables

Design variables are only supported for the OptiStruct and Nastran user profiles.

1. Create design variable.
   - In the browser, right-click on a sub-type under the design variable branch and select **New** from the context menu.
   - In the browser, right-click on a sub-type under the design variable branch, then select **New** from the File menu.
   - In the browser, right-click on a sub-type under the design variable branch, then click (New) on the toolbar.

   Only sub-type available is DESVAR.

A new design variable appears under the selected branch.

A default name and ID are assigned to newly created design variable.

The newly created design variable is automatically selected in the Laminate browser and the Define/Edit pane takes on the appropriate configuration.

2. If desired, a new name for the laminate may be entered in the Desvar field.
3. Initial, lower bound, and upper bound values for the design variable can be entered in the appropriate data fields.
4. Checking the **Move limit** box activates the Move limit field, where a move limit value other than the default of 0.5 may be entered.
5. Checking the **Ddval ID** box activates the Ddval ID field, where the ID of a discrete value list may be entered.
6. Click **Apply** to save the changes for the current HyperLaminate session.

> **Note:** The HyperMesh database is only updated on exit from HyperLaminate.

Edit Design Variable

1. In the Laminate Browser, select the design variable to be edited.
   The Define/Edit and Review panes are populated with the selected design variable definition.
2. Edit the data fields in the Define/Edit pane.
   Data may be cut, copied from, pasted to, or deleted from the data fields using the toolbar, pull-down Edit menu or Ctrl+X, Ctrl+C, Ctrl+V, and Ctrl+D, respectively.
3. Clicking **Clear** will reset all fields to their default values.
   Each change is reflected in the Review pane.
4. Click **Apply** to save the changes for the current HyperLaminate session.

> **Note:** The HyperMesh database is only updated on exit from HyperLaminate.

The final design variable definition is displayed in the Review tab.

Import Geometry

Import geometry into your current HyperMesh session, and set advanced import options.

1. From the menu bar, click **File** > **Import** > **Geometry**.
   The Import browser opens, and the Import Type is automatically set to Geometry.
2. For File type, select the file type you wish to import.

   > **Tip:** Select an input translator automatically by choosing **Auto Detect**.

3. Click to choose the file to import.
   Selected files are listed.
4. Set advanced import options by clicking **Import Options** to expand the panel.

Option	Description
Set the scale factor	1. For Target units, select a unit type. 2. In the Scale factor field, enter a scale factor. This is useful when converting a model created in English units to Metric, for example.
Define cleanup tolerance	Cleanup tolerance determine if two surface edges are the same and if two surface vertices are the same. For example, cleanup tolerance determines if two surface edges are close enough to be automatically combined to shared edges (green edges) and if a surface is degenerate and should be removed. 1. For Cleanup tol, select a method for defining the cleanup tolerance. • Choose **Automatic** to take the complexity of the surfaces and edge geometries into account and to automatically define a tolerance to maximize shared edges (green edges). By default, the automatic cleanup tolerance value defaults to 100 times what is used internally by the translator. • Choose **Manual** to enter a tolerance. The value you enter must be greater than the default value. The translator modifies data only if the data stays within the original data tolerance. Increasing the tolerance can cause serious problems. When this value is set, any features equal to or less than the tolerance are eliminated. The translator does not include any edge less than tolerance long; if there are edges present that are important to the surface, that surface will be distorted, or will fail to trim properly. Surfaces smaller than the tolerance may not be imported. If the file you have read has many very short edges, it may be worthwhile to reread the file using a larger tolerance. The same holds true if surfaces appear to be "inside out" when surface lines are displayed. The tolerance value should not be set to a value greater than the node tolerance set in the options panel to be used for your element mesh.

Option	Description
	If you are reading a Catia file, you may need to override the tolerance in the file.
Define how blanked/ no show entities are imported	Import blanked entities in the IGES translator, as well as entities containing "NO SHOW" from the Catia translator by selecting the **Import hidden (blanked/no show) entities** checkbox.

5. Click **Import**.

Import Error Messages

When an external input translator is used to import data, a file for messages is created in the directory in which the program was started.

This file is named `translator.msg`, where translator is the name of the translation program being used. This file contains errors, warnings and useful information about the import process.

The translators also create a second file that contains extra data from the file being imported. This file is named `importfile.hmx`, where importfile is the name of the file being imported. This extra data could contain FEA data and keywords not supported and/or generic comments about the data.

If the **Display Import Errors** checkbox is selected on the Import tab, any error messages will display in the **Import Process Messages** dialog. From this dialog you can choose to display the `.hmx` file, save the message file or delete the message file. Clicking **Close** will close the dialog, but will not remove the message file from your directory.

Create Collectors

All entities in an HyperMesh database are stored in collectors. Based on the assigned template, each collector may use a dictionary or card image to define the attributes assigned to the collector. The definitions contained in the dictionaries or card image are used to translate models to external analysis codes.

1. From the menu bar, select **Collectors** > **Create** > **<collector type>**.
   The **Create** dialog opens.
2. In the Name field, enter a name for the collector.
3. Define additional fields accordingly.

   Fields vary based on the collector type.
4. Click **Close**.

Change the Current Collector

By default, when you create a new collector it automatically becomes the current collector.

New components, loads, beam sections, multibodies, and so on will be created and organized within the respective current collector.

The current collector status is indicated in bold.

From the Model Browser, right-click on the collector and select **Make Current** from the context menu.

> **Tip:** If you do not know the name of the collector to make current you can quickly find it by activating the selector ( ) in the Model Browser and selecting the collector in the modeling window. The selected collector becomes highlighted in the Model Browser.

Create Geometry Data

If geometry is not available from a CAD system, you can create or edit geometry in HyperMesh.

- Create and edit lines.

Option	Description
Lines	Create lines in a variety of methods, including: from points, at tangents, and at the intersection of other geometry.
Line Edit	Edit existing lines in a variety of methods such as combine, split, smooth, or extend.
Circles	Create circles or arcs.

- Create and edit surfaces.

Option	Description
Surfaces	Create surfaces from existing lines or nodes by different methods, such as spline, drag, or spin.
Primitives	Create standard shaped surfaces or solid entities, including squares, spheres, cones, and cylinders.
Surface Edit	Edit existing surfaces by trimming, extending, or shrinking.
Defeature	Edit existing surfaces by removing individual features such as holes or fillets.

- Create and edit points and nodes.

Option	Description
Nodes	Create new nodes. Several methods are available.
Temp Nodes	Add or remove nodes used only for geometry creation or editing.

Create NURBS Surfaces

A NURBS (non-uniform rational B-spline) surface is a parametric surface defined by control points, knots and weights.

Use the ruled, spline/filler, and drag/spin subpanels of the Surfaces panel to create NURBS surfaces.

The spline option creates a surface through 3D lines. If you select a set of lines that do not form a closed loop, HyperMesh will connect the disconnected lines with straight lines, and create a spline surface and/or mesh in the enclosed area. There is no limit on the number of lines used to create a mesh/surface.

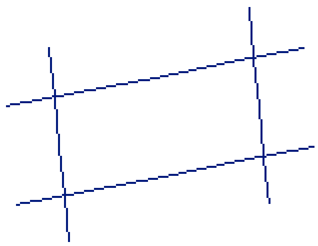


Figure 1574:
Lines form one path because they intersect at four points.

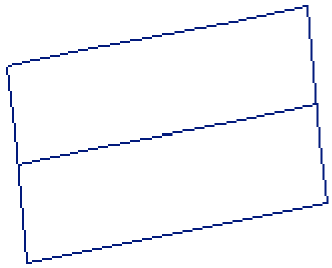


Figure 1575:
Lines form more than one path and cause an error.

The tolerance setting on the Options panel is used to determine the intersections between lines. If the tolerance is too small and an intersection cannot be found, HyperMesh reports an error when you attempt to create the surface.

Lines that contain sharp edges can cause problems when you create a surface. These lines result in a more complex surface, which takes longer to create, and slows the automeshing process. These sharp edges are sometimes the result of data created on other CAD/CAM systems and brought into HyperMesh via a translator. These lines may need to be "smoothed" by using the line edit panel or replaced with a new, smooth, line by using the Lines panel.

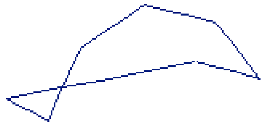


Figure 1576:
Creating a surface with these lines results in a relatively complex surface.

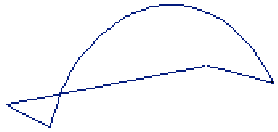


Figure 1577:

The "circular" shaped line has been replaced with a smooth line, which results in a much simpler surface. In some cases the sharp edges are required to represent the model and should not be smoothed.

The skin option can create a skinned surface through a set of lines.

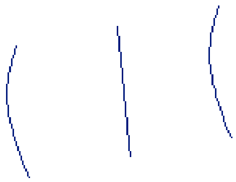


Figure 1578:
Lines used to define a skinned surface.

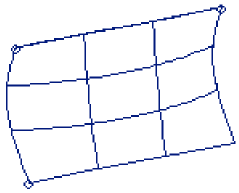


Figure 1579:
A skinned surface created from the lines.

The ruled option can create a ruled surface between two lines.

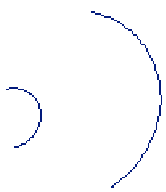


Figure 1580:
Lines used to create a ruled surface.

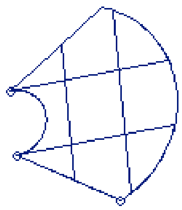


Figure 1581:
A ruled surface created from the lines.

Temporary Nodes

A temporary node list retains nodes that are not attached to an element, protecting them from automatic removal by the database management.

Nodes are not protected from automatic removal for panels that automatically clear all temporary nodes, such as edges, faces, edit elements. There may be times when you wish to use an unattached node later in the modeling process.

The Temp Nodes panel allows you to modify the temporary node list. In the Temp Nodes panel, there are three functions:

add
: Adds selected individual nodes to the temporary node list.

clear
: Removes selected individual nodes from the temporary node list.

clear all
: Removes all the temporary nodes from the database.

Select Surfaces

Overview of how to select surfaces in wireframe and shaded mode.

Surface edges may be used in the same way as lines in any surface creation panel, where appropriate. If you use any surface edge lines in the Line Edit panel, duplicates of the lines are created and the operation is applied to the duplicates.

Select surfaces.

Option	Description
Wireframe mode	The easiest method of selecting a surface is to pick the surface near its edges or surface visualization lines. If several surfaces share an edge, you can select any one of them by clicking on the edge, and while holding the mouse button down, moving the mouse slightly from side to side. Each surface highlights as selected. Release the mouse button when the desired surface is highlighted.
Shaded mode	Click anywhere on the surface to select it. Similar to wireframe mode, you can hold the left mouse button down until the surface of interest is highlighted, and release it to confirm the selection.

Edit Surfaces

Each surface contains one or more faces. It is usually preferable to combine multiple faces into one surface entity before you use the meshing tools.

This allows them to be meshed at the same time.

Use the Surface Edit panel to modify surface geometry when it is necessary to make changes before you generate a mesh.

For example, to trim a surface with a line, use the trim with lines subpanel of the Surface Edit panel. You must select the surface and the line and specify a direction vector. The surface is trimmed by sweeping the line along the vector and intersecting the surface with the sweep. If the sweep does not intersect the surface, the surface is not trimmed.

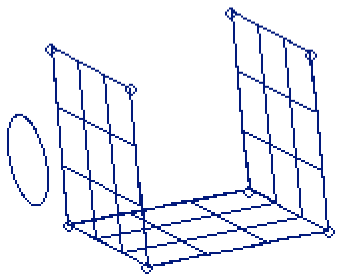


Figure 1582:
A circle and a surface (represented with surface lines) before trimming.

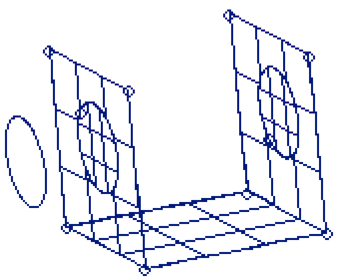


Figure 1583:
After the circle is used to trim the surface, two new surfaces are created (shown highlighted) and the original surface is trimmed.

To trim one surface with another, use the trim with surfs/planes subpanel.

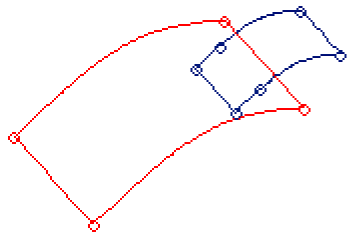


Figure 1584:

Two surfaces before trimming.

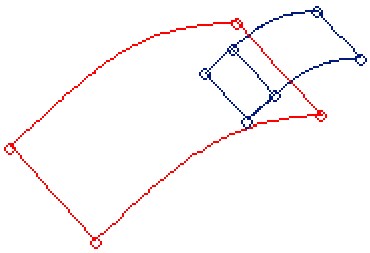


Figure 1585:

The smaller surface is split into two surfaces after it is trimmed with the larger surface.

Associativity

Nodes and elements can be associated to surfaces.

When you create a mesh with the automesher, the nodes are automatically associated to the surface. When nodes are associated to a surface, you can use the Smooth panel to smooth elements on the surface and the Node Edit panel to move the nodes along the surface. Associated nodes and elements can be selected by surface, which allows you to select all the nodes and/or elements associated to a surface. Some operations break associativity. If you transform, such as translate, a surface, node, or element, associativity is broken. However, if you transform a component that contains both a surface and its associated nodes/elements, the associativity is not broken.

Associativity is also broken if you trim a surface. To re-associate a node to a surface, use the Node Edit or Project panel.

Note: Re-associating nodes to a surface is usually a time consuming task.

Geometry Cleanup

Prepare surface geometry for meshing.

When designers create CAD geometry, their priorities are different from those of analysts trying to use the data. A single smooth surface is typically split into smaller patches, each a separate mathematical face. The juncture between two surfaces often contains gaps, overlaps, or other misalignments.

To make the geometry more appropriate for meshing, analysts need to combine a number of faces into a single smooth surface. This allows the elements to be created on the entire region at once, and prevents unnecessary artificial or accidental edges from being present in the final mesh.

The Quick Edit, Edge Edit, Point Edit, and Autocleanup panels contain tools to help you prepare surface geometry for meshing.

Figure 1586:

The initial CAD geometry often contains gaps, misalignments, or pinholes.These features can distort the elements or demand a finer mesh.

Figure 1587:

With the tools of the geometry cleanup panels, you can close the gaps between surfaces, combine surfaces into large meshing regions, and eliminate pinholes. Using the simpler, cleaner geometry, you can easily build a much better mesh.

Apply Loads

The final step in the model building process is to apply constraints and forces and to create or assign coordinate systems.

Before you apply loads, create a load collector. Loads are displayed in the color assigned to the load collector. The size of loads and constraints is based on model units and can be modified from within the boundary condition panels.

HyperMesh stores and displays all loads in the global coordinate system. Depending on the analysis code being used to calculate results, HyperMesh transforms the loads appropriately to any local nodal output coordinate system.

HyperMesh currently supports the following load types:

Accelerations
: Applies an acceleration at a node. Accelerations are displayed as a single-headed arrow with an optional label, A. The label may also display the magnitude of the acceleration.

Constraints
: Applies a constraint or enforced displacement at a node. Constraints are displayed as a triangle with an optional label that displays the degrees of freedom affected by the constraint.

Equations
: Applies a general equation constraint between nodes. Equations are displayed with the label, EQ.

Fluxes
: Applies a flux load at a node. Fluxes are displayed as a thick arrow with an optional label, flux. The label may include the magnitude of the flux.

Forces
: Applies a concentrated force along any user-defined vector at a node. Forces are displayed as a single-headed arrow with an optional label F. The label may include the magnitude of the force.

Linear interpolation input files for forces require 6 values with a space in between, as follows:

<X location> <Y location> <Z location> <X component> <Y component > <Z component >

Moments
: Applies a concentrated moment about a user-defined vector at a node. Moments are displayed as a double-headed arrow with an optional label, M. The label may include the magnitude of the moment.

Linear interpolation input files for moments require 6 values with a space in between, as follows:

<X location> <Y location> <Z location> <X component> <Y component > <Z component >

Pressures
: Applies a pressure on an element or geometry. Pressures are displayed as a single-headed arrow with an optional label, P. The label may include the magnitude of the pressure.

Linear Interpolation input files for Pressures require 4 values with a space in between, as follows:

<X location> <Y location> <Z location> <magnitude>

Temperatures

Applies a temperature constraint at a node. Temperatures are displayed as a straight line starting at the node at which the temperature is applied extending upward, with an optional label, T.

Linear Interpolation input files for Temperatures require 4 values with a space in between, as follows:

<X location> <Y location> <Z location> <magnitude>

Velocities

Applies a velocity at a node. Velocities are displayed as a single-headed arrow with an optional label, V. The label may include the magnitude of the velocity.

Note: Refer to the specific panel for detailed information about creating, reviewing, and updating loads and constraints.

Create Systems

Systems are referred to as coordinate systems and may be rectangular, cylindrical, or spherical.

Reference and analysis systems are supported. Reference systems transform geometric location or input vectors from the global system to a local system. Nodes, mass elements, forces, and other systems are eligible entities for a reference system. Analysis systems transform the output system of a node entity. Systems are built and referenced in the Systems panel.

> **Note:** System collectors collect system entities. A system collector must exist and be current in order to build a system.

Tools

Overview of tools used when setting up your model.

Setup Modal Analysis

Define modal analysis cards, such as extraction methods, frequency range, modes to expand, iterative solver tolerance, modes significance level and solution control options.

1. From the menu bar, click **Tools** > **Analysis Setup**.
2. In the **Analysis Options** dialog, set the modal analysis in the model.

   All commonly used cards and options for modal analysis in the ANSYS solver are listed in the dialog. You do not need to search for relevant cards in the control card list, and you do not need to know the control cards that are used for modal analysis. All other implied ANSYS cards, such as /SOLU, SOLVE are set up automatically. If any option or card is not required, then default values will be exported. For example, in the image shown below, if Mass and stiffness matrix multiplier value need not required.

   If modal analysis needs to be carried out in the ANSYS solver, this dialog needs to be set up before exporting the model to an ANSYS deck.

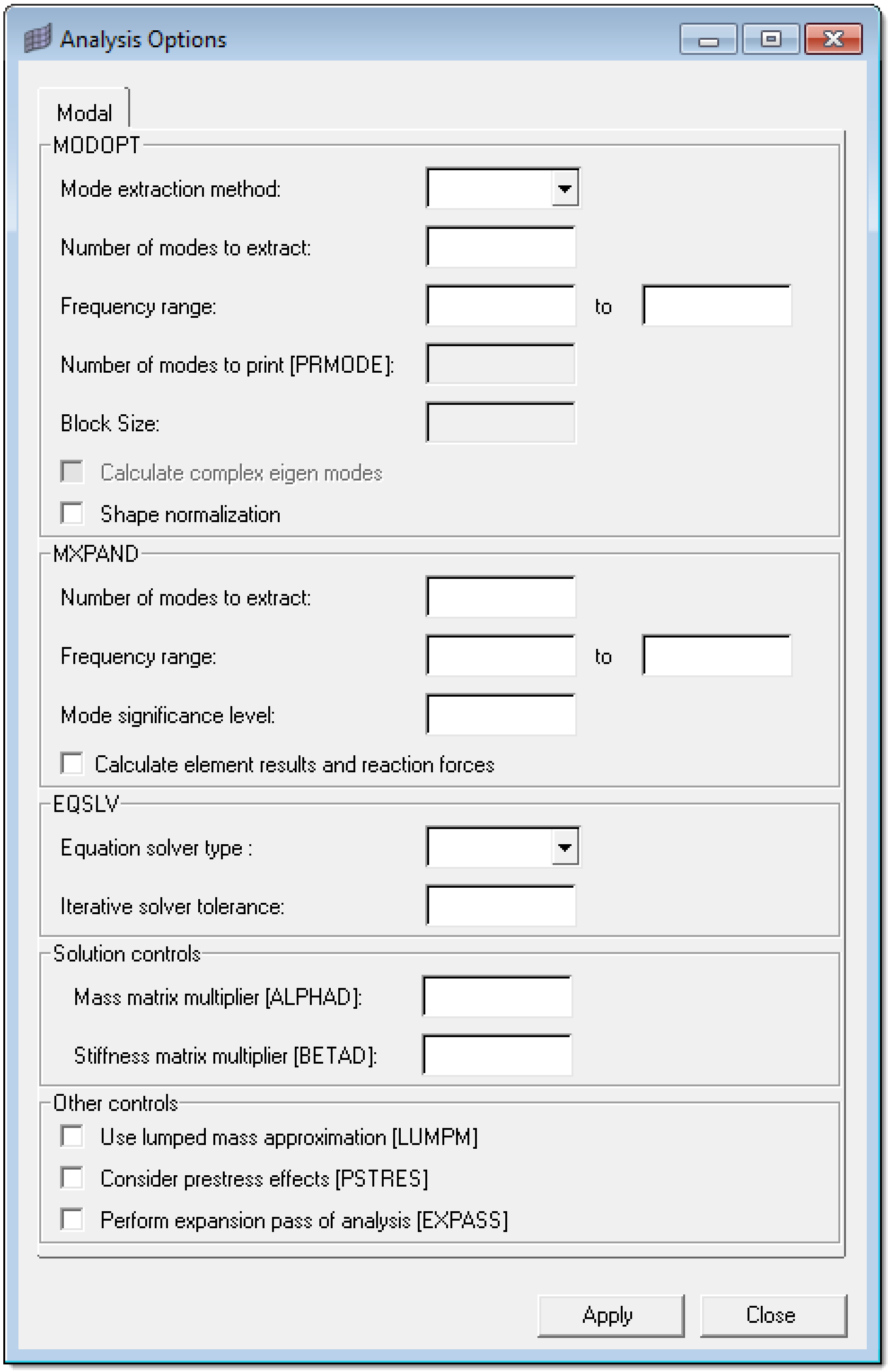


Figure 1588:

Compare Models

Perform a model-based CAD-CAD, CAD-FE or FE-FE comparison between two models, or two selections of entities, and find and report geometrical/shape differences.

When you are comparing entities, the entities must exist in the HyperMesh database. The Comparison tool generates results by comparing the source entities to the target entities. Currentl and 2D elements are supported in the Comparison tool. The results generated by this tool can be broken down into the following categories: Matched, Overlapped, Intersected, and Unmatched.

Open **Comparison** dialog.

- From the menu bar, click **Geometry** > **Check** > **Surfaces** > **Comparison**.
- From the menu bar, click **Mesh** > **Check** > **Elements** > **Comparison**.

Generate Comparison Results

In the Generate tab, define the following options, which will be used to generate the comparison results.

The options available in this tab will vary depending on the Location you select.

1. Click the **Generate** tab.
2. From the Location list, specify where the entities you are comparing can be found.
    - Choose **Auto** to generate results for entities positioned at any location, by automatically calculating a transformation matrix between the source and target entities. A maximum of one match for each source entity will be found. This option is only available for CAD-CAD comparison.
    - Choose **Position** to generate results for entities positioned at a location that you specify. For the Source entities and the Target entities collectors, select up to three locations on the source that correspond to the same three locations on the target. This option will use these locations to define the transformation of the source entities to the target entities. A maximum of one match for each source entity can be found.
    - Choose **Recursive** to generate results for entities positioned at any location, by automatically calculating transformation matrices between the source and target entities. Multiple matches can be found for each source entity. A source entity may match more than one target entity positioned at a different location. This option is only available for CAD-CAD comparison.
    - Choose **Rotate** to generate results for entities rotated about a vector. Multiple matches may be found for each source entity, depending:

      Axis
      Select a collector that defines the rotation axis/vector.

      Angle
      Specify the rotation angle to search and find target entities.

      Steps
      Specify the number of times to increment the angle value when searching.
      For example, in the image below, assume the red element is the source, the blue elements are the target, and the z-axis is the rotation vector. If the angle = 60 and the steps = 1, then only the first blue element will be found. If the steps = 2, then the first two blue elements will be found. If the steps = 3, then all three blue elements will be found. Similarly, if the angle = 30 and the steps = 1, then no blue elements will be found, but if the steps = 2, then the first blue element will be found.

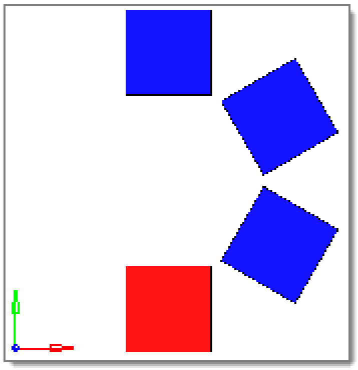


Figure 1589:

- Choose **Same side** to generate results for entities positioned at the same location. A maximum of one match can be found for each entity source.
- Choose **Symmetry** to generate results for entities positioned symmetrically about a plane. A maximum of one match can be found for each source entity. When you select this option, the Symmetry plane selector becomes active. Use the Symmetry plane selector to select the plane about which the symmetry exists.
- Choose **Translate** to generate results for the entities translated along a vector. Multiple matches may be found for each source entity, depending on:

  Direction
  : Select a collector that defines the translation vector.

  Distance
  : Specify the translation distance to search and find target entities.

  Steps
  : Specify the number of times to increment the distance value when searching.
  For example, in the below image, assume the red element is the source, the blue elements are the target, and the x-axis is the direction vector. If the distance = 2 and the steps = 1, then only the first blue element will be found. If the steps = 2, then the first two blue elements will be found. If the steps = 3, then all three blue elements will be found. Similarly, if the distance = 1 and the steps = 1, then no blue elements will be found, but if the steps = 2, then the first blue element will be found.

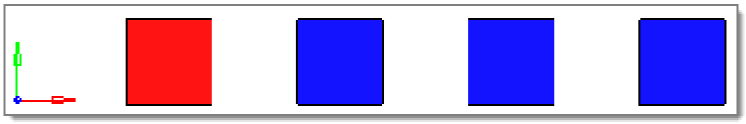


Figure 1590:

3. In the Tolerance field, enter the tolerance to use for comparing entities, in model units.

   A smaller value will lead to a more strict interpretation, while a larger value will lead to a looser interpretation.

The tolerance is not used to remove features smaller than the tolerance value. All features for a surface or mesh are considered valid for comparison regardless of their size, and must be within the tolerance of a similar feature on the compared entities for the surfaces or elements to be considered matched or overlapped.

4. For Result type, select the level of detail for the results.

Higher levels of detail require more calculation and processing time. Each level of detail produces different results based on the type of comparison you are performing. For detail description of the result types, refer to Comparison Result Types.

5. Optional: Perform a comparison a shells using their thickness.

a) Select the **use thickened shells** checkbox.

> **Restriction:** Only available when you are comparing FE-CAD/CAD-FE.

b) To keep the internally calculated fillet distance relatively small, meaning that elements near fillet regions must also be close to the surfaces to prevent them from being marked as intersected, select the **Match fillets** checkbox.

Fillets are assumed to lie nearby fillet roots, which are located at all t-intersections and bends in the mesh of at least 44 degrees. Any nodes within an internally calculated distance of a fillet root are treated as being inside the fillet, and therefore they cannot cause an element to be marked as intersected. When you clear the Match fillets checkbox, the internally calculated distance will be relatively large, preventing many elements in fillet regions from being marked as intersected if they are not close to the surfaces.

The internally calculated fillet distance is based on the thickness of the elements touching the fillet root and the average thickness in the model. It is not a precise determinant of the actual fillet region and is only intended to prevent elements that are not close to the surfaces due to the presence of fillets from being marked as intersected. As a result, most models with fillets will have elements in the fillet regions marked as overlapped or intersected rather than matched.

> **Note:** Clearing the Match fillets checkbox may cause the internally calculated fillet distance to be rather large for certain models, and may potentially mark elements as overlapped when they would otherwise be marked intersected due to the element's edges not matching surface lines.

c) Use the Ignore holes/slots option to specify which holes and/or slots in the surface data to ignored during the comparison.

If the features are smaller than the specified Holt/slot diameter and/or Slot length criteria, then they will be ignored. Element edges in the region of an ignored hole or slot are not required to be within the tolerance of a surface line. Element nodes over an ignored hole or slot cannot cause the element to be marked as intersected.

Figure 1591:

6. To use the normal projection of the centroid of an element to help determine with which surface an element should be associated, select the **Include element centroids** checkbox.

Without this option, elements are associated with a given surface based on the largest number of their nodes which are within the given tolerance of that surface. If an element has the same number of nodes within the given tolerance of more than one surface the association will be made arbitrarily. This option uses the projection of the element centroid to remove any ambiguity as to which surface is the best fit.

Additionally, elements that lie partially or completely off the edge of a surface will be marked as unmatched instead of intersected if the projection of the centroid does not lie on any surface. This option is particularly useful when comparing coarse meshes to highly curved surfaces.

This option is turned off by default because it can substantially increase the time spent comparing meshes if there are a large number of ambiguous elements.

> **Restriction:** Only available for CAD-FE/FE-CAD comparisons.

7. To refacet all the surfaces in the model so that their facet size will be roughly equal to that of the compared elements, select the **Refined facets** checkbox.

This option is particularly useful when comparing highly curved geometries to FE meshes with a tight tolerance.

The Comparison tool uses surface facets, rather than the surfaces themselves, to quickly compare elements to surfaces or surfaces to surfaces. Often the default facets can be long, thin triangles which only roughly capture the surface curvature in highly curved areas. While the default facets can often be "close enough" to the surfaces they represent, nodes of an element which lie exactly on the surface may actually lie outside the given tolerance of the nearest surface facet. If this happens, the elements attached to that node will be marked as intersected or unmatched instead of overlapped or matched.

One solution to this problem is to increase the tolerance, but there are times when using a large tolerance will be undesirable. For those times you can select this option which will recalculate the facets for the surfaces such that they more closely adhere to the actual surfaces. The result will be more accurate results with more elements marked as matched or overlapped instead of intersected or unmatched.

This option is turned off by default because it can substantially increase the time spent making the comparison.

> **Restriction:** Only available for CAD-FE/FE-CAD comparisons.

8. Click **Run**.

Comparison Result Types

Overview of the different result types you can select when working with the Comparison tool.

Standard Comparison

Table 263: Overview of Standard Comparison Result Type

	CAD-CAD	CAD-FE/FE-CAD	FE-FE
Basic	Matched (paired) Unmatched	Matched (paired) Unmatched	Matched Unmatched
Full	Matched (paired) Overlapped Intersected Unmatched	Matched (paired) Overlapped Intersected Unmatched	Matched Overlapped Intersected Unmatched
Detailed	Matched (paired) Overlapped (paired) Intersected (paired) Unmatched	Matched (paired) Overlapped (paired) Intersected (paired) Unmatched	Matched Overlapped Intersected Unmatched

CAD-CAD

Result Type	Description
Basic	**Matched** Occurs when a source or target surface is within the given tolerance of a compared surface using a direct surface to surface comparison. All points and lines comprising each surface must match between the surfaces. Each matched surface is placed in a separate match type group with the surface it matches. **Unmatched** Occurs when a source or target surface differs from the closest compared surface to a degree larger than the tolerance.
Full	**Matched** Occurs when a source or target surface is within the given tolerance of a compared surface using a direct surface to surface

Result Type	Description
	comparison. All points and lines comprising each surface must match between the surfaces. Each matched surface is placed in a separate match type group with the surface it matches. **Overlapped** Occurs when all facet nodes of a source or target surface are within the given tolerance of the compared surfaces and all of the facet nodes on the nearest compared surfaces' exterior edges are within the given tolerance of the source or target surface's exterior edges. All of the overlapped surfaces are placed in a single match type group. **Intersected** Occurs when at least one but not all facet nodes on a source or target surface are within the given tolerance of the compared surfaces or at least one of the facet nodes on the nearest compared surfaces' exterior edges is outside the given tolerance of the source or target surface's exterior edges. All of the intersected surfaces are placed in a single match type group **Unmatched** Occurs when there are no facet nodes on a source or target surface that are within the given tolerance of the compared surfaces, or all the criteria for the intersected match type are met and no interior facet nodes for the surface are within the tolerance of the compared surfaces.
Detailed	Behaves the same as Full, except overlapped and intersected match types are separated out into pairs, or contiguous groups which share the same match type.

CAD-FE/FE-CAD

Result Type	Description
Basic	**Matched** Occurs when a group of elements are found having all nodes within the given tolerance of a surface, the areas defined by the surface and the elements are within 3%, and all of the nodes on the elements' exterior edges are within the given tolerance of the surface's exterior edges. Each matched surface is placed in a separate match type group with the elements it matches. **Unmatched** Occurs for surfaces and elements when the matched criteria are not met.

Result Type	Description
Full	**Matched** Occurs when a group of elements are found having all nodes within the given tolerance of a surface, the areas defined by the surface and the elements are within 3%, and all of the nodes on the elements' exterior edges are within the given tolerance of the surface's exterior edges. Each matched surface is placed in a separate match type group with the elements it matches. **Overlapped** Occurs for a surface when all facet nodes on the surface are within the given tolerance of the elements and all nodes along the nearest FE mesh's exterior edges are within the given tolerance of the surface's exterior edges. It occurs for an element when all of the nodes of the element are within the given tolerance of the surfaces and all of the nodes of the element which touch the FE mesh’s exterior edges are within the given tolerance of the surfaces’ exterior edges. All overlapped surfaces and elements are placed in a single match type group. **Intersected** Occurs for a surface when at least one of the facet nodes on the surface is within the given tolerance of the elements and at least one point along the nearest FE mesh's exterior edges is outside the given tolerance of the surface's exterior edges. It occurs for an element when at least one node of the element is outside the given tolerance of the surfaces, or when at least one of its nodes which touch the FE mesh’s exterior edges is outside the given tolerance of the surfaces’ exterior edges. All of the intersected surfaces and elements are placed in a single match type group. The options Include Element Centroids and Refine Facets can improve the accuracy of the results in both Full and Detailed results modes. Without these options, some elements will be marked as intersected if one of their nodes lies outside of the tolerance of a coarse facet approximating a curved surface. Other elements may get marked as intersected if it is ambiguous to which surface they belong or if they have one or two nodes within the tolerance of any surface but do not overlap it at all. By including element centroids in the calculations and using refined facets, intersected elements and surfaces may be more accurately marked as overlapped or unmatched. **Unmatched** Occurs for a surface when there are no facet nodes on the surface that are within the given tolerance of the elements. It occurs for an element when no nodes of the element are within the given tolerance of the surfaces.

Result Type	Description
Detailed	Behaves the same as Full, except that each surface will be placed in a separate match type group with the elements it overlaps or intersects with. Due to the grouping process, surfaces that are not grouped with any elements are marked as unmatched. In Full results, such surfaces may be marked as intersected if a surface edge is within the given tolerance of an element but there is no overlap in the interior of either. A given surface may be in two match type groups: one with overlapped elements and another with intersected elements.

FE-FE

Result Type	Description
Basic	**Matched** Occurs when an element is within the given tolerance of another element using a direct node to node comparison. All matched elements are placed in a single match type group. **Unmatched** Occurs if at least one node for the element lies outside the given tolerance of the nodes of the compared FE mesh.
Full	**Matched** Occurs when an element is within the given tolerance of another element using a direct node to node comparison. All matched elements are placed in a single match type group. **Overlapped** Occurs when all nodes of a source or target element are within the given tolerance of the compared elements and all of the element’s nodes which touch the FE mesh’s exterior edges are within the given tolerance of compared FE mesh’s exterior edges. All of the overlapped elements are placed in a single match type group. **Intersected** Occurs when at least one node of a source or target element is outside the given tolerance of the compared elements or at least one of the element’s nodes which touches the FE mesh’s exterior edges is outside the given tolerance of the compared FE mesh’s exterior edges. All of the intersected elements are placed in a single match type group.

Result Type	Description
	Unmatched Occurs when there are no nodes of a source or target element that are within the given tolerance of the compared elements.
Detailed	Behaves the same as Full.

Thickened Shell Comparison

When you are performing a thickened shell comparison, Basic, Full, and Detailed all produce the same answer.

CAD

Matched
: There is no criteria for matched surfaces in a thickened shell comparison. However, a value is provided by dividing the matched FE area by two. Surfaces are set to 100% matched if the FE is also 100% matched. Surfaces are never categorized as matched, though, as the overlapped match type is the maximum category.

Overlapped
: Occurs when all points of a surface are within the tolerance of the elements (when offset is projected by thickness). The element edge nodes that are not within the tolerance of a surface edge may be projected on to the surface. This is calculated by dividing the overlapped FE area by two.

Intersected
: Occurs when at least one point of a surface is within the tolerance of the elements (when offset is projected by thickness). This value is adjusted to make sure the percentages add up to 100%.

FE

Matched
: Occurs when all of an element's nodes (when offset is projected by thickness) are within the tolerance of a surface on both sides. If a node is lying along the edge of a mesh, it must also be within the tolerance of a surface line on at least one side.

Overlapped
: Occurs when all of an element's nodes (when offset is projected by thickness) are within the tolerance of a surface on both sides, located over an ignored hole or slot, or within the internally calculated distance of the root of a fillet (any t-intersection or bend in the mesh of at least 44 degrees). If a node is lying along an edge, it must also be within the tolerance of a surface line on at least one side, located over an ignored hole or slot, or within the internally calculated distance of the root of a fillet.

Intersected
: Occurs when at least one of an element's nodes (when offset is projected by thickness) are within the tolerance of a surface on at least one side, located over an ignored hole or slot,

within an internally calculated distance of the root of a fillet (any t-intersection or bend in the mesh of at least 44 degrees), or within the tolerance of any surface edge.

Define Review Settings

In the Review tab, you can control the comparison review mechanism options.

The Review tab is enabled once comparison results are available.

1. Click the **Review** tab.
2. Define the type of review to perform.

To perform | **Do this**

No review

Steps to perform if you do not want to apply any graphical review.

1. For Review type, select **None**.

Comparison

Steps to perform if you want to generate a visual representation of the comparison results graphically.

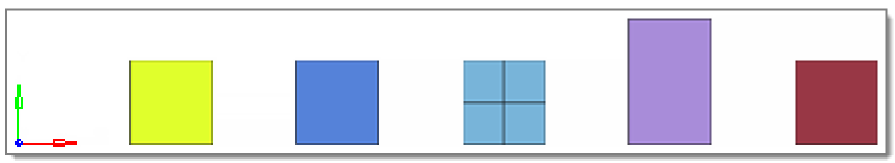


Figure 1592:

1. For Review type, select **Comparison**.
2. For Review colors, specify how to display the results of the match types.
   - Choose **Basic** to display all of the match types by grouping together the matched/overlapped as matched and the intersected/unmatched as unmatched.

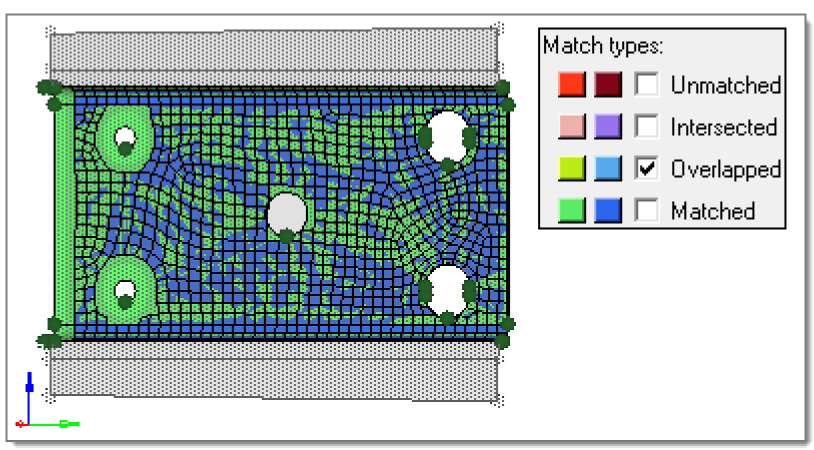


Figure 1593:

To perform	Do this
	• Choose **Detailed** to display all of the match types by their defined colors.

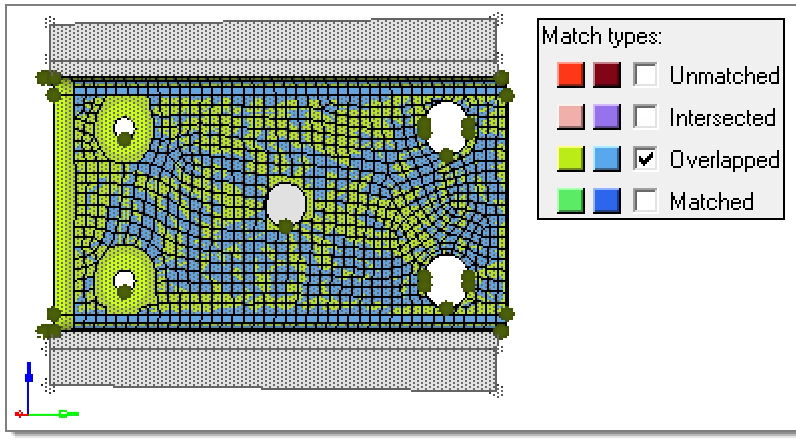


Figure 1594:

3. Under Source/target match types, click the color swatch and then select a new color from the box to control the display colors of each of the match types for both the source (left) and target (right) entities.

To control which match types display in the graphics area, select the check box next to each match type you would like to display. If any entity is part of two match types, then the "worse" type takes precedence.

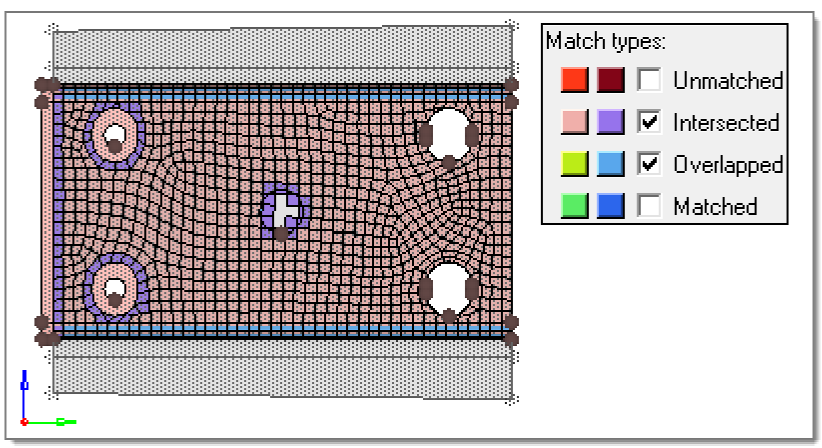


Figure 1595:

If you would like to reset the colors to their default values, click **Reset Colors**.

4. For Transparent, specify whether the source entities, target entities, or no entities at all will be drawn transparent. The entities displayed

To perform	Do this

that are not part of the comparison operation will always be drawn transparent.

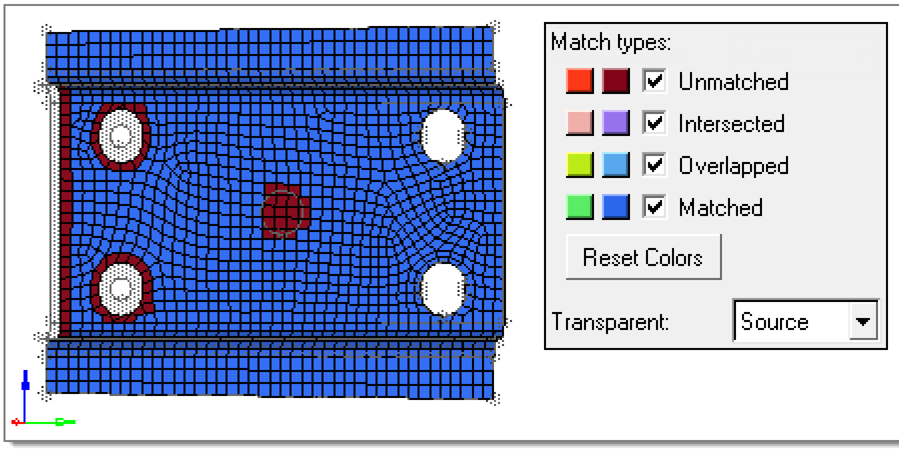


Figure 1596:

5. For Transparent color, specify how transparent entities will be drawn.

- Choose **Gray** to display transparent entities in the default HyperMesh gray color.

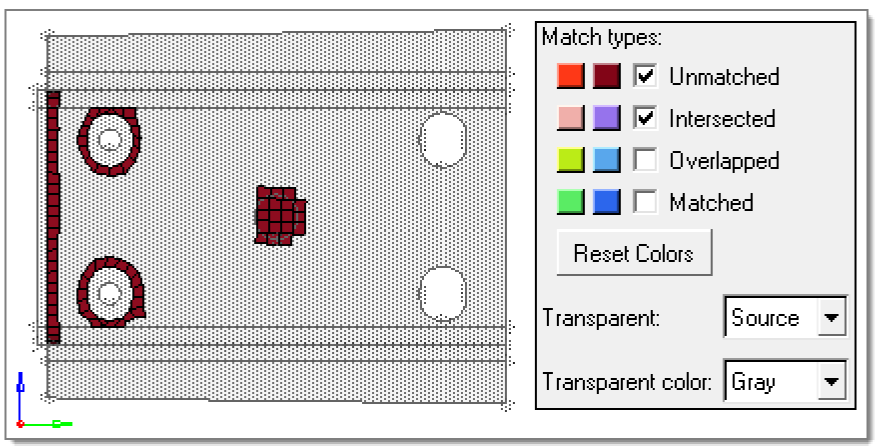
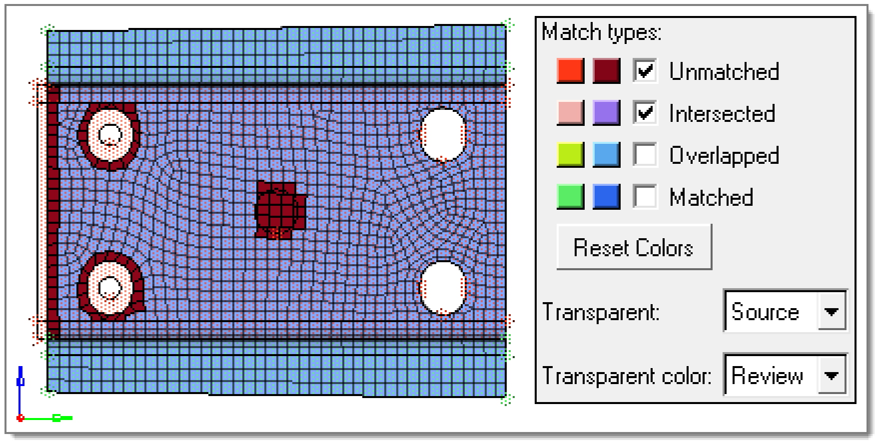
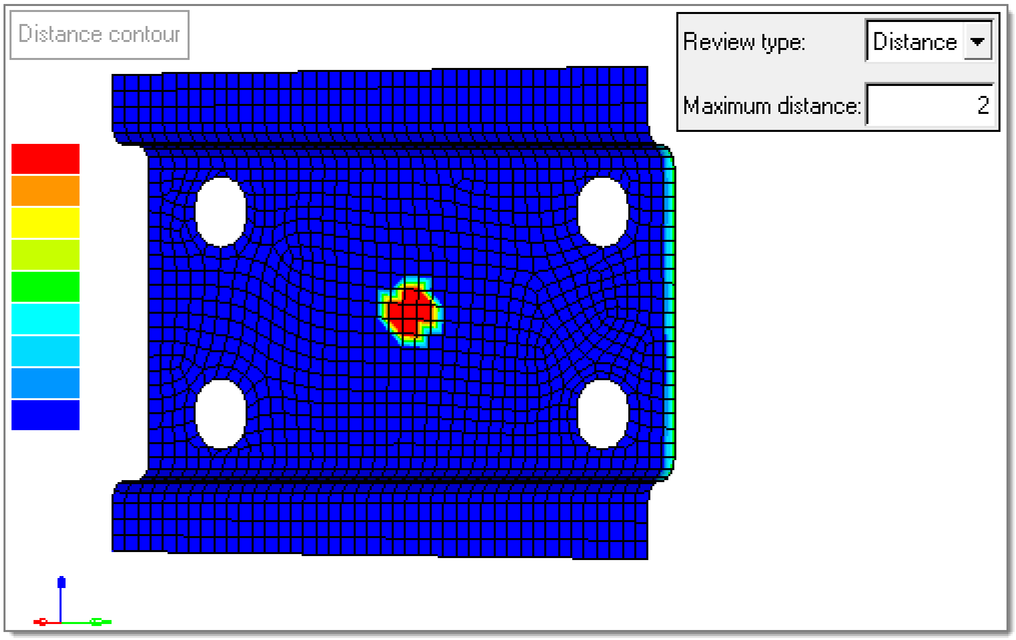


Figure 1597:

- Choose **Review** to display transparent entities in their default color.

To perform	Do this
	 *Figure 1598:*
Distance	Steps to perform if you want to generate a contour plot that shows the distance of the target entities from the source entities. **Restriction:** Not available for CAD-CAD comparisons. **1.** For Review type, select **Distance**. **2.** In the Maximum distance field, enter a maximum value to display in the contour legend.

To perform	Do this
	 *Figure 1599:*

Review Results

In the Results tab, you can review the area-based qualitative results for the comparison.

The Results tab is enabled once comparison results are available.

The area, as a percentage, for the source and target entities of each match type is reported in the Source area% and Target area% fields.

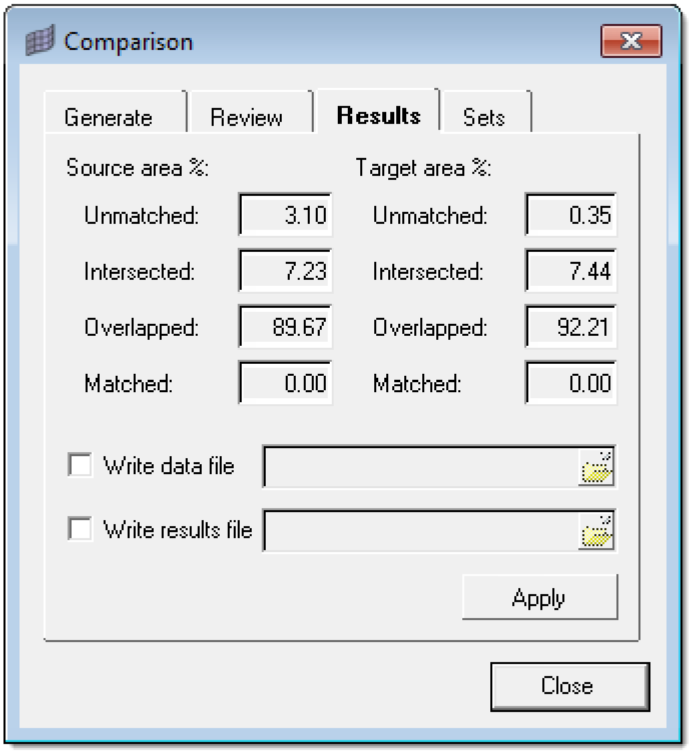


Figure 1600:

1. Click the **Results** tab.
2. Review results.

   When comparing CAD-CAD or FE-FE, the area percentages are calculated by summing the areas of all the entities in each category and dividing them by the total area. However, when comparing CAD-FE or FE-CAD, the Comparison tool uses the areas of the overlapped and intersected elements in place of the overlapped and intersected surfaces when making these calculations. While this may not be technically correct, it results in more accurate area percentage estimates.

   The overlapped and intersected categories are inherently imprecise. The actual lines of intersection between the compared entities seldom lie along the element and surface boundaries and thus cannot be easily determined. If the surfaces and elements were highly refined to a very small scale, you would have no overlapped or intersected entities at all. Instead the elements and surfaces would all be either matched or unmatched and the area percentages could be calculated to a high degree of accuracy. However, since this is not a practical solution, the Comparison Tool tries to get the most accuracy out of the entities you have.

   In general, intersections between CAD and FE usually involve only small areas of the model, for example, a hole in the CAD which is not represented in the FE. The Comparison Tool marks the entire surface as a single match type and chooses the most severe type: intersected. This is despite the fact that the majority of the surface may be overlapped with the FE mesh. If the entire surface is used when calculating the area percentage of the intersected surfaces, that figure may end up being misleadingly high since only a small part of the model is actually intersected. Also, the area percentages between the surfaces and the elements may end up being very different due to the relative size differences between the elements and surfaces, which can be confusing. Is the model 5% intersected as the elements show or 25% intersected as the surfaces indicate? Since elements are generally smaller than surfaces, they generally give a more accurate assessment of the actual amounts of the model which are intersected and overlapped. For this reason, the FE areas are used instead of the surface areas in the area calculations, with care taken to ensure that the percentages all add up to 100%. The result is more accurate results and tighter correlation between element and surface area percentages.

3. Export comparison results to external files.
   a) To write the current comparison data (transformations, match types, match entities, and so on) to a file, select the **Write data file** checkbox.
   b) To write the current comparison results (area%, entity groupings) to a file, select the **Write results file** checkbox.

   Both distance and comparison results can also be exported to an H3D file by clicking **File** > **Export** > **Model** from the menu bar and enabling the **Write results** checkbox.

   For comparison results, the following special behaviors apply:

   - If the source, target, or both are not transparent, and if a match type is turned off, the entities belonging to that match type will not be exported.
   - If the source, target, or both are transparent, all match types are exported regardless of whether they are turned on or off. The components can be made transparent within HyperView or HyperView Player as desired.
   - If the transparent color is gray, the entities are shown as "no result" (which is also gray). Otherwise they are shown in color.
4. Click **Apply**.

Create Entity Sets

In the Sets tab, you can create entity sets from the generated comparison results.

The Sets tab is enabled once comparison results are available.

1. Click the **Sets** tab.
2. Under Source entities, select the corresponding **Source entities** and **Target entities** checkboxes for the entity sets you wish to create.

   One entity set is created for each unique location-type-transformation grouping. A relevant transformation matrix for each grouping is attached as metadata to each entity set created, with the name CompareTransformationMatrix.
3. For Name prefix, enter a prefix for the generated entity set names.

   HyperMesh generates the set names as:: <prefix>_Transform<#>-<Type>-<Source|Target>_<#>

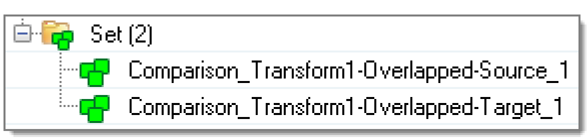


Figure 1601:

4. Click **Create**.

Setup DDAM Analysis

Setup a DDAM analysis for Abaqus, and use the Shock Spectra curves for a subsequent Response Spectum Analysis.

Before you can setup a DDAM analysis, a normal mode result must be available for the model. Abaqus `*.dat` files are supported.

1. From the menu bar, click **Tools** > **DDAM**.
   The **HM DDAM Utility** dialog opens.
2. In the Import Result File field, browse to the location of the results file.
3. Click **Import**.

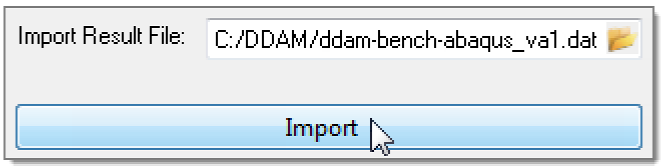
   

   Figure 1602:

4. Select shock modes for the DDAM process.
   a) Under Shock Modes, select a type of mode selection.
      - Choose **Significant** to automatically select modes based on the modal weight until a cumulative contribution is met.
      - Choose **All** to automatically select all of the modes. This option is not realistic for very large models and frequency ranges due to the size of output files. It is recommended to use this option with smaller output files.
      - Choose **Manual** to manually select any selection of modes.
   b) Under Direction, select a direction.
   c) If you selected Manual, select modes from the table.

      Note: By default, modes that have the largest contribution until the minimum cumulative model weight of 80% is achieved are selected. These modes are highlighted in grey. By default, these modes are shown for the contribution in the X-Direction.

      The summation of percent modal weight currently selected is displayed in the the Percent Selected Modal Weight field. Green indicates the percentage is greater than the minimum cumulative modal weight, yellow indicates the percentage is close to the minimum cumulative modal weight, and red indicates more modes are needed.

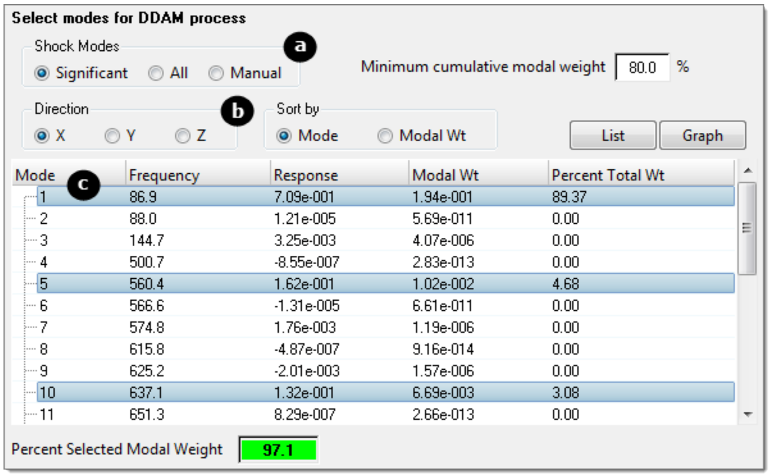


Figure 1603:

5. Determine DDAM loading values.

a) Select a **Ship Type**.

b) Select a **Mounting System**.

c) Select a **Deformation Type**.

d) To manually define loading values, select **Specify values**.

> **Note:** By default, the loading values are pre-populated with the NREL 1396 values.

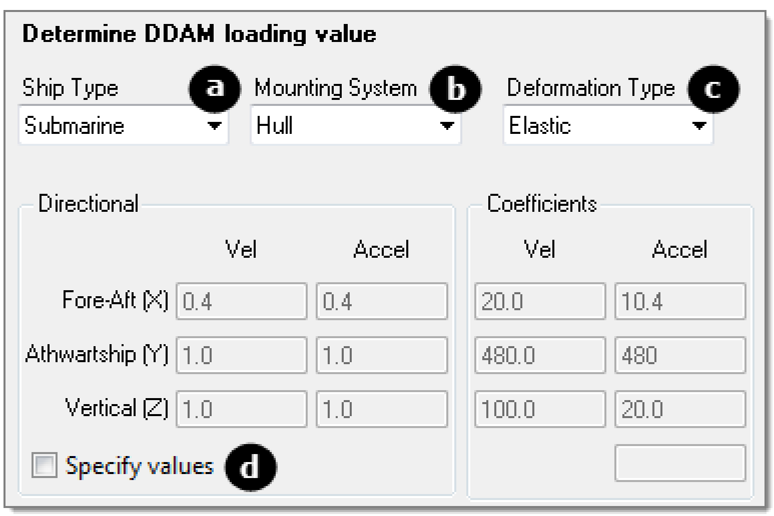
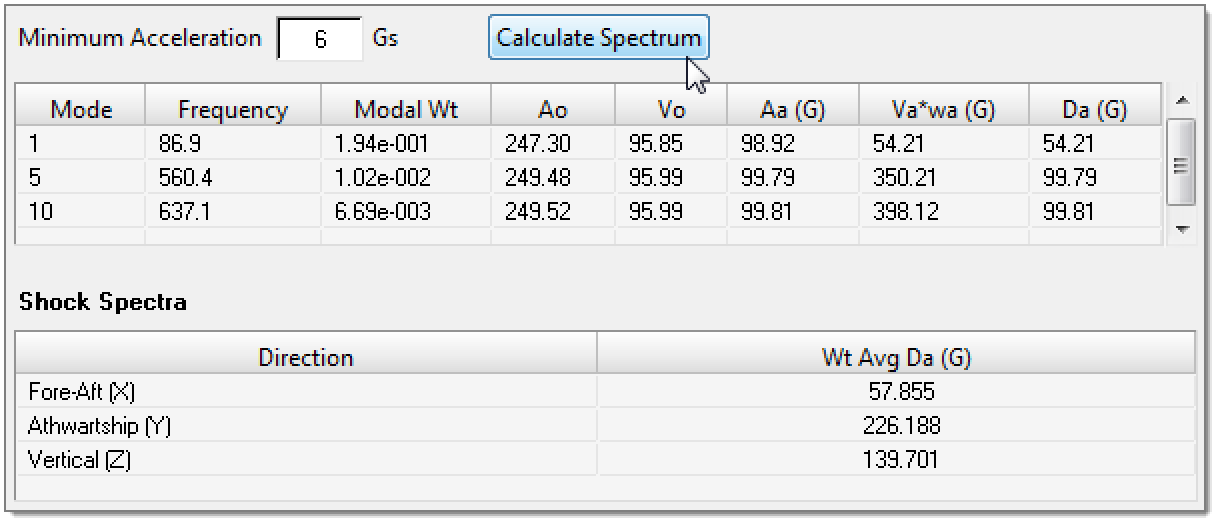


Figure 1604:

6. Click **Calculate Spectrum**.
The Shock Spectra curves are calculated, and the mode table and weighted average Da values are populated.



Minimum Acceleration 6 Gs Calculate Spectrum

Mode	Frequency	Modal Wt	Ao	Vo	Aa (G)	Va*wa (G)	Da (G)
1	86.9	1.94e-001	247.30	95.85	98.92	54.21	54.21
5	560.4	1.02e-002	249.48	95.99	99.79	350.21	99.79
10	637.1	6.69e-003	249.52	95.99	99.81	398.12	99.81

Shock Spectra

Direction	Wt Avg Da (G)
Fore-Aft (X)	57.855
Athwartship (Y)	226.188
Vertical (Z)	139.701

Figure 1605:

7. View a plot of the Shock Spectra curves.

a) Go to the Plots tab and select a **Plot Type**.

- Choose **Acceleration vs. Frequency** to display a line chart, which is only available after shock spectra calculation.
- Choose **Frequency vs. Mode** to display a bar chart, which is available after importing a results file.
- Choose **Effective weight vs. Frequency** to display a bar chart, which is available after importing a results file.

b) Select a **Shock Mode**.

c) Select a **Direction**.

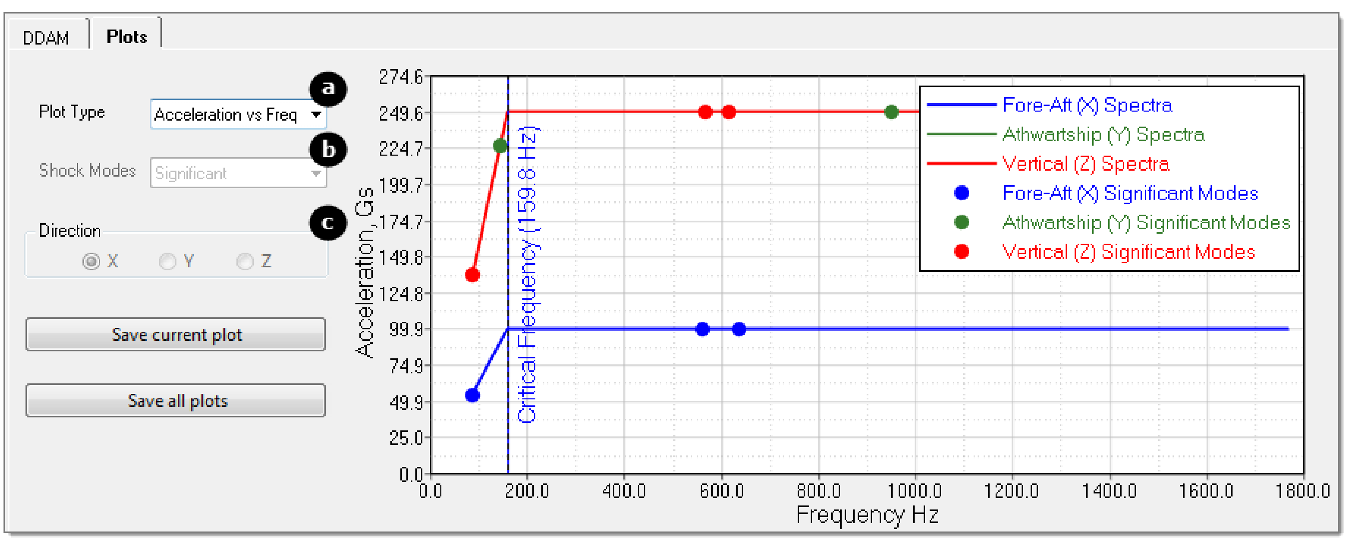


Figure 1606:

8. Go to the DDAM tab and click **Create Loadsteps**.
   Shock Spectra curves are created in the form of load steps, and organized in the Model Browser, Load Step folder.

Fatigue Process

Create or open a fatigue analysis setup process.

Restriction: Only available in the OptiStruct solver interface.

Create New Session

1. From the menu bar, click **Tools** > **Fatigue Process** > **Create New**.
2. The **Create New Session** dialog opens.
3. Create a new fatigue analysis setup process and save it to your working directory.
4. Click **Create**.

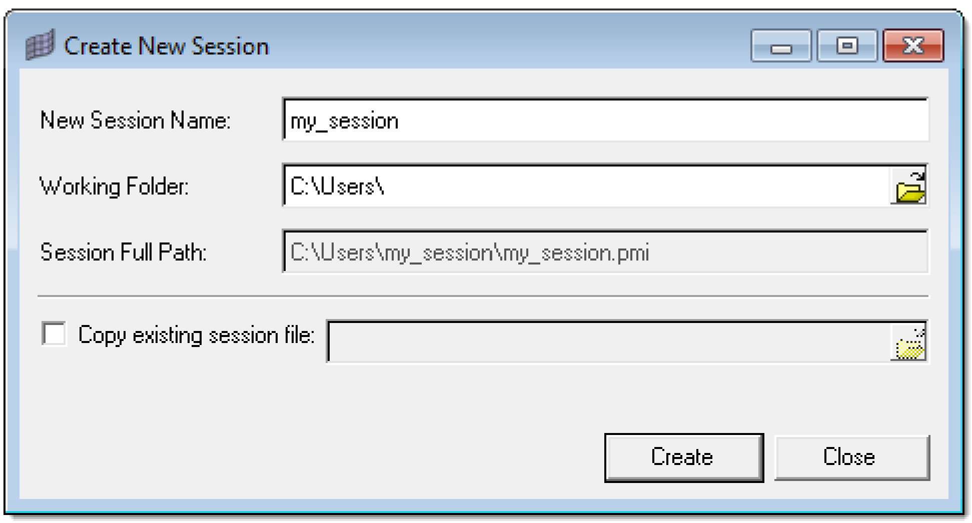


Figure 1607:

Load Existing Session

1. From the menu bar, click **Tools** > **Fatigue Process** > **Load Exist**.
2. The Process Manager browser opens.
3. In the Load template field, load a fatigue analysis setup process that has been previously saved to a file.

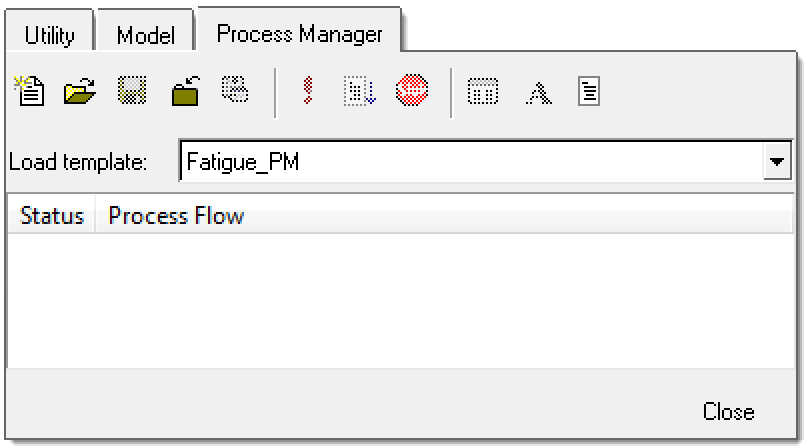


Figure 1608:

Frequency Response Process

Setup a frequency response analysis with a series of step by step process templates.

These process templates prompt you to enter generic engineering data, and once the data is available, automatically create frequency response analysis loadcases and associated solver cards. The templates share a set of common process steps that are only described the first time they are used.

Setup a frequency response analysis by clicking **Tools** > **Freq Resp Process** from the menu bar, and selecting one of the step by step process templates.

> **Restriction:** Only available in the OptiStruct and Nastran user profiles.

Setup a Normal Model Process

The Normal Modes loadcase Process Manager automatically generates a loadcase template, which can be used in the Analysis Manager for future normal modes loadcases without going through the Process Manager again.

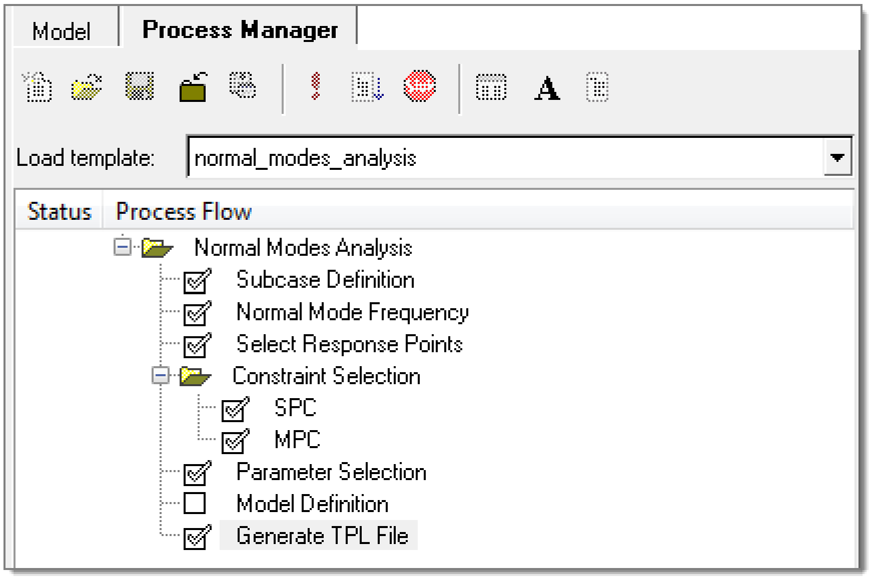


Figure 1609:

1. From the menu bar, click **Tools** > **Freq Resp Process** > **Normal Modes**.

   A normal modes analysis process template is loaded and opens in the Process Flow browser.
2. In the Subcase Definition task, create a new subcase or edit an existing subcase.
   a) Create or edit a subcase.
      - To create a new subcase, edit the optional subcase label and click **Add**.
      - To edit an existing subcase, highlight an existing subcase in the list box, edit the optional subcase label and click **Update**.

   b) Once input to the task is complete, click **Apply** to proceed.

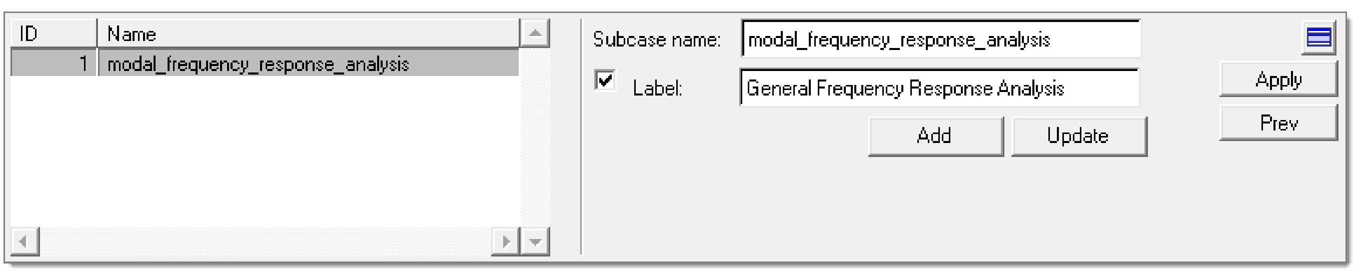


Figure 1610:

3. In the Normal Mode Frequency task, define Normal Mode Frequency.
   a) Enter normal modes extraction parameters, such as Min. and Max. frequency, or the number of modes desired.

      In most cases, only input for the Max. frequency field is needed. When left blank, the Min. frequency is interpreted as 0 (Hz.), and the No. of modes is however many found between 0 and the Max. frequency. This task gives you separate control over modal extraction in the structural and fluid domains.

b) Once input to the task is complete, click **Apply** to proceed.

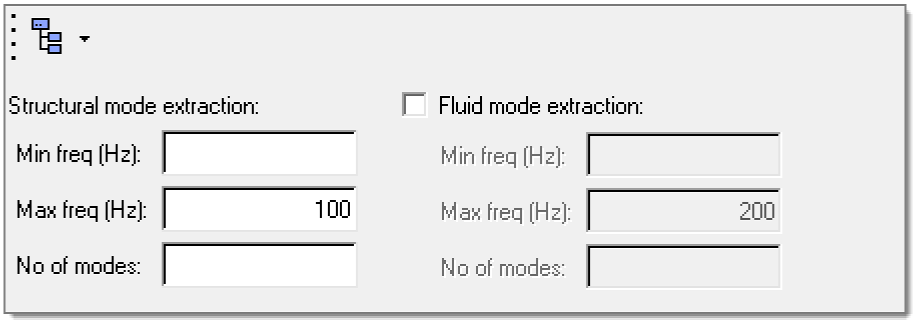


Figure 1611:

4. In the Select Response Points task, select response points for output.

a) Select a Response type, such as Displacement.

b) Use the Entity selector to select nodes, a node set, or tags to be included in a particular response set.

You can add or delete multiple response sets from the list using the **Add row** and **Delete row** icons to the right of the list.

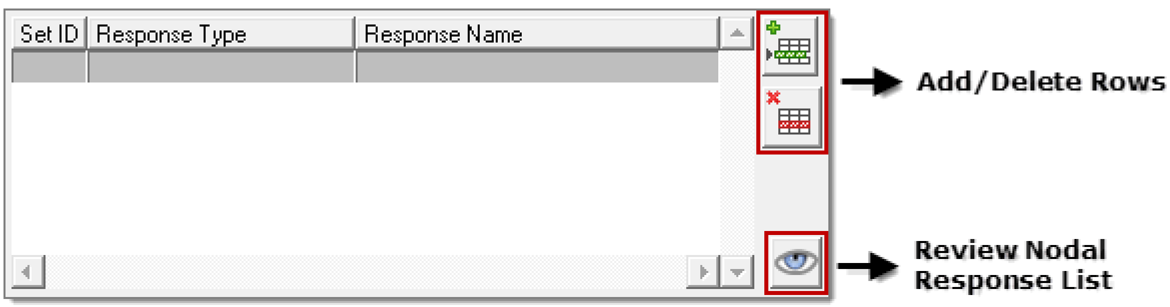


Figure 1612:

c) Under Output data format, select the complex frequency response data format (real/imaginary or magnitude/phase).

d) Under Output file format, select the output file format (h3d, punch, or op2).

e) Once all the required responses have been defined, click **Apply** to proceed.

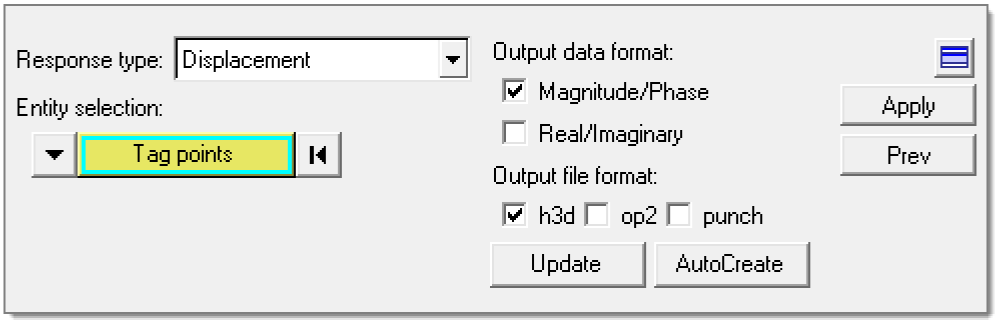


Figure 1613:

5. In the Constraint selection, SPC task, select the boundary condition of the frequency response analysis.

a) You can select existing SPCs by checking the corresponding box under the Active column, or click **Create SPC** to go to the Constraints panel and define a new SPC.

b) Once the boundary condition has been fully defined, click **Apply** to proceed.

6. In the Constraint selection, MPC task, select MPC equations to turn on for the frequency response analysis.

a) Select MPCs.

- NVH user profile: There are two options for managing MPCs, the Analysis Manager and the Process Manager. If you select the Analysis Manager, the Select subcase option and the MPCs list are disabled. If you select the Process Manager, after you select a subcase you can then select existing MPCs by checking the corresponding box under the Active column.
- HyperMesh user profile: The Analysis Manager option is not available, and the MPCs can only be managed through the Process Manager.

b) Once the MPC equations have been selected, click **Apply** to proceed.

7. In the Parameter Selection task, select typical solution parameters, such as title, singular point constraints, and so on.

a) Check all boxes under the Active column to activate the desired solution option.

b) Once the parameters have been selected, click **Apply** and a Process Manager message box pops up informing you that the process has come to an end.

c) Click **Yes** to close the template, or **No** to review or edit the process steps.

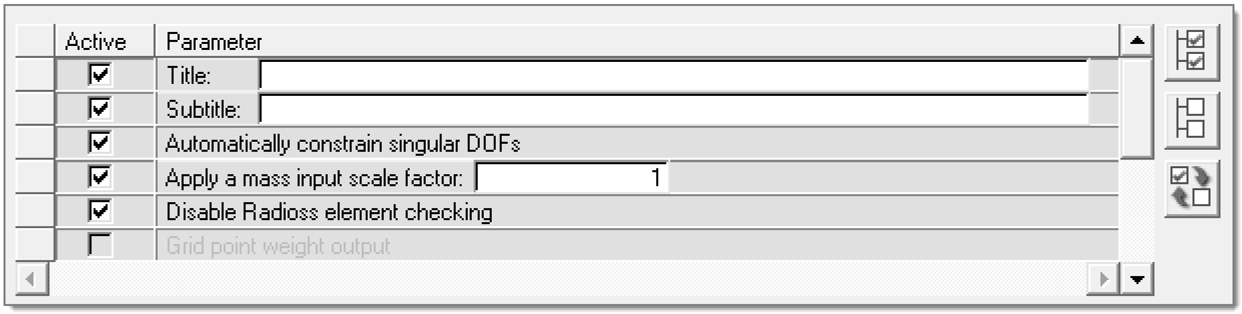


Figure 1614:

8. In the Generate TPL file task, generate and save the parameters defined in a standard template file at your selected location.

In the Analysis Manager, the generated template file can be selected as loadcase and solver decks can be exported for normal modes analysis.

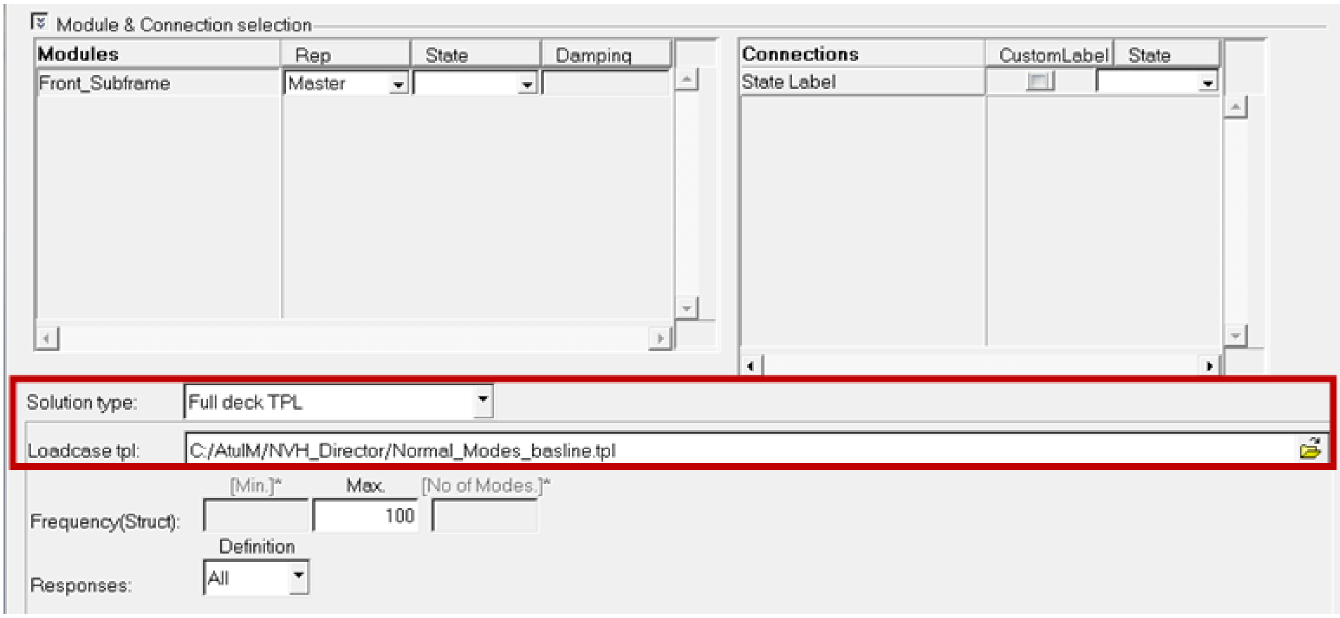


Figure 1615:

Setup CMS SE Generatin Process

The CMS SE Generation process template helps you generate a CMS SuperElement (SE) modal model from a finite element based model.

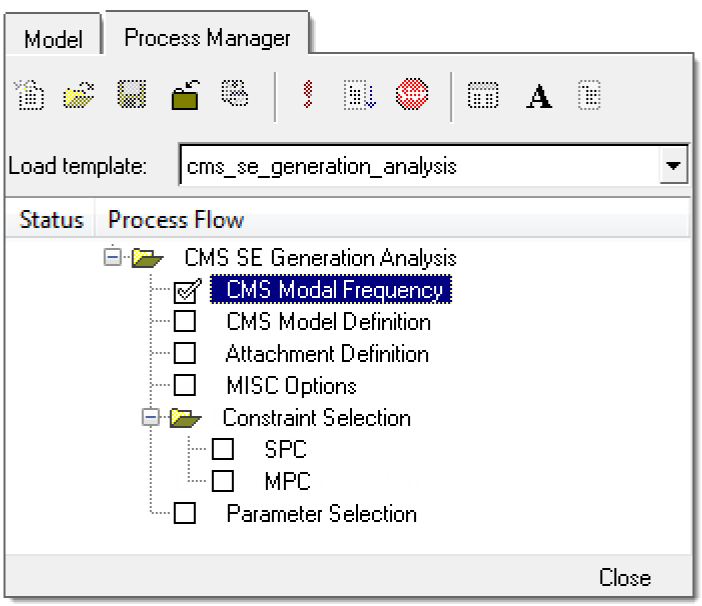


Figure 1616:

1. From the menu bar, click **Tools** > **Freq Resp Process** > **CMS SE Generation**.

2. In the CMS Modal Frequency step, define the CMS modal frequency.

a) For CMS method, select a type of CMS SE.

- Choose **Craig-Bamption (CBN)** to create a fixed boundary Craig-Bamption (CBN) CMS superelement.
- Choose **GM - general modal method** to create a mixed (free-free or fixed) boundary (GM - general modal method) CMS superelement.

b) Define the modes to be included by specifying either the upper frequency or the number of modes, as well as Spoint starting IDs to be assigned to the modes.

The Spoint ID fields are auto filled with the first available ID provided by the HyperMesh database. Each mode added to the CMS SE will be assigned a Spoint ID sequentially from the starting ID, and the total number of Spoint IDs actually used will be the same as the number of modes found.

c) Fluid modes, as well as fluid-structure coupling matrix can be included in the CMS SE by selecting the **Coupled fluid-structure SE** checkbox, and providing the above mentioned frequency and Spoint ID definition for the fluid modes.

d) Once the task has been completed, click **Apply** to proceed.

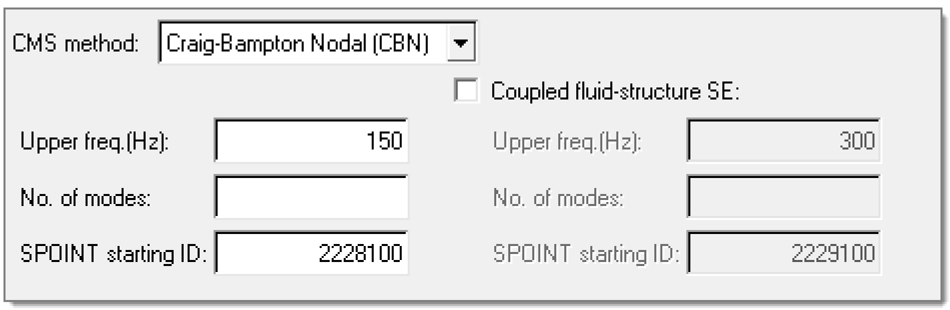


Figure 1617:

3. In the CMS Modal Definition task, specify what recovery information is to be stored in the CMS SE.

a) Select a set of elements, for which the keyword Plotel is a valid specification, or a set of grids.

b) To exclude/include rigid elements from the recovery set, select the **Rigid** checkbox.

c) Once the model recovery set has been defined, click **Apply** to proceed.

Figure 1618:

4. In the Attachment Definition task, specify attachment point sets.

a) Both fixed and free-free attachments can be specified for a mixed (GM) CMS SE, while only fixed attachments can be specified for a fixed (CBN) CMS SE.

b) Once the attachment set has been defined, click **Apply** to proceed.

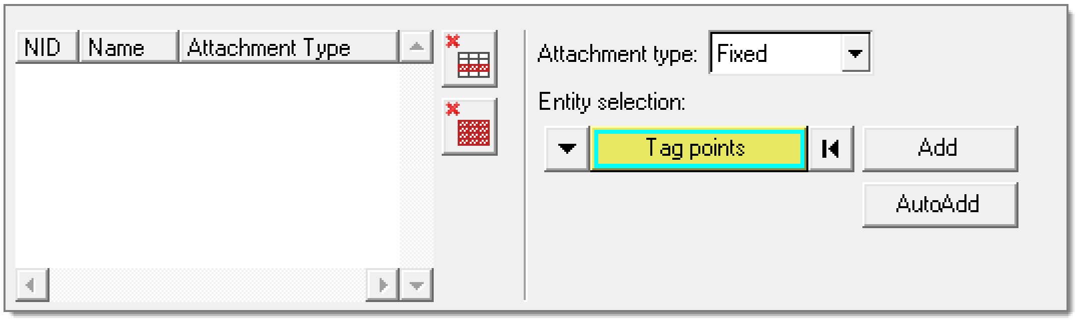


Figure 1619:

5. In the MISC Options task, specify structural damping and fluid-structure coupling options.

a) You can specify a global material damping to be assigned to all structural parts, or select the option of **No damping** to avoid adding any damping globally if damping has been specified at the material level for all components.

> **Note:**
>
> Global modal viscous damping or global fluid damping cannot be stored in a CMS SE, but can be applied in assembly (residual) runs using the CMS SE as a component.

b) For fluid-structure coupling, choose **Solver auto-search** driven by the ACMODL card, or **Akusmod** which assumes that coupling is provided in a binary file named `ftn.70`.

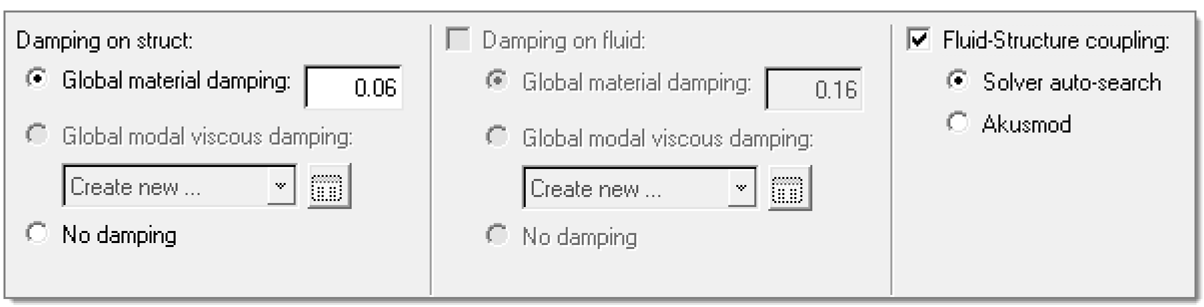


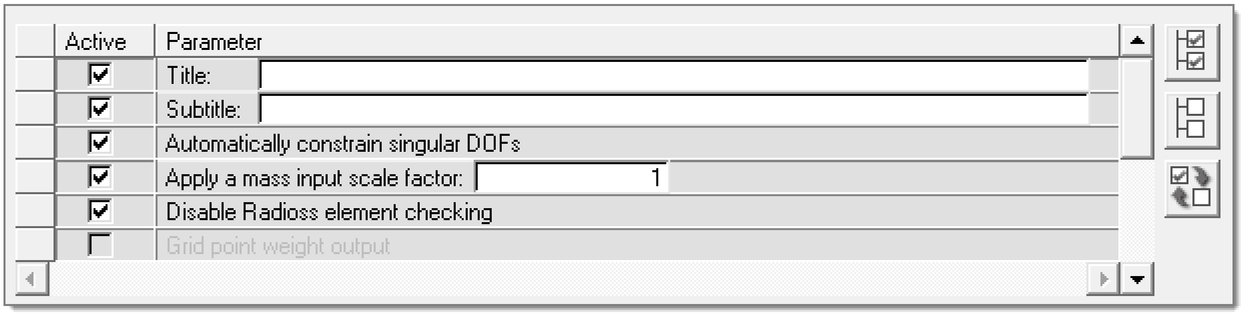
Figure 1620:

6. In the Parameter Selection task, select typical solution parameters, such as title, singular point constraints, and so on.

a) Check all boxes under the Active column to activate the desired solution option.

b) Once the parameters have been selected, click **Apply** and an Process Manager message box pops up informing you that the process has come to an end.

c) Click **Yes** to close the template, or **No** to review or edit the process steps.



Setup CDS SE Generation Process

Generate a CDS SuperElement (SE) from a finite element based model.

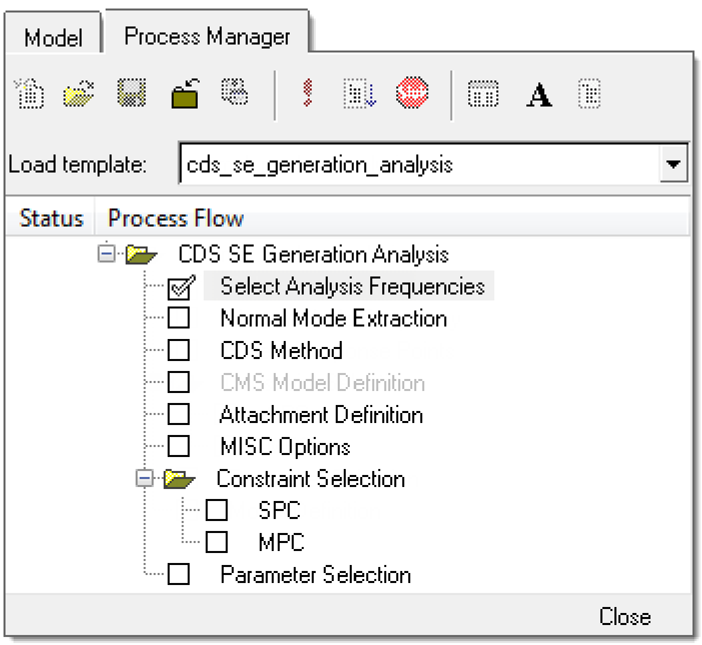


Figure 1621:

1. From the menu bar, click **Tools** > **Freq Resp Process** > **CDS SE Generation**.
2. In the Select Analysis Frequencies task, enter the frequencies for which response solution is needed.
    a) Define the frequency set.
        - To define the min, max, and a linear step, select **Frequency range**. For Incr type, select **Linear** and then fill in the required fields.
        - To define min, max, and a number of increments with logarithmic spacing, select **Frequency range**. For Incr type, select **Logarithmic** and then fill in the required fields.
        - To define an arbitrary list of frequencies, select **Frequency list** and enter a list of arbitrary frequencies.

b) Click **Update**.

A frequency set entry is created in the list box to the left. You can add additional frequency sets, or delete one from the list using the Add row and Delete row icons to the right of the list.

c) Once the frequency set(s) have been defined, click **Apply** to proceed.

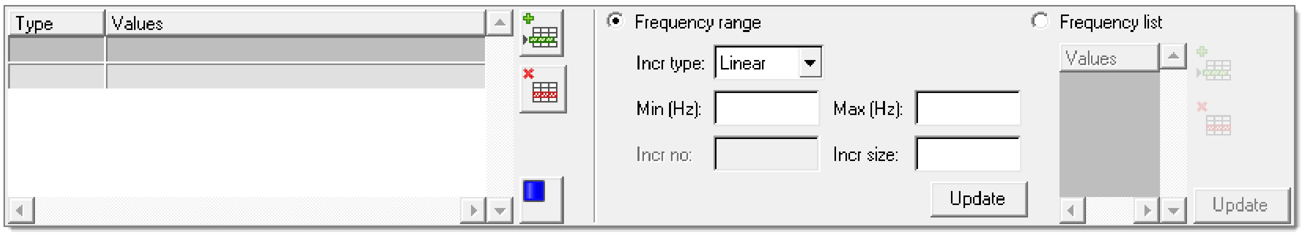


Figure 1622:

3. In the Normal Mode Extraction task, the default frequencies filled in are based on the max. frequency you filled in for the previous step.

a) Modify the values based on the specific requirements of the case under study.

These values are merely the suggested values based on general use cases.

b) Click **Apply** to proceed.

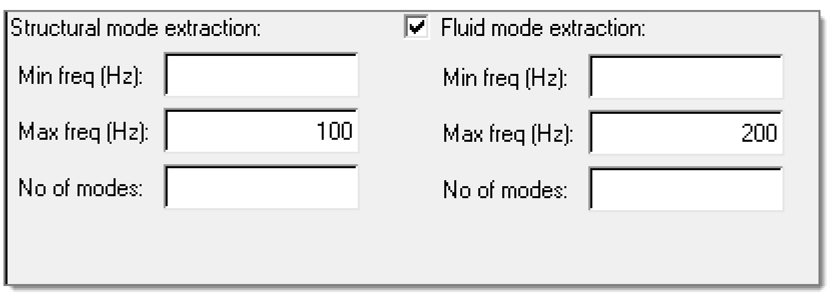


Figure 1623:

4. In the CDS Method task, define the CDS method.

a) For CDS method, select a type of CDS SE.

Currently only the BME method is available.

b) To store transfer functions in CDS superelement, select the **Store transfer functions in CDS superelement** checkbox.

c) To generate a CMS-SE modal model result file after solving, select the **Export CMS superelement file** checkbox.

d) Click **Apply** to proceed.

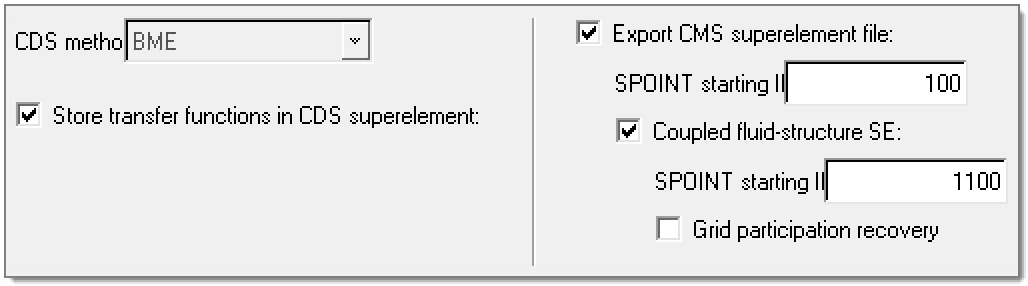


Figure 1624:

5. In the CMS Model Definition task, specify which recovery information is to be stored in the CMS SE.

> **Restriction:** This task is only activated when the **Export CMS superelement file** checkbox is selected.

a) Select a set of elements, for which Plotel is a valid specification, or a set of grids.
b) To include or exclude rigid elements from the recovery set, select the **Rigid** checkbox.
c) Once the model recovery set has been defined, click **Apply** to proceed.

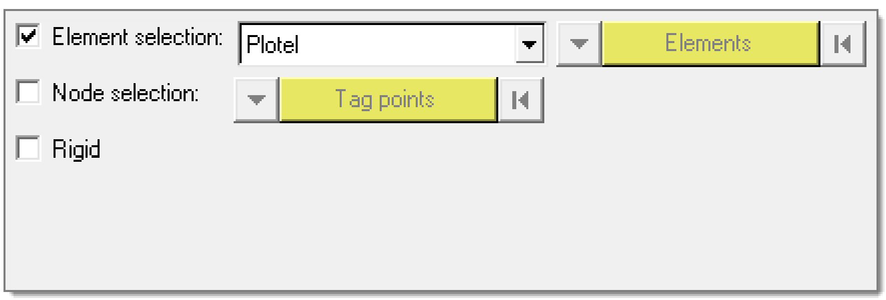


Figure 1625:

6. In the Attachment Definition task, specify attachment point sets.
a) Both fixed and free-free attachments can be specified for a mixed (GM) CMS SE, while only fixed attachments can be specified for a fixed (CBN) CMS SE.
b) Once the attachment set has been defined, click **Apply** to proceed.

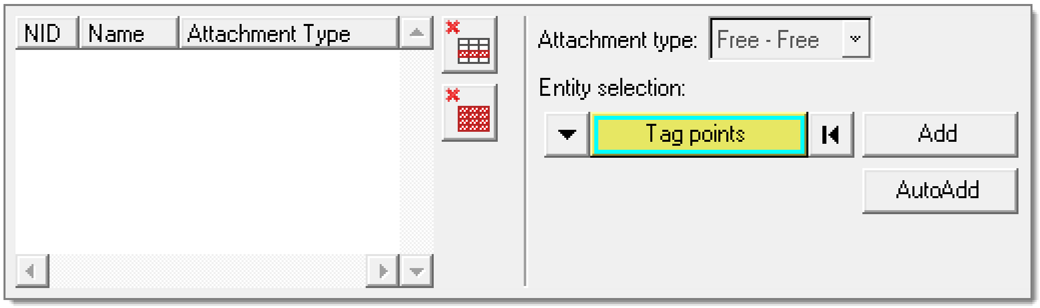


Figure 1626:

7. In the MISC Options task, specify structural damping and fluid-structure coupling options.

a) You can specify a global material damping to be assigned to all structural parts, or select the option of **No damping** to avoid adding any damping globally if damping has been specified at the material level for all components.

> **Note:**
>
> Global modal viscous damping or global fluid damping cannot be stored in a CMS SE, but can be applied in assembly (residual) runs using the CMS SE as a component.

b) For fluid-structure coupling, choose **Solver auto-search** driven by the ACMODL card, or **Akusmod** which assumes that coupling is provided in a binary file named `ftn.70`.

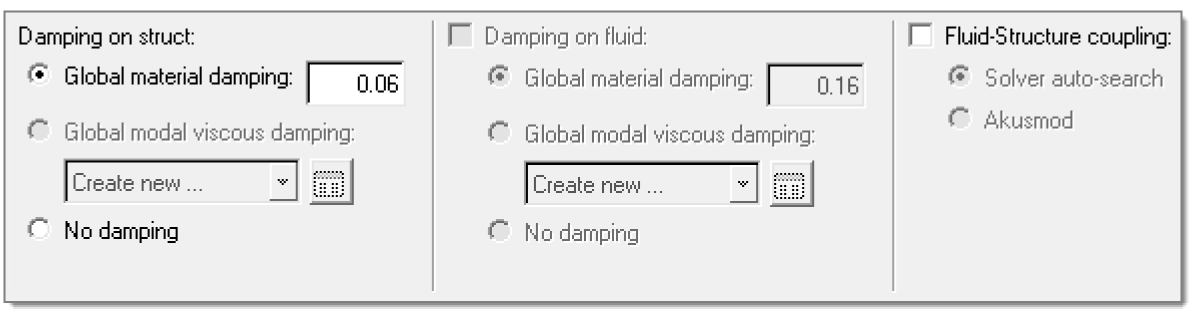


Figure 1627:

8. In the Parameter Selection task, select typical solution parameters, such as title, singular point constraints, and so on.

a) Check all boxes under the Active column to activate the desired solution option.

b) Once the parameters have been selected, click **Apply** and an Process Manager message box pops up informing you that the process has come to an end.

c) Click **Yes** to close the template, or **No** to review or edit the process steps.

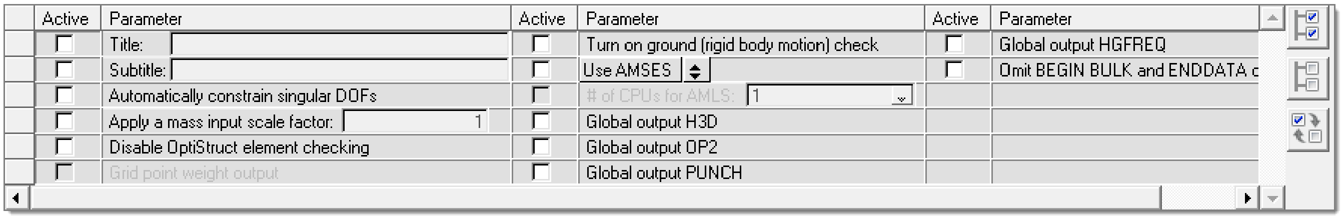


Figure 1628:

Setup Unit Input Frequency Response Process

Set up subcases with unit inputs. In practice, this is common to generate vibration and noise sensitivity results.

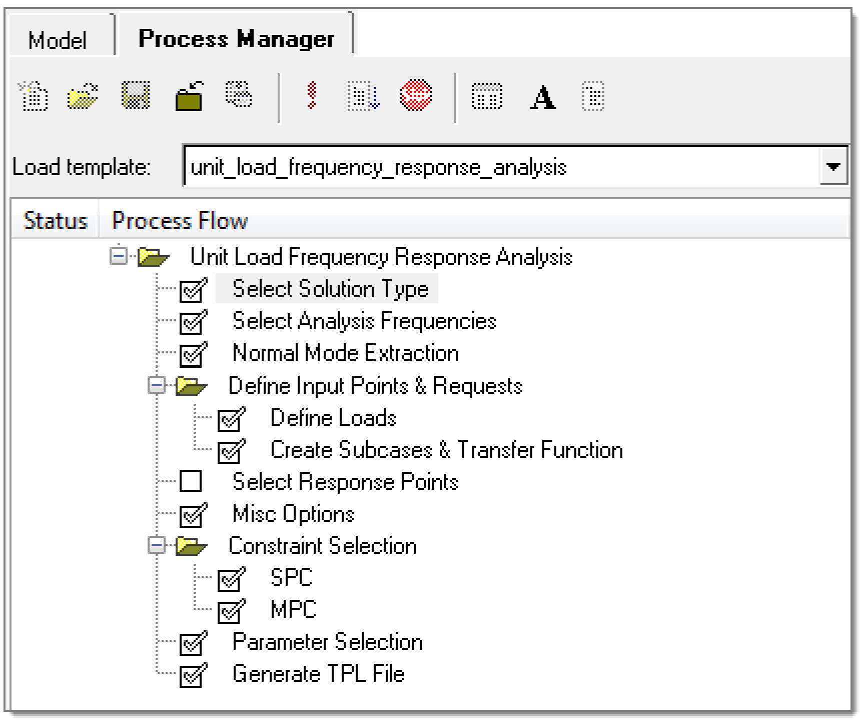


Figure 1629:

1. From the menu bar, click **Tools** > **Freq Resp Process** > **Unit Input Frequency Response**.
2. In the Select Solution Type task, select a solution method.
   a) Select either the **Direct Frequency Response solution** method, or the **Modal Frequency Response solution** method. For large problems involving more than a few frequencies, the modal solution is typically the most efficient solution.
   b) Click **Apply** to proceed.

3. In the Select Analysis Frequencies task, enter the frequencies for which response solution is needed.
   a) Define the frequency set.
      - To define the min, max, and a linear step, select **Frequency range**. For Incr type, select **Linear** and then fill in the required fields.
      - To define min, max, and a number of increments with logarithmic spacing, select **Frequency range**. For Incr type, select **Logarithmic** and then fill in the required fields.
      - To define an arbitrary list of frequencies, select **Frequency list** and enter a list of arbitrary frequencies.

   b) Click **Update**.

      A frequency set entry is created in the list box to the left. You can add additional frequency sets, or delete one from the list using the Add row and Delete row icons to the right of the list.
   c) Once the frequency set(s) have been defined, click **Apply** to proceed.

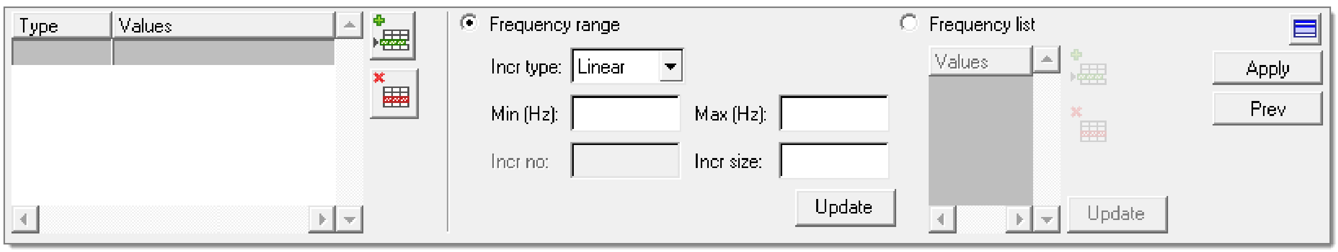


Figure 1630:

4. In the Normal Mode Extraction task, the default frequencies filled in are based on the max. frequency you filled in for the previous step.
   a) Modify the values based on the specific requirements of the case under study.

      These values are merely the suggested values based on general use cases.
   b) Click **Apply** to proceed.

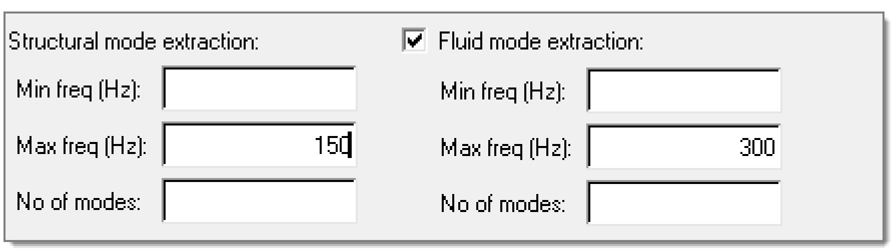


Figure 1631:

5. In the Define Input Points & Requests, Define Loads task, define what type of load is applied.

   The first four types are only applicable for structural nodes, and the last one for fluid nodes.

a) For Load type, select **Force**, **Enforced motion** (displacement, velocity, or acceleration), or **Acoustic source**.

b) Use the Nodes selector to select nodes, a node set, or tags in the modeling window, then click **Add**.
The table on the left hand side is populated with additional degree of freedom (DOF) checkboxes.

c) You can make a single or multiple row selection within the table, and then right-click to access the DOF row selection options. Alternatively, make a single or multiple row selection within the table, and then right-click to access the column selection options.
All selected DOFs will be checked to indicate locations and directions where unit input are to be applied.

d) Click **Apply** to proceed.
One loadcase will be created for each DOF indicated.

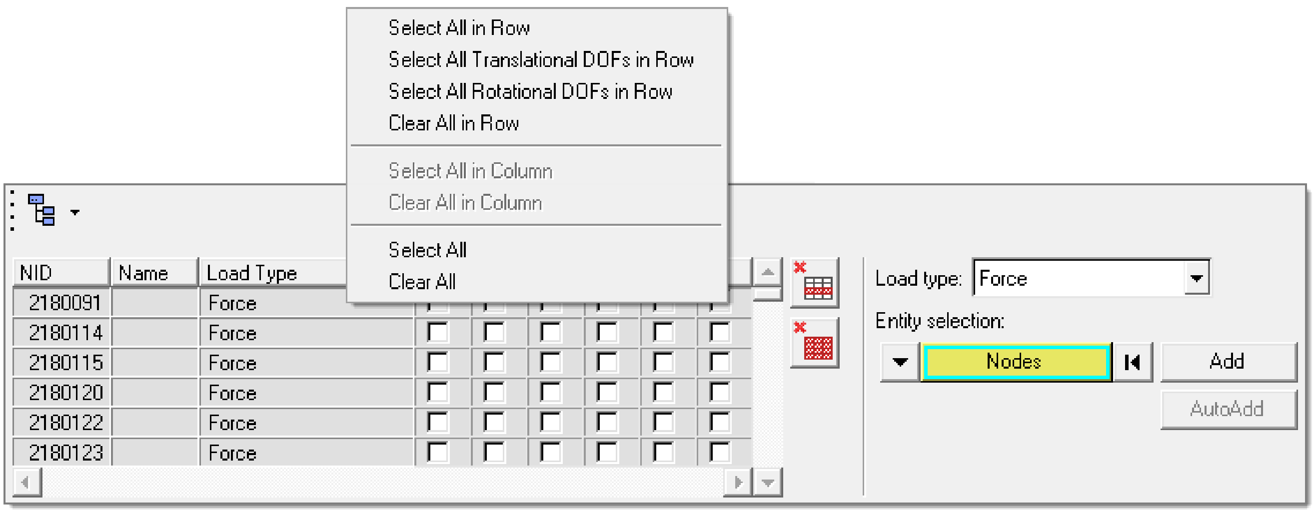


Figure 1632:

6. In the Define Input Points & Requests, Create Subcases & Transfer Function task, select to output transfer function between input points.

Of particular interest are driving point (response taken at the same point as input) transfer functions, which are commonly used as a measure for local dynamic stiffness, or full matrix (all possible pairs of input point combinations) output, which is sometimes used as input for FRF based substructuring analysis.

It is also possible to create new subcase groups of input to be used in individual subcases. New subcase groups can be added by clicking the Add Group icon . You can make a single or multiple row selections within the table, and then select the newly created Subcase group and click **Update**. All input dofs belonging to one subcase group will be used as simultaneous excitations in one subcase.

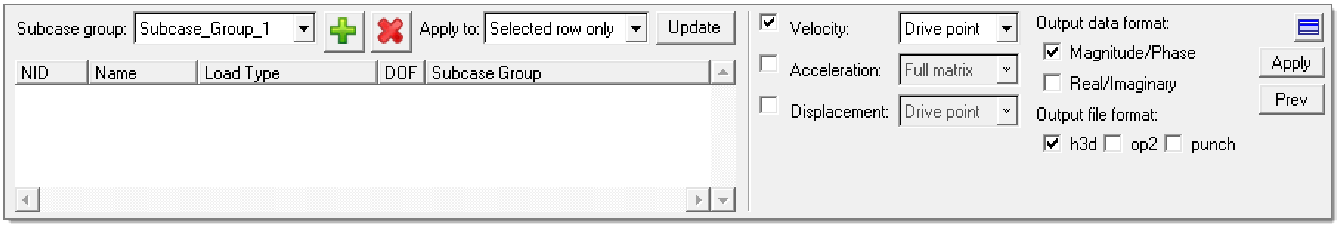


Figure 1633:

7. In the Select Response Points task, select response points for output.

a) Select a Response type, such as Displacement.

b) Use the Entity selector to select nodes, a node set, or tags to be included in a particular response set.

You can add or delete multiple response sets from the list using the **Add row** and **Delete row** icons to the right of the list.

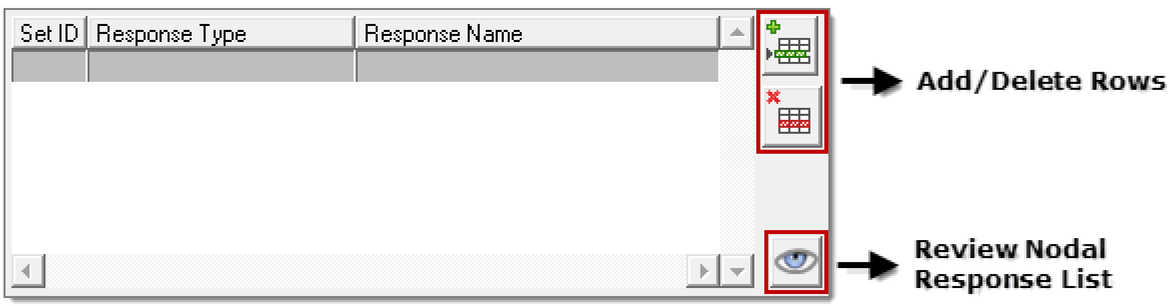


Figure 1634:

c) Under Output data format, select the complex frequency response data format (real/ imaginary or magnitude/phase).

d) Under Output file format, select the output file format (h3d, punch, or op2).

e) Once all the required responses have been defined, click **Apply** to proceed.

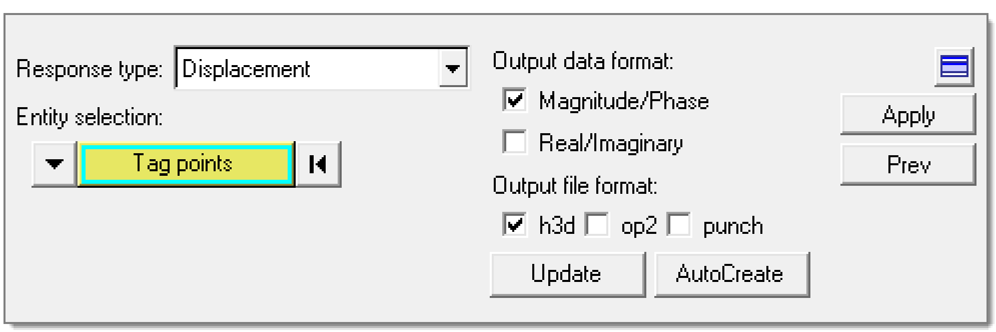


Figure 1635:

8. In the Misc Options task, select from the damping options that are available, including the global modal viscous damping on the structure side, and global material and viscous damping on the fluid side.

Figure 1636:

9. In the Constraint selection, SPC task, select the boundary condition of the frequency response analysis.

a) You can select existing SPCs by checking the corresponding box under the Active column, or click **Create SPC** to go to the Constraints panel and define a new SPC.

b) Once the boundary condition has been fully defined, click **Apply** to proceed.

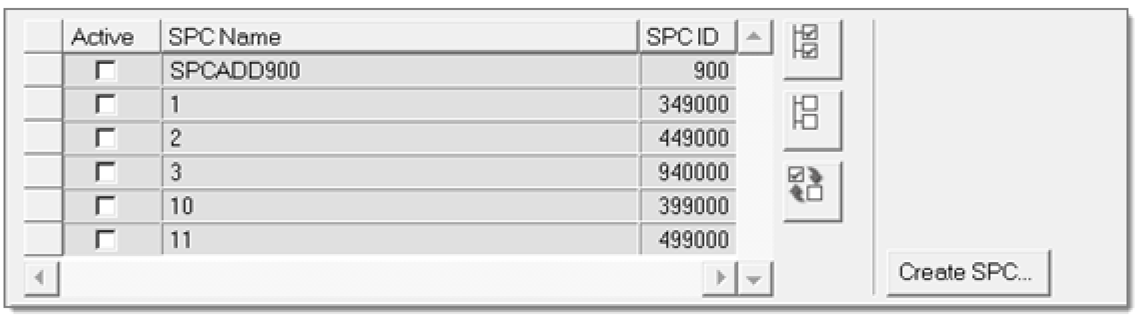


Figure 1637:

10. In the Constraint selection, MPC task, select MPC equations to turn on for the frequency response analysis.

a) Select existing MPCs by checking the corresponding box under the Active column.

b) Once the MPC equations have been selected, click **Apply** to proceed.

11. In the Parameter Selection task, select typical solution parameters, such as title, singular point constraints, and so on.

a) Check all boxes under the Active column to activate the desired solution option.

b) Once the parameters have been selected, click **Apply** and an Process Manager message box pops up informing you that the process has come to an end.

c) Click **Yes** to close the template, or **No** to review or edit the process steps.

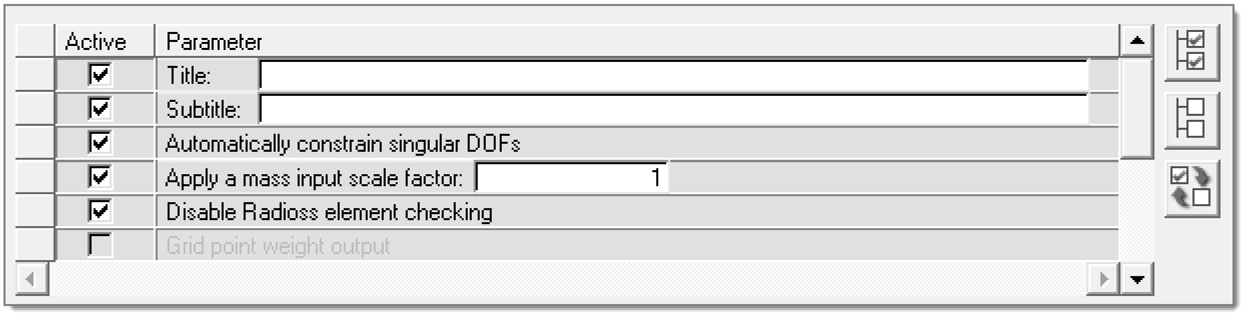


Figure 1638:

12. In the Generate TPL file task, generate and save the parameters defined in a standard template file at your selected location.

- Choose **Full deck TPL** to access the file that has all of the parameters for the complete unit load FRF process manager. In the Analysis Manager, the generated template file can be selected as loadcase for the Solution type Full deck TPL and solver decks can be exported for the unit FRF.

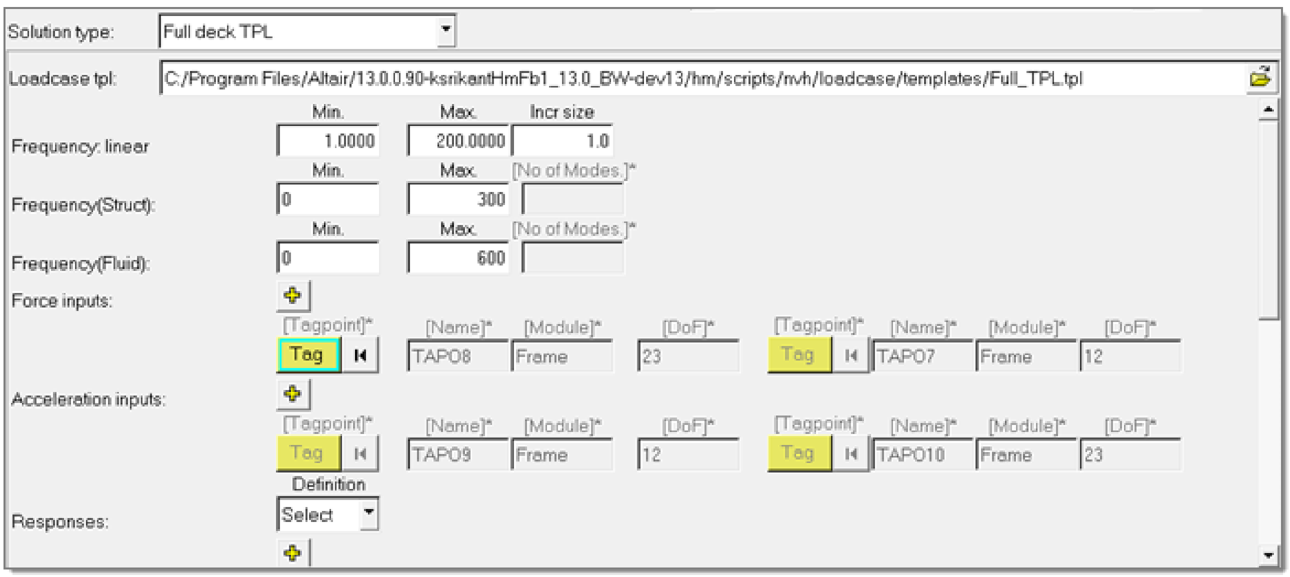


Figure 1639:

- Choose **Loads Only TPL** to access the file that contains only the parameters for the loadcase portion of the unit load FRF process manager. In the Analysis Manager, the generated template file can be selected as the loadcase for the Solution type Frequency Response and solver decks can be exported for general FRF.

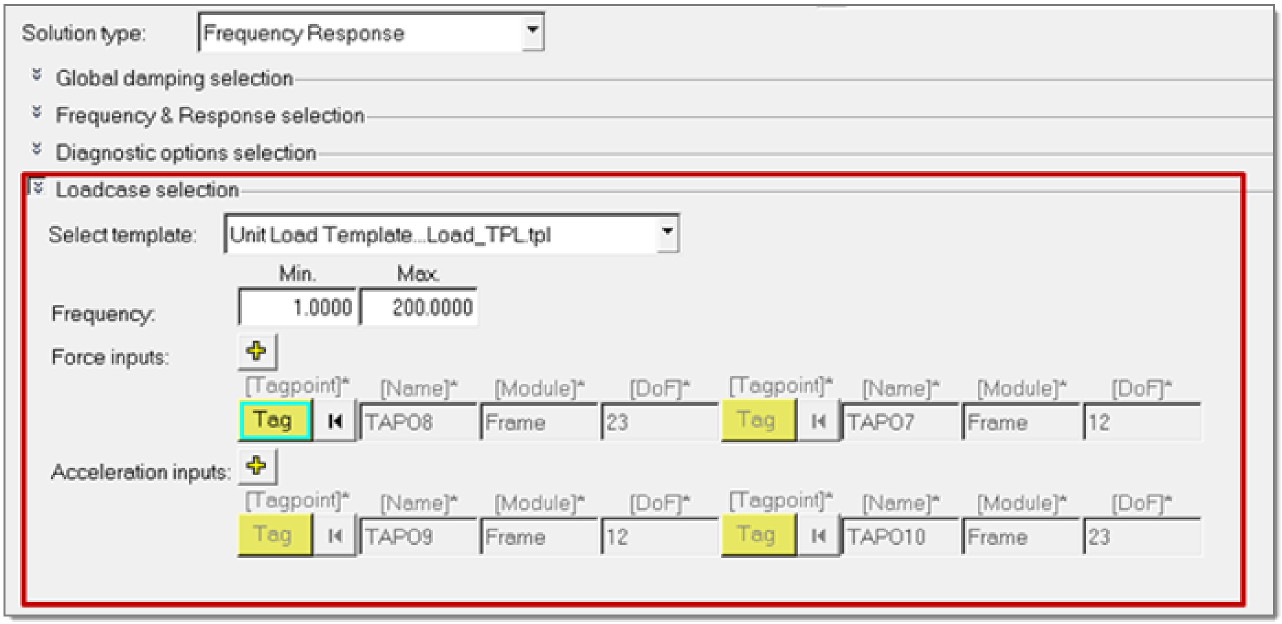


Figure 1640:

Setup Random PDS Frequency Response Process

Perform random frequency response analysis.

This analysis includes two steps, the first step is to define unit input frequency response subcase, and the second is to perform random response analysis by combining the unit input subcases with the auto and cross PSD matrix.

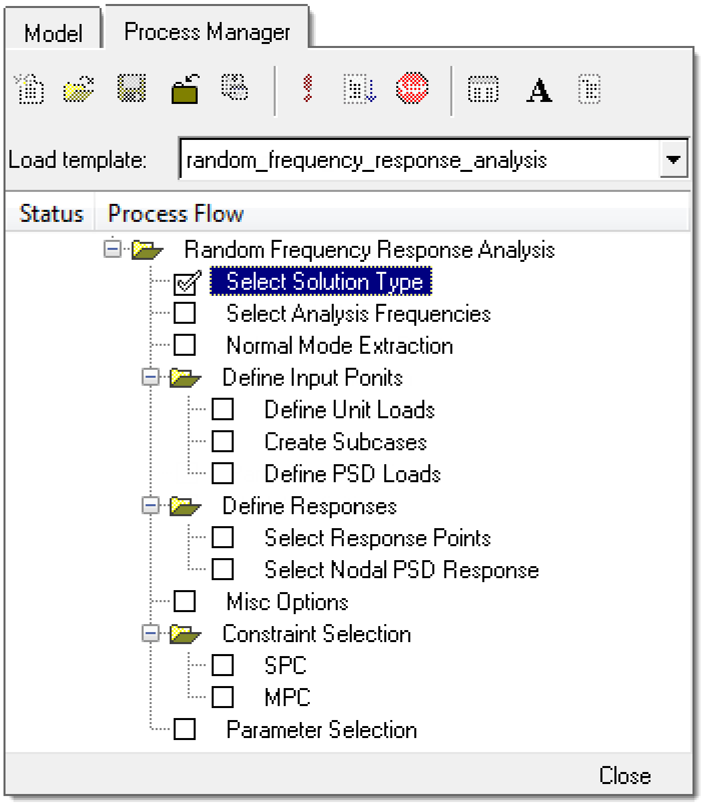


Figure 1641:

1. From the menu bar, click **Tools** > **Freq Resp Process** > **Random PSD Frequency Response**.
2. In the Select Solution Type task, select a solution method.
   a) Select either the **Direct Frequency Response solution** method, or the **Modal Frequency Response solution** method. For large problems involving more than a few frequencies, the modal solution is typically the most efficient solution.
   b) Click **Apply** to proceed.
3. In the Select Analysis Frequencies task, enter the frequencies for which response solution is needed.
   a) Define the frequency set.
      - To define the min, max, and a linear step, select **Frequency range**. For Incr type, select **Linear** and then fill in the required fields.
      - To define min, max, and a number of increments with logarithmic spacing, select **Frequency range**. For Incr type, select **Logarithmic** and then fill in the required fields.
      - To define an arbitrary list of frequencies, select **Frequency list** and enter a list of arbitrary frequencies.
   b) Click **Update**.

      A frequency set entry is created in the list box to the left. You can add additional frequency sets, or delete one from the list using the Add row and Delete row icons to the right of the list.
   c) Once the frequency set(s) have been defined, click **Apply** to proceed.

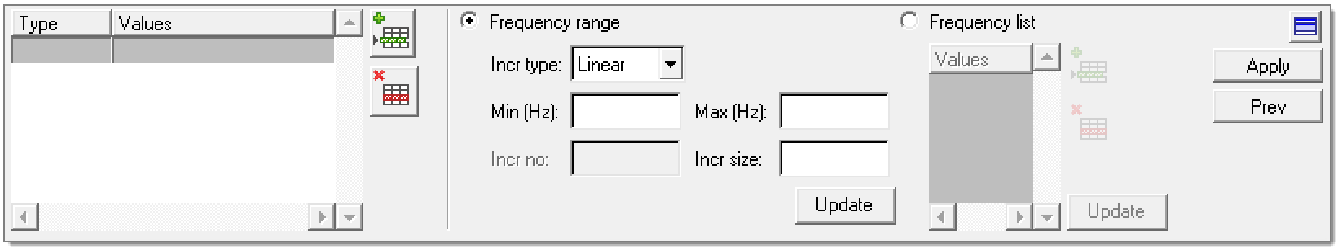


Figure 1642:

4. In the Normal Mode Extraction task, the default frequencies filled in are based on the max. frequency you filled in for the previous step.

a) Modify the values based on the specific requirements of the case under study.

These values are merely the suggested values based on general use cases.

b) Click **Apply** to proceed.

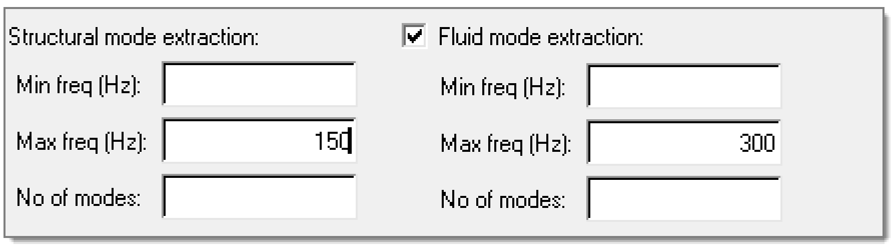


Figure 1643:

5. In the Define Input Points, Define Unit Loads task, define what type of load is applied.

The first four types are only applicable for structural nodes, and the last one for fluid nodes.

a) For Load type, select **Force**, **Enforced motion** (displacement, velocity, or acceleration), or **Acoustic source**.

b) Use the Nodes selector to select nodes, a node set, or tags in the graphics window, then click **Add**.
The table on the left hand side is populated with additional degree of freedom (DOF) checkboxes.

c) You can make a single or multiple row selection within the table, and then right-click to access the DOF row selection options. Alternatively, make a single or multiple row selection within the table, and then right-click to access the column selection options.
All selected DOFs will be checked to indicate locations and directions where unit input are to be applied.

d) Click **Apply** to proceed.
One loadcase will be created for each DOF indicated.

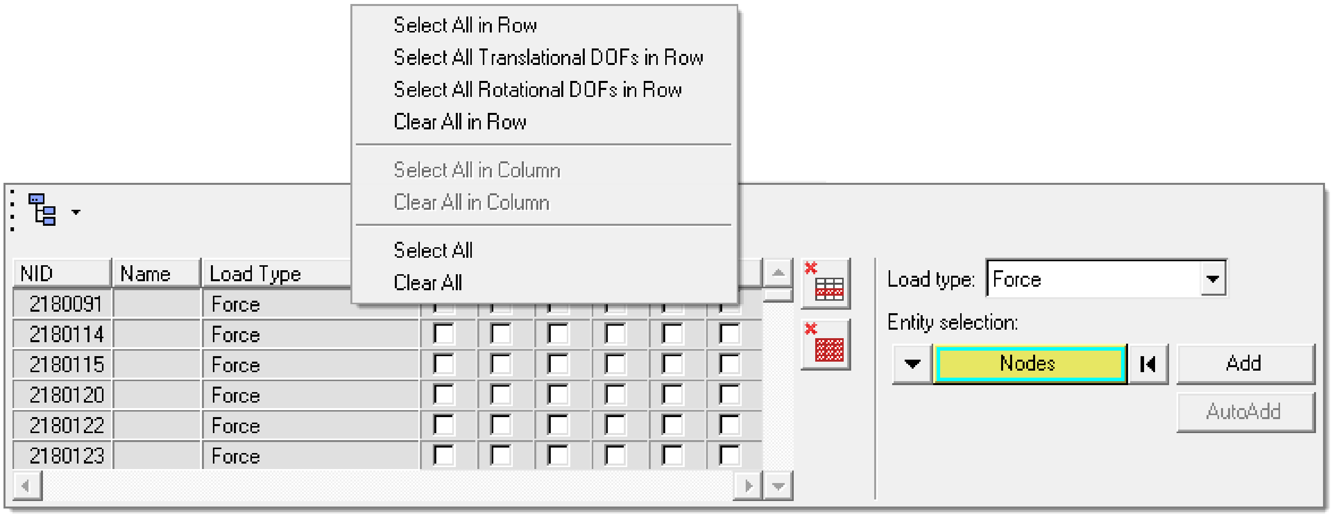


Figure 1644:

6. In the Define Input Points, Create Subcases task, select to output transfer function between input points.

Of particular interest are driving point (response taken at the same point as input) transfer functions, which are commonly used as a measure for local dynamic stiffness, or full matrix (all possible pairs of input point combinations) output, which is sometimes used as input for FRF based substructuring analysis.

It is also possible to create new subcase groups of input to be used in individual subcases.

New subcase groups can be added by clicking the Add Group icon . You can make a single or multiple row selections within the table, and then select the newly created Subcase group and click **Update**. All input dofs belonging to one subcase group will be used as simultaneous excitations in one subcase.

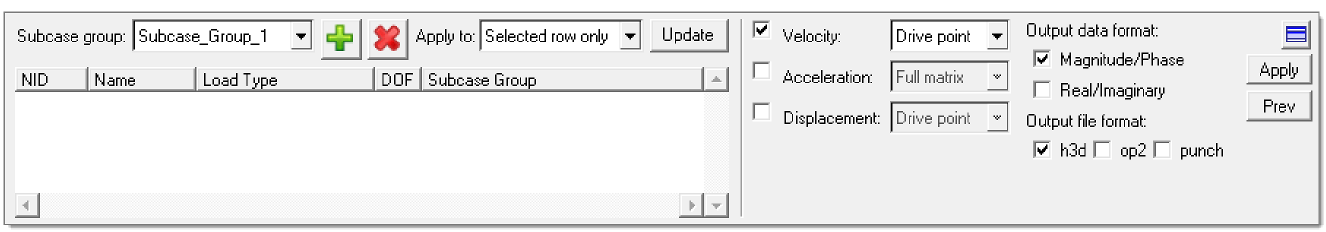


Figure 1645:

7. In the Define Input Points, Define PDS Loads task, input the NxN 2 dimensional PSD matrix (here N is the number of subcases whose response are used in the random PSD calculations.

a) The diagonal cells are used to define auto PSD terms, and the off diagonal cells are used to define cross PSD terms. For cross PSD, only the bottom triangle of the matrix needs to be defined. To input a PSD term, click one of the cells, and then specify the real and imaginary scaling factor and the frequency table.

b) Once the PSD matrix input is completed, click **Apply** to proceed.

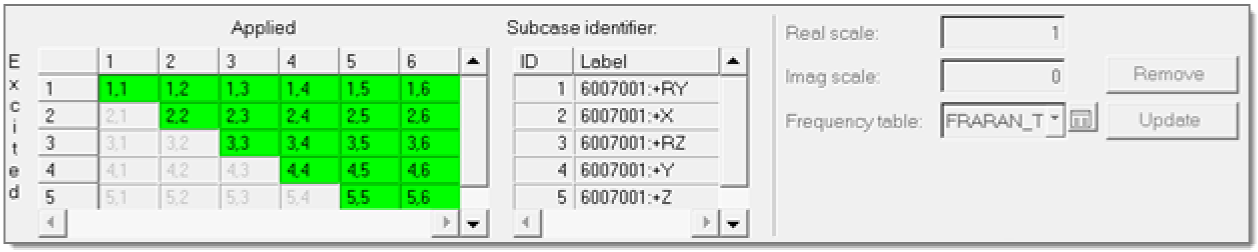


Figure 1646:

8. In the Define Responses, Select Response Points task, select response points for output.

a) Select a Response type, such as Displacement.

b) Use the Entity selector to select nodes, a node set, or tags to be included in a particular response set.

You can add or delete multiple response sets from the list using the **Add row** and **Delete row** icons to the right of the list.

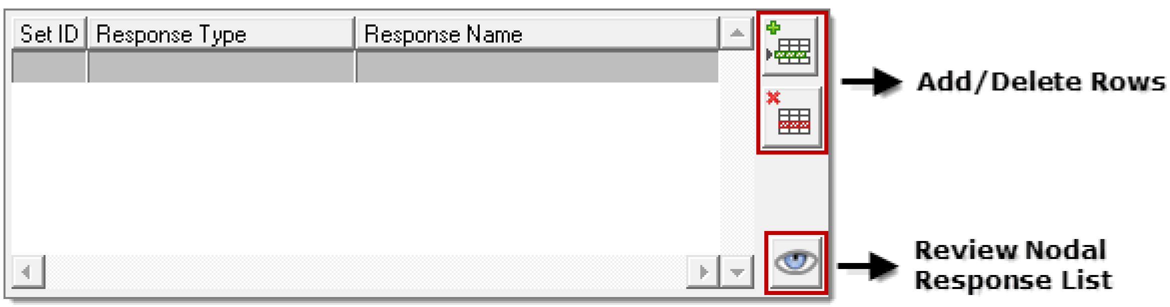


Figure 1647:

c) Under Output data format, select the complex frequency response data format (real/imaginary or magnitude/phase).

d) Under Output file format, select the output file format (h3d, punch, or op2).

e) Once all the required responses have been defined, click **Apply** to proceed.

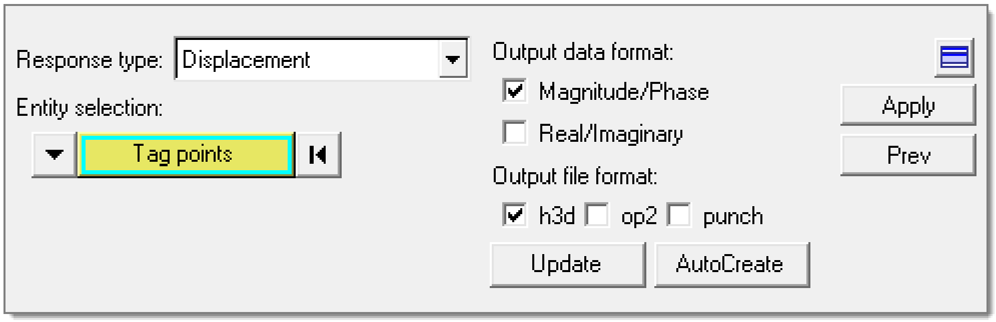


Figure 1648:

9. In the Define Responses, Select Nodal PSD Response task, define the response points and DOFs to be output.

Response points and DOFs can be output into the following formatted files: XYPUNCH, XYPLOT, and XYPEAK.

a) Select a response type and then select nodes, set of nodes, or tags, and then click **Add**. The list box is populated to allow you to further the selection to a set of DOFs.

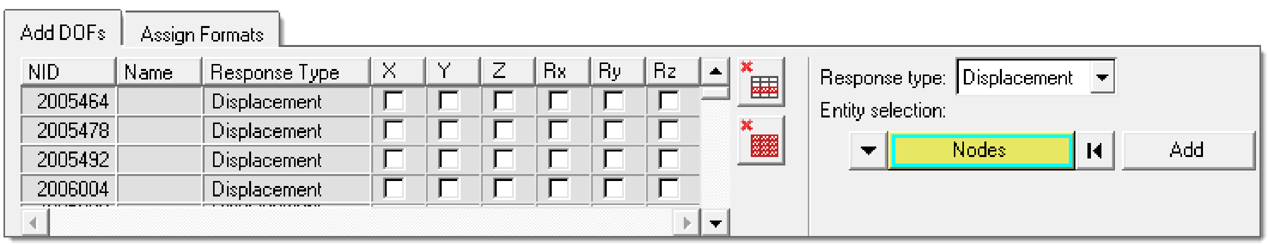


Figure 1649:

b) For each DOF, select the output formats.

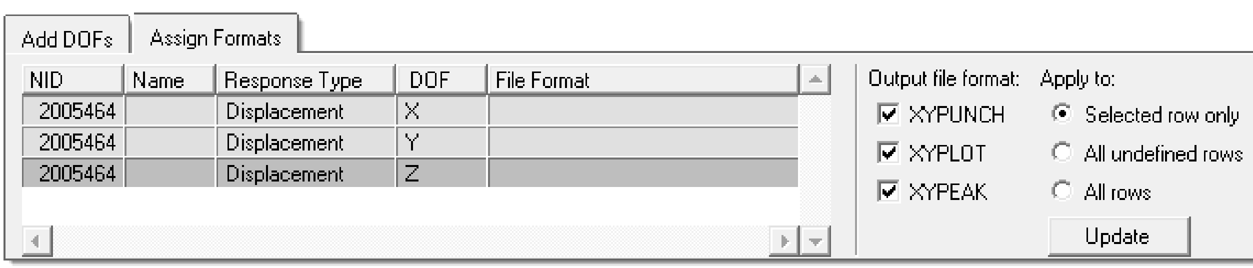


Figure 1650:

c) Once the PSD response selection is completed, click **Apply** to proceed.

10. In the Misc Options task, select from the damping options that are available, including the global modal viscous damping on the structure side, and global material and viscous damping on the fluid side.

Figure 1651:

11. In the Constraint selection, SPC task, select the boundary condition of the frequency response analysis.

a) You can select existing SPCs by checking the corresponding box under the Active column, or click **Create SPC** to go to the Constraints panel and define a new SPC.

b) Once the boundary condition has been fully defined, click **Apply** to proceed.

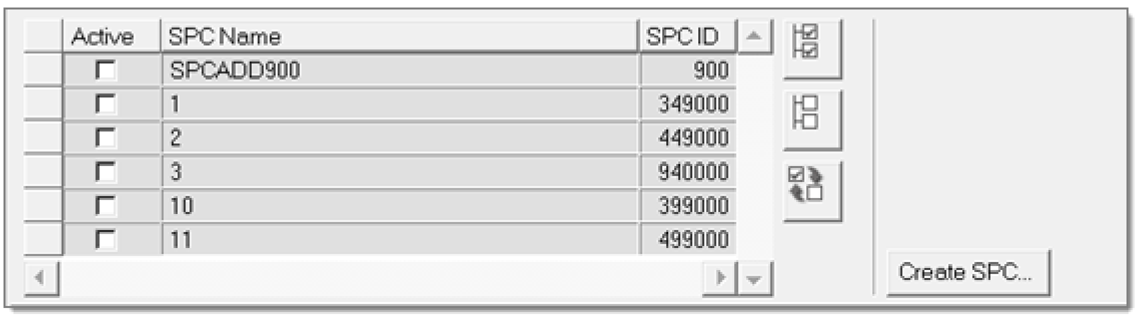


Figure 1652:

12. In the Constraint selection, MPC task, select MPC equations to turn on for the frequency response analysis.

a) Select existing MPCs by checking the corresponding box under the Active column.

b) Once the MPC equations have been selected, click **Apply** to proceed.

13. In the Parameter Selection task, select typical solution parameters, such as title, singular point constraints, and so on.

a) Check all boxes under the Active column to activate the desired solution option.

b) Once the parameters have been selected, click **Apply** and an Process Manager message box pops up informing you that the process has come to an end.

c) Click **Yes** to close the template, or **No** to review or edit the process steps.

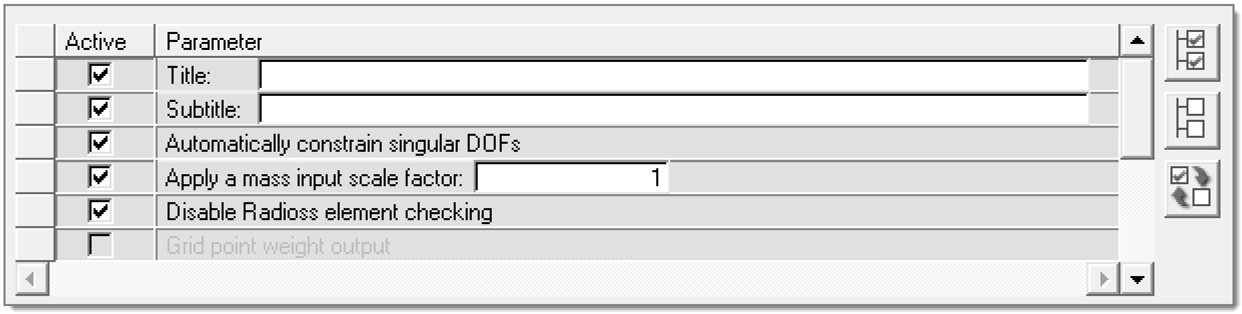


Figure 1653:

Setup General Frequency Response Process

Apply loads at multiple DOFs simultaneously with arbitrary magnitude and delay (relative phase).

In the Unit Input Frequency Response process and the Random PSD Frequency Response processes, loads are limited to single DOF with unit inputs subcases.

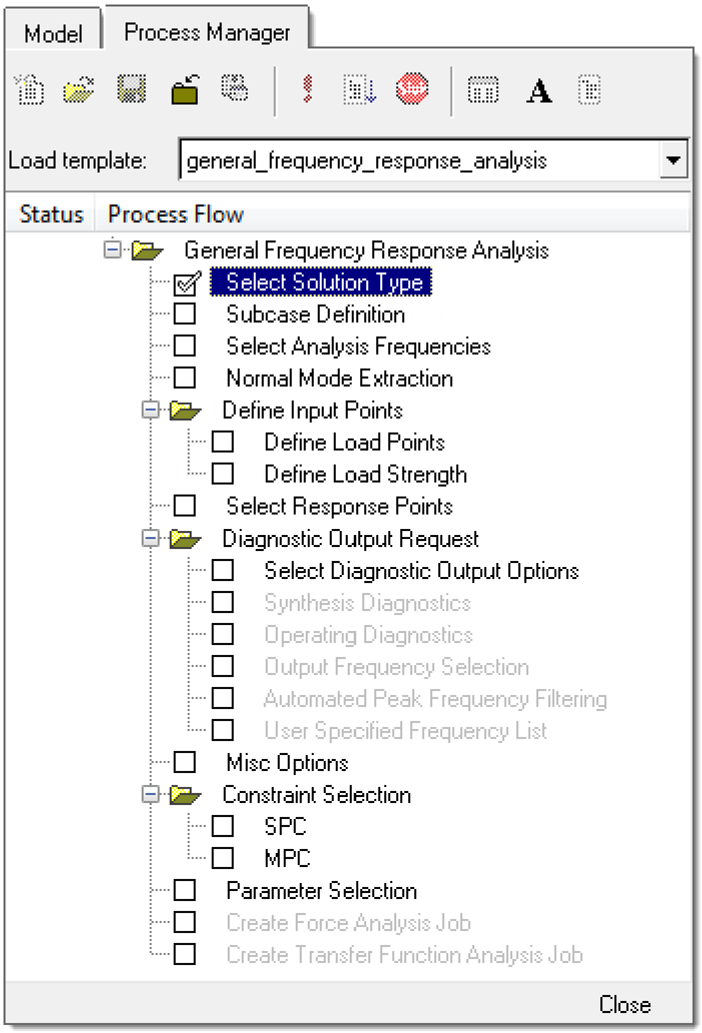


Figure 1654:

1. From the menu bar, click **Tools** > **Freq Resp Process** > **General Frequency Response**.
2. In the Select Solution Type task, select a solution method.
   a) Select either the **Direct Frequency Response solution** method, or the **Modal Frequency Response solution** method. For large problems involving more than a few frequencies, the modal solution is typically the most efficient solution.
   b) Click **Apply** to proceed.
3. In the Subcase Definition task, create a new subcase or edit an existing subcase.
   a) Create or edit a subcase.
      - To create a new subcase, edit the optional subcase label and click **Add**.
      - To edit an existing subcase, highlight an existing subcase in the list box, edit the optional subcase label and click **Update**.
   b) Once input to the task is complete, click **Apply** to proceed.

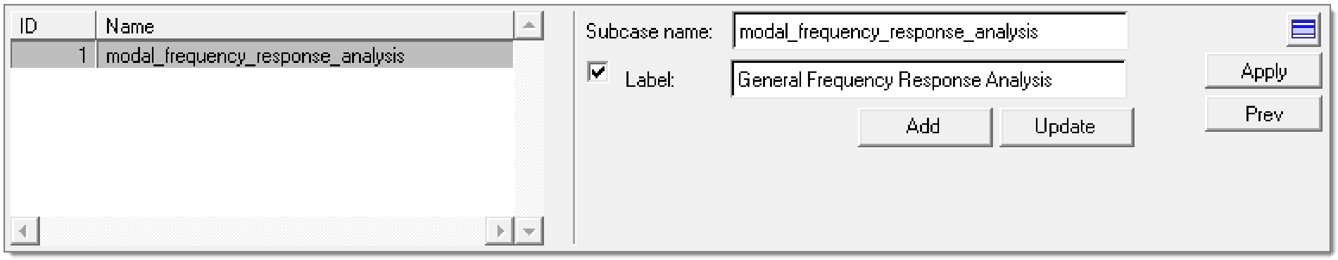


Figure 1655:

4. In the Select Analysis Frequencies task, enter the frequencies for which response solution is needed.

a) Define the frequency set.

- To define the min, max, and a linear step, select **Frequency range**. For Incr type, select **Linear** and then fill in the required fields.
- To define min, max, and a number of increments with logarithmic spacing, select **Frequency range**. For Incr type, select **Logarithmic** and then fill in the required fields.
- To define an arbitrary list of frequencies, select **Frequency list** and enter a list of arbitrary frequencies.

b) Click **Update**.

A frequency set entry is created in the list box to the left. You can add additional frequency sets, or delete one from the list using the Add row and Delete row icons to the right of the list.

c) Once the frequency set(s) have been defined, click **Apply** to proceed.

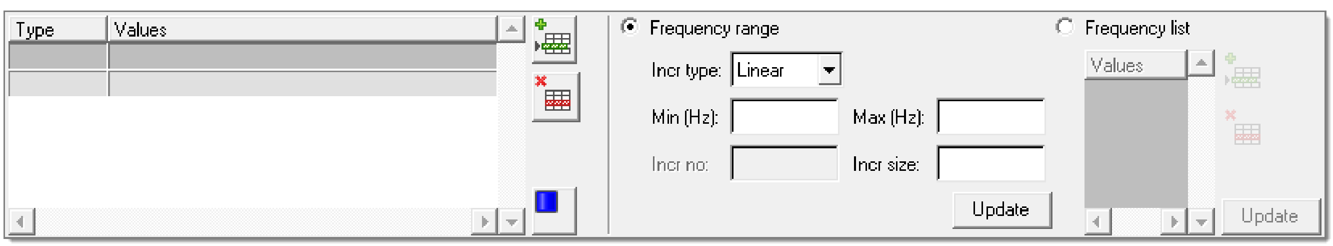


Figure 1656:

5. In the Normal Mode Extraction task, the default frequencies filled in are based on the max. frequency you filled in for the previous step.

a) Modify the values based on the specific requirements of the case under study.

These values are merely the suggested values based on general use cases.

b) Click **Apply** to proceed.

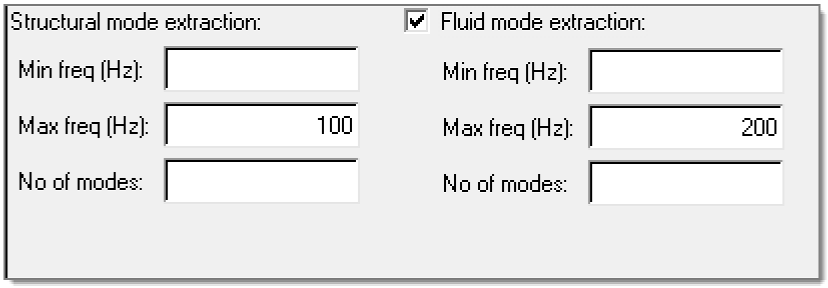


Figure 1657:

6. In the Define Input Points, Define Load Points task, define what type of load is applied.

The first four types are only applicable for structural nodes, and the last one for fluid nodes.

a) For Load type, select **Force**, **Enforced motion** (displacement, velocity, or acceleration), or **Acoustic source**.

b) Use the Nodes selector to select nodes, a node set, or tags in the modeling window, then click **Add**.
The table on the left hand side is populated with additional degree of freedom (DOF) checkboxes.

c) You can make a single or multiple row selection within the table, and then right-click to access the DOF row selection options. Alternatively, make a single or multiple row selection within the table, and then right-click to access the column selection options.
All selected DOFs will be checked to indicate locations and directions where unit input are to be applied.

d) Click **Apply** to proceed.
One loadcase will be created for each DOF indicated.

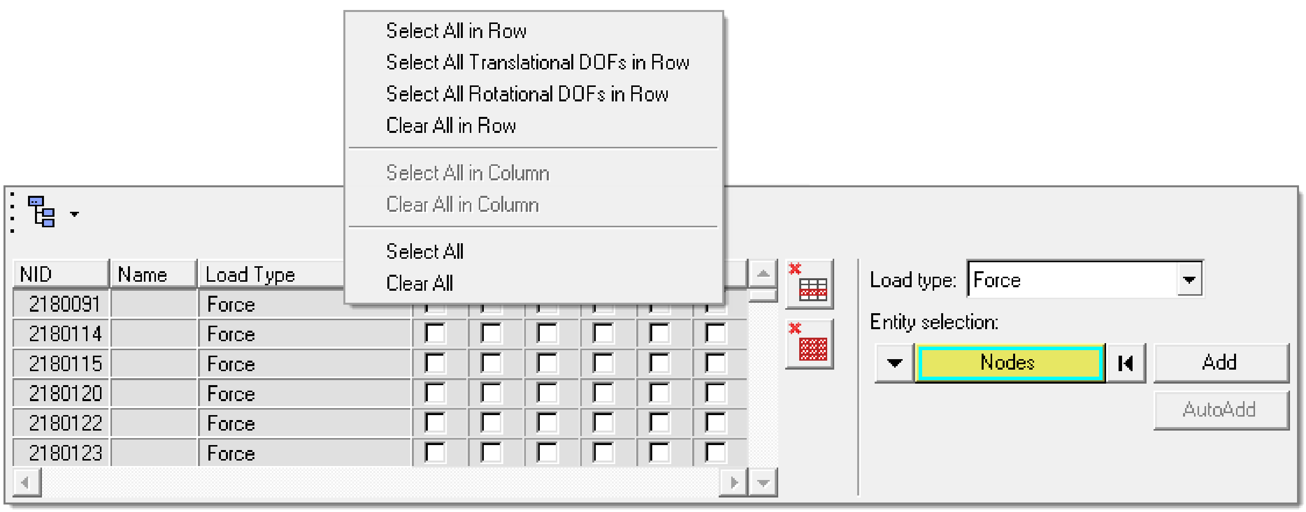


Figure 1658:

7. In the Define Input Points, Define Load Strength task, define frequency dependent load strength and delay for individual subcases or DOFs in one subcase.

- For subcase based definition, once the subcase is selected, select a DOF row from the list box, and then add load strength frequency tables in either real/imaginary or magnitude/phase form.

  Delay or phase relative to a reference input DOF can optionally be specified as well. Loading strength can be defined manually, with external files in csv/text format and also universal (`.unv`) files from external data acquisition tools like LMS, IDEAS, and B&K. Import the `.csv/text` or universal file, select the relevant loading strength and click **Save** to define it. You can select one of the radio buttons on the right to control how the definitions provided are filled into various rows. It is also possible to go back to the Subcase Definition panel by clicking the Create Subcase icon  to add new subcases or edit existing subcases. If subcases are added or edited, then you will be redirected to the Select Analysis Frequencies task to repeat all the steps prior to defining load strengths. If subcases are not edited then you will come back to the Define Load Strength task.

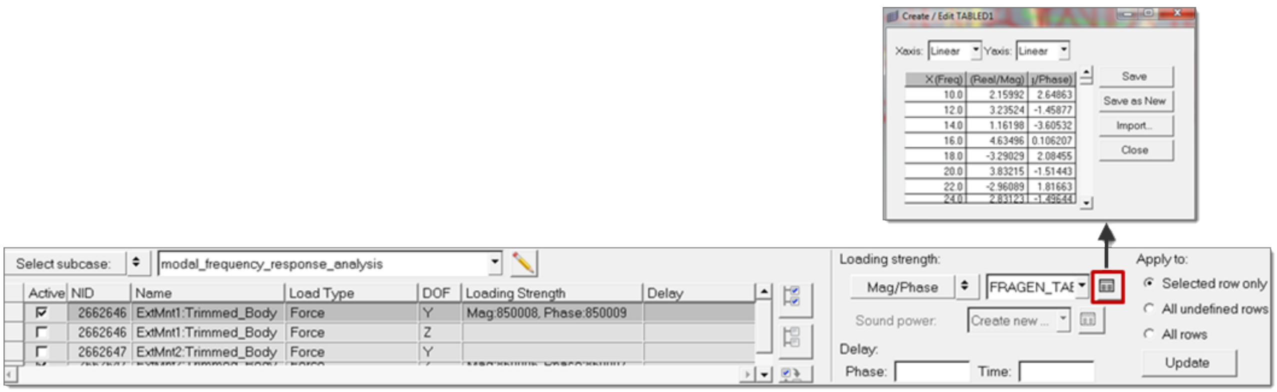


Figure 1659:

- For DOF based definition, once the DOF is selected, select a subcase row from the list and follow the same process described in the above section. With this option it is not possible to add or edit subcases.

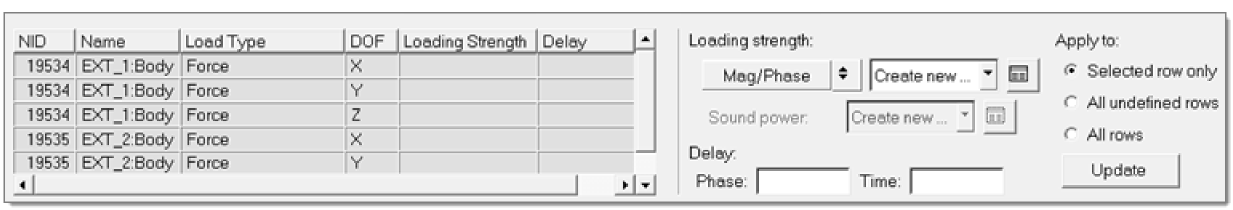


Figure 1660:

8. In the Select Response Points task, select response points for output.

Response selection can be global or subcase specific.

a) Select a Response type, such as Displacement.

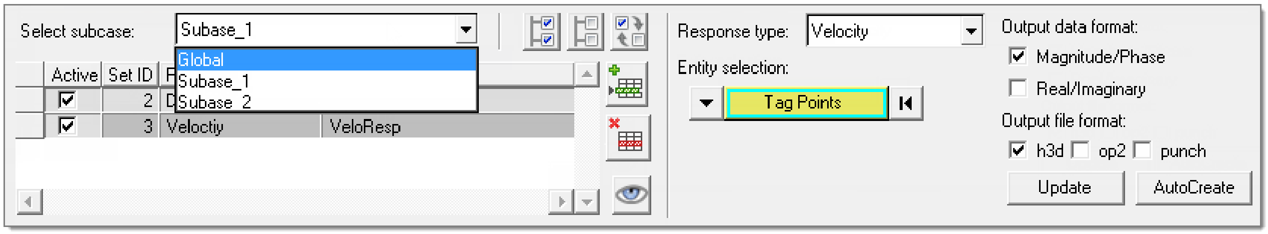


Figure 1661:

b) Use the Entity selector to select nodes, a node set, or tags to be included in a particular response set.

You can add or delete multiple response sets from the list using the **Add row** and **Delete row** icons to the right of the list.

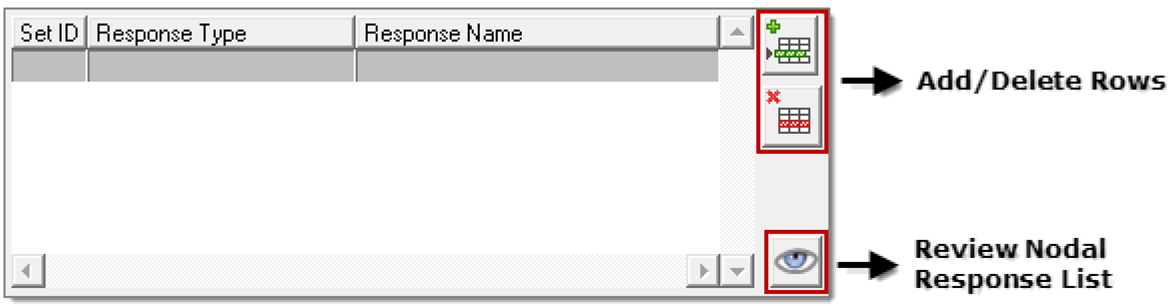


Figure 1662:

c) Under Output data format, select the complex frequency response data format (real/imaginary or magnitude/phase).

d) Under Output file format, select the output file format (h3d, punch, or op2).

e) Once all the required responses have been defined, click **Apply** to proceed.

Responses selected for the global subcase are by default available for other individual subcases. Additional responses can be added and used for an individual subcase. It is also possible to add duplicate response types to any individual subcase, in addition to those in the global subcase. In this case a separate response set will be created for that subcase which will be a union of entities in global and individual subcases.

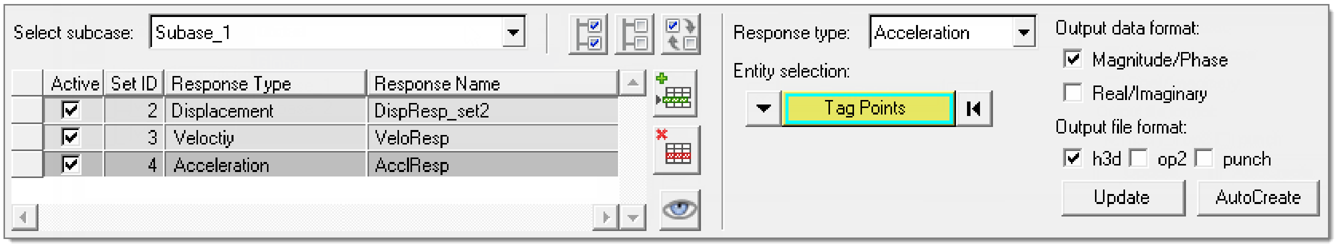


Figure 1663:

9. In the Diagnostic Output Request, Select Diagnostic Output Options task, control if diagnostic outputs are to be generated.

a) Select **No** to generate diagnostic output requests and the process will go directly to the Miscellaneous Options task. Select **Yes** to request diagnostic output only at selected response peaks.

b) Click **Apply** to proceed.

Figure 1664:

10. In the Select Diagnostic Output Request, Synthesis Diagnostics task, select the Synthesis diagnostic type which presents a breakdown (participation factors) to the response, such as modal participations.

This type will be output only at the peak frequencies of the corresponding response DOF. The selection of the Synthesis diagnostic type can be Global or Subcase Specific.

a) First select the subcase, either **Global** or **Individual Subcase**.

b) Select response DOFs by clicking **Response points**.

In a case of individual subcases the list of responses will be a union of response entities for Global and Individual Subcase. There is a specific option of **Attachment points on control volume** in Auto TPA and Traditional TPA. In Auto TPA you can select a connection node set. The elements attached to connection nodes are automatically segregated into Non-Rigid Element set (CONEL) and Rigid Element set (CONREL) based on the element type. In Traditional TPA, you can create a node set at which attachment forces are to be calculated in the assembled state.

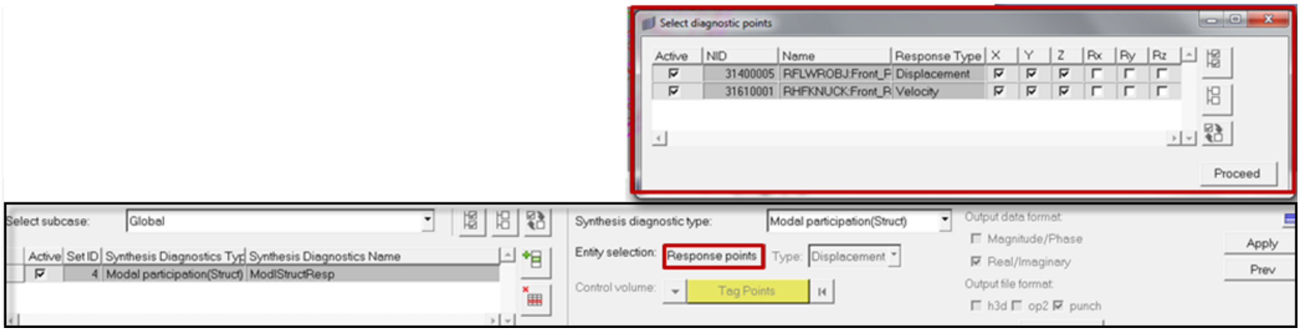


Figure 1665:

Selecting Traditional TPA as the diagnostic type enables Create Force Analysis Job and Create Transfer Function Analysis Job tasks in the process manager tree. Create Force Analysis Job allows you to export a solver deck or submit the job related to the first step of traditional TPA for calculating attachment forces in the fully assembled state through the Analysis Manager.

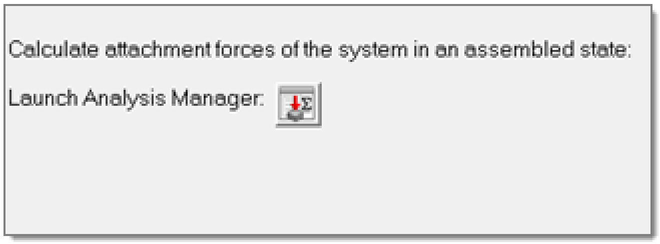


Figure 1666:

c) Click **Apply** to invoke the Create Transfer Function Analysis Job task.

This will delete all the created entities, and entities related to unit transfer functions are created. This allows you to export the solver deck or submit a job related to the second step of TPA for calculating transfer function in Control Volume through the Analysis Manager.

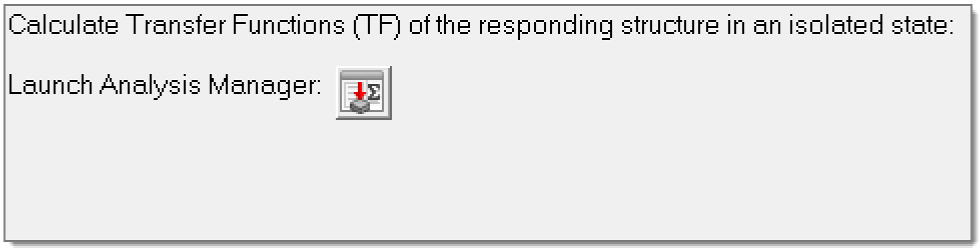


Figure 1667:

11. In the Select Diagnostic Output Request, Operating Diagnostics task, select the Operating diagnostic type which is not response specific, such as ODS animation or energy.

Output in this case will be generated at the super set of the peak frequencies of all selected response DOFs. In this case the selection of operating diagnostic type can be global or subcase specific.

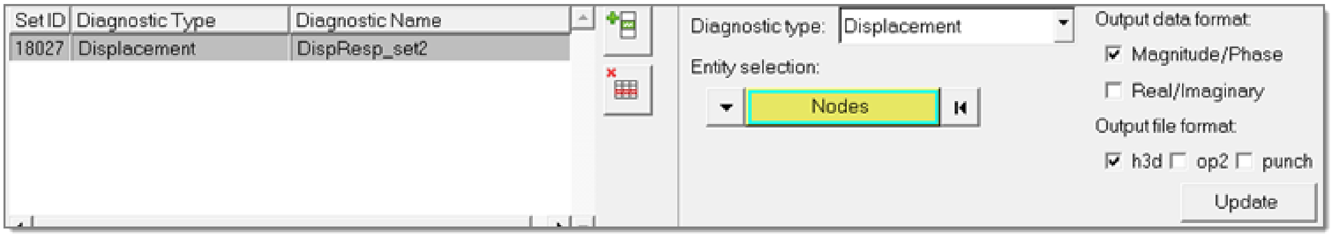


Figure 1668:

12. In the Select Diagnostic Output Request, Output Frequency Selection task, control to select frequency at which diagnostic output is to be requested.

- Select **Automated Peak Frequency Filtering** to customize options used to define response peak frequencies through the PEAKOUT card.
- Select **User Specified Frequency List** to enter a list of frequencies for each subcase, which will be referenced by a OFREQ card in a separate diagnostic output subcase.

13. In the Select Diagnostic Output Request, Automated Peak Frequency Filtering task, select the customized options specific for structural and acoustic responses to be used for selected response peak frequencies.

a) Select customized options.

Frequency selection for this option can also be global or subcase specific. The PEAKOUT card specified for the Global subcase is also used for those subcases for which a separate PEAKOUT card is not specified. For those subcases where specific peak frequency selection control is needed, a separate PEAKOUT card should be specified with appropriate parameters.

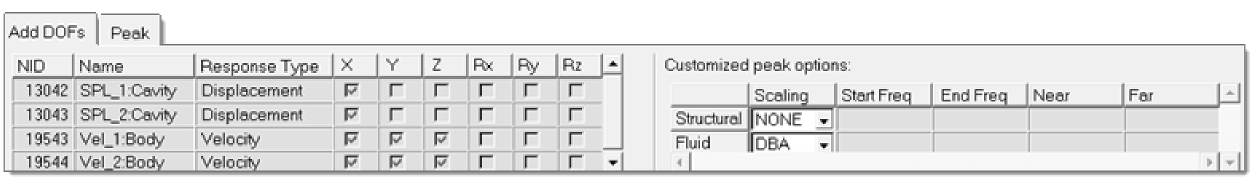


Figure 1669:

b) Specify a single threshold value or a frequency table.

Only peaks above the threshold are retained in the search process as candidates.

c) Specify the number of peaks (default =5) per response DOF to be kept in the peak frequency set.

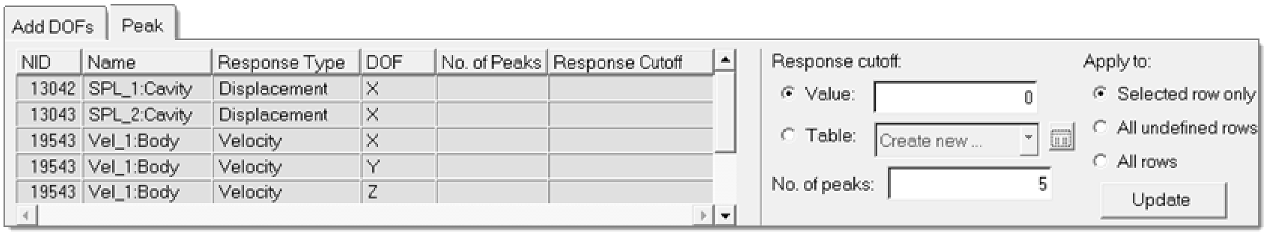


Figure 1670:

14. In the Select Diagnostic Output Request, User Specified Frequency List task, specify a list of frequencies at which diagnostic output is requested.

This definition is subcase specific. For subcases where a Frequency List is entered in the Select Analysis Frequencies task, the same frequency list values are automatically entered by default.

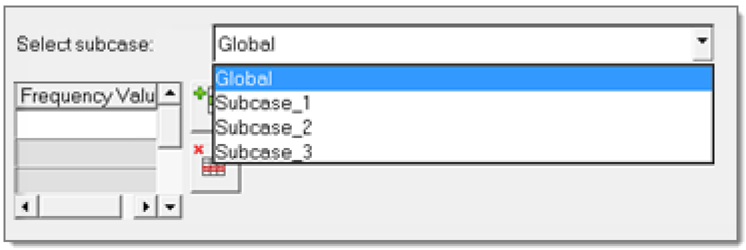


Figure 1671:

15. In the Misc Options task, select from the damping options that are available, including the global modal viscous damping on the structure side, and global material and viscous damping on the fluid side.

Figure 1672:

16. In the Constraint selection, SPC task, select the boundary condition of the frequency response analysis.

a) You can select existing SPCs by checking the corresponding box under the Active column, or click **Create SPC** to go to the Constraints panel and define a new SPC.

b) Once the boundary condition has been fully defined, click **Apply** to proceed.

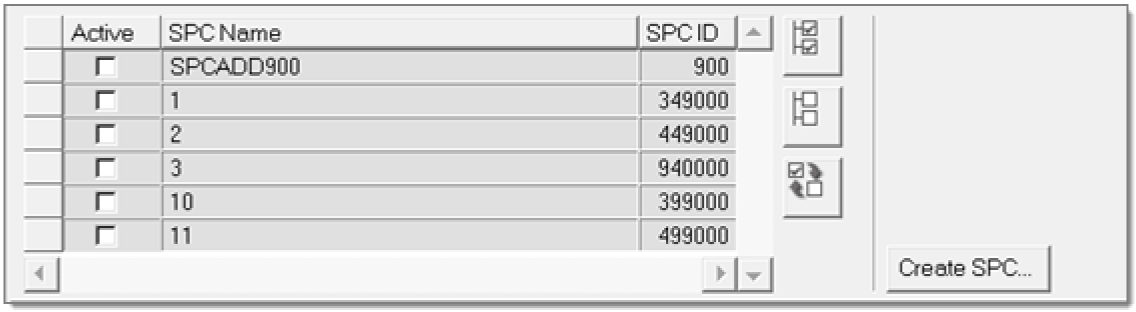


Figure 1673:

17. In the Constraint selection, MPC task, select MPC equations to turn on for the frequency response analysis.

a) Select existing MPCs by checking the corresponding box under the Active column.

b) Once the MPC equations have been selected, click **Apply** to proceed.

18. In the Parameter Selection task, select typical solution parameters, such as title, singular point constraints, and so on.

a) Check all boxes under the Active column to activate the desired solution option.

b) Once the parameters have been selected, click **Apply** and an Process Manager message box pops up informing you that the process has come to an end.

c) Click **Yes** to close the template, or **No** to review or edit the process steps.

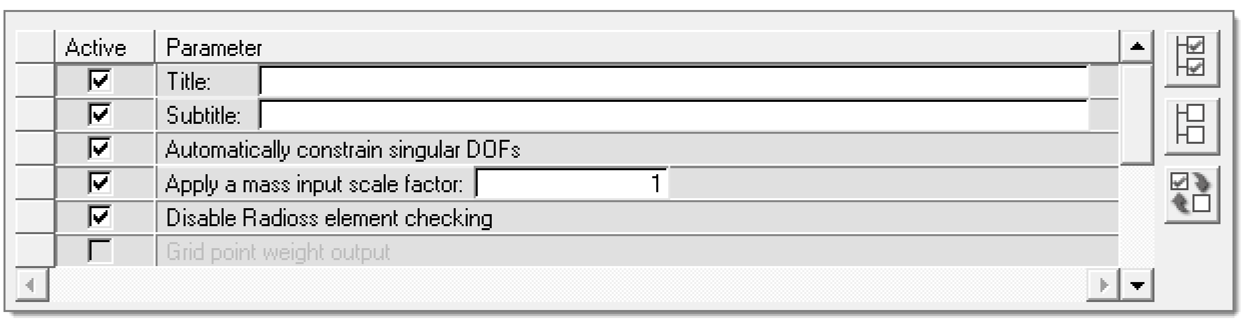


Figure 1674:

ID-Management

Manage all of the IDs for the entities that you create, and define ID ranges for all of the entities in each Include file in relation to the full model in order to avoid ID duplication.

Perform ID-management in the ID Manager.

> **Note:** When importing and/or creating new IDs, as well as editing entities in the Master Model or Include files, the ID Manager updates accordingly.

Once the ID ranges are defined for each Include file and child entity type, they will remain active and any new entities created will be assigned an ID within the ID range following the new ID rules. Entities that are imported into the Include file that have ID ranges already defined, will have the IDs that overflow after import automatically corrected by Move After Max option.

Aside from new entity creation and import, the ID Manager will not actively correct all of the IDs in the FE model. If there are entity IDs that result in overflow because of organization issues in the Include file such as renumbering, they will remain as overflow and must be corrected manually.

FE entities in CAE models are labeled and tagged using IDs and names. IDs are used by the solvers during analysis and then passed on by the solver to the results. This process enables you to interpret the results and relate them to the FE model. In most cases, the IDs of the FE entities in a FE model must be unique for efficient identification. In large models that involve several parts assembled into sub-assemblies and main assemblies, it is essential to have a tool that manages a model's IDs consistently in the part level, sub-assembly level, and in the global level for ease of identification using IDs.

In the Abaqus user profile, the following entity types are supported and impacted by the operations performed during ID-management: Assembly, Component, Element, Node, Property, Material, Beam Collector, Beam Section and Tags. Only these entity types will honor the Min and Max ID ranges defined in the ID Manager. Corrections made to an Include file only impact these entity types. During the creation of new entities that are not supported by the ID Manager, the Max ID + 1 is used to assign new IDs.

Manage IDs

Learn how to perform varios ID-management operations.

Create/Delete Included Files

1. From the menu bar, click **Tools** > **ID Manager**.
   The **ID Manager** opens.
2. Create or delete Include files.

Option	Description
Create	From the right-click context menu, select **Create** > **Include File**.
Delete	1. Right-click on an Include file and select **Delete** from the context menu. 2. In the **Delete Entity** dialog, select **Also delete children entities** to delete the selected Include file(s) along with its child entities.

Option	Description
	3. Click **OK**.

Create ID Ranges for New Entity Types

ID ranges can be defined for entity types that do not yet exist in the Master Model or in an Include file.

1. From the menu bar, click **Tools** > **ID Manager**.
   The **ID Manager** opens.
2. Right-click on the Master Model or an Include file and select **Create** > **ID-Range** from the context menu.
3. In the **Create** dialog, select an entity type and enter a Minimum ID and Maximum ID.
4. Click **OK**.

Edit ID Ranges

Edit the minimum and maximum ID range of the Master Model, Include files and entities.

1. From the menu bar, click **Tools** > **ID Manager**.
   The **ID Manager** opens.
2. Double-click the Min and Max columns and enter new values.

Clear ID Ranges

1. From the menu bar, click **Tools** > **ID Manager**.
   The **ID Manager** opens.
2. Clear ID ranges in the following ways:

Option	Description
Clear all ID ranges	Clear all defined ID ranges across all Include files in your model by right-clicking on the Master Model and selecting **Clear All ID Ranges** from the context menu.
Clear selected ID ranges	Clear ID ranges for a single Include file or entity by right-clicking on the Include file or entity and selecting **Clear ID-Ranges** from the context menu.

Note: Clearing an Include file's ID range also clears ID ranges defined for child entities. If you clear the ID range defined for a child entity, when it is different from the Include range, then the range defined for the Include file will be updated for the child entity. Clearing ID ranges will not remove exclusions.

Exclude Entities

1. From the menu bar, click **Tools** > **ID Manager**.
   The **ID Manager** opens.
2. Exclude entities in the following ways:
   - To exclude a specific entity, right-click on a Master Model, Include file, or entity and select **Exclude** > **Exclude Only** from the context menu.

     If you right-click on a Master Model or Include file you must select a type of entity to exclude in the **Exclude** dialog.

     Exclusion is automatically applied if you right-click on an entity. When an entity is excluded, an X icon ( ) displays in the Excluded column.
   - To exclude IDs of the same entity type across all Include files, right-click on an entity and select **Exclude** > **Exclude Similar** from the context menu.

Tip: Excluded entities can be hidden in the ID Manager by selecting the **Hide Excluded Entities** checkbox.

Clear Exclusions

1. From the menu bar, click **Tools** > **ID Manager**.
   The **ID Manager** opens.
2. Clear exclusions in the following ways:
   - To clear excluded IDs for a specific entity, right-click on one or more entities and select **Clear Exclusion** > **Clear Exclusion Only** from the context menu.
   - To clear excluded IDs of the same entity type across all Include files, right-click on an entity and select **Clear Exclusion** > **Clear Exclusion Similar** from the context menu.

   Note: Multiple entities across Include files can be selected.

Lock/Unlock IDs

Lock IDs to prevent them from being changed by future ID management and renumbering operations, or unlock IDs so they can be changed.

- Lock IDs.

a) From the menu bar, click **Tools** > **ID Manager**.
The **ID Manager** opens.
b) Lock IDs.
    - To lock all IDs, right-click on a Master Model, Include file, or entity and select **Lock** > **All IDs** from the context menu.
    - To lock discrete IDs, right-click on an entity and select **Lock** > **Discrete IDs** from the context menu, then select specific IDs to lock in the panel area.
    - To lock IDs of the same entity type across all Include files, right-click on an entity and select **Lock** > **All Similars** from the context menu.

- Unlock IDs.
    a) From the menu bar, click **Tools** > **ID Manager**.
    The **ID Manager** opens.
    b) Unlock IDs.
        - To unlock all IDs, right-click on a Master Model, Include file, or entity and select **Unlock** > **All IDs** from the context menu.
        - To unlock discrete IDs, right-click on an entity and select **Unlock** > **Discrete IDs** from the context menu, then select specific IDs to unlock in the panel area.
        - To unlock IDs of the same entity type across all Include files, right-click on an entity and select **Unlock** > **All Similars** from the context menu.

Reserve/Unreserve IDs

Reserve and unreserve available IDs for future use.

When reserving discrete IDs, if the ID or set of IDs being reserved does not fall within the defined ID range, then those IDs will still be reserved but will be listed under the Master Model.

If you reserve IDs for an entity that supports ID pools, the reserved IDs will be applied to each ID pool. For example, the count for elements reserves in LS-DYNA includes reserves for each element pool available for the given ID range.

After IDs are reserved, they cannot be changed from renumbering operations. When an ID range is cleared, reserved IDs will be moved to the Master Model. If the ID range is re-entered, reserved IDs will be moved back to the entity it was originally reserved for. If you create an exclusion or delete an ID range at the entity level, the reserve range defined and the reserves defined within that range will be removed.

- Reserve IDs.
    a) From the menu bar, click **Tools** > **ID Manager**.
    The **ID Manager** opens.
    b) Reserve IDs.
        - To reserve all IDs, right-click on a Master Model, Include file, or entity and select **Reserve** > **All IDs** from the context menu.

All IDs within the defined ID range are automatically reserved if you right-click on an entity. If you right-click on the Master Model or an Include file you must select a type of entity to reserve all IDs for in the **Reserve All** dialog.

The Reserve All option can only be used when there is an ID range defined at the entity level. When you reserve all IDs, the Min Reserved and Max Reserved columns are populated using the Min and Max ID range defined at the entity level. The Min and Max Reserved columns can be manually edited as long as your modifications fall within the Min and Max ID range. IDs within the reserve range that already exist in the model will be populated in the Conflicts column.

- To reserve discrete IDs, right-click on a Master Model, Include file, or entity and select **Reserve** > **Discrete IDs** from the context menu.

  In the **Reserve** dialog, select an entity type and correction option, and enter IDs to be reserved.

- Unreserve IDs.
  a) From the menu bar, click **Tools** > **ID Manager**.
     The **ID Manager** opens.
  b) Unreserve IDs.
     - To unreserve all IDs, right-click on a Master Model, Include file, or entity and select **Un-Reserve** > **All IDs** from the context menu.
     - To unreserve discrete IDs, right-click on a Master Model, Include file, or entity and select **Un-Reserve** > **Discrete IDs** from the context menu. In the **Un-Reserve** dialog, select an entity type and enter IDs to be unreserved.

Compact Entities

IDs that are already occupied in an Include File can be compacted.

Overflowed IDs are then placed into the ID range, starting from the Min ID occupied without any gaps. Locked IDs in an Include file will not be renumbered.

1. From the menu bar, click **Tools** > **ID Manager**.
   The **ID Manager** opens.
2. Right-click on a Master Model, Include File, or entity and select **Compact Entities** from the context menu.

   > **Note:** This option is only available if you have selected single/multiple Include files, single/multiple child entities, or an Include file plus a child entity that has the ID range defined.

Review ID Overflow

1. From the menu bar, click **Tools** > **ID Manager**.
   The **ID Manager** opens.

2. Right-click on the Master Model or an Include file and select **ID Overflow List** from the context menu.
3. In the **ID Overflow List** dialog, review all of the overflow IDs in the Master Model or Include file.

   The Name column lists all of the entities in the Master Model/Include file that contains overflow IDs, along with the number of overflow IDs in each entity within parentheses. The Correction Option column displays the correction option currently assigned to each entity.
4. Review overflow IDs in the graphics area.
   - Left-click on each entity in the Name column to highlight it in the graphics area.
   - Right-click on each entity to access additional display options from the context menu, such as **Show**, **Hide**, or **Isolate**.
5. Correct ID overflow.
   - To correct the ID overflow for a specific entity, right-click on the entity and select **Correct** from the context menu.
   - To correct all of the ID overflow listed, click **Correct All**.

Correct ID Overflow

1. From the menu bar, click **Tools** > **ID Manager**.
   The **ID Manager** opens.
2. Right-click on an Include file or entity and select **Correct** > **Overflow** from the context menu.

Overflow IDs are corrected using the correction option assigned in the Correction Option column. If you do not right-click on an Include file or entity folder, then this operation will be performed on all of the overflow IDs inside of the model.

A message will appear before this action is performed, warning you that all of the overflow entities in the model will be corrected once you click OK.

If the ID Manager cannot correct all of the IDs, a warning message will be displayed and none of the overflow IDs within an Include file/entity folder will be corrected.

Correct Reserve Conflicts

1. From the menu bar, click **Tools** > **ID Manager**.
   The **ID Manager** opens.
2. Right-click on the entity folder that contains conflicts and select **Correct** > **Reserve Conflicts** from the context menu.

   > **Note:** Nodes and element entities are isolated in the graphics area.
3. In the panel area, use the selector or other extended entity selection options to select IDs to correct.
4. Click **proceed**.
5. In the **Correct Reserve Conflicts** dialog, select a correction method to resolve conflicts with.

- Choose **Move Out** to move the selected IDs out of reserve range and renumbers them using the max +1 rule, which is the maximum ID available plus 1. If the ID range is full, corrected IDs will be marked as overflow. The conflict icon will only be removed if you resolve all of the conflicts.
- Choose **Lock** to lock IDs in the reserve range and populates the Lock column. The conflict icon will display, and all of the IDs listed in the reserve range are locked.

6. Click **OK**.

Import/Export a CSV File of ID Management Rules

Import/export a `.csv` file of ID management rules

ID management rules can be exported for only the entities that are currently displayed in the ID Manager, or for all of the entities in the ID Manager including those excluded by ID management filters.

ID management data related to overflow, min/max occupied IDs, number of conflicts, number of reserves, and number of locks will be calculated while reading the `.csv` file.

When you are importing a `.csv` file, you will be notified if overlapped ID ranges are identified.

1. From the menu bar, click **Tools** > **ID Manager**.
   The **ID Manager** opens.
2. Right-click and select **Import CSV file** or **Export CSV file** from the context menu.

ID Manager Dialog

Overview of the ID Manager user interface.

Access the ID Manager by clicking **Tools** > **ID Manager** from the menu bar when the Abaqus, LS-DYNA, Nastran, OptiStruct, PAM-CRASH 2G and Radioss user profiles are loaded.

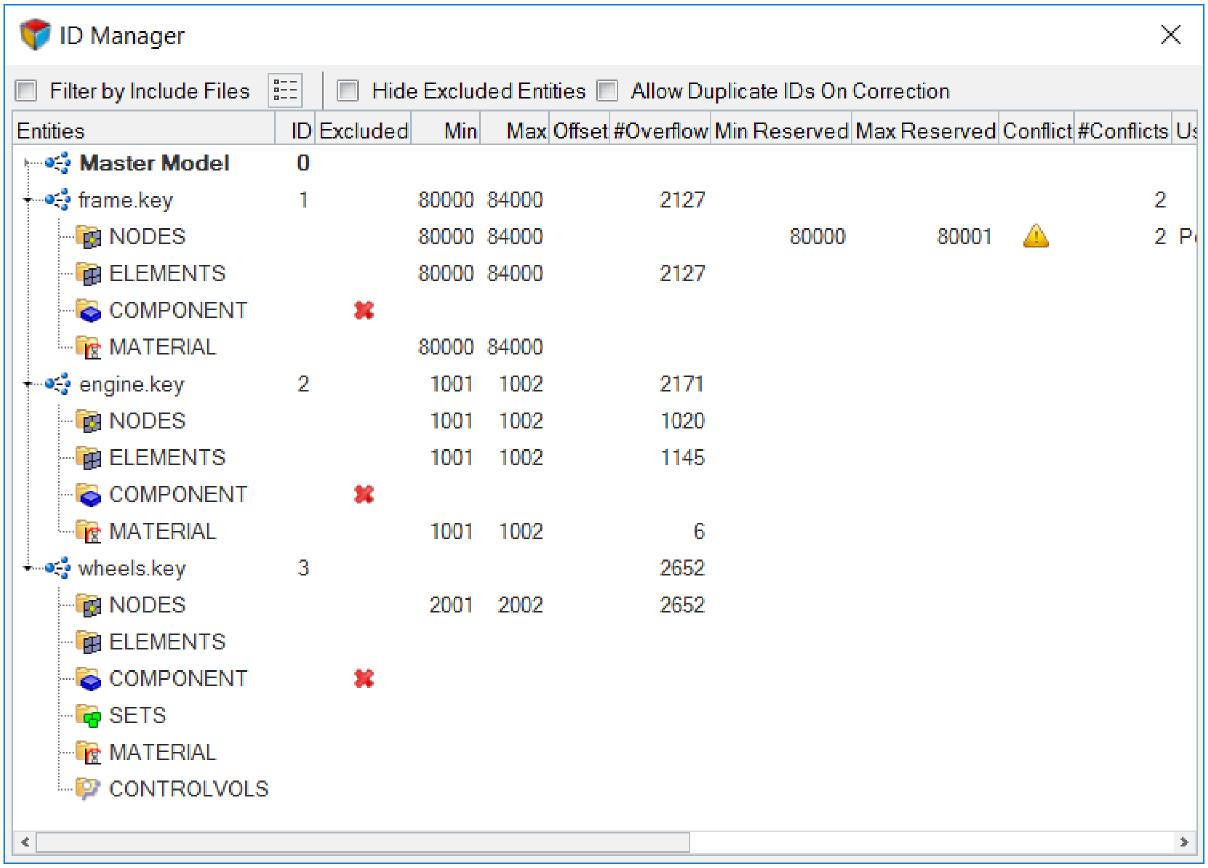


Figure 1675:

Table 264: ID Manager Options

Option	Description
Filter by Include Files	When enabled, Include files selected in the Select Includes dialog are filtered from the ID Manager. Click [icon] to open the **Select Includes** dialog. Data related to filtered Include files is saved to an `.hm` file.
Hide Excluded Entities	When enabled, entities marked as "Excluded" are hidden in the ID Manager. Data related to "Excluded" entities is saved to an `.hm` file.
Allow Duplicate IDs on Correction	When enabled, duplicate ID pool's for supported entities are permitted to have the same IDs.
Entities	Displays the Master Model and its child entities, as well as each Include file and its child entities.
ID	Displays the ID of each Include file.

Option	Description
Excluded	Displays an X icon ( ) when an entity is excluded from ID management operations.
Min	Enter a minimum integer value for the ID range that does not overlap with any other range definitions in the model. The Min should always be: • greater than the Max of the previous range defined in the model. • less than the Min of the next range defined in the model. • less than the Max of the current range.
Max	Enter a maximum integer value for the ID range that does not overlap with any other range definitions in the model. The Max should always be: • greater than the Max of the previous range defined in the model. • less than the Min of the next range defined in the model. • greater than the Min of the current Include file.
#Overflow	Displays the number of IDs in an Include file or entity folder that are outside of the defined range.
Min Reserved	Minimum integer value for the reserve ID range. You can manually edit the Min Reserved column as long as your modifications do not violate the Min and Max ID range. When you reserve all IDs, the Min Reserved and Max Reserved columns are populated using the Min and Max ID range defined at the entity level.
Max Reserved	Maximum integer value for the reserve ID range. You can manually edit the Max Reserved column as long as your modifications do not violate the Min and Max ID range. When you reserve all IDs, the Min Reserved and Max Reserved columns are populated using the Min and Max ID range defined at the entity level.
Conflict	Displays a conflict icon ( ) when there are IDs in the reserve range that already exist in the model.
#Conflicts	Displays the number of conflicts in a reserve range.
User Status	Indicates the status of reserve conflicts.

Option	Description
	Pending Pending displays as soon as conflicts occur. When conflicts are resolved with the use of the Move Out correction method, the user status automatically resets to a blank status. **WIP (Work in Progress)** When the conflict icon is displayed and conflicts have not been addressed or are not completely resolved, change the status from Pending to WIP. **Completed** When conflicts are resolved with the use of the Lock correction method, change the status to Completed. The conflict icon will still be displayed, but changing the status to Completed indicates conflicts have been resolved.
#Reserved	Displays the number of IDs reserved at the entity level. When IDs are reserved, this column is populated for the corresponding entity. All IDs within a defined ID range for an entity can be reserved, or discrete IDs within a defined ID range can be reserved. For IDs being reserved that do not fall into the defined ID range, these entities will be moved into the #Reserved column of the Master Model. If an entity type supports ID pools, the #Reserved column will display the total number of reserves applied to each ID pool in that entity type.
Min Occupied	Displays the minimum ID occupied by all of the entities in an Include file. For individual entities, it displays the corresponding minimum ID occupied in the Include file.
Max Occupied	Displays the maximum ID occupied by all of the entities in an Include file. For individual entities, it displays the corresponding maximum ID occupied in the Include file.
#Entities	Displays the total number of entities in Include files and entity folders.
#Locks	Displays the number of IDs that have been locked from renumbering.
New ID	Determines where new IDs will be placed within the current Include file range. **After Max** Uses any free IDs available starting after the maximum ID occupied within the defined range. **Before Min** Uses any free IDs available starting before the minimum ID occupied within the defined range.

Option	Description
	Fill in Gaps Uses any free IDs available, starting after the minimum ID within the defined range. If the ID Manager cannot find any free IDs using the **After Max** and **Before Min** options, then newly created entities will be assigned IDs using the **Fill in Gaps** option. If the ID Manager cannot find any free IDs within the defined ID range using any of the above options, then newly created entities will be assigned IDs beyond the Max ID occupied or the Max ID defined in the model, whichever one is being the highest.
Correction	Determines how duplicate IDs are renumbered to correct overflow. **None** Does not correct overflow IDs. **Compact And Fit** Compacts the IDs that are already occupied in an Include file, and moves the overflow IDs into the range starting from the Min ID in the defined ID range. **Move After Max** Moves the overflow IDs after the Max ID occupied in the defined ID range. **Move Before Min** Moves the overflow IDs before the Min ID occupied in the defined ID range. **Insert In Gaps** Moves the overflow IDs into gaps within the defined ID range. **Note:** IDs that are locked or reserved in an Include file will not be renumbered.

ID Manager Examples

Examples to provide you with a more in-depth understanding of the ID-management functionality.

New ID

The example below explains how new components are assigned an ID after you have defined an ID range and established rules for ID management.

An Include file contains components with the following IDs: 4, 5, 10 through 50, and 80 through 97. From the ID Manager, an ID range of 1 through 100 is defined for the Include file. When new components are added to the Include file, they will be assigned an ID based on the **New ID** option selected.

After Max

New components will be assigned IDs from 98, 99, and 100. Once the Max ID of 100 reached, any new component created will use the **Fill in Gap** option and be assigned IDs starting from 1.

Before Min

New components will be assigned IDs from 3, 2, and 1. Once the Min ID of 1 reached, any new component created will use the **Fill in Gap** option and be assigned IDs starting from 1.

Fill in Gaps

New components will be assigned available IDs starting from the Min ID of 1, 2, 3, 6, and so on.

ID Correction

The example below explains how the different correction options correct overflow IDs.

An Include file contains components with the following IDs: 4, 5, 20 through 60, 80 through 90, 101, 105, 110, and 117. From the ID Manager, an ID range of 10 through 100 is defined for the Include file, resulting in six overflow components. To correct the overflow IDs, a correction option is selected. The behavior of each correction option is as follows:

None

No correction is performed and the overflow IDs remain the same.

Compact and Fit

Compacts the IDs that are already occupied in the Include, and places the overflow IDs into the range starting from the Min ID of the defined range. The overflow IDs will be assigned the following new IDs:

IDs Before Correction	IDs After Correction
4	10
5	11
20 -60	12 - 52
80 - 90	53 - 63
101	64
105	65
110	66
117	67

Move After Max

Places the IDs that overflow after the Max ID occupied in the defined range. The overflow IDs will be assigned the following new IDs:

IDs Before Correction	IDs After Correction
4	91
5	92
20 - 60	20 - 60
80 - 90	80 - 90
101	93
105	94
110	95
117	96

Move Before Min

Places the IDs that overflow before the Min ID occupied in the defined range. The overflow IDs will be assigned the following new IDs:

IDs Before Correction	IDs After Correction
4	19
5	18
20 - 60	20 - 60
80 - 90	80 - 90
101	17
105	16
110	15
117	14

Insert in Gaps

Places the IDs that overflow into the gaps in the defined range. The overflow IDs will be assigned the following new IDs:

IDs Before Correction	IDs After Correction
4	10

IDs Before Correction	IDs After Correction
5	11
20 -60	20 - 60
80 - 90	80 - 90
101	12
105	13
110	14
117	15

Reserve ID

The following examples explain how to reserve IDs using the ID Manager.

An Include file contains components with the following IDs: 4, 5, 20 - 60, 80 - 90, 101, 105, 110, and 117. From the ID Manager, an ID range of 10 through 100 is defined for the Include file, resulting in six overflow components. To correct the overflow IDs, a correction option is selected.

By reserving all of the IDs within the range 1-100, the Number of ID reserves will be populated to 46, which is number of available IDs for the ID range 1-100 (available IDs include: 1-3, 6-19, 61-79 and 91-100).

If a reserve is applied to an entity that supports ID pools, the reserves will be applied to each ID pool of that entity. For example, consider a LS-DYNA model that has 10 ID pools for properties. If an ID range of 1-100 as defined, as in the above example with the same individual IDs for properties as opposed to components, the total count of reserves will be 460 (46 per pool and a total 10 pools).

Manage Penetrations/Intersections

Check components or groups for element penetrations and intersections using the Penetration Check tool.

Penetration and intersection can be used individually or collectively. Penetration is defined as the overlap of the material thickness of shell elements, while intersection is defined as elements passing completely through one another.

1. From the menu bar, click **Tools** > **Penetration Check**.

   > **Restriction:** Only available in the Radioss and LS-DYNA user profiles.

   The **Penetrations** browser opens.
2. Setup the Penetration Check and check for intersections/penetrations.
   a) In the Penetrations browser, click to invoke the collision setup widget.

b) In the Check type field, select the type of collision to check.

c) In the Entity type field, select the type of entity to be checked for intersections and/or penetrations.

d) In the Selection field, select contacts (groups) or components to check.

Selecting one of more contacts will perform the penetration check according to the rules enforced by the solver. In any case, the penetrating elements will be found, and the results will display in the Penetration Browser. Results will be listed by pairs of components regardless of the entity type that was used to select penetration candidates.

e) Select a Thickness option.

> **Restriction:** Only available for Component checks.

- Choose **Component thickness** to apply no adjustments as it uses the thickness value specified in a component's property card for each element within that component.
- Choose **Thickness multiplier** to multiply the selected entities' thickness by the value entered in the Thickness multiplier field for purposes of the penetration check. Fractional values are acceptable, but negative values are not.
- Choose **Uniform thickness** to ignore the existing component's thickness, and instead uses the value entered in the Uniform Thickness field for all of the components in the model.

  > **Note:** Use Uniform thickness as a workaround to the lack of thickness information in the default HyperMesh user profile, or when working with models that do not have a thickness specified.

f) Select a **Thickness** > **size** option.

> **Restriction:** Only available for Component checks.

- Full thickness, but ignore neighborhood (slow, but accurate if thickness > element size)
- Reduce thickness to 40% of elem size (fast)
- Full thickness, consider neighborhood (special usage only)

g) To consider edge penetrations, select the **Consider edge penetrations** checkbox.

h) Select a treatment algorithm for the boundary shell edge.

- Choose **Flat edges** to consider external borders of the components flat.

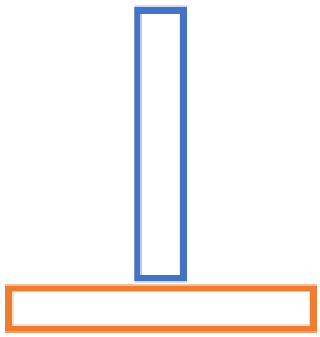


Figure 1676: Flat Edges

- Choose **Rounded edges** to extend external borders of the component by a cylinder having the diameter of the component thickness.

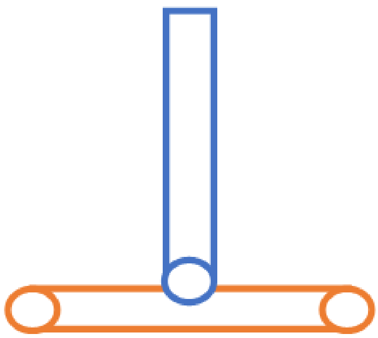


Figure 1677: Rounded Edges

i) Click **Check**.
Once the check is complete, the browser populates with detected intersections and/or penetrations.

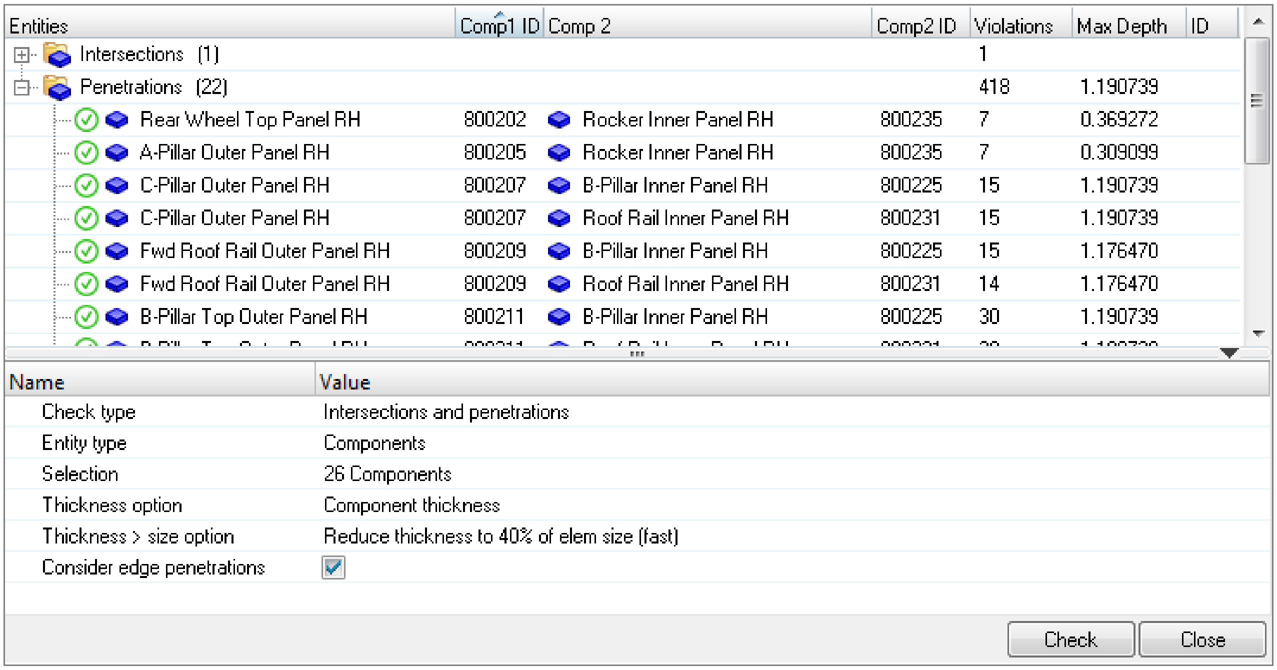


Figure 1678:

3. Review the penetrations/intersections using the view controls in the browser.

Figure 1679:

4. Fix the penetrations/intersections.

To fix	**Do this**
Manually	**1.** Select groups/components to fix in the browser. **2.** Click on the collision toolbar. Additional tools display.  *Figure 1680:* **3.** Select the elements or nodes to move. **4.** Move and/or translate the selected elements/nodes. **5.** Click to recheck that the intersection/penetration no longer remains.

To fix	Do this
Automatically	**1.** Select groups/components to fix in the browser. **2.** Click on the collision toolbar.

Tip:

- If a component is intersecting with another, right-click on the component and select **Find Matching Penetrating Component Pair** from the context menu to find the same pair of components in the penetrations list. If the pair does not penetrate, a message will display.
- To keep a specific component from changing when performing de-penetration fixes, right-click on that component and select **Lock Component** from the context menu. A red padlock displays on the component name in the browser to indicate that it has been locked. The nodes in a locked component cannot be moved by the Collision tool. To unlock, right-click a locked component and select **Unlock Component** from the context menu.

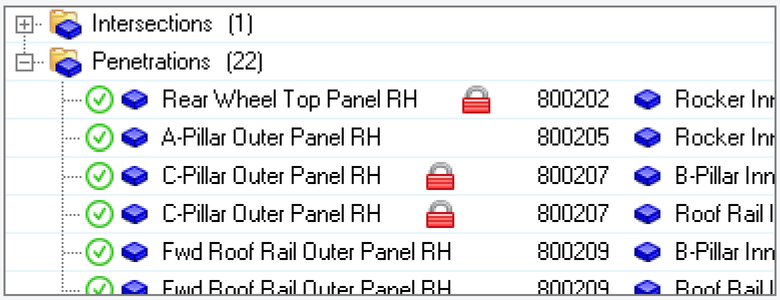


Figure 1681:

- Sort the columns in the browser by clicking the column headings. For example, clicking the **Violations** heading sorts the parent components according to their number of violations. A small triangular arrow in the column heading indicates whether the components are sorted in ascending or descending order; repeated clicks toggle between these two options.

Penetration Check Browser

Overview of the Penetration Check Browser user interface.

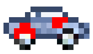
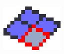
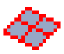
Collision Toolbar

Option	Description
	Invokes the collision setup widget, which can be used to setup the Penetration Check and check for intersections/penetrations.
	Reruns the collision check. This is recommended when you modify any attributes that control the collision check, or when a mesh modification has occurred.
	Automatically attempts to fix the intersections/penetrations you have selected in the browser, based on the settings found in the **Options** dialog.

Option	Description
	You can perform a fix on all of the intersections/penetrations listed in the browser, but it is highly recommended that all intersections are resolved before any automatic penetration fix is executed.
	Enables you to manually fix intersections/penetrations. When selected, additional tools display in the browser that can be used to perform manual, rather than automatic, penetration fixes.
	Exports the result of the collision run to a `.txt` or `.csv` file. Nodes and nodes + element pairs are supported.
	When multiple collisions are displayed in graphics area, click this button to identity the collision pairs in the collision browser after making a seletion in the graphics area.
	Displays a node List for selected penetration that reports penetration depth (thickness minus residual distance), thickness, relative penetration, (penetration divided by thickness) and residual distance for all penetrating nodes.

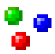
View Controls

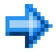
Button	Action
	Highlights all elements that caused penetrations or intersections when you select a component in the browser.
	Makes all elements transparent (wireframe) except for the interacting elements of the component that you selected in the browser, which display in a solid color.
	Displays a color gradient of the penetrating elements in the selected component, which indicates the severity (degree) of penetration for the interacting elements. This mode is not available for intersections because their depth cannot be determined.
	Displays a color gradient of relative penetrating elements.
	Displays individual vectors for each penetrating element in the component that you select in the browser. These vectors indicate the direction and depth of penetration for both the selected component and its interacting components. This mode is not available for intersections because their depth cannot be determined.
	Fits the failed elements to the display. In large models, this can be very helpful in finding and viewing small areas of minor penetration. Note that while this option is active, the view automatically fits to the

Button	Action
	penetrating elements of any component that you click in the browser. Click the option again to deactivate the fit mode.
	Displays all elements, by unmasking all elements in the model, but not other masked entities such as model geometry.
	Masks everything in the model except for components with penetrations or intersections. Note that this option only applies to components for which you have run the current penetration check; other components may be interacting, but if you have not run a check on them they do not appear as interacting, and they will be masked.
	Masks everything in the model, including the interacting components, except for the specific elements that penetrate or intersect.

Manual Fix Tools

Button	Action
	Select Elements By Tree. When enabled, clicking the lowest-level component in the browser selects all of its failed elements.
	Select Elements Manually. When enabled, you can click each desired element belonging to the lowest-level component in the browser. This includes the ability to select non-failed elements or a sub-set of the failed elements.
	Select Nodes By Tree. When enabled, clicking the lowest-level component in the browser selects all of the penetrating nodes in its failed elements. This differs from Select Elements By Tree in that individual nodes can be moved to fix a penetration, thus changing the shape of a failed element, instead of moving entire elements.
	Select Nodes Manually. When enabled, you can click each desired node belonging to the lowest-level component in the browser. This includes the ability to select individual nodes of non-failed elements, or a sub-set of nodes belonging to the failed elements.
	Determines the direction that you wish to manually move the selected nodes or elements. Click the small triangle, in the bottom corner, to select one of the following:  Move along the average normal of the selected elements, or related elements in the case of selected nodes.  Move along a fixed vector.

Button	Action
	 Move along the X axis.  Move along the Y axis.  Move along the Z axis.  Move along an already-existing vector entity in your model that you select.  Move along nodes in your model that you select to define the direction vector. If you pick two nodes, they define the direction. If you pick three nodes, the direction is the normal of the plane that these three nodes define (picking more than three nodes uses only the last three picked.)
	Moves the selected nodes/elements by the negative amount specified in the numeric text box.

Button	**Action**
	Moves the selected nodes/elements by the positive amount specified in the numeric text box.

Review Orientation

Display/review and modify frequently used coordinate systems used in CAE analysis.

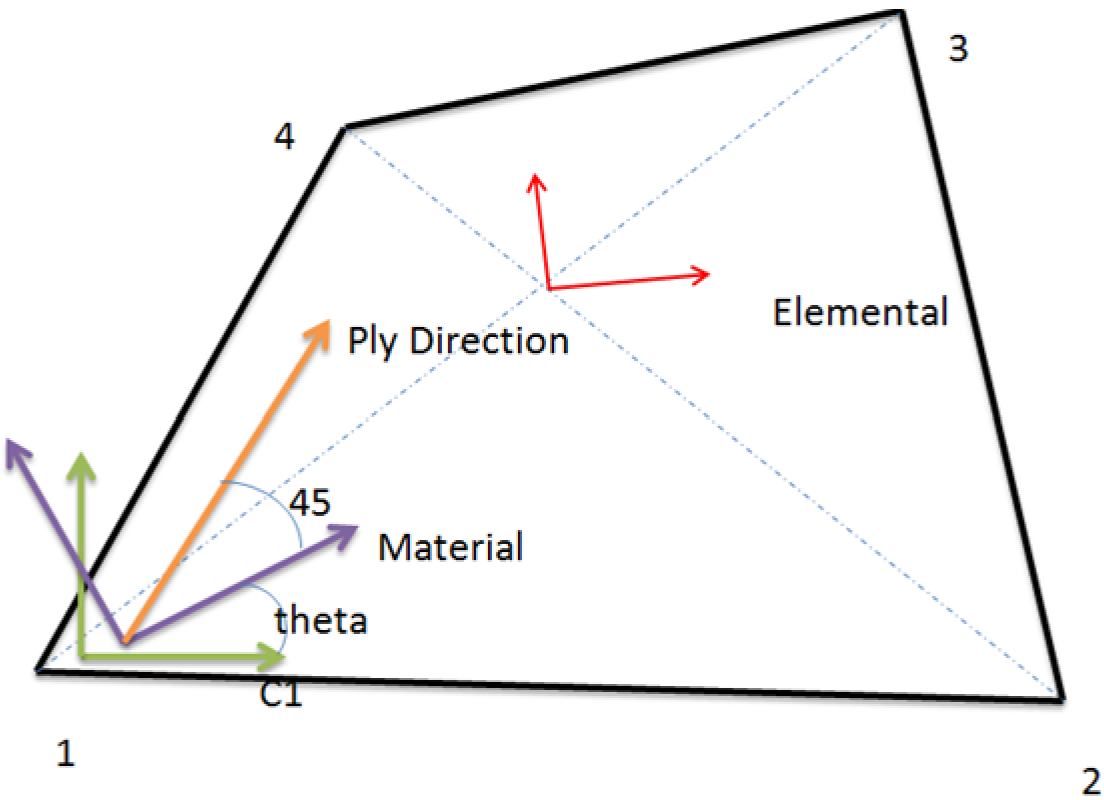


Figure 1682:

Review Elemental Systems

Select 1D, 2D, or 3D elements and display the coordinate system.

Elemental systems are dependent on the solver type. For example, for Nastran and for quad elements the X axis is determined using the bisector of the diagonal line.

1. From the menu bar, click **Tools** > **Orientation Review**.
   The **Orientation Review** dialog opens.
2. Click the **Elemental system** tab.
3. Use the Elements selector to select elements.

4. Define display options.
5. Click **Apply** to show the system display.

The mode element can be added and the display can be super imposed. Clicking **Clear** removes the information that you added, and clicking **Clear All** removes the system.

Review Material Systems

1. From the menu bar, click **Tools** > **Orientation Review**.
   The **Orientation Review** dialog opens.
2. Click the **Material system** tab.
3. Use the Elements selector to select elements.
4. Define display options.
5. Click **Apply** to show the system display.

By default the material system aligns with the C1 direction of the element edge. C1 is the direction from node 1 to node 2 of the element. You can super impose the elemental system on the same plot of the material system.

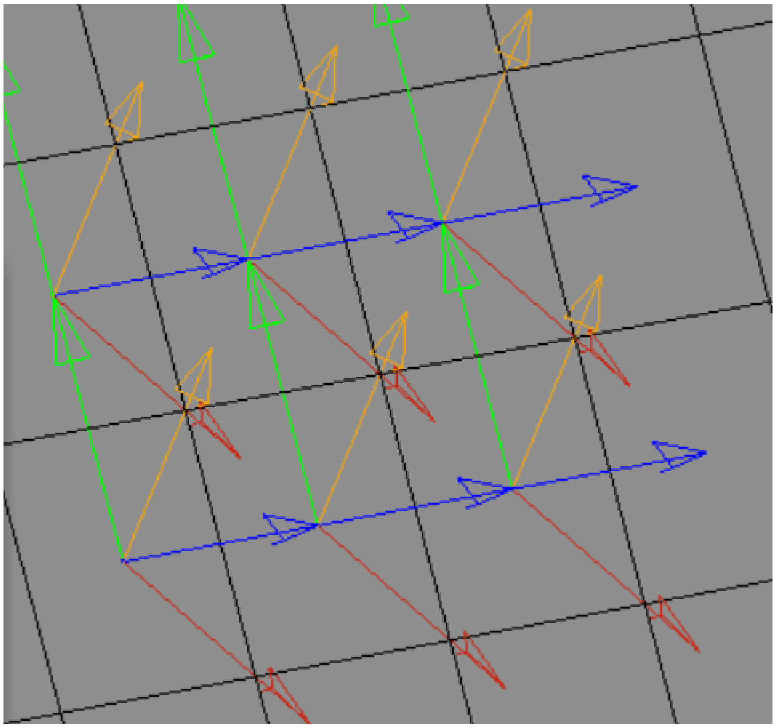


Figure 1683:

The blue and green arrows represent the elemental system (x and y) and the red and brown arrows represent the material system (x and y).

Review Ply Directions

During CAD import of the composite data, each ply is associated with one or a separate system (one system for several plies or a system for each ply). You can select a ply and review its system.

This is important to align the element normal to match the ply normal, or correct the ply angles if the normals are in the opposite direction.

1. From the menu bar, click **Tools** > **Orientation Review**.
   The **Orientation Review** dialog opens.
2. Click the **Ply Directions** tab.
3. Use the ply selector to select plies.
4. Define ply options.
   - To show fiber direction without the correction for drape, select the **Fiber orientation** checkbox.
   - To show fiber direction with drape correction, select the **Drape fiber orientation** checkbox.
   - To show a ply system for each ply, select the **System** checkbox.
5. Define display options.
6. Click **Apply** to show the ply system.

Figure 1684:
The green arrows show the material X direction. The blue arrow shows the ply 5 fiber direction without drape. The red arrows shows ply 5 fiber direction with drape.

Review Orientation

Display and correct element orientation.

Element orientation is determined by the order of the nodes in the element connectivity. This is sometimes referred to as C1, C2, C3, or element local system.

1. From the menu bar, click **Tools** > **Orientation Review**.

The **Orientation Review** dialog opens.

2. Click the **Orientation** tab.
3. Use the elements selector to select elements.
4. Select a function.
   - Choose **By element** to select an element with the correct orientation.
   - Choose **By edge_node** to select edge nodes with the correct direction.
   - Choose **By System** to select a system with the correct 1, 2, 3 direction.
   - Choose **Vector** to select a vector direction.
   - Choose **Align N1-N2 dir** to select a selection method.
5. Define display options.
6. Click **Apply** to show the element orientation.

The orientation of the rest of the elements will be determined based on the orientation or direction of the selected entity.

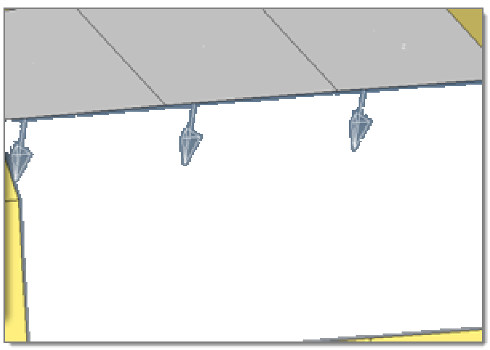


Figure 1685: Display Normal and Reverse Normal 2D Elements

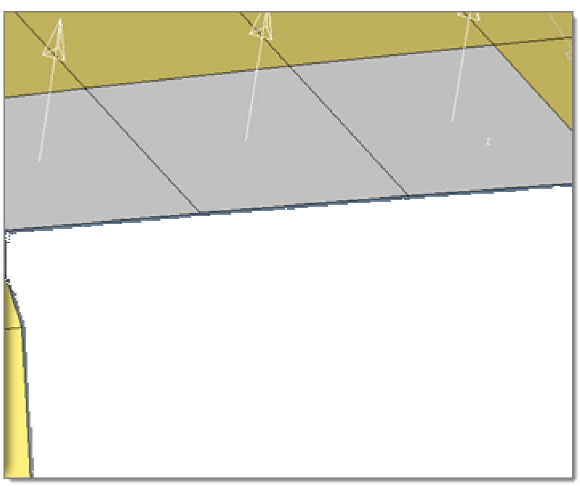


Figure 1686: Reverse Normals

The element already selected during the display normal will be reversed.

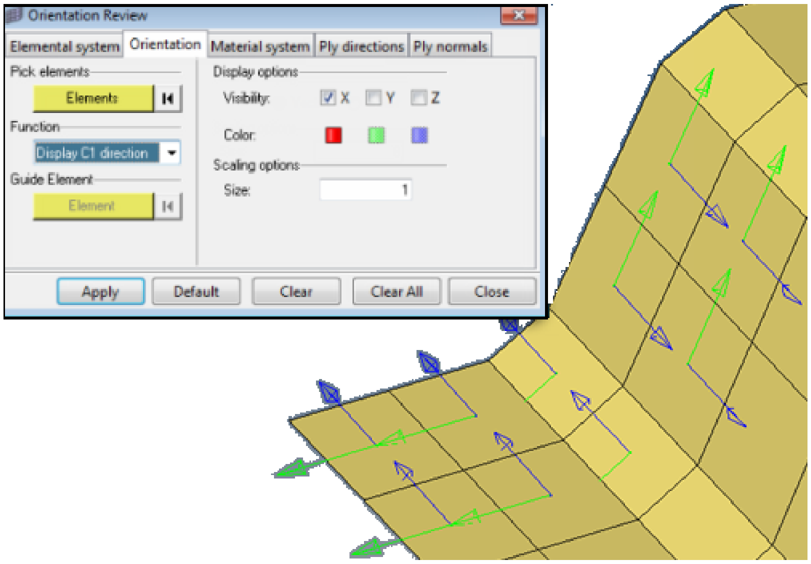


Figure 1687: C1, C2, C3 Direction

This displays the element orientation of the nodes. C1 is the direction from node 1 to node 2 using the element connectivity order.

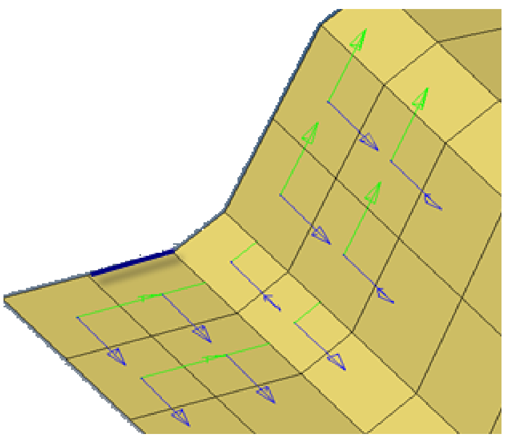


Figure 1688: Resequence Nodes

Align C1 direction of 2D quad elements. Using the display C1 direction, some of the elements may not be aligned, as shown in the previous image. Select a guide element as the reference element and the rest of the element node connectivity will be changed to align the C1 direction.

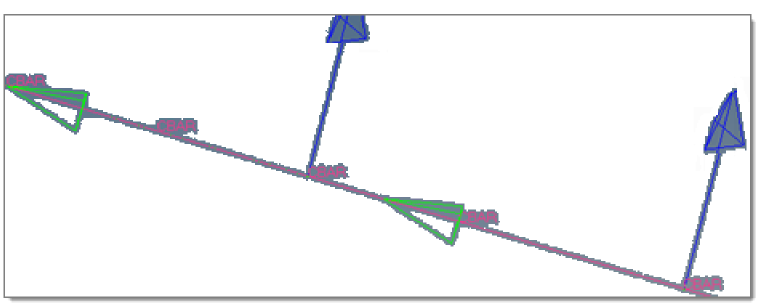


Figure 1689: Reverse 1D Element Direction

T function reverses the X direction (node 1 to node 2) of a 1D element. This image shows before reversing the elements.

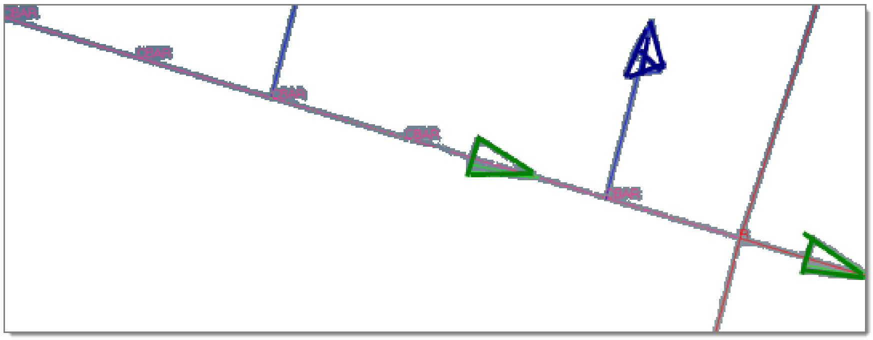


Figure 1690: After Reversing Elements

Review Ply Normals

Indicate which ply angles need to be reversed for which elements when they do not match the ply normal.

CAD based ply angles and orientation are transformed with respect to the element material system for analysis solvers. Most of the time you can correct the element normal to match the ply normal from the CAD. There are times when it is not possible to change the element normal to match the ply normal, as shown in the diagram below. In these cases, you need to change the ply angles with respect to the element normal without changing the element normal. For example, a +45 degree ply will become a -45 degree ply if the element normal and ply normal are in opposite directions.

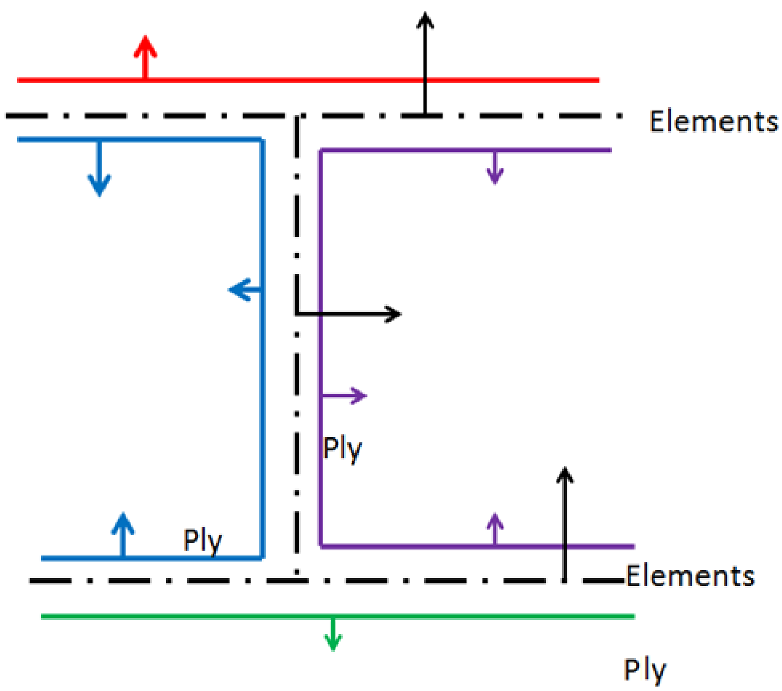


Figure 1691:

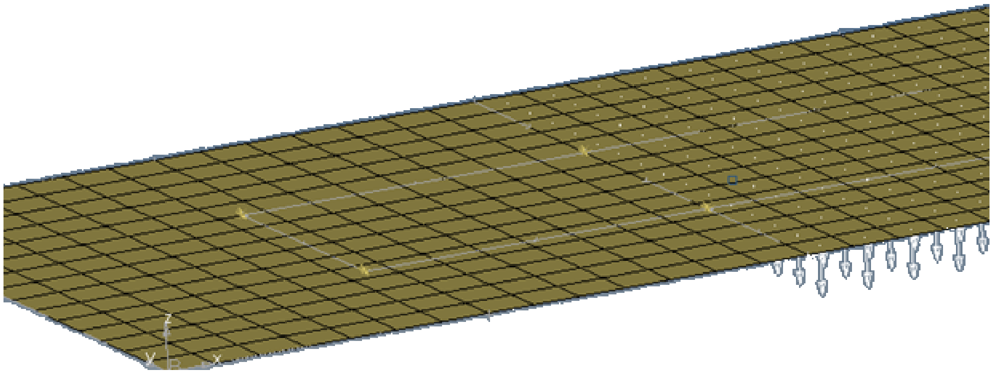


Figure 1692:
The ply normal does not match the element normal.

1. From the menu bar, click **Tools** > **Orientation Review**.
   The **Orientation Review** dialog opens.
2. Click the **Ply Normals** tab.
3. Use the ply selector to select the single ply for which the element normal needs to be reversed.
4. Use the Elements selector to select the elements that need to be reversed.
5. Under Noramls, specify how to display normal.
   - Choose **Display** to display normals.
   - Choose **Reverse** to reverse the display of the normal if they do not match.
6. Define display options.
7. Click **Apply** to show the ply normals.
8. Repeat these steps for each ply.
   The element normal for the ply is now reversed.

   > **Note:** The actual element normal is not changed. The software takes a note (sets a flag internally) for a particular ply, and the angle needs to be reversed for that element, instead of reversing the normal. This is reflected in the solver data.

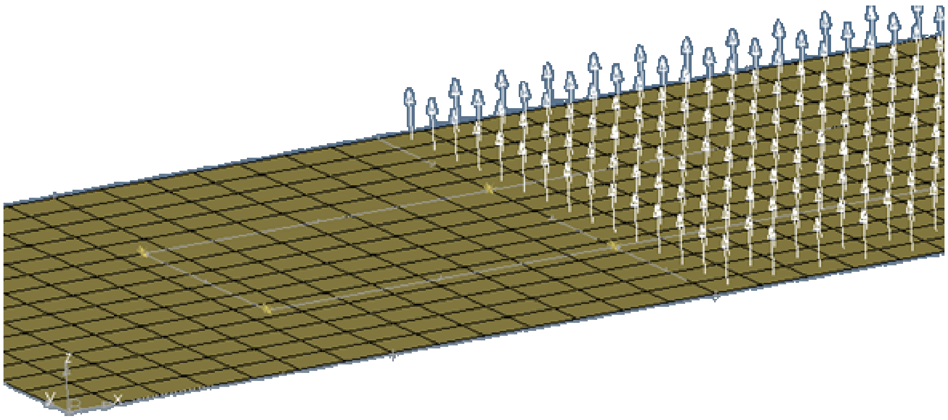


Figure 1693:

Manage Transformations

Manage transformations and/or positions.

Manage Transformations with the Transformation Manager

Manage the definition of transformation applied on the nodes and elements in order to easily position the entity using a combination of scale, rotate, and translate and symmetry functions.

These operations are usually applied on standard parts such as dummies, barriers, and so on for positioning in different crash setups. Also they can be used for repositioning components with strain history between multiple steps of simulation.

Access the Transformation Manager by selecting **Tools** > **Transformation Manager** from the menu bar.

> **Restriction:** The Transformation Manager is available LS-DYNA and PAM-CRASH 2G solver profiles.

The Transformation Manager has two basic folders/sections.

Transformation Definition
: Lists the transformations defined in the model in a tree structure detailing the cross referencing between the entity and the transformation.

Transformation Repository
: Holds the candidates on which the transformations can be applied which is solver dependent. In the PAM-CRASH 2G solver profile, the repository contains all the node sets defined in the model.

In the LS-DYNA solver profile, the repository contains node sets and include files organized in two different folders.

Toolbars

The Transformation Manager includes toolbars and a context menu. Toolbars provide the ability to switch between the two views, show or hide entities on which transformation is applied within the model, and activate or deactivate the transformation applied on an entity. The context menu provide basic functions such as the creation, addition of entities, activation/inactivation of transformation, deletion, card edit, manipulating the sequence of transformation, display control, and review.

Icon	Function
	Activate the selected transformation. If none are selected, activate all the transformations defined.
	Inactive the selected transformation. If none are selected, inactivate all the transformations defined.
	Active or inactive state depending in its current state for the selected transformation. If none is selected, it is applied on all the transformations defined.
	Switches to transformation – entity view
	Switches to entity – transformation view

Context Menu

A context menu is available for any selected item in the Transformation Manager tree. To view the context menu right-click on an entity folder, an individual entity, or in an empty space in the browser. Clicking in the empty space provides options applicable to all the entities in the transformation definition. The functions vary depending on whether it is performed on an entity in the transformation repository or in a transformation definition. There are also subtle differences in functions performed depending on the view.

Transformation Definition

Table 265: Entity- Transformation View

Item	Definition
Create New	Creates new entity on which transformation will be applied. Includes node sets for the PAM-CRASH 2G solver profile. Includes node sets and include files for the LS-DYNA user profile.
Add entity	Allows you to edit the entity on which transformation is defined. Available only on entity.

Item	Definition
Card edit	Opens the corresponding solver Card Image panel. Available on both entity and transformation.
Delete	Deletes the selected entity. Available on both entity and transformation.
Transform	Creates a new transformation and applies it on the selected entity. If transformation already exists for the entity, it appends to it. Available on the entity.
Rename	Renames the selected entity/ transformation. Available on both entity and transformation.
Show, Hide, Isolate, Show all, Review, Reset review:	Similar to Model Browser functions. Available for both transformation and entity.
Move UP	Swaps the ID of the selected transformation with the next smaller ID that is attached to the same entity.
Move up	Swaps the ID of the selected transformation with the next higher ID that is attached to the same entity.
Collapse All	Closes all of the folders in the tree structure, so that only the top-most level of items displays.
Expand All	Opens all of the folders in the entire tree structure, exposing every item nested at every level.

Table 266: Transformation - Entity View

Item	Definition
Create new	Creates new transformation.
Add entity	Allows to add a new entity on to the selected transformation. If the entity is selected, you can edit the content. Available on both entity and transformation.
Card edit	Opens the corresponding solver Card Image panel. Available on both entity and transformation.
Delete	Deletes the selected entity. Available on both entity and transformation.
Transform	Appends the defined transformation on the selected transformation. Available only on the transformation.

Item	Definition
Rename	Renames the selected entity. Available on both entity and transformation.
Show, Hide, Isolate, Show all, Review, Reset review	Similar to Model Browser functions. Available for both transformation and entity.
Move Up	Swaps the ID of the selected transformation with the next smaller ID that is attached to the same entity.
Move Down	Swaps the ID of the selected transformation with the next higher ID that is attached to the same entity.
Collapse All	Closes all of the folders in the tree structure, so that only the top-most level of items displays.
Expand All	Opens all of the folders in the entire tree structure, exposing every item nested at every level.

Transformation Repository

Item	Definition
Transform	Creates new transformation on the selected entity. The transformation and the entity show up in the transformation definition folder.
Card Edit	Opens the corresponding solver Card Image panel.
Rename	Renames the selected entity.
Show, Hide, Isolate, Review, Reset review:	Similar to Model Browser functions.

Supported Solver Keywords

LS-DYNA

NODE_TRANSFORM

INCLUDE_TRANSFORM

PAM-CRASH 2G

TRSFM _/_

Manage Positions/Transformations

Apply a transformation sequence on a set of nodes to change their position in the model.

Related reference

Positions

Transformations

Abaqus

Transformations are applied in the sequence in which they are listed in the **Select Transformations** dialog. The list of transformations can have unique transformations, or a given transformation can be repeated as many times as required. If there are changes made to the applied list of transformations, for example if a Transformation was deleted, the Position entity will be automatically updated to incorporate these changes. If you clear the **Applied** checkbox, the transformations will be undone in the reverse order in which they were applied.

1. Open the Entity Editor for the Position entity you are adding a list of transformations to.

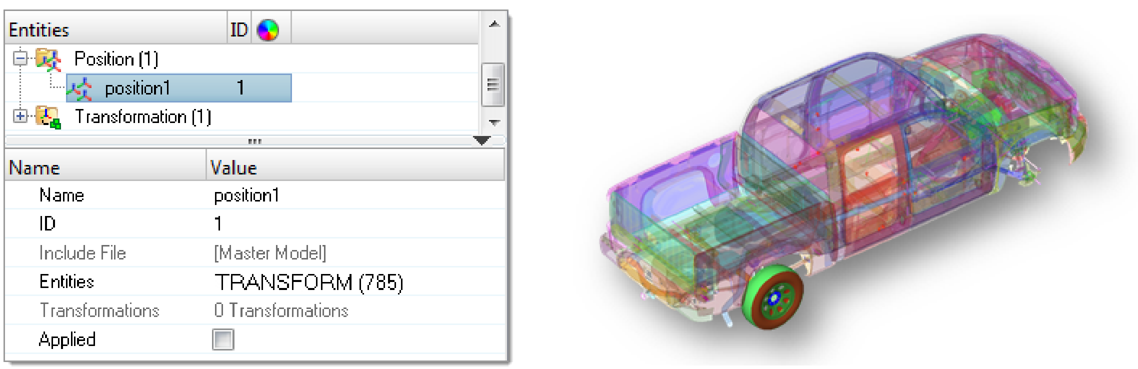


Figure 1694: Position Contains a Node Set for the Right, Rear Tire

2. In the Entity Editor, right-click on **Transformations** and select **Select Ordered List** from the context menu.
3. In the **Select Transformations** dialog, click to add transformations.
4. In the panel area, click **transformations**.
5. Select a transformation.
6. Click **proceed**.
7. If necessary, add additional transformations.

   To remove transformations, select a transformation and click .

To change the sequence of transformations, click .

8. Click **OK**.

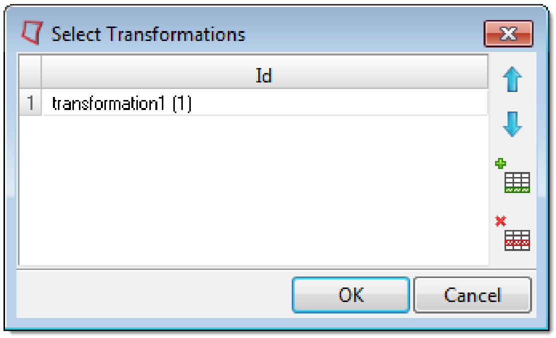


Figure 1695:

9. In the Entity Editor, select the **Applied** checkbox to apply the list of transformations for the Node Set in the Position entity.

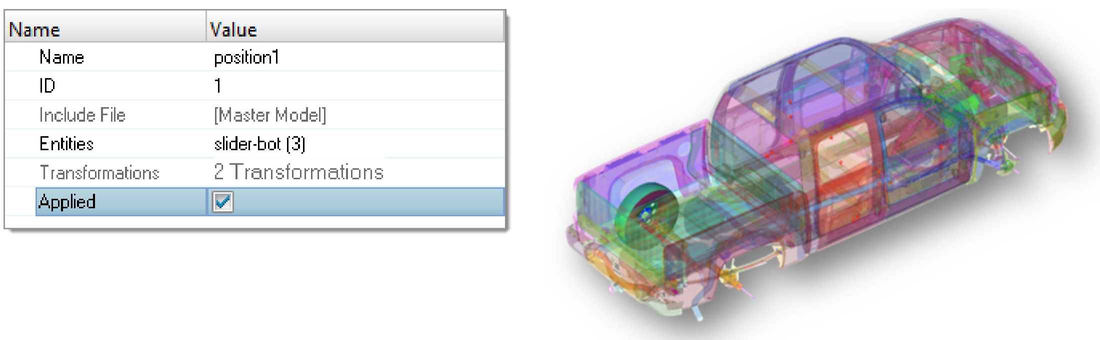


Figure 1696: Transformations Applied

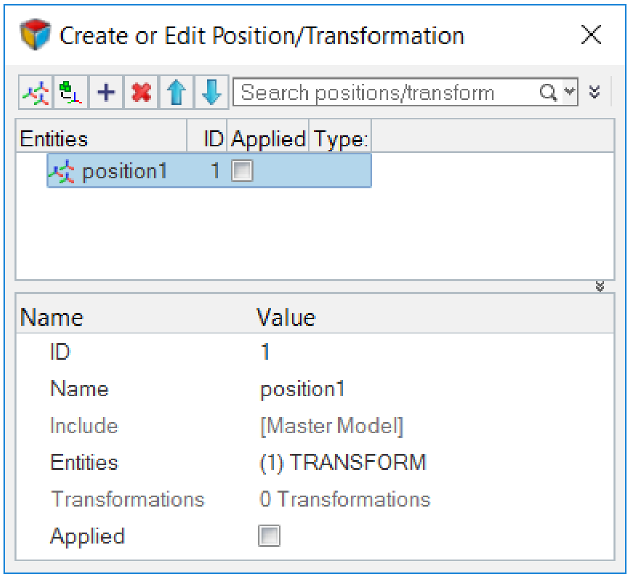
The right, rear tire moved to the left, rear side of the truck as a result of the applied transformations.

Radioss

Apply a transformation sequence on a set of nodes to change their position in the model.

In the Radioss solver profile, there is no solver keywords mapped to the position entity. Use the position entity to manage the transformations applied on a set of nodes or a Submodel. Position and transformations are managed in the **Position/Transformation** tool.

1. In the Model browser, right-click and select **Position/Transformation** from the context menu. The **Create or Edit Position/Transformation** dialog opens.
2. Create a position.
   a) Click .
      A new position is created, and a list of entity attributes are displayed in bottom pane.
   b) Edit the position's attributes accordingly.



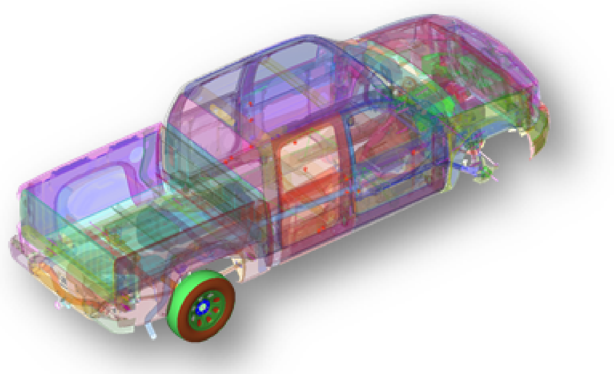


Figure 1697:

3. Apply a transformation to the position.

a) Select the position entity.

b) Apply a new or existing transformation to the position.

- Create a new transformation by clicking . Edit the transformation's attributes accordingly in the bottom pane.
- Add an existing transformation by clicking and selecting a transformation from the **Select Transformation** dialog. Click **OK** when finished.

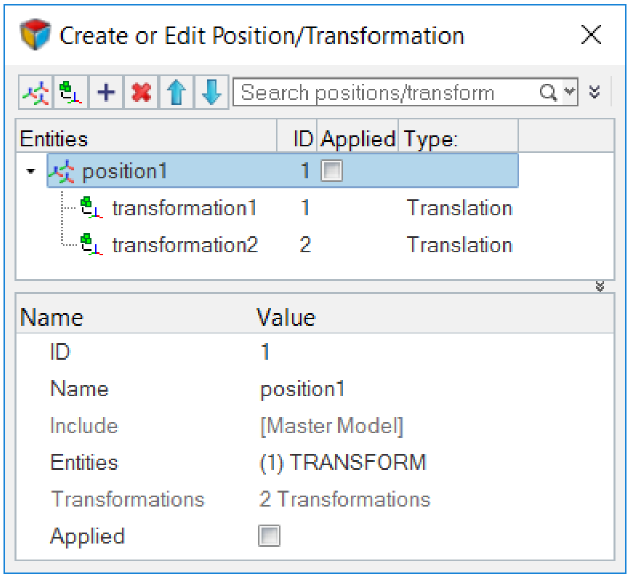


Figure 1698:

4. Select the **Applied** checkbox for the position entity to apply the list of transformations.

   You can select the Applied checkbox from the Applied column in the first pane, or by modifying the position's attributes in the second pane.

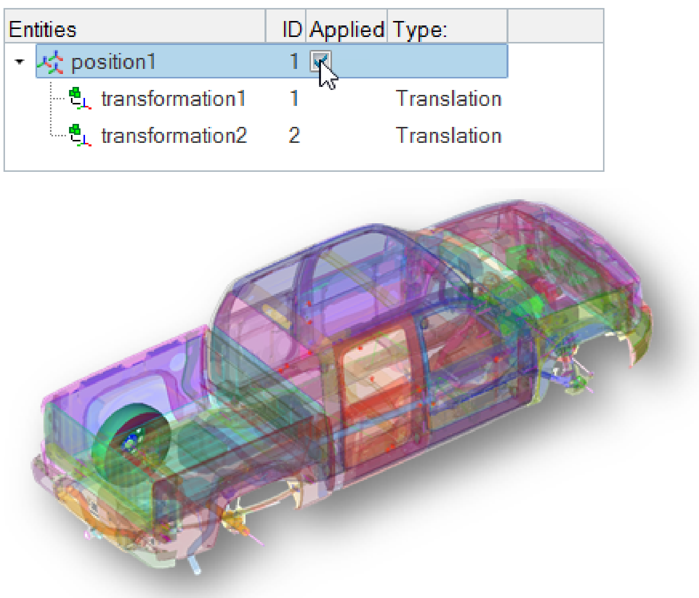


Figure 1699:

Tip:

- Search for positions/transformations using the search field.
- Change the sequence of transformations by clicking .
- Remove transformations by selecting a transformation and clicking .

Transform Elements

Translate, scale, reflect or rotate elements with multiple numbers of copies, including properties and loads, attached to them.

Loads will also be copied to the new transformed elements with the possibility to reverse them. ID's can also be assigned with offset for the newly created elements. Multiple instances (number of copies) can also be created.

1. From the menu bar, click **Tools** > **Transformation Tool**.
   The **Transformation Tool** dialog opens.
2. In the Action field, select an action to perform on elements.
3. In the Entities field, select source elements, nodes, components, or systems.
4. To create multiple instances, select the **Duplicate** checkbox.
5. In the Destination Component field, select where to organize elements after the transformation.
6. Define additional options accordingly.
7. Click **Apply**.

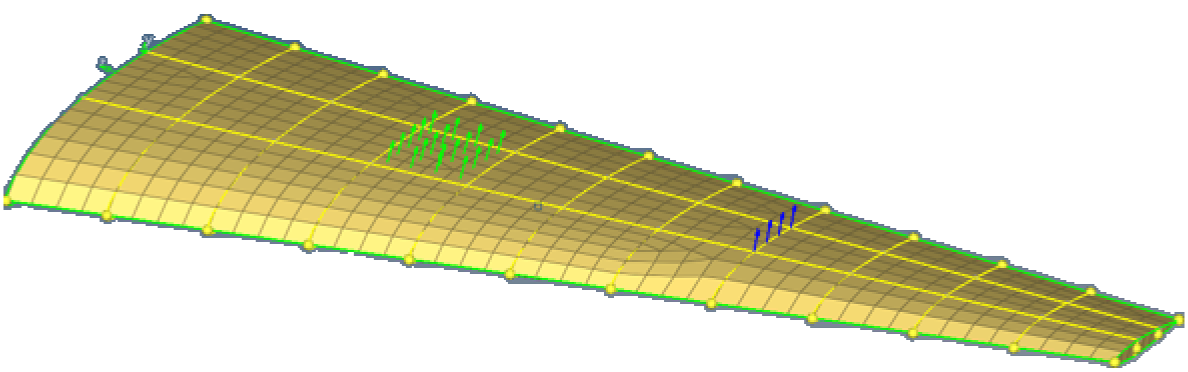


Figure 1700: Element to be Transformed with Loads

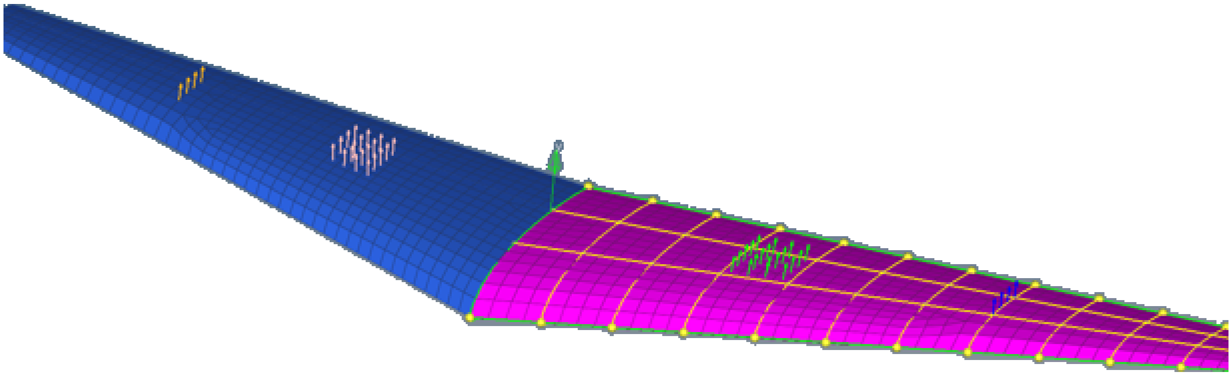


Figure 1701: Reflected Wing with Loads in a New Load Collector

Create Pretension Bolt

Create pretension loads in 1D and 3D bolts.

ANSYS: Create Pretension Bolt

Create pretension loads in 1D and 3D bolts of ANSYS models using the ANSYS PRETS179 element type.

Element PRETS179 is a bar element with a third node used for pre-loading. Use the Pretension Bolt tool to get a pre-loaded bolt with PRETS179 elements created at the desired location with a third node. A pretension section is created and associated with the pretension elements created. Also, the pretension load card SLOAD is created with the defined load in the utility, as well as a load steps on and off values. You can edit this load card after completing this process.

> **Restriction:** The Pretension Bolt tool is only available in the ANSYS solver interface.

1. From the menu bar, click **Tools** > **Pretension Bolt**.
   The **Pretension Bolt** dialog opens.
2. For Pretension Type, select the type of bolt to create.
3. In the Node ID field, enter the node ID for the third node of the pretension element.

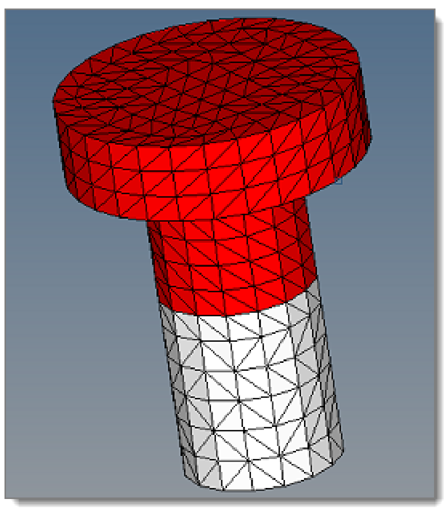
   > **Note:** If you do not enter a node ID, a default node number will be assigned.

4. In the Section Name field, enter a name for the pretension section to be created.

   > **Note:** If you do not enter a section name, a default name will be assigned.

5. Toggle the type of method for loading the bolt to either **Force** or **Displacement**, then enter a corresponding value.
6. In the Loadstep ID to activate [LSLOAD] field, enter the load step ID to which the load is to be applied.

7. In the Loadstep ID to lock [LSLOCK] field, enter the load step ID to which the displacements or force loads needs to be locked.
8. Click **Create**.
9. Select source entities for the pretension bolt.

Option	Description
1D pretension bolt	1. In the panel area, use the Nodes selector to select the nodes of the 1D elements where the pretension element needs to be created and then click **proceed**.
3D pretension bolt	1. In the panel area, use the Comps selector to select the bolt component and then click **proceed**. **Note:** Multiple components are not allowed to be selected. If more than one bolt is selected, they will all be placed under one component. 2. Use the elems selector to select the elements which form the cut section of the bolt and then click **proceed**. Pretension elements are created at this section.  *Figure 1702: Example of a Typical Section* 3. Use the node selector to select two or three nodes that define the load direction for the pretension load and then click **proceed**.

For 1D pretension bolts, PRETS179 elements are created at the node locations, with the pretension section created and associated to these elements. A SLOAD card with the given pretension load is also created.

For 3D pretension bolts, PRETS179 elements are created at the cut section with the pretension section created and associated to these elements. A SLOAD card with the given pretension load is also created.

10. To edit the SLOAD card, click **Edit SLOAD**.

Samcef: Create Pretension Bolt

Create and edit 1D and 3D pretension bolt loads and bolt sections.

Use the Pretension Manager to create and edit pretension bolts. Existing bolts and newly created bolts are displayed in the table within the **Pretension Manager** dialog. The card attribute's of every bolt present in the model are displayed. You can edit the output codes for each bolt in the Output column. Any changes made to the Output column will be automatically updated in the respective BOLT cards.

> **Restriction:** The Pretension Bolt tool is only available in the Samcef solver interface.

Create a 1D Pretension Bolt

1. From the menu bar, click **Tools** > **Pretension Manager**.
   The **Pretension Manager** dialog opens.
2. Click **Create New**.
3. Set the Bolt type to **1D**.
4. In the Name field, enter a name for the bolt.
5. Click **Element** > **Select Existing Element**.

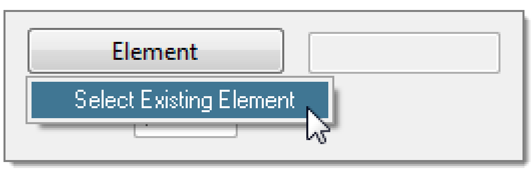


Figure 1703:

6. In the panel area, use the elems selector to select a single beam, bar, or rod element and click **proceed**.
   The ID of the selected element is displayed in the field to the right of the Element definition button.
7. To automatically generate a pretension node at the second node of the selected 1D element, select the **Auto generate pretension node** checkbox.

Figure 1704:

Once the 1D element is selected, the generated pretension node's ID is displayed in the Node ID field.

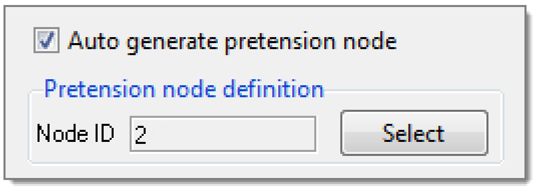


Figure 1705:

8. Select whether to apply a **Force** or **Displacement** load on the bolt, then enter a corresponding magnitude.

   > **Note:** A load is not created until a load collector is specified.

9. Click **Function Time**.
10. In the panel area, use the curves selector to select the required curve for the load and click **proceed**.
    The selected curve and its corresponding ID are displayed in the field to the right of the Function Time definition button.
11. Click **Load Collector** and select where to store the defined load.
    - Choose **Select Existing Collector** to open the panel area where you can use the loadcol selector to select an existing load collector.
    - Choose **Create New Collector** to create a new load collector with the prefix PRETENS_#. The number appended to the end of the load collector name depends on the number of load collectors currently present in the model.

    The load collector is created along the 1D bolt.

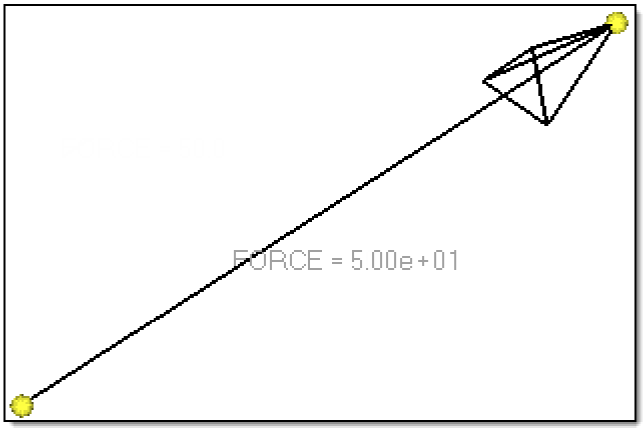


Figure 1706:

12. To create a FIX parameter, select the **FIX parameter** checkbox and enter a FIX value.
13. Select where the different codes will be internally written in the solver deck.
    - **Total axial force in the bolt (9524)**
    - **Relative displacement in the cut (9530)**
14. Click **Create**.

A BOLT is created as a group entity.

The macro element ID for the first bolt that was created is written out as the maximum element ID in the model + the number of nodes present in model. 1000 is added to the macro element ID for consecutive bolts, that is, if the first bolt's macro ID is 3, then the next bolt will be 1003 (3 + 1000).

The macro element ID of a bolt is reflected in its `.SAI` command, which is written along with the bolt's corresponding output-code. It is preceded by the comment !!HM_TEMP_OUTPUTBLOCK, which enables the model-reader to ignore the output block upon import and not create any output blocks in HyperMesh. A bolt card that does not contain an output definition will not have a `.SAI` command written out.

All such commented commands are automatically exported out again upon re-export in the same format.

The load applied on a bolt is written out in the solver deck. A load's magnitude is written out after VAL, and any curve attached to the load using Function Time is written after the keyword NF.

Create a 3D Pretension bolt

1. From the menu bar, click **Tools** > **Pretension Manager**.
   The **Pretension Manager** dialog opens.
2. Click **Create New**.
3. Set the Bolt type to **3D**.
4. In the Name field, enter a name for the bolt.
5. Click **Contact Surface** and select where to create a new surface or select an existing surface.

- Choose **Create New Surface** to open the panel area where you can use the node list selector to select two consecutive nodes to generate a contact surface. The first node should be a base node in the desired contact surface. The second node should lie in the direction of the contact surface's normal. A default name is assigned to the new contact surface with the prefix PRETENS_#. The number appended to the end of the name depends on the number of contact surfaces currently present in the model.

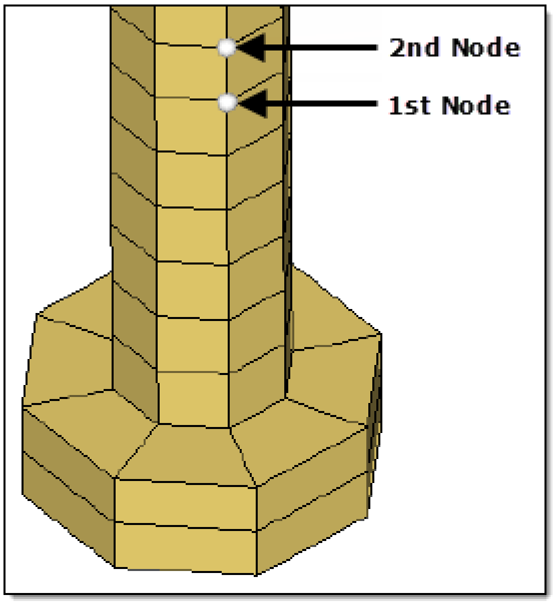


Figure 1707:

- Choose **Select Existing Surface** to open the panel area where you can use the contactsurf selector to select an existing contact surface.

6. Select a Method for defining the bolt.
- Choose **1** to create a .MCT contact between the two faces of the cut.
- Choose **2** to create a MEAN element and a new node on each side of the cut.

7. Select a pretension node.
- Choose **Manually**, then clear the **Auto generate pretension node** checkbox and click **Select**. In the panel area, use the nodes selector to select two consecutive nodes. The first node should be a base node in the desired contact surface. The second node should lie in the direction of the contact surface's normal. The approximate center of the contact surface where the pretension node will be created on is automatically estimated.
- Choose **Auto-Generate**, then select the **Auto generate pretension node** checkbox. A pretension node will be automatically generated in the center of the contact surface.

8. Select whether to apply a **Force** or **Displacement** load on the bolt, then enter a corresponding magnitude.

A load is not created until a load collector is specified. The load acts in the direction normal to the contact surface. A load (force and displacement) is always created for bolts in the X-direction (COMP1), regardless of the model.

9. Click **Function Time**.

10. In the panel area, use the curves selector to select the required curve for the load and click **proceed**.
    The selected curve and its corresponding ID are displayed in the field to the right of the Function Time definition button.
11. Click **Load Collector** and select where to store the defined load.
    - Choose **Select Existing Collector** to open the panel area where you can use the loadcol selector to select an existing load collector.
    - Choose **Create New Collector** to create a new load collector with the prefix PRETENS_#. The number appended to the end of the load collector name depends on the number of load collectors currently present in the model.

    The load collector is created along the 3D bolt.
12. To create a FIX parameter, select the **FIX parameter** checkbox and enter a FIX value.
13. Select where the different codes will be written internally in the solver deck.
    - **Total axial force in the bolt (9524)**
    - **Relative displacement in the cut (9530)**
14. Click **Create**.

A BOLT is created as a group entity.

The macro element ID of the first bolt that was created is written out as the maximum element ID in the model + the number of nodes present in model. 1000 is added to the macro element ID for consecutive bolts, that is, if the first bolt's macro ID is 117653, then the next bolt will be 118653 (117653 + 1000).

The macro element ID of a bolt is reflected in its .SAI command, which is written along with the bolt's corresponding output-code. It is preceded by the comment !!HM_TEMP_OUTPUTBLOCK, which enables the model-reader to ignore the output block upon import and not create any output blocks in HyperMesh. A bolt card that does not contain an output definition will not have a .SAI command written out.

All such commented commands are automatically exported out again upon re-export in the same format.

The load applied on a bolt is written out in the solver deck. A load's magnitude is written out after VAL, and any curve attached to the load using Function Time is written after the keyword NF.

Manage Sets in the Sets Browser

Automate the grouping and display of model components through the entity set functionality using the Sets Browser.

Open the Set Browser by selecting **Tools** > **Set Browser** from the menu bar.

The Sets Browser consists of a tree structure listing the current entity sets in the model, along with the entity set display and export states. It also includes functions for displaying, creating, deleting, renaming, appending entities to, and changing the export state of entity sets.

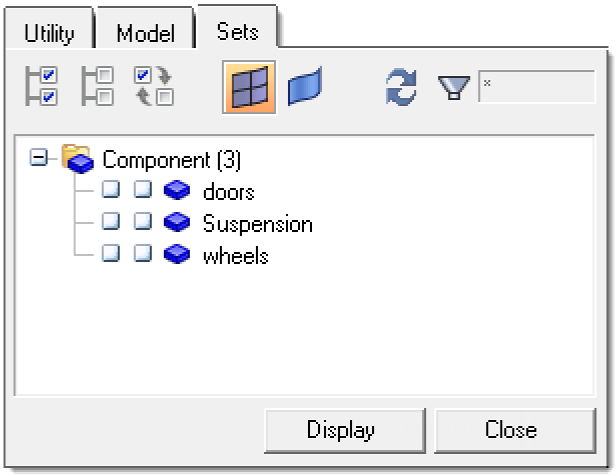


Figure 1708:

Synch Set Browser with Graphics Display

The Sets Browser is meant to allow you to easily control the display and review of entity sets for model grouping and visualization purposes. For large models, constantly synchronizing the display state of entity sets with the current display can introduce performance issues.

To remedy these occurrences, the Sets Browser utility does not automatically synchronize the display states of entity sets with the current display.

Synch the Sets Browser with the graphics display.

- Click the **Display** button at the bottom of the Sets Browser to update the display to match the Sets Browser settings. When the Display button is used to update the display to the current Sets Browser selection, the Sets Browser and the display remain synchronized until another selection is made within the Sets Browser.
- Click the **Synch** button in the Sets Browser toolbar to update the Sets Browser to match the current state of the display.

Set Display Options for Sets Browser

1. Open the Sets Browser.
   The Sets Browser displays in the tab area.
2. Use the toolbar buttons in the Sets Browser tab to manipulate the display options, as desired.

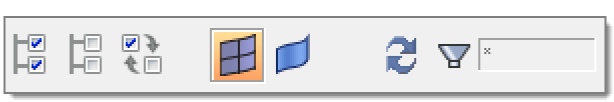
   

   Figure 1709:

   - Use Ctrl-click and Shift-click to select multiple items in the tree structure. For large numbers of selections, use the select all ( ), select none ( ), reverse ( ) buttons.

- The Entity Type options allow you to control the entities (elements or geometry) that the selection buttons apply to. These buttons are toggles; you can have one or both active at the same time. In the screenshot above, the Element () button is active, but the Geometry () one is not.
- The Sync ( ) button synchronizes the entity set display states with the current display.

  This means that if you have changed the display states of various entities, for example from within the Model Browser, you can use this button to force the selection of entities in the Sets Browser to match the current display states.
- The Name filter uses standard filtering syntax in the text box; click the funnel () icon to activate the text box, type in the string you wish to filter by, and then press Enter. This can limit the entities which display in the tree structure. To undo the filter, click the funnel icon again to disable the text box.

Change Sets Browser Display and Export States

1. Open the Sets Browser.
   The Sets Browser displays in the tab area. Its tree structure lists all entity set currently existing in the model, grouped in folders by type.
2. The display states of entity sets are controlled by clicking the checkboxes located next to each set on or off.

Option	Description
	The checked state signifies that all entities in the entity set are currently displayed, after clicking the Display button.
	The blank state signifies that one or more of the entities in that entity set are not displayed, after clicking either the Display button.

3. Once the display checkboxes are changed, click the **Display** button at the bottom of the browser to update the display with the current selection.

Use the Sets Browser Right-Click Functionality

1. Open the Sets Browser.
   The Sets Browser displays in the tab area. Its tree structure lists all entity set currently existing in the model, grouped in folders by type.
2. Right-click anywhere within the tree structure to open the right-click menu.

   There are many functions available, accessed by right-clicking in the background, on folders, or on individual or multiple items within folders. Most options require that you click on a folder or one or more items, and are grayed out of no selection is made; exceptions are specifically noted below. The graphic above shows the available options, including:

Option	Description
Create	Create a new entity set of the specified type. You are prompted to type in a name for the set or accept a default name. Supported entity set types are shown above. This option does not require any existing sets to be selected.
Edit	Edit the element set, by picking a different group of elements to assign to it.
Delete	Deletes the currently selected set(s). Multiple sets may be selected by using standard Ctrl/Shift-click functionality.
Card edit	Edit the property card assigned to the set (but not the entities within it).
Rename	Rename the selected set.
Delete Reference	Removes a set reference from an entity set type of sets.
Add Entities from Set	Adds entities into the currently selected set. This operation brings up an entity selector to select entities to add to the set.
Remove Entities from Set	Removes entities from the currently selected set. This operation brings up an entity selector to select entities to remove from the set.
Show	This operation adds the entities contained in the selected set(s) to the display.
Hide	This operation removes the entities contained in the selected set(s) from the display.
Isolate	This operation turns off (masks) the display of all entities not currently selected, so that only the selected entities display.
Collapse All	Collapses all branches (folders) of the tree. This option does not require any selection.
Expand All	Expands all branches (folders) of the tree. This option does not require any selection.
Display Options	Determines how the sets are labeled in the Sets Browser tree. Available options are shown above. This option does not require any selection.
Display IDs	Displays a popup window showing the IDs of all entities contained in the selected set.
Export Session File	Saves a session file (`.ses`), containing group definitions for the selected node or element sets, to the disk.

Option	Description
Import Session File	Loads a session file (`.ses`) containing group definitions. These group definitions will be converted into entity sets. This option does not require any selection.

Find Connectivity

Check the connectivity between two or more components.

Certain types of connections can be included or excluded while checking for connectivity.

> **Restriction:** The Find Connectivity tool is only available in the LS-DYNA and Radioss user profiles.

1. From the menu bar, click **Tools** > **Find Connectivity**.
   The Find Connectivity Browser opens.
2. Invoke the Find Connectivity controller by right-clicking in the browser and selecting **Find Connectivity** from the context menu.

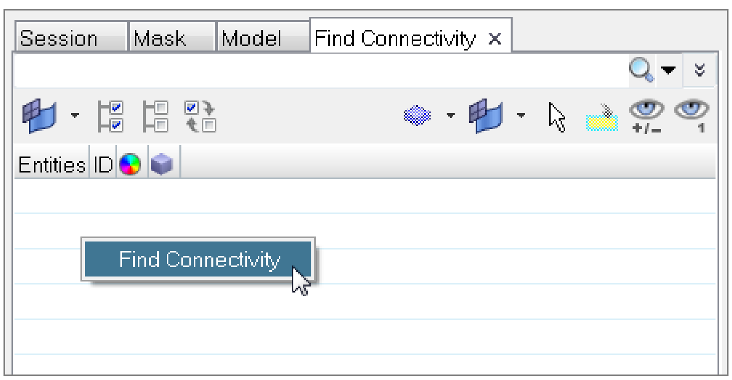


Figure 1710:

3. In the Find Connectivity controller, enable the type(s) of connectivity to include in the check.
   By default, all types of connectivity are enabled by except Connectors.

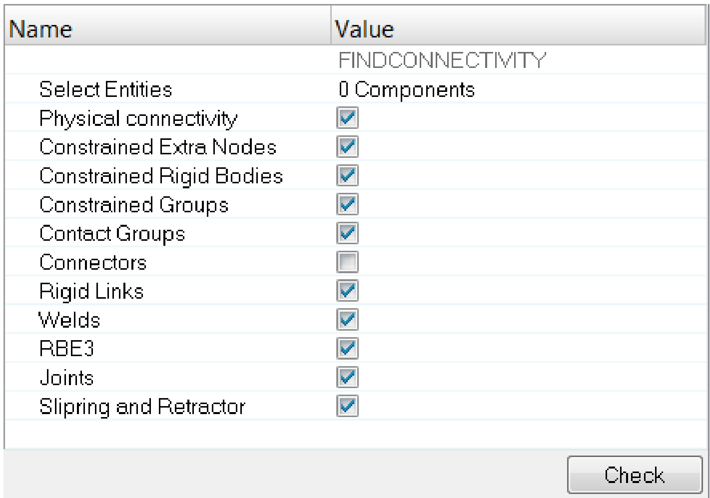


Figure 1711:

4. Use the Select Entities selector to select two or more components to check the connectivity between.

   Connectivity check works on all the selected components regardless of its display state. Connectivity due to mass elements are completely excluded when checking connectivity.

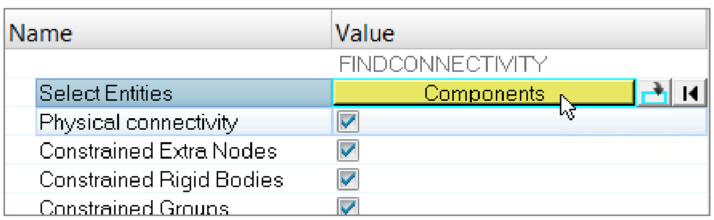


Figure 1712:

5. Click **Check**.

   If two or more parts are connected, the number of connected parts are displayed in the browser. Expand the folders to see individual components. If the parts are free, the free part is displayed separately.

Results are displayed in the browser.

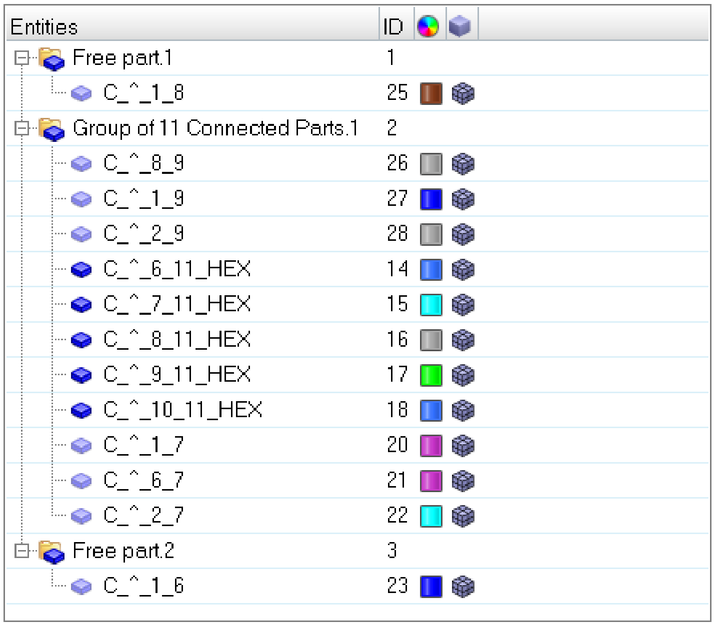


Figure 1713:

Connectivity Types

Radioss

Physical connectivity
Checks for node to node connectivity (for 1D, 2D and 3D).

RBE2
Checks for RBE2 connectivity.

Rigids
Checks for RBODY connectivity.

Interface2
Checks for contacts with INTER/TYPE2.

Connectors
Checks for FE realized connections with connectors.

Rigid Links
Checks for RLINK connectivity.

RBE3
Checks for RBE32 connectivity.

Joints

Checks for CYLJOINT connectivity.

LS-DYNA

Physical connectivity

Checks for node to node connectivity (for 1D, 2D and 3D).

Constrained Extra Nodes

Checks for connections using constrained extra nodes and extra noded joints.

*CONSTRAINED_EXTRA_NODES

Constrained Rigid Bodies

Checks for connections using constrained rigid bodies.

*CONSTRAINED_RIGID_BODIES

Constrained Groups

*CONSTRAINED_TIE-BREAK

*CONSTRAINED_TIED_NODES_FAILURE

Contact Groups

*CONTACT_SPOTWELD

NodesToSurface (only following types)

*CONTACT_TIED_NODES_TO_SURFACE

*CONTACT_TIEBREAK_NODES_TO_SURFACE

*CONTACT_TIED_SHELL_EDGE_TO_SURFACE

SurfaceToSurface (only following types)

*CONTACT_TIEBREAK_SURFACE_TO_SURFACE

*CONTACT_TIED_SURFACE_TO_SURFACE

SlidingOnly

*CONTACT_SLIDING_ONLY

Connectors

Checks for FE realized connections with connectors.

Rigid Links

*CONSTRAINED_NODAL_RIGID_BODY

*CONSTRAINED_NODE_SET_ID

*CONSTRAINED_GENERALIZED_WELD

*CONSTRAINED_SHELL_TO_SOLID

Welds

*CONSTRAINED_SPOTWELD

RBE3

*CONSTRAINED_INTERPOLATION

Joints

Checks for connections with Joints

Slipring and Retractor

Checks for connections with sliprings and Retractor

*ELEMENT_SEATBELT_SLIPRING

*ELEMENT_SEATBELT_RETRACTOR

Review Mass Summary

Review details about Mass, Center of Gravity (Cog) and Inertia of your model.

1. From the menu bar, click **Tools** > **Mass Details** and select one of the following:
    - Choose **Mass only** to calculate only the mass of the model.
    - Choose **Mass, Cog and Inertia** to calculate the mass, center of gravity, and inertias.

    Calculations are performed, and the **Mass summary** dialog opens.

    Individual entity details can be reviewed in a tabular format from the browser area, and global values of the whole model for Mass, Center of Gravity (Cog) and Inertia can be reviewed in the lower part of the dialog.
2. Optional: Export the results to a CSV or HTML file by right-clicking in the browser and selecting **Export** > **<file format>** from the context menu.

> **Tip:**
> - Manage the columns that are visible in the browser by right-clicking on a column header and selecting the columns to display.
> - Turn the display of entities ON/OFF in the browser and graphics window using the **Show**, **Hide**, **Isolate**, and **Reverse** options available in the right-click context menu.
> - Review only the selected entities, and make all other entities transparent using the **Review** option available in the right-click context menu.

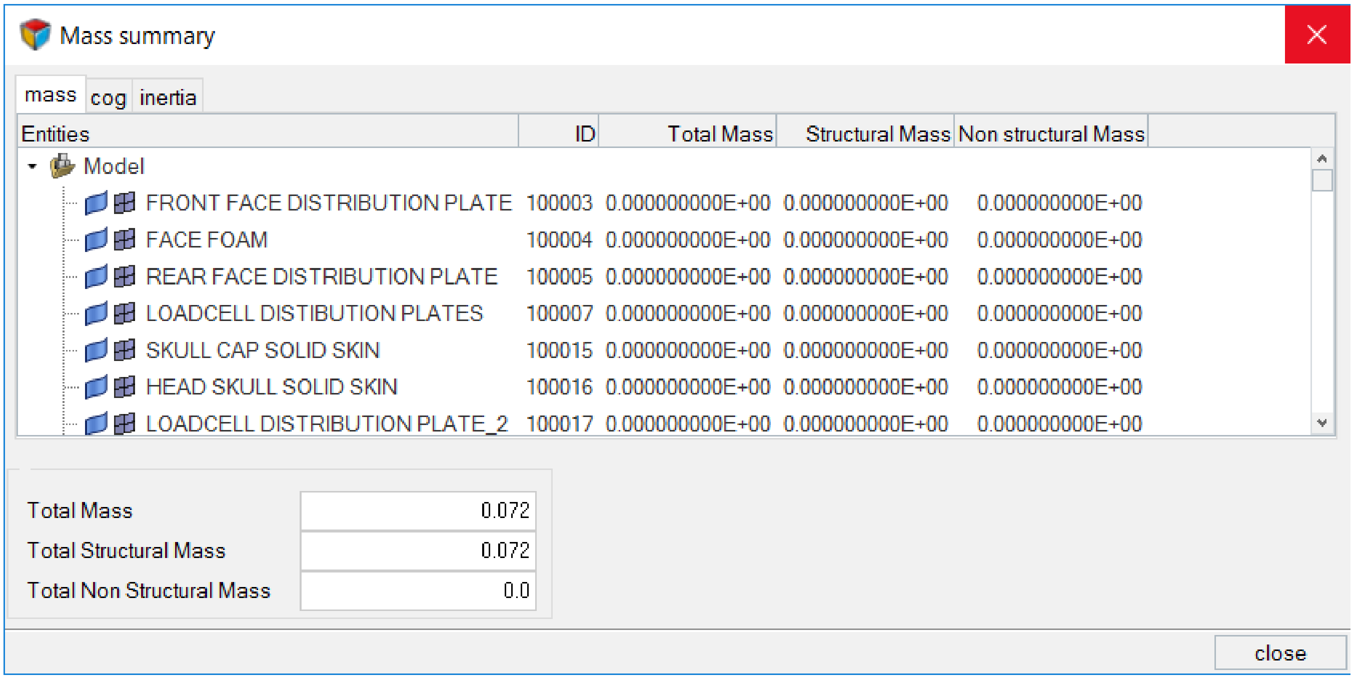


Figure 1714: Summary of Mass

Supported Mass Values

Mass values supported in the **Mass Summary** dialog.

Structural Mass

Mass of the component defined by the mesh volume and material density.

Non Structural Mass

Mass added on the component, provided by following keywords:

LS-DYNA

*ELEMENT_MASS_PART
*ELEMENT_MASS_PART_SET
MAREA defined in *SECTION_SHELL
NSM defined in *SECTION_BEAM

Radioss

/ADMAS defined on PART

Lumped Mass

Nodal distributed mass provided by following keywords:

LS-DYNA

*ELEMENT_MASS
*ELEMENT_MASS_NODE_SET
*ELEMENT_INERTIA
*ELEMENT_SEATBELT_ACCELEROMETER

Radioss
/ADMAS defined on NODES

Mass in CNRB
Constrained_Nodal_Rigid_Bodies

LS-DYNA
Total mass transferred to the *CONSTRAINED_NODAL_RIGID_BODYs in the model.

Mass on RBODY

Radioss
Total mass transferred to the RBODYs in the model.

Transferred Mass
For each component, this is the nodal massed transferred to the rigid bodies.

LS-DYNA
Deformable mass transferred to CONSTRAINED_NODAL_RIGID_BODY or to the rigid parts of a *CONSTRAINEED_EXTRA_NODES or to the rigid parts of *CONSTRAINED_RIGID_BODIES

Radioss
Deformable mass transferred to RBODY

Distributed Mass

LS-DYNA
Mass in rigid components coming from free nodes linked by *CONSTRAINED_EXTRA_NODES

Engineering Mass

LS-DYNA
Engineering Mass = Structural Mass + Non structural Mass + Lumped Mass + Distributed Mass

Radioss
Engineering Mass =Structural Mass + Non structural Mass + Lumped Mass

Solver Part Mass
Solver Part Mass = Engineering Mass + Transferred Mass

Total Mass
Total Mass = Solver Part Mass

Manage Rigid Bodies

Manage and review all rigid bodies in the model using the RBODY Manager.

Access the RBODY Manager by selecting **Tools** > **Rbody Manager** from the menu bar.

> **Restriction:** Only available in the Radioss user profile.

The RBODY Manager provides the following features in one convenient tab:

- Display all rigid bodies in the model
- Display individual rigid bodies
- Create new, and edit existing, simple and complex rigid body formulations
- View and update details of individual rigid bodies, though the card editor and the Rigid panel

Existing rigid bodies are shown in the table. For each rigid body, the display status, ID number, name, master node ID, and type is shown.

Table 267:

Column	Description
Disp	Indicates whether the rigid body is displayed in the graphics area.
ID	The ID number of the rigid body.
Title	The descriptive name of the rigid body.
Master Node	The ID of the node that serves as the master node of the rigid body.
Type	S or C. S indicates a simple rigid body, which is a typical spider formulation. C indicates a complex formulation, such as an RBODY that points to a part or a set of sets.

Highlight individual entries or groups of entries to perform an action on the rigid body. Actions are available from the context menu (by right-clicking over the table entries) or the toolbar buttons. These actions are described below:

Table 268:

Icon	Name	Description
	Review Option	Customize the way the selected rigid bodies are displayed. Options include transparency and auto-review selections.
	Review	Highlights the nodes to which the selected RBODY is attached. The master node is shown in blue and the slave nodes are shown in red.
	Find Attached	Highlights the elements that are attached to the selected rigid body.
	Edit	Modify the definition of the rigid body through the Rigid panel.
	Card Edit	Opens the RBODY card in the **Card Editor**.
	Delete	Deletes the selected rigid body.

Icon	Name	Description
	Refresh	Update the table of rigid bodies.

New rigid bodies can be created with the **RBODY Manager**. The following fields are available at the bottom of the **RBODY Manager** tab, which enable you to supply all the basic data needed to create a new RBODY. Nodes, parts, materials, properties, and GRNODs can be used to define the slave nodes. Once the RBODY is created, click **refresh** to list it in the table. Then you can select the RBODY to edit the card image, display the RBODY, and so on.

Note:

- When a large number of slave nodes are attached to a master node, the connecting lines are not displayed in the graphical model.
- The table of rigid bodies can be sorted by the ID, title, mater node, and type columns.
- Select **Show Details** from the context menu to display a summary of details about the rigid body including the ID, name, master node ID, and number of slave nodes.
- Select **Editable** from the context menu to make the title column editable. When the Title column is editable you can modify the names of the rigid bodies.

The tool is also available in the PAM-CRASH 2G user profile and offers similar features.

Analyze Part Information

Analyze a summary of a part's statistics using the Part Info tool.

Restriction: Only available in the Radioss user profile.

1. From the menu bar, select **Tools** > **Part Info**.
2. Click **component** on the main menu area to select a component or click a component in the graphics area to select it.
3. Click **proceed**.
   The **Part Information** dialog opens, which lists the part ID, name, thickness, and material type.
4. To view additional statistics about the part, click **More Detail**.
5. To display statistics for a different part, select the part in the graphics area or the components selector and click **proceed** again.

   Tip: Click the middle mouse button instead of the proceed button to quickly select components.

Relative Displacement

Create the /TH/SPRING card, which supports time histories with spring output using the Relative Displacement tool.

Access the **Relative Displacement** tool by selecting **Tools** > **Relative Displacement** from the menu bar. The tool opens as a tab named Relative Displacement.

Restriction: Only available in the Radioss user profile.

Create Time Histories of Springs

1. Type a name in the Time History Name field and click **Create/Edit**.
   A pop-up menu appears from which you can choose either to create the spring and edit the card that is generated or simply create the spring.
2. The time history appears in the Output Blocks list. Right-click the list item and select **Add Springs** to create springs for that time history.
3. Pick nodes from the model to define springs.

   Note: The first node you select is used as Node1 in the definition of the spring.

Edit Time Histories of Springs

1. Right-click its name in the Output Blocks list and select **Edit**.
   The card for that time history opens in the main panel area. From here you can provide names for each element, and add or modify variables.
2. To add variables, click **NUM_VA**.
   A pop-up window of numbers opens.
3. Click the number of variables you want to include in the card.

   Var fields appear where you can type the variable names.

   When you use the Relative Displacement tool to create time histories of springs, you also create a component named Comp_Rel_Disp. This component is of the Springs/Rivets type and no material is assigned. The component contains one property, SPR_GENE, with only the mass value specified.

Manage Boundary Conditions

Create boundary conditions of any load collector type other than ACTIV using the BCs Manager tool.

Access the **BCs Manager** by selecting **Tools** > **BCs Manager** from the menu bar. The tool appears as a tab in the tab area.

Restriction: Only available in the Radioss user profile.

This tool combines required actions from several panels into one convenient tab interface, including the ability to create a boundary condition from both sets and individual nodes.

Create New Boundary Conditions

1. Select the type of boundary condition you want to create in the Select type field.
2. In the Name field, type a name for the load collector.
3. Select the type from the drop-down menu in the Select type field.

   Based on the type selected, the options that display may change.
4. Use the selectors to pick sets and/or individual nodes to define boundary conditions. The switch allows you change the selector to access parts, nodes, materials, properties, GRNOD (set), and GRNOD (box). Pick the type of entity to be selected from the pull down menu and click the yellow button to open the Selector panel.
5. Enter the loading conditions in the options.

   Empty fields require user input, and yellow buttons provide links to another entity that needs to be linked; namely curve, system or sensor. Each yellow tab has two options:
   - Choose **Create/select** to directly create the entity from this GUI.
   - Choose **Select** to select an entity already defined.
6. Click **Create** to create the database with the boundary condition data entered. Click **Cancel** to cancel the creation and **Close** to close the dialog.

Update Existing Boundary Conditions

1. Highlight a boundary condition in the list to display its properties.
   This opens the Review (editing) mode of the dialog.
2. In Review mode, every defined entity/entry in the field can replaced with new entity or modified. To replace the entity on which boundary condition is define, set the GRNOD tab to the desired entity to be selected and make the selection.
   The selection will replace the existing entity on which the BC is defined. Similarly the curve, system and sensor can be changed.
3. The table appears with list of entities that are referred in the selected boundary condition. To edit the existing entity, select the entity, right-click it and select **edit**.
   This opens a corresponding panel with editing features.
   - Click **Update** to update the changes made to the selected boundary condition. Click **return** to go back to Create mode.
   - Click **Cancel** to nullify all updates made to the selected boundary condition. Click **return** to go back to Create mode.
   - Click **Close** to nullify all updates to the selected boundary condition and close the dialog.

- Click **Review** to highlight the entities on which the selected boundary condition is defined.

Table Functions

The table in the dialog lists the boundary conditions in the model by default.

This can be limited to the desired boundary condition using the **Select type** option on the top of the table.

Right-clicking on each entity in the table provides the following functions:

Table 269:

Function	Description
Refresh list	Refreshes the table based on the option in the Select type field.
Card edit	Opens the selected boundary condition's card image panel.
Delete	Deletes the selected boundary condition.
Review	Highlights the entities on which the selected boundary condition is defined and grays out the others.
Clear review	Returns the graphics window to regular mode.
Show all	No pre-selection is needed. Displays the part on which the boundary conditions in the table are defined and shows the load with handles.
Show	Displays the part on which the selected boundary condition is defined, if hidden, and shows the load with handle.
Isolate	Isolates the part on which the selected boundary condition is defined in the graphics and shows the load with handle.

Control Cards

Control cards allow you to add input and output parameters to a model, including location and names of the input, output and scratch files; the type of run (analysis, check or restart); overall running of the analysis or optimization; and type, format and frequency of the output.

Control cards are assigned to your model from within the Control Cards panel. This panel lists all of the control cards defined for the solver/user profile that you currently have loaded; you can disable, enable, or delete cards as desired.

Use the Card Previewer

A control card may be in one of three states:

State	Color	Explanation
Undefined	Gray	The control card was either never created or has been deleted.
Defined (See note.)	Green	Any control card viewed in the card previewer is activated.
Inactive	Red	A card that has been defined may be disabled. The attributes for that card remain; however, the control card is not output.

Note: Those control cards that are defined (green in the control card editor) are output.

Default values for attributes are common throughout the card previewer. A default value field has two states:

State	Description
Default = ON	In this state, the field label color is yellow and no data entry is allowed.
Default = OVERRIDDEN	To override a default value field, pick the yellow field label. When you override a default value field, the label text color changes to cyan and allows you to enter data in the field.

Boundary Conditions

Boundary Conditions define limits as well as loads on geometry and mesh.

Loads on Geometry

You can apply loads to geometrical entities and map them to the FE mesh using the Load on Geom panel.

One advantage is that you can remesh a model without deleting complicated loads or boundary conditions. After remeshing, loads or boundary conditions that have been applied to geometrical entities can be remapped to the new mesh.

You can apply loads to geometry by using the following panels: Forces, Moments, Constraints, Pressures, Temperatures, Flux, Velocities, and Accels. These are the same panels used to apply loads to a mesh.

There are two ways to map loads on geometry to the mesh associated with this geometry (loads on mesh):

- Manually, using the Load on Geom panel
- Automatically, by exporting the FE deck, using the Export tab.

The Model Browser allows separate or simultaneous visualization of loads on mesh and loads on geometry.

To visualize loads on mesh and/or loads on geometry, right-click on the load in the Model Browser.

From the toolbar, click (Elements/Geometry). This icon determines what the other buttons act on; right-click the button (or left-click the small triangular downward arrow) to reveal a drop-down menu of options. You can select **Elements**, **Geometry**, or **both**.

- When Elements is selected, you control the display of loads applied to elements.
- When Geometry is selected, you control the display of loads applied to geometric entities.
- Both means that you can control the display of both types of loads independently, and load collectors may contain one type or both types simultaneously.
- Use the **none**, **all** and **reverse** buttons to assist in selecting which loadcols should be displayed.

Note:

Loads on mesh and loads on geometry can be displayed together, similar to the simultaneous display of both elements and geometry belonging to a specific component.

A geometrical entity can be associated with one mesh or multiple meshes (component or components) and/or with one load collector or multiple load collectors.

One load collector stores both loads on geometry and loads on mesh. The mesh, or multiple meshes, is associated with the geometrical entities to which the loads on geometry have been applied. Each load type is stored in a dedicated section of the same load collector.

Terminology and Definitions

geometrical entities

A point, a line, or a surface.

loads on geometry or geometry loads

Loads applied to geometrical entities.

loads on mesh or mesh loads

Loads applied to mesh (nodes or element).

Loads can be applied directly to mesh or applied by mapping them from loads on geometry.

load mapping

The process of mapping geometrical loads to mesh loads. The loads are mapped from the geometrical entities (to which the geometrical loads are applied) to the mesh that is associated with the geometrical entities.

Application of Loads to Geometry

You can apply loads to geometrical entities in a way similar to the manner in which loads are applied to mesh. The process includes two basic steps.

1. Creating a load collector by using the Collector panel.
2. Applying loads to the geometry using one of the following panels on the Analysis page: Forces, Moments, Constraints, Pressures, Temperatures, Flux, Velocities, and Accels.

   To apply a load to a geometrical entity, first create a load collector in which the loads applied to geometrical entities will be stored. Next, access an HyperMesh load panel (such as Forces, Constraints) located on the Analysis page, and choose the create subpanel. Third, select a geometrical entity on which the loads will be applied (points, lines, or surfaces) using the panel selection box, define the load or boundary condition parameters in the same way you would for the application of the load or boundary condition on a FE mesh entity, such as a node, and click **create**. HyperMesh stores the loads/boundary conditions in the database and displays them in the modeling window.

   The following chart specifies the geometrical entities to which loads can be applied, in each of the load application panels listed above.

Panel	Geometrical Entities
Accels Panel	points, lines and surfaces
Constraints Panel	points, lines and surfaces
Flux Panel	points
Forces Panel	points

Panel	Geometrical Entities
Moments Panel	points
Pressures Panel	surfaces nodes on edge: lines (for 2D solid elements) nodes on face: surfaces (for 3D solid elements)
Temperatures Panel	points, lines and surfaces.
Velocities Panel	points, lines and surfaces.

Refer to the specific panel for detailed information about creating, reviewing and updating loads and constraints.

Exporting Loads

Export sessions that contain loads on geometry, loads on mesh that have been applied directly to mesh, and loads on mesh that have been mapped from loads on geometry.

When saving the model as an HM database, all load types are saved and are retrieved when you open the `.hm` file. When exporting the model using an export template, only the loads on mesh are exported. The loads on mesh that are exported may have been applied directly to mesh, mapped from geometry to mesh, or both.

The **all/displayed** option on the Export tab allows you to determine which loads are exported.

If all is selected, all the loads on geometry that have not been mapped (if any), are mapped to loads on mesh and all the loads on mesh are exported.

If displayed is selected, all the displayed loads on mesh are exported. All the loads on mesh, both displayed and hidden, that are associated with the displayed loads on geometry are exported as well. If any loads on geometry are displayed and have not been mapped, they will automatically be mapped to loads on mesh and exported as well.

Visualization of Loads on Geometry and Loads on Mesh

The Display panel allows you to visualize loads on mesh and loads on geometry.

Visualize loads on mesh and loads on geometry either individually or together by setting the collector type to **loadcols** and using the toggle between **elems** and **geoms**.

The elems option controls the display of loads on mesh while geom controls the display of loads on geometry. A simultaneous display is similar to the display of both elements and geometry belonging to a specific component.

Note: A major graphical display difference between loads on geometry and loads on mesh is the density of the arrows. Multiple arrows represent loads on mesh (one arrow per node or element); a single arrow for each geometrical entity represents loads on geometry. The basic length of the arrow also differs. For the same arrow magnitude percentage setting or uniform size setting within the load application panels, an arrow that represents a load on geometry is longer than arrows representing loads on mesh.

Create Load Collectors

1. In the Model Browser, right-click on the white area and select **Create** > **Load Collectors**.
2. In name field, enter a load collector name.
3. Click color and select a color from the pop-up menu.
4. If creating a generic load collector:
   a) Click the switch and select **no card image**.
   b) Click **create**.
   c) Click **return**
5. If creating a specific load collector:
   a) Click the switch and select **card image**.
   b) Click card image = and select the card image type.
   c) Click **create/edit**.
   d) Enter the relevant data in the card image.
   e) Click **return**.

Morphing 13

Morph the shape of your finite element model.

This chapter covers the following:

Using morphing tools in HyperMesh you can rapidly change the shape on the FE mesh without severely sacrificing the mesh quality.

During the morphing process, you can create shapes which can be used for subsequent design optimization studies.

Approaches to Morphing

HyperMorph utilizes exclusive HyperMesh morphing entities; domains, handles, morph constraints, morph volumes, shapes, and symmetries.

While all the entities and functions are fully compatible, and may be used in a complementary fashion, they can be divided into three approaches to morphing; the domains and handles concept, the morph volume concept, and the freehand concept.

Domains and Handles Approach

The Domains and Handles approach involves dividing the mesh into domains containing elements or nodes and placing handles at the corners of those domains.

The domains and handles approach also allows for parametric morphing of lengths, angles, radii, and arc angles as well as morphing the mesh to match geometric data and other meshes. This approach is most useful for making detailed changes to any mesh (local domains) as well as general changes to space frame type meshes (global domains).

You can automatically divide the mesh into logical domains or you can manually define your own domains and handles. When the handles are moved, the shape of the mesh changes according to the domain boundaries. When the handles associated with a domain move, the shape of the domain changes, which in turn changes the positions of the nodes inside those domains. During the morphing process the mesh morphs in a logical way with nodes near the moving handles moving more and nodes near the stationary handles moving less. In the areas between the handles, the mesh is stretched or compressed to match the desired shape.

The amount each node moves with respect to each handle is relative to an internally calculated influence coefficient. The process for calculating the influence coefficients is somewhat time consuming, but once they are calculated they can be stored and applied rapidly. Thus, when handles and domains are initially set up or edited, HyperMorph spends an amount of time, proportional to the size of the new domains, calculating the handle influences. However, when handles are moved to morph the model, no calculations are necessary and the actual morphing occurs quickly. The advantage of this approach is that it makes morphing an interactive process, even for large models.

For very large domains, calculating influence coefficients can be time consuming. For domains that have more than 50,000 elements, although you can change this default limit, the large domain solver is used. The large domain solver much faster at morphing large domains, but the drawback is that it must be invoked every time you wish to morph, thus making morphing slower. However, for very large domains, the process of calculating influences can be too slow or too memory intensive, therefore the large domain solver makes it possible to morph such domains.

Domains and handles are divided into two basic groups, global domains and local domains.

Global Domains and Handles

Each global domain is associated with any number of global handles. Global handles will only influence the nodes contained within their associated global domains. Global domains and handles are best for making large scale shape changes to the model.

Global domains are represented by a cube made up of dashed lines, and located at the centroid of the nodes which make up the global domain.

Global handles are the largest handles in the model. Handles are colored red if they are not dependent on other handles, and they are colored yellow, cyan, or violet if they are dependent on other handles. The handle color indicates their level of dependency. Dependent global handles are also smaller than the handles on which they are dependent. The base size of all the handles in the model can be set on the morphing Visualization Controls tab accessed by using the Visualization options ( ) on the Visualization toolbar. The size given is used as the radius for the independent global handles. You cannot edit the color of the handles nor the relative size between the dependent and independent handles. However, you can edit the color of the domains in the morphing Visualization Controls tab.

The Domains panel is used to create, edit, and organize global domains. When a global domain is created with the **create handles** option selected, HyperMorph generates several global handles. Global handles are generated at each of the eight corners of a box surrounding the model laid out along the global axes. These global handles are named corner followed by a number from one to eight. HyperMorph also places at least one global handle within the global domain box in areas of peak nodal density within the model. HyperMorph generally creates no more than about 30 global handles within the global domain box. These handles are named global followed by a number. The automatic global handle generation works well for space frame models such as full car models. If the handles are not generated in the positions where you want them to be, you can always delete them, reposition them, or create new handles using the Handles panel.

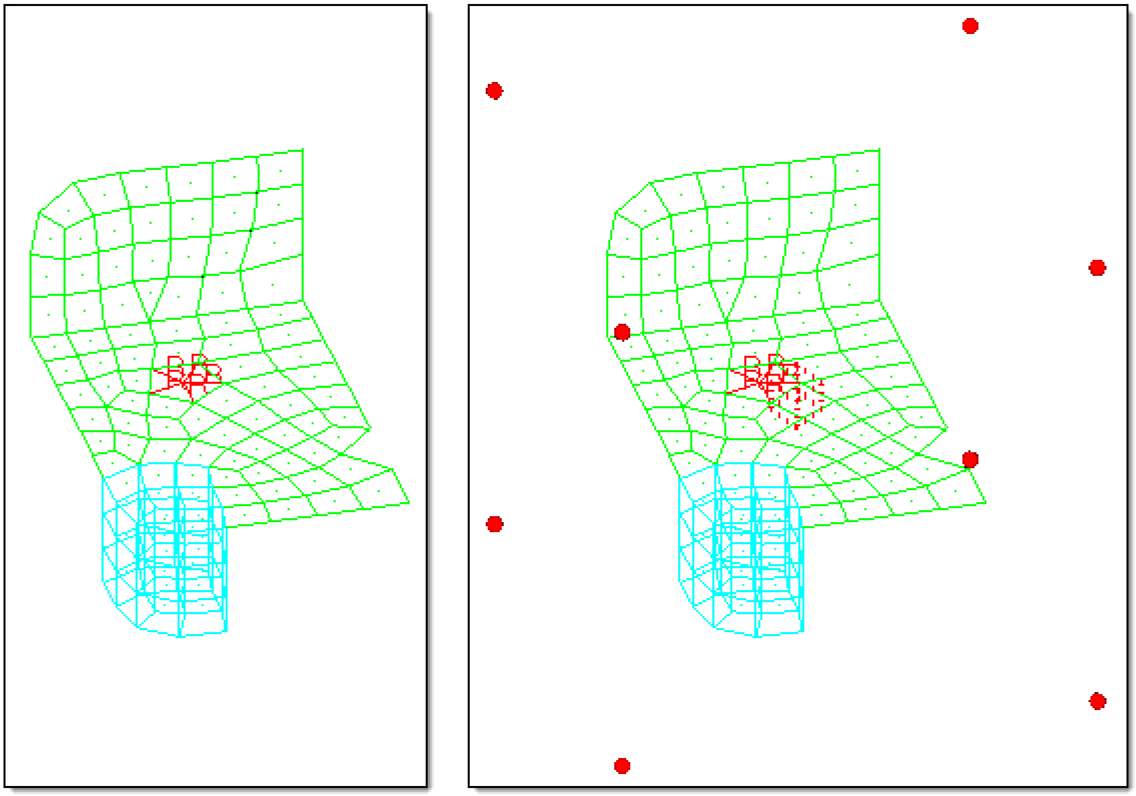


Figure 1715: Global Domain and Global Handles

Eight handles are placed at the corners of a box enclosing the model. By moving the handles you can stretch or deform the model along all three axes.

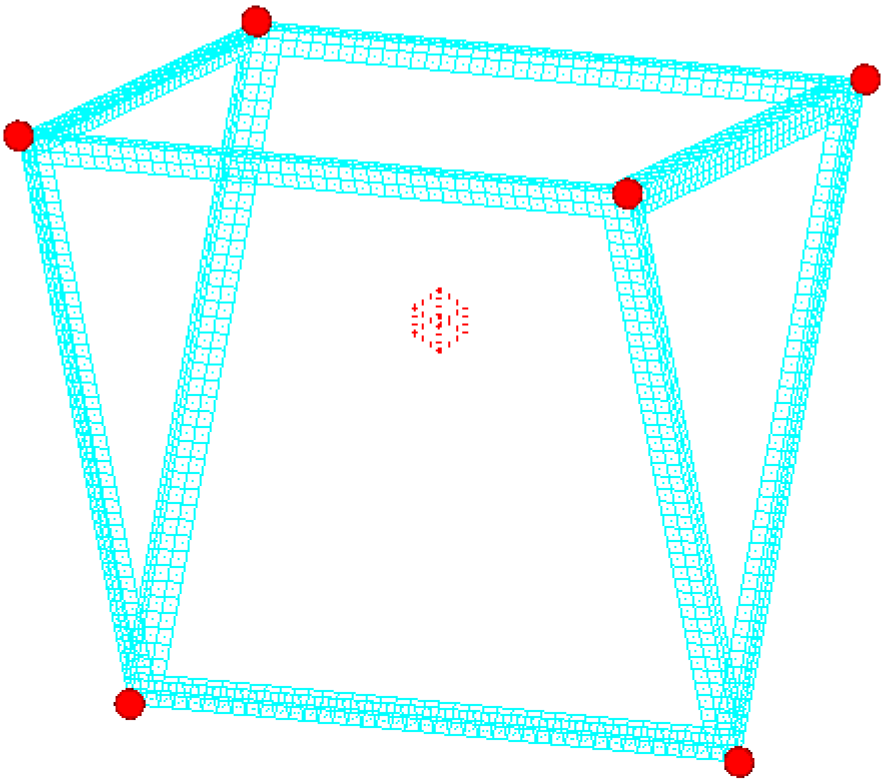


Figure 1716: Space Frame with Six Manually Created Global Handles

When the handles are moved, the space frame morphs in a way such that the bars run between the handles.

The method for determining how global handles associated with global domains influence the mesh can be selected in the Morph Options panel, Global subpanel.

Hierarchical

Global handles influence the local handles found at nodes inside the global domain, which in turn influence nodes within the local domains.

Direct (default)

Global handles influence the nodes in the model directly even if the nodes are not in a local domain.

Mixed

Global handles will influence every node inside the global domain using the hierarchical method if the node is inside a local domain, or the direct method if the node is not in a local domain.

There are subtle differences in how the global handles influence the nodes for each method with the main difference being that the parts of the model defined by local edge domains have their shape preserved when using the hierarchical method. Straight edges will remain straight and circular holes will remain circular for the hierarchical method, while the direct method may bend or warp these features into curved edges and elliptical holes. You should select which method is right for the type of morphing that you want to perform. If you wish to preserve the local geometry, choose the hierarchical or mixed method. If you are willing to accept distortions in the local geometry, choose the direct method.

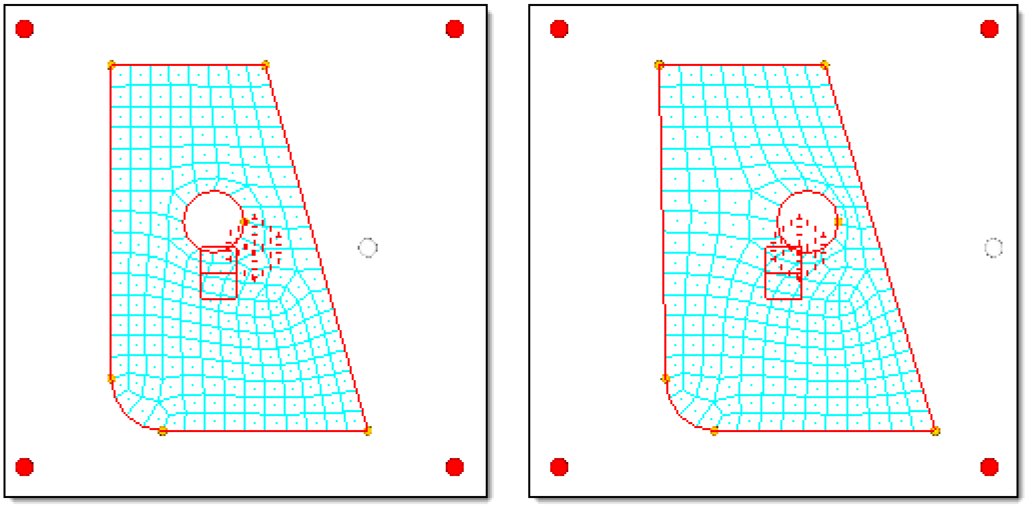


Figure 1717: Global Morphing using the Hierarchical Method

When the highlighted (white) handle is moved to the right, it moves the local handles, which move the mesh. Note how the straight edge remains straight and the circle remains round.

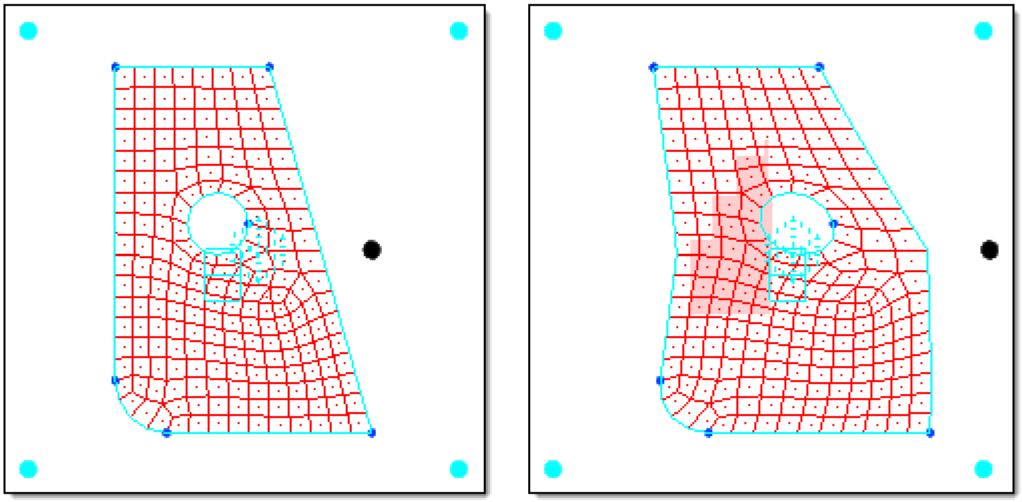


Figure 1718: Global Morphing using the Direct Method

When the highlighted (white) handle is moved to the right, the mesh is affected directly. Note the resulting distortion of the edge and circle.

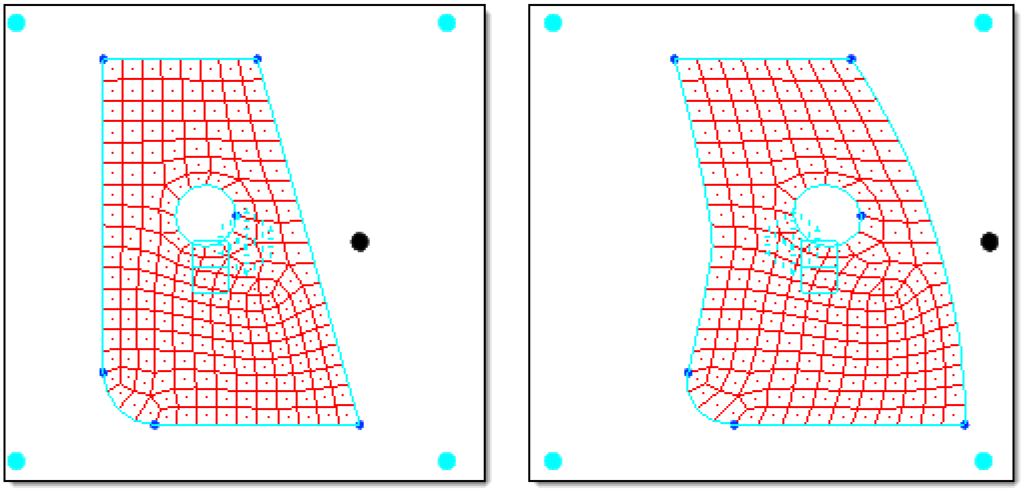


Figure 1719: Global Morphing using the Direct Method and Biasing Factors

By increasing the biasing factor for the highlighted (white) handle, the angular shape of the morph becomes rounded.

The influences between the global handles and local handles using the hierarchical method or nodes using the direct method can be calculated using either the spatial method or the geometric method. Both methods attempt to determine how a global handle affects nodes or local handles in the space surrounding it. The spatial method is the default, and is the fastest and most robust method for generating global influences based on a spatial formulation for the entire model. The geometric method can be slow for large models or large numbers of global handles, but may produce more desirable influences. The geometric method is the method that was originally used in HyperMesh and generates influences based on the geometric relationship between a given node or local handle and the surrounding global handles. The method used can be selected in the global subpanel of the Morph Options panel.

Local Domains and Handles

Each local domain is associated with any number of local handles. Local handles will only influence nodes contained within their associated local domains. Local handles are intended to be used to make small scale, parametric changes to the model.

A model can contain both global and local handles and domains, which allows for both large and small scale morphs. It is not necessary to have both types of domains and handles in a model.

Local domains are represented by a single rectangle for 1D domains, two joined rectangles for 2D domains, a cube for 3D domains, four joined rectangles for general domains, and a line for edge domains.

Local handles are colored orange if they are not dependent on other handles. Local handles are colored green, blue, or pink if they are dependent on other handles, the color indicating their level of dependency. The base size of all the handles in the model can be set on the morphing Visualization Controls tab accessed by using the Visualization Options icon ( ) on the Visualization toolbar. The size given is used as the diameter for the independent local handles. You cannot edit the color of the handles nor the relative size between the dependent and independent handles. However, you can edit the color of the domains in the morphing Visualization Controls tab.

Local domains can be created individually by selecting nodes or elements in the Domains panel, Create subpanel. When local domains are created, HyperMorph automatically places local handles at the ends of all edge domains. These local handles are named local followed by a number. The placement of local handles depends on the type of domain created and the partitioning options, if partitioning is selected.

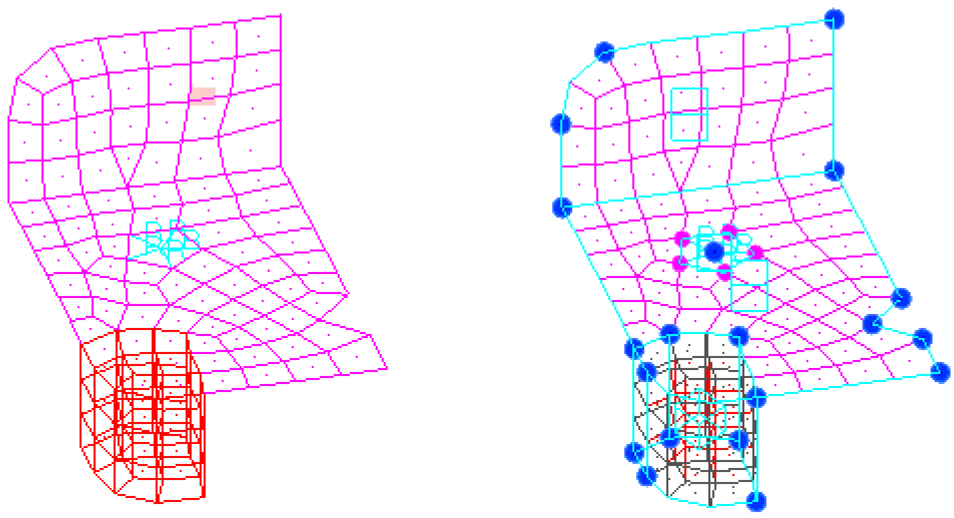


Figure 1720: Local Domains and Local Handles with Partitioning

In Figure 1720, the rigid elements have been placed in a 1D domain with the center node having an independent (orange) handle and the other nodes having dependent (green) handles. The shell elements have been placed in two 2D domains separated at the bend line due to partitioning. The solid elements have been placed in a 3D domain. Shell elements have been created on the faces of the 3D domain. These elements are placed in a component named ^morphface. 2D domains have been created on the faces of the 3D domain and that edge domains have been created on the edges of all the 2D domains. Handles have been placed at the ends of all the edge domains.

1D Domains

1D domains are made up of 1D elements, such as bars and rigid elements.

When automatically creating local 1D domains, 1D elements that share common nodes are grouped together into 1D domains. An independent local handle is placed at the centermost node of the 1D domain and dependent local handles are placed at every other node of the elements in the 1D domain. The independent handle is larger and orange, while the dependent handles are smaller and green. All the dependent handles in a given 1D domain are directly dependent on the independent handle. This dependency relationship means that moving the independent handle also results in moving the dependent handles the same amount in the same direction. This is done to preserve the unique relationship established for groups of 1D elements. Additionally, the bias factors for the dependent handles for a 1D domain are given an initial value of 3. All other handles in the model are given a biasing factor of 1. A higher biasing factor means that a given handle will have greater influence over the surrounding mesh than the others. The higher biasing factor given to dependent handles on 1D domains is intended to prevent mesh distortion when the 1D elements connect to nodes in 2D and 3D domains.

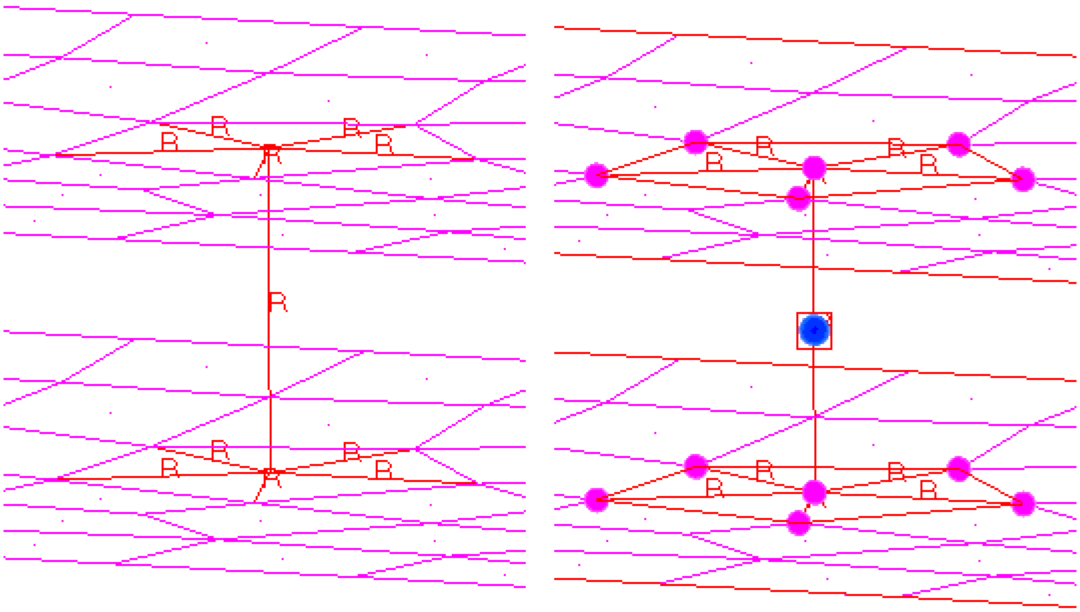


Figure 1721: 1D Domain Created from Rigid Spider Elements

An independent local handle (orange) is placed at the centroid of the 1D domain and dependent handles (green) are placed at each node. By moving the orange handle, the entire spider is moved, maintaining the proper shape and connectivity for the rigid spider.

2D Domains

2D domains are made up of shell elements.

When automatically creating local 2D domains, shell elements that share common nodes are grouped together into 2D domains. If **partitioning** has been selected, these domains are subdivided into smaller domains along break angles and curvature changes according to the partitioning parameters. Edge domains are placed along the edges of the 2D domains and are also partitioned. Local handles are placed at the ends of all the edge domains. In general, the local handles are placed at the corners of the 2D domains and at other useful positions. The intent is to make it faster and easier for you to apply parametric changes to the model. Since you morph the model by moving handles, it helps to have handles already at the positions where you want them. HyperMorph tries to predict where the handles should be placed to reduce the amount of time it takes to prepare your model for morphing. If the handles or domains are not laid out in the positions where you want them to be, you can delete them, edit them, or create new ones. Also, even though the generated local handles are associated with the edge domains, they will influence the nodes in any domain that shares the node at which it is placed. This is true even if the handle is associated with the 2D domain. A handle associated with any domain will always influence the nodes in domains that it is touching. Note that it is possible to create a handle on a node that is not touching the domain to which it is associated. This allows you to place a handle outside of a domain, such as floating in space near the domain, and have it influence the nodes within its domain.

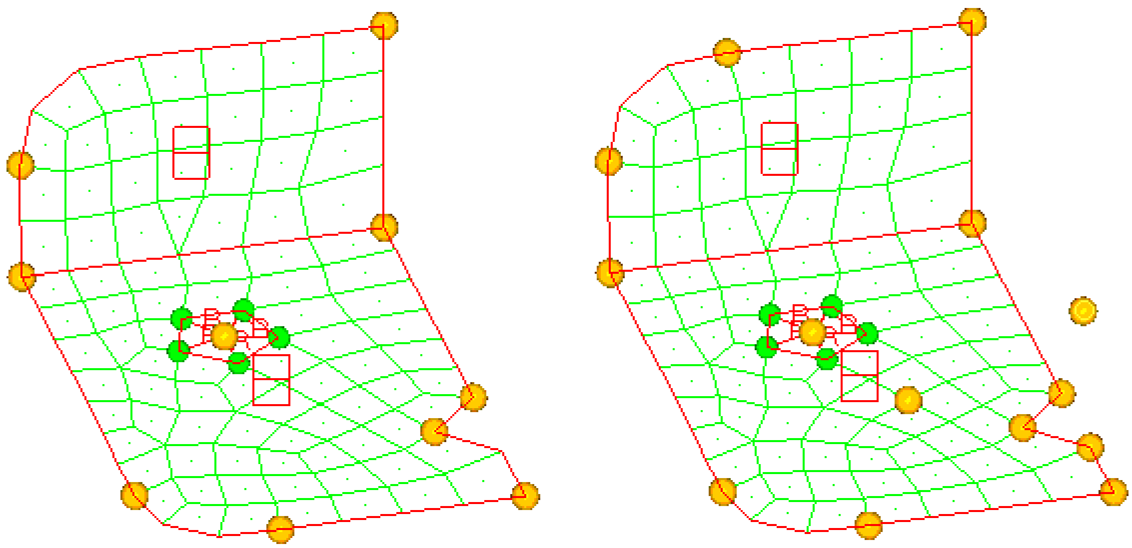


Figure 1722: 2D Domains with Edge Domains and Handles

The model on the left shows the initial handle positions. The model on the right shows the addition of four new handles. Handles can be placed anywhere, even at nodes not on the associated domain.

3D Domains

3D domains are made up of solid elements.

When automatically creating local 3D domains, solid elements that share common nodes are grouped together into 3D domains. Shell elements are created on the faces of each 3D domain and placed into a component called ^morphface. It is recommended that you do not delete or edit these elements nor rename or delete the ^morphface component. However, if you do, these elements and their 2D domains will be regenerated the next time you enter or exit a HyperMorph panel or the Delete panel. The shell elements on the face of each 3D domain are placed into a 2D domain that is then partitioned if the **partitioning** option is active. Edge elements are placed around each 2D domain and local handles are created at the ends of each edge domain. In cases where shell elements that are attached to the faces of solid elements are present in the model, HyperMorph will not create ^morphface elements coincident with the existing elements. The color of the ^morphface component can be changed on the morphing Visualization dialog accessed by using the Visualization Options icon on the Visualization toolbar.

> **Note:** The face elements in the ^morphface component will not be written out to any FEM formatted deck since the component name begins with a "^".

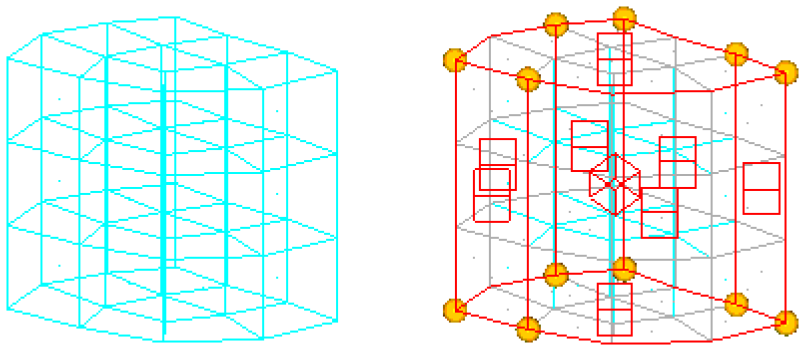


Figure 1723: 3D Domain Created from Block of Solid Elements

The gray shell elements on the face of the 3D domain are the ^morphface component. The ^morphface component has been partitioned into 2D domains. Handles are created at the corners of the 2D domains.

Edge Domains

Edge domains are made up of a list of nodes.

When automatically creating local edge domains, edge domains are placed around the edges of all 2D domains. When you are selecting domains and are holding the mouse button down while placing the mouse over the icon of a 2D or 3D domain, or an element in the domain, HyperMesh will highlight both the domain icon and the surrounding edge domains. This makes it easier for you to tell which domain you are selecting. When you release the mouse button, only the icon for the domain remains highlighted.

Edge domains and 2D domains on the faces of 3D domains play an important function in determining the influences for the handles over a given domain. Nodes on edge domains will only move as a function of the handles touching the edge domain. No other handles will affect the nodes on the edges. Similarly, nodes in a 2D domain on the face of a 3D domain will only move as a function of the handles touching the 2D domain. This preserves the boundaries of 2D and 3D domains such that straight edges remain straight, flat surfaces remain flat, and curved edges retain their curvature. It allows you to move handles within a 2D or 3D domain without affecting the edges. If you do not want to have the boundaries of a domain preserved, you can delete the edges for a given domain, or choose to create the domain as a general domain instead. Also, non-reflective symmetries allow the influences of handles to extend through edges and faces depending on the type of symmetry. For domains that have non-reflective symmetry types, the boundaries may not be preserved during morphing.

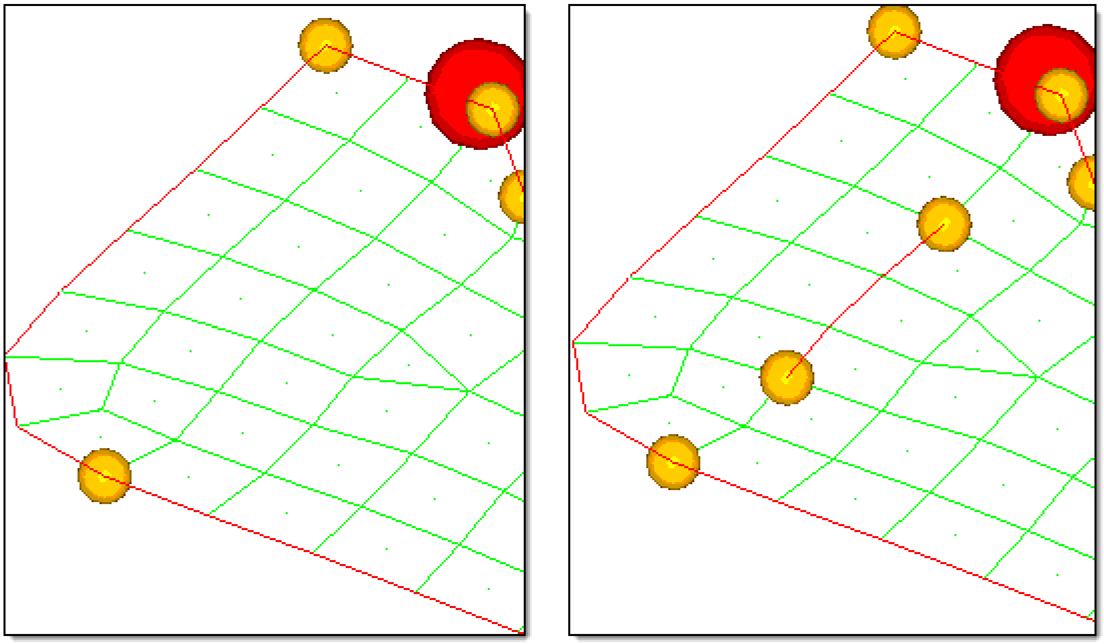


Figure 1724: Edge Domains

Edge domains are placed around the edges of 2D domains. In the model at the right an edge domain has been created inside a 2D domain. When an edge domain is created, it is partitioned and handles are placed at the ends and joints.

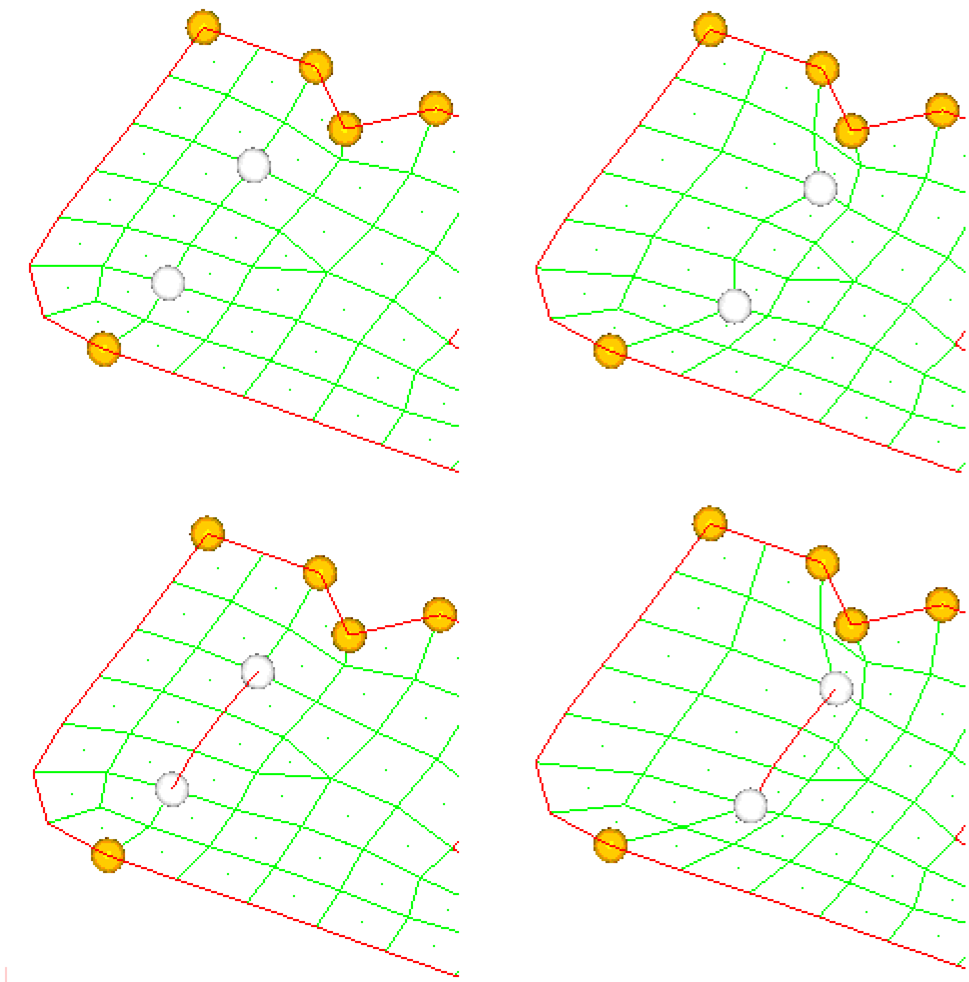


Figure 1725: Edge Domains after Morphing

In the top two frames two handles inside a 2D domain are created and moved. In the bottom frames, two handles connected by an edge domain are created and moved. The edge domain remains straight, preserving the shape of the feature.

General Domains

General domains are made up of any combination of 1D, 2D, and 3D elements.

General domains are not automatically created when automatically generating local domains. Like all other domains, the elements within a single general domain must touch one another.

When a general domain is created, no 2D domains are created on the faces of any 3D elements and no edge domains are created either, thus no handles are created for the domain. However, general domains respect all neighboring edge domains and 2D domains and thus if you create 2D and edge domains for your general domains they will impose restrictions on handle influences for the general domain. Otherwise, handles on a general domain freely influence all of the nodes inside the general domain, allowing it to stretch and deform in an unbounded manner with morphing extending across differences in element type. General domains are very useful for realized connectors which are often represented as clusters of different element types. Another use is for meshes where precise changes are required for one section, where 1D, 2D, and 3D domains are used, but the rest of the mesh, where a general domain is used, can simply follow along.

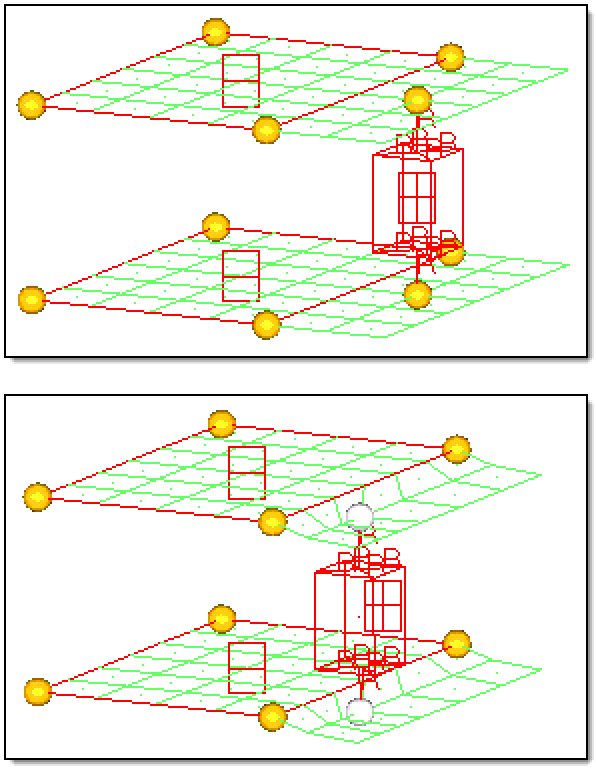


Figure 1726: Interaction between General Domain and 2D Domains

In the top frame, two 2D domains are created for parts of two shell meshes and a general domain is creating from the remaining rigid, shell, and solid elements. Two handles have been placed within the general domain at the ends of the rigid spiders. In the bottom frame the two handles inside the general domain are translated. Note how the shell elements in the general domain morph, bounded only by the edge of the 2D domains with the other domain morph, bounded only by the edge of the 2D domains with the other edges free to follow the handles.

Partitioning

Partitioning is a method of dividing 2D domains into smaller 2D domains at logical places, such as at the edges of surfaces associated with the mesh, or where the angle between elements exceeds a certain value, or where the domain changes from flat to curved.

Partitioning allows you to prepare your model for morphing more quickly and easily since it divides your model into sections where parametric changes can be applied.

Partitioning can be applied directly to 2D domains and indirectly to 3D domains. 3D domains are created with 2D domains on their faces.

You can invoke partitioning when creating 2D or 3D domains by activating the **partition 2D domains** checkbox on the Domains panel. If there are no surfaces in the model, or the use geometry option in the Domains panel, Partitioning subpanel is unchecked, partitioning will divide your model such that every radius and straight or flat section is placed into a separate domain. Partitioning is not always exact, therefore it is possible that there will be areas where elements are not placed into the desired domains. If you are unsatisfied with the partitioning, you may change the partitioning parameters in the

Domains panel, Partitioning subpanel and try again, using the redo last button, or edit the domains by hand using the create and organize subpanels in the Domains panel.

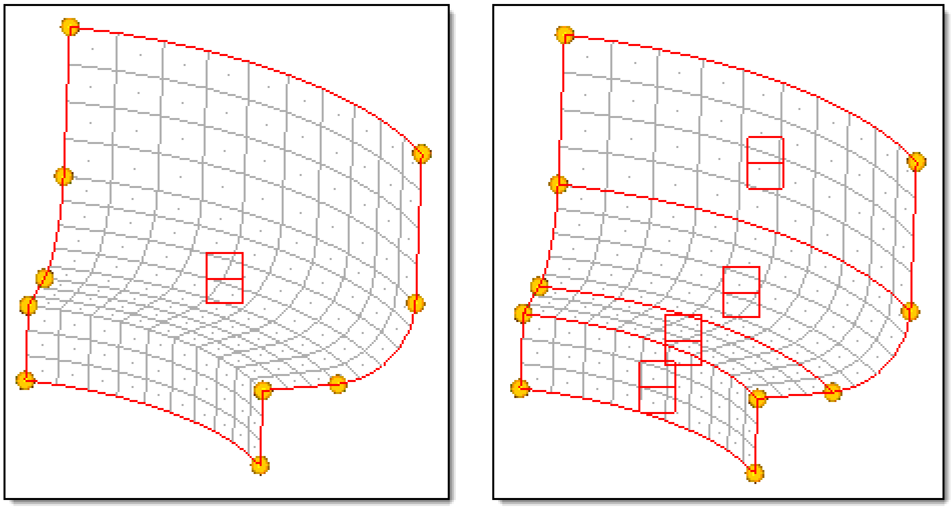


Figure 1727: Partitioning

For the model on the left, the 2D domain was created without partitioning. For the model on the right, partitioning was used. Note how the 2D domains are divided along angle and curvature change boundaries. Also note that the edge domains are partitioned regardless of whether the partitioning option is on or off.

There are two algorithms you can use to partition, element-based and node-based. These can be set individually for quad/mixed meshes and for tria/tetra meshes. In general, the element-based algorithm works better for quad/mixed meshes and second order meshes, while the node-based algorithm works better for tria/tetra meshes.

There are also several parameters that govern the creation of domains for either algorithm. They are found in the of the Domains panel, Partitioning subpanel.

If you have selected **use geometry**, all elements whose nodes are associated to surfaces in the model will be partitioned along the edges of the surfaces. All other elements will be partitioned using one of the partitioning algorithms. If you have also selected **add to geometry**, then any partitions created outside of the surfaces will be added to the partitions created using the surfaces if the partitioning algorithm does not find a break along the edges or the surfaces. This option is helpful when surface data is incomplete of some of the nodes have been moved away from their surfaces.

Partitioning can be angle-based or curvature-based. In either case, the domain angle controls the break angle along which a partitioning break is made. If the angle between the normal vectors between two elements is greater than this value, a new domain is created with an edge running between the two elements. When using curvature-based partitioning, the curve tolerance controls the angle of which values less than it are considered straight for curvature measuring purposes. If the angle between the normal vectors between two elements is less than this value, they are considered flat, otherwise they are considered to be curved. If the curvature changes from straight to curved, changes direction, or changes curvature by more than the curvature tolerance, a new domain is created with an edge running between the two elements. Note that in order for a new partition to be created, a break due to angle or curvature must be found along its entire edge.

For the node based method, domain angle and curve tolerance have a roughly similar meaning as the element based method. The node based method tends to create fewer partitions than the element based method, although exact performance for each method depends heavily on the features in your model. For instance, the node based method seems to work better on first order tria and tetra meshes while the element based method seems to work better on mixed quad and tria meshes.

Dependent Handles

Handle can be made dependent on one or more other handles, and then those handles can be made dependent on one or more other handles, and so on.

You can create any number of dependency layers. Handles that are dependent on other handles appear smaller and in a color different from the handles on which they are dependent. In the Handles panel, Update subpanel, clicking **review** allows you to view the handles on which a specific handle is dependent. Making a handle dependent has no affect on the way it influences nodes.

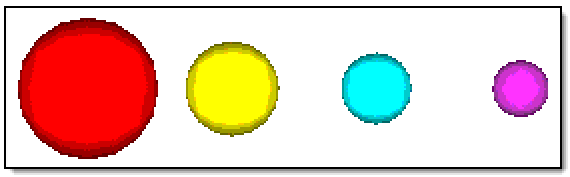


Figure 1728: Global Handles
Independent (red) and dependent (yellow, cyan, and violet).

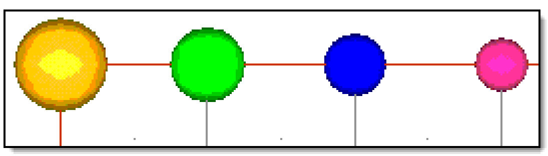


Figure 1729: Local Handles
Independent (orange) and dependent (green, blue, and pink).

The conditions for handle dependency are as follows:

- A handle that is dependent on another handle inherits the movements applied to the higher level handle.
- If a handle is dependent on only one other handle, it inherits the full movement of the higher level handle.
- If a handle is dependent on more than one handle, it will inherit a percentage of the movements applied to each higher level handle. The percentage is based on the distance between the dependent and independent handles.
- A handle may be dependent on any number of handles, but dependency loops are not allowed.
- A dependent handle can be moved independently of the handles on which it is dependent. This means that movements applied to the dependent handle are not applied to the independent handles. This allows you to add the movements of dependent and independent handles in a logical manner.

- In the hierarchical method, all local handles are dependent on global handles. These dependencies are calculated internally and cannot be modified manually, biasing will affect them.

Handle dependencies are useful for several different applications.

Transparent control of domain edges and faces

You can create a dependent handle on an edge domain that is dependent on the handles at the ends of the domain. When the dependent handle is moved, the shape of the edge can be changed. When the handle at either end of the edge domain is moved, the dependent handle moves along as if it was not there. This allows you to combine the changes easily without having to apply separate perturbations for all of the handles.

Grouping features together to move as a unit

You can make all the handles at one cross section of a beam dependent on a single handle. This allows you to move an entire cross section while only having to select one handle.

Linking several domains together

You can make all of the handles within several domains dependent on a few at the corners of the domain. This allows you to stretch all of the domains uniformly by moving the independent handles, in essence, performing localized "global" morphing.

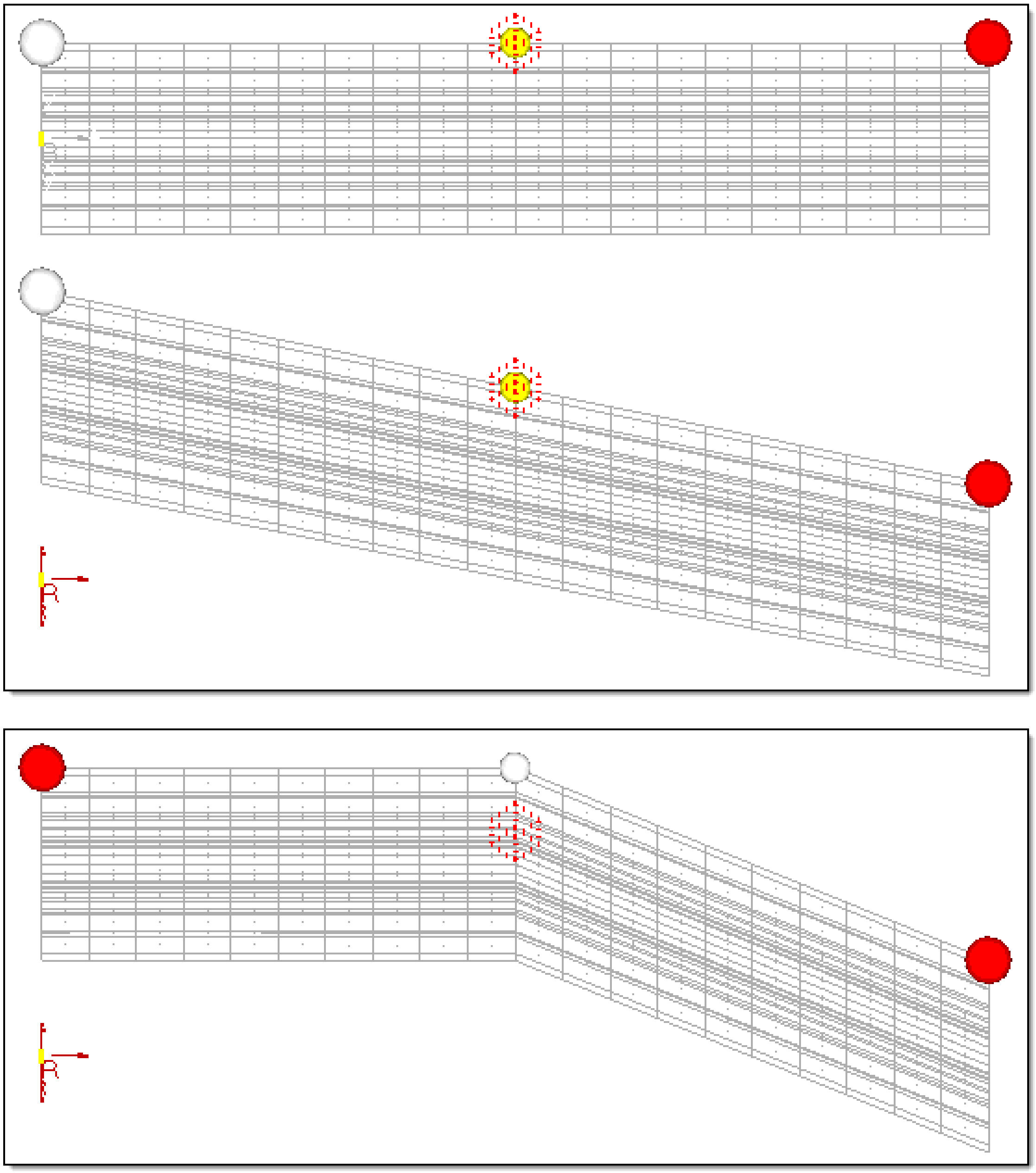


Figure 1730: Dependencies

The center global handle is dependent on the two outer global handles. When the highlighted handle on the left is moved (center frame), the center handle follows along. In the lower frame, the center handle is moved independently.

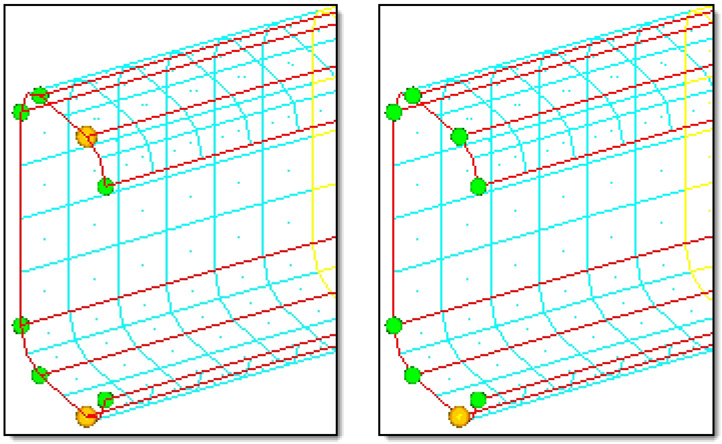


Figure 1731: Dependencies

In the model on the left, the three green handles on the top are dependent on the orange handle on the top. The bottom has similar dependencies. The top and bottom halves of the cross sections are controlled by just two handles. In the model on the right, all of the green handles are dependent on the orange handle. The entire cross section is controlled by one handle. Note that the dependencies can extend beyond the 2D domain boundaries.

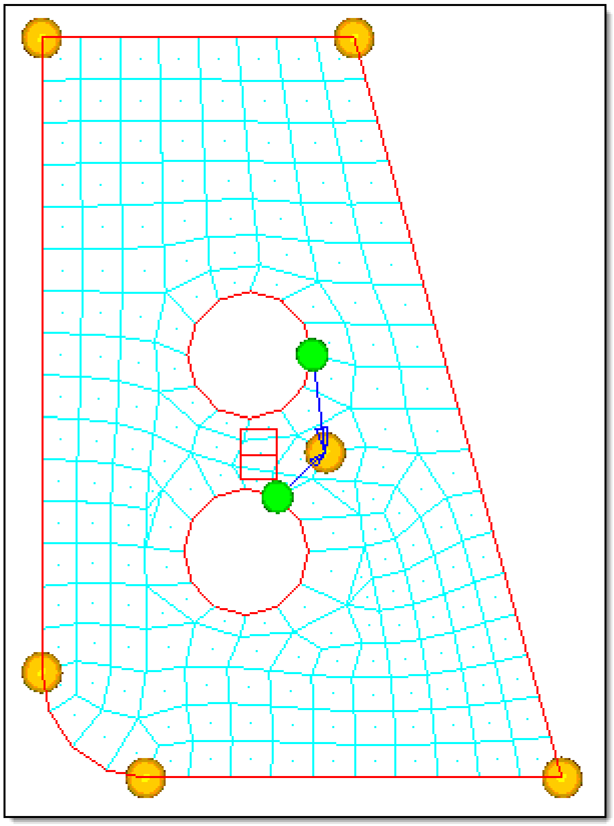


Figure 1732: Dependencies

An independent handle was created between the two holes and the handles governing the positions of the holes are made dependent on it. When the independent handle is moved, both holes move with it. Also, each hole can be positioned separately by moving the dependent handle associated with it.

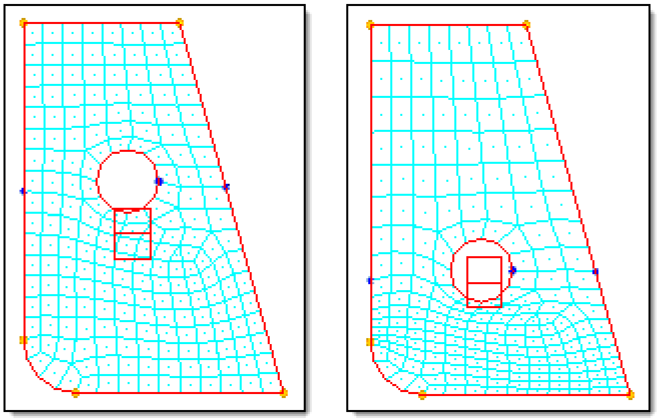


Figure 1733: Dependencies used to Reduce Mesh Distortion

Two dependent handles were created on the edges of the part near the center hole. The dependent handles were constrained along vectors parallel to the sides of the part. When the handle at the hole is moved downward, the dependent handles follow and reduce mesh distortion by spreading the morph across the entire part instead of only around the hole.

Shapes

Shapes are collections of handle and/or node perturbations from the initial configuration of the FE mesh before the morph.

When you morph your model, HyperMorph stores the morph internally as a collection of perturbations which you can then undo or redo. When you create a shape using the save shape subpanel on the Morph panel or Freehand panels, the handle and/or node perturbations are stored in the new shape entity along with biasing factors for the handle perturbations and details such as the biasing style. HyperMorph takes the difference between the initial state of the model and the current state of the model when creating a new shape. If you save the model using the **save each morph step** option in the Shapes panel, each morph on the undo/redo list will be saved as a separate shape. To get to the current state of the model from the initial state, all of these shapes must be applied. Creating shapes allows you to generate shape variables for optimization and store model changes for parametric studies.

For many morphing operations, the morph consists only of handle perturbations. However, if constraints are being used, or the morph is a mapping or radius changing operation, node perturbations are required to fully describe the shape. In the case of freehand morphing, the morph consists only of node perturbations. When you create a shape, vectors are drawn for each handle and node perturbation for the shape. The vectors are drawn the exact length of the perturbation and the vectors for the handle perturbations are drawn with thicker lines to denote that they are different from node perturbations.

> **Note:** While shapes with handle perturbations will move nodes when they are applied, those shapes do not contain node perturbations and thus vectors are not drawn at those nodes.

When you are saving a shape, you can select whether to save it as:

as handle perturbations

Save shape as either handle perturbations only, or a combination of handle and node perturbations if node perturbations are required to describe the shape.

Shapes saved as handle perturbations will differ from shapes that have been saved with changes to the handle influences.

node perturbations

Save shape as node perturbations only.

Shapes saved as node perturbations are not affected by changes to domains and handles.

Whenever you make a change to your model, HyperMorph will ask you if you want to preserve any existing shapes saved as handle perturbations by converting them to node perturbations. If you plan to make changes to domains and handles, you should save shapes as node perturbations. If not, save shapes as handle perturbations and they will require less memory and disk space. If you later decide that you want to change a shape from node perturbations to handle perturbations or vice versa you can do so in the Shapes panel, Convert subpanel.

Once a shape is saved, you can apply it to your model with any given scaling factor. Applying a shape in this way is like any other morphing operation and can be undone, redone, or saved as part of another shape.

Convert Saved Shapes

Convert shapes saved with handle perturbations to shapes saved with node perturbations, or vice-versa.

1. From the Tool page, click **HyperMorph** module.
2. Click **Shapes**.
3. Select the **Convert** subpanel.
4. Select the type of conversion to perform.
5. Select the shapes to be converted.
6. Click **convert**.

Optimization Setup

Morphing can be used to create shape variables for optimization.

> **Note:** A shape is not a shape variable, but by adding a desvar which points to the shape, it becomes a shape variable.

Create Shape Variables for Optimization Runs

Morph your model into the shape of the first shape variable.

1. From the Tool page, click **HyperMorph** module.
2. Click **Morph**.
3. Select the **Save Shape** subpanel.

4. Save morph shape for each shape variable you want to create.
    a) Save your morph as a shape.
    b) Click undo all to return to your base model shape.
5. From the Analysis page, click **Optimization** module.
6. Click **Shape**.
7. Select **multiple desvars**.
8. Use the shapes selector to select the shapes for which you want to create shape variables.
9. Click **create**.

    A desvar for each shape is created with the initial value and bounds in the panel. Each desvar is given a unique name.
10. Animate the shape variables.
    a) Click **undo morphing** if you did not click undo all after saving the last shape.
    b) Click **animate**.

Once you have created shape variables for your shapes, you can set up the rest of your optimization problem within the optimization module.

Morph Volume Approach

The Morph Volume approach involves surrounding the mesh with one or more morph volumes, which are highly deformable six-sided prisms.

A number of methods exist to create the morph volumes, including single and matrix creation as well as the interactive on-screen method. Morph volumes support tangency between adjoining edges and allow for multiple control points along their edges. Handles placed at the corners and along the edges of the morph volumes allow for the morphing of the morph volumes which in turn morphs the mesh inside the morph volumes. The morph volume approach is quick and intuitive and is most useful for making large scale changes to complex meshes.

Freehand Approach

The Freehand approach involves morphing by moving the nodes directly without the need to create any HyperMesh morphing entities.

You define the nodes which will move, the nodes which will stay fixed, and the affected elements, which manually allows for rapid changes to any mesh. You have great flexibility in how the moving nodes are moved, such as translation, rotation, and projection to geometry as well as using a tool to "sculpt" the mesh into the desired shape. You are also able to turn node manipulations made in any panel, such as scaling or node projection, into morphs using the record subpanel. The freehand approach is an ideal introduction to HyperMorph since it allows morphing without the creation of any HyperMesh morphing entities while employing the concepts of domains and handles. The freehand approach also allows for "customized" morphing, allowing you to do virtually any kind of morphing.

Space Frame Model Strategies using Global Domains

Space frames are models that have a sparse distribution of elements, such as a car body. Space frame models can generally have element counts in the hundreds of thousands, but their basic structure is rather simple.

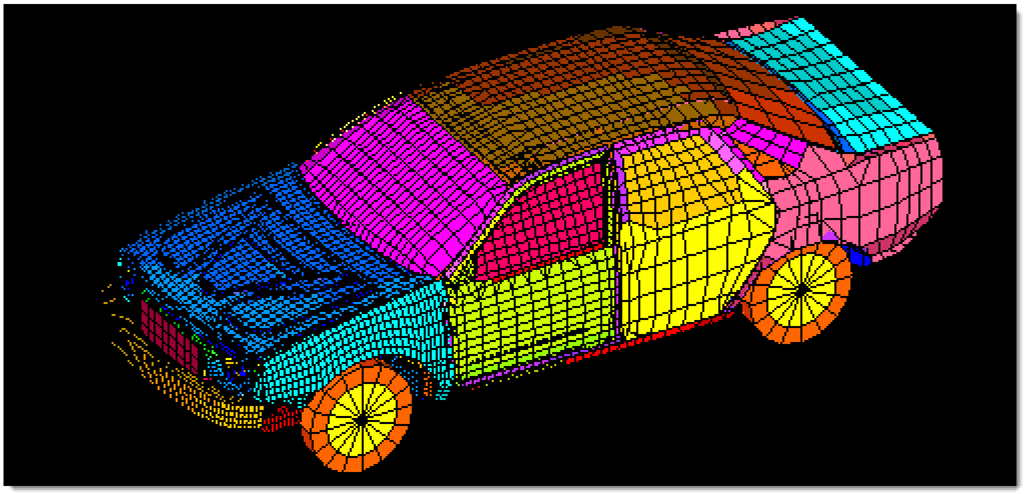
Often the desired shape changes are general, such as making it smaller, shorter, wider, or altering the basic positions of components within the frame. In many instances, these changes can be performed by placing a handle at each joint in the frame and moving those handles to the desired locations. For these types of models, all that is necessary is to create a global domain and global handles. Local handles are not required since local changes to the frame components are not necessary. Since local handles and domains for large models can consume a great deal of resources, you should avoid creating them unless it is necessary.

Create Handles and Domains for Space Frame Model

Create a global domain and global handles at useful positions throughout the space frame.

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Create** subpanel.
4. Set the selector to **global domain**.
5. Set the toggle to **all nodes**.
6. Set the toggle to **create handles**.
7. Click **create**.
   A global domain and global handles are created at useful positions throughout the space frame.
8. If the handles are not where you want them to be, delete the handles and add global handles elsewhere.
   a) Open the Delete panel by pressing **F2**.
   b) Delete unwanted handles.
   c) Go to the HyperMorph module, Handles panel, Create subpanel.
   d) In the name field, enter a name for the handle.
   e) Select an xyz position or any number of nodes where you want global handles.
   f) Click **create**.

A new global handle is created at each node or at the specified xyz location. If more than one handle is created at a time, the handles will each be given a unique name by appending a number after the name you have given. You should place global handles both in areas where you want to apply perturbations and in areas that you want to stay fixed. You can also use morph constraints to fix nodes in place during global morphing but if you want them to affect the surrounding mesh you must select the stretch mesh around nodes option when creating the morph constraint. If you want a part of your model to move as a rigid body, such as a wheel or the engine block, use a cluster type morph constraint.



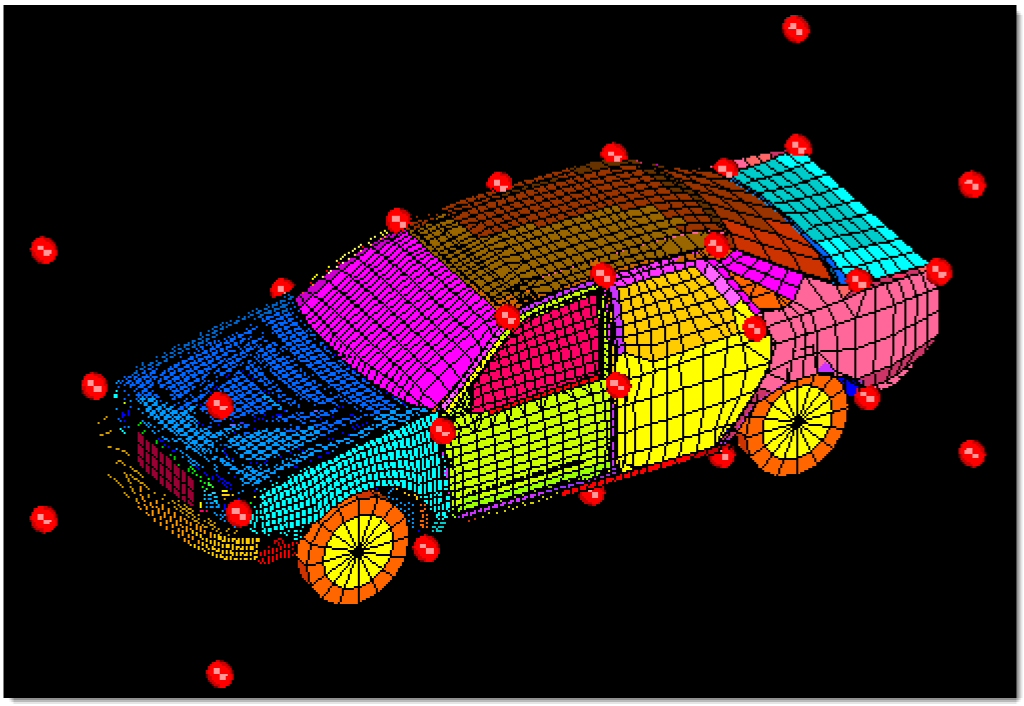


Figure 1734: Global Domain and Global Handles for Full Car Model

Exiting any panel in the HyperMorph module or the Delete panel automatically triggers HyperMorph to refresh the handle influences, if necessary. Adding, editing, or deleting handles, domains, or symmetries makes it necessary for HyperMorph to refresh the handle influences. For large models or large changes, this can be time consuming, so you will want to make all the changes you desire within each panel before exiting.

Match Mesh, Line, or Surface Data

Move the handles into positions that change the shape of the model to match the mesh or geometry data. If you are going to match a mesh you need to make sure that the mesh does not get morphed when you are moving the handles.

This can be accomplished by constraining the nodes on the target mesh.

Constrain Nodes on the Target Mesh

Constrain all of the nodes in the target mesh to remain fixed during morphing operations.

1. From the Tools page, click **HyperMorph**.
2. Click **Morph Constraints**.
3. Select the **Create/Update** subpanel.
4. Select the nodes on the target mesh.
5. Switch the selector to **fixed**.
6. Clear the **stretch mesh around nodes** checkbox.
7. Click **create**.

   All the nodes in the target mesh are constrained to remain fixed during morphing operations as long as the constraint is active and the **use constraints** checkbox is selected. See the Morph Options panel.

   > **Note:** If you select the **stretch mesh around nodes** checkbox, the nodes between the constrained nodes and the handles will be affected regardless of whether the mesh is continuous between them.

Morph Interactively

Morph interactively by moving the handle and releasing the mouse button.

One of the most enjoyable ways to morph is interactively. As you drag a handle across the screen and you can watch the mesh move along with it. For large models it may be too slow to morph interactively in real time, but you can still morph interactively with any size model by setting HyperMorph to perform the morphing after you move the handle and release the mouse button.

You can also select other features to drag the handle along such as a line, a plane, or a surface. HyperMorph uses the position of the mouse on the screen to figure out where you want to move the handle. You can use this feature to position a handle anywhere you want line or surface data.

1. From the Tools page, click **HyperMorph**.
2. Click **Morph**.

3. Select the **move handles** subpanel.

4. Change the upper middle selector to **interactive**.

5. Change the rightmost toggle from real time to **on release**.

6. Change the lower middle selector from on domains to **along vector**.

7. Select a vector.

8. Morph.

   a) Click **morph**.

   b) Select a handle on the screen and hold the mouse button down.

   c) Move the handle to the new location and release the mouse button.

   As you drag the mouse, the handle follows along the selected vector. Since on release was selected, only the graphics for the handle are updated, which leaves a dark trail through the mesh. When you release the mouse button, the morph is applied to the model and the graphics are updated for the entire model. If the handle position needs to be changed again, repeat step 8.

Tip: To save time, move more than one handle at the same time by selecting several handles before performing step 8. When you release the mouse, all of the selected handles are moved the same distance in the same direction.

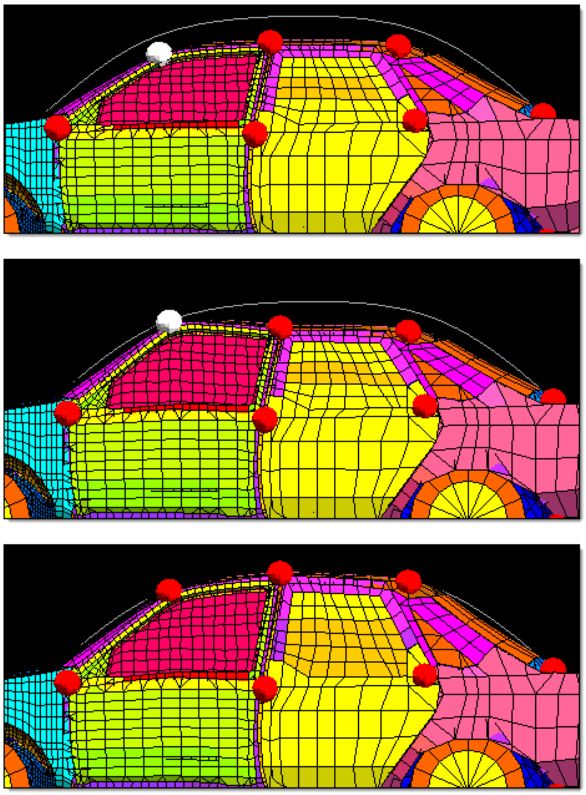
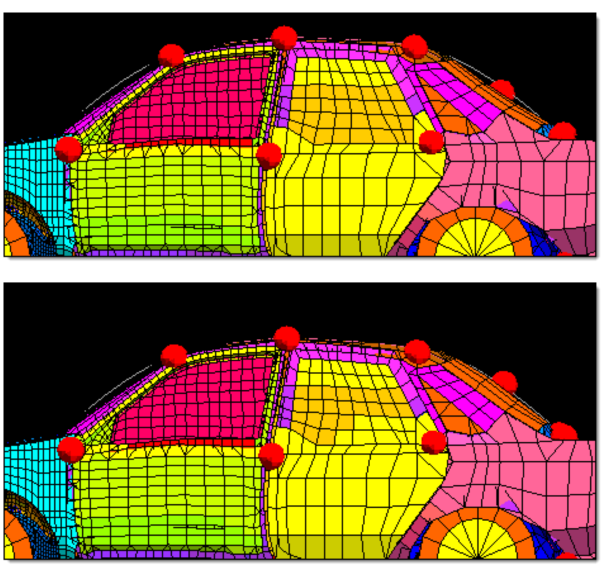


Figure 1735: Morph to Profile Line

In the top frame, the global handles on either side of top of the windshield are selected. In the middle frame they are interactively moved upwards along a vector to a point matching with the profile line. In the bottom frame the process has been repeated for the other handles on the roof. The result is a morphed vehicle model that closely matches the profile line.



2. Click **Morph**.
3. Select the **move handles** subpanel.
4. Change the upper middle selector from interactive to **move to node**.
5. Select a handle.
6. Select a node.

The handle is moved to the position where the node was prior to morphing and the rest of the mesh morphs accordingly.

Create Nodes on the Fly

Create nodes on the fly on lines and surfaces.

1. From the Tools page, click **HyperMorph**.
2. Click **Morph**.
3. Hold the mouse button down and drag the mouse over a line or surface until it is highlighted.
4. Click on the line or surface where you want the node.

A node will be created and the handle will immediately be moved to the node.

Make Parametric Changes

Dimensions such as distance and angle can be changed easily in HyperMorph.

Translate/Rotate Handles

1. From the Tools page, click **HyperMorph**.
2. Click **Morph**.
3. Select the **Move Handles** subpanel.
4. Translate the handles.
   a) Change the upper middle selector from move to node to **translate**.
   b) Use the handles selector to select handles.
   c) Select a vector and distance.
   a) Select the desired xyz translation.
   b) Click **translate**.
   The handles move the specified distance in the specified direction and the model morphs accordingly.
5. Rotate the handles.
   a) Change the upper middle selector from translate to **rotate**.
   b) Use the handles selector to select handles.
   c) Select an axis of rotation.
   d) Set the rotation angle.

e) Click **rotate**.
The handles rotate about the axis the specified angle and the model morph accordingly.

6. Specify dimensions more precisely

a) Select the **Alter Dimensions** subpanel.
b) Set the upper left selector to **distance**.
c) Change the middle left selector to **nodes and handles**.
d) Select node a and node b at nodes whose distance you want to change
e) Select follower handles for node a that are near node a.
f) Select follower handles for node b that are near node b.
g) Change the distance value.
h) Click **morph**.
HyperMorph moves the follower handles for node a as a group and the follower handles for node b as a group either towards each other or away from each other so that the new distance between node a and node b is equal to the specified distance. If the left selector is set to hold end a, node a will not move, same for node b. If the left selector is set to hold middle, both node a and node b will move the same distance.

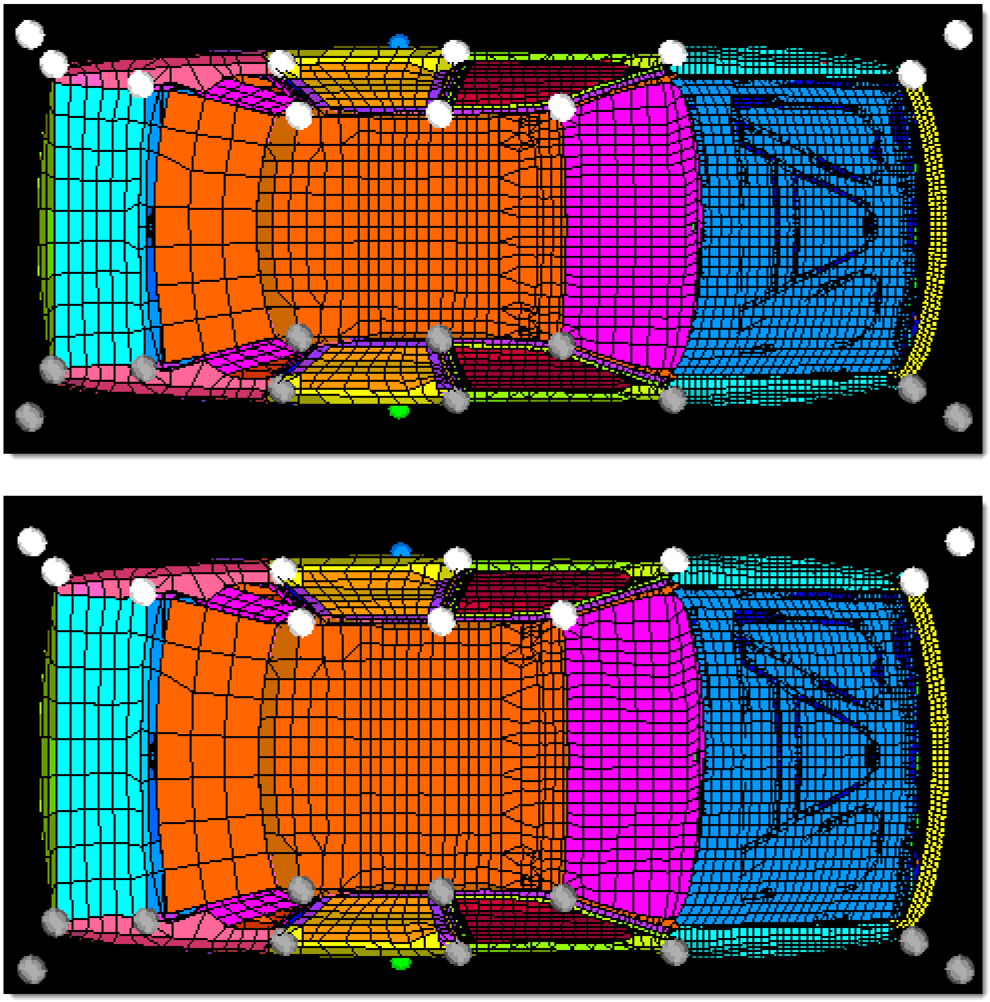


Figure 1737: Morph by Altering Distance between Two Nodes

The width of the car is found by placing node a (green dot) on the right hand door and node b (blue dot) on the left hand door. The handles on the right side of the model are selected as followers for node a and the handles on the left side of the model are selected as followers for node b. The distance is changed and the model morphs.

Change Angles

1. From the Tools page, click **HyperMorph**.
2. Click **Morph**.
3. Select the **Alter Dimensions** subpanel.
4. Set the upper left selector to **angle**.
5. Change the middle left selector to **nodes and handles**.
6. Select node a, vertex, and node b at nodes whose angle you want to change.
7. Select follower handles for node a that are near node a.
8. Select follower handles for node b that are near node b.

9. Change the angle value.
10. Click **morph**.
    HyperMorph moves the follower handles for each end in a way so that the new angle between node a, the vertex, and node b are the specified angle. If necessary, HyperMorph will iterate to achieve the desired angle, or at least get close. If node a and node b are selected coincident with one of the follower handles, iteratation is not necessary.

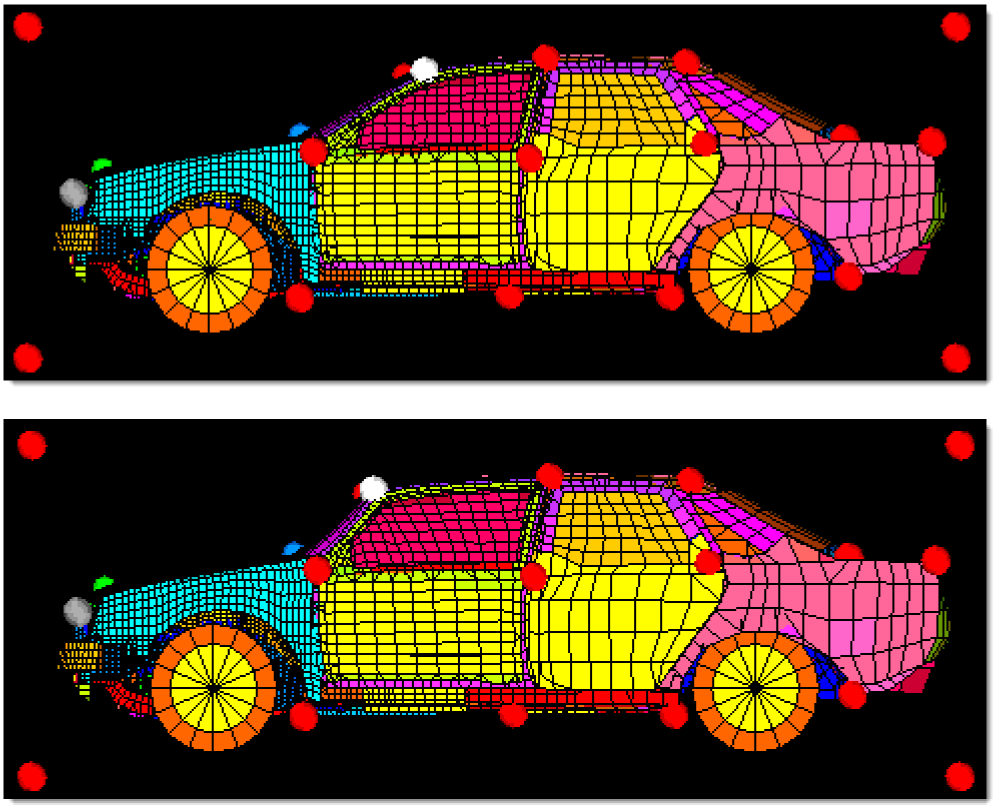


Figure 1738: Morph by Altering Angle Formed by Three Nodes

The slope of the windshield is altered by defining an angle using three nodes, green, blue, and red, selecting two handles at the front of the car as followers for node a (green), and selecting two handles on either side of the windshield as followers for node b (red node). The angle is changed from 160 degrees to 150 degrees. Note that the handles on either side of the windshield were constrained to move along the x-axis (front to back) thus maintaining the height of the roof.

Global Morphing with Handle Placement

Global morphing differs from local morphing in that there are no definite boundaries between the handles that restrict their zones of influence. When you perform global morphing operations, the parts of the model that are morphed are those that lie between the handles that are moving and those that are not.

For the general space frame cases, positioning handles at the joints between the members of the space frame restricts the handle influences to the parts of the frame that they are touching. However, for cases where you are trying to morph a mesh that covers a wide area, you will need to place several handles across both of sides of the zone of influence.

You can visualize the handles as places on a sheet of rubber where you are placing your fingers. If you place three fingers on one side and two on the other and try to stretch the sheet, the space between your fingers on the two finger side will be pulled towards the three finger side. By placing three fingers on each side, you allow for even stretching to occur between each set of fingers. In morphing this is accomplished by placing handles evenly along both sides of the mesh to be stretched.

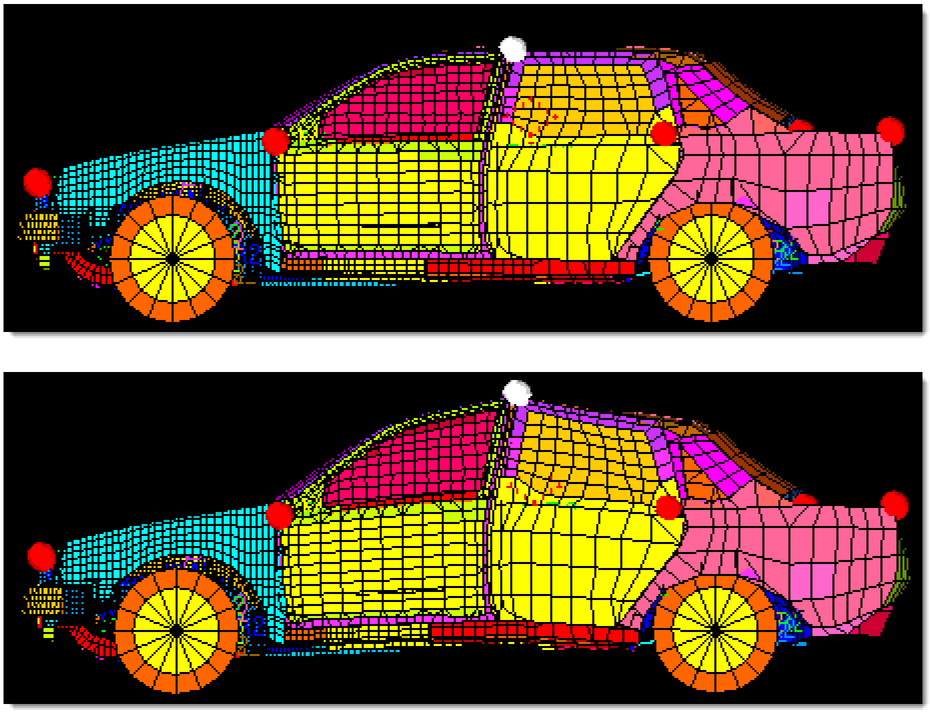


Figure 1739: Control Global Morphing with Handles

The handle on the roof is moved upwards and the center section of the car is morphed along with it.

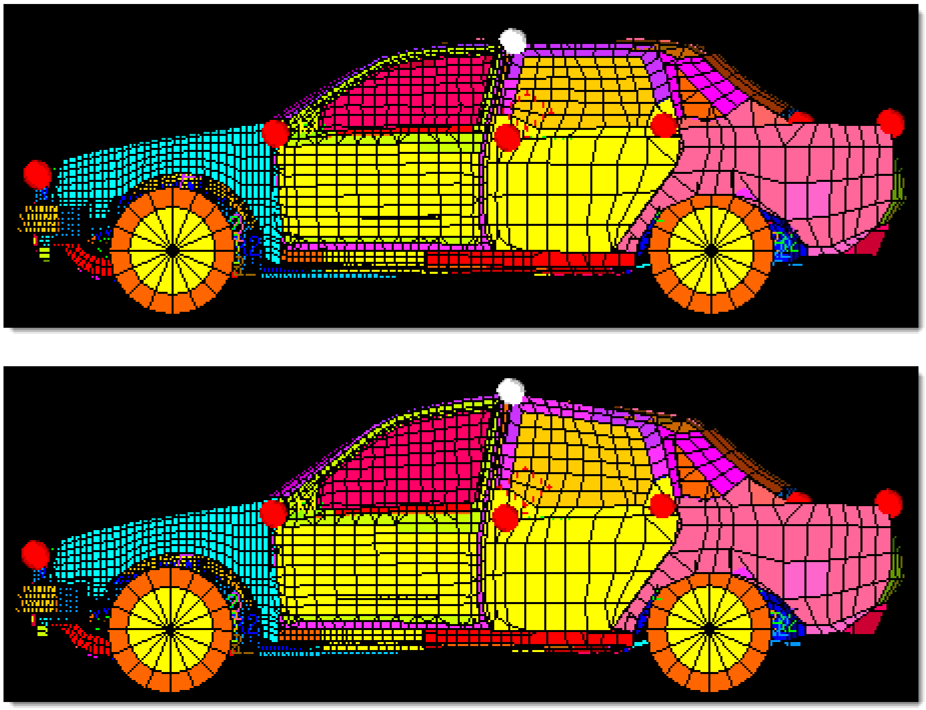
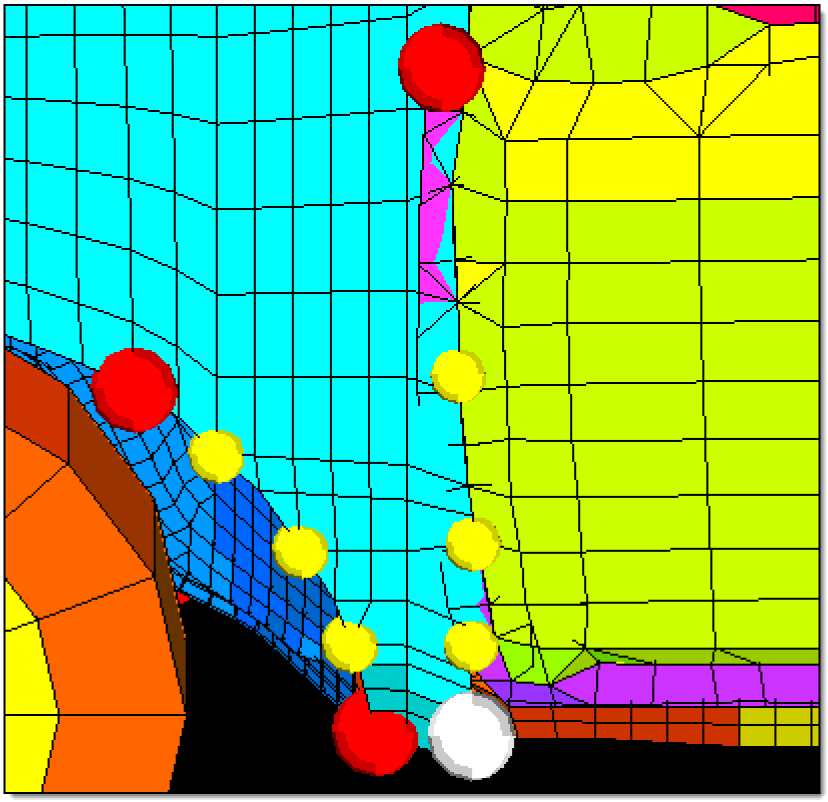
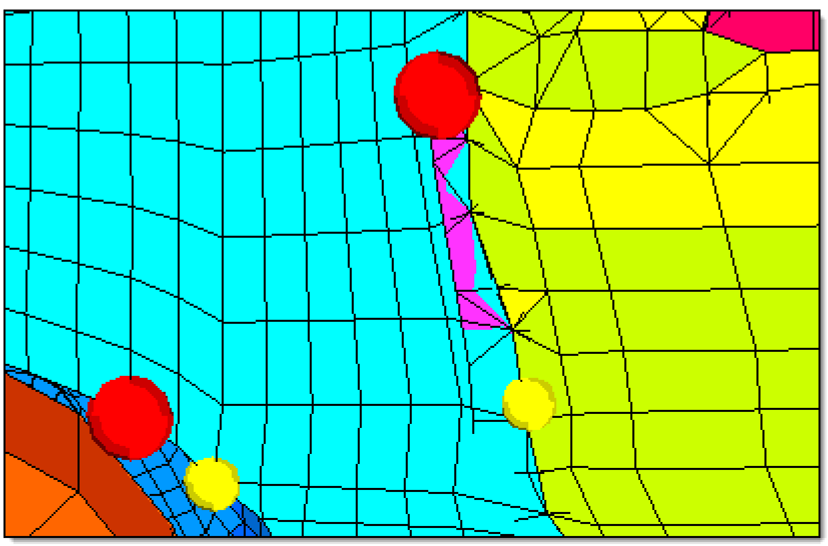


Figure 1740: Control Global Morphing with Handles

A handle is added directly below the handle on the roof near the center of the car. Now when the handle on the roof is moved upwards, only the part of the car between the roof and the handles along the midline of the car is stretched.





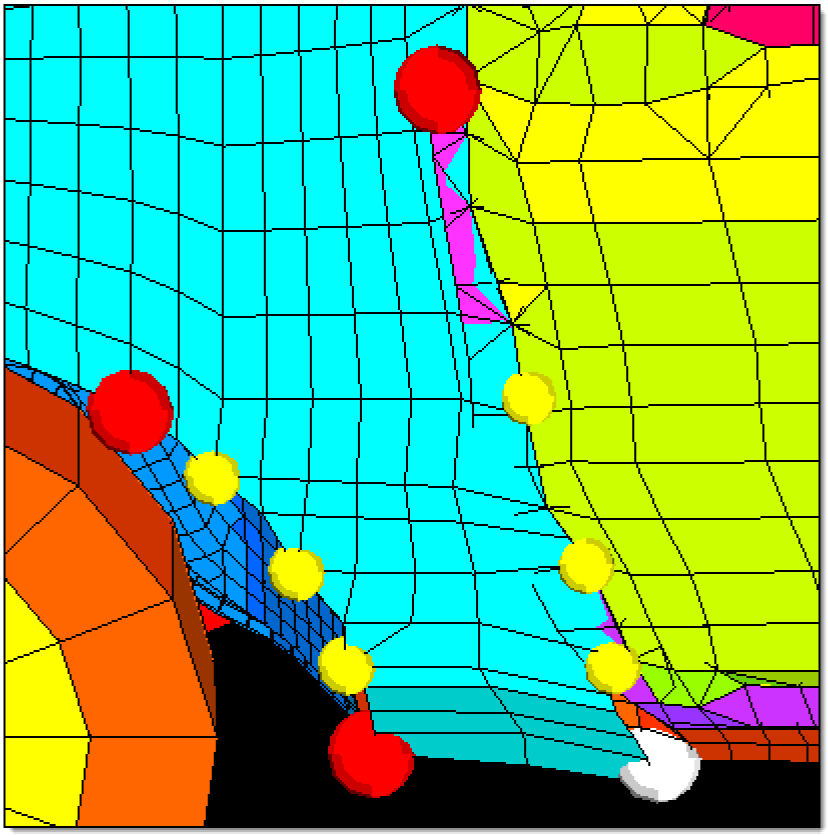


Figure 1741: Add Handles to Control Global Morphing

Using several handles on either side, the fender of the model is morphed. Note that dependent handles are used to simplify the morphing operation. Also note that in cases where detailed shape changes are required, morph volumes will usually yield better results.

Mirror Images using 1 Plane Symmetry

If your space frame is symmetric, you can create a plane of symmetry at the center of your space frame and have your morphs applied in a symmetric fashion.

1. From the Tools page, click **HyperMorph**.
2. Click **Systems**.
3. Create a system at a node where the plane of symmetry is to be located and have the x-axis pointing normal to the plane to be created.
4. Click **return** to go to the HyperMorph module.
5. Click **Symmetry**.
6. In the name field, enter a name.
7. Select the global domain icon.
8. Switch the selector from none to **1 plane**.
9. Select the system you created.
10. Select x-axis as the axis to align the symmetry.

11. Change the left toggle from approximate to **enforced**.

12. Click **create**.

A plane of symmetry is created at the origin of the system and based perpendicular to the x-axis. The icon for a 1-plane symmetry is a rectangle positioned like a small mirror for the symmetry system. HyperMorph also links any handles that it finds that are reflections of the other. Since enforced was selected, HyperMorph creates new handles that are reflections of ones that are not linked to any others and creates a symmetric link between them. When handles are created or deleted, the enforced option will automatically create or delete handles on the other side of the symmetric link in order to enforce symmetry of the handles.

The mesh itself does not need to be symmetric to use the symmetry options. The symmetry will be applied to the handles and handle perturbations that will influence the mesh in a symmetric fashion. If you want to add handles to one side of the plane of symmetry and not the other, yet still have symmetry active for the symmetric handles, use the approximate option instead.

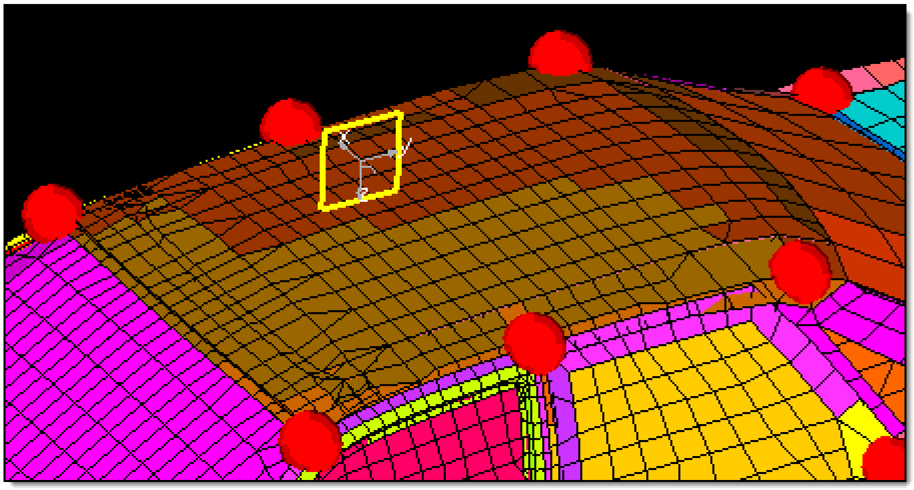
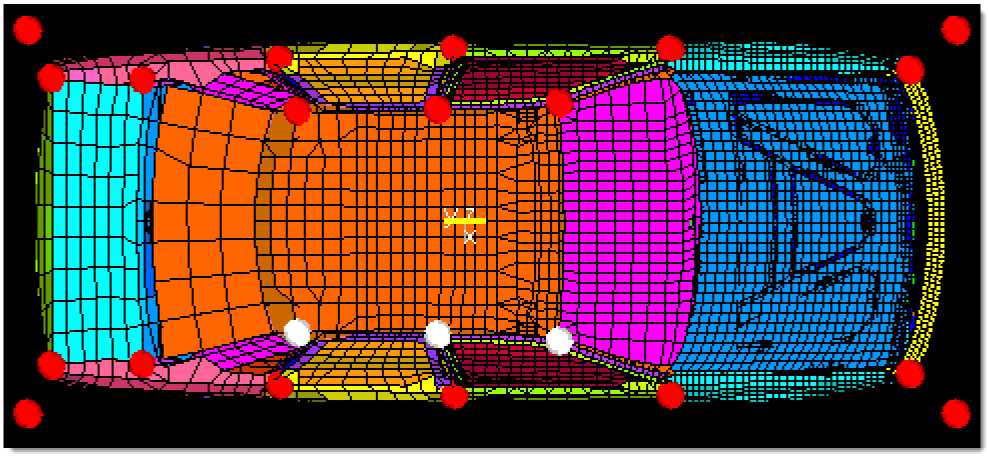


Figure 1742: System and 1-Plane Symmetry

The plane of symmetry is positioned at the origin of the system and perpendicular to the x-axis. The perturbations applied to handles on one side of the plane of symmetry will be mirrored on to the other side.

Now when you perform a morphing operation you only need to move the handles on one side of the plane of symmetry. If you have the symmetry links checkbox activated, HyperMorph automatically applies the handle movements to the handles on the other side of the plane of symmetry through the symmetry link. As a result, the model maintains symmetry across the symmetry plane.



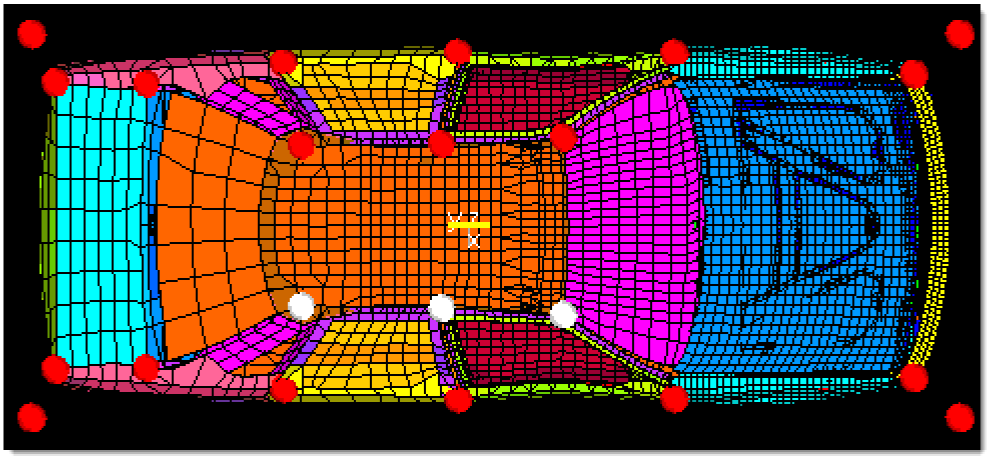


Figure 1743: Using 1-Plane Symmetry

Three handles on the right hand side of the roof are selected and moved towards the centerline. HyperMorph automatically moves the corresponding nodes on the left hand side of the roof in a symmetric fashion.

Reduce 3D to 2D using Linear Symmetry

Use linear symmetry to apply morphs to the model in such a way that the model is essentially reduced to two dimensions.

1. From the Tools page, click **HyperMorph**.
2. Click **Systems**.
3. Create a system with the x-axis pointing along the dimension to be reduced.
4. Click **return** to go to the HyperMorph module.
5. Click **Symmetry**.
6. Select the **create** subpanel.
7. In the name field, enter a name.
8. Select the global domain icon.
9. Switch the selector from 1 plane to **linear**.

10. Select the system you created.
11. Select x-axis as the axis to align the symmetry.
12. Click **create**.

A linear symmetry is created along the x-axis of the system. The icon for a linear symmetry consists of two parallel lines along the dimension to be reduced. The origin of the system is irrelevant. Now each handle acts on the mesh as if it were a line extending along the system x-axis. If two handles lie along a line parallel to the system x-axis, they will be linked through symmetry. When you move a handle, all the nodes and handles with the same y and z coordinates will move along with it.

Note: Since linear is a non-reflective type of symmetry, leaving symlinks unchecked will not prevent the handles from having linear influences. However, it will stop movements from one handle from being applied to others that are linked via the symmetry. If you wish to turn the symmetry off for a given morphing operation, make the symmetry inactive in the Morph Options panel.

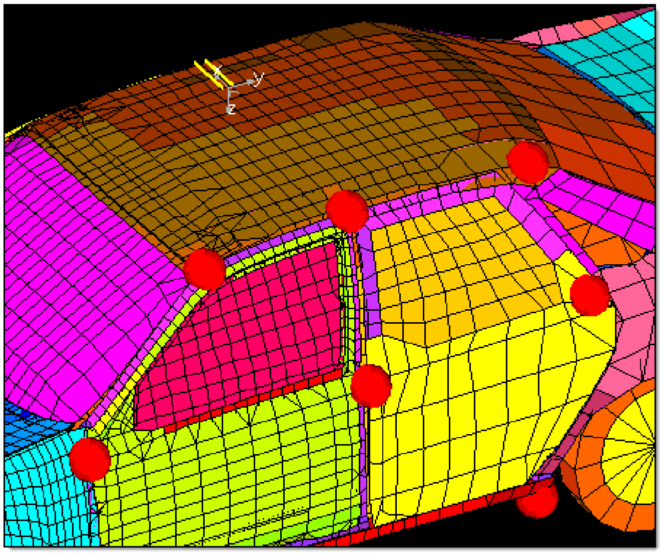


Figure 1744: System and Linear Symmetry

The linear symmetry icon consists of two parallel lines along the system x-axis. Note that the placement of a linear symmetry system does not matter, the effect of the linear symmetry system is determined only by the direction of the x-axis.

Applying a linear symmetry is very useful for making profile changes to a space frame model. It does not matter where the handles are placed along the x-axis, greatly simplifying the model set up. You only need to look at the model from one view to set up the handles and to morph the model. For models with a large number of elements this can save a great deal of time.

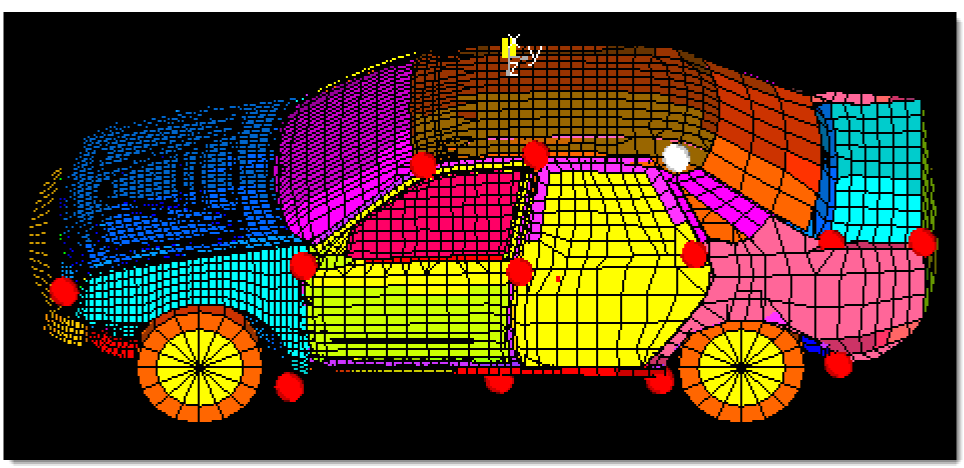


Figure 1745: Using Linear Symmetry

The handle on the rear part of the roof is selected and the entire rear portion of the roof is morphed along with it. With linear symmetry you only need to place handles on one side of the model to affect the entire profile.

2. Click **Systems**.
3. Create a system with the x-axis pointing along the dimension to be retained.
4. Click **return** to go to the **HyperMorph** module.
5. Click **Symmetry**.
6. Select the **create** subpanel.
7. In the name field, enter a name.
8. Select the global domain icon.
9. Switch the selector from linear to **planar**.
10. Select the system you created.
11. Select the x-axis as the axis to align the symmetry.
12. Click **create**.
13. Click **return** to go to the **HyperMorph** module.
14. Click **Symmetry**.
15. Select the **update by domain** subpanel.
16. Select the global domain.
17. Select the planar symmetry.
18. Click **update**.

A planar symmetry is created and the other two symmetries from the global domain are removed. You are allowed to have any number of symmetries associated with a domain and all will apply, but combining linear and planar symmetry in the same direction results in an unrealistic situation and poor influence calculations.

The planar symmetry icon is displayed as a filled-in rectangle perpendicular to the system x-axis. Now each handle acts on the mesh as if it were a plane perpendicular to the x-axis. If two handles lie in a plane perpendicular to the system x-axis, they will be linked through symmetry. When you move a handle, all the nodes and handles with the same x coordinates will move along with it.

Note: Since planar is a non-reflective type of symmetry, leaving symlinks unchecked will not prevent the handles from having linear influences. However, it will stop movements from one handle from being applied to others. If you wish to turn the symmetry off for a given morphing operation, make the symmetry inactive in the Morph Options panel.

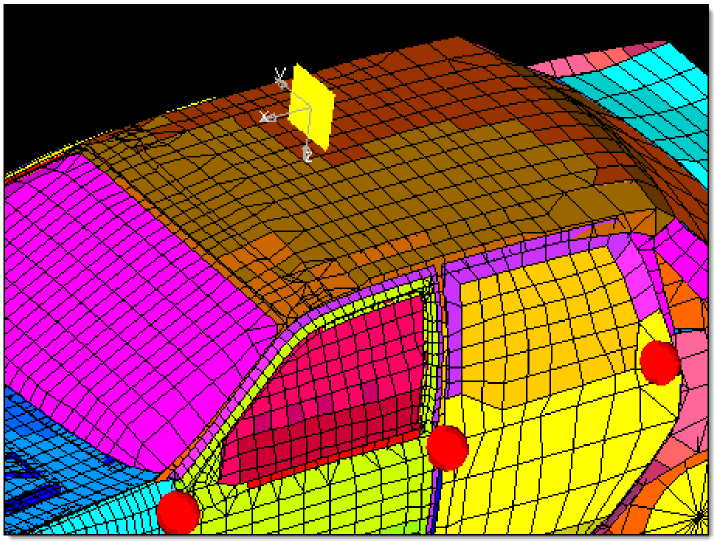
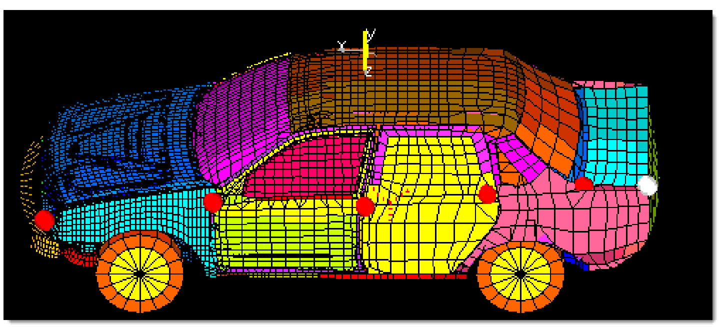
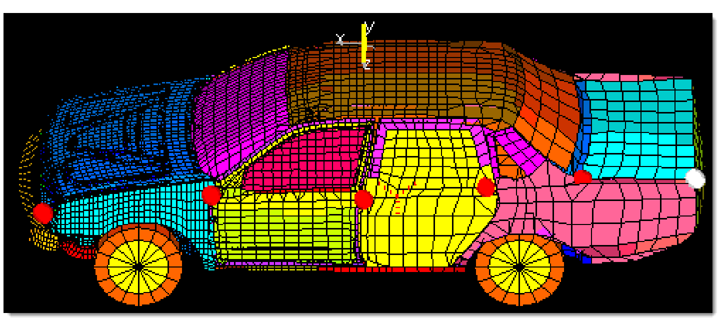


Figure 1746: System and Planar symmetry

The planar symmetry icon is a plane perpendicular to the system x-axis. Note that the placement of a planar symmetry system does not matter, the effect of the planar symmetry system is determined only by the direction of the x-axis.

Applying a planar symmetry greatly simplifies a model. Essentially, it reduces the model to a lying along single axis. This symmetry type is very useful for changing dimensions along one axis through the entire model.





Shell Model Strategies using Local Domains

Shell models are models that are made up primarily of shell elements, namely, quads, and trias. In general, a shell model represents many parts, each with numerous features such as holes and edges, and connected together using 1D elements such as bars and rigids.

HyperMorph is designed to make it easy to change the size and shapes of the shell model features. This is done using one of the following methods:

- Moving the handles on the part to new locations
- Moving the global handles around the parts to new locations
- Altering the radius or curvature of curved edges of the parts, or mapping the nodes of a part to line or surface data

For most models you only need to create 2D domains for the entire part, but you can also add a global domain and global handles for shape alterations of a general nature.

Manage Handles and Domains for Shell Models

Create handles and domains, divide your shell model, and group, split, merge domains.

Create Handles and Domains for Shell Models

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **create** subpanel.
4. Set the selector to **2D domains**.
5. Change the toggle to **all elements** or select all the elements in the model.
6. Click **create**.
   A 2D domain is created for each group of continuous shell elements. Parts joined by 1D or 3D elements are separated into different domains. If partition domains is checked, the 2D domains will be partitioned according to the settings selected in the partitioning subpanel of the Domains panel. Once partitioned, edge domains are placed around the 2D domains and handles are placed at the ends of the edge domains. All of this is automatic, but 1D and 3D elements will not be placed into 1D and 3D domains unless you set the selector to local domains instead of 2D domains. In many cases, the domains and handles will be generated where you want them to be. If not you can always add, edit, or delete the handles and domains to meet your needs.

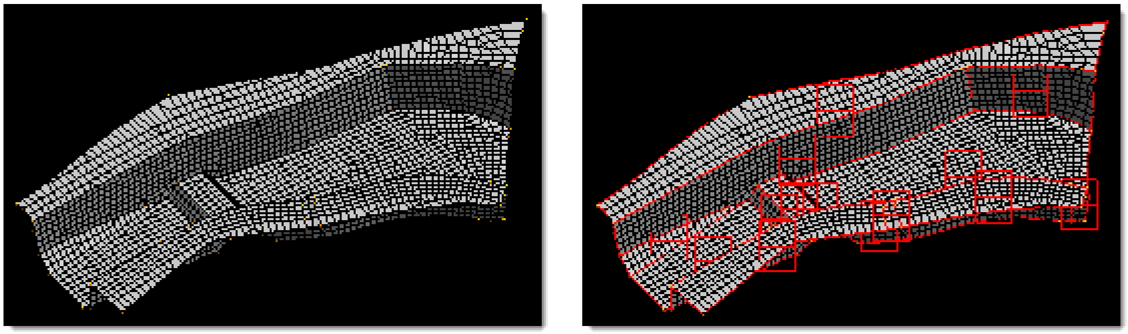


Figure 1748: Shell Model Partitioned into 2D Domains

7. If you wish to generate a global domain as well as local domains for your model with a single button click, either change the selector to **global and local** and click **create**, or to **auto functions** and click **generate**.

   In the case of the generate auto function, if there are any domains or handles in the model, HyperMorph will first ask if you want to delete all the current morphing entities. If you say "yes", or if there are no morphing entities in the model, HyperMorph automatically generates 1D, 2D, 3D, and edge domains for the entire model and a global domain and handles as well.

   For tria meshes which lack underlying geometry, both the node-based and element based partitioning algorithms may prove unsatisfactory. In these cases you may find it more effective to ignore curvature when partitioning. To accomplish this, go to the partitioning subpanel, select element based as the algorithm for tria/tetra meshes, and change the uppermost toggle from curvature based to angle based. You may also want to lower the domain angle to 30 degrees. HyperMorph will then only make partitions along edges in the model where the domain angle is exceeded. You can then go in and manually divide the 2D domains where the curvature breaks should go. This method is almost mandatory for meshes that began as first order meshes but were transformed into second order meshes. For these meshes, HyperMorph will detect a curvature break at every element along a curve if the midpoint nodes of the elements have not been modified to capture the curvature. The result will be a domain for every element on a curve which makes morphing impractical.

   Solving the influence coefficients for 2D domains which contain more than 20,000 elements can become very time consuming even though it is only done after domain editing and during morphing operations such as radius change and map to geom. In these cases you may want to divide the large domains into multiple domains or lower the limit for the large domain solver. The large domain solver limit can be found in the global subpanel of the Morph Options panel. However, even though influence calculations for large domains are more rapid, morphing using the large domain solver can be time consuming, and thus subdividing 2D domains can often be the best solution for efficient morphing.

Divide Shell Model

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.

3. Select the **create** subpanel.
4. Set the selector to **2D domains**.
5. Select the elements to be placed into a new 2D domain.
6. Click **create**.

When selecting the elements for the new domain you do not need to select only shell elements. HyperMorph automatically removes any other elements before creating the domain. It does not matter if the elements selected are already in a 2D domain. When the new domain is created, the elements are moved from the old domains to the new domain.

Handle influences need to be recalculated every time handles, domains, or symmetries are added, edited, or deleted. They are also recalculated during radius changes and geometry mapping. These calculations occur when you enter or leave any HyperMorph panel or when you leave the Delete panel. Thus, for models with large domains you will want to make all of your domain changes before exiting the Domains panel. HyperMorph only recalculates the handle influences for handles in regions that have been edited.

If the domains are not created exactly how you want them to be, you can edit them in the Domains panel. The create subpanel allows you to create new domains. The organize subpanel allows you to edit domains by adding and removing elements to or from a domain and by grouping domains together. The edit edges subpanel allows you to split, merge, and place handles along edge domains. It is suggested that you create and edit all the 2D domains, then create and edit the edge domains. This order works better since creating or editing 2D domains will result in the regeneration of the surrounding edge domains with the previous modifications to those edge domains being discarded.

Sometimes partitioning does not divide the mesh in the ways that would be most useful to you. Occasionally, elements end up in domains adjacent to where you want them or placed in their own domain. Partitioning is not an exact science, so some cleanup is sometimes required.

Organize Elements in Domains

Move elements from one domain to another.

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **organize** subpanel.
4. Change the selector to **add nodes/elems**.
5. Change the toggle to **local domain**.
6. Select the elements to be moved.
7. Select the target domain.
8. Click **organize**.

Elements are moved from the domain that they are currently in to the selected domain. HyperMorph also refreshes the edge domains around both domains as well as the edge domains at the interface. New handles may also be created during this process, and if retain handles is not checked, handles may be deleted. It is suggested that you keep retain handles unchecked unless you have created shapes for the model that use the handles on the domains that you are editing.

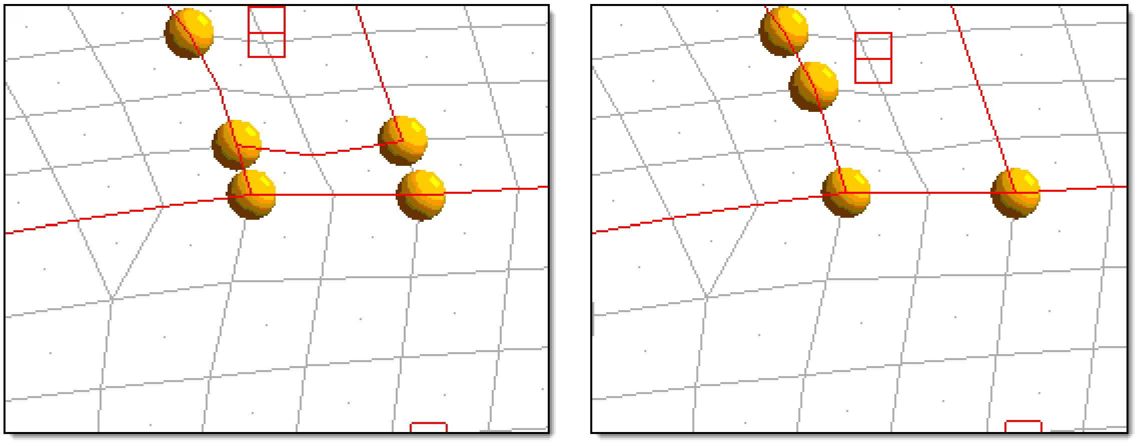


Figure 1749: Partitioning Problems

The model on the left shows problems that partitioning can encounter for some meshes. The model on the right has been corrected using the organize subpanel of the domains panel. For this example the retain handles option was left unchecked resulting in the elimination of handles that are no longer on the corners of the 2D domains. Note that the edge domains are always partitioned for any new domain and handles are placed at the end of the edge domains. For the example above, a handle was created in a new location due to the edge partitioning being different for the two domain configurations.

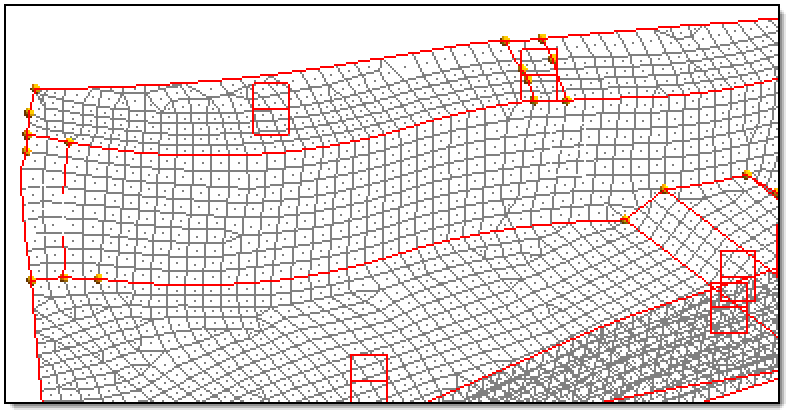
When you hold the mouse button down and the mouse is either over the icon for a 2D domain or over an element inside a domain, the edge domains surrounding the domain are highlighted as well. This allows you to better visualize the domain that you are selecting. The domain icon is placed at the centroid of the domain, and some domains can end up away from the elements of the domain and near other domain icons. Having the edges for the domain highlighted during selection is often necessary to tell which icon goes with which group of elements.

Group Domains

Group two or more domains together.

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **organize** subpanel.
4. Change the selector to **combine domains**.
5. Select the domains to be grouped together.
6. Click **organize**.

The selected domains are combined into a single domain and the surrounding domains and handles are updated.



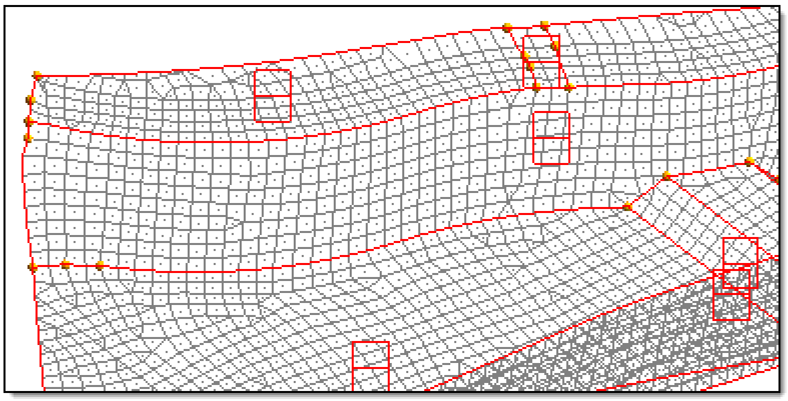


Figure 1750: Two Domains Organized into One

Edge domains are automatically partitioned when they are created. They are also updated whenever a change occurs for a domain of which they are on the edge. This is why any editing of the edge domains should come after the editing of the other domains. If you do your edge editing first, your changes may be erased when you edit the 2D domains.

Edge domains are used to make radius changes, so it is important to make sure that any radius in the model that you intend to change be captured correctly by edge domains. HyperMorph tries to partition edge domains where curvature begins and ends, but in some cases it may not identify the proper starting and ending points. You may need to correct this by hand.

Split Edge Domains

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Edit Edges** subpanel.
4. Change the selector to **split**.
5. Select an edge domain.
6. Select a node on that domain that is not on the edge.
7. Click **split**.

The selected edge domain is split into two edge domains at the selected node. A handle is created at the selected node.

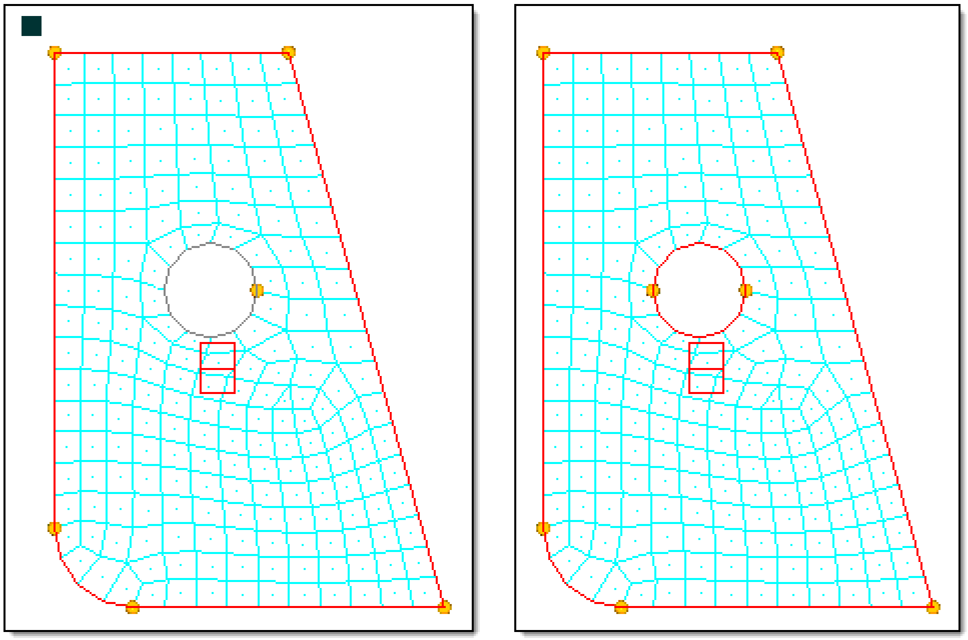


Figure 1751: Splitting an Edge Domain

A circular edge domain is divided into two half circles. A handle was created at the joint to allow you to manipulate the edges.

Merge Edge Domains

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Edit Edges** subpanel.
4. Change the selector to **merge**.
5. Select any number of connected edge domains.
6. Click **merge**.

The two edge domains are merged into one edge domain. This function only allows you to merge edge domains that lie end to end such that the resultant edge domain is a continuous series of nodes.

Note: You can also merge edge domains in the organize subpanel.

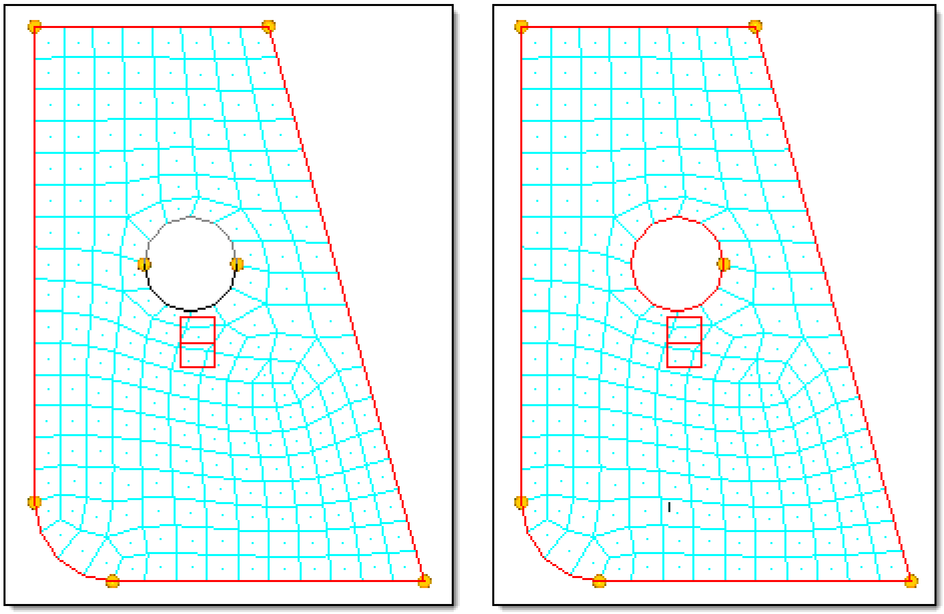


Figure 1752: Merge Two Edge Domains

The two half circles are merged into a single domain. Since retain handles was unchecked, the handle at the joint was deleted.

You may also create dependent handles along an edge domain. This feature is quite useful for saving time when you are changing the radius for the edge domain. If the domain containing the radius to be changed is very large you may find it more efficient to place dependent handles on the edge domains whose radii you wish the change before you go into the Morph panel.

Create Dependent Handles along Edge Domains

Create dependent handles on edge domains whose radii you wish the change.

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Edit Edges** subpanel.
4. Change the selector to **add handles**.
5. Select the domain(s).
6. Click **create**.

   The dependent handles are created on the selected edge domains. These handles are dependent on the independent handles to either side of them along the edge domain.

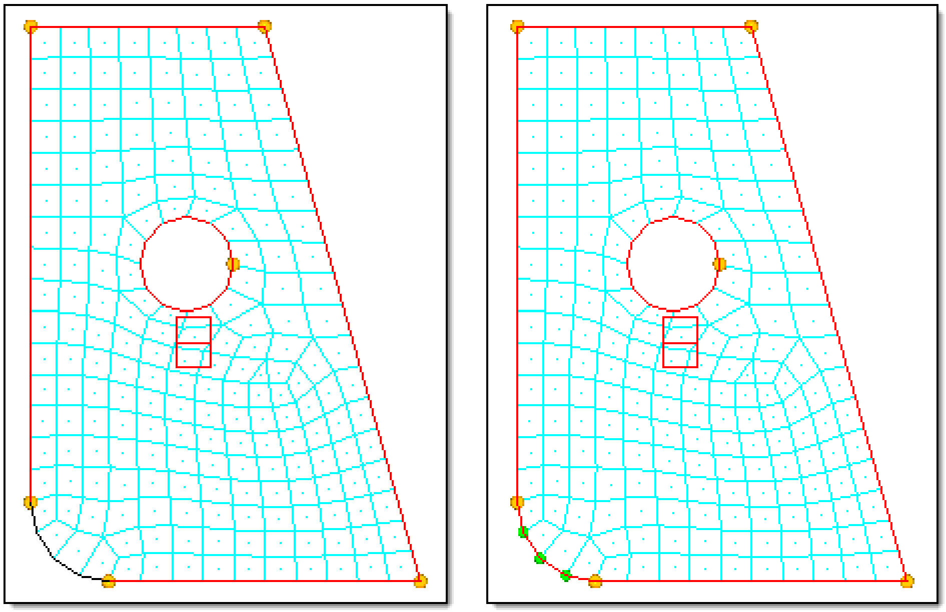


Figure 1753: Create Handles on Edge Domain

A dependent handle is created on each node of the edge domain.

Creating dependent handles in this way has two significant effects. The first is that since they are dependent, movements applied to any of the independent handles on the edge will be transparently applied to the dependent handles. It will be as if they were not there. Secondly, when you make a radius change to an edge domain that has a handle at each of its nodes, the influences do not need to be recalculated, which makes the radius change process much faster for large models.

7. When you are satisfied with your domains, click **return**.
HyperMorph calculates the influences for the handles and you are ready to begin morphing. During influence calculation you might run out of available memory. This generally happens when a given domain is too large and it contains too many handles. In these cases you should divide large domains, delete unnecessary handles, or lower the limit of the large domain solver.

Local Domains Moprhing

You can change the shape of a model with local domains and handles.

Moprhing can be accomplished using one or more of the following methods:

- Moving the local handles
- Changing a distance or angle
- Changing the radius, curvature, or arc angle of an edge domain
- Mapping nodes to a line, plane, surface, or mesh
- Using section mapping, line and surface difference, and element offset
- Using freehand morphing capabilities such as move nodes, record, and sculpting

You can move handles in the Morph panel, Move Handles subpanel using the following options:

interactive

Move handles interactively by dragging the mouse across the screen. You select an entity such as a vector, line, plane, surface, or domains, to orient the mouse location in 3D space, and move a handle by clicking on it and dragging it to a new location. Interactive morphing is most effective for visualizing how the mesh will react when a handle is moved and for making approximate shape changes. If you want to move a handle a specific distance or to a specific position, it is better to use a non-interactive option.

translate

Translate handles along a vector or element normals.

rotate

Rotate handles about an axis.

move to XYZ

Position handles at specific XYZ locations or place them on lines, surfaces, or another mesh.

move to node, move to point

Position handles at specific node or point locations, or place them on lines, surfaces, or another mesh.

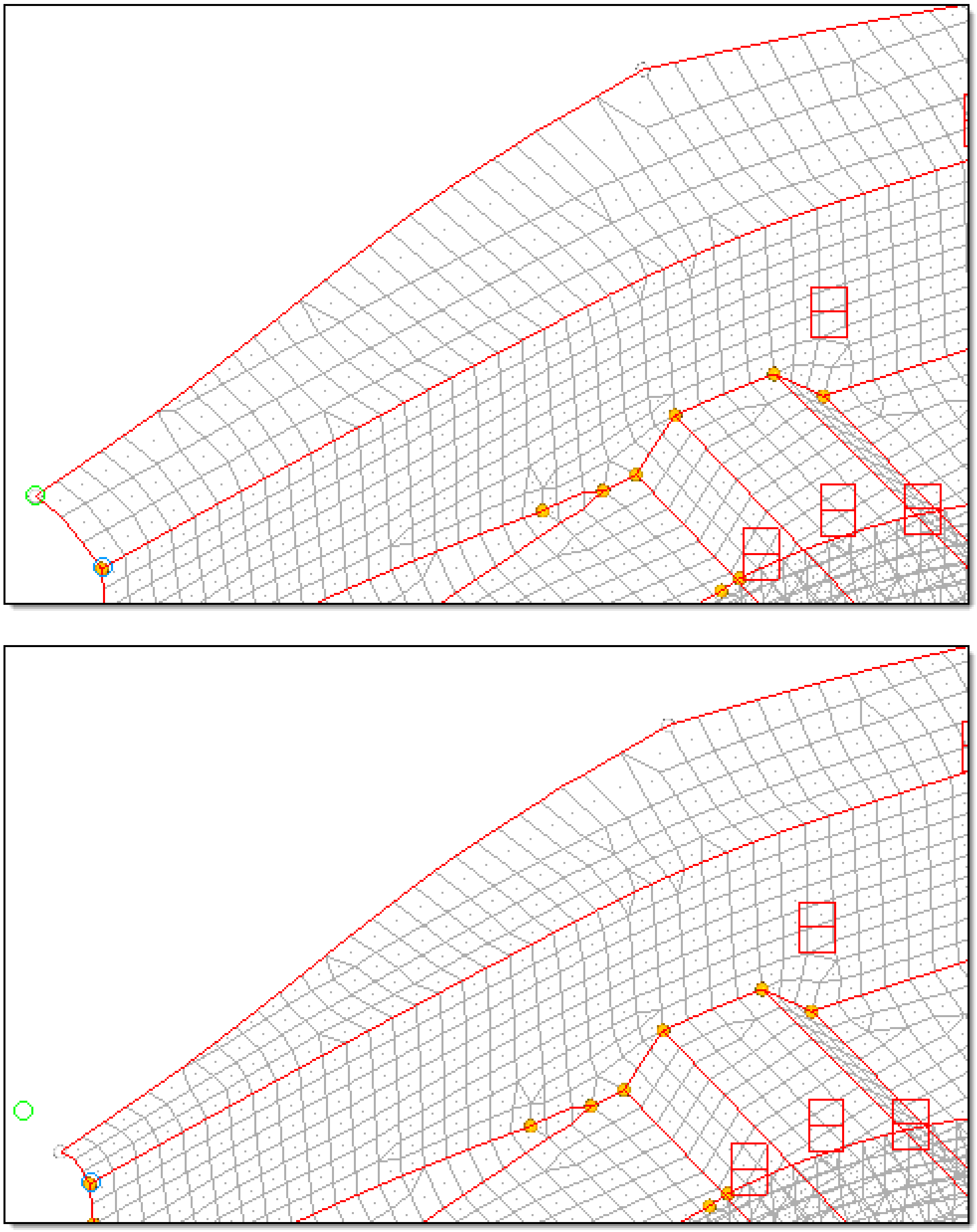


Figure 1754: Morph by Translating Handles along Edge

By selecting the two handles along the edge of the flange and translating them along a vector defined at the end of the section (green and blue nodes), the length of the flange is reduced.

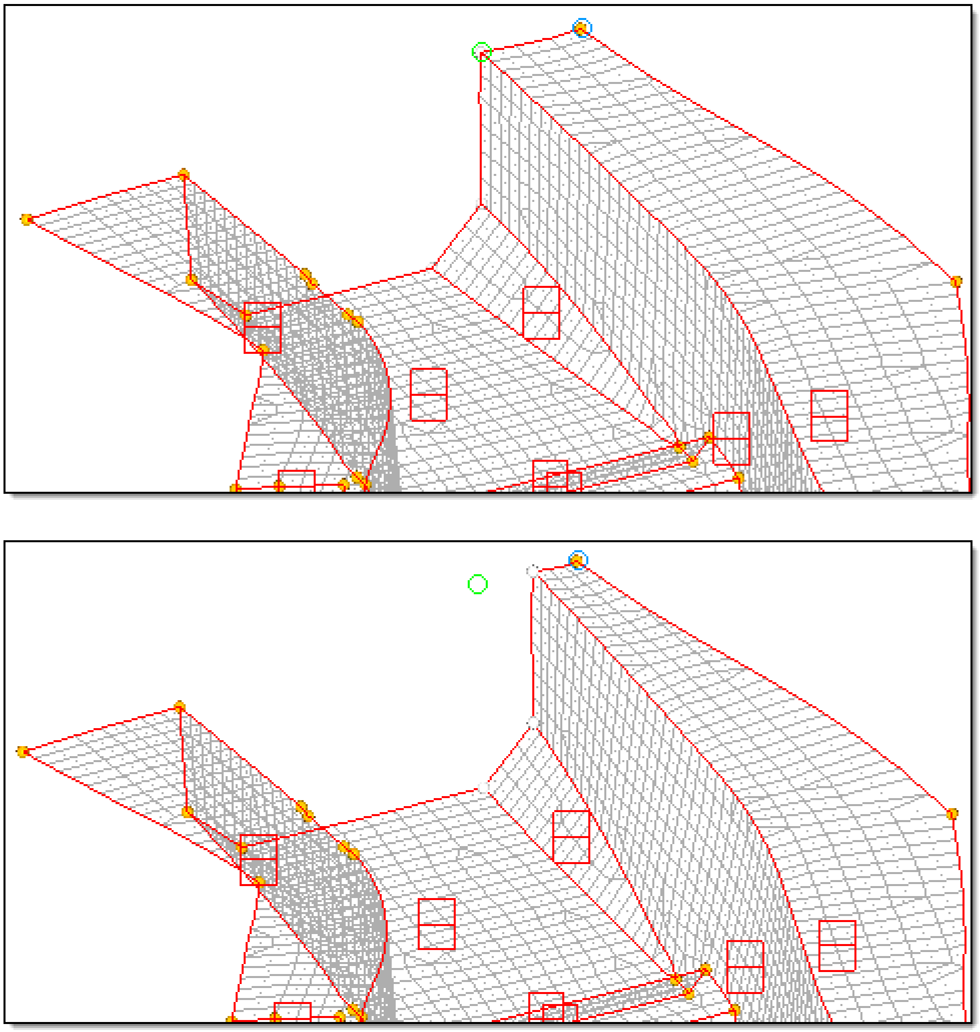


Figure 1755: Morph by Translating Handles along Vector

By selecting the three handles and translating them along a vector defined at the end of the section, the width of the channel is increased.

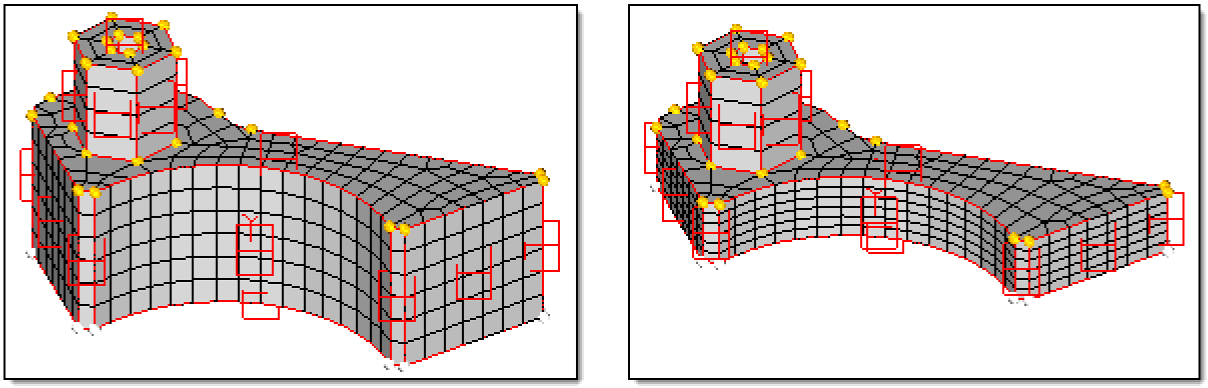


Figure 1756: Morph by Translating Handles at Bottom of Part

By selecting the handles at the bottom of the part and translating them upwards, the thickness of the lower section is reduced.

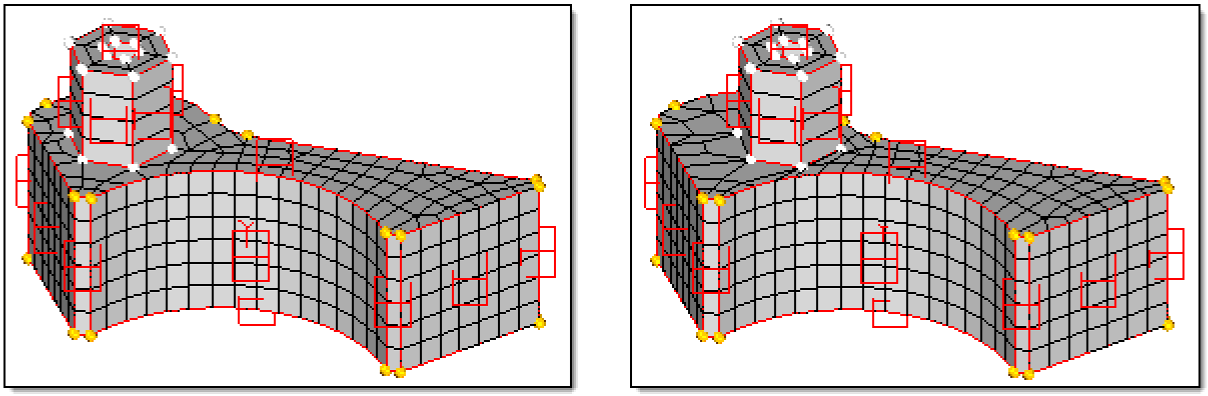


Figure 1757: Morph by Translating Handles around Bolt Boss

By selecting all the handles around the bolt boss and translating them horizontally, the position of the bolt boss is modified.

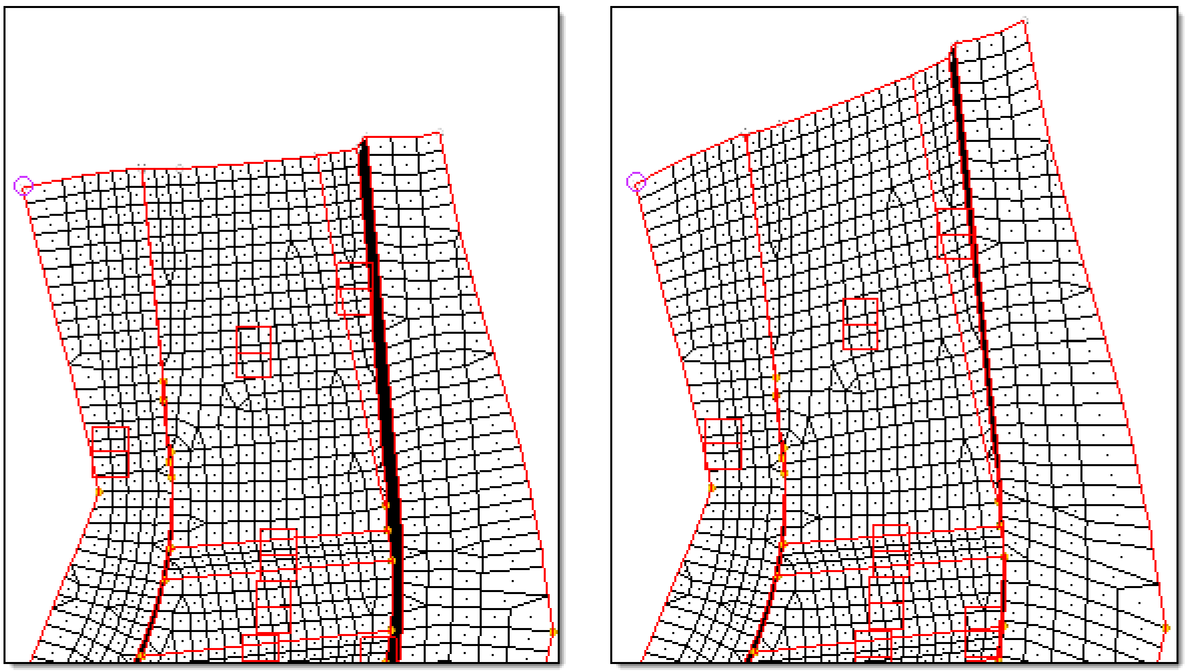


Figure 1758: Morph by Rotating Handles at End of Section - Constant

By selecting all the handles at the end of the section and rotating them about a point (violet node), the end angle of the section is modified.

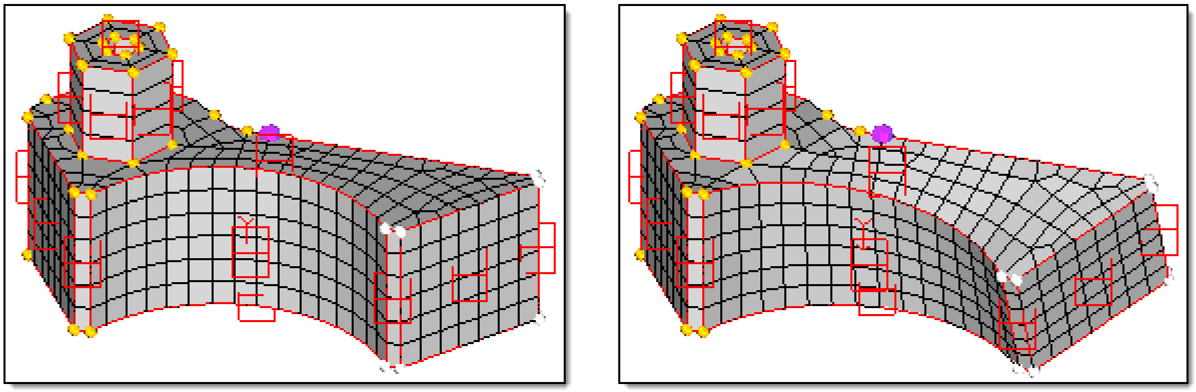


Figure 1759: Morph by Rotating Handles at Right End of Block - Constant

The right end of the block is given a constant rotation.

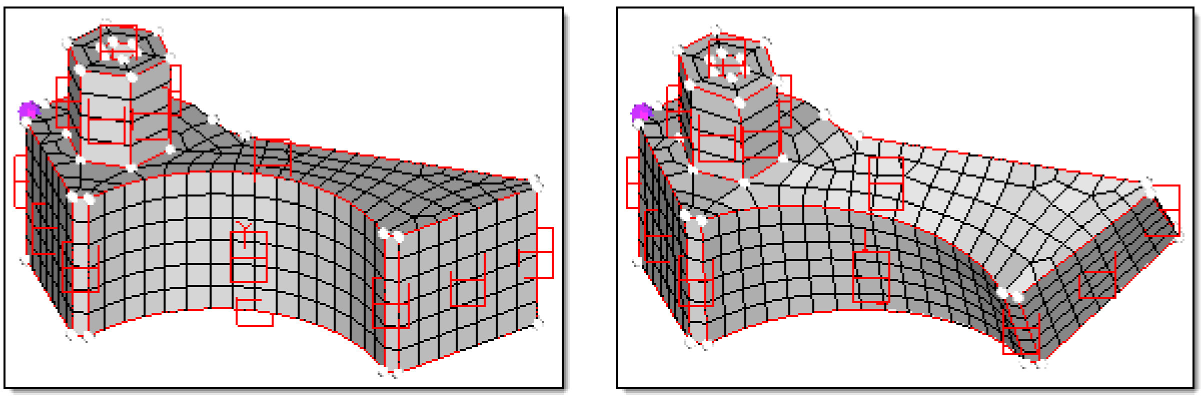


Figure 1760: Morph by Rotating Handles for Entire Block - Linear

The entire block is given a linear rotation. Note how the magnitude of the twist increases linearly with the distance from the base (purple) node.

When applying handle perturbations to your model, it is important to note that the nodes in the model follow the movements of the handles according to the influence coefficients. This concept comes into play when you are using the rotate function. After rotating handles you may find areas in the model, particularly those defined by curved edges, that are not rotated the same as the neighboring handles. This is because the nodes have followed the handles instead of being rotated about the axis. To correct this situation, select the **true rotation** checkbox. This will cause the nodes to be rotated as well as the handles with the amount of rotation being equal to the influence coefficient. Although it could be argued that true rotation is the "correct" way to morph via rotation of the handles, not all morphing applications are best done using true rotation.

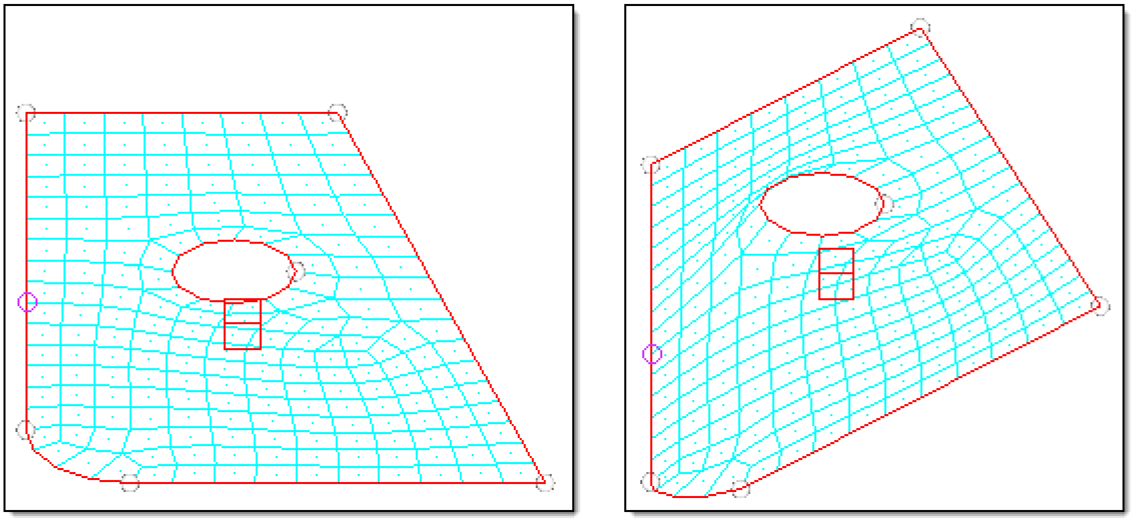


Figure 1761: Morph by Rotation Handles - Normal
Although the highlighted handles are rotated, the circle at the center of the model remains on the same plane as before.

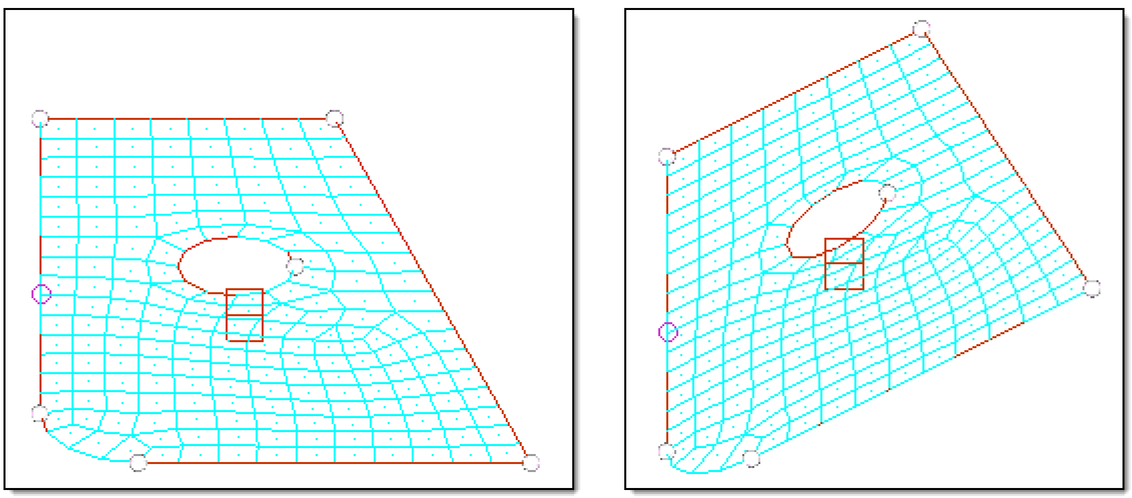


Figure 1762: Morph by Rotating Handles - True Rotation
During "true rotation" the nodes rotate along with the handles.

While morphing a model, the following message may be displayed: "Some handles selected are dependent on others. Would you like to ignore dependencies for this operation?". This occurs when both a dependent handle and the handle on which it is dependent are selected to be morphed. If you click **yes** the given perturbation is applied to each handle and the dependent handles are not given an additional perturbation inherited from another handle. If you click **no**, the given perturbation and any inherited perturbation is applied to each dependent node. For most cases you will want to click **yes**.

The Morph panel, Alter Dimensions subpanel allows you to change one of the parameters in the model, such as the distance between nodes, the angle between nodes, or the radius or curvature of an edge domain. The basic concept is as follows:

- Select two nodes (node a and node b).
- Select handles corresponding to those nodes.

The handles selected are the ones that will move to make the distance between node a and node b, or angle with a vertex selected, equal the specified value. You must select at least one handle for each end and the handle may be coincident with one of the nodes. For solid models, controlling a particular dimension often involves moving more than one handle for each end.

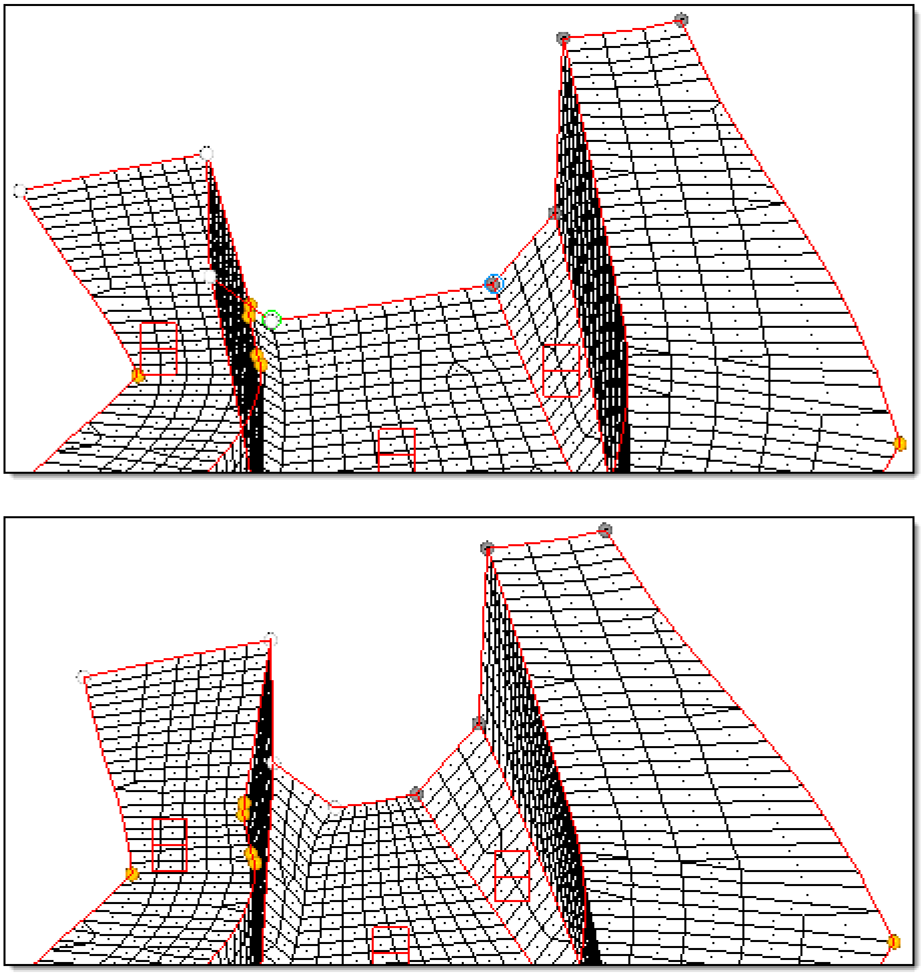


Figure 1763: Morph by Altering Dimensions using Channel's Bottom Width - Distance

By selecting the width of the bottom of the channel as the desired distance to alter (green and blue nodes) and by selecting the handles on the left (highlighted) to follow the green node and the handles on the right (shown as gray) to follow the blue node, the width of the bottom of the channel can be changed from 60 to 30 with the rest of the channel changing along with it.

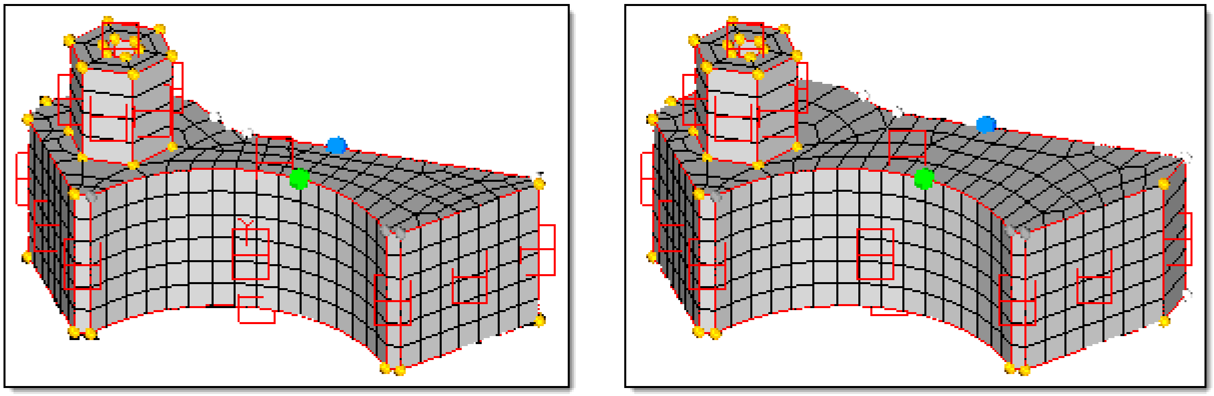


Figure 1764: Morph by Altering Dimensions using Block Thickness - Distance

By selecting the thickness of the block as the desired distance to alter (green and blue nodes) and by selecting the handles on the radius (shown as gray) to follow the green node and the handles on the back face (highlighted) to follow the blue node, the thickness of the block between the radius and the back face is altered from 15 to 25 by moving the entire back face.

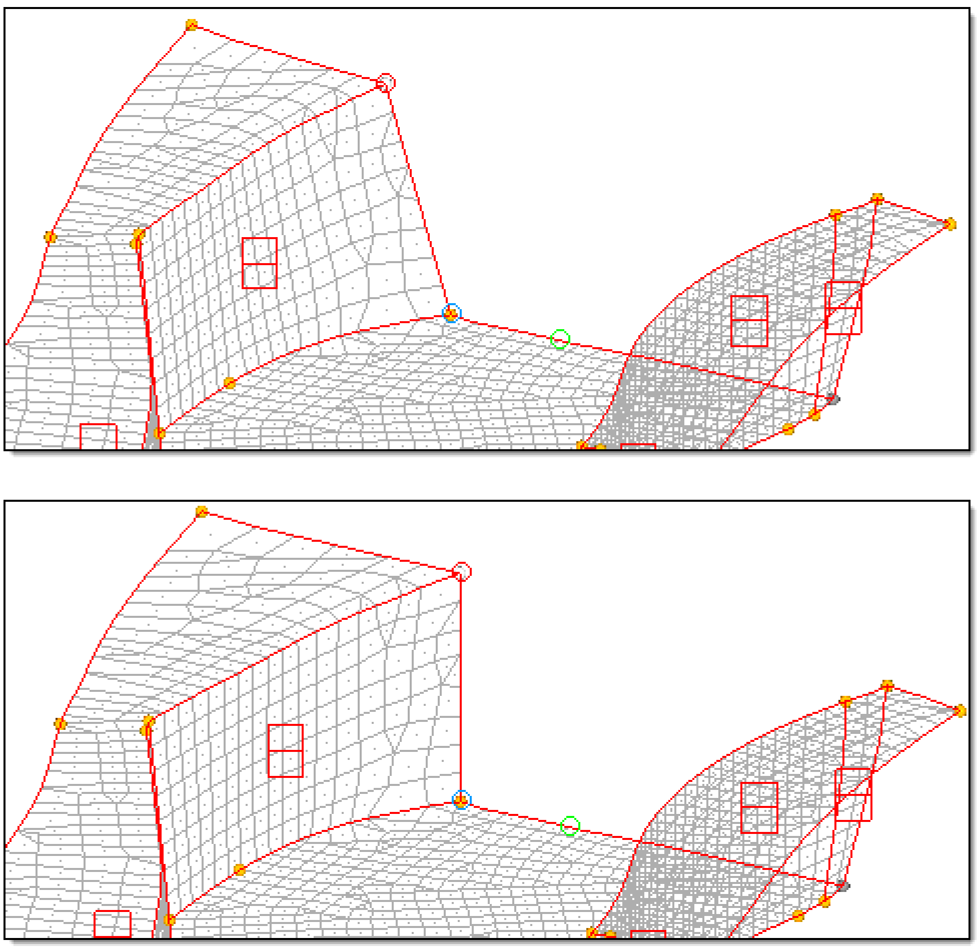


Figure 1765: Morph by Altering Dimensions using Angle of Channel's Left Side - Angle

By selecting the angle of the left side of the channel (green, blue, and red nodes) and by selecting the handle at the bottom right of the channel (shown as gray) to follow the green node and the handle at the red node (highlighted) to follow the red node, the angle of the left side of the section is changed from 110 degrees to 90 degrees.

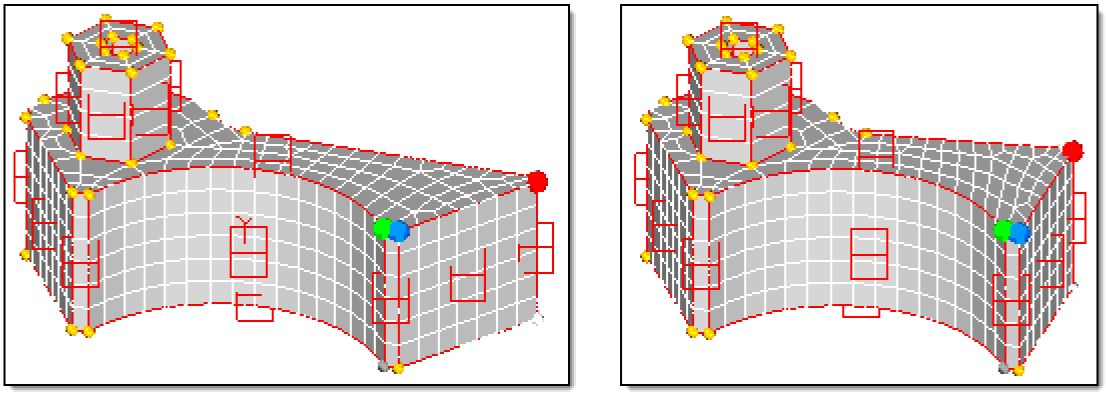


Figure 1766: Morph by Altering Dimensions using Angle between Two Faces of Block - Angle

By selecting the angle between two faces of the block (green, blue, and red nodes) and by selecting the handles at and directly below the green node (shown as gray) to follow the green node and the handles at, near, and below the red node (highlighted) to follow the red node, the angle is altered from 126 degrees to 90 degrees.

The radius, curvature, and arc angle options are used as follows. You select any number of curved edge or 2D domains, select the **center calculation and style** options, set the new radius, curvature multiplication, or arc angle factor for them, and click **morph**. All the domains are changed simultaneously.

> **Note:** The curvature tool scales your radius by a factor rather than a set radius, so if you want to change a radius from 5.0 to 8.0, you need to set the curve ratio to 1.6. The curvature tool is intended for domains that do not have constant curvature. Making the bias factor retroactive does not work for radius changes.

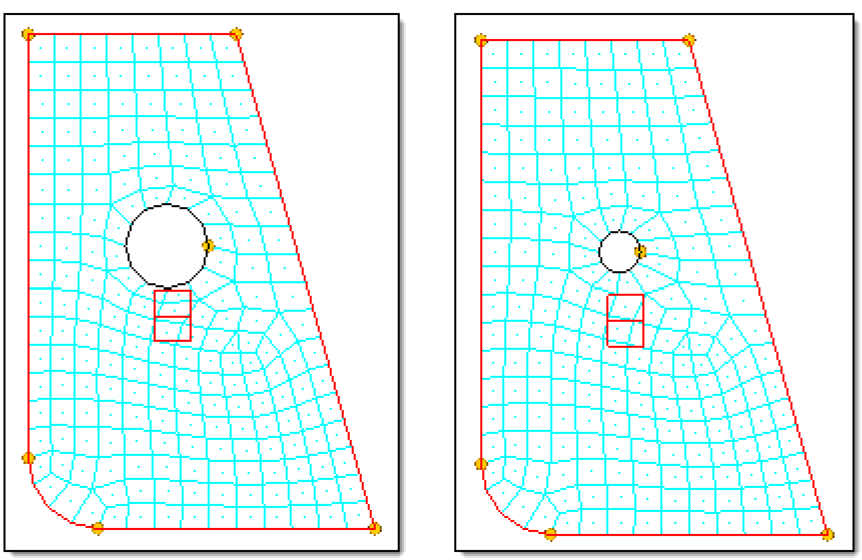


Figure 1767: Morph by Altering Dimensions - Radius and Center

By selecting the edge domain around the edge of the hole, the radius is changed from 3 to 1.5.

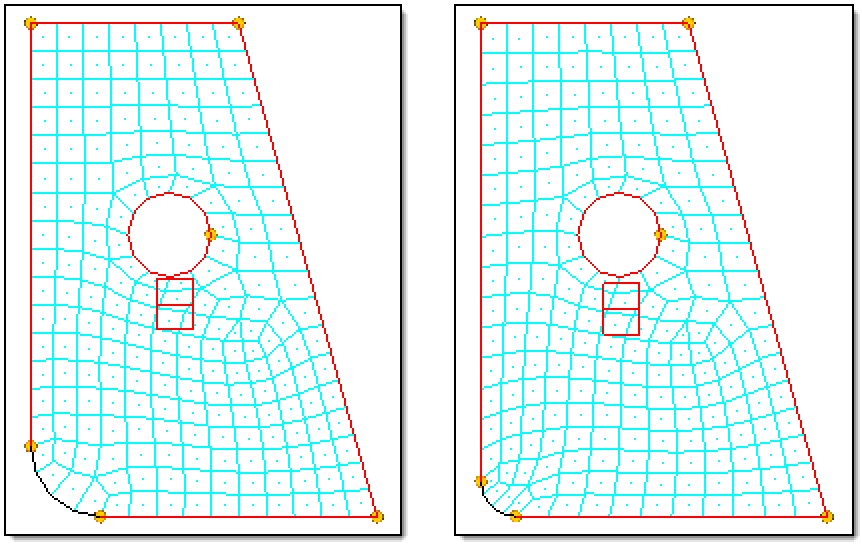


Figure 1768: Morph by Altering Dimensions - Radius and Fillet

By selecting the edge domain at the corner of the part and selecting the fillet option, the radius is changed from 5 to 2.5 and kept in line with the edges at either end.

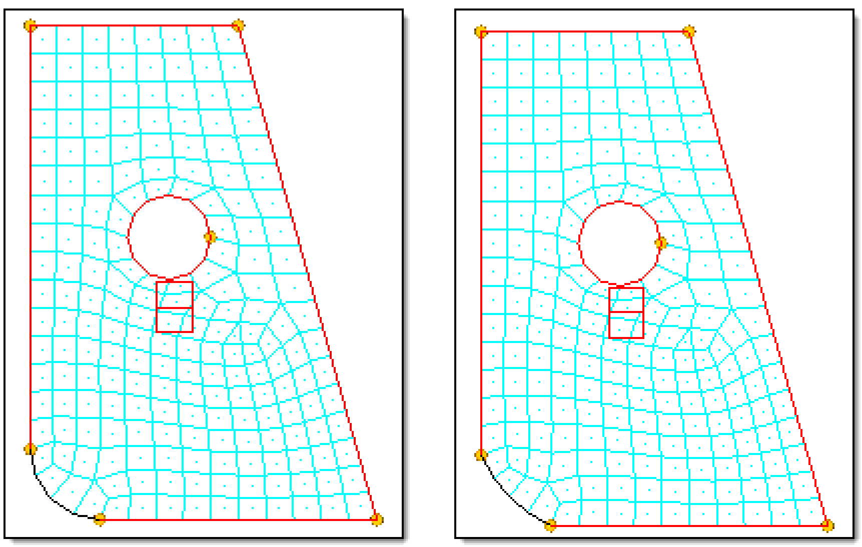


Figure 1769: Morph by Altering Dimensions - Radius and Hold Ends

By selecting the edge domain at the corner of the part and selecting the hold ends option, the radius is changed from 5 to 10 with the ends held in place.

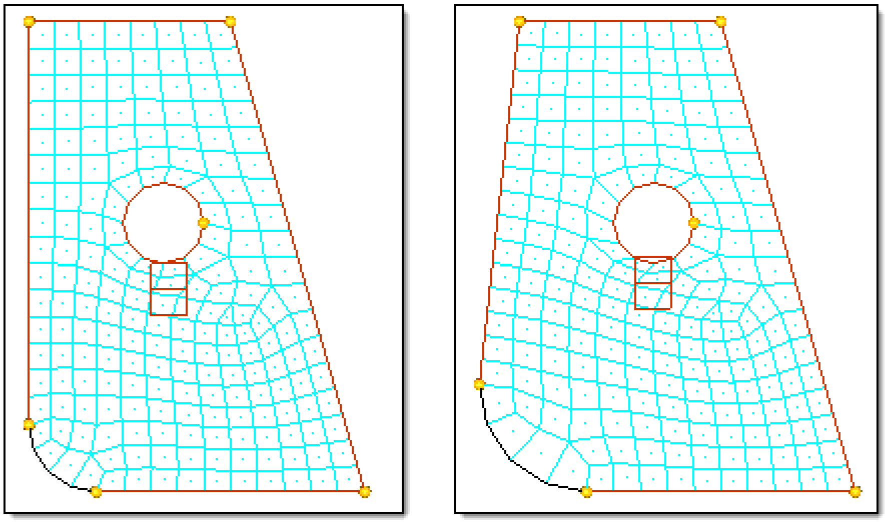


Figure 1770: Morph by Altering Dimensions - Radius and Hold End

By selecting the edge domain at the corner of the part, selecting the hold end option, and selecting a node at the end of the edge domain, the radius is changed from 5 to 8 while the held end remains in place.

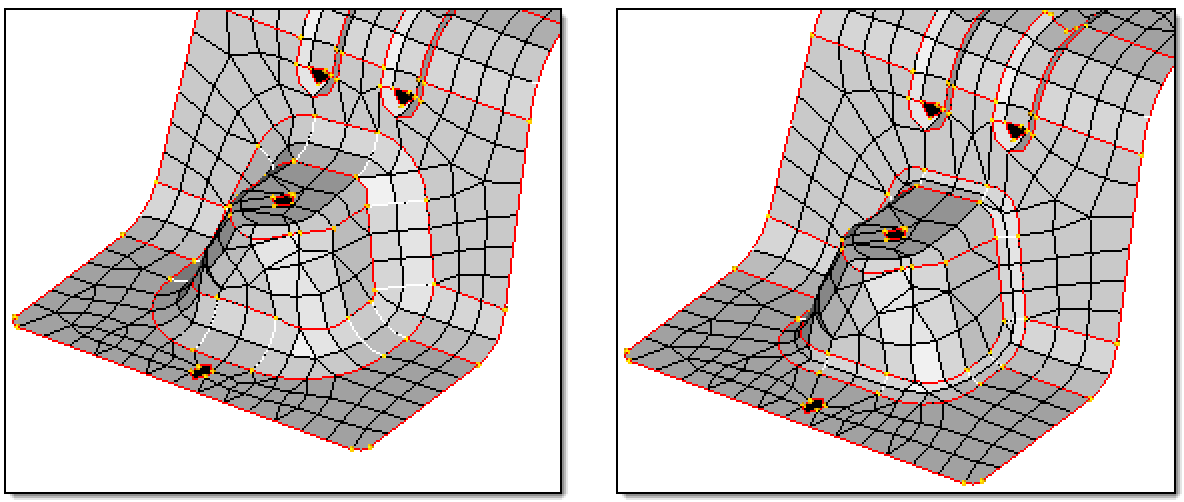


Figure 1771: Morph by Altering Dimensions - Radius and Fillet

By selecting all of the edge domains that form the fillet between the flat sections and the round section and changing them simultaneously, the fillet is reduced from 20 to 8.

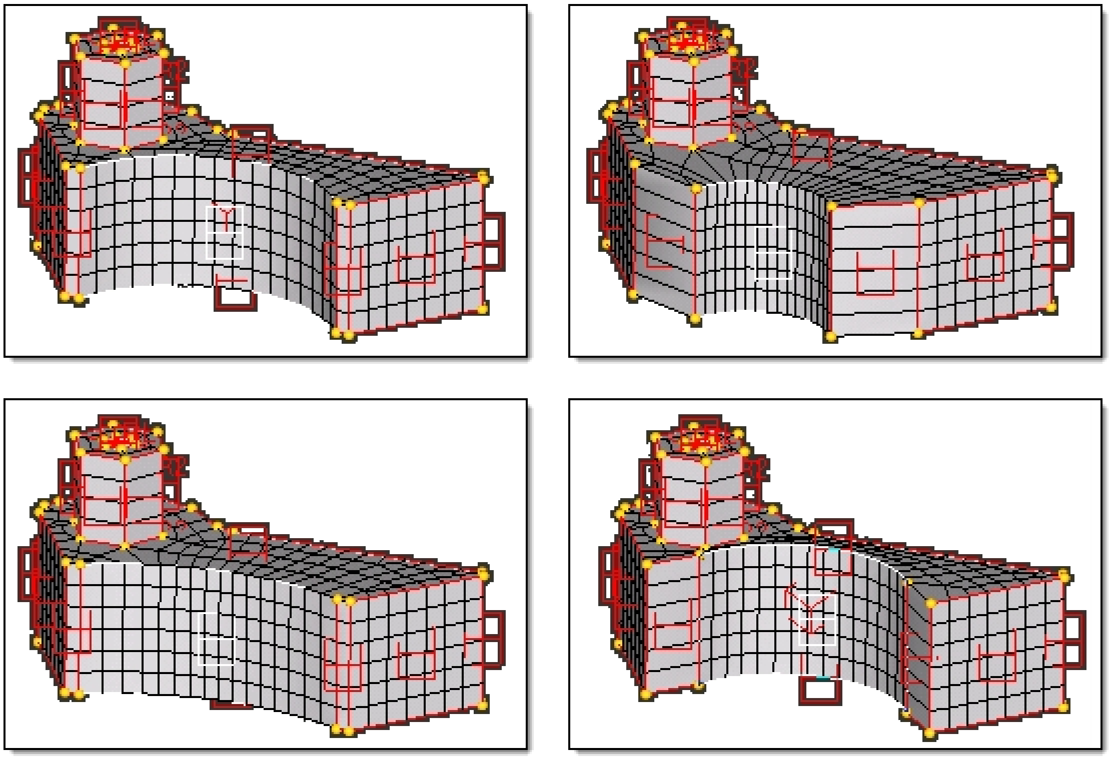


Figure 1772: Morph by Altering Dimensions - Radius

The radius is changed in three different ways. At the top right, the hold center option is used. At the lower left, the hold ends option is used. At the lower right, the fillet option is used. In all cases, both the top and bottom edge domains were selected as well as the 2D domain and the by normals option was used for center calculation. This option will directly calculate the radii for the nodes on the 2D domain instead of inferring them from the edge domains which makes this approach more accurate for 2D domains as well as more reliable for non-uniform meshes.

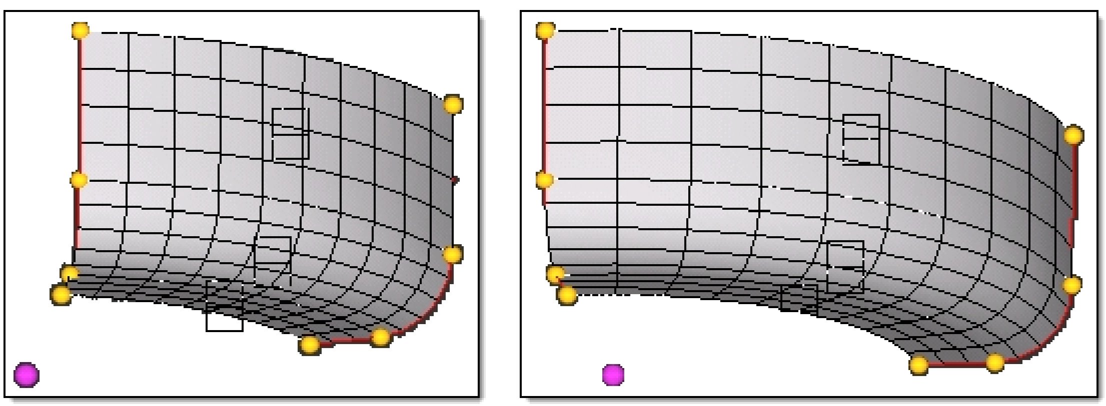


Figure 1773: Morph by Altering Dimensions - Arc Angle

The arc angle of the mesh is changed from 60 to 90 degrees using by axis (the vertical axis and violet base node) to calculate the center of curvature.

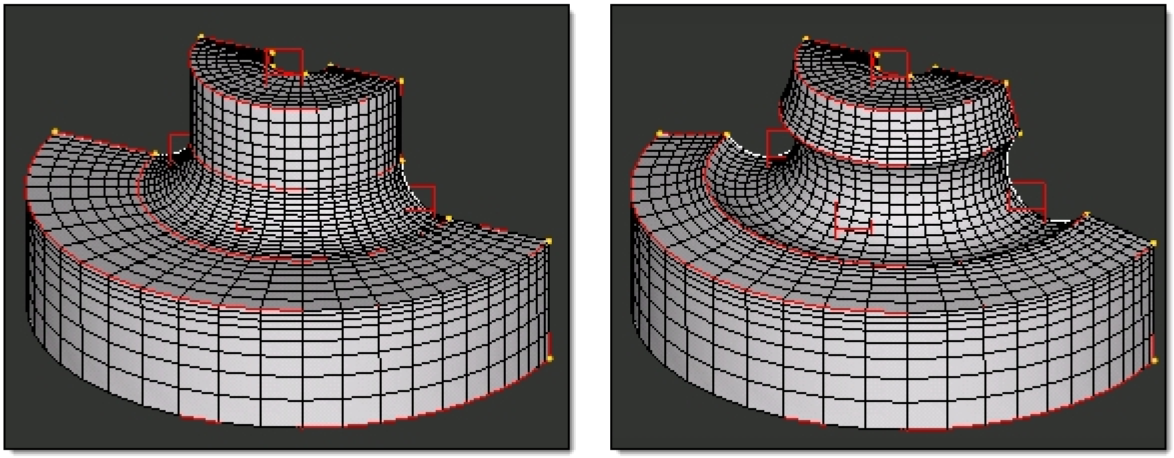


Figure 1774: Morph by Altering Dimensions - Arc Angle

The arc angle of the fillet is changed from 90 to 180 degrees using by normals to calculate the center of curvature.

Methods available for calculating the center of curvature for the selected domains include:

by normals (Default)
: Uses the element normals to approximate where the center of curvature is for each node in the selected domains. This method is not always accurate, but often gives good results for regular meshes.

by axis
: you may select an axis which will serve as the center of curvature.

by line
: You may select a line which will serve as the center of curvature.

by node
: You may select a node which will serve as the center of curvature.

by edges
: Uses the edge domains to calculate the center of curvature with the center lying in the plane of the edge domains. The symmetry option refers to how the morphing of the edge domains is applied to neighboring 2D domains. The auto-symmetry option was the default for HyperMorph prior to version 8.0. In 8.0 you may choose to turn off symmetry when using this option.

For auto-symmetry, the changes in the radii of the edge domains are applied to any 2D domain, depending on the number of edge domains you change for the 2D domains. If you change only one edge domain for a given 2D domain, the radius change will not be applied linearly across the 2D domain. If you change the radii of two edges for any given 2D domain, either a linear or planar temporary symmetry is created between the two edge domains for the 2D domain that will apply radius changes more linearly across the 2D domain. This works best if the mesh is regular. If you are changing only one edge for a 2D domain, you can increase the bias factor of any handles on an edge domain to yield a more even distribution.

Mapping an edge domain to a line or a 2D mesh to a plane, surface, or mesh is done using the Map to Geom panel. This option is very effective for fitting a mesh to new geometric data. When mapping a domain to a geometric feature, all the nodes in neighboring domains are stretched along with it, minimizing mesh distortion. You have several options for determining how the nodes for the mapped domain are placed on the geometry. When mapping an edge domain or node list the nodes can be moved normal to the line, along a vector to the line, or distributed along the full length of the line. When mapping a 2D domain or selection of nodes to a plane, surface, or mesh, the nodes can be moved normal to the target, normal to the elements of the 2D domain or selected nodes, or along a vector. If you wish to fit a mesh to a surface, there is no option to do this automatically, however, with multiple mapping operations, or using the **user control** option you can fit a 2D domain to a surface.

Furthermore, you have the option of creating a morph constraint between the nodes and the map target automatically after mapping. This constraint will allow you to do further morphing operations while maintaining the constrained nodes on the geometry.

The map to geom panel is also effective for solid model meshing. You can create a block of solid elements roughly in the shape of the geometry that you are trying to mesh, and then use map to surface to morph the faces of the block to the geometry.

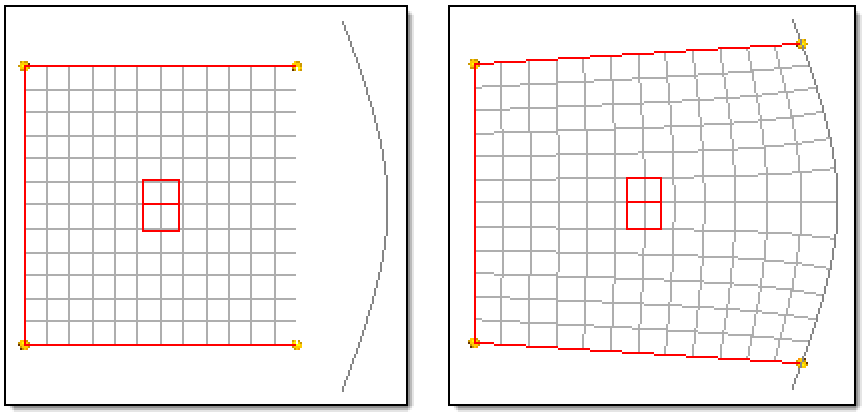


Figure 1775: Morph by Mapping to Line - Automap and Normal to Geom
The edge domain is mapped to a line by moving the nodes normal to the line.

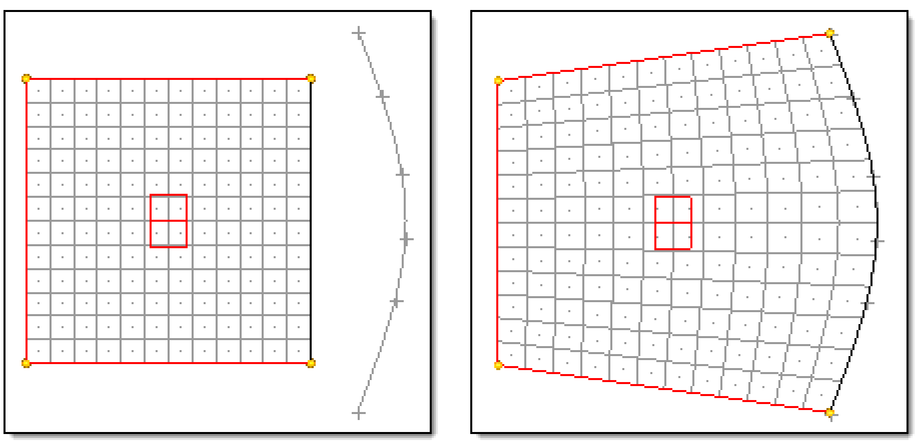


Figure 1776: Morph by Mapping to Line - Automap and Fit to Line
The edge domain is mapped to the line by fitting them along the line. Any proportional spacing between the nodes will be maintained after mapping.

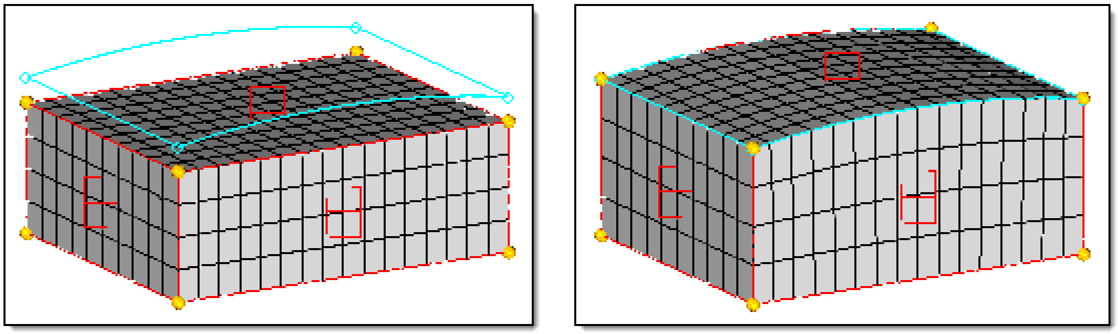


Figure 1777: Morph by Mapping to Surface

By selecting the 2D domain on the top of the solid block to be mapped to the surface, the entire solid block is morphed to match the surface.

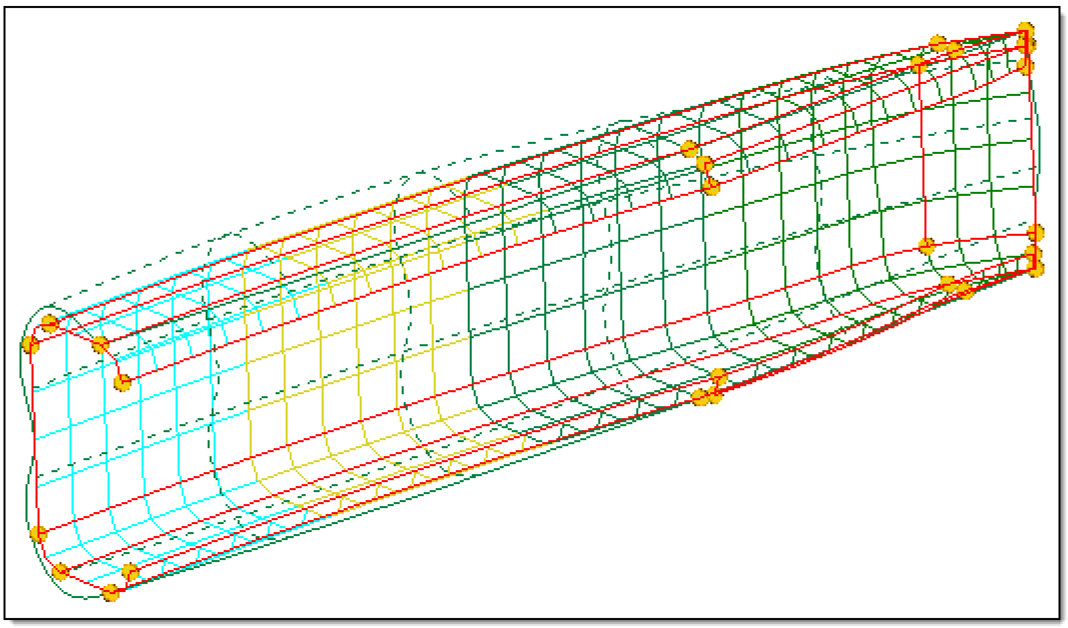


Figure 1778: Morph by Mapping to Surface

A rectangular C-section is mapped to a curved surface.

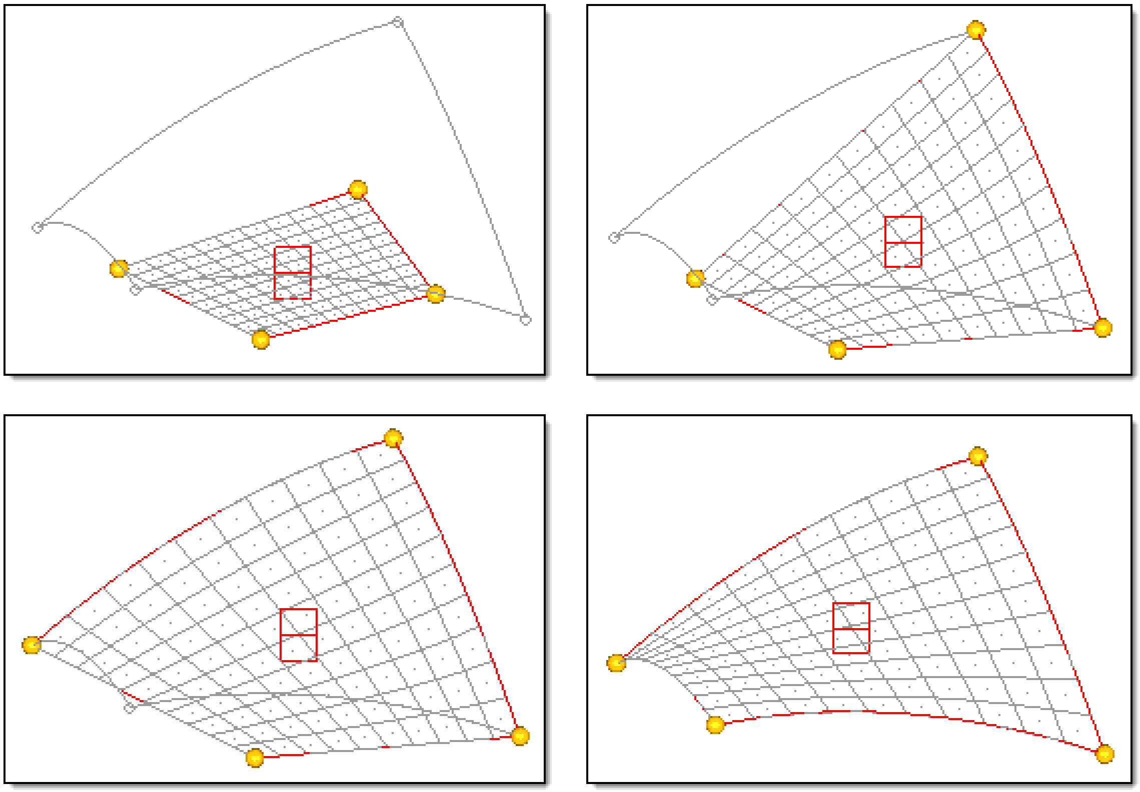


Figure 1779: Morph by Mapping to Surface - User Control Approach

The surface and 2D domain are selected and the user control button is clicked. This brings up a new panel which allows you to place handles or map edges prior to the surface mapping operation. One by one each edge domain is placed on one of the lines around the target surface using the fit to line option. This stretches the 2D domain to match the surface more closely than before. When the map button is clicked, the domain is the mapped to the surface, fitting it perfectly to the geometry.

Global Handles Morphing

Global handles are most effective when used to make general shape changes for a model, such as changing the basic shape of a model, stretching parts of a model, or making changes that involve the movement of many local handles.

There are three methods available for affecting the way global handles influence the model, the direct method, the hierarchical method and the mixed method. The default is the direct method, where the global handles move the nodes directly. In the hierarchical method, the global handles move the local handles which in turn move the nodes, but if any nodes lie outside of local domains they will be unaffected. In the mixed method, the hierarchical method is used for all nodes in local domains and the direct method is used for all other nodes. The hierarchical method maintains the shape of edge domains in the model, but if local handles are not evenly placed throughout the model, some parts will become distorted. The direct method gives you what you expect but often distorts the shape of the edge domains. For shell and solid models, better morphing is more likely to occur if you use the hierarchical method, and place local handles in areas where there is distortion.

Morph Constraints

Morph constraints are a powerful tool that can be used to restrict the movement of nodes during morphing operations.

The following types of constraints can be applied to any node: fixed, cluster, along vector, on plane, along line, on surface, and on elements. Whenever a handle is moved that influences a node which is constrained, the node is moved according to the handle perturbation and is then projected back onto the feature to which it is constrained. This allows the nodes to slide across vectors, lines, planes, surfaces, and meshes, to remain fixed when handles are moved, or to move as a cluster along with other nodes. You may also constrain nodes where handles are located which, in effect, constrains the handles. When a perturbation is applied to a constrained handle, the handle are moved along the constraint feature regardless of the applied perturbation. This means that if you apply a translation in the x direction on a handle that is constrained along a vector x - y = 0, the handle moves along both the x and y axes.

There are also morph constraints that can be applied to domains, such as the smooth constraint, which applies spline-based smoothing along the constrained edge domains, and model constraints, which allow you to set a given parametric target, such as length, angle and mass, and have HyperMorph adjust the model to meet that target. These constraints as well as bounded and set distance options for the node constraints are described more fully in the panel help.

Morph constraints can be very useful for morphing a mesh that has been mapped to, projected to, or created upon a surface. Note that the map to geom operation allows you to have a morph constraint automatically created after mapping. Once you have done so, the nodes will remain on the surface during morphing operations.

Note: Although morph constraints can keep nodes on a curved line or surface during morphing operations, when morphs are saved as shapes and then turned into shape variables for optimization, the nodes will not stay on the line or surface during optimization. This is because optimization is a linear process and the shapes will be treated as linear, meaning that the nodes will move directly from their original point to their maximally perturbed point without moving along any constraint.

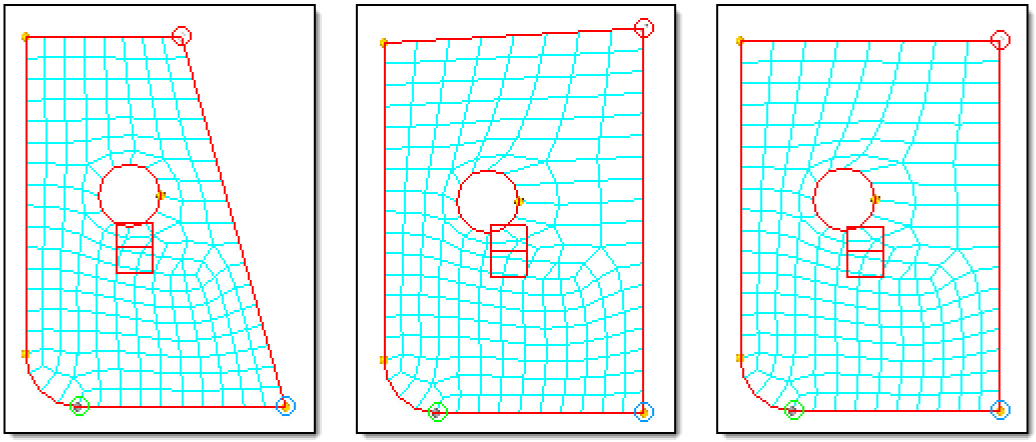


Figure 1780: Control Handle Positions with Morph Constraints

The angle of the lower right corner is changed from 74 to 90 degrees using the alter dimensions (angle) operation. The middle frame shows the result with no constraints. The frame on the right shows the result with the node in the upper right corner constrained to move along a vector that lies along the top edge.

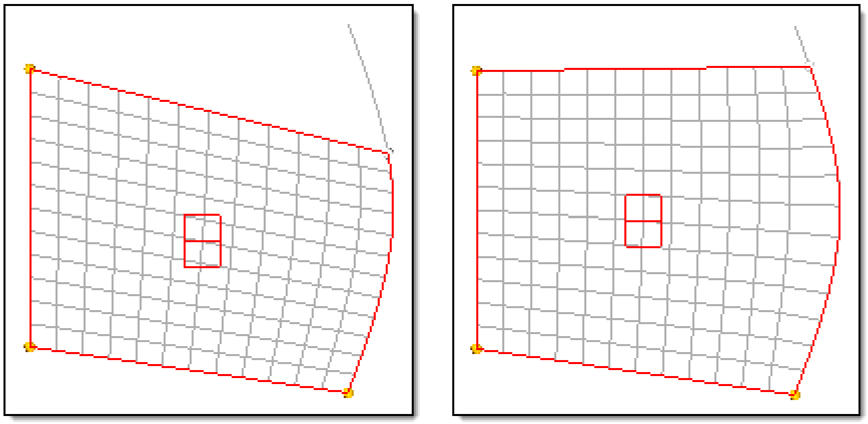


Figure 1781: Nodes Tracking a Line During Morphing

The nodes along the right edge domain are constrained to the line. When the handle is moved, it and the other constrained nodes move along the line.

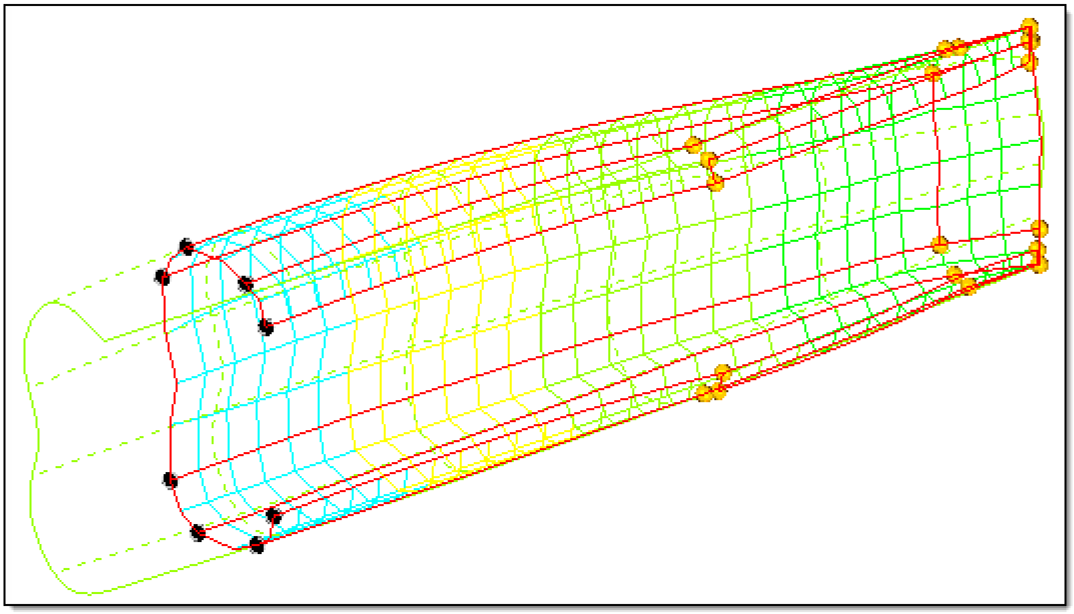


Figure 1782: Morph after Mapping to Surface

All mapped nodes are automatically constrained to the surface. When the handles are translated, the nodes are moved along the surface a distance corresponding to the applied handle perturbations. If the handles were also part of the map to geom operation, they too will be moved along the surface regardless of the applied perturbation. In this example, the handles were translated linearly. HyperMorph automatically placed the handles back on the surfaces after applying the translation so that the constraint was obeyed.

Biasing

Biasing allows you to control the shape of a mesh when applying handle perturbations.

Biasing increases or decreases the influence of a handle over the nodes within its area of influence. If the biasing values for all of the handles are equal to 1.000, which is the default value for all handles except for dependent handles on 1D domains, the morphing between the handles is linear, provided both handles are global handles or they are located on edge domains. Higher biasing values generate a smooth curvature near the handles, while lower biasing values generate harsh corners near the handles. To smoothly change the shape of a domain it is recommended that you use a biasing factor of 1.000 at the corners, 2.000 at the edges, and 3.000 in the middle.

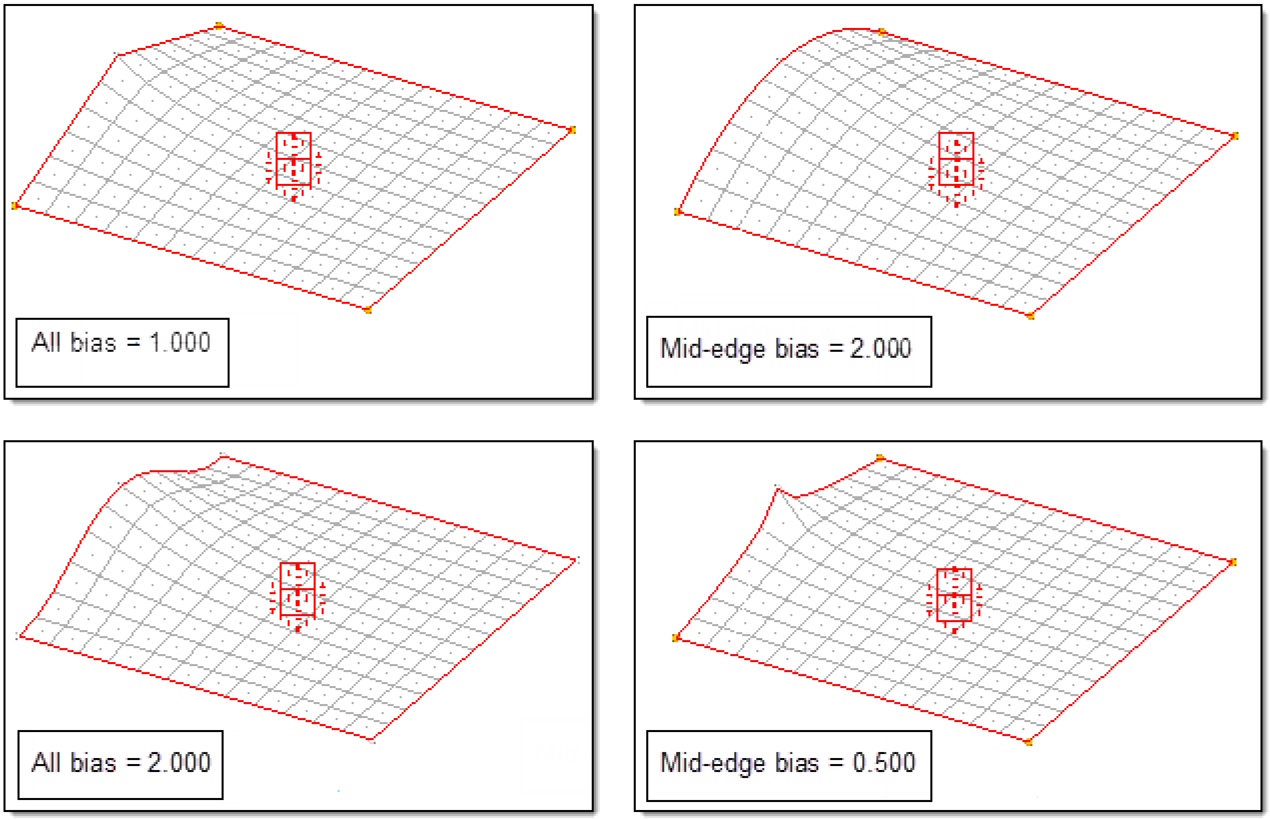


Figure 1783: Biasing for a 2D Domain

The model at the upper left has all five handles with the default biasing value of 1.000. The model at the upper right shows the four corner handles with a biasing value of 1.000, and the mid-edge handle with a biasing value of 2.000. The model at the lower left has all five handles with the default biasing value of 2.000. The model at the upper right shows the four corner handles with a biasing value of 1.000 and the mid-edge handle with a biasing value of 0.500.

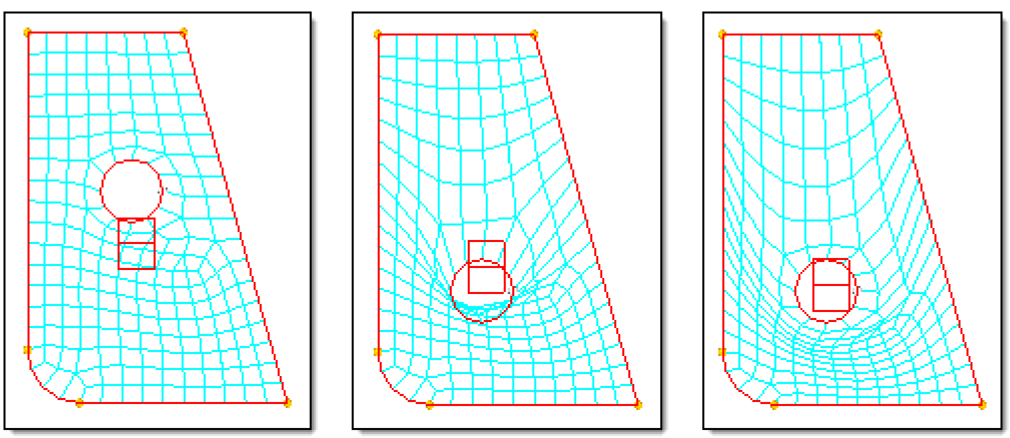


Figure 1784: Biasing to Reduce Mesh Distortion

When the hole is moved downward with a biasing factor of 1.000 for the handle at the hole, the mesh folds over due to the influences of the other handles (middle frame). When the biasing value of the handle at the hole is increased to 3.000, the mesh unfolds (right frame).

Biasing can be applied retroactively after a morphing operation. After applying a morph, you can change the biasing value by selecting the **make retroactive** checkbox, and have the current list of applied morphs updated to reflect the new biasing values. This is useful in selecting a good biasing value to apply for a given morph. Apply the morphs and change the biasing values retroactively until you get the shape that you want.

Solid Model Strategies

Solid models are models that are made up of solid elements, namely, tetras, pentas, and hexas. In general, a solid model represents a single part with numerous features such as holes, edges, bosses, flanges and ribs.

HyperMorph is designed to make it easy to change the size and shape of features in a solid model. This is done using one of the following methods:

- Moving the handles on the part to new locations
- Moving the global handles around the part to new locations
- Altering the radius or curvature of curved edges of the part
- Mapping the nodes of the part to line or surface data.

For solid models, it is only necessary to create a single 3D domain for the entire part. You can also add a global domain and global handles for shape alterations of a general nature.

Manage Handles and Domains for Solid Models

Create handles and domains, divide your solid model, and group, split, merge domains.

You can create a single 3D domain consisting of all the elements in a model. If the model is made up of more than one part, each part is placed in its own 3D domain. The surfaces of each 3D domain are covered with shell elements that are placed in a component named ^morphface. The elements in ^morphface covering each 3D domain are placed into 2D domains. If **partition 2D domains** is checked, these 2D domains are partitioned according to the settings selected in the parameter subpanel of the Domains panel. Once partitioned, edge domains are placed around the 2D domains and handles are placed at the ends of the edge domains. This procedure is automatic. In many cases, the domains and handles are generated where you want them to be. If they are not, you can add, edit, or delete the handles and domains to meet specific needs.

Create Single 3D Domains

Create a single 3D domain that consists of all the elements in a model.

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Create** subpanel.
4. Set the selector to **3D domains**.
5. Change the toggle to **all elements**, or manually select all of the elements in the model.
6. Click **create**.

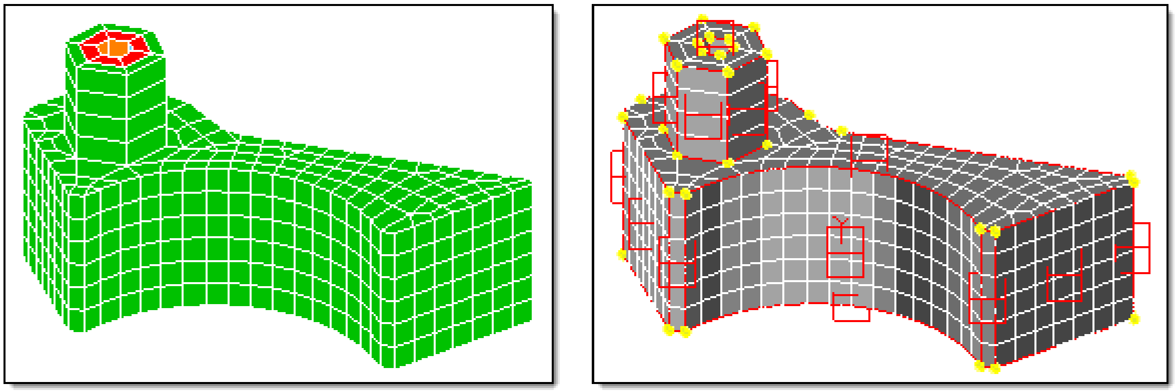


Figure 1785: 3D Domain Created for Solid Model

The automatic creation and partitioning of 2D domains on the face of the solid and the creation of edge domains and handles for the 2D domains.

Create 3D Domains with Global Domains and Global Handles

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Create** subpanel.
4. Set the selector to **auto functions**.
5. Click **generate**.

If there are any domains or handles in the model, you are asked if you want to delete all the current morphing entities. If you click yes, or if there are no morphing entities in the model, 1D, 2D, and 3D domains are automatically generated for the entire model, as well as a global domain and handles.

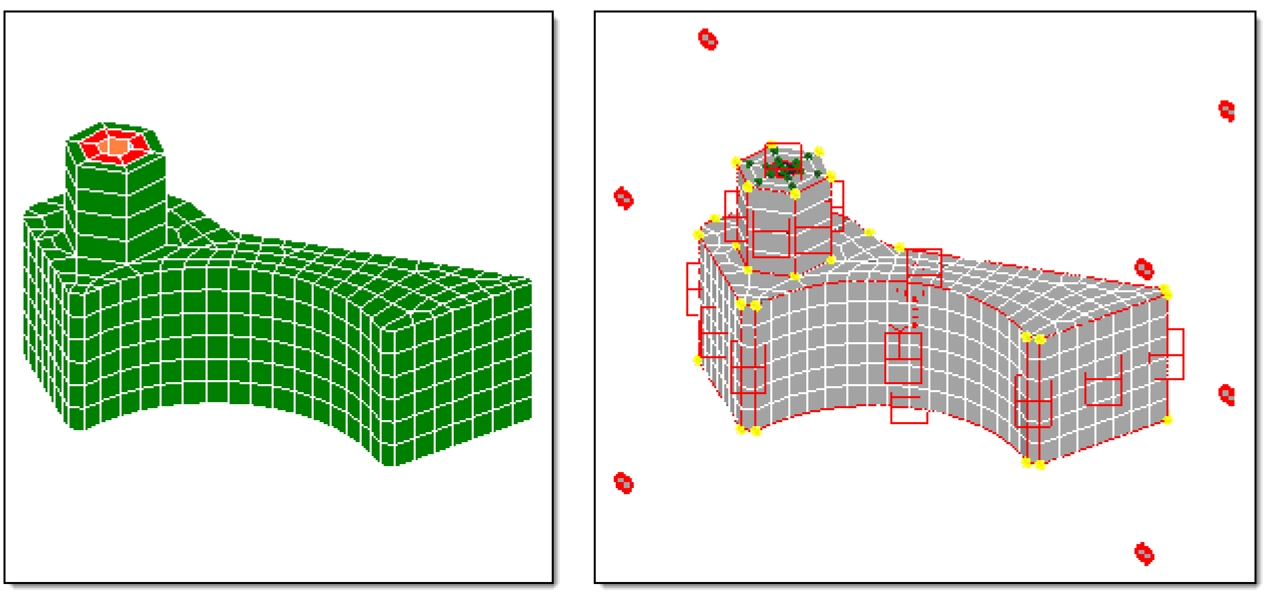


Figure 1786: Automatic Generation of Domains on Solid Model

The addition of a global domain, global handles, and 1D domain, which produces dependent (green) handles.

If you do not select partition 2D domains when you generate a 3D domain, the 2D domain made up of the elements on the surface of the 3D domain will not have edge domains and thus no handles will be generated for it. Without handles, morphing cannot be performed. However, this approach will give you a "blank slate" 2D domain that you can partition by hand. For meshes on which the automatic partitioning does not work well, such as first order tetra meshes, you may find it easier to start with a blank slate rather than editing the automatically created domains. Be sure to try both methods of partitioning, element based and node based, before deciding to partition by hand.

> **Note:** The element based method sometimes works better on second order tetras since it accounts for element curvature. However, if the second order tetras are converted first order tetras and thus have no curvature, the node based partitioning will work better.

Also, for first order tetra meshes, you may find it more effective to ignore curvature when automatically partitioning. To do this, in the parameters subpanel, change the uppermost toggle from curvature based to angle based. You may also want to lower the domain angle to 30 degrees. Partitions will be made only along edges in the model where the domain angle is exceeded. You can then manually divide the 2D domains where the curvature breaks should be located. This method is very helpful for meshes that began as first order tetra meshes but then were then transformed into second order meshes. For these meshes, a curvature break is detected at every element along a curve if the midpoint nodes of the elements have not been modified to capture the curvature. This results in a domain for every element on a curve which makes morphing impractical.

Solving the influence coefficients for 3D domains which contain more than 20,000 elements can become very time consuming even though it is only done after domain editing and during morphing operations such as radius change and map to geom. In these cases you may want to divide the domain into multiple domains using the subdivide 3D function in the update subpanel of the Domains panel, or lower the limit for the large domain solver. The large domain solver limit can be found in the global subpanel of the Morph Options panel. However, even though influence calculations for large domains are more rapid, morphing using the large domain solver can be time consuming, and thus subdividing 3D domains can often be the best solution for efficient morphing. Additionally, if you are only going to morph a part of your 3D mesh, you only need to create domains for that part.

Subdivide Solid Models

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Update** subpanel.
4. Set the selector to **subdivide 3D**.
5. Select the 3D domains to be subdivided.
6. Select any 2D domains on the surface of the 3D domain that are permissible for HyperMorph to split into more than one 2D domain.
7. Click **subdivide**.

HyperMorph automatically subdivides the 3D domains into one or more 3D domains while leaving the 2D domains not selected as being divisible unchanged. Not that in some cases HyperMorph will not be able to subdivide a 3D domain without dividing an indivisible 2D domain. In these cases the 3D domain will be left undivided.

Divide Solid Models Manually

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Create** subpanel.
4. Set the selector to **3D domains**.
5. Select the elements to be placed into a new 3D domain.
6. Click **create**.

When selecting elements for the new domain, you do not need to select only solid elements, other elements are automatically removed before the domain is created. Therefore, you do not need to be concerned about selecting the morphface elements.

Also, it does not matter if the selected elements are already in a 3D domain. When the new domain is created, the elements are moved from the old domains to the new domain. Morphface elements are placed at the internal interface between the new domain and the other domains and create a 2D domain for the interface, but it will not partition the interface. This better accommodates the division of tetra meshes that cannot be divided along flat or curved internal faces and thus would be partitioned into many domains.

> **Note:** When you divide a 3D domain into parts, it has the effect of partitioning the surface of the original 3D domain along seams where the divisions were made. So when you divide your model into 3D domains, make sure that you divide it along lines where you want your 2D domains on the surface to be.

Dividing a 3D domain into many 3D domains can be very useful for controlling the movement of nodes within the domain when the handles on the surface are moved. When some meshes are morphed, the internal elements can become distorted. This is generally caused by handle influences extending too far through the 3D domain. You can divide your 3D domains to restrict the handle influences and control mesh distortion.

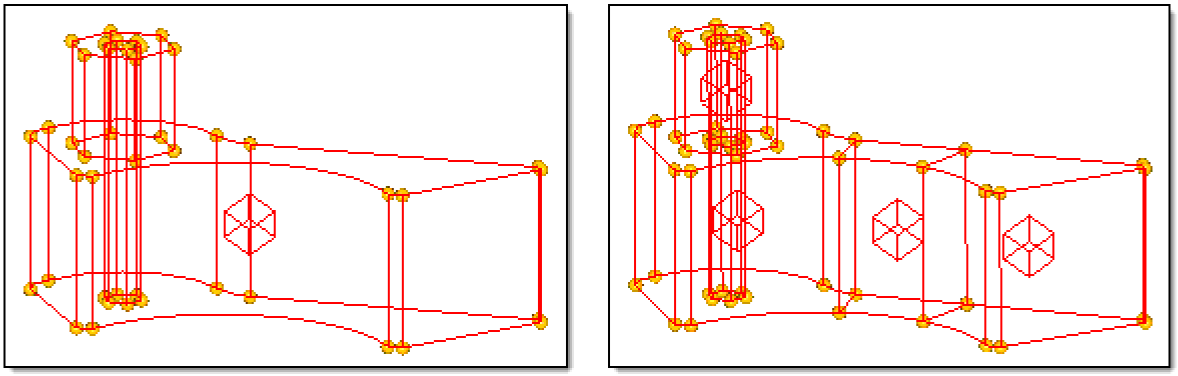


Figure 1787: 3D Domain Split into Four 3D Domains

The influences of the handles will not extend across the boundaries between the domains.

Influences must be recalculated every time handles, domains, or symmetries are added, edited, or deleted. They are also recalculated during radius changes and geometry mapping. These calculations occur when you enter or leave a HyperMorph panel or when you leave the delete panel. For large models you will want to make all of your domain changes before exiting the Domains panel. The influences for handles are only recalculated in regions that have been edited.

If the domains are not created exactly the way you want them, you can edit them in the Domains panel. The create subpanel allows you to create new domains. The organize subpanel allows you to edit domains by adding and removing elements to or from a domain and by grouping domains together. The edit edges subpanel allows you to split, merge, and place handles along edge domains. Since creating or editing 3D domains results in the creation of 2D and edge domains, and creating or editing 2D domains results in the creation and deletion of edge domains, you should perform the tasks in the following order:

1. Create and edit all the 3D domains that you want first.
2. Create and edit the 2D domains.
3. Create and edit the edge domains.

Automatic partitioning does not always divide a mesh in the most useful ways. Occasionally, elements end up in domains adjacent to where you want them or placed in their own domain. Some cleanup may be required.

Organize Elements in Domains

Move elements from one domain to another.

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Organize** subpanel.
4. Change the selector to **add nodes/elements**.
5. Select the elements to be moved.
6. Select the target domain.
7. Click **organize**.

The elements are moved from their current domain to the selected domain. The edge domains around both domains are refreshed, as well as the 2D domains at the interface if solid elements are being organized. New handles may also be created during this process, and if retain handles is not checked, handles may be deleted. You should keep retain handles unchecked unless you have created shapes for the model that use the handles on the domains that you are editing.

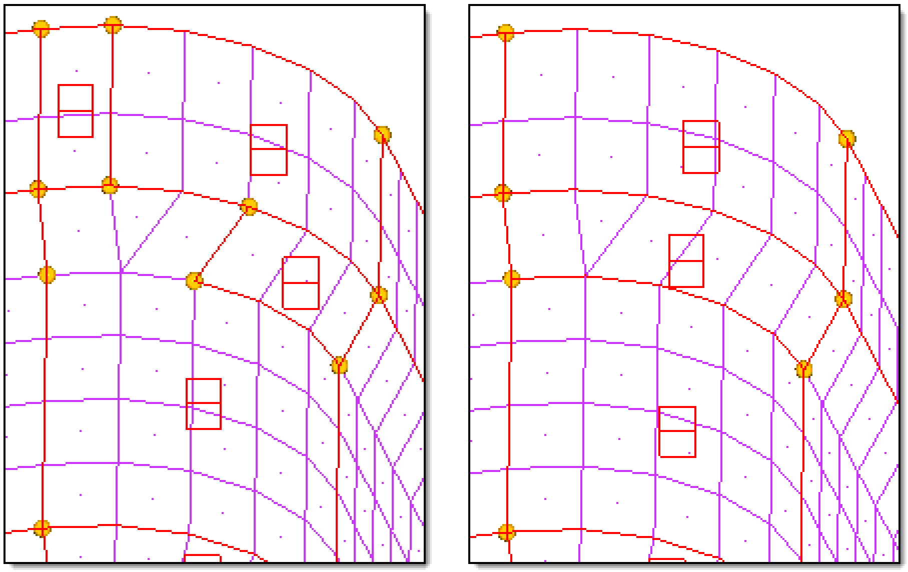


Figure 1788: Move Elements

The model on the left shows problems that partitioning can encounter for some meshes. The model on the right has been corrected using the organize subpanel of the Domains panel. For this example, the retain handles option was left unchecked, resulting in the elimination of handles that are no longer on the corners of the 2D domains.

> **Note:** Holding the mouse button down when the mouse is either over the icon for a 2D or 3D domain or over an element inside a domain, will highlight the edge domains surrounding the domain. This allows you to visualize the domain that you are selecting. The domain icon is placed at the centroid of the domain, and for some domains it can end up away from the elements of the domain and near other domain icons. Having the edges for the domain highlighted during selection is often necessary to tell which icon goes with which group of elements.

Group Domains

Group two or more domains.

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Organize** subpanel.
4. Change the selector to **combine domains**.
5. Select the domains to be grouped.
6. Click **organize**.

The selected domains are combined into a single domain, and the surrounding domains and handles are updated.

Edge domains are automatically partitioned when they are created. They are also updated whenever a change occurs for a domain of which they are on the edge. This is why you should edit the edge domains after the other domains have been edited. If you perform edge editing first, your changes may be erased when you edit the 2D and 3D domains.

Edge domains are used to make radius changes, so it is important to make sure that any radius in the model that you intend to change be captured correctly by edge domains. HyperMorph attempts to partition edge domains where curvature begins and ends, but in some cases, it will not identify the proper starting and ending points. You will need to correct this by hand.

Split Edge Domains

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Edit Edges** subpanel.
4. Change the selector to **split**.
5. Select an edge domain.
6. Select a node on that domain that is not on the edge.
7. Click **split**.

The selected edge domain is split into two edge domains at the selected node. A handle is created at the selected node.

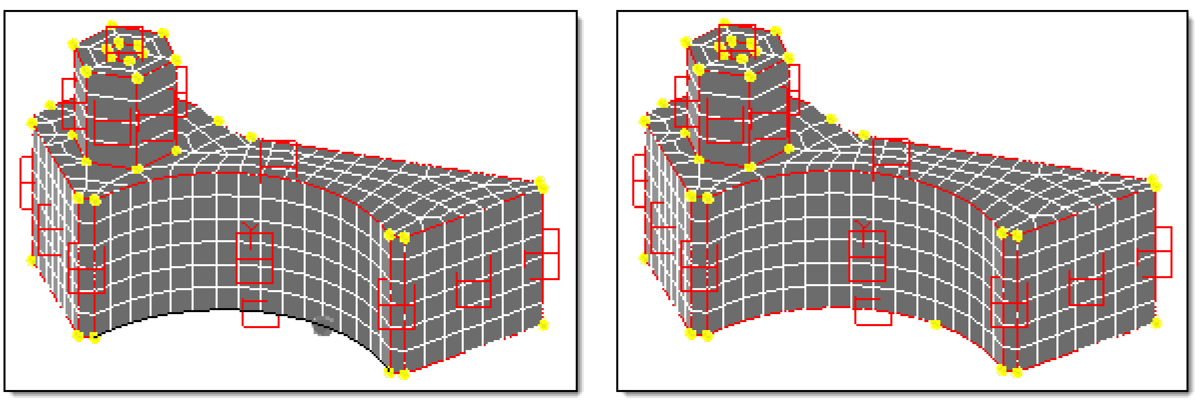


Figure 1789: Split Edge Domains

The lower edge domain has been split at the gray node (left model), which becomes a handle (right model). Now the radius of each new edge domain may be modified independently of the other.

Merge Edge Domains

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.

3. Select the **Edit Edges** subpanel.
4. Change the selector to **merge**.
5. Select any number of edge domains.
6. Click **merge**.

The two edge domains are merged into one edge domain. This function only allows you to merge edge domains that lie end-to-end such that the resultant merged edge domain is a continuous series of nodes.

You may also create dependent handles along an edge domain. This feature helps save time when you are changing the radius for the edge domain. If a model is very large, you may find it more efficient to place dependent handles on the edge domains whose radii you wish to change before you enter the morphing panel.

> **Note:** You can also merge edge domains in the organize subpanel.

Create Dependent Handles along Edge Domains

1. From the Tools page, click **HyperMorph**.
2. Click **Domains**.
3. Select the **Edit Edges** subpanel.
4. Change the selector to **add handles**.
5. Select one or more domains.
6. Click **create**.

   Dependent handles are created on the selected edge domains. These handles are dependent on the independent handles to either side of them along the edge domain.

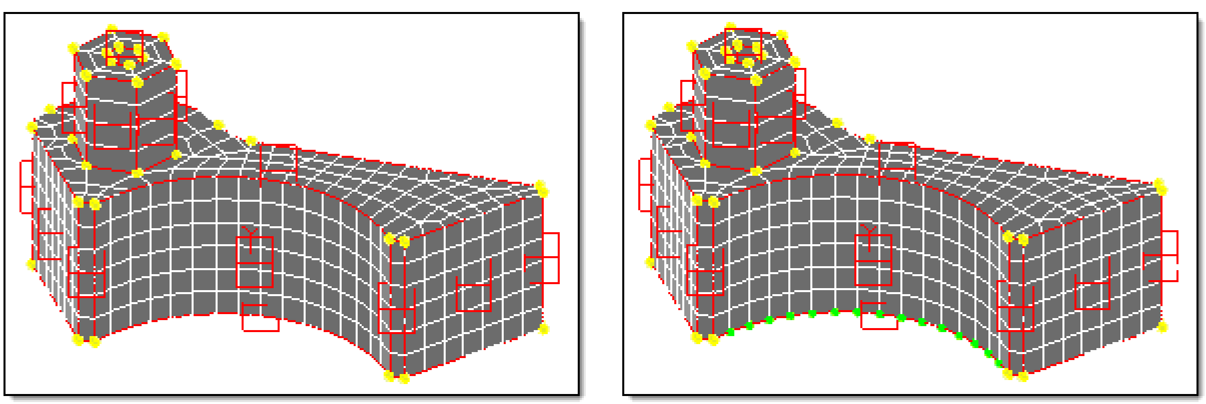


Figure 1790: Dependent Handles Created using the Handles on Edge Feature

Creating dependent handles in this way has two significant effects. The first is that since they are dependent, movements applied to any of the independent handles on the edge are transparently applied to the dependent handles. It will be as if they were not there. Secondly, when you make a radius change to an edge domain that has a handle at each of its nodes, the influences do not need to be recalculated, which makes the radius change process much faster for large models.

7. When you are satisfied with your domains, click **return**.

   The influences for the handles is calculated and you are ready to begin morphing.

   > **Note:** During influence calculation for large models you might run out of available memory. This generally happens when a given domain is too large and it contains too many handles. In these cases, you should divide large domains, delete unnecessary handles, or lower the limit of the large domain solver.

Solid Model Display Controls

The HyperMesh graphics engine supports different visual options for viewing models as you work on them.

Surface-Only Wire Frame

In this default mode, your model is displayed as a wire frame, but only the surface elements are drawn because in a solid model, a full wire frame can make it very difficult to visualize the model because every element in the model is displayed. Since HyperMorph creates a component called ^morphface, which contains shell elements on the surface of the 3D domains, the default setting is to display only that component, thus showing only the outer surface of your model and making it easier to work on. However, since the viewing mode is still wire frame, you will see the two sides of your model superimposed over each other.

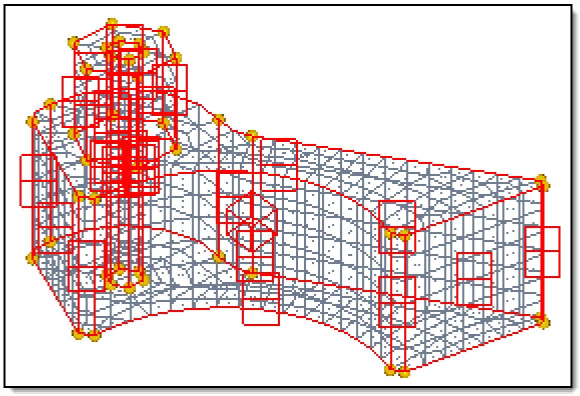


Figure 1791: Surface-Only Wire Frame

Solid Fill

The option produces a display that is similar to what you see when you perform a fill plot in the Hidden Line panel. You only see the side of the model that is facing you, as if your model was a real part. You can still display the surface mesh, if desired, as shown.

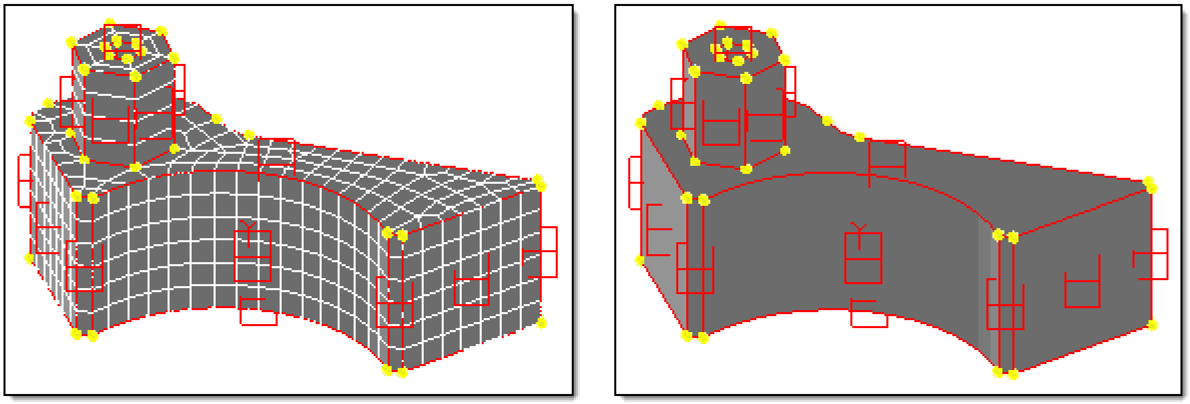


Figure 1792:

You can also view a solid model for morphing by turning off all the components and looking at only the domains and handles. This is similar to looking at the model in a meshless wire frame mode. Partitioning generally captures all the features on the surface of a solid, so by viewing only the domains you can visualize the model with minimal clutter.

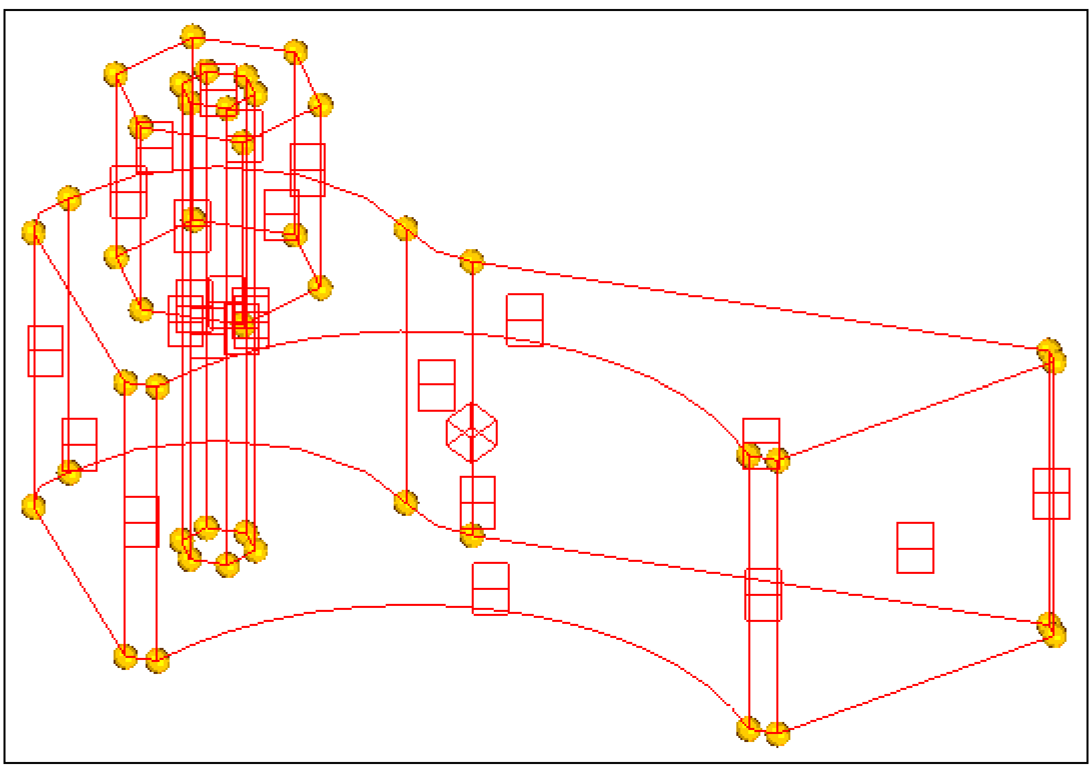


Figure 1793:

Optimization 14

Setup an Optimization in HyperMesh.

This chapter covers the following:

HyperMesh Optimization technology is supported by OptiStruct. OptiStruct is a finite element and multibody dynamics software application which can be used to design and optimize structures and mechanical systems.

OptiStruct uses the analysis capabilities of Radioss and MotionSolve to compute responses for optimization.

Structural Design and Optimization

Structural design tools include topology, topography, and free-size optimization.

Sizing, shape and free shape optimization are available for structural optimization. In the formulation of design and optimization problems, the following responses can be applied as the objective or as constraints: compliance, frequency, volume, mass, moment of inertia, center of gravity, displacement, velocity, acceleration, buckling factor, stress, strain, composite failure, force, synthetic response, and external (user defined) functions. Static, inertia relief, nonlinear gap, normal modes, buckling, and frequency response solutions can be included in a multi-disciplinary optimization setup.

Setup Optimization in HyperMesh

Simple steps for setting up your optimization in HyperMesh.

1. Create a running model of your analysis type supported by Radioss or MotionSolve.
2. Define or create your Design space.
3. Create responses.
4. Define constraints.
5. Define objectives.

Design Interpretation - OSSmooth

OSSmooth is a semi-automated design interpretation software, facilitating the recovery of a modified geometry resulting from a structural optimization, for further use in the design process and FEA reanalysis.

The OSSmooth tool has two incarnations: a standalone version that comes with the OptiStruct installation, and a dependent version that is embedded in HyperMesh.

OSSmooth can be used in for OSSmooth for geometry, FEA topology reanalysis, and FEA topography reanalysis.

OSSmooth (for geometry) can be used to:

- Interpret topology optimization results, creating an iso-density boundary surface (Iso-surface).
- Interpret topography optimization results, creating beads or swages on the design surface.
- Recover and smooth geometry resulting from a shape optimization.
- Reduce the amount of surface data from a given set of triangular patches by combining smaller patches.
- Smooth surface data given as triangular patches.

For FEA topology reanalysis and FEA topography reanalysis, OSSmooth can be used to:

- Preserve component boundaries for multiple design components.
- Recover geometry with or without an artificial layer of elements around a non-design space optionally.
- Tetramesh Iso-surfaces ‘by property’.
- Preserve boundary conditions upon geometry recovery to enable quick reanalysis.

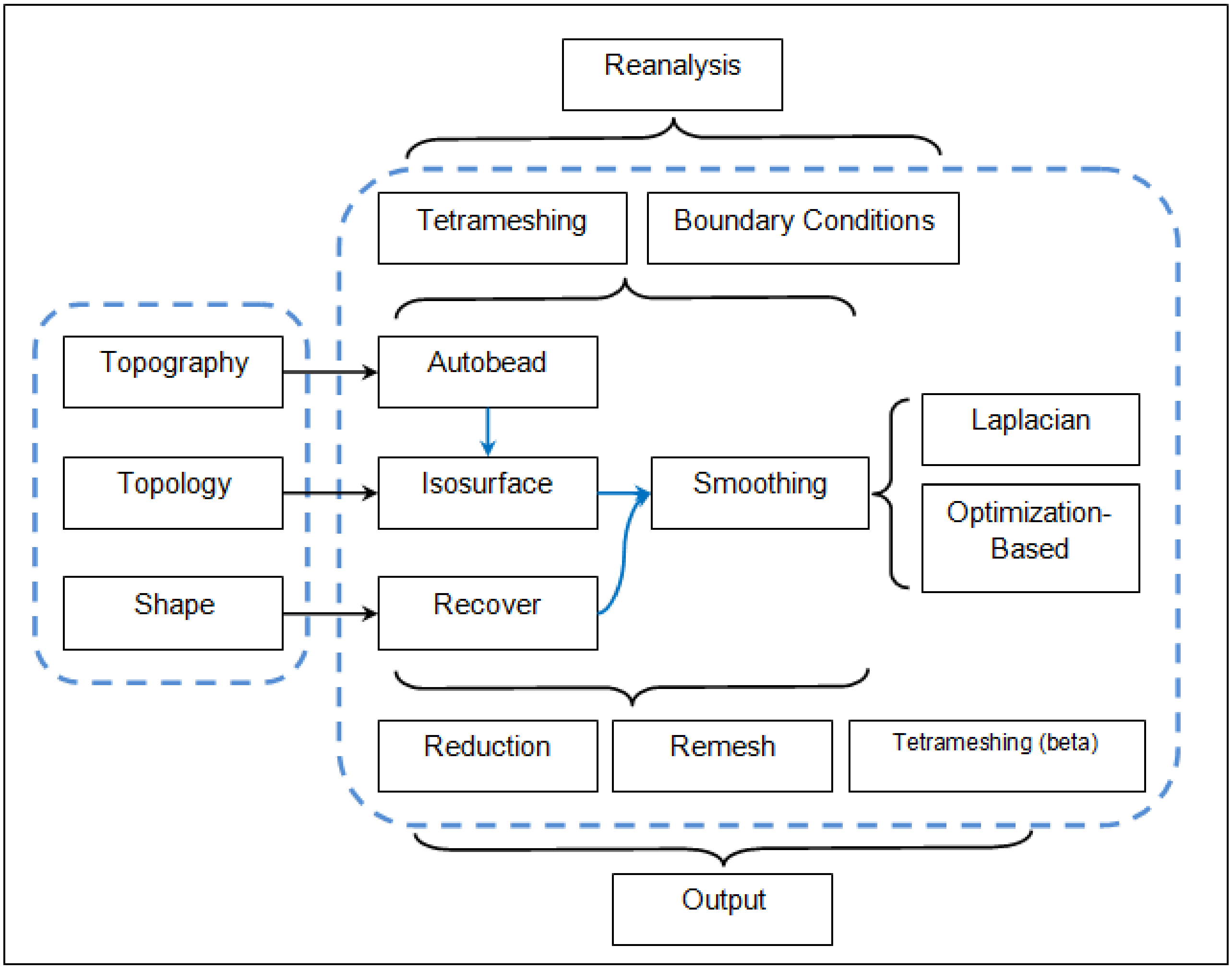


Figure 1794: OSSmooth Workflow

Overview of how OSSmooth works to interpret optimization results from OptiStruct.

Each of the three applications of OSSmooth has a corresponding sub-panel in the OSSmooth panel in HyperMesh. OSSmooth (for geometry) is generally used to recover geometry by interpreting topology, topography, and shape optimization results, while FEA topology and FEA topography are used to generate recovered geometry with boundary conditions for FEA reanalysis.

OSSmooth (for geometry) requires a parameter file, generally has the file extension `.oss` to run. This parameter file may be generated from the OSSmooth panel in HyperMesh, or it may be generated manually through a text editor. At the completion of an optimization run, OptiStruct automatically exports an OSSmooth parameter file `<prefix>.oss` with certain default settings depending on the type of optimization run. The second order option has been added for parameters in the OSSmooth panel.

In addition to the parameter file, OSSmooth (for geometry) also requires the input file (`<prefix>.fem`), the shape file (`<prefix>.sh`), and/or the grid file (`<prefix>.grid`) from an OptiStruct run. The grid file `<prefix>.grid` contains the grid point locations after a topography or shape optimization and is output at the end of a topography or shape optimization run. The shape file, `<prefix>.sh`, contains the element density information of a topology optimization and is output at the end of a topology optimization run. If you want to use OSSmooth for topology results which is not from OptiStruct, simple xy data is accepted.

FEA topology requires the input model (`<prefix>.fem`) to be loaded into HyperMesh before running, which is different from OSSmooth (for geometry). It also requires the shape file (`<prefix>.sh`) generated by a topology optimization. For processing of the non-design elements, two options, Keep smooth narrow layer around and Split all quads, are provided to recover geometry.

FEA topography requires a grid file (`<prefix>.grid`) to run. Similar to FEA topology, it also requires that the input model (`<prefix>.fem`) be loaded into HyperMesh first, with the option for iso-surface that performs the same functionality as FEA topology.

> **Note:** OSSmooth currently does not recognize OptiStruct long-format input data. A possible work-around for this problem is to import the long-format input file into HyperMesh and export it using the regular OptiStruct template before running OSSmooth. The interpreted design from OSSmooth can be exported as a finite element mesh in the bulk data format, as IGES surfaces, as a stereolithography file, or as a Hyper3D file.

OSSmooth Parameter File

Parameter statements in the OSSmooth parameter file.

OSSmooth file parameter statements each have following format:

```
parameter_name          arg1,arg2,...,argn
```

The `parameter_name` and arguments can be separated either by spaces or commas. The file is not case sensitive.

Comment lines in the OSSmooth parameter file should start with either '#' or '$'.

Supported Parameters

input_file

Identifies the files to be interpreted by OSSmooth.

arg1

The file name (without extension) of the OptiStruct `.fem`, `.sh`, and/or `.grid` files to be interpreted by OSSmooth.

output_file

Name of the file to be output by OSSmooth.

arg1

Full name of the file output by OSSmooth.

output_code

Identifies the type of output.

arg1

Output format for iso-surface

"1" Bulk data trias

"2" IGES patches

"3" STL trias [default]

"4" H3D trias

arg2

Output control for tet-meshing of volume enclosed by the iso-surface.
[Default: no tet-meshing]

"1" Tetra4 + Tria3 elements

"2" Tetra10 + Tria6 elements

"3" Tetra4 elements

"4" Tetra10 elements

The tet-mesh is written in OptiStruct input format to a file <input_file>_mesh.fem. In this file, the design space of the file <input_file>.fem is replaced with a tet-mesh for immediate reanalysis. Loads, boundary conditions, and non-design elements are carried over from the input file.

units

Defines output units for IGES format. This information gets written to the header of the IGES file and may be recognized by your CAD system.

1 (default)

inch

2

mm

4

foot

10

cm

autobead

Improves the recovered geometry from a topography optimization by applying automatic geometry creation.

arg1

Operation flag [integer]

"0" autobead off

"1 (default)" autobead on

arg2

Threshold value for creating autobead.
[real, between 0.0 and 1.0, default = 0.3]

arg3

Bead layer [integer]

"1 (default)" create 1 layer bead

"2" create 2 layers bead

isosurface

Generate threshold surface from a topology optimization by applying automatic geometry creation.

arg1

Operation flag [integer]

"0" isosurface off

"1 (default)" isosurface on

arg2

Type of surface created [integer]

"0" isosurface only

"1" isosurface with Optimization-based smoothing

"2" Element threshold surface

"3 (default)" isosurface with Laplacian smoothing

For topology results, smoothing maintains the topology as suggested by OptiStruct, but it can deviate from the given density distribution. If option 1 or 3 is used, check the maximum and average smoothing error output by OSSmooth.

arg3

Density threshold for creating isosurface.
[real, between 0.0 and 1.0, default = 0.3]

opti_smoothing

Optimization-based smoothing [Only used if isosurface C2=1].

arg1

Unit-less surface distance coefficient
[real, default = 0.0]
Defines closeness of the smooth surface from the threshold surface. The effect of this coefficient varies for different input meshes. Higher magnitudes, both positive and negative, give smoother results, but the surface deviates more from the original density distribution. The recommended range is from -50 to 50. When the coefficient is set between 0 and 50, the surface usually tends to smooth and shrink. When the coefficient is set between 0 and -50, the surface usually tends to smooth and expand.

arg2

Smooth isosurface boundary flag [integer]

"0 (default)" boundary not included in smoothing

"1" boundary included in smoothing

laplacian_smoothing

Laplacian smoothing [Only used if isosurface C2=3].

arg1

Number of iteration for Laplacian smoothing
[integer > 0, default = 10]

arg2

Feature angle threshold in degrees

[real, default = 30.0]
The feature angle is defined as the angle of normal between two intersected element planes. All corners with a feature angle larger than the threshold will be preserved in the smoothing process.

arg3

Smooth isosurface boundary flag [integer]

"0" boundary not included in smoothing

"1 (default)" boundary included in smoothing

remesh

Remesh autobead surface and/or isosurface flag [integer]

0 (default)

remesh off

1

remesh on

Remesh detects 2-layer elements around bead shape and/or boundary of isosurface. Use mixed type remesh if input mesh contains any QUAD elements, otherwise remesh with TRIA elements.

surface_reduction

Reduces the number of surfaces representing the geometry. Can reduce the number of surfaces by up to 80 percent.

arg1

Surface reduction flag [integer]

"0 (default)" no surface reduction

"1" do surface reduction

arg2

Feature angle threshold in degrees
[real, default = 10.0]
The feature angle is defined as the angle formed by the surface normal of two adjacent elements. The surface reduction will be performed on any two adjacent elements in which the feature angle between the two elements is smaller than the threshold. The greater the threshold, the more surface reduction will be conducted. The valid range of the threshold is [1.0, 80.0].

pure_surf_smoothing

Surface smoothing only.

arg1

Pure surface smoothing flag [integer]

"0 (default)"

"1" Optimization-based smoothing

"2" Laplacian smoothing

arg2

Number of iteration [Only used if G1=2]
[integer > 0, default = 10]

arg3

Feature angle threshold in degrees [Only used if G1=2]
[real, default = 30.0]
The feature angle is defined as the angle of normal between two intersected element planes. All corners with a feature angle larger than the threshold will be preserved in the smoothing process.

pure_surf_reduction

Surface reduction only.

arg1

Pure surface reduction flag [integer]

"0 (default)" no surface reduction

"1" do surface reduction

arg2

Feature angle threshold in degrees
[real, default = 10.0]
The feature angle is defined as the angle formed by the surface normal of two adjacent elements. The surface reduction will be performed on any two adjacent elements in which the feature angle between the two elements is smaller than the threshold. The greater the threshold, the more surface reduction will be conducted. The valid range of the threshold is [1.0, 80.0].

OSS Example Input File

input_file example

Identifies the root of the input files as example, so OSSmooth will look for the files `example.fem`, `example.grid`, and `example.sh`.

output_file example.stl

The resulting output will be `example.stl`.

output_code 3

The output will be in stereolithography format.

Autobead 1 0.3 1

Topography results will be interpreted using the autobead feature with a threshold value of 30% creating single depth beads.

Isosurface 1 3 0.3

Topology results will be interpreted by creating an iso-density boundary surface with at a density value of 30% and smooth using laplacian smoothing.

laplacian_smoothing 10 30 1

The Laplacian smoothing will run for 10 iterations, consider a feature angle of 30-degrees and including the boundary in the smoothing.

Remesh 1

The two rows of elements around the recovered geometry will be remeshed in an attempt to smooth the mesh transition.

Run OSSmooth

- Run OSSmooth from the command line.
  a) Enter `ossmooth <prefix>.oss`.
- Run OSSmooth from the HyperMesh Solver panel.
  a) From the menu bar, click **Tools** > **Solver**.
  b) Set switch to **OSSmooth**.
  c) In the input file field, enter `<prefix>.oss`.
  d) Click **solve**.

     OSSmooth standalone, which can also be invoked from the Solver panel in HyperMesh, checks out 50 HyperWorks Units.
- Run OSSmooth from the HyperMesh ossmooth panel.
  a) From the Post page, click **ossmooth**.
  b) Choose either OSSmooth (for geometry), FEA topology or FEA topography.
  c) In the file field, select the OptiStruct input file (`<prefix.fem>` and/or `<prefix.sh>` and/or `<prefix.grid >`).
  d) Edit the OSSmooth input data by making selections on the screen.
  e) Click **ossmooth**.

  OSSmooth (for geometry) will write a new `<prefix.oss>` file with the screen settings and load the geometry recovered into HyperMesh if the data format is IGES, STL, or Nastran. FEA topology and FEA topography will update the model in HyperMesh without outputting the result; if required, data can be exported from HyperMesh.

  OSSmooth invoked from the ossmooth panel in HyperMesh checks out 42 HyperWorks Units (21 leveled and 21 stacked).
- Run OSSmooth from the HyperWorks Run Manager.
  a) In the Input file(s) field, select the `.oss` file.
  b) Verify that the Options field is empty.
  c) Click **Run**.

Interpretation of Topology Optimization Results

Topology optimization results are interpreted to provide an iso-density surface based on the volumetric density information of a topology optimization, which is conducted using OptiStruct.

OSSmooth can handle both shell and solid elements with the same parameter setting.

```
#general parameters
```

```
input_file              mattel
output_file             mattel.stl
output_code             3

#specific parameters
isosurface              1  3  0.300
laplacian_smoothing     10  30.000  1
surface_reduction       1  10.000
```

Figure 1795: Parameter File for Post-Processing of Shell Element Topology Optimization

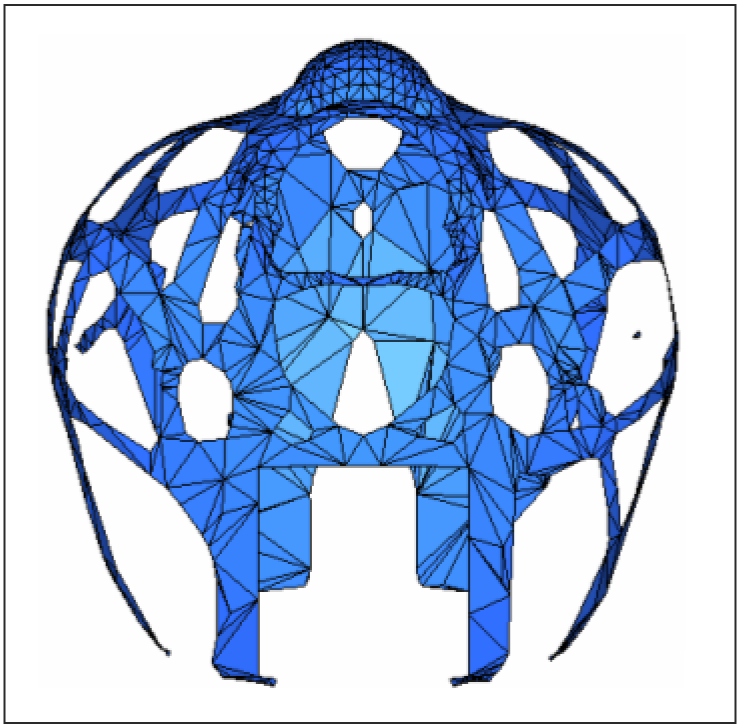


Figure 1796: Surface Reconstruction of Shell Element Topology Optimization

The parameter `laplacian_smoothing` is used for additional smoothing. In most cases, the threshold surface (isosurface with second argument 0) already creates a smooth shape. Additional smoothing (isosurface with second argument 3) maintains the topology as suggested by OptiStruct, but it can deviate from the given density distribution. If this option is used, the maximum and average smoothing error output by OSSmooth should be checked. The `surface_reduction` parameter is used to reduce the number of elements.

Connection Detection

While the iso-density surface is very useful in visualizing the resulting topology, simply using a density threshold value for design interpretation can be misleading. This is due to the fact that when a density threshold value is defined as the cut-off, all design elements with density values less than the threshold are not considered in the interpreted design. This is the intent of the feature, but in many cases, there are elements with densities lower than the threshold value that do contribute significantly to the structural integrity of the design. Ignoring these sections could lead to design failures or an interpreted design that is infeasible. Therefore, it is very important that these significant lower-density regions be captured as well in the design interpretation process. These regions could be interpreted as thinner or weaker sections, but are necessary from a structural perspective nonetheless. The iso-surface design interpretation capability of OSSmooth was therefore enhanced through the implementation of the connection detection feature.

Once the density threshold is applied, certain strategies are used to detect areas with potential connections or bridges between regions of material to preserve structural integrity of the interpreted design. However, there may be many such potential connections, most of which typically have elements with very low densities. Therefore, a lower bound threshold value can be defined to avoid finding the unnecessary connections.

Both threshold value (T) and lower threshold value (t) refer to element density, and range from 0.0 to 1.0. The lower threshold value, t, should be smaller than the threshold value, T, defined. For example, if T is 0.3, the lower threshold value should lie between 0.0 and 0.3. The default value for the lower threshold is calculated internally as t=0.667*T. However, selection of this value is model-dependant, so when necessary you can override the default by setting the lower threshold value based on your requirements.

If the resulting iso-surface with default settings has unwanted connections, a larger lower threshold value can simply be defined. On the other hand, if the final iso-surface does not contain the important connections needed, the lower threshold value can be made smaller.

Connection detection was implemented to facilitate a more robust design interpretation process by capturing important material distribution that could otherwise be missed due to the definition of a single density cut-off value.

From Figure 1797 and Figure 1798, it is evident that by selecting a certain threshold value, certain critical structural members may be lost in the design interpretation process.

Figure 1799 and Figure 1800 show the interpretation of the same density threshold values as figures 1 and 2 respectively, but with connectivity detection switched on. It is evident that important structural members are preserved even at higher threshold values.

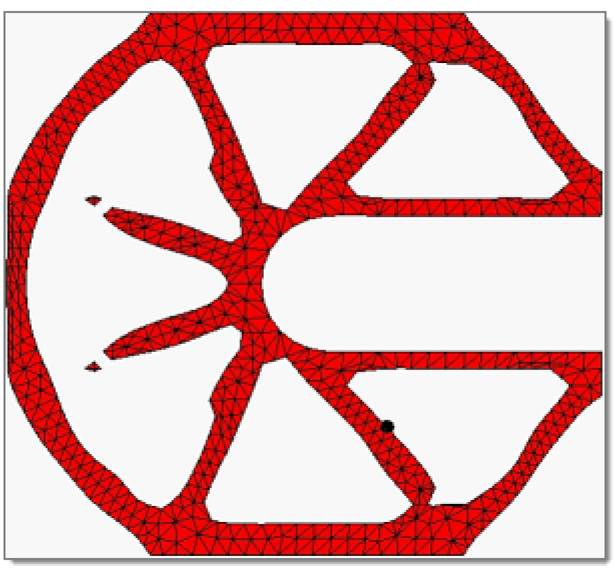


Figure 1797: Density Threshold = 0.5

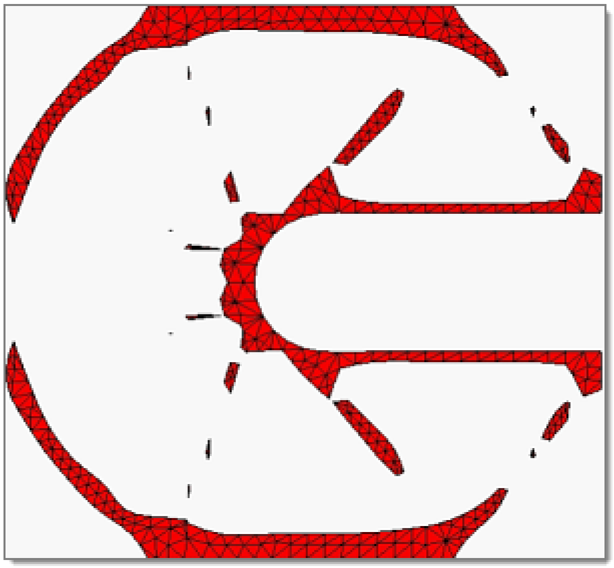


Figure 1798: Density Threshold = 0.7

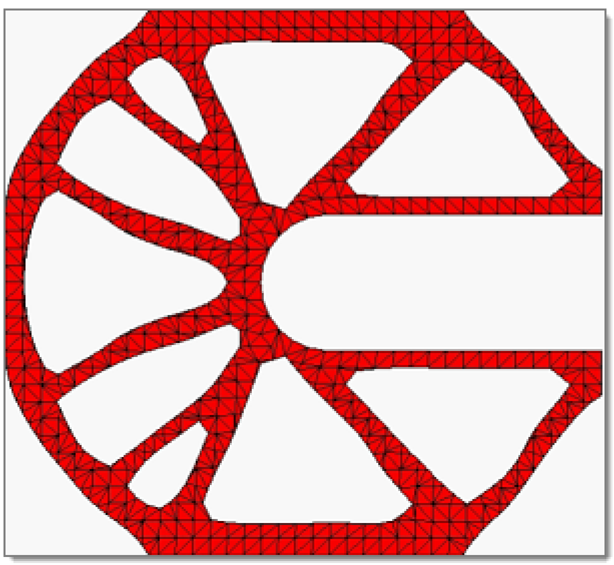


Figure 1799: Connection Detection = 0.5

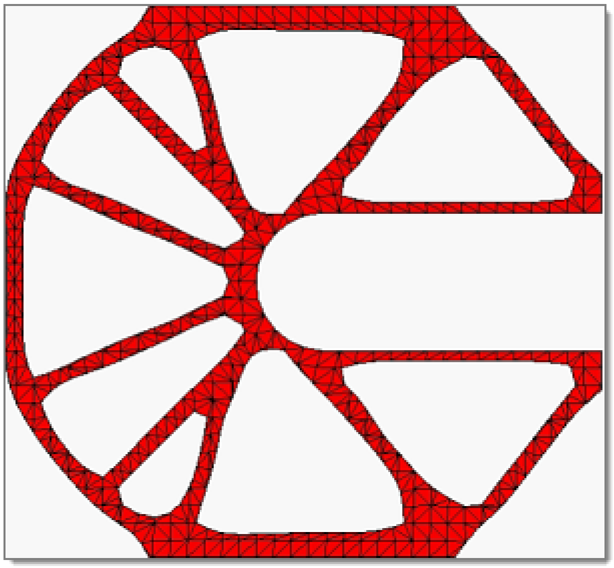


Figure 1800: Connection Detection = 0.7

Draw Recovery

To further enhance the quality of recovered geometry from a topology optimization, a new draw recovery option is implemented. By simply selecting the draw recovery checkbox in the OSSmooth panel, the draw recovery option is enabled in the geometry extraction process. A draw direction can be defined in the optimization formulation as a manufacturing constraint and the draw recovery option uses this information in the interpretation of the geometry.

From the results of a topology optimization, for iso-surfaces near-parallel to the draw direction vector, the draw recovery option creates smooth, straight surface walls in the draw direction, in the isosurface interpretation. Figure 1801 showcases an iso-surface extraction without the use of draw recovery, and Figure 1802 shows the same result with draw recovery. Straight surface walls in the draw direction are interpreted with the draw recovery option. This simplifies the design interpretation task.

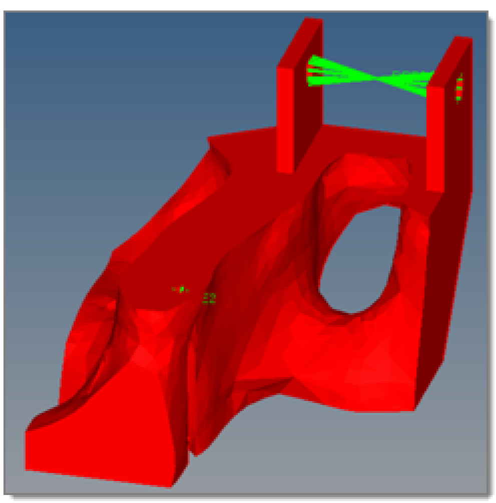


Figure 1801: No Draw Recovery

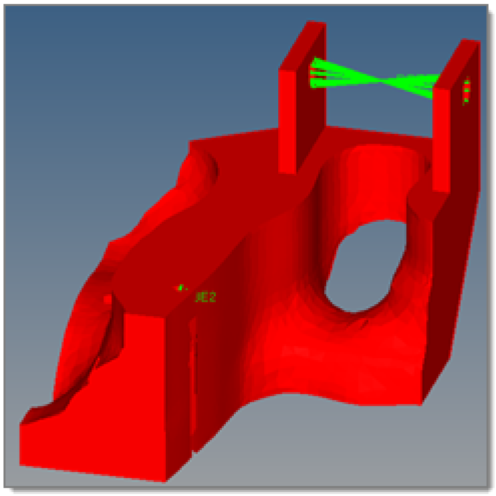


Figure 1802: With Draw Recovery

Laplacian Smoothing

Laplacian smoothing can be used in the smoothing of the results of topology optimization.

The *laplacian_smoothing* statement controls the iteration number of when the Laplacian smoothing will be performed and the feature angle threshold to preserve normal discontinuity at corners.

```
#general parameters
input_file            surf
output_file           surf.stl
output_code           3
isosurface            1   3   0.300

#specific parameters
laplacian_smoothing   10   30.000   1
```

Figure 1803: Parameter File for Smoothing Result

Laplacian smoothing creates smooth boundary iso-surface by entering 1 as the 3rd argument of the *laplacian_smoothing* parameter statement. Comparing Figure 1804 and Figure 1805, notice that Figure 1805 is almost ready for casting.

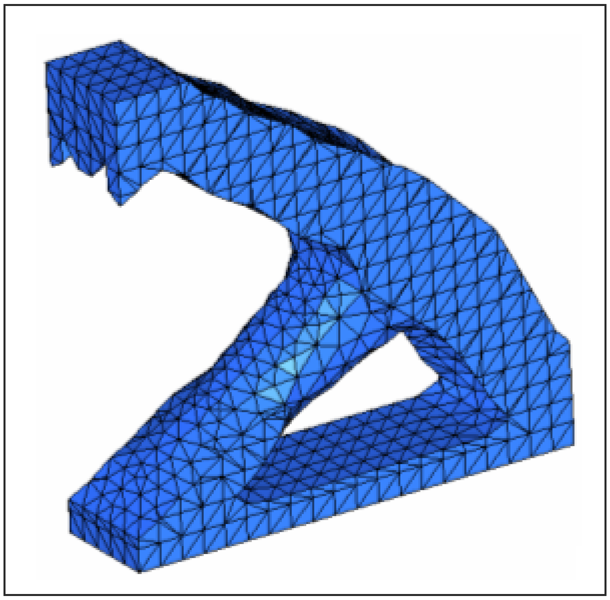


Figure 1804: Fix Boundary of Iso-Surface

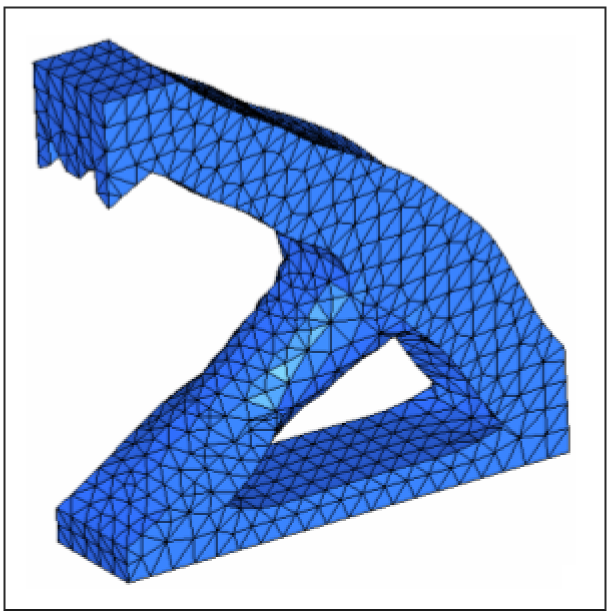


Figure 1805: Smooth Boundary of Iso-surface

The advantages of the `laplacian_smoothing` statement in OSSmooth include:

- The flexibility of controlling the number of smoothing iterations to obtain different degrees of smoothing (possibly a smoothing quality ready for casting). Normally, the iteration number ranges from 5 to 20.

- Smooth boundary of iso-surface with feature angle constrain are seamlessly incorporated into the smoothing process, which is more challenging in a pure CAD system.

Interpretation of Topography Optimization Results

The autobead feature of OSSmooth allows OptiStruct topography optimization results to be interpreted as one or two level beads.

Figure 1806 shows the level of detail captured in both cases; while the 2-level approach captures more details, it is more complicated to manufacture than the 1-level interpretation, often without significant performance gain.

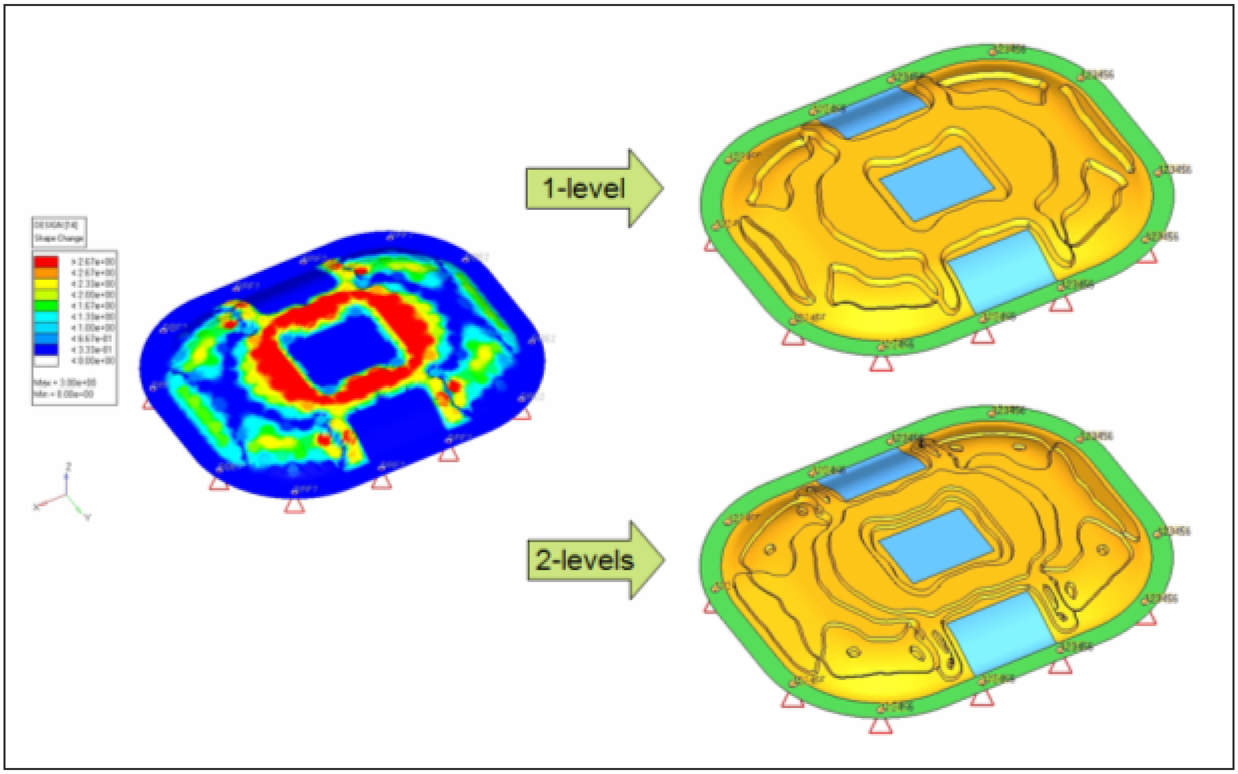


Figure 1806: Autobead Interpretation of Topography Optimization Result

Example: Autobead Result from Topograph Optimizationn

```
#general parameters
input_file             decklid
output_file            decklid.fem
output_code            1

#specific parameters
autobead               1  0.300  1
remesh                 1
```

Figure 1807: Parameter File

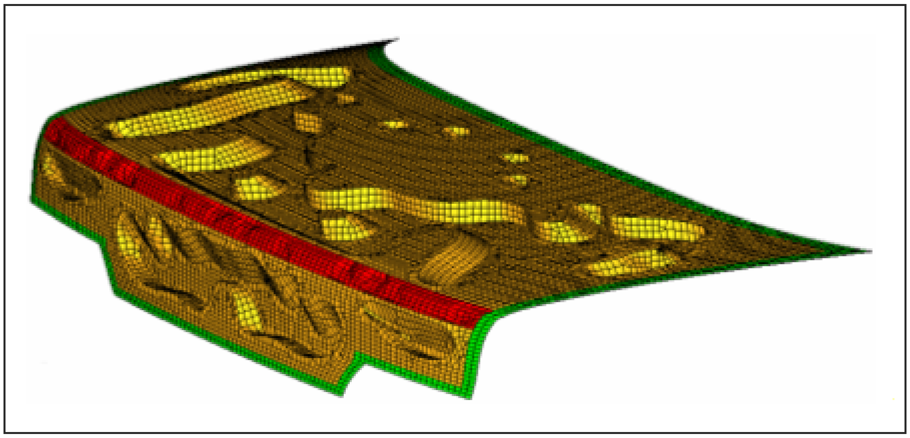


Figure 1808: Autobead Result from Topography Optimization

Example: 2-Layer Autobead Result from Topography Optimization

Some topography performances are relying on the half translation part. OSSmooth can interpolate topography optimization results to 2-layer autobead (autobead third argument 2).

```
#general parameters
input_file              decklid
output_file             decklid.nas
output_code             1

#specific parameters
autobead                1   0.300   2
```

Figure 1809: Parameter File

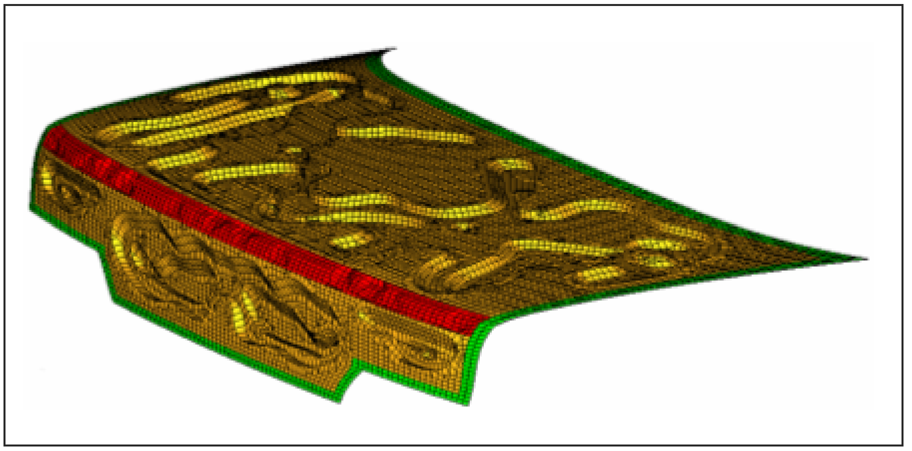


Figure 1810: 2-Layer Autobead Result From Topography Optimization

Interpretation of Composite Free Sizing Optimization Results

A free sizing optimization on composite materials is run to determine the optimal ply shapes and fiber orientation/material selection.

After this optimization phase is run, the next step is to:

- Interpret the optimized ply shapes,
- Smooth the ply boundaries, and
- Setup a subsequent, more detailed size optimization run to determine the number of plies that are required for each ply shape and material.

With the input file for the size optimization phase automatically generated by OptiStruct, handling the ply interpretation and smoothing process can be a tedious, manual process. Altair's ply shape interpretation functionality streamlines this design interpretation process, by allowing the interpretation of optimized ply shapes from a composite free sizing optimization.

During the interpretation process, the boundaries of the ply shapes are smoothed, and options are available to create separate ply entities for disconnected ply patches and for the treatment of very small ply coverage regions.

Additionally, after the smoothing and interpretation process is complete, a ready-to-run OptiStruct input deck is generated using the smoothed ply shapes and updated ply definitions, all while preserving the material definition, properties and fiber orientation.

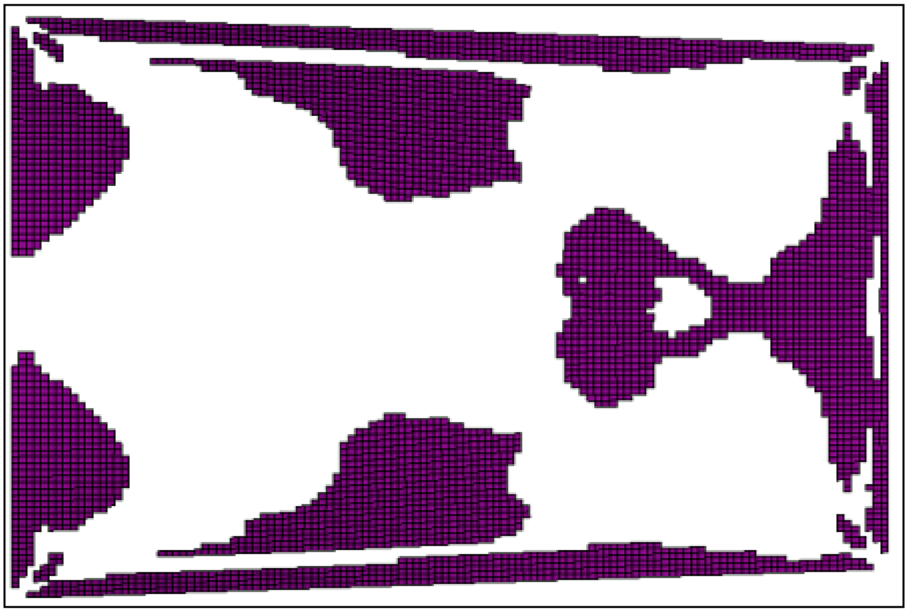


Figure 1811: Ply Shape from Free Size Optimization

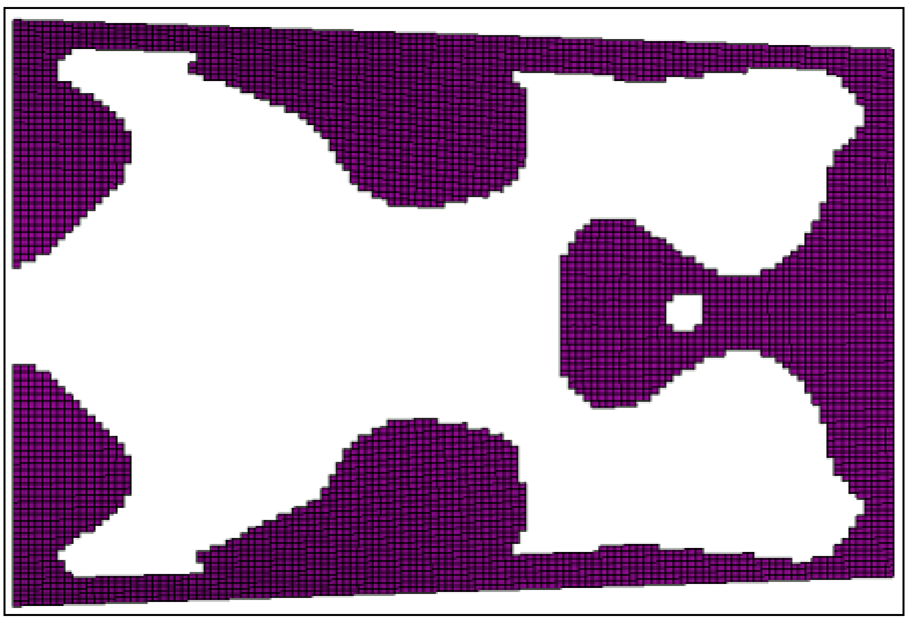


Figure 1812: Smoothed Ply Boundary after Running OSSmooth

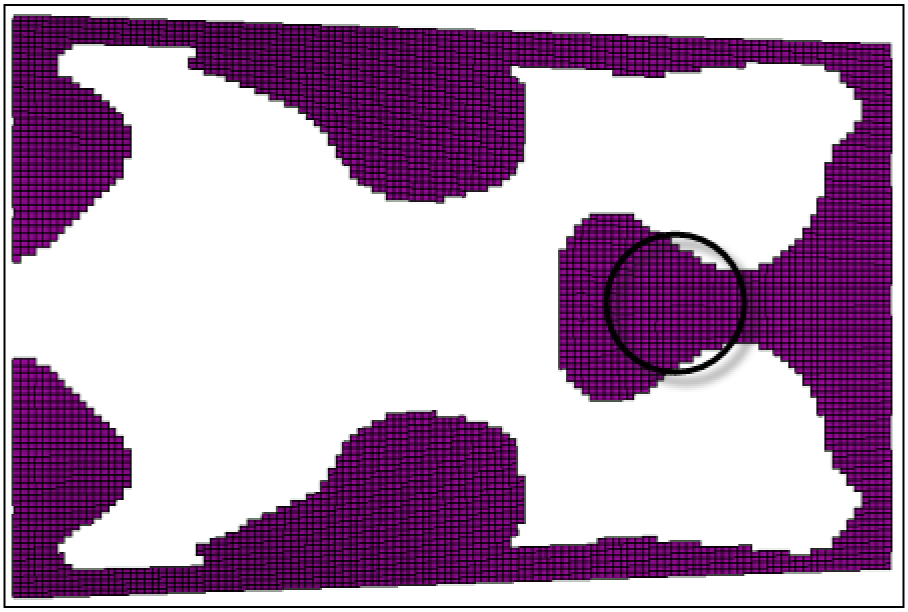


Figure 1813: Additional Treatment of Small Regions

The smooth iterations option defines the number of smoothing iterations to be run. Greater smoothing can be achieved by increasing the number of iterations. The default number of iterations is 20.

The small region option determines how very local and small ply coverage regions are treated. Regions smaller than the threshold value will be eliminated during the smoothing process. Two methods of determining what constitutes a small region are available:

Area ratio
: Local region area / total design area (real value between 0.0 and 1.0. Default=0.01)

Elem count
: Small region identified by number of elements (integer >=0. Default=15)

The split disconnected option can be used to create separate ply entities from disconnected ply coverage regions.

Shape Optimization Results, Surface Reduction and Surface Smoothing

OSSmooth may also be used to reduce and smooth surfaces or the surfaces of a domain.

The parameter statements *pure_surf_reduction* and *pure_surf_smoothing* can be used for this purpose.

The file defined by *input_file* must be in the OptiStruct bulk data format, and OSSmooth can smooth the surface or the surfaces of a domain of the model.

```
#general parameters
input_file            surf
output_file           surf.stl
```

```
output_code          3

#specific parameters
pure_surf_smoothing   2  10  30.000
pure_surf_reduction   1  10.000
```

Figure 1814: Parameter File

FEA Topology for Reanalysis

FEA topology for reanalysis provides an iso-density surface based on the volumetric density information from a topology optimization. Through tetrameshing for 3D models and inheriting boundary conditions, the results from FEA topology can be used for quick reanalysis.

FEA topology can handle both shell and solid elements. For 3D models, the recovered iso-surface can be tetrameshed-by-property automatically. FEA topology provides two options for the processing of non-design elements:

Keep smooth narrow layer
: Retain an artificial layer of elements around the non-design space in the interpretation.

Split all quads
: Split quad elements in the non-design space, if present, to generate a tetra connection between design and non-design regions.

FEA topology preserves boundary conditions by inheriting them from the original model (`<prefix>.fem`). Those boundary conditions unattached to nodes/elements after geometry recovery are deleted to ensure reanalysis.

Figure 1815 and Figure 1816 show FEA topology for reanalysis with following input data definition:

File
: block

Density threshold
: 0.300

Figure 1815 was run with Keep smooth narrow layer around disabled and Split to quads enabled.

-

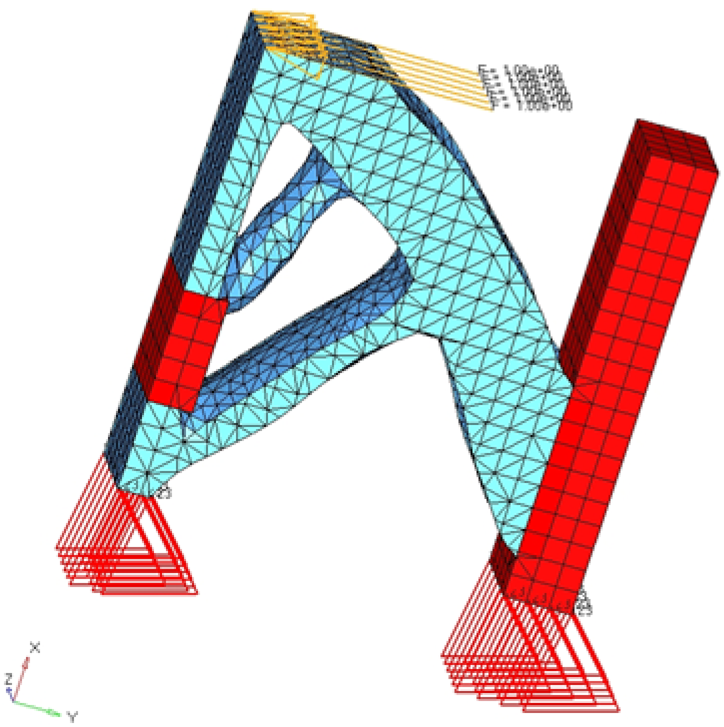


Figure 1815: Result of FEA Topology for Reanalysis

Figure 1816 was run with Keep smooth narrow layer around enabled, and Split all quads disabled. This approach creates a layer of elements around the non-design region and pyramids around the quad elements, if quads exist, to connect to the design space tetrahedral elements.

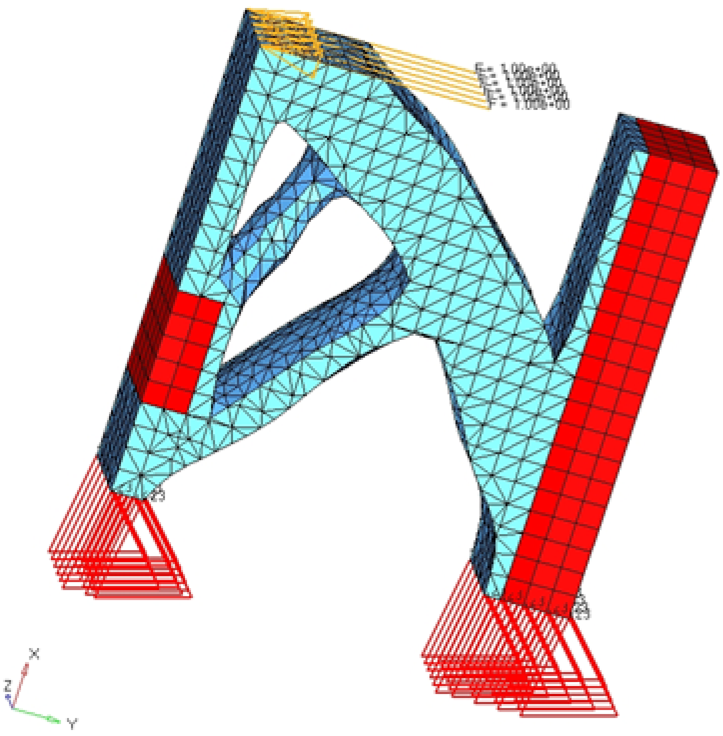


Figure 1816: Result of FEA Topology with a Layer of Elements around Non-Design Space

Tetramesh will be applied on the iso-surface result if there is one close volume at least. The advantages of the tetramesh in FEA topology include:

- Tetramesh can be performed by property.
- The flexibility of controlling the number of tetramesh retries by perturbing the density threshold value, in cases where tetramesh sometimes fails.

FEA Topography for Reanalysis

The FEA topography option in OSSmooth allows the results from an OptiStruct topography optimization to be interpreted as one or two level beads and recover boundary conditions upon geometry extraction. An option for iso surface is also provided for combined use, which performs the same functionality as FEA topology, with FEA topography.

Figure 1817 and Figure 1818 show the level of detail captured in a 1-level bead and 2-level bead case while preserving boundary conditions for quick reanalysis.

The input data definition for a 1-level Autobead extraction is as follows:

Grid file
    brkt

Threshold
    0.300

Layers

1

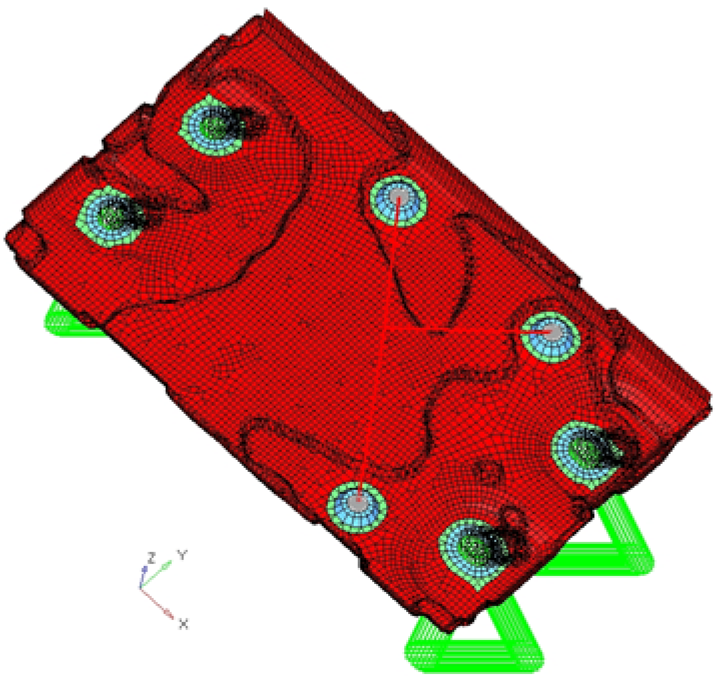


Figure 1817: 1-Level Autobead Result from FEA Topography

A 2-level Autobead extraction is activated with the following input data:

Grid file

brkt

Threshold

0.300

Layers

2

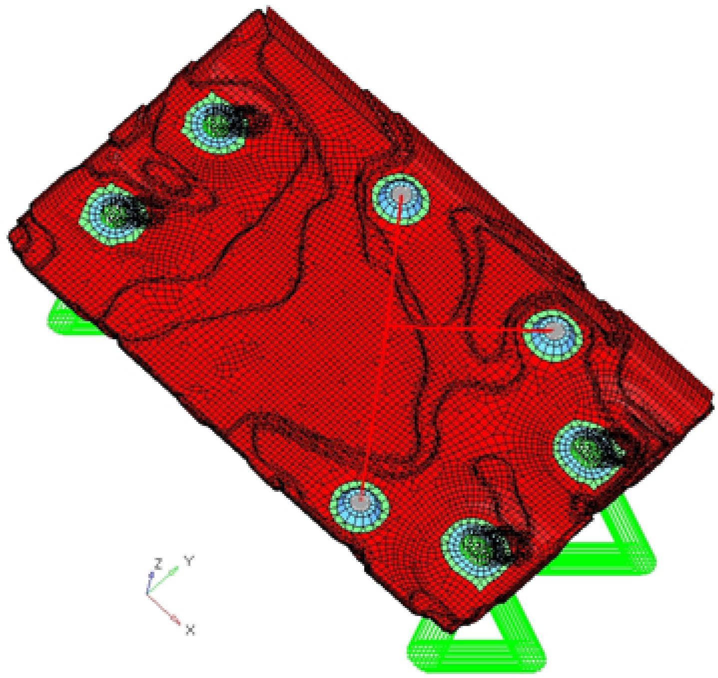


Figure 1818: 2-Level Autobead Result from FEA Topography

Conversion Between Solver Formats

15

Convert finite element models to another solver format.

This chapter covers the following:

Finite element models defined for a particular card image solver generally cannot exchange information with other card image or dictionary format solvers without manipulation of the data.

Convert Model File Solver Format

Convert model files from one solver format to another.

> **Note:** Depending on the solver conversion, certain options, such as defining a source template, a destination template, or a configuration file, may not be present.

1. Load your model in HyperMesh.
2. From the menu bar, select **Tools** > **Convert** > **<solver format to convert from>** > **<solver format to convert to>**.
3. In the that opens asking if you want to load the chosen solver's user profile to continue, click **Yes**. The conversion browser opens.
4. In the Source template field, select the appropriate template for your source solver.
5. In the Destination template field, select the appropriate template for your destination solver. The Configuration File field is automatically populated. The default file, `ConfigurationFile.txt`, provides a valid mapping scheme for the conversion.

   Individual entities are listed in the browser, and color coded green, orange or red to highlight to which extent the card is converted to the targeted solver.
6. Define a Log file location.
7. Click **Conversion Options** to view more options.
8. Review entities.
   - Click an entity to display which card the current entity is mapped to, as well as which parameters are translated.
   - Open the Card Edit panel by right-clicking on an entity and selecting **Edit Card** from the context menu.
9. Click **Convert**.
   A dialog opens, informing you of the status of the conversion. After the conversion, the user profile is automatically updated to the selected solver's format. Click **Close** to close the status window.

Batch Conversion Tools

Batch mode conversion is available for all solver interfaces.

To use batch mode conversion you must run a `.tcl` file in batch mode using `hmbatch.exe`.

Example: Convert Abaqus to OptiStruct

When running this `.tcl` file in batch mode, use the following commands in Windows and Linux:

```
<altair_home>/hm/bin/<platform>/hmbatch.exe -tcl </home/user/my_script.tcl>
```

```
## Bath mode conversion for Abaqus to Optistruct
## Note - Please keep the source model and script in the same folder
## Please replace XXXXX with name of the source deck and the destination files.

set altairHome [ hm_info -appinfo "ALTAIR_HOME" ]
set scriptFolder [ file dirname [info script] ]

*templatefileset "${altairHome}/templates/feoutput/abaqus/standard.3d"

*createstringarray 5 "Abaqus " "Standard3D " "LOADCOLS_DISPLAY_SKIP "
 "CONTACTSURF_DISPLAY_SKIP " "SYSTCOLS_DISPLAY_SKIP "
*feinputwithdata2 "#abaqus/abaqus" "${scriptFolder}/XXXX.inp" 0 0 0 0 0 1 5 1 0

# *readfile "${scriptFolder}/XXXXX.hm"

*solverconvert "Abaqus" "optistruct" "Standard3D" "optistruct" "1"

hm_answernext "yes"
*createstringarray 1 "HM_NODEELEMS_SET_COMPRESS_SKIP "
*feoutputwithdata "${altairHome}/templates/feoutput/optistruct/optistruct"
 "${scriptFolder}/XXXX.fem" 0 0 2 1 1
hm_answernext "yes"
```

Figure 1819: Sample .tcl File

Abaqus Conversion

Abaqus to Nastran Conversion Mapping

The Abaqus to Nastran conversion uses an open conversion scheme; you can specify different mappings in the configuration file.

Care has to be taken so that the element and property mappings are consistent. A valid mapping scheme is provided in the `ConfigurationFile.txt` file. This document explains the scope and limitations of the mapping scheme.

Elements

HyperWorks elements have two basic attributes – configuration (or config) and type. The "config" defines the basic geometrical shape of an element. For example, tria3 configuration is a 3 node triangular element and hexa8 is an 8-node hexahedral element. The "type" defines the solver specific element type of a particular configuration. For example, the 4-node quadrilateral (quad4) element in Abaqus can be any of the following types: S4, S4R, M3D4, R3D4 and so on. The Element Types panel shows all supported element configurations and their types for a user profile.

For a specific configuration, you can map any supported Abaqus element type to any supported Nastran element type, or vice versa. For example, for an Abaqus to Nastran direction, several 2-noded element configurations such as spring, rigid, bar2, rid, and so on are supported. Because all of them are 2-noded elements, conversion across these configurations is also allowed for some element types. For example, CBUSH is of "spring" configuration in the Nastran user profile and CONN3D2 is of "rod" configuration in the Abaqus user profile. It is possible to map a CBUSH to CONN3D2 even though their configurations are different. The element mapping scheme must be under the *ElemTypeConversion block in the `ConfigurationFile.txt` file. You need to provide both configuration and type information to specify the element mapping scheme as shown for the Abaqus to Nastran direction below:

Table 270: Supported Element Mappings

HM configuration	Abaqus type	Nastran type
Mass	MASS	CONM2
	ROTARYI	CONM2
	SPRING1	CELAS1, CELAS2, CBUSH
	DASHPOT1	CDAMP1
	CONN3D2	CBUSH
	CONN2D2	CBUSH
rigid	BEAM	RBE2

HM configuration	Abaqus type	Nastran type
	LINK	RBE2
	PIN	RBE2
	TIE	RBE2
	KINCOUP	RBE2
	COUP_KIN	RBE2
	COUP_DIS	RBE2
	RB3D2	RBE2
	R2D2	RBE2
	RAX2	RBE2
	RB2D2	RBE2
rbe3	DCOUP3D	RBE3
	COUP_DIS	RBE3
	DCOUP2D	RBE3
rigidlink	KINCOUP	RBE2
	RB3D2	RBE2
	BEAM	RBE2
	LINK	RBE2
	PIN	RBE2
	TIE	RBE2
	COUP_KIN	RBE2
	COUP_DIS	RBE2
	R2D2	RBE2
	RAX2	RBE2
spring	SPRING2	CELAS1,CBUSH
	SPRINGA	CBUSH
	DASHPOT2	CDAMP1, CBUSH

HM configuration	Abaqus type	Nastran type
	DASHPOTA	CBUSH
	JOINTC	CBUSH
bar2	B31	CBAR,CBEAM
	B31H	CBAR,CBEAM
	B33	CBAR,CBEAM
	B33H	CBAR,CBEAM
	B31OS	CBAR,CBEAM
	B31OSH	CBAR,CBEAM
	PIPE31	CBAR,CBEAM
	PIPE31H	CBAR,CBEAM
	ELBOW31	CBAR,CBEAM
	ELBOW31B	CBAR,CBEAM
	ELBOW31C	CBAR,CBEAM
	AC1D2	CBAR,CBEAM
	GK3D2	CBAR,CBEAM
	GK3D2N	CBAR,CBEAM
	SAX1	CBAR,CBEAM
	B21	CBAR,CBEAM
	B21H	CBAR,CBEAM
	B23	CBAR,CBEAM
	B23H	CBAR,CBEAM
	PIPE21	CBAR,CBEAM
	PIPE21H	CBAR,CBEAM
	F2D2	CBAR,CBEAM
	FAX2	CBAR,CBEAM
bar3	B32	CBAR,CBEAM

HM configuration	Abaqus type	Nastran type
	B32H	CBAR,CBEAM
	B32OS	CBAR,CBEAM
	B32OSH	CBAR,CBEAM
	PIPE32	CBAR,CBEAM
	PIPE32H	CBAR,CBEAM
	ELBOW32	CBAR,CBEAM
	AC1D3	CBAR,CBEAM
	MGAX2	CBAR,CBEAM
	SFMAX2	CBAR,CBEAM
	SFMGAX2	CBAR,CBEAM
	SAX2	CBAR,CBEAM
	B22	CBAR,CBEAM
	B22H	CBAR,CBEAM
	PIPE22	CBAR,CBEAM
	PIPE22H	CBAR,CBEAM
rod	T3D2	CROD
	T3D2H	CROD
	T3D2T	CROD
	T3D2E	CROD
	MGAX1	CROD
	SFMAX1	CROD
	SFMGAX1	CROD
	CONN3D2	CBUSH
	T2D2	CROD
	T2D2H	CROD
	T2D2T	CROD

HM configuration	Abaqus type	Nastran type
	T2D2E	CROD
	GK2D2	CROD
	GK2D2N	CROD
	CONN2D2	spring CELAS1,CELAS2,CBUSH
gap	GAPUNI	CGAP
	GAPCYL	CGAP
	GAPSPHER	CGAP
tria3	S3	CTRIA3,CTRIAR
	S3R	CTRIA3,CTRIAR
	STRI3	CTRIA3,CTRIAR
	M3D3	CTRIA3,CTRIAR
	SFM3D3	CTRIA3,CTRIAR
	R3D3	CTRIA3,CTRIAR
	DS3	CTRIA3,CTRIAR
	CPE3	CTRIA3,CTRIAR
	CPE3H	CTRIA3,CTRIAR
	CPE3E	CTRIA3,CTRIAR
	CPS3	CTRIA3,CTRIAR
	CPS3E	CTRIA3,CTRIAR
	CAX3	CTRIA3,CTRIAR
	CAX3H	CTRIA3,CTRIAR
	CAX3E	CTRIA3,CTRIAR
	CGAX3	CTRIA3,CTRIAR
	CGAX3H	CTRIA3,CTRIAR
	AC2D3	CTRIA3,CTRIAR
	ACAX3	CTRIA3,CTRIAR

HM configuration	Abaqus type	Nastran type
	DCAX3	CTRIA3,CTRIAR
	DCAX3E	CTRIA3,CTRIAR
	DC2D3	CTRIA3,CTRIAR
	DC2D3E	CTRIA3,CTRIAR
quad4	S4	CQUAD4,CQUADR
	S4R	CQUAD4,CQUADR
	S4R5	CQUAD4,CQUADR
	M3D4	CQUAD4,CQUADR
	M3D4R	CQUAD4,CQUADR
	SFM3D4	CQUAD4,CQUADR
	SFM3D4R	CQUAD4,CQUADR
	R3D4	CQUAD4,CQUADR
	DS4	CQUAD4,CQUADR
	GK3D4L	CQUAD4,CQUADR
	GK3D4LN	CQUAD4,CQUADR
	F3D4	CQUAD4,CQUADR
	CPE4I	CQUAD4,CQUADR
	CPE4	CQUAD4,CQUADR
	CPE4H	CQUAD4,CQUADR
	CPE4IH	CQUAD4,CQUADR
	CPE4R	CQUAD4,CQUADR
	CPE4RH	CQUAD4,CQUADR
	CPE4T	CQUAD4,CQUADR
	CPE4HT	CQUAD4,CQUADR
	CPE4E	CQUAD4,CQUADR
	CPS4	CQUAD4,CQUADR

HM configuration	Abaqus type	Nastran type
	CPS4I	CQUAD4,CQUADR
	CPS4R	CQUAD4,CQUADR
	CPS4T	CQUAD4,CQUADR
	CPS4E	CQUAD4,CQUADR
	CAX4	CQUAD4,CQUADR
	CAX4H	CQUAD4,CQUADR
	CAX4I	CQUAD4,CQUADR
	CAX4IH	CQUAD4,CQUADR
	CAX4R	CQUAD4,CQUADR
	CAX4RH	CQUAD4,CQUADR
	CAX4T	CQUAD4,CQUADR
	CAX4HT	CQUAD4,CQUADR
	CAX4E	CQUAD4,CQUADR
	CAXA4N	CQUAD4,CQUADR
	CAXA4HN	CQUAD4,CQUADR
	CAXA4RN	CQUAD4,CQUADR
	CAXA4RHN	CQUAD4,CQUADR
	CGAX4	CQUAD4,CQUADR
	CGAX4H	CQUAD4,CQUADR
	CGAX4R	CQUAD4,CQUADR
	CGAX4RH	CQUAD4,CQUADR
	AC2D4	CQUAD4,CQUADR
	ACAX4	CQUAD4,CQUADR
	DC2D4	CQUAD4,CQUADR
	DC2D4E	CQUAD4,CQUADR
	DCAX4	CQUAD4,CQUADR

HM configuration	Abaqus type	Nastran type
	DCAX4E	CQUAD4,CQUADR
	DCCAX4	CQUAD4,CQUADR
	DCCAX4D	CQUAD4,CQUADR
	GKPS4	CQUAD4,CQUADR
	GKPE4	CQUAD4,CQUADR
	GKPS4N	CQUAD4,CQUADR
tria6	STRI65	CTRIA6
	M3D6	CTRIA6
	SFM3D6	CTRIA6
	DS6	CTRIA6
	CPE6	CTRIA6
	CPE6H	CTRIA6
	CPE6M	CTRIA6
	CPE6MH	CTRIA6
	CPS6	CTRIA6
	CPS6M	CTRIA6
	AC2D6	CTRIA6
	ACAX6	CTRIA6
	DCAX6	CTRIA6
	DC2D6	CTRIA6
	DCAX6E	CTRIA6
	DC2D6E	CTRIA6
	CAX6	CTRIA6
	CAX6H	CTRIA6
	CAX6M	CTRIA6
	CAX6MH	CTRIA6

HM configuration	Abaqus type	Nastran type
	CGAX6	CTRIA6
	CGAX6H	CTRIA6
quad8	S8R	CQUAD8
	S8R5	CQUAD8
	S8RT	CQUAD8
	M3D8	CQUAD8
	M3D8R	CQUAD8
	SFM3D8	CQUAD8
	SFM3D8R	CQUAD8
	DS8	CQUAD8
	CPE8	CQUAD8
	CPE8H	CQUAD8
	CPE8R	CQUAD8
	CPE8RH	CQUAD8
	CPS8	CQUAD8
	CPS8R	CQUAD8
	AC2D8	CQUAD8
	ACAX8	CQUAD8
	DC2D8	CQUAD8
	DCAX8	CQUAD8
	DCAX8E	CQUAD8
	DC2D8E	CQUAD8
	CAX8	CQUAD8
	CAX8H	CQUAD8
	CAX8HT	CQUAD8
	CAX8R	CQUAD8

HM configuration	Abaqus type	Nastran type
	CAX8RH	CQUAD8
	CAX8RHT	CQUAD8
	CAX8RHT	CQUAD8
	CGAX8	CQUAD8
	CGAX8H	CQUAD8
	CGAX8R	CQUAD8
	CGAX8RH	CQUAD8
	CAXA8N	CQUAD8
	CAXA8HN	CQUAD8
	CAXA8PN	CQUAD8
	CAXA8RN	CQUAD8
	CAXA8RHN	CQUAD8
	CAXA8RPN	CQUAD8
tetra4	C3D4	CTETRA
	C3D4H	CTETRA
	C3D4E	CTETRA
	AC3D4	CTETRA
	DC3D4	CTETRA
	DC3D4E	CTETRA
penta6	C3D6	CPENTA
	C3D6H	CPENTA
	C3D6E	CPENTA
	AC3D6	CPENTA
	GK3D6	CPENTA
	GK3D6N	CPENTA
	SC6R	CPENTA

HM configuration	Abaqus type	Nastran type
	COH3D6	CPENTA
hex8	C3D8I	
	C3D8	
	C3D8T	
	C3D8H	
	C3D8HT	
	C3D8IH	
	C3D8R	
	C3D8RH	
	C3D8E	
	AC3D8	
	DC3D8	
	DC3D8E	
	DCC3D8	
	DCC3D8D	
	SC8R	
	COH3D8	
tetra10	C3D10	DC3D10
	C3D10H	DC3D11
	C3D10M	DC3D12
	C3D10MH	DC3D13
	C3D10E	DC3D14
	DC3D10E	DC3D15
	AC3D10	DC3D16
	DC3D10	DC3D17
penta15	C3D15	CPENTA

HM configuration	Abaqus type	Nastran type
	C3D15H	CPENTA
	C3D15E	CPENTA
	AC3D15	CPENTA
	DC3D15	CPENTA
	DC3D15E	CPENTA
hex20	C3D20	CHEXA
	C3D20H	CHEXA
	C3D20R	CHEXA
	C3D20RH	CHEXA
	C3D20E	CHEXA
	C3D20RE	CHEXA
	C3D20T	CHEXA
	C3D20HT	CHEXA
	C3D20RT	CHEXA
	C3D20RHT	CHEXA
	DC3D20	CHEXA
	AC3D20	CHEXA
	DC3D20E	CHEXA

Abaqus CONN3D2 connector elements are converted to Nastran CBUSH, PBUSH using the following guidelines:

- Connector1 types converted:
  - AXIAL: Active = [1], Rigid = [-]
  - CARTESIAN, PROJECTION CARTESIAN: Active = [123], Rigid = [-]
  - JOIN: Active = [-], Rigid = [123]
  - RADIAL-THRUST: Active = [13]*, Rigid = [-]

    Note: Requires cylindrical system

- SLIDE-PLANE: Active = [23], Rigid = [1]
- SLOT: Active = [1], Rigid = [23]

- Connector2 types converted:
  - ALIGN: Active = [-], Rigid = [456]
  - CARDAN, EULER, ROTATION, FLEXION-TORSION, PROJECTION FLEXION-TORSION: Active = [456], Rigid = [-]
  - REVOLUTE: Active = [4], Rigid = [56]
- Special assembled Connector1 types:
  - BEAM, WELD = (JOIN + ALIGN): Active = [-], Rigid = [123456]
  - CYLINDRICAL = (SLOT + REVOLUTE): Active = [14], Rigid = [2356]
  - HINGE = (JOIN + REVOLUTE): Active = [4], Rigid = [12356]
  - PLANAR = (SLIDE-PLANE + REVOLUTE): Active = [234], Rigid = [156]
  - TRANSLATOR = (SLOT + ALIGN): Active = [1], Rigid = [23456]
  - BUSHING = (PROJECTION CARTESIAN + PROJECTION FLEXION-TORSION): Active = [123456], Rigid = [-]
- PBUSH stiffness and damping values (Ki, Bi) for active DOFs are mapped from *CONNECTOR BEHAVIOR material data. Rigid DOFs map to RIGID option inside PBUSH.
- CBUSH orientation is mapped from *CONNECTOR SECTION Orientation system.
  - Only 1 Orientation system can be mapped to CBUSH CID.
  - If 2 Orientation systems are present in the Abaqus card, HyperMesh only maps the first one.

It is also possible to use a simplified conversion of Abaqus connectors (CONN3D2) to rbe2 elements when modifying the `ConfigurationFile.txt` file. Change the entry for rod element type configuration to: rod,CONN3D2 rigid,rbe2

CONN3D2 elements will now be converted to RBE2 elements. Depending on the connection type set in the CONNECTOR SECTION, such as AXIAL or HINGE, degrees of freedom will be set for the RBE2 element. If systems are associated to the connector elemental nodes they will be assigned to the nodes of the RBE2 as well. Not all connection types are supported. If a system is ignored by a particular CONNECTOR SECTION, it will not be assigned to the nodes of the RBE2 either.

These connector types are currently considered in conversion: AXIAL, JOIN, LINK, SLIDE-PLANE, SLOT, ALIGN, REVOLUTE, BEAM, CYLINDRICAL, HINGE, PLANAR, TRANSLATOR, WELD.

*COUPLING/*KINEMATIC constraints with element based surfaces, currently mapped to groups in HyperMesh, are converted into RBE2 rigid elements. *COUPLING/*DISTRIBUTING constraints are converted to RBE3 elements.

All SPRING and DASHPOT related conversions (including JOINTC) map to CELAS1, CDAMP1, or CBUSH/PBUSH using the following guidelines:

- SPRING1/2 without ORIENTATION converts to CELAS1
- SPRINGA or SPRING1/2 with ORIENTATION converts to CBUSH/PBUSH/PBUSHT with K/KN lines. For SPRING1/2, ORIENTATION maps to CBUSH, CID.
- DASHPOT1/2 without ORIENTATION converts to CDAMP1

- DASHPOTA or DASHPOT1/2 with ORIENTATION converts to CBUSH/PBUSH/PBUSHT with B line. For DASHPOT1/2, ORIENTATION maps to CBUSH, CID.

Sectional Properties

Some of the properties in one solver can be converted to two different sections in the other solver. For an Abaqus to Nastran conversion, for example, *DASHPOT can be converted to *PELAS or PDAMP. The property mapping scheme can be edited under the *PropertyConversion block in the `ConfigurationFile.txt` file.

Please note that the property conversion scheme and corresponding element conversion scheme must be consistent. For example, if you define *CONNECTOR SECTION to PBUSH at the property mapping scheme, the corresponding element CONN3D2 must map to CBUSH in the element mapping scheme.

For SOLID SECTION the converter will always convert to PSOLID unless the property has a data line indicating a cross-sectional area for a truss element. In this case, conversion results in a PROD property.

For BEAM (GENERAL) SECTION the algorithm automatically decides which property to convert to depending on the element type chosen in the ElementTypeConversion section of the `ConfigurationFile.txt`. For example, if you want to convert B31 elements to CBAR, the beam property will get converted to a PBAR or PBARL property. If you choose to convert B31 elements to CBEAM, then the converter creates PBEAM or PBEAML properties accordingly. The same logic applies to B32 elements; the difference is that they are changed to first order beam elements first on conversion.

Table 271: Supported Sectional Property Mappings

Abaqus type	Nastran type
*BEAM GENERAL SECTION	PBAR(L), PBEAM(L)
*BEAM SECTION	PBAR(L), PBEAM(L)
*CONNECTOR SECTION	PELAS,PBUSH
*DASHPOT	PELAS,PDAMP
*GAP	PGAP
*MASS	CONM2
*MEMBRANE SECTION	PSHELL
*ROTARY INERTIA	CONM2
*SHELL GENERAL SECTION	PSHELL
*SHELL SECTION	PSHELL
*SOLID SECTION	PSOLID
*SPRING	PELAS, PBUSH

Abaqus type	Nastran type
*SOLID SECTION (Homogeneous)	PROD
*SHELL GENERAL SECTION (Homogeneous)	PSHELL
*SHELL SECTION (Homogeneous)	PSHELL
*SHELL GENERAL SECTION (User)	PSHELL
*SHELL SECTION (Composite)	PCOMP, PCOMPG
*SHELL GENERAL SECTION (Composite)	PCOMP, PCOMPG

Materials

The material mapping scheme can be edited under *PropertyConversion block in the `ConfigurationFile.txt` file.

Table 272: Supported Material Mappings

Abaqus	Nastran type
*MATERIAL	MAT1, MAT4, MATT1, MATS1
*CONNECTOR BEHAVIOR	PBUSH, PELAS

Loads

HyperMesh loads have two basic attributes – configuration (or config) and type. The supported load "configs" are: force, moment, constraint, pressure, temperature, flux, velocity, acceleration and equation. The load "type" defines the solver specific type of a particular configuration. For example, pressure load can be any of the following Nastran types: PLOAD, PLOAD2 or PLOAD4. The Load Types panel shows all supported load configurations and their types for a user profile.

The converter also converts distributed surfaces loads (*DLSOAD) applied on faces of shell or solid elements into pressure loads (PLOAD4).

For a specific configuration, you can map any supported Abaqus load type to any supported Nastran load type. The conversion tool does not support conversion across load configurations. The load mapping scheme can be edited under the *BCsTypeConversion block in the `ConfigurationFile.txt` file. You need to provide both configuration and type information to specify the mapping scheme as shown below:

Table 273: Supported Load Mappings

HM configuration	Abaqus type	Nastran type
temperature	TEMPERATURE	TEMP

HM configuration	Abaqus type	Nastran type
pressure	DLOAD	PLOAD,PLOAD2,PLOAD4
Constraint	ACCELERATION	SPCD
	VELOCITY	SPCD
	BOUNDARY	SPC,SUPORT
moment	CLOAD	MOMENT
force	CLOAD	FORCE
equation	EQUATION	MPC

Sets

Table 274: Supported Set Mappings

Abaqus type	Nastran type
*NSET	SET
*ELSET	SET

Systems

Table 275: Supported System Mappings

Abaqus type	Nastran type
*ORIENTATION	CORD2C,CORD2R,CORD2S
*SYSTEM	CORD2C,CORD2R,CORD2S
*TRANSFORM	CORD2C,CORD2R,CORD2S
*TRANSFORM- USER DEFINED NSET	CORD2C,CORD2R,CORD2S

Load Steps and Analysis Type

The conversion tool maps between Abaqus steps and Nastran subcases. It does not convert Abaqus analysis type to the solution type. You must define it manually using the Load Step Browser.

The converter converts *STEP into SUBCASE. Load collector references are maintained upon conversion. If multiple load collectors of a particular step contain constraints, a SPCADD card is created automatically. The same happens in case of loads in separate load collectors; a new LOAD card is created on conversion.

Abaqus to Radioss Conversion Mapping

Elements

Table 276: Supported Element Mappings

Abaqus (Explicit)	Radioss
C3D10M	/BRICK
C3D4	/BRICK
C3D6	/BRICK
C3D8R	/BRICK
S3R	/SH3N
S4R	/SHELL
SC8R	/BRICK
SC6R	/PENTA6
R3D3	/SH3N + /RBODY
R3D4	/SHELL + /RBODY
MASS	/ADMAS
M3D4R/M3D3	/PROP/SHELL

Properties

Table 277: Supported Property Mappings

Abaqus (Explicit)	Radioss
*SOLID_SECTION and *CONTROL_SECTION	/PROP/SOLID
*SHELL_SECTION and *CONTROL_SECTION	/PROP/SHELL or PROP/TSHELL
*RIGIDBODY	PROP/SHELL or /PROP/SOLID
*MASS	/ADMAS

Abaqus (Explicit)	**Radioss**
*SHELL SECTION COMPOSITE	/PROP/TSH_COMP(P22_TSH_COMP)

Materials

Table 278: Supported Material Mappings

Abaqus (Explicit)	**Radioss**
ELASTIC, ISOTROPIC	/MAT/ELASTIC
PLASTIC, ISOTROPIC	/MAT/PLAS_TAB
Hyperelastic, Mooney, Rivlin	MAT/OGDEN
Hyperelastic, Ogden	MAT/OGDEN
PLASTIC,JOHNSON COOK	M2PLAS_JOHN_ZERIL
HYPERELASTIC,MARLOW	/MAT/LAW70
HYPERFOAM, VISCOELASTIC	/MAT/LAW62

Functions

Table 279: Supported Function Mappings

Abaqus (Explicit)	**Radioss**
*AMPLITUDE	/FUNCT, /MOVE_FUNCT

Groups

Table 280: Supported Group Mappings

Abaqus (Explicit)	**Radioss**
*NSET	/GRNOD
*ELSET	/GSHEL, /GRSH3N, /GRBRIC

Abaqus (Explicit)	Radioss
*SURFACE	/GRNOD, /SURF

Loads

Table 281: Supported Load Mappings

Abaqus (Explicit)	Radioss
*BOUNDARY	/IMPDISP, /IMPVEL, /IMPACC, /BCS
*INITIAL CONDITION	/INIVEL
*CLOAD	/CLOAD
*DLOAD	/GRAV, /PLOAD

Contacts

Table 282: Supported Contact Mappings

Abaqus (Explicit)	Radioss
*TIED, *CONTACT_CLEARANCE, *SURFACE_INTERACTION	/INTER/TYPE2
*CONTACT_CLEARANCE, *SURFACE_INTERACTION	/INTER/TYPE7
*CONTACT PAIR	/INTER/TYPE24
GENERAL CONTACT	/INTER/TYPE24

Constraints

Table 283: Supported Constraint Mappings

Abaqus (Explicit)	Radioss
*COUPLING KINEMATIC, *COUPLING DYNAMIC	/RBE2, /RBE3
*MPC BEAM	

Abaqus (Explicit)	Radioss
	/RBODY

Output Blocks

Table 284: Supported Output Block Mappings

Abaqus (Explicit)	Radioss
*STEP, STEP NAME	/TITLE and /RUN
*DYNAMIC, T_period	/TSTOP
*VARIABLE MASS SCALING	/DT/NODA/CST
*OUTPUT FIELD	/ANIM/DT, /ANIM/VECT/(ACC, VEL, DISP, FINT, PCONT), /ANIM/BRICK/TENS/(STRESS,STRAIN) / ANIM/SHELL/TENS/(STRESS,STRAIN), /ANIM/ ELEM/(VONMISES, PRESSURE,EPSP, ENER, DENS)
*OUTPUT HISTORY	/TFILE, /TH/NODE, /TH/SHEL, /TH/SH3N, /TH/ BEAM, /TH/BRIC

Abaqus to OptiStruct Conversion Mapping

The Abaqus to OptiStruct conversion uses an open conversion scheme; you can specify different mappings in the configuration file.

Care has to be taken so that the element and property mappings are consistent. A valid mapping scheme is provided in the `ConfigurationFile.txt` file. This document explains the scope and limitations of the mapping scheme.

Contact and Pretension Groups

- Contacts
  - NLPARM card is automatically created and assigned to a nonlinear quasi-static subcase for models containing contacts.
  - *SURFACE INTERACTION property with clearance/pressure vs conductivity is mapped to PCONTHT with TABLED1 cards.
  - *MODELCHANGE conversion is supported for both element set and contact group.
  - *CONTACT PAIR will be mapped to FINITE when SMALL SLIDING is not defined.
  - *CONTACT PAIR will be mapped to its default (SLIDE) when SMALL SLIDING is defined.
  - *CONTACT CONTROLS are converted to CNTSTB and is mapped to loadstep.

- Bolt Pretension
  - *PRETENSION SECTION group maps to a PRETENS entity set with reference to a 1D pretension element or a 3D pretension surface (SURF).
  - *CLOAD applied to a pretension node converts to PTFORCE.
  - *BOUNDARY applied to a pretension node converts to PTADJST.
  - *BOUNDARY, FIXED option in step converts to STATSUB(PRETENS) on a preceding pretension subcase.

Table 285: Supported Contact and Pretension Group Mappings

Abaqus type	HM config, OptiStruct type
*SURFACE (node based)	set, SET
*SURFACE (element based)	contact surface, SURF
*CONTACT PAIR	group, CONTACT
*TIE	group, TIE
*PRETENSION SECTION	set, PRETENS contact surface, SURF
*SURFACE INTERACTION, Separatiopn, No/Yes	Property ,PCONT, Separation, No/Yes
*CONTACT CONTROLS	Load collector, CNTSTB
*MODEL CHANGE	MODCHG
* RIGID BODY	rbody, RBODY
*SHELL TO SOLID COUPLING	Group, TIE

Elements

HyperMesh elements have two basic attributes – configuration (or config) and type. The "config" defines the basic geometrical shape of an element. For example, tria3 configuration is a 3 node triangular element and hexa8 is an 8-node hexahedral element. The "type" defines the solver specific element type of a particular configuration. For example, the 4-node quadrilateral (quad4) element in Abaqus can be any of the following types: S4, S4R, M3D4, R3D4, and so on. The Element Types panel shows all supported element configurations and their types for a user profile.

For a specific configuration, you can map any supported Abaqus element type to any supported OptiStruct element type, or vice versa. For example, for an Abaqus to OptiStruct direction, several 2-noded element configurations such as spring, rigid, bar2, rid, and so on, are supported. Because all of them are 2-noded elements, conversion across these configurations is also allowed for some element types. For example, CBUSH is of "spring" configuration in the OptiStruct user profile and CONN3D2 is of "rod" configuration in the Abaqus user profile. It is possible to map a CBUSH to CONN3D2 even though their configurations are different. The element mapping scheme must be under the *ElemTypeConversion block in the `ConfigurationFile.txt` file. You need to provide both configuration and type information to specify the element mapping scheme.

Table 286: Supported Element Mappings

HM configuration	Abaqus	OptiStruct type
Mass	MASS	CONM2
	ROTARYI	CONM2
	SPRING1	CELAS1, CELAS2, CBUSH
	DASHPOT1	CELAS1, CELAS2, CBUSH
	CONN3D2	CBUSH
	CONN2D2	CBUSH
rigid	BEAM	RBE2
	LINK	RBE2
	PIN	RBE2
	TIE	RBE2
	KINCOUP	RBE2
	COUP_KIN	RBE2
	COUP_DIS	RBE2
	RB3D2	RBE2
	R2D2	RBE2
	RAX2	RBE2
	RB2D2	RBE2
rbe3	DCOUP3D	RBE3
	COUP_DIS	RBE3
	DCOUP2D	RBE3
rigidlink	KINCOUP	RBE2
	RB3D2	RBE2
	BEAM	RBE2

HM configuration	Abaqus	OptiStruct type
	LINK	RBE2
	PIN	RBE2
	TIE	RBE2
	COUP_KIN	RBE2
	COUP_DIS	RBE2
	R2D2	RBE2
	RAX2	RBE2
	RB2D2	RBE2
spring	SPRING2	CELAS1, CBUSH
	SPRINGA	CBUSH
	DASHPOT2	CDAMP1, CBUSH
	DASHPOTA	CBUSH
	JOINTC	CBUSH
bar2	B31	CBAR,CBEAM
	B31H	CBAR,CBEAM
	B33	CBAR,CBEAM
	B33H	CBAR,CBEAM
	B31OS	CBAR,CBEAM
	B31OSH	CBAR,CBEAM
	PIPE31	CBAR,CBEAM
	PIPE31H	CBAR,CBEAM
	ELBOW31	CBAR,CBEAM
	ELBOW31B	CBAR,CBEAM
	ELBOW31C	CBAR,CBEAM
	AC1D2	CBAR,CBEAM
	GK3D2	CBAR,CBEAM

HM configuration	Abaqus	OptiStruct type
	GK3D2N	CBAR,CBEAM
	SAX1	CBAR,CBEAM
	B21	CBAR,CBEAM
	B21H	CBAR,CBEAM
	B23	CBAR,CBEAM
	B23H	CBAR,CBEAM
	PIPE21	CBAR,CBEAM
	PIPE21H	CBAR,CBEAM
	F2D2	CBAR,CBEAM
	FAX2	CBAR,CBEAM
bar3	B32	CBAR,CBEAM
	B32H	CBAR,CBEAM
	B32OS	CBAR,CBEAM
	B32OSH	CBAR,CBEAM
	PIPE32	CBAR,CBEAM
	PIPE32H	CBAR,CBEAM
	ELBOW32	CBAR,CBEAM
	AC1D3	CBAR,CBEAM
	MGAX2	CBAR,CBEAM
	SFMAX2	CBAR,CBEAM
	SFMGAX2	CBAR,CBEAM
	SAX2	CBAR,CBEAM
	B22	CBAR,CBEAM
	B22H	CBAR,CBEAM
	PIPE22	CBAR,CBEAM
	PIPE22H	CBAR,CBEAM

HM configuration	Abaqus	OptiStruct type
rod	T3D2	CROD
	T3D2H	CROD
	T3D2T	CROD
	T3D2E	CROD
	MGAX1	CROD
	SFMAX1	CROD
	SFMGAX1	CROD
	CONN3D2	CROD
	T2D2	CROD
	T2D2H	CROD
	T2D2T	CROD
	T2D2E	CROD
	GK2D2	CROD
	GK2D2N	CROD
	CONN2D2	spring CELAS1,CELAS2,CBUSH
gap	GAPUNI	CGAP
	GAPCYL	CGAP
	GAPSPHER	CGAP
tria3	S3	CTRIA3,CTRIAR
	S3R	CTRIA3,CTRIAR
	STRI3	CTRIA3,CTRIAR
	M3D3	CTRIA3,CTRIAR
	SFM3D3	CTRIA3,CTRIAR
	R3D3	CTRIA3,CTRIAR
	DS3	CTRIA3,CTRIAR
	CPE3	CTRIA3,CTRIAR

HM configuration	Abaqus	OptiStruct type
	CPE3H	CTRIA3,CTRIAR
	CPE3E	CTRIA3,CTRIAR
	CPS3	CTRIA3,CTRIAR
	CPS3E	CTRIA3,CTRIAR
	CAX3	CTRIA3,CTRIAR
	CAX3H	CTRIA3,CTRIAR
	CAX3E	CTRIA3,CTRIAR
	CGAX3	CTRIA3,CTRIAR
	CGAX3H	CTRIA3,CTRIAR
	AC2D3	CTRIA3,CTRIAR
	ACAX3	CTRIA3,CTRIAR
	DCAX3	CTRIA3,CTRIAR
	DCAX3E	CTRIA3,CTRIAR
	DC2D3	CTRIA3,CTRIAR
	DC2D3E	CTRIA3,CTRIAR
quad4	S4	CQUAD4,CQUADR
	S4R	CQUAD4,CQUADR
	S4R5	CQUAD4,CQUADR
	M3D4	CQUAD4,CQUADR
	M3D4R	CQUAD4,CQUADR
	SFM3D4	CQUAD4,CQUADR
	SFM3D4R	CQUAD4,CQUADR
	R3D4	CQUAD4,CQUADR
	DS4	CQUAD4,CQUADR
	GK3D4L	CQUAD4,CQUADR
	GK3D4LN	CQUAD4,CQUADR

HM configuration	Abaqus	OptiStruct type
	F3D4	CQUAD4,CQUADR
	CPE4I	CQUAD4,CQUADR
	CPE4	CQUAD4,CQUADR
	CPE4H	CQUAD4,CQUADR
	CPE4IH	CQUAD4,CQUADR
	CPE4R	CQUAD4,CQUADR
	CPE4RH	CQUAD4,CQUADR
	CPE4T	CQUAD4,CQUADR
	CPE4HT	CQUAD4,CQUADR
	CPE4E	CQUAD4,CQUADR
	CPS4	CQUAD4,CQUADR
	CPS4I	CQUAD4,CQUADR
	CPS4R	CQUAD4,CQUADR
	CPS4T	CQUAD4,CQUADR
	CPS4E	CQUAD4,CQUADR
	CAX4	CQUAD4,CQUADR
	CAX4H	CQUAD4,CQUADR
	CAX4I	CQUAD4,CQUADR
	CAX4IH	CQUAD4,CQUADR
	CAX4R	CQUAD4,CQUADR
	CAX4RH	CQUAD4,CQUADR
	CAX4T	CQUAD4,CQUADR
	CAX4HT	CQUAD4,CQUADR
	CAX4E	CQUAD4,CQUADR
	CAXA4N	CQUAD4,CQUADR
	CAXA4HN	CQUAD4,CQUADR

HM configuration	Abaqus	OptiStruct type
	CAXA4RN	CQUAD4,CQUADR
	CAXA4RHN	CQUAD4,CQUADR
	CGAX4	CQUAD4,CQUADR
	CGAX4H	CQUAD4,CQUADR
	CGAX4R	CQUAD4,CQUADR
	CGAX4RH	CQUAD4,CQUADR
	AC2D4	CQUAD4,CQUADR
	ACAX4	CQUAD4,CQUADR
	DC2D4	CQUAD4,CQUADR
	DC2D4E	CQUAD4,CQUADR
	DCAX4	CQUAD4,CQUADR
	DCAX4E	CQUAD4,CQUADR
	DCCAX4	CQUAD4,CQUADR
	DCCAX4D	CQUAD4,CQUADR
	GKPS4	CQUAD4,CQUADR
	GKPE4	CQUAD4,CQUADR
	GKPS4N	CQUAD4,CQUADR
tria6	STRI65	CTRIA6
	M3D6	CTRIA6
	SFM3D6	CTRIA6
	DS6	CTRIA6
	CPE6	CTRIA6
	CPE6H	CTRIA6
	CPE6M	CTRIA6
	CPE6MH	CTRIA6
	CPS6	CTRIA6

HM configuration	Abaqus	OptiStruct type
	CPS6M	CTRIA6
	AC2D6	CTRIA6
	ACAX6	CTRIA6
	DCAX6	CTRIA6
	DC2D6	CTRIA6
	DCAX6E	CTRIA6
	DC2D6E	CTRIA6
	CAX6	CTRIA6
	CAX6H	CTRIA6
	CAX6M	CTRIA6
	CAX6MH	CTRIA6
	CGAX6	CTRIA6
	CGAX6H	CTRIA6
quad8	S8R	CQUAD8
	S8R5	CQUAD8
	S8RT	CQUAD8
	M3D8	CQUAD8
	M3D8R	CQUAD8
	SFM3D8	CQUAD8
	SFM3D8R	CQUAD8
	DS8	CQUAD8
	CPE8	CQUAD8
	CPE8H	CQUAD8
	CPE8R	CQUAD8
	CPE8RH	CQUAD8
	CPS8	CQUAD8

HM configuration	Abaqus	OptiStruct type
	CPS8R	CQUAD8
	AC2D8	CQUAD8
	ACAX8	CQUAD8
	DC2D8	CQUAD8
	DCAX8	CQUAD8
	DCAX8E	CQUAD8
	DC2D8E	CQUAD8
	CAX8	CQUAD8
	CAX8H	CQUAD8
	CAX8HT	CQUAD8
	CAX8R	CQUAD8
	CAX8RH	CQUAD8
	CAX8RHT	CQUAD8
	CAX8RT	CQUAD8
	CGAX8	CQUAD8
	CGAX8H	CQUAD8
	CGAX8R	CQUAD8
	CGAX8RH	CQUAD8
	CAXA8N	CQUAD8
	CAXA8HN	CQUAD8
	CAXA8PN	CQUAD8
	CAXA8RN	CQUAD8
	CAXA8RHN	CQUAD8
	CAXA8RHN	CQUAD8
tetra4	C3D4	CTETRA
	C3D4H	CTETRA

HM configuration	Abaqus	OptiStruct type
	C3D4E	CTETRA
	AC3D4	CTETRA
	DC3D4	CTETRA
	DC3D4E	CTETRA
	C3D4T	CTETRA
penta6	C3D6	CPENTA
	C3D6H	CPENTA
	C3D6H	CPENTA
	C3D6H	CPENTA
	DC3D6	CGASK6
	DC3D6E	CGASK6
	GK3D6	CPENTA
	GK3D6N	CPENTA
	SC6R	CPENTA
	COH3D6	CPENTA
hex8	C3D8I	CHEXA
	C3D8	CHEXA
	C3D8T	CHEXA
	C3D8H	CHEXA
	C3D8HT	CHEXA
	C3D8IH	CHEXA
	C3D8R	CHEXA
	C3D8RH	CHEXA
	C3D8E	CHEXA
	AC3D8	CHEXA
	DC3D8	CHEXA

HM configuration	Abaqus	OptiStruct type
	DC3D8E	CHEXA
	DCC3D8	CHEXA
	DCC3D8D	CHEXA
	GK3D8	CGASK8
	GK3D8N	CGASK8
	SC8R	CHEXA
	COH3D8	CHEXA
tetra10	C3D10	DC3D10
	C3D10H	DC3D11
	C3D10M	DC3D12
	C3D10MH	DC3D13
	C3D10E	DC3D14
	DC3D10E	DC3D15
	AC3D10	DC3D16
	DC3D10	DC3D17
penta15	C3D15	CPENTA
	C3D15	CPENTA
	C3D15	CPENTA
	C3D15	CPENTA
	C3D15	CPENTA
	C3D15	CPENTA
hex20	C3D20	CHEXA
	C3D20	CHEXA
	C3D20R	CHEXA
	C3D20RH	CHEXA
	C3D20E	CHEXA

HM configuration	Abaqus	OptiStruct type
	C3D20RE	CHEXA
	C3D20T	CHEXA
	C3D20HT	CHEXA
	C3D20RT	CHEXA
	C3D20RHT	CHEXA
	DC3D20	CHEXA
	AC3D20	CHEXA
	DC3D20E	CHEXA
Pyramid	C3D8HT	CPYRA
	C3D20R	CPYRA

Abaqus CONN3D2 connector C3D20R elements are converted to OptiStruct CBUSH, PBUSH or JOINTG using the following guidelines:

- Connector1 types converted:
    - AXIAL: Active = [1], Rigid = [-]
    - CARTESIAN, PROJECTION CARTESIAN: Active = [123], Rigid = [-]
    - JOIN: Active = [-], Rigid = [123]
    - RADIAL-THRUST: Active = [13]*, Rigid = [-]

      Note: Requires cylindrical system
    - SLIDE-PLANE: Active = [23], Rigid = [1]
    - SLOT: Active = [1], Rigid = [23]
- Connector2 types converted:
    - ALIGN: Active = [-], Rigid = [456]
    - CARDAN, EULER, ROTATION, FLEXION-TORSION, PROJECTION FLEXION-TORSION: Active = [456], Rigid = [-]
    - REVOLUTE: Active = [4], Rigid = [56]
- Special assembled Connector1 types:
    - BEAM, WELD = (JOIN + ALIGN): Active = [-], Rigid = [123456]
    - CYLINDRICAL = (SLOT + REVOLUTE): Active = [14], Rigid = [2356]
    - HINGE = (JOIN + REVOLUTE): Active = [4], Rigid = [12356]
    - PLANAR = (SLIDE-PLANE + REVOLUTE): Active = [234], Rigid = [156]

  - TRANSLATOR = (SLOT + ALIGN): Active = [1], Rigid = [23456]
  - BUSHING = (PROJECTION CARTESIAN + PROJECTION FLEXION-TORSION): Active = [123456], Rigid = [-]
- PBUSH stiffness and damping values (Ki, Bi) for active DOFs are mapped from *CONNECTOR BEHAVIOR material data. Rigid DOFs map to RIGID option inside PBUSH.
- CBUSH orientation is mapped from *CONNECTOR SECTION Orientation system.
  - Only 1 Orientation system can be mapped to CBUSH CID.
  - If 2 Orientation systems are present in the Abaqus card, HyperMesh only maps the first one.

It is possible to use a simplified conversion of Abaqus connectors (CONN3D2) to rbe2 elements when modifying `ConfigurationFile.txt` in the following way (change the entry for rod element type configuration: rod,CONN3D2 rigid,rbe2

CONN3D2 elements will now be converted to RBE2 elements. Depending on the connection type set in the CONNECTOR SECTION (such as AXIAL or HINGE), degrees of freedom will be set for the RBE2 element. If systems are associated to the connector elemental nodes they will be assigned to the nodes of the RBE2 as well. Not all connection types are supported. If a system is ignored by a particular CONNECTOR SECTION, it will not be assigned to the nodes of the RBE2 either.

These connector types are currently considered in conversion: AXIAL, JOIN, LINK, SLIDE-PLANE, SLOT, ALIGN, REVOLUTE, BEAM, CYLINDRICAL, HINGE, PLANAR, TRANSLATOR, WELD.

*KINEMATIC COUPLING constraints with element based surfaces (currently mapped to groups in HyperMesh) are converted into RBE2 rigid elements. *DISTRIBUTING COUPLING constraints are converted to RBE3 elements.

All SPRING and DASHPOT related conversions (including JOINTC) map to CELAS1, CDAMP1, or CBUSH/PBUSH using the following guidelines:

- SPRING1/2 without ORIENTATION converts to CELAS1
- SPRINGA or SPRING1/2 with ORIENTATION converts to CBUSH/PBUSH/PBUSHT with K/KN lines. For SPRING1/2, ORIENTATION maps to CBUSH, CID.
- DASHPOT1/2 without ORIENTATION converts to CDAMP1
- DASHPOTA or DASHPOT1/2 with ORIENTATION converts to CBUSH/PBUSH/PBUSHT with B line. For DASHPOT1/2, ORIENTATION maps to CBUSH, CID.
- Convertor has an additional option: **Convert *SPRING defined with Orientation to CBUSH**. If this option is enabled, *SPRING is converted to CBUSH if not enabled to CELAS.

Refer to the following table for Connector Section values:

Connector Section	Values
AXIAL	AXIAL
AXIAL, ALIGN	AXIAL, ORIENT
BEAM (JOIN + ALIGN)	BALL + ORIENT OR CBUSH, Pbush (RIGID all)
LINK	RROD

Connector Section	Values
LINK, ALIGN	RROD, ORIENT
JOIN	BALL
JOIN, ROTATION	BALL, CARDAN
HINGE (JOIN + REVOLUTE)	REVOLUTE, BALL
SLOT	INLINE
CYLINDRICAL (SLOT + REVOLUTE)	INLINE + REVOLUTE
TRANSLATOR (SLOT + ALGIN)	INLINE, ORIENT
CARTESIAN	CARTESIAN
SLIDE-PLANE	INLINE
UJOINT (JOIN + UNIVERSAL)	BALL, UNIVERSAL
HINGE	RBAR + REVOLUTE

Nodal thickness is mapped to respective grid post conversions.

Configuration file paths are selected in batch mode conversion. Conversion options can be used in batch mode conversion by editing the configuration file.

Sectional Properties

Some of the properties in one solver can be converted to two different sections in the other solver. For an Abaqus to OptiStruct conversion, for example, *DASHPOT can be converted to *PELAS or PDAMP. The property mapping scheme can be edited under the *PropertyConversion block in the `ConfigurationFile.txt` file.

The property conversion scheme and corresponding element conversion scheme must be consistent. For example, if you define *CONNECTOR SECTION to PBUSH at the property mapping scheme, the corresponding element CONN3D2 must map to CBUSH in the element mapping scheme.

For SOLID SECTION the converter will always convert to PSOLID unless the property has a data line indicating a cross-sectional area for a truss element. In this case conversion results in a PROD property.

For BEAM (GENERAL) SECTION the algorithm automatically decides which property to convert to depending on the element type chosen in the ElementTypeConversion section of the `ConfigurationFile.txt`. For example, if you want to convert B31 elements to CBAR, the beam property will get converted to a PBAR or PBARL property. If you choose to convert B31 elements to

CBEAM, then the converter creates PBEAM or PBEAML properties accordingly. The same logic applies to B32 elements; the difference is that they are changed to first order beam elements first on conversion.

Table 287: Supported Sectional Property Mappings

Abaqus type	**OptiStruct type**
*SURFACE INTERACTION	PCONT
*SURFACE BEHAVIOR, PRESSURE -OVERCLOSURE = EXPONENTIAL	PCONT SOFT YES + STFEXP
*FRICTION	PCONT
*CLEARENCE	PCONT
*GASKET SECTION	PGASK
*BEAM GENERAL SECTION	PBAR(L), PBEAM(L)
*BEAM SECTION	PBAR(L), PBEAM(L)
*CONNECTOR SECTION	PELAS,PBUSH
*DASHPOT	PELAS,PDAMP
*GAP	PGAP
*MASS	CONM2
*MEMBRANE SECTION	PSHELL
*ROTARY INERTIA	
	CONM2
*SHELL GENERAL SECTION	PSHELL
*SHELL GENERAL SECTION	PSHELL
*SHELL SECTION	PSOLID
*SOLID SECTION	PELAS, PBUSH, PBUSHT
*SPRING	PROD
*SOLID SECTION (Homogeneous)	PSHELL
*SHELL GENERAL SECTION (Homogeneous)	PSHELL
*SHELL GENERAL SECTION (User)	PSHELL

Abaqus type	OptiStruct type
*SHELL SECTION (Composite)	PCOMP, PCOMPG
*SHELL GENERAL SECTION (Composite)	PCOMP, PCOMPG

Materials

The material mapping scheme can be edited under *PropertyConversion block in the `ConfigurationFile.txt` file.

Table 288: Supported Material Mappings

Abaqus type		OptiStruct type
*MATERIAL	*ELASTIC, ISOTROPIC	MAT1
	*ELASTIC, ISOTROPIC E, poisson's ratio, T	MATT1 with TABLEM1 for E and poisson's ratio
	*ELASTIC, LAMINA	MAT8
	*PLASTIC stress (x), plastic strain (y)	MATS1 with TYPSTRN=1 for plastic strain. TABLES1 with stress (y) vs plastic strain (x)
	*PLASTIC stress (x), plastic strain (y), Temp	For each T, need a separate TABLES1. TABLEST to define T and corresponding TABLES1.
	*SPECIFIC HEAT	MAT4, CP
	*CONDUCTIVITY	MAT4, K
	*EXPANSION expansion coeff, T	MATT1 with TABLEM1 for expansion coeff (A)
*GASKET BEHAVIOR	*GASKET ELASTICITY, COMPONENT = MEMBRANE	MGASK + MAT1
	*GASKET ELASTICITY, COMPONENT = TRANSVERSE SHEAR	MGASK + MAT1 GPL field of MGASK card

Abaqus type		**OptiStruct type**
	*GASKET THICKNESS BEHAVIOR, DIRECTION = LOADING pressure (x), closure (y)	TABLES1 curve with pressure (y) vs closure (x) definition TABLES1 referred in TABLED field of MGASK
	*GASKET THICKNESS BEHAVIOR, DIRECTION = LOADING for TYPE = ELASTO-PLASTIC	BEHAV = 0 in MGASK
	*GASKET THICKNESS BEHAVIOR, DIRECTION = LOADING for TYPE = DAMAGE	BEHAV = 1 in MGASK
	*GASKET THICKNESS BEHAVIOR, DIRECTION=LOADING for TYPE = DAMAGE or ELASTO-PLASTIC with TENSILE STIFFNESS FACTOR	EPL in MGASK
	*GASKET THICKNESS BEHAVIOR, DIRECTION = UNLOADING pressure, closure, Max closure	For "n" max/plastic closure values, creates "n" TABLES1 for individual unloading pressure vs closure curves. TABLES1 referred in TABLU1 TABLUn fields in MGASK.
	*EXPANSION expansion coeff, T	
*MATERIAL	*CONDUCTIVITY	MATT4,TABLEM1
*MATERIAL	*HYPERELASTIC,OGDEN	MATHE,OGDEN,TABLES1
*ELASTIC, TYPE = ENGINEERING CONSTANTS		MAT9ORT
*CONNECTOR BEHAVIOR		PBUSH, PELAS

Loads

HM loads have two basic attributes – configuration (or config) and type. The supported load "configs" are: force, moment, constraint, pressure, temperature, flux, velocity, acceleration and equation. The load "type" defines the solver specific type of a particular configuration. For example, pressure load can be any of the following OptiStruct types: PLOAD, PLOAD2, or PLOAD4. The Load Types panel shows all supported load configurations and their types for a user profile.

The converter also converts distributed surfaces loads (*DLSOAD) applied on faces of shell or solid elements into pressure loads (PLOAD4).

Temperature with *INTIAL condition is converted to TEMP(INTIAL) and mapped to Global Case Control.

FILM loads are converted to CHBDYE elements, sink temperatures are converted to SPC and heat transfer coefficient(H) with PCONV.

SFILM loads are converted to CHBDYE elements, sink temperatures are converted to SPC and heat transfer coefficient(H) with PCONV.

Distributed surfaces loads (*DLSOAD) applied on faces of shell or solid elements into pressure loads (PLOAD4).

For a specific configuration, you can map any supported Abaqus load type to any supported OptiStruct load type. The conversion tool does not support conversion across load configurations. The load mapping scheme can be edited under the *BCsTypeConversion block in the `ConfigurationFile.txt` file. You need to provide both configuration and type information to specify the mapping scheme.

Table 289: Supported Load Mappings

HM configuration	Abaqus type	OptiStruct type
temperature	TEMPERATURE	TEMP
pressure	DLOAD	PLOAD,PLOAD2,PLOAD4
pressure	DLOAD, ROTA	RACC
pressure	CENTRIFUGAL	RFORCE
pressure	FILM	SPC
	SFILM	SPC
	DFLUX	QBDY1
Constraint	ACCELERATION	SPCD
	VELOCITY	SPCD
	BOUNDARY	SPC,SUPORT
	BOUNDARY on pretension node	PTADJST
moment	CLOAD	MOMENT
force	CLOAD	FORCE
	CLOAD on pretension node	PTFORCE

HM configuration	Abaqus type	OptiStruct type
equation	EQUATION	MPC
temperature	BOUNDARY	SPC
Connector Load	CONNECTOR LOAD	LOADJG
Connector Motion	CONNECTOR MOTION	MOTIONJG

Sets

Table 290: Supported Set Mappings

Abaqus type	OptiStruct type
*NSET	SET
*ELSET	SET

Control Cards

- Node set and element set from output block for displacement and stress are mapped.
- PARAM, INREL, -2 is created if there is *INERTIA RELIEF in the OptiStruct deck.
- PARAM, RBE3COL is created if there are rbe3 elements in the OptiStruct deck.

Systems

Table 291: Supported System Mappings

Abaqus type	OptiStruct type
*ORIENTATION	CORD2C,CORD2R,CORD2S
*SYSTEM	CORD2C,CORD2R,CORD2S
*TRANSFORM	CORD2C,CORD2R,CORD2S
*TRANSFORM- USER DEFINED NSET	CORD2C,CORD2R,CORD2S

Control Cards

Displacement, Stress and Contact pressure output card is created on conversion and is mapped to respective loadstep.

If OUTPUTBLOCK is not mapped to any loadstep these are updated in GLOBAL CASE CONTROL.

*RESTART,WRITE,OVRELAY is converted to OS RESTARTW= 1 ,COVER

Load Steps and Analysis Type

The converter converts *STEP into respective supported SUBCASE. Load collector references are maintained upon conversion. If multiple load collectors of a particular step have constraints, an SPCADD card is created automatically. Similarly, a LOAD card is created for homogeneous loads and mapped to step.

For models containing contacts, NLPARM card is automatically created and assigned to a nonlinear quasi-static subcase.

Abaqus models that contain both conductivity and temperature defined under *CONDUCTIVITY in *MATERIAL is converted to a non-linear Heat transfer (NLHEAT) analysis type in OptiStruct.

NLGEOM is set to YES in a loadstep with id 'i', CNTNLSUB will be set checked on, OPTION set to YES and SCID points to the previous loadstep in loadstep with id 'i+1'. The chain continues for the loadsteps.

*PERTURBUTION steps are converted to linear static.

*FREQUENCY RESIDUAL=YES is converted to RESVEC=YES on conversion.

*FREQUENCY (analysis type) is converted to normal modes with the EIGRL card (load collector) mapped in the loadstep on conversion.

*STATIC load step wuth Int_INC is mapped to NLPARM load collector with NINC=1/Int_INC.

*STATIC load step with MinIcr is mapped to NLADAPT,DTMIN load collector.

*STATIC load step with MaxIncr is mapped to NLADAPT,DTMAX load collector.

*STEP with unsymm=yes is converted to PARAM, UNSYMSLV.

ANSYS Conversion

ANSYS to Abaqus Conversion Mapping

Materials

Modulus of Elasticity, Poisson's ratio and density are mapped.

Table 292: Supported Material Mappings

ANSYS type	Abaqus type	HM Configuration
MASS21	MASS	Mass
COMBI14	SPRING2	Spring
BEAM4	B31	Bar2
LINK8	T3D2	Rod
SHELL43, SHELL63, SHELL181	S3	Tria3
SHELL43, SHELL63, SHELL181	S4	Quad4
SHELL93	STRI65	Tria6
SHELL93	S8R	Quad8
SOLID45, SOLID62, SOLID64, SOLID69, SOLID70, SOLID96, SOLID97, SOLID164, SOLID185	C3D4	Tetra4
SOLID62, SOLID70, SOLID96, SOLID97, SOLID164	C3D8	Pyramid5
SOLID5, SOLID45, SOLID62, SOLID64, SOLID69, SOLID70, SOLID96, SOLID97, SOLID164, SOLID185	C3D6	Penta6
SOLID5, SOLID45, SOLID62, SOLID64, SOLID69, SOLID70, SOLID96, SOLID97, SOLID164, SOLID185	C3D81	Hex8
SOLID87, SOLID90, SOLID92, SOLID95, SOLID98, SOLID117,	C3D10	Tetra10

ANSYS type	Abaqus type	HM Configuration
SOLID148, SOLID168, SOLID186, SOLID187, SOLID191		
SOLID90, SOLID95, SOLID117, SOLID186	C3D20	Pyramid13
SOLID90, SOLID95, SOLID117, SOLID147, SOLID186, SOLID191	C3D15	Penta15
SOLID90, SOLID95, SOLID117, SOLID147, SOLID186, SOLID191		Hex20
MPC184	BEAM_RIGID	Rigid
RBE3, CONTA175, TARGE170	COUP_DIS	RBE3
CP	COUP_KIN	Rigid
COMBIN39	SPRINGA	Spring

Properties

Table 293: Supported Property Mappings

ANSYS type	Abaqus type	Convert Parameters
MASS21p	MASS	none
COMBI14p	SPRING	none
BEAM4p, LINK8p	BEAMSECTION	none
SHELL43p, SHELL63p, SHELL93p	SHELLSECTION	Shell Thickness
SOLID45p, SOLID70p, SOLID95p, SOLID117p, SOLID191p	SOLIDSECTION	none
BEAM4p	BEAMGENSECTION	Area, Ixx,Iyy,Izz

> **Note:** If ANSYS solid elements do not have a property, then one SOLIDSECTION property will be created by using this conversion tool.

Loads and Boundary Conditions

Loads and boundary conditions will not be converted on using this conversion tool.

ANSYS to Nastran Conversion Mapping

Conversion Notes

- All ANSYS solid and shell elements that are currently supported are included in the conversion tool.
- Material properties – Thermal Conductivity, specific heat, thermal expansion co-efficient can be now converted.
- Nonlinear material data and temperature dependent materials defined through TB tables are converted to TABLESx cards inside a MATS1.
- 2D and 3D Orthotropic properties of E, NU, and Alpha can be converted.
- If components or properties do not have a material associated to them on conversion, then MID = 1 will be associated.
- On conversion, a model will be cleaned by removing unused sets, ET types, and contact elements.
- COMBIN14 elements are now converted to CELAS1 if key option2 for COMBIN14 in ANSYS is defined. If key option 3 is defined, it is converted to 3 coincident CELAS (1 per DOF).
- ANSYS BEAM188 converts to CBEAM now [please note section properties defined using SECDATA card in ANSYS are not yet converted].
- Forces, Moments, Pressures, Temperatures and Constraints are automatically converted. MODOPT (control cards) will be mapped to EIRGL. ACEL (Control Cards) will be converted to GRAV.
- BEAM44 element type end release options defined in KEYOPTIONS 7 and 8 are converted to values in fields PA and PB respectively in the CBEAM card.
- BEAM188 section properties are converted from the BeamSection Collector data and not from the EtType Sensor (this is how the BEAM188 section data is modeled in the HyperMesh ANSYS User Profile).
    - SECTYPE with TYPE equal to BEAM and SUBTYPEs equal to RECT, CSOLID, CTUBE, I, L, T convert to a PBEAML property with corresponding standard section definition.
    - SECTYPE with TYPE equal to BEAM and SUBTYPEs equal to ASEC, QUAD, CHAN, Z, HATS, HREC convert to a PBEAM property with corresponding cross section property values.
- MPC184 converts to either a RBE2 rigid element or a CBUSH/PBUSH bushing element depending on the KEYOPT(1) value, as follows:
    - KEYOPT(1) equal to 0 converts to RBE2 with constraint DOFs equal to [123]
    - KEYOPT(1) equal to 1 converts to RBE2 with constraint DOFs equal to [123456]
    - KEYOPT(1) equal to 6 converts to CBUSH + PBUSH
        - K4 equals <blank>/0.0

        - K1, K2, K3, K5, K6 equals 1e10
    - KEYOPT(1) equal to 8 converts to CBUSH + PBUSH
        - K1, K4, K5, K6 equals <blank>/0.0
        - K2, K3 equals 1E10
    - KEYOPT(1) equal to 9 converts to CBUSH + PBUSH
        - K2, K3, K4, K5, K6 equals <blank>/0.0
        - K1 equals 1E10
    - KEYOPT(1) equal to 10 converts to CBUSH + PBUSH
        - K1 equals <blank>/0.0
        - K2, K3, K4, K5, K6 equals 1E10
    - KEYOPT(1) equal to 11 converts to CBUSH + PBUSH
        - K1, K4 equals <blank>/0.0
        - K2, K3, K5, K6 equals 1E10
    - KEYOPT(1) equal to 12 converts to CBUSH + PBUSH
        - K2, K3, K4 equals <blank>/0.0
        - K1, K5, K6 equals 1E10
    - KEYOPT(1) equal to 13 converts to RBE2 with constraint DOFs equal to [123456]
    - KEYOPT(1) equal to 14 converts to RBE2 with constraint DOFs equal to [456]
    - KEYOPT(1) equal to 15 converts to RBE2 with constraint DOFs equal to [123]

Assumptions

- The Conversion tool will only convert entities imported by the ANSYS FE input reader.
- For shell elements, constant thickness parts are assumed.
- Analysis types assumed: Static – gravity, forces, pressures and constraints; Modal.

Elements

Table 294: Supported Element Mappings

ANSYS	Nastran
mass, CONTAC175	sets, GRID
mass, MASS21	mass, CONM2
rigid, CP	equation, MPC
rigid, CERIG	rbe2, RBE2
rigidlink, CP	equation, MPC
rigidlink, CERIG	equation, MPC
rbe3/RBE3	rbe3/RBE3

ANSYS	Nastran
spring, CONBIN14	spring, CBUSH
bar2, BEAM44	bar2, CBEAM
bar2, BEAM188	bar2, CBEAM
bar2, PIPE16	bar2, CBEAM
bar2, BEAM4	bar2, CBEAM
rod, LINK8	rod, CROD
rod, MPC184	rbe2,RBE2 / spring,CBUSH
tria3, PLANE182	tria3, CTRIA3
tria3, SHELL181	tria3, CTRIA3
tria3, SHELL163	tria3, CTRIA3
tria3, PLANE162	tria3, CTRIA3
tria3, SHELL157	tria3, CTRIA3
tria3, SHELL143	tria3, CTRIA3
tria3, SHELL131	tria3, CTRIA3
tria3, PLANE75	tria3, CTRIA3
tria3, PLANE67	tria3, CTRIA3
tria3, SHELL63	tria3, CTRIA3
tria3, SHELL57	tria3, CTRIA3
tria3, PLANE55	tria3, CTRIA3
tria3, SHELL43	tria3, CTRIA3
tria3, PLANE42	tria3, CTRIA3
tria3, SHELL41	tria3, CTRIA3
tria3, PLANE25	tria3, CTRIA3
tria3, PLANE13	tria3, CTRIA3
quad4, PLANE182	quad4, CQUAD4
quad4, SHELL181	quad4, CQUAD4

ANSYS	Nastran
quad4, SEHLL163	quad4, CQUAD4
quad4, PLANE162	quad4, CQUAD4
quad4, SHELL157	quad4, CQUAD4
quad4, SHELL143	quad4, CQUAD4
quad4, SHELL131	quad4, CQUAD4
quad4, PLANE75	quad4, CQUAD4
quad4, PLANE67	quad4, CQUAD4
quad4, SHELL63	quad4, CQUAD4
quad4, SHELL57	quad4, CQUAD4
quad4, PLANE55	quad4, CQUAD4
quad4, SHELL43	quad4, CQUAD4
quad4, PLANE42	quad4, CQUAD4
quad4, SHELL41	quad4, CQUAD4
quad4, SHELL28	quad4, CQUAD4
quad4, PLANE25	quad4, CQUAD4
quad4, PLANE13	quad4, CQUAD4
tetra4, SOLID185	tetra4, CTETRA
tetra4, SOLID164	tetra4, CTETRA
tetra4, SOLID97	tetra4, CTETRA
tetra4, SOLID96	tetra4, CTETRA
tetra4, SOLID73	tetra4, CTETRA
tetra4, SOLID72	tetra4, CTETRA
tetra4, SOLID70	tetra4, CTETRA
tetra4, SOLID69	tetra4, CTETRA
tetra4, SOLID64	tetra4, CTETRA
tetra4, SOLID62	tetra4, CTETRA

ANSYS	Nastran
tetra4, SOLID46	tetra4, CTETRA
tetra4, SOLID45	tetra4, CTETRA
tetra4, SOLID285	tetra4, CTETRA
penta6, SOLSH190	penta6, CPENTA
penta6, SOLID185	penta6, CPENTA
penta6, SOLID164	penta6, CPENTA
penta6, VISCO107	penta6, CPENTA
penta6, SOLID97	penta6, CPENTA
penta6, SOLID96	penta6, CPENTA
penta6, SOLID73	penta6, CPENTA
penta6, SOLID70	penta6, CPENTA
penta6, SOLID69	penta6, CPENTA
penta6, SOLID64	penta6, CPENTA
penta6, SOLID62	penta6, CPENTA
penta6, SOLID46	penta6, CPENTA
penta6, SOLID45	penta6, CPENTA
penta6, SOLID5	penta6, CPENTA
hex8, SOLSH190	hex8, CHEXA
hex8, SOLID185	hex8, CHEXA
hex8, SOLID164	hex8, CHEXA
hex8, SOLID97	hex8, CHEXA
hex8, SOLID96	hex8, CHEXA
hex8, SOLID73	hex8, CHEXA
hex8, SOLID70	hex8, CHEXA
hex8, SOLID69	hex8, CHEXA
hex8, SOLID64	hex8, CHEXA

ANSYS	Nastran
hex8, SOLID62	hex8, CHEXA
hex8, SOLID46	hex8, CHEXA
hex8, SOLID45	hex8, CHEXA
hex8, SOLID5_1	hex8, CHEXA
tria6, PLANE183	tria6, CTRIA6
tria6, SHELL150	tria6, CTRIA6
tria6, PLANE145	tria6, CTRIA6
tria6, SHELL132	tria6, CTRIA6
tria6, SHELL281	tria6, CTRIA6
tria6, PLANE121	tria6, CTRIA6
tria6, SHELL99	tria6, CTRIA6
tria6, SHELL93	tria6, CTRIA6
tria6, SHELL91	tria6, CTRIA6
tria6, PLANE83	tria6, CTRIA6
tria6, PLANE82	tria6, CTRIA6
tria6, PLANE78	tria6, CTRIA6
tria6, PLANE77	tria6, CTRIA6
tria6, PLANE53	tria6, CTRIA6
tria6, PLANE35	tria6, CTRIA6
tria6, PLANE2	tria6, CTRIA6
quad8, PLANE183	quad8, CQUAD8
quad8, SHELL150	quad8, CQUAD8
quad8, PLANE145	quad8, CQUAD8
quad8, SHELL132	quad8, CQUAD8
quad8, SHELL281	quad8, CQUAD8
quad8, PLANE121	quad8, CQUAD8

ANSYS	Nastran
quad8, SHELL99	quad8, CQUAD8
quad8, SHELL93	quad8, CQUAD8
quad8, SHELL91	quad8, CQUAD8
quad8, PLANE83	quad8, CQUAD8
quad8, PLANE82	quad8, CQUAD8
quad8, PLANE78	quad8, CQUAD8
quad8, PLANE77	quad8, CQUAD8
quad8, PLANE53	quad8, CQUAD8
tetra10, SOLID191	tetra10, CTETRA
tetra10, SOLID187	tetra10, CTETRA
tetra10, SOLID186	tetra10, CTETRA
tetra10, SOLID148	tetra10, CTETRA
tetra10, SOLID117	tetra10, CTETRA
tetra10, SOLID98	tetra10, CTETRA
tetra10, SOLID95	tetra10, CTETRA
tetra10, SOLID92	tetra10, CTETRA
tetra10, SOLID90	tetra10, CTETRA
tetra10, SOLID87	tetra10, CTETRA
penta15, SOLID191	penta15, CPENTA
penta15, SOLID186	penta15, CPENTA
penta15, SOLID147	penta15, CPENTA
penta15, SOLID117	penta15, CPENTA
penta15, SOLID95	penta15, CPENTA
penta15, SOLID90	penta15, CPENTA
hex20, SOLID191	hex20, CHEXA
hex20, SOLID191	hex20, CHEXA

ANSYS	Nastran
hex20, SOLID191	hex20, CHEXA
hex20, SOLID191	hex20, CHEXA
hex20, SOLID191	hex20, CHEXA
hex20, SOLID191	hex20, CHEXA

Properties

Table 295: Supported Property Mappings

ANSYS	Nastran
LINK8P	LINK8P
BEAM4P	PBEAM
BEAM44P	PBEAM
SOLID45P	PSOLID
SOLID46P	PSOLID
SOLID70P	PSOLID
SOLID95P	PSOLID
SOLID97P	PSOLID
SOLID117P	PSOLID
SOLID185P	PSOLID
SOLID191P	PSOLID
SHELL28P	PSHELL
SHELL41P	PSHELL
SHELL43P	PSHELL
SHELL57P	PSHELL
SHELL63P	PSHELL
SHELL91P	PCOMP
SHELL93P	PSHELL

ANSYS	Nastran
SHELL99P	PCOMP
SHELL131P	PCOMP
SHELL132P	PCOMP
SHELL143P	PSHELL
SHELL150P	PSHELL
SHELL157P	PSHELL
SHELL181P	PSHELL
SHELL281P	PSHELL
PLANE2P	PSHELL
PLANE42P	PSHELL
PLANE82P	PSHELL
PLANE145P	PSHELL
PLANE183P	PSHELL
PIPE16P	PBEAML
COMBIN14P	PBUSH
MASS21P	CONM2

Materials

Table 296: Supported Material Mappings

ANSYS	Nastran
MP	MAT1

ANSYS	Nastran
MPDATA	MAT1

Loads

Table 297: Supported Load Mappings

ANSYS	Nastran
temperature, BF_TEMP	temperature, TEMP
temperature, IC_TEMP	temperature, TEMP
temperature, D_TEMP	temperature, TEMP
pressure, HFLUX	pressure, PLOAD2
pressure, ConvBulkTe	pressure, PLOAD
pressure, PRESSURE	pressure, PLOAD4
constraint, D_VOLT	constraint, ASET1
constraint, IC_CONSTRNT	constraint, SPCD
constraint, D_CONSTRAIN	constraint, SPC
moment, FORCE2	moment, MOMENT

ANSYS	Nastran
force, FORCE	force, FORCE

Sets

Table 298: Supported Set Mappings

ANSYS	Nastran
set	set

Cards

Table 299: Supported Card Mappings

ANSYS	Nastran
ACEL	GRAV

Sensors

Table 300: Supported Sensor Mappings

ANSYS	Nastran
Link8	PROD
BEAM4	PBEAM
BEAM44	PBEAM
BEAM188	PBEAML
SOLID5	PSOLID
SOLID5	PSOLID
SOLID46	PSOLID
SOLID62	PSOLID
SOLID64	PSOLID
SOLID69	PSOLID
SOLID70	PSOLID

ANSYS	Nastran
SOLID87	PSOLID
SOLID90	PSOLID
SOLID92	PSOLID
SOLID95	PSOLID
SOLID96	PSOLID
SOLID97	PSOLID
SOLID98	PSOLID
SOLID117	PSOLID
SOLID147	PSOLID
SOLID148	PSOLID
SOLID164	PSOLID
SOLID185	PSOLID
SOLID186	PSOLID
SOLID187	PSOLID
SOLID191	PSOLID
SOLSH190	PSOLID
SOLID226	PSOLID
SOLID227	PSOLID
SOLID285	PSOLID
SHELL28	PSHELL
SHELL41	PSHELL
SHELL43	PSHELL
SHELL51	PSHELL
SHELL57	PSHELL
SHELL63	PSHELL
SHELL91	PCOMP

ANSYS	Nastran
SHELL93	PSHELL
SHELL99	PCOMP
SHELL131	PCOMP
SHELL132	PCOMP
SHELL143	PSHELL
SHELL150	PSHELL
SHELL157	PSHELL
SHELL181	PCOMP OR PSHELL
SHELL281	PCOMP OR PSHELL
PLANE2	PSHELL
PLANE13	PSHELL
PLANE25	PSHELL
PLANE35	PSHELL
PLANE42	PSHELL
PLANE67	PSHELL
PLANE75	PSHELL
PLANE77	PSHELL
PLANE78	PSHELL
PLANE82	PSHELL
PLANE83	PSHELL
PLANE121	PSHELL
PLANE145	PSHELL
PLANE162	PSHELL
PLANE183	PSHELL
PLANE223	PSHELL
PIPE16	PBEAM

ANSYS	Nastran
COMBIN14	PBUSH
MASS21	CONM2
MPC184	PBUSH

ANSYS to OptiStruct Conversion Mapping

Conversion Notes

- All ANSYS solid and shell elements that are currently supported are included in the conversion tool.
- Material properties – Thermal Conductivity, specific heat, thermal expansion co-efficient can be now converted.
- Nonlinear material data and temperature dependent materials defined through TB tables are converted to TABLESx cards inside a MATS1.
- 2D and 3D Orthotropic properties of E, NU, and Alpha can be converted.
- If components or properties do not have a material associated to them on conversion, then MID equal to 1 will be associated.
- On conversion, a model will be cleaned by removing unused sets, ET types, and contact elements.
- COMBIN14 elements are now converted to CELAS1 if key option2 for COMBIN14 in ANSYS is defined. If key option 3 is defined, it is converted to 3 coincident CELAS (1 per DOF).
- Forces, Moments, Pressures, Temperatures and Constraints are automatically converted. MODOPT (control cards) will be mapped to EIRGL. ACEL (Control Cards) will be converted to GRAV.
- Surface to Surface and Node to Surface contact pairs can be converted to OptiStruct. Contact element pairs CONTA173-TARGE170 and CONTA174-TARGE170 are converted to SURF (contact surfaces) in OptiStruct, and CONTA175 is converted to Grid SET. A CONTACT group for each contact pair is created during conversion.
- Composite data conversion: ANSYS composite elements and layer information is converted to OptiStruct composite elements and composite property information.
- BEAM44 element type, end release options defined in KEYOPTIONS 7 and 8 are converted to values in fields PA and PB respectively in CBEAM card.
- BEAM188 section properties are converted from the BeamSection Collector data and not from the EtType Sensor (this is how the BEAM188 section data is modeled in the HyperMesh ANSYS User Profile).
    - SECTYPE with TYPE equal to BEAM and SUBTYPEs equal to RECT, CSOLID, CTUBE, I, L, T convert to a PBEAML property with corresponding standard section definition.
    - SECTYPE with TYPE equal to BEAM and SUBTYPEs equal to ASEC, QUAD, CHAN, Z, HATS, HREC convert to a PBEAM property with corresponding cross section property values.

- MPC184 converts to either a RBE2 rigid element or a CBUSH/PBUSH bushing element depending on the KEYOPT(1) value, as follows:
    - KEYOPT(1) equal to 0 converts to RBE2 with constraint DOFs equal to [123]
    - KEYOPT(1) equal to 1 converts to RBE2 with constraint DOFs equal to [123456]
    - KEYOPT(1) equal to 6 converts to CBUSH + PBUSH
        - K4 equals <blank>/0.0
        - K1, K2, K3, K5, K6 equals RIGID
    - KEYOPT(1) equal to 8 converts to CBUSH + PBUSH
        - K1, K4, K5, K6 equals <blank>/0.0
        - K2, K3 equals RIGID
    - KEYOPT(1) equal to 9 converts to CBUSH + PBUSH
        - K2, K3, K4, K5, K6 equals <blank>/0.0
        - K1 equals RIGID
    - KEYOPT(1) equal to 10 converts to CBUSH + PBUSH
        - K1 equals <blank>/0.0
        - K2, K3, K4, K5, K6 equals RIGID
    - KEYOPT(1) equal to 11 converts to CBUSH + PBUSH
        - K1, K4 equals <blank>/0.0
        - K2, K3, K5, K6 equals RIGID
    - KEYOPT(1) equal to 12 converts to CBUSH + PBUSH
        - K2, K3, K4 equals <blank>/0.0
        - K1, K5, K6 equals RIGID
    - KEYOPT(1) equal to 13 converts to RBE2 with constraint DOFs equal to [123456]
    - KEYOPT(1) equal to 14 converts to RBE2 with constraint DOFs equal to [456]
    - KEYOPT(1) equal to 15 converts to RBE2 with constraint DOFs equal to [123]

Asseumptions:

- The Conversion tool only converts entities imported by the ANSYS FE input reader.
- For shell elements, constant thickness parts are assumed.
- Analysis types assumed: Static – gravity, forces, pressures and constraints; Modal.

Elements

Table 301: Supported Element Mappings

ANSYS	OptiStruct
mass, CONTAC175	sets, GRID
mass, MASS21	mass, CONM2
rigid, CP	equation, MPC

ANSYS	OptiStruct
rigid, CERIG	rbe2, RBE2
rigidlink, CP	equation, MPC
rigidlink, CERIG	equation, MPC
rbe3/RBE3	rbe3, RBE3
spring, COMBIN14	spring, CBUSH
bar2, BEAM44	bar2, CBEAM
bar2, BEAM188	bar2, CBEAM
bar2, PIPE16	bar2, CBEAM
bar2, BEAM4	bar2, CBEAM
rod, LINK8	rod, CROD
rod, MPC184	rbe2,RBE2 / spring,CBUSH
tria3, PLANE182	tria3, CTRIA3
tria3, SHELL181	tria3, CTRIA3
tria3, CONTA173	contact surf
tria3, TARGE170	contact surf
tria3, SHELL163	tria3, CTRIA3
tria3, PLANE162	tria3, CTRIA3
tria3, SHELL157	tria3, CTRIA3
tria3, SHELL143	tria3, CTRIA3
tria3, SHELL131	tria3, CTRIA3
tria3, PLANE75	tria3, CTRIA3
tria3, PLANE67	tria3, CTRIA3
tria3, SHELL63	tria3, CTRIA3
tria3, SHELL57	tria3, CTRIA3
tria3, PLANE55	tria3, CTRIA3
tria3, SHELL43	tria3, CTRIA3

ANSYS	OptiStruct
tria3, PLANE42	tria3, CTRIA3
tria3, SHELL41	tria3, CTRIA3
tria3, PLANE25	tria3, CTRIA3
tria3, PLANE13	tria3, CTRIA3
quad4, PLANE182	quad4, CQUAD4
quad4, SHELL181	quad4, CQUAD4
quad4, CONTA173	contact surf
quad4, TARGE170	contact surf
quad4, SEHLL163	quad4, CQUAD4
quad4, PLANE162	quad4, CQUAD4
quad4, SHELL157	quad4, CQUAD4
quad4, SHELL143	quad4, CQUAD4
quad4, SHELL131	quad4, CQUAD4
quad4, PLANE75	quad4, CQUAD4
quad4, PLANE67	quad4, CQUAD4
quad4, SHELL63	quad4, CQUAD4
quad4, SHELL57	quad4, CQUAD4
quad4, PLANE55	quad4, CQUAD4
quad4, SHELL43	quad4, CQUAD4
quad4, PLANE42	quad4, CQUAD4
quad4, SHELL41	quad4, CQUAD4
quad4, SHELL28	quad4, CQUAD4
quad4, PLANE25	quad4, CQUAD4
quad4, PLANE13	quad4, CQUAD4
tetra4, SOLID185	tetra4, CTETRA
tetra4, SOLID164	tetra4, CTETRA

ANSYS	OptiStruct
tetra4, SOLID97	tetra4, CTETRA
tetra4, SOLID96	tetra4, CTETRA
tetra4, SOLID73	tetra4, CTETRA
tetra4, SOLID72	tetra4, CTETRA
tetra4, SOLID70	tetra4, CTETRA
tetra4, SOLID69	tetra4, CTETRA
tetra4, SOLID64	tetra4, CTETRA
tetra4, SOLID62	tetra4, CTETRA
tetra4, SOLID46	tetra4, CTETRA
tetra4, SOLID45	tetra4, CTETRA
tetra4, SOLID285	tetra4, CTETRA
penta6, SOLSH190	penta6, CPENTA
penta6, SOLID185	penta6, CPENTA
penta6, SOLID164	penta6, CPENTA
penta6, VISCO107	penta6, CPENTA
penta6, SOLID97	penta6, CPENTA
penta6, SOLID96	penta6, CPENTA
penta6, SOLID73	penta6, CPENTA
penta6, SOLID70	penta6, CPENTA
penta6, SOLID69	penta6, CPENTA
penta6, SOLID64	penta6, CPENTA
penta6, SOLID62	penta6, CPENTA
penta6, SOLID46	penta6, CPENTA
penta6, SOLID45	penta6, CPENTA
penta6, SOLID5	penta6, CPENTA
hex8, SOLSH190	hex8, CHEXA

ANSYS	OptiStruct
hex8, SOLID185	hex8, CHEXA
hex8, SOLID164	hex8, CHEXA
hex8, SOLID97	hex8, CHEXA
hex8, SOLID96	hex8, CHEXA
hex8, SOLID73	hex8, CHEXA
hex8, SOLID70	hex8, CHEXA
hex8, SOLID69	hex8, CHEXA
hex8, SOLID64	hex8, CHEXA
hex8, SOLID62	hex8, CHEXA
hex8, SOLID46	hex8, CHEXA
hex8, SOLID45	hex8, CHEXA
hex8, SOLID5_1	hex8, CHEXA
tria6, PLANE183	tria6, CTRIA6
tria6, CONTA174	contact surf
tria6, TARGE170	contact surf
tria6, SHELL150	tria6, CTRIA6
tria6, PLANE145	tria6, CTRIA6
tria6, SHELL132	tria6, CTRIA6
tria6, SHELL281	tria6, CTRIA6
tria6, PLANE121	tria6, CTRIA6
tria6, SHELL99	tria6, CTRIA6
tria6, SHELL93	tria6, CTRIA6
tria6, SHELL91	tria6, CTRIA6
tria6, PLANE83	tria6, CTRIA6
tria6, PLANE82	tria6, CTRIA6
tria6, PLANE78	tria6, CTRIA6

ANSYS	OptiStruct
tria6, PLANE77	tria6, CTRIA6
tria6, PLANE53	tria6, CTRIA6
tria6, PLANE35	tria6, CTRIA6
tria6, PLANE2	tria6, CTRIA6
quad8, PLANE183	quad8, CQUAD8
quad8, CONTA174	contact surf
quad8, TARGE170	contact surf
quad8, SHELL150	quad8, CQUAD8
quad8, PLANE145	quad8, CQUAD8
quad8, SHELL132	quad8, CQUAD8
quad8, SHELL281	quad8, CQUAD8
quad8, PLANE121	quad8, CQUAD8
quad8, SHELL99	quad8, CQUAD8
quad8, SHELL93	quad8, CQUAD8
quad8, SHELL91	quad8, CQUAD8
quad8, PLANE83	quad8, CQUAD8
quad8, PLANE82	quad8, CQUAD8
quad8, PLANE78	quad8, CQUAD8
quad8, PLANE77	quad8, CQUAD8
quad8, PLANE53	quad8, CQUAD8
tetra10, SOLID191	tetra10, CTETRA
tetra10, SOLID187	tetra10, CTETRA
tetra10, SOLID186	tetra10, CTETRA
tetra10, SOLID148	tetra10, CTETRA
tetra10, SOLID117	tetra10, CTETRA
tetra10, SOLID98	tetra10, CTETRA

ANSYS	OptiStruct
tetra10, SOLID95	tetra10, CTETRA
tetra10, SOLID92	tetra10, CTETRA
tetra10, SOLID90	tetra10, CTETRA
tetra10, SOLID87	tetra10, CTETRA
penta15, SOLID191	penta15, CPENTA
penta15, SOLID186	penta15, CPENTA
penta15, SOLID147	penta15, CPENTA
penta15, SOLID117	penta15, CPENTA
penta15, SOLID95	penta15, CPENTA
penta15, SOLID90	penta15, CPENTA
hex20, SOLID191	hex20, CHEXA
hex20, SOLID186	hex20, CHEXA
hex20, SOLID147	hex20, CHEXA
hex20, SOLID117	hex20, CHEXA
hex20, SOLID95	hex20, CHEXA
hex20, SOLID90	hex20, CHEXA

Properties

Table 302: Supported Property Mappings

ANSYS	OptiStruct
LINK8P	LINK8P
BEAM4P	PBEAM
BEAM44P	PBEAM
SOLID45P	PSOLID
SOLID46P	PSOLID
SOLID70P	PSOLID

ANSYS	OptiStruct
SOLID95P	PSOLID
SOLID97P	PSOLID
SOLID117P	PSOLID
SOLID185P	PSOLID
SOLID191P	PSOLID
SHELL28P	PSHELL
SHELL41P	PSHELL
SHELL43P	PSHELL
SHELL57P	PSHELL
SHELL63P	PSHELL
SHELL91P	PCOMP
SHELL93P	PSHELL
SHELL99P	PCOMP
SHELL131P	PCOMP
SHELL132P	PCOMP
SHELL143P	PSHELL
SHELL150P	PSHELL
SHELL157P	PSHELL
SHELL181P	PSHELL
SHELL281P	PSHELL
PLANE2P	PSHELL
PLANE42P	PSHELL
PLANE82P	PSHELL
PLANE145P	PSHELL
PLANE183P	PSHELL
TARGE170P	PCONT

ANSYS	OptiStruct
CONTAC173P	PCONT
CONTAC174P	PCONT
PIPE16P	PBEAML
COMBIN14P	PBUSH
MASS21P	CONM2

Materials

Table 303: Supported Material Mappings

ANSYS	OptiStruct
MP	MAT1
MPDATA	MAT1

Loads

Table 304: Supported Load Mappings

ANSYS	OptiStruct
temperature, BF_TEMP	temperature, TEMP
temperature, IC_TEMP	temperature, TEMP
temperature, D_TEMP	temperature, TEMP
pressure, HFLUX	pressure, PLOAD2
pressure, ConvBulkTe	pressure, PLOAD
pressure, PRESSURE	pressure, PLOAD4
constraint, D_VOLT	constraint, ASET1
constraint, IC_CONSTRNT	constraint, SPCD
constraint, D_CONSTRAIN	constraint, SPC
moment, FORCE2	moment, MOMENT

ANSYS	OptiStruct
force, FORCE	force, FORCE

Sets

Table 305: Supported Set Mappings

ANSYS	OptiStruct
set	set
mass, CONTA175	set, GRID

Cards

Table 306: Supported Card Mappings

ANSYS	OptiStruct
ACEL	GRAV

Sensors

Table 307: Supported Sensor Mappings

ANSYS	OptiStruct
Link8	PROD
BEAM4	PBEAM
BEAM44	PBEAM
BEAM188	PBEAML
SOLID5	PSOLID
SOLID45	PSOLID
SOLID46	PSOLID
SOLID62	PSOLID
SOLID64	PSOLID
SOLID69	PSOLID

ANSYS	OptiStruct
SOLID70	PSOLID
SOLID87	PSOLID
SOLID90	PSOLID
SOLID92	PSOLID
SOLID95	PSOLID
SOLID96	PSOLID
SOLID97	PSOLID
SOLID98	PSOLID
SOLID117	PSOLID
SOLID147	PSOLID
SOLID148	PSOLID
SOLID164	PSOLID
SOLID185	PSOLID
SOLID186	PSOLID
SOLID187	PSOLID
SOLID191	PSOLID
SOLSH190	PSOLID
SOLID226	PSOLID
SOLID227	PSOLID
SOLID285	PSOLID
SHELL28	PSHELL
SHELL41	PSHELL
SHELL43	PSHELL
SHELL51	PSHELL
SHELL57	PSHELL
SHELL63	PSHELL

ANSYS	OptiStruct
SHELL91	PCOMP
SHELL93	PSHELL
SHELL99	PCOMP
SHELL131	PCOMP
SHELL132	PCOMP
SHELL143	PSHELL
SHELL150	PSHELL
SHELL157	PSHELL
SHELL181	PCOMP OR PSHELL
SHELL281	PCOMP OR PSHELL
PLANE2	PSHELL
PLANE13	PSHELL
PLANE25	PSHELL
PLANE35	PSHELL
PLANE42	PSHELL
PLANE67	PSHELL
PLANE75	PSHELL
PLANE77	PSHELL
PLANE78	PSHELL
PLANE82	PSHELL
PLANE83	PSHELL
PLANE121	PSHELL
PLANE145	PSHELL
PLANE162	PSHELL
PLANE183	PSHELL
PLANE223	PSHELL

ANSYS	OptiStruct
TARGE170	PCONT
CONTAC173	PCONT
CONTAC174	PCONT
PIPE16	PBEAM
COMBIN14	PBUSH
MASS21	CONM2
MPC184	PBUSH

LS-DYNA Conversion

LS-DYNA to Nastran Conversion Mapping

Elements

Table 308: Supported Element Mappings

LS-DYNA	Nastran
*ELEMENT_MASS	CONM2
*ELEMENT_INERTIA	CONM2
*ELEMENT_BEAM	CBAR
*ELEMENT_SHELL	CTRIA3,CQUAD4, CQUAD8, CTRIA6
*ELEMENT_SOLID	CTETRA4/CHEXA8/CPENTA6
*CONSTRAINED_SPOTWELD	RBAR
*ELEMENT_PLOTEL	PLOTEL
*CONSTRAINED_INTERPOLATION	RBE3
*CONSTRAINED_NODE_SET	RBE2
*ELEMENT_DISCRETE	CELAS1

Properties

Table 309: Supported Property Mappings

LS-DYNA	Nastran
*SECTION_SOLID	PSOLID
*SECTION_SHELL	PSHELL
*SECTION_BEAM	PBAR, PBARL

*SECTION_BEAM with ELFORM 1 is mapped to the PBARL property.

*SECTION_BEAM with ELFORM 2 is mapped to the PBAR property.

Materials

All the LS-DYNA materials are mapped MAT1 in Nastran.

Boundary Conditions

Table 310: Supported Boundary Condition Mappings

LS-DYNA	Nastran
*BOUNDARY_SPC	SPC
*LOAD_NODE_POINT	FORCE, MOMENT
*LOAD_SEGMENT	PLOAD4
*INITIAL_TEMP	TEMP

Coordinate System

Table 311: Supported Coordinate System Mappings

LS-DYNA	Nastran
*DEFINE_COORDINATE_NODE	CORD1R
*DEFINE_COORDINATE_SYSTEM	CORD2R

LS-DYNA to Radioss Conversion Mapping

Constrained Definitions

Table 312: Supported Constrained Definition Mappings

LS-DYNA	Radioss
*CONSTRAINED_JOINT_REVOLUTE	/SPRING,/PROP/KJOINT2/ property type 2
*CONSTRAINED_JOINT_SPHERICAL	/SPRING,/PROP/KJOINT2/ property type 1
*CONSTRAINED_JOINT_CYLINDRICAL	/SPRING,/PROP/KJOINT2/ property type 3
*CONSTRAINED_JOINT_TRANSLATIONAL	/PROP/KJOINT2/ property type6

LS-DYNA	Radioss
*CONSTRAINED_JOINT_STIFFNESS_GENERALIZED	/SPRING
*CONSTRINED_RIGID_BODY	/RBODY

Control Cards

Table 313: Supported Control Card Mappings

LS-DYNA	Radioss
*CONTROL_ADAPTIVE	ADMESH_GLOBAL, ADMESH_SET

Contact Interfaces

Table 314: Supported Contact Interface Mappings

LS-DYNA	Radioss
*CONTACT_AUTOMATIC_GENERAL	INTER/TYPE7
*CONTACT_AUTOMATIC_GENERAL_INTERIOR	INTER/TYPE7
*CONTACT_AUTOMATIC_NODES_TO_SURFACE	INTER/TYPE7
*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_S	INTER/TYPE7
*CONTACT_AUTOMATIC_SINGLE_SURFACE	INTER/TYPE7
*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE	INTER/TYPE7
*CONTACT_INTERIOR	INTER/TYPE7
*CONTACT_AUTOMATIC_SINGLE_SURFACE	INTER/TYPE7

Components

Table 315: Supported Component Mappings

LS-DYNA	Radioss
*DAMPING_PARTS_OPTIONS	PART
*PART_OPTIONS	PART

LS-DYNA	Radioss
*INCLUDE_STAMPED_PART	PART

Curves

Table 316: Supported Curve Mappings

LS-DYNA	Radioss
*DEFINE_CURVE	FUNCT

Elements

Table 317: Supported Element Mappings

LS-DYNA	Radioss
*ELEMENT_SHELL	SHELL, SH3N
*ELEMENT_SOLID	BRICK, TETRA44
*ELEMENT_MASS	ADMAS
*ELEMENT_BEAM	BEAM
*ELEMENT_DISCRETE	SPRING2N
*ELEMENT_SEATBELT_ACCELEROMETER	ACCEL

Loads

Table 318: Supported Load Mappings

LS-DYNA	Radioss
*BOUNDARY_SPC_NODE/SET	BCS
*LOAD_NODE_POINT/SET	CLOAD

LS-DYNA	Radioss
*INITIAL_VELOCITY_NODE/GENERATION/ RIGID BODY	INIVEL

Materials

Table 319: Supported Material Mappings

LS-DYNA	Radioss
*MAT_ELASTIC	M1_ELASTIC
*MAT_PLASTIC_KINEMATIC	M36_PLASTIC_TAB
*MAT_NULL	M0_VOID
*MAT_RIGID	M1_ELASTIC (nodes of material defined as / RBODY)
*MAT_PIECEWISE_LINEAR_PLASTICITY	M36_PLAS_TAB
*MAT_SIMPLIFIED_JOHNSON_COOK	M2_PLAS_JOHNS_ZERIL
*MAT_BLATZ_-KO_RUBBER	M42_OGDEN
*MAT_SPRING_NONLINEAR_ELASTIC	P13_SPR_BEAM
*MAT_SPOTWELD_TITLE	P13_SPR_BEAM
*MAT_SPRING_ELASTIC	/PROP/TYPE13
*MAT_SEATBELT	/PROP/SPRING
*MAT_MOONEY_RIVLIN_RUBBER	/M42_OGDEN /M69
*MAT_LOW_DENSITY_FOAM	/MAT/LAW38
*MAT_SOLID/TYPE45	/MAT/LAW70
*MAT_FABRIC/MAT_034	/MAT/FABRI
MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM_TITLE	/SPRING,/PROP/SPR_BEAM
MAT_VISCOELASTIC_OPTION/MAT_006	MAT/LAW42
MAT_SPRING_GENERAL_NONLINEAR/MAT_S06	/PROP/SPR_BEAM

LS-DYNA	Radioss
*MAT_DAMPER_VISCOUS	/SPRING,/PROP/SPR_BEAM

Properties

Table 320: Supported Property Mappings

LS-DYNA	Radioss
*SECTION_BEAM	P3_BEAM
*SECTION_DISCRETE	P4_SPRING
*SECTION_SHELL	P1_SHELL
*SECTION_SOLID	P14_SOLID

Rigid Walls

Table 321: Supported Rigid Wall Mappings

LS-DYNA	Radioss
*RIGIDWALL_GEOMETRIC_FLAT_OPTIONS	RWALL
*RIGIDWALL_PLANAR_OPTIONS	RWALL

Sets

Table 322: Supported Set Mappings

LS-DYNA	Radioss
*SET_NODE_ADD	/GRNOD/NODE

LS-DYNA	Radioss
*SET_NODE_ADD_ADVANCED	/GRNOD/GRNOD

Systems

Table 323: Supported System Mappings

LS-DYNA	Radioss
*DEFINE_COORDINATE_NODES	SKEW/MOVE/
*DEFINE_COORDINATE_SYSTEM	SKEW/FIXED

Time History Definitions

Table 324: Supported Time History Definition Mappings

LS-DYNA	Radioss
*DATABASE_HISTORY_BEAM	TH/BEAM
*DATABASE_HISTORY_NODE	TH/NODE
*DATABASE_HISTORY_SHELL	TH/SHEL
*DATABASE_HISTORY_SOLID	TH/BRIC
*DATABASE_HISTORY_TSHELL	TH/THSH
*DATABASE_HISTORY_NODE_SET_LOCAL	/TH/NODE
*DATABASE_EXTENT_BINARY	/ANIM/SHELL/TENS/STRESS/UPPER
*DATABASE_BINARY_D3PLOT	/TFILE/1
*DATABASE OPTION	/TFILE/1

LS-DYNA	Radioss
*DATABASE_CROSS_SECTION_PLANE	/SECT

LS-DYNA to OptiStruct Conversion Mapping

Elements

Table 325: Supported Element Mappings

LS-DYNA	OptiStruct
*ELEMENT_MASS	CONM2
*ELEMENT_BEAM	CBAR
*ELEMENT_SHELL	CTRIA3,CQUAD4, CQUAD8, CTRIA6
*ELEMENT_SOLID	CTETRA4/CHEXA8/CPENTA6
*CONSTRAINED_SPOTWELD	RBAR
*ELEMENT_PLOTEL	PLOTEL
*CONSTRAINED_INTERPOLATION	RBE3
*CONSTRAINED_NODE_SET	RBE2
*ELEMENT_DISCRETE	CELAS1

Properties

Table 326: Supported Property Mappings

LS-DYNA	OptiStruct
*SECTION_SOLID	PSOLID

LS-DYNA	OptiStruct
*SECTION_SHELL	PSHELL

Materials

All the LS-DYNA materials are mapped MAT1 in OptiStruct.

Boundary Conditions

Table 327: Supported Boundary Condition Mappings

LS-DYNA	OptiStruct
*BOUNDARY_SPC	SPC
*LOAD_NODE_POINT	FORCE, MOMENT
*LOAD_SEGMENT	PLOAD4
*INITIAL_TEMP	TEMP

Coordinate Systems

Table 328: Supported Coordinate System Mappings

LS-DYNA	OptiStruct
*DEFINE_COORDINATE_NODE	CORD1R
*DEFINE_COORDINATE_SYSTEM	CORD2R

Nastran Conversion

Nastran to Abaqus Conversion Mapping

Elements

HyperMesh elements have two basic attributes – configuration (or config) and type. The "config" defines the basic geometrical shape of an element. For example, tria3 configuration is a 3 node triangular element and hexa8 is an 8-node hexahedral element. The "type" defines the solver-specific element type of a particular configuration. For example, the 4-node quadrilateral (quad4) element in Abaqus can be any of the following types: S4, S4R, M3D4, R3D4, and so on. The Elem Types panel shows all supported element configs and their types for a user profile.

For a specific configuration, you can map any supported Nastran element type to any supported Abaqus element type. Several 2-noded element configurations such as spring, rigid, bar2, rid, and so on are supported. Because all of them are 2-noded elements, conversion across these configurations is also allowed for some element types. For example, CBUSH is of "spring" config in the Nastran user profile and CONN3D2 is of 'rod" config in the Abaqus user profile. It is possible to map a CBUSH to CONN3D2 even though their configs are different. The element mapping scheme must be under the *ElemTypeConversion block in the `ConfigurationFile.txt` file. You need to provide both config and type information to specify the element mapping scheme.

Table 329: Supported Element Mappings

Nastran	Abaqus
tria3, CTRIA3	tria3, S3
tria3, CTRIAR	tria3, S3R
quad4, CQUAD4	quad4, S4
quad4, CQUADR	quad4, S4R
quad4, CSHEAR	quad4, M3D4
tetra4, CTETRA	tetra4, C3D4
penta6, CPENTA	penta6, C3D6
hex8, CHEXA	hex8, C3D8
tria6, CTRIA6	tria6, STRI65
quad8, CQUAD8	quad8, S8R
tetra10, CTETRA	tetra10, C3D10

Nastran	Abaqus
penta15, CPENTA	penta15, C3D15
hex20, CHEXA	hex20, C3D20
mass, CONM2	mass, MASS
mass, CELAS1	mass, SPRING1
mass, CELAS2	mass, SPRING1
rigid, RBE2	rigid, COUP_KIN
rigidlink, RBE2	rigidlink, COUP_KIN
rbe3, RBE3	rbe3, COUP_DIS
spring, CELAS1	spring, SPRING2
spring, CELAS2	spring, SPRING2
spring, CDAMP1	spring, DASHPOT1
spring, CDAMP2	spring, DASHPOT2
spring, CBUSH	rod, CONN3D2
bar2, CBEAM	bar2, B31
bar2, CBAR	bar2, B31
rod, CROD	rod, T3D2
rod, CONROD	rod, T3D2
gap, CGAP	gap, GAPUNI
weld, RBAR	rigid, KINCOUP

Note:

- The CELAS1 or CELAS2 elements in Nastran have both spring stiffness and damping attributes. If both spring and damping values are present and the mapping scheme is CELAS1 to SPRING1, the conversion tool will automatically create an extra DASHPOT element.
- Similarly, the CONM2 elements in Nastran have both translational and rotational mass values. If both translational and rotational values are present and the mapping scheme is CONM2 to MASS, the conversion tool will automatically create an extra ROTARY1 element.
- *COUPLING/*KINEMATIC constraints with element based surfaces, currently mapped to groups in HM, are converted into rigid elements. Currently conversion of *COUPLING/*DISTRIBUTING with element based surfaces is not supported.
- The configuration file can be updated such that config can be changed to DCOUP3D for Rbe3 with COUP_DIS as the default conversion.
- In the Nastran to Abaqus Conversion browser, the conversion option Define Direction Cosines in Property after conversion for PBar, PBeam, PBarL, and PBeamL is selected by default. If your provided direction cosines for beams in Nastran then this option will transfer this information to *Beam Section in Abaqus.

Contacts

BSURF is converted to *SURFACE ELEMENT

BCBODY in BCTABLE is mapped to CONTACT PAIR

Friction in BCTABLE is mapped to *SURFACE INTERACTION

Sectional Properties

Some of the properties in one solver can be converted to two different Abaqus sections in the other solver. For a Nastran to Abaqus conversion, for example, PSHELL can be converted to *SHELL SECTION or *SHELL GENERAL SECTION. In the mapping scheme, you must select one of them. The property mapping scheme must be under the *PropertyConversion block in the `ConfigurationFile.txt` file.

Abaqus beam section axes are defined at the element level in Nastran. They are in the sectional property level in Abaqus unless the beam axis is defined by a third node in element connectivity. This means that several elements with different beam axis directions can point to the same PBEAM, PBEAML, PBAR or PBARL property in Nastran. But in Abaqus, all elements under a *BEAM SECTION or *BEAM GENERAL SECTION property have one beam axis orientation. If a third node is used to define the beam axis, even Abaqus beams with a different axis can belong to a single *BEAM SECTION property. Use the conversion tool to select an extra (1 or 0) argument to define the beam axis conversion mechanism.

If the argument is 0 (or not defined), the conversion tool will take the beam axis direction of the first element corresponding to a PBEAM, PBEAML, PBAR or PBARL property and map that to the corresponding *BEAM SECTION or *BEAM GENERAL SECTION card. The beam axis vectors of other elements with the same property will be ignored.

If the argument is 1, the conversion tool will create a third node for each element to define the equivalent beam axis vector. As a result, the axis direction for each element will be maintained after the conversion. Because this option updates each element, the conversion process might take a considerable amount of time for models with a large number of beams.

Therefore for CELAS1 two options can be set in the `ConfigurationFile.txt` (1 or 0). If the option is 1, one property per element will be created (default). If the flag is set to 0, one property per PELAS card will be created. In this case, the settings of the first element found on this property will be translated. From CELAS2 elements you always create a *SPRING and *DASHPOT or *CONNECTOR SECTION property per element.

Table 330: Supported Section Property Mappings

Nastran	Abaqus	Beam axis option
PSOLID	*SOLID SECTION	
PSHELL	*SHELL SECTION or *SHELL GENERAL SECTION	
PBEAM	*BEAM GENERAL SECTION	1 or 0
PBEAML	*BEAM SECTION	1 or 0
PBAR	*BEAM GENERAL SECTION	1 or 0
PBARL	*BEAM SECTION	1 or 0
PROD	*SOLID SECTION	
PBUSH	*CONNECTOR SECTION	
PBUSHT(KN)	*CONNECTOR PLASTICITY	
PELAS	(*SPRING + *DASHPOT) or *CONNECTOR SECTION	1 or 0
PDAMP	*DASHPOT or *CONNECTOR SECTION	
PGAP	*GAP PROPERTY	
CELAS2	(*SPRING + *DASHPOT) or *CONNECTOR SECTION	
CDAMP2	*DASHPOT or *CONNECTOR SECTION	
CONM2	(*MASS +*ROTARY INERTIA)	

Note:

- CELAS2, CDAMP2 and CONM2 are elements in Nastran, but they are sectional properties in Abaqus. Therefore, the mapping for them must also be defined under *PropertyConversion.
- The PELAS or CELAS2 in Nastran have both spring stiffness and damping attributes. If both spring and damping values are present and they are mapped to *SPRING, the conversion tool will automatically create an extra *DASHPOT property. The elements will both be kept in the same component and the property will be directly assigned to the *SPRING or *DASHPOT element.
- Similarly, the CONM2 in Nastran has both translational and rotational mass values. If both translational and rotational values are present and it is mapped to *MASS, the conversion tool will automatically create an extra *ROTARY INERTIA component.
- The property conversion scheme and corresponding element conversion scheme must be consistent. For example, if you define PBUSH to *CONNECTOR SECTION at the property mapping scheme, the corresponding element CBUSH must map to CONN3D2 in the element mapping scheme.
- A Nastran model with PBUSHT KN referencing TABLED1 is converted to CONNECTORSECTION & CONNECTOR_BEHAVIOR ,and TABLED1 is mapped to CONNECTOR PLASTICITY.

Materials

The material mapping scheme must be defined under *PropertyConversion block in the `ConfigurationFile.txt` file.

Table 331: Supported Material Mappings

Nastran	Abaqus	
MAT1	*MATERIAL	*ELASTIC, TYPE=ISO; *EXPANSION, TYPE=ISO; and *DENSITY (G is used only for *BEAM GENERAL SECTION)
MAT2	*MATERIAL	When used alone in a PSHELL, MAT2 is translated to *ELASTIC, TYPE=LAMINA or *ELASTIC, TYPE=ANISOTROPIC
MAT8	*MATERIAL	ELASTIC, TYPE=LAMINA; *EXPANSION, TYPE=ORTHO; and *DENSITY
MAT9	*MATERIAL	*ELASTIC, TYPE=ANISOTROPIC unless the data are found to be orthotropic, in which case

Nastran	Abaqus	
		the data are analyzed to create *ELASTIC, TYPE=ENGINEERING CONSTANTS. Also *DENSITY; and *EXPANSION, TYPE=ANISO or ORTHO.
MAT9ORT	*MATERIAL	*ELASTIC,TYPE=ENGINEERING CONSTANTS

Note: If a PBEAM or PBAR is mapped to a *BEAM GENERAL SECTION, the material properties defined in the corresponding Nastran material are mapped to the *BEAM GENERAL SECTION card. No *MATERIAL is created in this case.

Nastran to ANSYS Conversion Mapping

Materials

Table 332: Supported Material Mappings

Nastran	ANSYS	Convert Parameters
MAT1	MPTEMP/MPDATA	E, G, NU, RHO, A, GE, TREF
MATT1	MPTEMP/MPDATA	T(E), T(G), T(NU), T(RHO), T(A), T(GE)
MAT2	MATERIAL	density
MAT4	MATERIAL	density

Elements

Table 333: Supported Element Mappings

Nastran	ANSYS	HM Configuration
CONM2	MASS21	Mass
CELAS1	COMBI14	Spring
CBAR, CBEAM	BEAM44	Bar2

Nastran	ANSYS	HM Configuration
CROD	LINK8	Rod
CTRIA3, CTRIAR	SHELL63	Tria3
CQUAD4, CQUADR, CSHEAR	SHELL63	Quad4
CTRIA6	SHELL93	Tria6
CQUAD8	SHELL93	Quad8
CTETRA	SOLID45	Tetra4
CTETRA	SOLID95	Tetra10
CHEXA	SOLID45	Hex8
CHEXA	SOLID95	Hex20
CPENTA	SOLID45	Penta6
CPENTA	SOLID95	Penta15
CGAP	CONTACT52	Gap
CBUSH	COMBI14	Spring

Properties

Table 334: Supported Property Mappings

Nastran	ANSYS	Convert Parameters
PBAR, PBARL, PBEAM, PBEAML	BEAM44p	none, only creates property card with BEAM44p
PBUSH	COMBI14p	k1 to k6, GE1 to GE6
PELAS	COMBI14p	k, GE
PGAP	CONTACT52p	U0, KA, KT, MU1
PROD	LINK8p	none
PSHELL, PSHEAR (tria3, quad4)	SHELL63p	Shell Thickness
PSHELL, PSHEAR (tria6, quad8)	SHELL93p	Shell Thickness
PSOLID (tetra4, penta6, hex8)	SOLID45p	none

Nastran	ANSYS	Convert Parameters
PSOLID (tetra10, penta15, hex20)	SOLID95p	none

ETType of ANSYS

If the model contains Nastran elements which were mapped to MASS21, LINK8, COMBI14, BEAM44, SHELL63, SHELL93, SOLID45 and SOLID95 elements, respective ETTYPES will be created and assigned to the component. Key options of ETTYPE will not be updated.

Components

If the component collector contains different type of configuration of elements, new components will be created for the respective configurations and those elements will be moved into the new component. ETTYPE, material and realset IDs will be assigned to the new component.

Loads and Boundary Conditions

Loads and boundary conditions will not be converted by using this conversion tool.

Nastran to LS-DYNA Conversion Mapping

Element

Table 335: Supported Element Mappings

Nastran	LS-DYNA
CONM2	*ELEMENT_MASS or *ELEMENT_INERTA
CWELD	Meshless weld (feabsorb, fe realize using HEXA)
CDAMP1/CDAMP2 (0 length)	*ELEMENT_DISCRETE (card edit for ground option check box)
PLOTEL	*ELEMENT_PLOTEL
RBAR/RBE2/RJOINT	*CONSTRAINED_NODAL_RIGID_BODY
RBE3	*CONSTRAINED_INTERPOLATION
CBAR/CBEAM/CROD/CTUBE/CBUSH CELAS1/CELAS2	*ELEMENT_BEAM

Nastran	LS-DYNA
CTRIA3/CTRIAR CQUAD4, CQUADR, CSHEAR CQUAD8, CTRIA6 (Change to 1st order then convert)	*ELEMENT_SHELL
CTETRA4/CHEXA8/CPENTA6 CTETRA10/CHEXA20/CPENTA15 (Change to 1st order then convert)	*ELEMENT_SOLID

Properties

Table 336: Supported Property Mappings

Nastran	LS-DYNA
PBAR/PBEAM/PROD (Only single section PBEAM is supported)	*SECTION_BEAM
PBARL/PBEAML/PRODL (ROD, TUBE, BAR)	*SECTION_BEAM
PBUSH	*SECTION_BEAM + *MAT_GENERAL_SPRING_DISCRETE_BEAM
PELAS	*SECTION_BEAM + *MAT_1DOF_GENERALIZED_SPRING
PDAMP	*SECTION_DESCRETE
PSHELL/PSHEAR	*SECTION_SHELL
PSOLID	*SECTION_SOLID
PCOMP PCOMPG	*PART_COMPOSITE +*ELEMENT_SHELL with angle option; all materials referred

Nastran	LS-DYNA
	in PCOMP, PCOMPG converted to *MAT_ORTHOTROPIC_THERMAL

Materials

Table 337: Supported Material Mappings

Nastran	LS-DYNA
MAT1	*MAT_ELASTIC
MAT2	*MAT_PIECEWISE_LINEAR _PLASTICITY
MAT4	*MAT_PIECEWISE_LINEAR _PLASTICITY
MAT8	*MAT_ORTHOTROPIC_ELASTIC
MAT9	*MAT_ANISOTROPIC_ELASTIC

Coordinate Systems

Table 338: Supported Coordinate System Mappings

Nastran	LS-DYNA
CORD1R	*DEFINE_COORDINATE_NODE
CORD2R	*DEFINE_COORDINATE_SYSTEM

Loads

Table 339: Supported Load Mappings

Nastran	LS-DYNA
SPC	*BOUNDARY_SPC
FORCE	*LOAD_NODE_POINT
MOMENT	*LOAD_NODE_POINT
PLOAD4	*LOAD_SEGMENT

Nastran	LS-DYNA
TEMP	*INITIAL_TEMP

Nastran to Radioss Conversion Mapping

Elements

Table 340: Supported Element Mappings

Nastran	Radioss
CONM2	/ADMAS
CWELD	Mesh less weld (feabsord, fe realize using HEXA)
CELAS1/CELAS2 CDAMP1/CDAMP2 (0 length)	/SPRING
PLOTEL	Plot element
RBAR	/RIVET
RBE2/RBE3	/RBODY
RJOINT	/RLINK
CBUSH (When using vector)	/ SPRING3N
CBUSH1D	/SPRING2N
CBAR/CBEAM/CBEND	BEAM
CROD	TRUSS
CONROD	TRUSS and /PROP/TRUSS
CTRIA3/CTRIAR CTRIA6 (Change to 1st order before convert)	/SH3N

Nastran	Radioss
CQUAD4, CQUADR, CSHEAR CQUAD8 (Change to 1st order then convert)	/SHELL
CTETRA4 CTETRA10 (Change to 1st order then convert)	/TETRA4
CHEXA8/ CPENTA6 CHEXA20/ CPENTA15 (Change to 1st order then convert)	/BRICK

Properties

Table 341: Supported Property Mappings

Nastran	Radioss
PBAR/PBEAM (Only single section PBEAM is supported)	/PROP/BEAM
PBUSH	/PROP/SPR_GENE
PBUSH1D/PDAMP/PELAS	/PROP/SPRING
PROD	/PROP/TRUSS
PSHELL/PSHEAR	/PROP/SHELL
PSOLID	/PROP/SOLID

Materials

Table 342: Supported Material Mappings

Nastran	Radioss
MAT1	/MAT/ELASTIC
MAT8	/MAT/LAW19

OptiStruct Conversion

OptiStruct to Abaqus Conversion Mapping

The OptiStruct to Abaqus conversion uses an open conversion scheme; you can specify different mappings in the configuration file.

Care has to be taken so that the element and property mappings are consistent. Altair provides a valid mapping scheme in the `ConfigurationFile.txt`. This document explains the scope and limitations of the mapping scheme.

Elements

HyperMesh elements have two basic attributes – configuration (or config) and type. The "config" defines the basic geometrical shape of an element. For example, tria3 configuration is a 3 node triangular element and hexa8 is an 8-node hexahedral element. The "type" defines the solver specific element type of a particular configuration. For example, the 4-node quadrilateral (quad4) element in Abaqus can be any of the following types: S4, S4R, M3D4, R3D4, and so on. The Element Types panel shows all supported element configurations and their types for a user profile.

For a specific configuration, you can map any supported Nastran element type to any supported Abaqus element type. For example, for an OptiStruct to Abaqus conversion, several 2-noded element configurations such as spring, rigid, bar2, rid, and so on are supported. Because all of them are 2-noded elements, conversion across these configurations is also allowed for some element types. For example, CBUSH is of "spring" configuration in the HyperMesh user profile and CONN3D2 is of ‘rod" configuration in the Abaqus user profile. It is possible to map a CBUSH to CONN3D2 even though their configurations are different. The element mapping scheme must be under the *ElemTypeConversion block in the `ConfigurationFile.txt` file. You need to provide both configuration and type information to specify the element mapping scheme.

Table 343: Supported Element Mappings

OptiStruct	Abaqus
tria3, CTRIA3	tria3, S3
tria3, CTRIAR	tria3, S3R
quad4, CQUAD4	quad4, S4
quad4, CQUADR	quad4, S4R
quad4, CSHEAR	quad4, M3D4
tetra4, CTETRA	tetra4, C3D4
penta6, CPENTA	penta6, C3D6

OptiStruct	Abaqus
hex8, CHEXA	hex8, C3D8
tria6, CTRIA6	tria6, STRI65
quad8, CQUAD8	quad8, S8R
tetra10, CTETRA	tetra10, C3D10
penta15, CPENTA	penta15, C3D15
hex20, CHEXA	hex20, C3D20
mass, CONM2	mass, MASS
mass, CELAS1	mass, SPRING1
mass, CELAS2	mass, SPRING1
rigid, RBE2	rigid, COUP_KIN
rigidlink, RBE2	rigidlink, COUP_KIN
rbe3, RBE3	rbe3, DCOUP3D
spring, CELAS1	spring, SPRING2
spring, CELAS2	spring, SPRING2
spring, CDAMP1	spring, DASHPOT2
spring, CDAMP2	spring, DASHPOT2
spring, CBUSH	rod, CONN3D2
bar2, CBEAM	bar2, B31
bar2, CBAR	bar2, B31
rod, CROD	rod, T3D2
rod, CONROD	rod, T3D2
gap, CGAP	gap, GAPUNI
weld, RBAR	rigid, KINCOUP

Note:

- The CELAS1 or CELAS2 elements in OptiStruct have both spring stiffness and damping attributes. If both spring and damping values are present and the mapping scheme is CELAS1 to SPRING1, the conversion tool will automatically create an extra DASHPOT element.
- Similarly, the CONM2 elements in OptiStruct have both translational and rotational mass values. If both translational and rotational values are present and the mapping scheme is CONM2 to MASS, the conversion tool will automatically create an extra ROTARY1 element.
- *COUPLING/*KINEMATIC constraints with element based surfaces (currently mapped to groups in HyperMesh) are converted into rigid elements. Currently conversion of *COUPLING/*DISTRIBUTING with element based surfaces is not supported.

Sectional Properties

Some of the properties in one solver can be converted to two different Abaqus sections in the other solver. For a OptiStruct to Abaqus conversion, for example, PSHELL can be converted to *SHELL SECTION or *SHELL GENERAL SECTION. In the mapping scheme, you must select one of them. The property mapping scheme must be under the *PropertyConversion block in the `ConfigurationFile.txt` file.

Abaqus beam section axes are defined at element level in OptiStruct. They are in the sectional property level in Abaqus unless the beam axis is defined by a third node in element connectivity. This means that several elements with different beam axis direction can point to the same PBEAM, PBEAML, PBAR or PBARL property in OptiStruct. But in Abaqus, all elements under a *BEAM SECTION or *BEAM GENERAL SECTION property have one beam axis orientation. If a third node is used to define the beam axis, even Abaqus beams with a different axis can belong to a single *BEAM SECTION property. Use the conversion tool to select an extra (1 or 0) argument to define the beam axis conversion mechanism.

If the argument is 0 (or not defined), the conversion tool will take the beam axis direction of the first element corresponding to a PBEAM, PBEAML, PBAR or PBARL property and map that to the corresponding *BEAM SECTION or *BEAM GENERAL SECTION card. The beam axis vectors of other elements with the same property will be ignored.

If the argument is 1, the conversion tool will create a third node for each element to define the equivalent beam axis vector. As a result, the axis direction for each element will be maintained after the conversion. Because this option updates each element, the conversion process might take a considerable amount of time for models with a large number of beams.

The system for CELAS1 or CELAS2 elements is sitting on the grid nodes. Thus, every element can have a different system. Ideally, on conversion one *SPRING (and *DASHPOT) or *CONNECTOR SECTION per element needs to be created. For large models this can be time-consuming.

Therefore for CELAS1 two options can be set in the `ConfigurationFile.txt` (1 or 0). If the option is 1, one property per element will be created (default). If the flag is set to 0, one property per PELAS card will be created. In this case, the settings of the first element found on this property will be translated. From CELAS2 elements you always create a *SPRING and *DASHPOT or *CONNECTOR SECTION property per element.

Composite sections PCOMP and PCOMPG can be converted to Abaqus as well. From `ConfigurationFile.txt` a user can select to convert to SHELL SECTION or SHELL GENERAL SECTION properties. Besides individual layers, the conversion takes care of system assignments, offsets, SYM, BEND and other similar parameters. In PCOMPG a global ply id (GPLYID) number is honored in the ply name in Abaqus user profile after conversion.

Table 344: Supported Section Property Mappings

OptiStruct	**Abaqus**	**Beam axis/property option**
PSOLID	*SOLID SECTION	
PSHELL	*SHELL SECTION or *SHELL GENERAL SECTION	
PCOMP(G)	*SHELL SECTION or *SHELL GENERAL SECTION (COMPOSITE)	
PBEAM	*BEAM GENERAL SECTION	1 or 0
PBEAML	*BEAM SECTION	1 or 0
PBAR	*BEAM GENERAL SECTION	1 or 0
PBARL	*BEAM SECTION	1 or 0
PROD	*SOLID SECTION	
PBUSH	*CONNECTOR SECTION	
PELAS	(*SPRING + *DASHPOT) or *CONNECTOR SECTION	1 or 0
PDAMP	*DASHPOT or *CONNECTOR SECTION	
CELAS2	(*SPRING + *DASHPOT) or *CONNECTOR SECTION	
CDAMP2	*DASHPOT or *CONNECTOR SECTION	
CONM2	(*MASS + *ROTARY INERTIA)	

Note:

- CELAS2, CDAMP2 and CONM2 are elements in OptiStruct but they are sectional properties in Abaqus. Therefore, the mapping for them must also be defined under *PropertyConversion
- The PELAS or CELAS2 in OptiStruct both spring stiffness and damping attributes. If both spring and damping values are present and they are mapped to *SPRING, the conversion tool will automatically create an extra *DASHPOT property. The elements will both be kept in the same component and the property will be directly assigned to the *SPRING or *DASHPOT element.
- Similarly, the CONM2 in OptiStruct has both translational and rotational mass values. If both translational and rotational values are present and it is mapped to *MASS, the conversion tool will automatically create an extra *ROTARY INERTIA component.
- The property conversion scheme and corresponding element conversion scheme must be consistent. For example, if you define PBUSH to *CONNECTOR SECTION at the property mapping scheme, the corresponding element CBUSH must map to CONN3D2 in the element mapping scheme.

Materials

The material mapping scheme must be defined under *PropertyConversion block in the `ConfigurationFile.txt` file.

Table 345: Supported Material Mappings

OptiStruct	Abaqus	Notes
MAT1	*MATERIAL	*ELASTIC, TYPE=ISO; *EXPANSION, TYPE=ISO; and *DENSITY (G is used only for *BEAM GENERAL SECTION)
MAT2	*MATERIAL	When used alone in a PSHELL, MAT2 is translated to *ELASTIC, TYPE=LAMINA or *ELASTIC, TYPE=ANISOTROPIC
MAT8	*MATERIAL	ELASTIC, TYPE=LAMINA; *EXPANSION, TYPE=ORTHO; and *DENSITY
MAT9	*MATERIAL	*ELASTIC, TYPE=ANISOTROPIC unless the data are found to be orthotropic, in which case the data are analyzed to create *ELASTIC, TYPE=ENGINEERING CONSTANTS. Also *DENSITY;

OptiStruct	Abaqus	Notes
		and *EXPANSION, TYPE=ANISO or ORTHO.

Note: If a PBEAM or PBAR is mapped to a *BEAM GENERAL SECTION, the material properties defined in the corresponding OptiStruct material are mapped to the *BEAM GENERAL SECTION card. No *MATERIAL is created in this case.

Loads

HyperMesh loads have two basic attributes – configuration (or config) and type. The supported load "config" are: force, moment, constraint, pressure, temperature, flux, velocity, acceleration and equation. The load "type" defines the solver specific type of a particular configuration. For example, pressure load can be any of the following Abaqus types: DLOAD, FILM, DFLUX, and so on. The Load Types panel shows all supported load configurations and their types for a user profile.

The converter also converts distributed surfaces loads (*DLSOAD) applied on faces of shell or solid elements into pressure loads (PLOAD4).

For a specific configuration, you can map any supported OptiStruct load type to any supported Abaqus load type. The conversion tool does not support conversion across load configurations. The load mapping scheme is valid for either direction and must be under the *BCsTypeConversion block in the `ConfigurationFile.txt` file. You need to provide both configuration and type information to specify the mapping scheme.

Table 346: Supported Load Mappings

OptiStruct	Abaqus
force, FORCE	force, CLOAD
moment, MOMENT	moment, CLOAD
const, SPC	const, BOUNDARY
const, SPCD	const, VELOCITY
const, SUPORT	const, BOUNDARY
pressure, PLOAD	pressure, DLOAD
pressure, PLOAD2	pressure, DLOAD
pressure, PLOAD4	pressure, DLOAD
temp, TEMP	temp, TEMPERATURE

OptiStruct	Abaqus
equation, MPC	equation, EQUATION

In addition to the above load types, the conversion tool also converts OptiStruct Dload, with corresponding Rload1, Rload2, DAREA, TABLED1, TABLED2, TABLED3, to Abaqus *BOUNDARY or *CLOAD (with corresponding *AMPLITUDE curve). No mapping scheme needs to be specified for this conversion; the conversion is done automatically if present in the model.

Load Steps and Analysis Type

The conversion tool maps between OptiStruct subcases and Abaqus steps. It does not convert the solution type from/to any Abaqus analysis type. You must define it manually using the Abaqus Step Manager or the OptiStruct Load Step Browser.

Systems and Masses

The conversion tool converts OptiStruct system types into the corresponding Abaqus system (*SYSTEM, *TRANSFORM or *ORIENTATION). It also converts the NSM into *NONSTRUCTURAL MASS and assigns them to the relevant properties.

Table 347: Supported System and Mass Mappings

OptiStruct	Abaqus
NSM	*NONSTRUCTURAL MASS
NSM1	
NSML	
NSML1	
NSMADD	
GRID	*NODE AND *SYSTEM
CORD1R	*SYSTEM for nodes *TRANSFORM if referred to on GRID *ORIENTATION for elements
CORD1C	
CORD1S	
CORD2R	
CORD2C	

OptiStruct	Abaqus
CORD2S	

WTMASS

If the `WTMASS` parameter is defined in the OptiStruct model, it is used to modify density, mass, and inertia values during conversion.

OptiStruct to ANSYS Conversion Mapping

Conversion Notes

1. If CBEAM is associated with PBEAM/PBEAML, then the properties within those cards will be converted to ANSYS SECTION with type BEAM. OptiStruct beam cross sections of types: BAR, CHAN, HAT, I, L, T, ROD, and TUBE are converted to ANSYS equivalent beam sections.

   OptiStruct Z and BOX sections are converted to ANSYS ASEC beam sections. CHAN, Z, and BOX sections are only converted when PBEAML has a HyperBeam section attached. PBEAM is converted to ANSYS ASEC when HyperBeam sections are not attached to a property card. OptiStruct NSM is converted to ANSYS ADDMAS.

   Limitation: In the current version of the conversion tool, section offsets are not considered.
2. PSHELL and PSHEAR properties are converted to ANSYS SECTIONS with type SHELL. Shell thickness and material from PSHELL/PSHEAR are mapped to TK and Material of SHELL section. After conversion, section offset is always set to MID.
3. Conversion of contact surfaces: If contact surfaces in OptiStruct are defined with node sets, then the surfaces will not be converted.

Materials

Table 348: Supported Material Mappings

OptiStruct	ANSYS	Convert Parameters
MAT1	MPTEMP/MPDATA	E, G, NU, RHO, A, GE, TREF
MATT1	MPTEMP/MPDATA	T(E), T(G), T(NU), T(RHO), T(A), T(GE)
MAT2	MATERIAL	density

OptiStruct	ANSYS	Convert Parameters
MAT4	MATERIAL	density

Elements

Table 349: Supported Element Mappings

OptiStruct	ANSYS	HM Configuration
CONM2	MASS21	Mass
CELAS1	COMBI14	Spring
CBEAM	BEAM188	Bar2
CBAR	BEAM44	Bar2
CROD	LINK8	Rod
CTRIA3, CTRIAR	SHELL181	Tria3
CQUAD4, CQUADR, CSHEAR	SHELL181	Quad4
CTRIA6	SHELL281	Tria6
CQUAD8	SHELL281	Quad8
CTETRA	SOLID45	Tetra4
CTETRA	SOLID95	Tetra10
CHEXA	SOLID45	Hex8
CHEXA	SOLID95	Hex20
CPENTA	SOLID45	Penta6
CPENTA	SOLID95	Penta15
CGAP	CONTACT52	Gap

OptiStruct	ANSYS	HM Configuration
CBUSH	COMBI14	Spring

Properties

Table 350: Supported Property Mappings

OptiStruct	ANSYS	Convert Parameters
PBEAM, PBEAML	SECTION	.
PBAR, PBARL	BEAM44p	None. Only creates a property card.
PBUSH	COMBI14p	k1 to k6, GE1 to GE6
PELAS	COMBI14p	k, GE
PGAP	CONTACT52p	U0, KA, KT, MU1
PROD	LINK8p	None
PSHELL, PSHEAR (tria3, quad4)	SECTION	.
PSHELL, PSHEAR (tria6, quad8)	SECTION	
PSOLID (tetra4, penta6, hex8)	SOLID45p	None

OptiStruct	ANSYS	Convert Parameters
PSOLID (tetra10, penta15, hex20)	SOLID95p	None

ETType of ANSYS

If the model contains OptiStruct elements which were mapped to MASS21, LINK8, COMBI14, BEAM44, SHELL181, SHELL281, SOLID45 and SOLID95 elements, respective ETTYPES will be created and assigned to the component. Key options of ETTYPE are not updated.

Components

If the component collector contains different type of configuration of elements, new components will be created for the respective configurations and those elements will be moved into the new component. ETTYPE, material, and realset section IDs will be assigned to the new component.

Loads and Boundary Conditions

Loads and boundary conditions will not be converted by using this conversion tool.

Contact Surfaces

Conversion of contact surfaces: If in OptiStruct a contact pair's master and slave are both contact surfaces (CSURF), then the contact surfaces and the pair will be retained after conversion. However, in ANSYS, you must add the details for the surfaces, such as Et type and property card. If the friction coefficient MU1, available in the OptiStruct property card, is attached to the contact pair, then a new material card with a MPDATA card image will be created, and the value will be mapped to MU in the ANSYS format after conversion.

OptiStruct to LS-DYNA Conversion Mapping

Elements

Table 351: Supported Element Mappings

OptiStruct	LS-DYNA
CONM2	*ELEMENT_MASS
CBAR	*ELEMENT_BEAM
CTRIA3,CQUAD4, CQUAD8, CTRIA6	*ELEMENT_SHELL
CTETRA4/CHEXA8/CPENTA6	*ELEMENT_SOLID
RBAR	*CONSTRAINED_SPOTWELD

OptiStruct	LS-DYNA
PLOTEL	*ELEMENT_PLOTEL
RBE3	*CONSTRAINED_INTERPOLATION
RBE2	*CONSTRAINED_NODE_SET
CELAS1	
	*ELEMENT_DISCRETE

Properties

Table 352: Supported Property Mappings

OptiStruct	LS-DYNA
PSOLID	*SECTION_SOLID
PSHELL	*SECTION_SHELL

Materials

All the LS-DYNA materials are mapped MAT1 in OptiStruct.

Boundary Conditions

Table 353: Supported Boundary Condition Mappings

OptiStruct	LS-DYNA
SPC	*BOUNDARY_SPC
FORCE, MOMENT	*LOAD_NODE_POINT
PLOAD4	*LOAD_SEGMENT
TEMP	*INITIAL_TEMP

Coordinate Systems

Table 354: Supported Coordinate System Mappings

OptiStruct	LS-DYNA
CORD1R	*DEFINE_COORDINATE_NODE

OptiStruct	LS-DYNA
CORD2R	*DEFINE_COORDINATE_ SYSTEM

OptiStruct to Radioss Conversion Mapping

Elements

Table 355: Supported Element Mappings

OptiStruct	Radioss
CONM2	`/ADMAS`
CWELD	Mesh less weld (feabsord, fe realize using HEXA)
CELAS1/CELAS2 CDAMP1/CDAMP2 (0 length)	`/SPRING`
PLOTEL	Plot element
RBAR	`/RIVET`
RBE2/RBE3	`/RBODY`
RJOINT	`/RLINK`
CBUSH (When using vector)	`/SPRING3N`
CBUSH1D	`/SPRING2N`
CBAR/CBEAM/CBEND	`/BEAM`
CROD	`/TRUSS`
CONROD	`/TRUSS` and `/PROP/TRUSS`
CTRIA3/CTRIAR CTRIA6 (Change to 1st order before convert)	`/SH3N`

OptiStruct	Radioss
CQUAD4, CQUADR, CSHEAR CQUAD8 (Change to 1st order then convert)	/SHELL
CTETRA4 CTETRA10 (Change to 1st order then convert)	/TETRA4
CHEXA8/ CPENTA6 CHEXA20/ CPENTA15 (Change to 1st order then convert)	/BRICK

Properties

Table 356: Supported Property Mappings

OptiStruct	Radioss
PBAR/PBEAM (Only single section PBEAM is supported)	/PROP/BEAM
PBUSH	/PROP/SPR_GENE
PBUSH1D/PDAMP/PELAS	/PROP/SPRING
PROD	/PROP/TRUSS
PSHELL/PSHEAR	/PROP/SHELL
PSOLID	/PROP/SOLID

Materials

Table 357: Supported Material Mappings

OptiStruct	Radioss
MAT1	/MAT/ELASTIC
MAT8	/MAT/LAW19

PAM-CRASH 2G Conversion

PAM-CRASH 2G to Radioss Conversion Mapping

Elements

Table 358: Supported Element Mappings

PAM-CRASH 2G	Radioss
BAR	TRUSS
BEAM	BEAM
SPRING	SPRING
SPRBM	SPRING+PROP/SPR_BEAM (TYPE 13)
MEMBR	SH3N, SHELL
SHELL	SH3N, SHELL
SOLID	BRICK
TETRA44	TETRA44
MASS	ADMAS
PLINK	SPRING+INTER/TYPE2
KJOIN	SPRING+PROP/KJOINT(TYPE33)

Boundary Conditions

Table 359: Supported Boundary Condition Mappings

PAM-CRASH 2G	Radioss
INIVEL	INIVEL
BOUNC	BCS

PAM-CRASH 2G	Radioss
CONLO	CLOAD

Systems

Table 360: Supported System Mappings

PAM-CRASH 2G	Radioss
FRAME	FRAME

Materials

Table 361: Supported Material Mappings

PAM-CRASH 2G	Radioss
MAT_1D TYPE 201	M1_ELASTIC
MAT_1D TYPE 220	/PROP/SPRING
MAT_1D TYPE 223	/PROP/SPR_BEAM
MAT_2D TYPE 100	M0_VOID
MAT_2D TYPE 101	M1_ELASTIC
MAT_2D TYPE 102	M36_PLAS_TAB
MAT_2D TYPE 103	M36_PLAS_TAB, M44_, COWPER

PAM-CRASH 2G	Radioss
	M2_PLAS_JOHNS

Curves

Table 362: Supported Curve Mappings

PAM-CRASH 2G	Radioss
FUNCT	FUNCT

Control Cards

Table 363: Supported Control Card Mappings

PAM-CRASH 2G	Radioss
TITLE	TITLE
OCTRL/	
DSYOUPUT	/ANIM/DT

PAM-CRASH 2G	**Radioss**
THPOUTPUT	/TFILE

Components

Table 364: Supported Component Mappings

PAM-CRASH 2G	**Radioss**
PART	PART

Contact Interfaces

Table 365: Supported Contact Interface Mappings

PAM-CRASH 2G	**Radioss**
CNTAC36	INTER/TYPE7

Rigid Walls

Table 366: Supported Rigid Wall Mappings

PAM-CRASH 2G	**Radioss**
RWALL	RWALL

Time History

Table 367: Supported Time History Mappings

PAM-CRASH 2G	**Radioss**
THELE	TH/SHEL, SH3N, SPRING, BRIC
THNODE	TH/NODE

PAM-CRASH 2G	Radioss
THLOC	TH/FRAME/ACCELEROMETER

Section

Table 368: Supported Section Mappings

PAM-CRASH 2G	Radioss
SECFO	SECT

Group

Table 369: Supported Group Mappings

PAM-CRASH 2G	Radioss
GROUP	GRNOD

Airbags

Table 370: Supported Airbag Mappings

PAM-CRASH 2G	Radioss
BAGIN	MONVOL

Permas Conversion

Permas to OptiStruct Conversion Mapping

Elements

Table 371: Supported Element Mappings

Permas	OptiStruct
BECOC	CBEAM
MASS3	CONM2
MASS6	CONM2
TRIA3	CTRIA3
QUAD4	CQUAD4
SPRING1	CBUSH
SPRING3	CBUSH
SPRING6	CBUSH
DAMP1	CBUSH
DAMP3	CBUSH
DAMP6	CBUSH
TET4	CTETRA
TET10	CTETRA
PENTA6	CPENTA
HEXE8	CHEXA
HEXE20	CHEXA
$MPC RIGID	RBE2

Permas	OptiStruct
$MPC WLSCON, DOFTYPE=DISP	RBE3

Properties

Table 372: Supported Property Mappings

Permas	OptiStruct
BEAM	PBEAM
DAMPER	PBUSH
SHELL	PSHELL
SOLID	PSOLID
SPRING	PBUSH

Materials

Table 373: Supported Material Mappings

Permas	OptiStruct
MATERIAL	MAT1
+ $CONDUCTIVITY	+ MAT4
+ $HEATCAP	+ MAT4
+ $ELASTIC, INPUT=TABLE	+ MATT1
+ $THERMEXP, INPUT=TABLE	+ MATT1
+ $HARDENING	+ MATS1
+ $PLASTIC	+ MATS1

Radioss Conversion

Radioss to PAM-CRASH 2G Conversion Mapping

Elements

Table 374: Supported Element Mappings

Radioss	PAM-CRASH 2G
TRUSS	BAR
BEAM	BEAM
SPRING	SPRING
SPRING+PROP/SPR_BEAM (TYPE 13)	SPRBM
SH3N	SHELL
SHELL	SHELL
BRICK	SOLID
TETRA44	TETRA44
ADMAS	MASS
SPRING+INTER/TYPE2	PLINK
SPRING+PROP/KJOINT(TYPE33)	KJOIN
RBODY	RBODY

Boundary Conditions

Table 375: Supported Boundary Condition Mappings

Radioss	PAM-CRASH 2G
INIVEL	INVEL
BCS	BOUNC

Radioss	PAM-CRASH 2G
CLOAD	CONLO

Systems

Table 376: Supported System Mappings

Radioss	PAM-CRASH 2G
FRAME/FIX	FRAME
SKEW/MOV	FRAME
SKEW/FIX	FRAME

Materials

Table 377: Supported Material Mappings

Radioss	PAM-CRASH 2G
M1_ELASTIC	MAT_1D TYPE 201, MAT_2D TYPE 101
/PROP/SPRING	MAT_1D TYPE 220
/PROP/SPR_BEAM	MAT_1D TYPE 223
M0_VOID	MAT_2D TYPE 100
M36_PLAS_TAB	MAT_2D TYPE 102/103
M44_COWPER	MAT_2D TYPE 102/103
M2_PLAS_JOHNS	MAT_2D TYPE 102/103

Properties

Table 378: Supported Property Mappings

Radioss	PAM-CRASH 2G
PROP/BEAM	PART, TYPE= BEAM
PROP/KJOINT	PART, TYPE = KJOIN
PROP/SHELL	PART, TYPE = SHELL

Radioss	PAM-CRASH 2G
PROP/SOLID	PART, TYPE = SOLID
PROP/SPRING	PART, TYPE = SPRING
PROP/SPR_BEAM	PART, TYPE = SPRING
PROP/SPR_GENE	PART, TYPE = SPRING
PROP/TRUSS	PART, TYPE = BAR

Curves

Table 379: Supported Curve Mappings

Radioss	PAM-CRASH 2G
FUNCT	FUNCT

Control Cards

Table 380: Supported Control Card Mappings

Radioss	PAM-CRASH 2G
TITLE	TITLE
	OCTRL/
/ANIM/DT	DSYOUPUT

Radioss	PAM-CRASH 2G
/TFILE	THPOUTPUT

Components

Table 381: Supported Component Mappings

Radioss	PAM-CRASH 2G
PART	PART

Contacts

Table 382: Supported Contact Mappings

Radioss	PAM-CRASH 2G
CNTAC36	INTER/TYPE7

Rigid Walls

Table 383: Supported Rigid Wall Mappings

Radioss	PAM-CRASH 2G
RWALL	RWALL

Time History

Table 384: Supported Time History Mappings

Radioss	PAM-CRASH 2G
TH/SPRING	THELE
TH/NODE	THNODE

Radioss	PAM-CRASH 2G
TH/FRAME	THLOC

Sections

Table 385: Supported Section Mappings

Radioss	PAM-CRASH 2G
SECT	SECFO

Groups

Table 386: Supported Group Mappings

Radioss	PAM-CRASH 2G
GRNOD/NODE	GROUP
GRNOD/GENE	GROUP
GRNOD/SUBSET	GROUP
GRNOD/PART	GROUP
GRNOD/MAT	GROUP
GRNOD/PROP	GROUP
GRNOD/GRNOD	GROUP
GRNOD/SURF	GROUP
GRNOD/GRSHEL	GROUP
GRNOD/GRBRIC	GROUP
GRNOD/GRSPRI	GROUP
GRNOD/GRSH3N	GROUP
GRNOD/GRTRUS	GROUP
GRNOD/GRBEAM	GROUP
GENOD/NODENS	GROUP

Radioss	PAM-CRASH 2G
GRNOD/FORMULA	GROUP

Airbags

Table 387: Supported Airbag Mappings

Radioss	PAM-CRASH 2G
BAGIN	MONVOL

Radioss to OptiStruct Conversion Mapping

Elements

Table 388: Supported Element Mappings

Radioss	OptiStruct
BEAM	CBEAM
BRIC20	CHEXA
BRICK	CHEXA, CPENTA, CTETRA
PENTA6	CPENTA
SH3N	CTRIA3
SHELL	CQUAD4
SPRING + PROP/SPRING (TYPE 4)	CBUSH1D
SPRING	CBUSH
TETRA4	CTETRA

Radioss	**OptiStruct**
TRUSS	CROD

Properties

Table 389: Supported Property Mappings

Radioss	**OptiStruct**
PROP/BEAM (TYPE 3)	PBEAM + PBEAMX*
PROP/SH_SANDW (TYPE11)	PCOMP**
PROP/SHELL (TYPE 1)	PSHELL + PSHELLX*
PROP/SOLID (TYPE 14)	PSOLID + PSOLIDX*
PROP/SPR_BEAM (TYPE 13)	PBUSH + PBUSHT
PROP/SPR_GENE (TYPE 8)	PBUSH + PBUSHT
PROP/SPR_PUL (TYPE 12)	PBUSH
PROP/SPRING (TYPE 4)	PBUSH1D
PROP/TRUSS (TYPE 2)	PROD

*Only when you click **Yes** in the **RadiossBlock to OptiStruct conversion tool** dialog.

**Only material, ply thickness, and ply angle is converted.

Materials

Table 390: Supported Material Mappings

Radioss	**OptiStruct**
MAT/COMPSH (LAW 25)	MAT9ORT**
MAT/ELAST (LAW 1)	MAT1
MAT/HONEYCOMB (LAW 28)	MAT9ORT + MATX28*
MAT/PLAS_JOHNS (LAW 2)	MAT1 + MATX02*
MAT/PLAS_TAB (LAW 36)	MAT1 + MATX36*

Radioss	OptiStruct
MAT/VOID (LAW 0)	MAT1 + MATX0*

*Only when you click **Yes** in the **RadiossBlock to OptiStruct conversion tool** dialog.

**The values of NU23 and NU31 are set to default values on conversion.

Samcef Conversion

Convert Samcef to OptiStruct Mapping

Element

HyperMesh elements have two basic attributes, configuration (or config) and type. The "config" defines the basic geometrical shape of an element. For example, Tria3 configuration is a 3 node triangular element and hexa8 is an 8-node hexahedral element. The "type" defines the solver specific element type of a particular configuration. For example, the 4-node quadrilateral (quad4) element in Samcef can be any defined by different hypothesis: MINDLIN, VOLUMIC, and so on. The Element Types panel shows all supported element configurations and their types for a user profile.

Table 391: Supported Element Mappings

Samcef	OptiStruct
tria3, MINDLIN	tria3, CTRIA3
quad4, MINDLIN	quad4, CQUAD4
tetra4, VOLUMIC	tetra4, CTETRA
pyramid5, VOLUMIC	pyramid5, CPYRA
penta6, VOLUMIC	penta6, CPENTA
hex8, VOLUMIC	hex8, CHEXA
tria6, MINDLIN	tria6, CTRIA6
quad8, MINDLIN	quad8, CQUAD8
tetra10, VOLUMIC	tetra10, CTETRA
pyramid13, VOLUMIC	pyramid13, CPYRA
penta15, VOLUMIC	penta15, CPENTA
hex20, VOLUMIC	hex20, CHEXA
bar2, VOLUMIC	BARS, CBEAM
rod, VOLUMIC	rod, CWELD

Samcef	OptiStruct
rbe3, RBE3	rbe3, RBE3

Properties

Table 392: Supported Property Mappings

Samcef	OptiStruct
SHELLPHP	PSHELL
SOLIDMAT	PSOLID
BPR	PBEAML

Material

Table 393: Supported Material Mappings

Samcef	OptiStruct
ISOTROPIC	MAT1
ORTHOTROPIC	MAT9ORT

Load

Table 394: Supported Load Mappings

Samcef	OptiStruct
force, FORCE	force, FORCE
moment, MOMENT	moment, MOMENT
constraint, FIX/DEP	constraint, SPC

Samcef	OptiStruct
pressure, PRESSURE	pressure, PRESSURE

Contacts

Table 395: Supported Contact Mappings

Samcef	OptiStruct
MTI	CONTACT

XY Plotting 16

Study relationships between data vectors in results files.

This chapter covers the following:

Information about xy plots is stored in plot collectors, which are referred to as plots. Plots maintain a list of pointers to curves that are to be displayed on the plot. The plot may contain any number of curves. There is no limit to the number of plot collectors that an HM database may contain.

Information about curves is stored in curve collectors, which are referred to as curves. To display a curve, you must assign the curve collector to a plot. A curve may appear on more than one plot at a time and there is no limit to the number of curves that an HM database may contain.

You can create standard plots or dual plots that show real /imaginary or phase/magnitude data.

XY Plots Module

The XY Plots module is a group of panels that perform operations on plots and the curves displayed on those plots.

The XY Plots module can be accessed from the XY Plots menu.

Axis Labels Panel
: Modify the x and y axes titles and labels. You can also change the color and font size used to display these entities.

Axis Scaling Panel
: Modify the starting and ending values of the plot axes. You can set the values explicitly or implicitly by using the panel functions such as find curves, circle zoom, and zoom out.

Border Panel
: Change the thickness and color of the border around the plot. You may also specify whether the border is displayed and the size of the margin between the border and the plot.

Curve Attribs Panel
: Change the color, marker style (used to indicate the point location), thickness, and the line style (solid, dashed, and so on). You can apply a scaling factor to the original data points. You can also change the curve title that appears in the legend.

Edit Curves Panel
: Creates and modifies the curves in the database. This panel allows you to read data vectors from files as well as perform advanced mathematical operations on curves.

Grid Attribs Panel
: Change the color, line style, thickness of the grid lines, and the margin displayed around the grid lines.

Grid Labels Panel
: Change the color, font, and number of significant places in the labels. Grid labels appear along the x and y axes in the plot (tick marks).

Integrate Panel
: Calculates and displays the integral of a curve.

Legend Panel
: Change the location and the font used to display the legend.

Plot Titles Panel
: Change the plot title, subtitle, and label. In addition, you can change the color and font size used to display these entities.

Plots Panel
: Create an xy plot and assign curves to the xy plot.

Query Curves Panel
: Determine the coordinate values of points in a curve.

Read Curves Panel
: Reads curves from an ASCII file.

Rename Panel
Eename curves.

Results Curves Panel
Generates a curve from the currently-selected results file.

Simple Math Panel
Perform simple mathematical calculations on a curve.

In addition, you can use the Curve Editor to view and modify curves already defined in your HyperMesh model.

Create XY Plots

Create an entirely new plot or a new plot based on default values from an existing plot.

If you have complex data, you can create dual plots to display real/imaginary or phase/magnitude curve data. You can also select the curves that are to be displayed on the plot.

Plot attributes include the title, subtitle, and labels, and also the margin and border around the xy plot. These attributes can be adjusted before or after you add curves to the plot.

1. In the Model Browser, right-click and select **Create** > **Plot** from the context menu.
   A new curve opens in the Entity Editor.
2. In the Name field, enter a name for the plot.
3. Define plot attributes by modifying their corresponding fields.

   Plot attributes include a title, subtitle, axis title, and labels, and so on.
4. Select curves.
   a) Click **Curves** > **0 Curves**.
   b) Click the **curves** entity selector.
   c) In the **Select Curves** dialog, select the curves to display.
   d) Click **OK**.

> **Tip:** When several plots are contained within a database, you can modify one of the values on all of the plots. For example, you may wish to change the axis titles so that they are all the same. You can simultaneously modify one plot so that it has the desired values, and then apply those modifications to the other plots, or a subset of the plots. When you modify xy plots using the panels of the xy plots module, the plot = field allows you to select one plot and the plots entity selector allows you to select multiple plots.

Manage XY Plot Windows

Every xy plot is placed within a window to help you to control multiple plots by resizing and moving plots around the screen.

1. Press **W** to open the Window panel.
2. Modify XY plot windows.

Create Curves on XY Plots

Create Curves Manually

Create and define new curves.

1. In the Model Browser, right-click and select **Create** > **Curve** from the context menu.
   The **Curve editor** dialog opens.
2. Create curve.
   a) Click **New**.
   b) In the panel area, Name field, enter a name for the curve.
   c) Click **proceed**.
3. In the X and Y table, enter curves X and Y coordinates.
4. To display the curve in the graph area, in the Curve List, right-click on the curve and select **Display only** from the context menu.
5. Define curve attributes such as the title, the thickness of the curve line, the style of the line used to draw the curve, the color of the line, and the marker style used to identify the curve.

Edit a curve by right-clicking on it in the Model Browser and selecting **Edit** from the context menu. In the **Curve editor** dialog, edit the curve and click **Update**.

Read Curves from ASCII Files

Create curves by reading in an xy data set from an ASCII file.

Each curve in the file is defined in a block format. The block begins with the statement, XYDATA. After XYDATA, the title assigned to the curve, which is displayed in the legend, follows on the same line. Point data follows with a set of (x, y) data pairs on each line. The block ends with an ENDDATA statement. In the above example, there are two blocks of data, which define two curves.

```
XYDATA,TITLE
X1, Y1
X2, Y2
.
.
.
ENDDATA
XYDATA,TITLE
X1, Y1
X2, Y2
.
.
.
```

```
ENDDATA
```

Figure 1820: Format of the input file

1. From the Post page, click **XY Plots**.
2. Click **read curves**.
3. Click **plot=** and select the plot on which to display the new curve.
4. Double-click **file** and select the file that contains the xy data.
5. Click **input**.

The specified curve data is loaded, and a new curve is created from it.

Create Analysis Based Curves

Create analysis-based curves by extracting data from an HyperMesh binary results file.

When you create an analysis curve, you select entities of interest in your model, and then select a data type for the x axis data points and a data type for the y axis data points. After this information has been supplied, the required data is read from the results file and generates the appropriate curve.

This function is useful after running a model because it allows you to display the data in a two-dimensional format as an xy plot. Once the curve is created, you can integrate the curve and inspect points.

1. From the Post page, click **XY Plots**.
2. Click **results curves**.
3. Select the plot on which to display the new curve.
   - Choose **plot=** to select an existing plot.
   - Choose **create new plot** to create an entirely new plot.
4. Select data.

   If data is in vector format, you can extract the components or the magnitude of the vector. Click the switch after x data type/y data type to change the data that is extracted from the database.

   a) Click **x data type** and select the type of data that should appear on the x axis.
   b) Click **y data type** to select the type of data that should appear on the y axis.
5. Use the entity selector to select nodes or elements to include on the curve.
6. Define the simulation range.
   a) In the start with field, enter the first simulation to be used.
   b) In the end with field, enter the last simulation to be used.

7. Click **create**.

Create Curves using Simple Math Operators

Create curves using simple math operators.

You can combine two curves, transform a curve, or export the curve. For every operation, you can specify that the x or y values of the curve remain fixed.

You can also apply external filters to curves. Examples of external filters are in the filters subdirectory that is provided when this option is selected. Essentially, filters exchange data with HyperMesh, using the standard HyperMesh curve data file format.

1. From the Post page, click **XY Plots**.
2. Click **simple math**.
3. Click **plot=** and select the plot to edit.
4. Select curves.

   For example, when subtracting, the second curve is subtracted from the first curve.

   a) Select the initial curve by clicking **1st curve=** and selecting a curve.
   b) Select the secondary curve by clicking **2nd curve=** and selecting a curve.
5. Select the mathematical operation to perform.

   The available operations depend on the number of curves selected.
6. Specify which values of the curve should remain fixed during the operation.

   Only the values not chosen here are affected by the operation, for example, multiplied or subtracted.

   - Choose x-fixed to keep X values fixed.
   - Choose y-fixed to Y values fixed.
7. In the factor field, enter the amount, in model units, to translate or scale.

   Restriction: Only available when the operation is translate or scale.
8. In the filter field, specify a file.

   Restriction: Only available when the operation is external.
9. In the params field, enter any necessary parameters.

   Restriction: Only available when the operation is external.
10. Click **execute**.

The selected mathematical operations are performed on the selected curves.

Create Curves from Files or Math Expressions

Create new curves using either a data vector in a data file or a math expression.

For example, the data source for the x vector could be a file, and the data source for the y vector could be a math expression.

The x and y fields displays the data sources for the x and y vectors once they are selected.

1. From the Post page, click **XY Plots**.
2. Click **edit curves**.
3. Select the **create** subpanel.
4. Click **plot=** and select the plot on which to create the curve.

   An untitled plot is created if no plot currently exists.
5. Define data source.

To define data source using	Do this
Source file	1. Select **file**. 2. In the File field, navigate to the file that contains curve data. 3. Click **type=** and select a data type. 4. Click **req=** and select a request set. 5. Click **comp=** and select a component. **Tip:** For type, req, and comp, click **+** or **-** to scroll through the list of available options.
Math expressions	1. Select **math**. The math expression must evaluate to an array.

6. Click **create**.

Use the modify subpanel to edit an existing curve.

To access the x and y vectors of a curve, you must indicate the curve number and the x or y vector, in the format, c#.vector. For example:

c1.x
: Reference the x vector of curve 1.

c1.y
: Reference the y vector of curve 1.

If you try to reference a curve that has not been created with this panel, you will receive the message, "This curve has no defined source. Convert to math curve?" Respond by clicking **Yes** or **No**. If the selected curve has too many points, it cannot be converted.

Post-Processing Analysis 17

Learn how to use post-processing functions.

This chapter covers the following:

Result databases can be used during post-processing to create contour, assigned, deformed, and vector plots.

Review results files and databases generated by external codes using post-processing tools. Results files can be translated into HyperMesh results databases which are then read into HyperMesh for post-processing. This translation is done using result translators; for more information, refer to the individual translators in the Interface help system.

HyperMesh Results Database

An HyperMesh Results data base contains translated results files, which are read into HyperMesh for post-processing.

The structure of a results database allows you to access results by a method similar to that of the analysis code. A results database is divided into sections called simulations. Each simulation stores the results for a model as it responds to a loading condition. For example, if you run a linear statics problem and apply three different loading conditions to your model, the results file generated by the translator contains three simulations. If you run a nonlinear job, each load step (the response of the model to each incremental amount of load applied) translates to a simulation.

Each simulation in the results database is further subdivided into data types. Each data type found in a simulation contains a group of results of the same type. For example, each simulation in a results file may contain two data types: displacements and von Mises stress. A data type may contain only one type of result. Data types are one of the forms described below:

Nodal Displacement
Stores three floating point values at a node. This form of data type is usually used to store displacements or a vector quantity.

Nodal Value
Stores one floating point value at a node. This form of data type is used to store stress quantities or other types of results where a single value is needed at a node.

Element Value
Stores one floating point value at an element. This form of data type is used to store stress quantities or other types of results where a single value is needed at an element.

Complex Nodal Displacement
Stores a complex value (magnitude and phase) at a node. This form of data type is usually used to store displacements or a vector quantity.

Complex Nodal Value
Stores a complex value (magnitude and phase) at a node.

Complex Element Value
Stores a complex value (magnitude and phase) at an element.

Complex Nodal von Mises
Stores a complex von Mises value (magnitude, phase, offset) at a node.

Complex Element von Mises
Stores a complex von Mises value (magnitude, phase, offset) at an element.

Data types are not required to contain results for every node or element in the model, and may contain a subset of the total model, if this is appropriate. If this occurs, a message is printed indicating that results for some of the entities requested were not found in the database. In order to complete the post-processing function being executed, HyperMesh sets the results values needed for that function to zero for all of the nodes or elements that are missing.

Load Results File

In order to perform post-processing functions, you must first specify the name and location of the results database.

Select a results file.

- Click **File** > **Load** > **Results** and select the `.res` file.
- In the Global panel, for results file, enter the path and name of the results file or click browse to select a file using the browser.

Create Deformed Geometry Plots

Display the deformed geometry of your model statically, in either wire frame or hidden line mode.

In the Deformed panel, selected simulation must have a data type in it that contains nodal displacement records.
It is from the data contained with the nodal displacement records that allows the calculation of the deformed geometry of the structure.

Create Animations

View your model structure in motion.

Create animation.

Option	Description
Linear	Linear animation creates and displays an animation sequence that starts with the original position of the structure and ends with the fully deformed position of the structure. An appropriate number of frames are linearly interpolated between the first and the last positions. Linear animation is usually selected when results are from a static analysis. Linear animation sequences are generated in the Deformed panel.
Modal	Modal animation creates and displays an animation sequence that starts and ends with the original position of the structure. The deforming frames are calculated based on a sinusoidal function. Modal animation is most useful for displaying mode shapes. Modal animation sequences are generated in the Deformed panel.
Transient	Transient animation displays the structure in its timestep positions as calculated by the analysis code. Transient animation is used to animate the transient response of a structure. Transient animation sequences are generated in the Transient panel.

The selected simulations must include a data type that contains nodal displacement records in order to create an animation sequence. HyperMesh calculates the deformed geometry of the structure from the data contained within the nodal displacement records. For linear and modal animation, HyperMesh uses only one simulation and this simulation must include a data type that contains nodal displacement records. For transient animation, HyperMesh uses a range of simulations. In this case, each of the simulations used in the animation sequence must include a data type that contains nodal displacement records.

Create Vector Plots

Create analysis-based vector plots.

Use the Vector Plots panel to display the model with a vector at each node that has a result-based direction and magnitude. Vector plots are used to determine the direction of movement and allow you to verify the location of the center of rotation of a model. See the Vector Plot panel for more information.

Create Contour Plots

Generate color bands on a model, based on the values found in the results file.

The bands of color are created by calculating a value for each node in the model and then interpolating across each element.

The results file must include a simulation that contains one of the three forms of data types. Each data type is handled differently when it is used to generate a contour plot. When a contour function is performed, the objective is to take all of the results and place them at the nodes of the elements. In order to accomplish this, HyperMesh may have to average results before it can display the contour plot.

nodal values and displacements

The results are stored at the nodes. HyperMesh can create the contour plot without modifying any of the values in the results file.

element values

The values are located at the centroid of the element. HyperMesh averages the centroidal element values to the nodes of the elements. You should be aware that averaging is taking place when element centroid values are used to create a contour plot.

In the Contour panel, use the contour function to generate contour plots.

Create Assigned Plots

Assign a color to each element in the model, based on the values in the results file.

The elements are then displayed in the solid color assigned to them. This allows you to display elements that have values within a specified range.

The results file must include a simulation that contains one of the three forms of data types. Each data type is handled differently when it is used to generate a contour plot. When the assign function is performed, the objective is to take all of the results and place them at the centroid of the elements. In order to accomplish this, HyperMesh may have to average results before it can display the assigned plot.

element values

The results are already stored at the centroid of the element, so no further calculations are required.

nodal values and displacements

HyperMesh averages the results at the nodes to the centroid of the elements. For each element, this is accomplished by adding the results at each node and dividing by the number of nodes on the element. You should be aware that averaging is taking place when nodal values or nodal displacements are being used to create an assigned plot.

In the Contour panel, use the assign function to generate assigned plots.

Add Plot Identification

After you create a results-based plot, you can add titles, modify the colors used in the legend, and relocate the legend and the descriptor.

1. Temporary titles can be added to each type of plot by entering a title in the title field in the Contour panel.

   After you enter the title and create the plot, the temporary title is displayed on the upper left side of the screen.

   HyperMesh creates the "descriptor" in order to display the simulation and data type that were used to create the plot. By default, the descriptor is located in the upper left-hand corner of the plot above the legend.
2. To modify the descriptor, click within the descriptor to access the Title Edit panel, click **color** to change the color of the text of the descriptor, click **font** and select the size font you want to use in the descriptor.
   HyperMesh plots a legend if the results-based plot created requires it.
3. To modify a legend, click within the displayed legend to access the Legend Edit panel.

   Functions on this panel allow you to move the legend to a different location on the screen, change the color of the text in the legend, reverse the colors of the legend, change the font size, and also change the colors used in the legend that correspond to the model.

Inspect Results

A contour or assigned plot provides a fast, convenient way of viewing the results of a large number of elements.

When you want to determine the actual value that an analysis code has calculated for a node or element, you can select the node or element after the results-based plot has been created.
The ID, simulation and data type, and value of the node or element are displayed in the menu area.

Free Body Diagrams

Free Body Diagram (FBD) utilities facilitate the extraction and post-processing of grid point force results.

FBD extractions are typically utilized for breakout and/or sub-modeling analysis schemes, where balanced "free body" sub-cases are extracted from a coarse grid model and applied to a fine grid sub-model for eventual optimization and/or analysis. FBD is also used to extract cross-sectional resultant forces and moments, typically at the centroid of a cross-section, for use in traditional strength calculations.

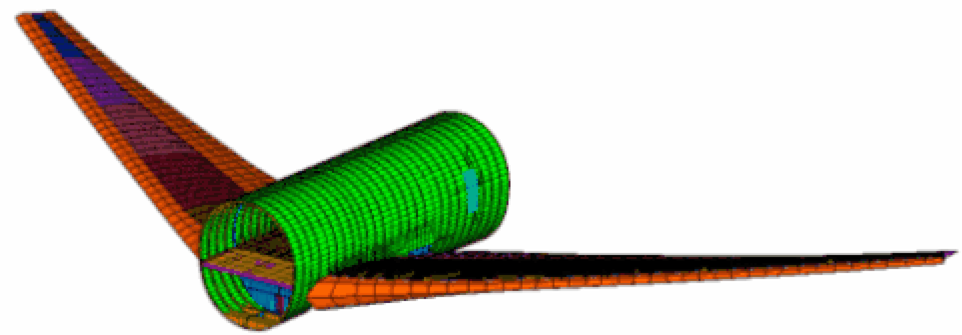


Figure 1821: Coarse Grid Model, Typical for FBD Extractions

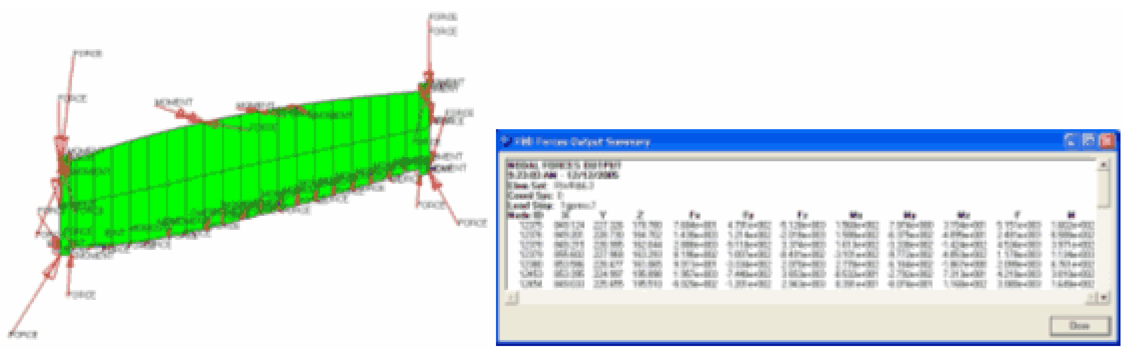


Figure 1822: Typical FBD – Forces Output on a Wing Rib

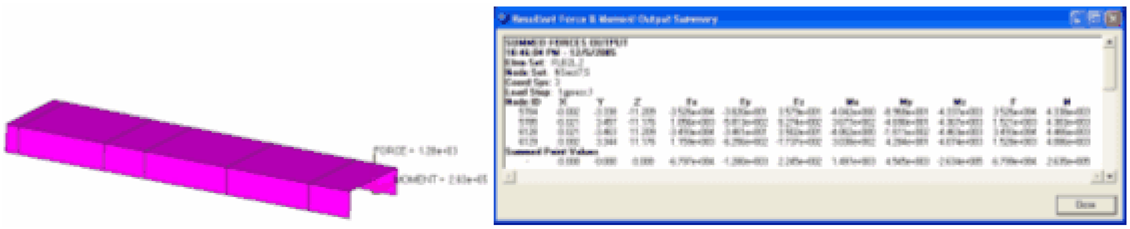


Figure 1823: Typical Result Force and Moment Output on a Floor Beam

Extract Displacement Data

Extracts displacement data for user defined node set(s).

Extracting displacement data is useful for doing breakout modeling within a sub-modeling scheme.

After you define an element set with an associated node set, all appropriate displacements and rotations are extracted. Results can be output to load collectors for graphical review, a text summary table, and a formatted `.csv` file which can be loaded into traditional spreadsheet software packages.

1. From the menu bar, click **Post** > **Free Body** > **Displacements**.
   The FBD Displacements tab opens.
2. In the .op2/.odb file field, specify the full path and filename of the results file containing the displacement output for the current model.
   Once an results file is selected, loadstep names and IDs with displacement output are saved with the rest of the HyperMesh database for use with all FBD utilities. If a new results file is required, or if the original results file changes, you must load the new results file into the database, overwriting the previously selected one.
3. In the Loadsteps section, select which loadstep(s) to extract displacement information.

   Only loadsteps with displacement results from the currently selected results file display for selection. You can select multiple loadsteps by Ctrl-clicking or Shift-clicking. Filter buttons allow for additional selection control, including a namefilter that uses standard HyperMesh filtering syntax. The loadstep display can be switched between **ID** and Name **(ID)**.

   The **ID** option lists the loadsteps as "SUB-CASE #". The **Name (ID)** option lists the loadstep as "SUBTITLE - LABEL(ID)". If no SUBTITLE exists, only the LABEL is used. See the OptiStruct online reference guide for more information regarding SUBTITLE and LABEL loadstep information cards.
4. Select entities to extract displacement data from.

   The entity selection section allows you to select and/or create the appropriate entities required to execute the FBD Displacements utility.

   a) Use the Element Set selector to define the elements that contain the nodes at which displacement data will be extracted.

      The Set Browser utility on the Tools menu can be used to create the necessary element sets.

   b) Use the Node Sets selector to define the nodes at which displacement data will be extracted.

      Only the nodes contained within the selected node set will be part of the extraction. If a node set is not selected, then all nodes within the element set are used.

c) To automatically finds the nodes attached to elements that are not contained within the currently selected element set, select the **Auto find interface nodes** checkbox.

This procedure selects the nodes interfacing with the remainder of the structure. You will be prompted to give the newly created node set a name. Additional nodes may be added to the node set once it is created by clicking the Node Setselector and picking additional nodes.

d) To automatically displays the entire model in transparency mode and highlights the currently selected element and node sets, select the **Show Model** checkbox.

This functionality allows you to verify which element and node sets are currently selected.

5. Define Output options.

The Output options section contains various options to review and display results of FBD Displacement extractions.

a) Use the **Coordinate Systems** selector to determine the coordinate system used to display the nodal coordinates (x,y,z) in the summary table and `.csv` file output options. Displacement data (Ux, Uy,…) is always output in the system that the results are stored with in the results file format. Results coordinate system transformations are not performed on displacement data.

The FBD Displacement utility extracts and applies the displacement and rotation results from the results file in the output coordinate system without any further coordinate system transformations. It is assumed that the output coordinate system assigned to each node in the HyperMesh database matches that used to run the analysis and generate the results file. Output coordinate systems are defined in HyperMesh by accessing the Systems panel. On the Setup menu, click **Coordinate Systems**, and toggle to the **assign** subpanel, select the required nodes and a coordinate system, and click **Set Analysis**. In OptiStruct and Nastran this operation sets the CD field on the GRID card(s). See the OptiStruct online reference guide for more information regarding the GRID bulk data card. If the output coordinate systems for each node in the HyperMesh database does not match those used to run the analysis then the extracted values will be incorrect. Situations when this behavior could occur include modification of nodal output system within HyperMesh without rerunning the analysis and/or loading a results file that does not match the currently loaded model.

If a coordinate system is not specified, the HyperMesh "base" system is used by default.

b) In the Zero Tolerance field, enter the cut-off point below which a result quantity is considered zero. All calculations are done with float point precision and the zero tolerance value is only used for controlling the output of results to the various formats. The option helps to eliminate "relatively small" values from being output to the result formats. To maintain float precision the default is set to 1.0e-6, otherwise modify the value as desired.

c) To extract the specified displacement data and display it in organized load collectors within HyperMesh for graphical visualization within the model window5, select the **Create Load Collectors** checkbox.

A single load collector, for the current element and node set, is created for each loadstep. The load collector name format is FBDD_E(#)_N(#)_S(#)_Disp. For example, FBDD_E(1)_N(1)_S(1)_Disp would be created for element set 1, node set 1 and loadstep 1. The loads in this load collector are created with the SPC load type. This collector can be referenced as the SPC in the loadstep panel.

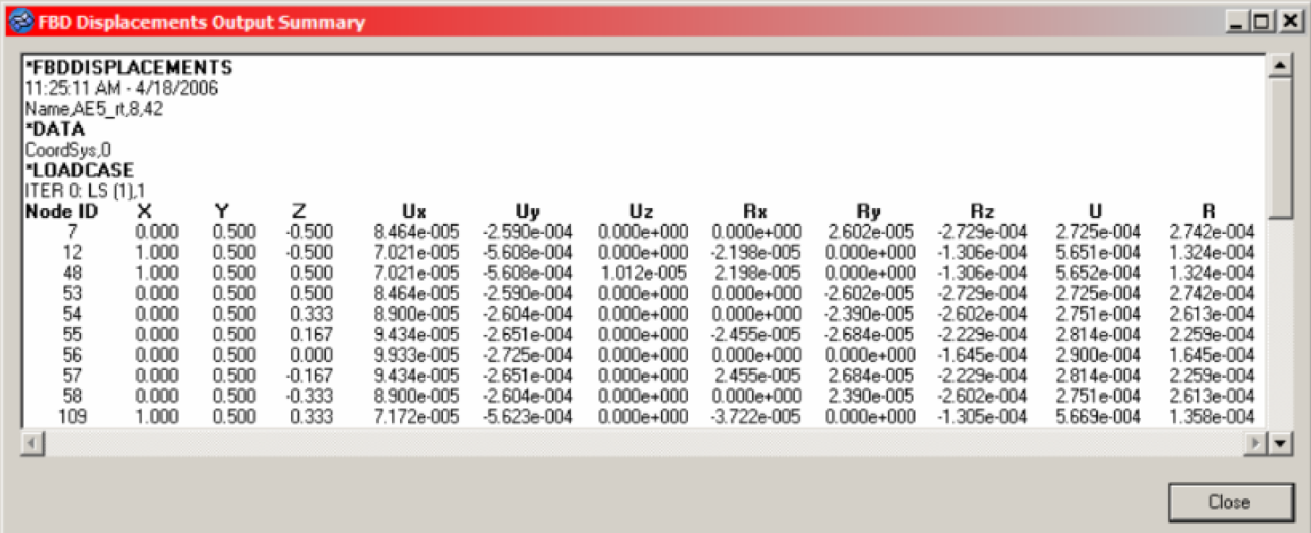
> **Tip:** To choose a color for all created load collectors, click the **Color** button. This color can be modified later using either the interface or the FBD Results Manager utility

d) To create a load collector with the name "FBDD_E(#)_N(#)_S(#)_SPCD", select the **Create SPCD** checkbox.

   Loads in this collector are created with the SPCD load type. This collector can be referenced as the LOAD in the loadstep panel.

e) To output the results to a popup window for instant review, select the **Show summary** checkbox.

   The table contains information about the loadsteps, element and node set(s), and detailed displacement data at each node.


FBD Displacements Output Summary

*FBDDISPLACEMENTS
11:25:11 AM - 4/18/2006
Name,AE5_rt,8,42
*DATA
CoordSys,0
*LOADCASE
ITER 0: LS (1),1

Node ID	X	Y	Z	Ux	Uy	Uz	Rx	Ry	Rz	U	R
7	0.000	0.500	-0.500	8.464e-005	-2.590e-004	0.000e+000	0.000e+000	2.602e-005	-2.729e-004	2.725e-004	2.742e-004
12	1.000	0.500	-0.500	7.021e-005	-5.608e-004	0.000e+000	-2.198e-005	0.000e+000	-1.306e-004	5.651e-004	1.324e-004
48	1.000	0.500	0.500	7.021e-005	-5.608e-004	1.012e-005	2.198e-005	0.000e+000	-1.306e-004	5.652e-004	1.324e-004
53	0.000	0.500	0.500	8.464e-005	-2.590e-004	0.000e+000	0.000e+000	-2.602e-005	-2.729e-004	2.725e-004	2.742e-004
54	0.000	0.500	0.333	8.900e-005	-2.604e-004	0.000e+000	0.000e+000	-2.390e-005	-2.602e-004	2.751e-004	2.613e-004
55	0.000	0.500	0.167	9.434e-005	-2.651e-004	0.000e+000	-2.455e-005	-2.684e-005	-2.229e-004	2.814e-004	2.259e-004
56	0.000	0.500	0.000	9.933e-005	-2.725e-004	0.000e+000	0.000e+000	0.000e+000	-1.645e-004	2.900e-004	1.645e-004
57	0.000	0.500	-0.167	9.434e-005	-2.651e-004	0.000e+000	2.455e-005	2.684e-005	-2.229e-004	2.814e-004	2.259e-004
58	0.000	0.500	-0.333	8.900e-005	-2.604e-004	0.000e+000	0.000e+000	2.390e-005	-2.602e-004	2.751e-004	2.613e-004
109	1.000	0.500	0.333	7.172e-005	-5.623e-004	0.000e+000	-3.722e-005	0.000e+000	-1.305e-004	5.669e-004	1.358e-004

Close

Figure 1824:

f) To create a `.csv` file that contains the same information as the summary table, but in a comma-separated file, select the **Create .csv file** checkbox, then select a new file or an existing file.

   If the data you are extracting already exists in the file, based on element set, node set and loadstep IDs, the existing block will be overwritten with the new data. If it does not exist, it will be appended to the end of the file.

   In any case, you will be warned that the file already exists and asked if you want to replace it. Selecting **yes** will not overwrite the file; it will append/replace the data.

6. Click **Accept**.

Extract Forces

Extract grid point force data, including forces and moments, for a user-defined element set.

Extracting FBD forces is useful for doing breakout modeling within a sub-modeling scheme.

Results can be output to load collectors for graphical review, a text summary table, and/or a formatted Comma-Separated Values (`.csv`) file which can be loaded into traditional spreadsheet software packages.

1. From the menu bar, click **Post** > **Free Body** > **Force**.
   The FBD Forces tab opens.
2. In the .op2/.odb file field, specify the full path and filename of the results file containing the forces output for the current model.

   Once you have selected an results file, loadstep names and IDs with grid point force output are saved to the database for use with all FBD utilities. The loadstep name and ID information is retained within the HyperMesh database once saved.

   If a new results file is required, or if the original results file changes, you must load the new results file into the database, overwriting the previously selected.
3. In the Loadsteps section, select which loadstep(s) to extract grid point force information.

   Loadsteps with grid point force results from the currently selected results file are displayed for selection only. Multiple loadsteps can be selected via Ctrl-click or Shift-click functionality. Filter buttons allow for additional selection control as shown including a name filter that uses standard filtering syntax. The loadstep list can be switched between ID and Name (ID).

   The ID option lists the loadsteps in the format SUBCASE ID. The Name (ID) option lists the loadstep in the format NAME – ITERATION (ID).
4. Select entities.
   a) Use the Element Set selector to define the elements that make up the free body and contain the nodes at which GPFORCE data will be extracted.

      The Set Browser tool can be used to create the necessary element sets.
   b) Use the Result System selector to define the coordinate system into which the grid point force and moment result vectors are transformed and output.

      If a results system is not specified, the HyperMesh "base" system is used by default.

      The FBD Forces utility extracts grid point force and moment results from an results file in the output coordinate system in which the solver output these results. HyperMesh assumes that the output coordinate system assigned to each node in the HyperMesh database matches that used to run the analysis and generate the results file. Output coordinate systems are defined in HyperMesh by accessing the Systems panel. On the assign subpanel, select the required nodes and a coordinate system, and click **Set Analysis**. In OptiStruct and Nastran this operation sets the CD field on the GRID card(s).

      If the output coordinate systems for each node in the HyperMesh database do not match those used to run the analysis, the extracted values will be incorrect. This could occur when modifying a nodal output system within HyperMesh without rerunning the analysis and, or when loading a results file that does not match the currently loaded model. In addition, results coming from, or output to, cylindrical or spherical result coordinate systems should be inspected for validity near the origin and along principal axes.

c) Use the Summation Node selector to define the node about which the GPFORCE data is summed for the selected element set.

   This is useful for verifying free body behavior through zero-sum values for all force and moment components about any node. It is also useful for calculating the result of applied or reaction forces about any node. If a node is not selected, the HyperMesh origin (0,0,0) is used by default.

d) To automatically display the entire model in transparency mode while highlighting the currently selected element set, result system and summation node, select the **Show Model** checkbox.

   This allows you to verify which element sets is currently selected.

5. Define output options.

a) For FBD type, select the type of grid point force and moment data to extract and utilize for FBD calculations for each node in the selected element set.

   - Choose **All Loads** to extract and utilize all element contribution, applied, SPC, and supported MPC grid point data for FBD calculations on the nodes in the selected element set.
   - Choose **Applied Loads Only** to extract and utilize only the applied loads grid point data for FBD calculations on the nodes in the selected element set.
   - Choose **Reaction Loads Only** to extract and utilize only SPC and supported MPC grid point data for FBD calculations on nodes in the selected set.

   MPC force and moment data are properly extracted for the following MPC constraint types: RBE2, RBE3, Rigidlink, RJOINT, RROD, and RBAR.

b) In the Zero Tolerance field, enter the cut-off point below which a result quantity is considered zero.

   All calculations are performed with floating point precision and the zero tolerance value is only used to control the output of results to the various formats. This option helps to prevent relatively small values from being output to the result formats. To maintain floating-point precision the default is set to 1.0e-6; modify the value as desired.

c) To extract the specified grid point data and display it in organized load collectors for visualization in the model window, select the **Create Load Collectors** checkbox.

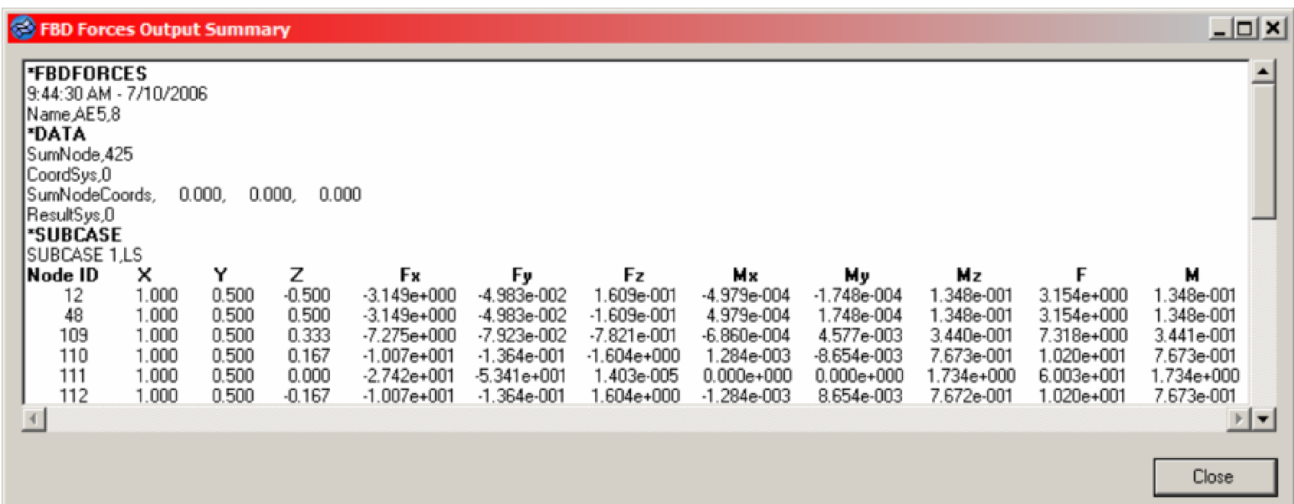
   Multiple load collectors are created — one for each force and moment component — for each selected loadstep of the current element set. The load collector name format is "FBDF_E(#)_S(#)_(compID)". For example FBDF_E(1)_S(1)_Fx would be created for element set 1, loadstep 1, and component Fx. In addition a load collector with the Nastran/OptiStruct `LOAD` card is also created, referencing the component force and moment load collectors. This load collector is named "FBDF_E(#)_S(#)_C" and can be referenced in the loadstep panel as the `LOAD` entry for the various loadstep definitions.

> **Tip:** To choose a color for all created load collectors, click the **Color** button. This color can be modified later using either the interface or the FBD Results Manager utility.

The FBD Results Manager can be used to review the load collectors generated from FBD Forces utility. When you save the database, all FBD Forces load collectors are saved to the database. This allows FBD information to be reviewed and utilized in the future without having to rerun the tool. Renumbering element or node sets after running the tool invalidates the link between the load collector names and the associated sets; therefore it is important to avoid renumbering any element or node sets for which FBD result must be retained as load collectors in HyperMesh.

d) To output the results to a popup window for instant review, select the **Show summary table** checkbox.

The table contains information about the loadsteps, element set(s), and detailed data from the grid point extraction at each node.


FBD Forces Output Summary

*FBDFORCES
9:44:30 AM - 7/10/2006
Name,AE5,8
*DATA
SumNode,425
CoordSys,0
SumNodeCoords, 0.000, 0.000, 0.000
ResultSys,0
*SUBCASE
SUBCASE 1,LS

Node ID	X	Y	Z	Fx	Fy	Fz	Mx	My	Mz	F	M
12	1.000	0.500	-0.500	-3.149e+000	-4.983e-002	1.609e-001	-4.979e-004	-1.748e-004	1.348e-001	3.154e+000	1.348e-001
48	1.000	0.500	0.500	-3.149e+000	-4.983e-002	-1.609e-001	4.979e-004	1.748e-004	1.348e-001	3.154e+000	1.348e-001
109	1.000	0.500	0.333	-7.275e+000	-7.923e-002	-7.821e-001	-6.860e-004	4.577e-003	3.440e-001	7.318e+000	3.441e-001
110	1.000	0.500	0.167	-1.007e+001	-1.364e-001	-1.604e+000	1.284e-003	-8.654e-003	7.673e-001	1.020e+001	7.673e-001
111	1.000	0.500	0.000	-2.742e+001	-5.341e+001	1.403e-005	0.000e+000	0.000e+000	1.734e+000	6.003e+001	1.734e+000
112	1.000	0.500	-0.167	-1.007e+001	-1.364e-001	1.604e+000	-1.284e-003	8.654e-003	7.672e-001	1.020e+001	7.673e-001

Close

Figure 1825:

e) To create a `.csv` file that contains the same information as the summary table, but in a comma-separated file, select the **Create .csv file** checkbox and select a new file or an existing file.

If the data you are extracting already exists in the file, based on element set, loadstep IDs, the existing block will be overwritten with the new data. If it does not exist, it will be appended to the end of the file.

You are asked if you wish to replace the existing file. However, selecting **yes** will not overwrite the file, it will append/replace the data as described above.

6. Click **Accept**.

Create and Manage FBD Cross-Section Definitions

Create and manage cross-section definitions that are used within the Resultant Force and Moment utility.

Cross-sections, are defined by an element set, node set, summation node, and a local result coordinate system.

The FBD Cross-section Manager features semi-automatic generation of element and node sets for defining cross-sections. Functionality is also available to reorder FBD cross-sections.

The FBD Cross-section Manager interface has two creation methods available for cross-section definition: manual and (semi-) automatic. The Advanced options section provides the means to semi-automatically create cross-section element and node sets for beam-like structures with regular meshes. This auto-create cross-section capability requires a continuous mesh with rows of nodes (of any orientation) to work properly. The mesh should not have any discontinuities (such as holes and gaps) and must have identifiable rows of nodes, starting from the selected nodes and progressing along the length of the selected elements.

Manually Define Cross Sections

1. From the menu bar, click **Post** > **Free Body** > **Cross Section**.
   The FBD Cross Section Manager tab opens.
2. Use the Element Set selector to specify the elements containing the nodes that define the cross-section.

   The Set Browser can be used to create the necessary element sets.

   If multiple element sets are selected, each set is added to the table as a separate cross-section definition which can be modified later by selection.
3. Use the Node Set selector to define the nodes in each currently selected element set at which grid point data will be extracted and summed from.

   Only the nodes contained within the selected node set will be part of the grid point extraction. Use the Set Browser to create the necessary node sets.

   If multiple node sets are selected for a single element set, HyperMesh adds separate cross-section definitions to the table with the original element set and each selected node set.
4. Use the Summation Node selector to define the node about which the grid point data will be summed.

   If no node is selected, the utility defaults to "Centroid". This option calculates the nodal averaged centroid of the coordinates of all of the nodes in the node set and creates a temporary node at that location.

   When using the "Centroid" option, a temporary node is created. If this node is deleted from the model, the loads associated with that node are also deleted.

5. Use the Result System selector to define the coordinate system into which the grid point vector results will be transformed and output.

   If a results system is not specified, the HyperMesh "base" system is used by default.
6. To display the element set, node set, result system and summation node which define the cross-section in the graphics display area, select the **Display sections** checkbox.
7. To display the entire model in transparency mode, highlighting the currently selected element set, node set, result system and summation node, select the **Show model** checkbox.

   This allows you to verify a cross-section definition.
8. Use the filter buttons on the top of the spreadsheet to select which cross-sections are required.

   Standard Ctrl/Shift–click functionality can be used to select cross-sections. Selected cross-sections can also be deleted from the database by using the **Remove** selection button on the right end of the filter buttons.

Each time a cross-section is created, modified, or deleted, the cross-section data is saved to the database. When the database is saved, all cross-section definitions are also saved. Therefore, cross-section definitions only need to be defined once and stored in the database. Cross-Sections can then be accessed from within the Resultant Force and Moment utility, which utilizes the cross-section definitions to perform these calculations.

Renumbering element or node sets after running the tool invalidates the link between the cross-section names and the associated sets. Therefore, it is important to avoid renumbering any element or node sets for which cross-sections are to be retained within the database.

Automatically Define Cross Sections

1. From the menu bar, click **Post** > **Free Body** > **Cross Section**.
   The FBD Cross Section Manager tab opens.
2. Use the Elements selector to choose the elements that define the entire "beam-like" component from which cross-sections will be generated.
3. Use the Nodes selector to pick nodes for the first node set (first cross-section).

   These nodes should be at one end of the beam.
4. To automatically display the entire model in transparency mode, highlighting the currently selected elements and nodes, select the **Show model** checkbox.
5. In the Element set prefix field, enter a prefix for the name of each generated element set.

   For example, you type in "ESET" each element set will be named ESET [#], where "#" increases with each new set generated.
6. In the Node set prefix field, enter a prefix for the name of each generated node set into the Node set prefix field.

   For example, you type in "NSET" each element set will be named NSET [#], where "#" increases with each new set generated.
7. In the Numbering offset field, enter an offset value for generated set names.

By default, the offset value is zero and HyperMesh generates numbered set names starting with one. If the offset value is set to a value greater than zero, the generated set names are numbered starting from that value.

8. To have each element set contain the elements from the previous set, select the **sets accumulate** checkbox.

   This determines whether each progressive set also contains the elements from the previous set, or only the new "row" of elements.

Extract Resultant Force and Moment Data

Extract grid point force data for user defined cross-sections created via the Cross-section Manager.

The Resultant Force and Moment utility generates input data for shear and moment (VMT) diagrams and/or to perform load-case screening with Potato plots in HyperGraph. Two utilities available within HyperGraph are available for these purposes.

Results can be output to load collectors for graphical review, a text summary table, and/or a formatted `.csv` file which can be loaded into traditional spreadsheet software packages.

1. From the menu bar, click **Post** > **Free Body** > **Resultant Force and Moments**.
   The Resultant Force and Moments tab opens.
2. In the .op2/.odb file field, specify the full path and filename of the results file containing the desired GPFORCE output for the current model.

   When a results file is selected, loadstep names and IDs with GPFORCE output are saved to the database for use with all FBD utilities. The loadstep name and ID information is retained within the database once saved. If a new results file is required, or if the original results file changes, you must load the new results file into the database, overwriting the previously selected.
3. In the Loadsteps section, select which loadstep(s) to extract GPFORCE information.

   Loadsteps with GPFORCE results from the currently selected results file display for selection only. Multiple loadsteps can be selected by Ctrl-clicking and Shift-clicking. The list can be filtered using the buttons provided, including a name filter that uses standard HyperMesh filtering syntax. The loadstep list can be organized by **ID** or **Name (ID)**.

   The **ID** option lists the loadsteps as "SUBCASE #". The **Name (ID)** option lists the loadstep as "SUBTITLE – LABEL (ID)". If no SUBTITLE exists, only the LABEL is used. See the OptiStruct online reference guide for more information regarding SUBTITLE and LABEL loadstep information cards.
4. In the Cross Sections section, select the cross-sections from which you want to calculate resultant force and moment results for each selected loadstep.

   Cross-sections are created using the Cross-section Manager. Multiple cross-sections can be selected by Ctrl-clicking and Shift-clicking. The list can be filtered using the buttons provided, including a name filter that uses standard HyperMesh filtering syntax.
5. Define Output options.

   The Output options section contains various options to review and display the results of Resultant Force and Moment extractions

a) Use the **Coordinate Systems** selector to determine the coordinate system used to output the nodal coordinates (x,y,z) in the summary table, `.csv` file, and `.fbd` file output options.

   Resultant force and moment vector results are always output in the result coordinate system defined for each cross-section. Result coordinate systems for cross-sections are defined using the Cross-section Manager. If a coordinate system is not specified, the HyperMesh "base" system is used by default.

   The Resultant Force and Moment utility extracts grid point force and moment results from the results file in the output coordinate system in which the solver output these results. It is assumed that the output coordinate system assigned to each node in the database matches that used to run the analysis and generate the results file. Output coordinate systems are defined by accessing the Systems panel. On the assign subpanel, select the required nodes and a coordinate system, and click **Set Analysis**. In OptiStruct and Nastran this operation sets the CD field on the GRID cards.

   If the output coordinate systems for each node in the database do not match those used to run the analysis, the extracted values will be incorrect. This can occur when modifying nodal output system without rerunning the analysis, and/or loading a results file that does not match the currently loaded model. In addition, results coming from or output to cylindrical or spherical result coordinate systems should be inspected for validity near the origin and along principal axes.

   MPC force and moment data are properly extracted for the following MPC constraint types: RBE2, RBE3, Rigidlink, RJOINT, RROD, RBAR.

b) In the Zero Tolerance field, enter the cut-off point below which a result quantity is considered zero.

   All calculations are done with float point precision and the zero tolerance value is only used to control the output of results to the various formats. The option helps to eliminate relatively small values from being output. To maintain float precision the default is set to 1.0e-6, otherwise modify the value as desired.

c) Extract the specified grid point data and display it in organized load collectors for graphical visualization within the model window. Select the **Create Load Collectors** checkbox.

   Multiple load collectors are created — one for each force and moment component — for each selected loadstep of the current cross-section, each made up of an element set and node set. The load collector name format is "RF&M_E(#)_N(#)_S(#)_(compID)". For example RF&M_E(1)_N(1)_S(1)_Fx would be created for element set 1, node set 1, loadstep 1, and component Fx.

   > **Tip:** To choose a color for all created load collectors, click the **Color** button. This color can be modified later using either the interface or the FBD Results Manager utility.

   You can also use the FBD Results Manager to review the load collectors generated from the Resultant Force and Moment utility. When the database is saved, all resultant force and moment load collectors are saved in the database. This allows resultant force and moment information to be reviewed and utilized in the future without having to rerun the tool. Renumbering element or node sets after running the tool invalidates the link between the load collector names and the associated sets and therefore it is important to not renumber any element or node sets for which resultant force and moment result are to be retained as load collectors in HyperMesh.

d) To output the results to a popup window for instant review, select the **Show summary** checkbox.

The table contains information about the loadsteps and cross-sections, and detailed data from the grid point extraction at each node.

e) To create a `.csv` file that contains the same information as the summary table, but in a comma-separated file, select the **Create .csv file** checkbox, then select a new or existing file.

If an existing file is selected, it is appended to.

f) To create a file that can be read into HyperGraph using the "Shear and Moment Plot" and "Potato Plot" utilities, select the **Create .fbd file** checkbox, then select a new or existing file.

6. Click **Accept**.

When saving over existing `.csv` or `.fbd` file, there are several items to note.

- If the data you are extracting already exists in the file, based on element set, loadstep IDs, the existing block will be overwritten with the new data. If it does not exist, it will be appended to the end of the file.
- You are asked if you wish to replace the existing file. However, selecting 'yes' will not overwrite the file; it will append/replace the data into the file as described above.
- Two utilities available within HyperGraph interact with data generated from the Resultant Force and Moment utility: Shear and Moment Diagrams (VMT Plots) and Potato Plot. These utilities are accessed from the Free Body Diagrams item within the HyperGraph Utilities menu.

Review Load Collectors

Graphically review and manage the load collectors generated from all FBD and Resultant Force and Moment utilities.

1. From the menu bar, click **Post** > **Free Body**.
   The FBD Results Manager tab opens.
2. Select one or more element sets.

   The Entity selection section allows you to select the element set for which to review FBD, Displacement, or Resultant Force and Moment load collector output.

   You must choose an existing element set for which you wish to the review the FBD, Displacement, and Resultant Force and Moment load collector output.

   The optional **Show model** checkbox, when checked, displays the entire model in transparency mode and highlights the currently selected element set.
3. Select the desired Results type.

   The Results selection section lets you select the FBD result type to review. Valid types include FBD Displacements, FBD Forces (All Loads), FBD Forces (Applied Loads), FBD Forces (Reaction Loads), and Resultant Force and Moment.

Selecting an FBD result type scans the HyperMesh database and updates the loadstepslist with available loadsteps for the currently selected element set and result type.

4. Select the desired loadsteps by clicking on them.

   Multiple loadsteps can be selected using standard Ctrl/Shift–click functionality. Filter buttons allow for additional selection control as shown including a name filter that uses standard HyperMesh filtering syntax.

   For FBD Displacements and Resultant Force and Moment Result types, the Results selection interface is modified to include a Node sets selector. This selector lists all of the node sets associated with the currently selected element set and loadsteps. If multiple loadsteps are selected, only the node sets that are common to all of them will be listed.
5. In the Display options section, select which force and moment components display in the graphics area for the current selection.
   - The **Fx**, **Fy**, **Fz** checkboxes determine which component/resultant vectors display when you click **Accept**. Grayed-out checkboxes indicate force and moment components or results that can't display for the currently selected element set/loadstep/node sets. These checkboxes are disabled for the FBD Displacements Result type.
   - The **Update load collector color** option will recolor the load collectors associated with the current selection to the color selected in the Color option. Click the **Color** box to pick a different color, if desired.
6. Choose a command.
   - Click **Accept** to display the selected result vectors on in the graphics area.
   - Click **Delete**to delete the load collectors associated with the current selection. A popup warning tells you what will be deleted and requires you to confirm the deletion.
   - Click **Reset** to clear the graphics area of all result vectors and resets all of the FBD Results Manager entry fields.

Export Load Collectors

Export FBD, Displacement, and/or Resultant Force and Moment load collectors generated by other FBD utilities.

After export, the exported data can be used for breakout modeling within a sub-modeling scheme.

1. From the menu bar, click **Post** > **Free Body**.
   The FBD Export Manager tab opens.
2. Use the Element set selector to specify the set for which you want to export data.
3. To display the entire model in transparency mode, highlighting the currently selected element set for review, select the **Show model** checkbox.
4. Choose the results type to export.

   This populates the list of loadsteps. Click the one(s) that you wish to export to highlight them; use <Ctrl>-click or <Shift>-click to select multiple results.

If you choose **FBD Displacements** or **Resultant Force and Moment** as the result type, an additional list of node sets displays. Select the desired node sets in the same fashion as the loadsteps.

5. Click **add to export** to add the highlighted results to the export batch.
6. Define export options.
   a) To create loadsteps upon export, select the **Create appropriate loadsteps** checkbox and then click the open-folder button in the Output file text box.

      This opens a standard file browser window that you can use to browse to the desired destination directory and either select an existing file, or type in a name for a new one.

      This option will also create SUBTITLE and LABEL cards if they are available from the loadstep information within the currently selected results file.

   b) Select the Results type to determine how loadsteps are created.
      - Choose **FBD Forces – All Loads** to create a SUBCASE with LOAD = assigned to the "FBDF_E(#)_S(#)_C" load collector which references the LOAD card pointing to each component "FBDF_E(#)_S(#)_(compID)" load collector for the selected loadstep.
      - Choose **FBD Displacements** to create a SUBCASE with SPC = assigned to the "FBDD_E(#)_N(#)_S(#)_Disp" load collector for the selected loadstep.
        - If the SPCD option was enabled when creating FBD Displacement loads, LOAD = will also be assigned to the "FBDD_E(#)_N(#)_S(#)_SPCD" load collector for the selected loadstep.
        - If multiple node sets are selected for export the following will occur:

      An SPCADD load collector will be created and the appropriate "FBDD_E(#)_N(#)_S(#)_Disp" load collectors will be assigned to it. The SPC = will reference the newly created SPCADD load collector.

      If the **SPCD** checkbox was selected when creating the FBD Displacement loads, a LOAD load collector will be created and the appropriate "FBDD_E(#)_N(#)_S(#)_SPCD" load collectors will be assigned to it. The LOAD = will reference the newly created LOAD load collector.

      If FBD Force and FBD Displacement load collectors from the same loadstep are selected for export, you are prompted to select one or the other from which to create the loadstep.

7. Choose a command.
   - Click **Export** to execute the export process, meaning that clicking it:
     - Turns off the display of all currently displayed elements
     - Creates temporary mass elements on the nodes where the selected FBD, Displacement, and/or Resultant Force and Moment loads are displayed
     - Exports the model with the "displayed" option
     - Deletes the temporary mass elements from the current model
     - Removes unnecessary header information from the output file

- Click **Reset** to clear all of the export criteria so that you can start over.

Review Grid Point Force Balance

The Grid Point Force Balance table is the data around which all FBD-Forces and Resultant Force and Moment utility calculations are performed.

See the OptiStruct online reference guide for more information regarding GPFORCE option cards.

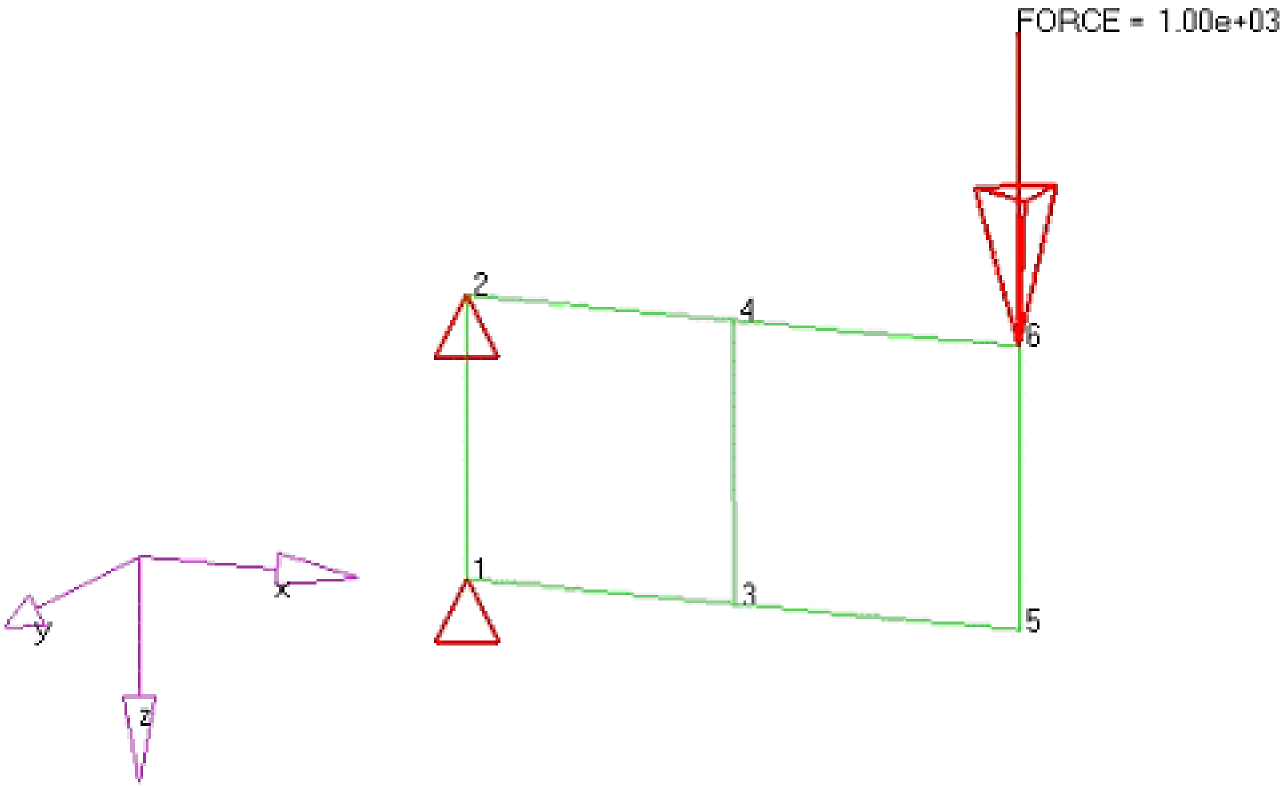


Figure 1826: Sample Model used to Demonstrate Grid Point Force Calculations

This model consists of two elements, a fixed support on the left end, and a point load on the right end.

The complete GPFORCE table for the above cantilever beam model is presented below.

> **Note:** For any given node within the GPFORCE table, several types of entries are possible depending on the forces acting at that node.
>
> - Applied forces and moments
> - SPC forces and moments
> - MPC forces and moments
> - Element forces and moments from elements attached to the node
> - Total summed values for each node, which in turn must sum to zero for the complete GPFORCE table.

```
Grid point forces for node        1 Subcase ID =        1
--------------------------------------------------------------------------------
Force  Element   x-force    y-force    z-force    x-moment   y-moment   z-moment
--------------------------------------------------------------------------------
SPC       --    2.121E+03  0.000E+00 -4.332E+02  0.000E+00  2.370E+01  0.000E+00
Appl.     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
F-MPC     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
Elem.      1 -2.121E+03  0.000E+00  4.332E+02  0.000E+00 -2.370E+01  0.000E+00
Total     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
--------------------------------------------------------------------------------

Grid point forces for node        2 Subcase ID =        1
--------------------------------------------------------------------------------
Force  Element   x-force    y-force    z-force    x-moment   y-moment   z-moment
--------------------------------------------------------------------------------
SPC       --   -2.121E+03  0.000E+00 -5.668E+02  0.000E+00  2.277E+01  0.000E+00
Appl.     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
F-MPC     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
Elem.      1    2.121E+03  0.000E+00  5.668E+02  0.000E+00 -2.277E+01  0.000E+00
Total     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
--------------------------------------------------------------------------------

Grid point forces for node        3 Subcase ID =        1
--------------------------------------------------------------------------------
Force  Element   x-force    y-force    z-force    x-moment   y-moment   z-moment
--------------------------------------------------------------------------------
SPC       --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
Appl.     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
F-MPC     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
Elem.      1    1.085E+03  0.000E+00 -6.426E+02  0.000E+00  8.871E+00  0.000E+00
Elem.      2 -1.085E+03  0.000E+00  6.426E+02  0.000E+00 -8.871E+00  0.000E+00
Total     --   -2.274E-13  0.000E+00 -4.775E-12  0.000E+00  9.948E-14  0.000E+00
--------------------------------------------------------------------------------

Grid point forces for node        4 Subcase ID =        1
--------------------------------------------------------------------------------
Force  Element   x-force    y-force    z-force    x-moment   y-moment   z-moment
--------------------------------------------------------------------------------
SPC       --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
Appl.     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
F-MPC     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
Elem.      1 -1.085E+03  0.000E+00 -3.574E+02  0.000E+00  1.024E+01  0.000E+00
Elem.      2    1.085E+03  0.000E+00  3.574E+02  0.000E+00 -1.024E+01  0.000E+00
Total     --   -1.592E-12  0.000E+00 -2.387E-12  0.000E+00  9.415E-14  0.000E+00
--------------------------------------------------------------------------------

Grid point forces for node        5 Subcase ID =        1
--------------------------------------------------------------------------------
Force  Element   x-force    y-force    z-force    x-moment   y-moment   z-moment
--------------------------------------------------------------------------------
SPC       --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
Appl.     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
F-MPC     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
Elem.      2 -3.977E-12  0.000E+00  1.407E-11  0.000E+00 -8.583E-14  0.000E+00
Total     --    3.977E-12  0.000E+00 -1.407E-11  0.000E+00  8.583E-14  0.000E+00
--------------------------------------------------------------------------------

Grid point forces for node        6 Subcase ID =        1
--------------------------------------------------------------------------------
Force  Element   x-force    y-force    z-force    x-moment   y-moment   z-moment
--------------------------------------------------------------------------------
SPC       --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
Appl.     --    0.000E+00  0.000E+00  1.000E+03  0.000E+00  0.000E+00  0.000E+00
F-MPC     --    0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00  0.000E+00
Elem.      2    2.493E-12  0.000E+00 -1.000E+03  0.000E+00 -3.411E-14  0.000E+00
Total     --   -2.493E-12  0.000E+00 -9.891E-12  0.000E+00  3.411E-14  0.000E+00
--------------------------------------------------------------------------------
```

Figure 1827:

From the menu bar, click **Post** > **Free Body**.

The FBD Forces and Resultant Force and Moment utilities use element and node set definitions to define what information to extract and sum from the GPFORCE table. This information is then used to produce free bodies and/or resultant force and moments.

FBD Forces

The FBD Forces utility uses an element set to define the values to extract from the GPFORCE table.

The FBD Forces utility uses an element set to define which values to extract from the GPFORCE table. This serves several purposes:

- Any nodes in the element set that have Applied loads will have those values extracted.
- Any nodes in the element set that have SPC loads will have those values extracted.
- Any nodes in the element set that have MPC loads will have those values extracted.
- Any nodes in the element set attached to one or more elements not in the element set will have the appropriate element forces extracted.

The following example of FBD-Forces extraction uses element 1 from the same 2-element image shown at the beginning of this topic.

Element Set

The element set contains only element 1. Element 1 has nodes 1, 2, 3 and 4.

X-Force

Node 1 is only attached to element 1 and has an SPC constraint. Since element 1 is in the element set, its force contributions are not considered. From the GPFORCE table, the x-force value that is extracted for node 1 is the SPC force 2.121e+03.
Node 2 is only attached to element 1 and has an SPC constraint. Since element 1 is in the element set, its force contributions are not considered. From the GPFORCE table, the x-force value that is extracted for node 2 is the SPC force -2.121e+03.
Node 3 is attached to elements 1 and 2. Since element 1 is in the element set, its force contributions are not considered. Since element 2 is not in the element set, its force contributions will be considered. From the GPFORCE table, the x-force value that is extracted for node 3 is the element 2 force -1.085e+03.
Node 4 is attached to elements 1 and 2. Since element 1 is in the element set, its force contributions are not considered. Since element 2 is not in the element set, its force contributions will be considered. From the GPFORCE table, the x-force value that is extracted for node 4 is the element 2 force 1.085e+03.
These values sum to 0.

Y-Force

All values are zero in this model.

Z-Force

Node 1 is only attached to element 1 and has an SPC constraint. Since element 1 is in the element set, its force contributions are not considered. From the GPFORCE table, the x-force value that is extracted for node 1 is the SPC force -4.332e+02.
Node 2 is only attached to element 1 and has an SPC constraint. Since element 1 is in the element set, its force contributions are not considered. From the GPFORCE table, the x-force value that is extracted for node 2 is the SPC force -5.668e+02.

Node 3 is attached to elements 1 and 2. Since element 1 is in the element set, its force contributions are not considered. Since element 2 is not in the element set, its force contributions will be considered. From the GPFORCE table, the x-force value that is extracted for node 3 is the element 2 force 6.426e+02.
Node 4 is attached to elements 1 and 2. Since element 1 is in the element set, its force contributions are not considered. Since element 2 is not in the element set, its force contributions will be considered. From the GPFORCE table, the x-force value that is extracted for node 4 is the element 2 force 3.574e+02.
These values sum to 0.

X-Moment
All values are zero in this model.

Y-Moment
Node 1 is only attached to element 1 and has an SPC constraint. Since element 1 is in the element set, its moment contributions are not considered. From the GPFORCE table, the y-moment value that is extracted for node 1 is the SPC moment 2.370e+01.
Node 2 is only attached to element 1 and has an SPC constraint. Since element 1 is in the element set, its moment contributions are not considered. From the GPFORCE table, the y-moment value that is extracted for node 2 is the SPC moment 2.277e+01.
Node 3 is attached to elements 1 and 2. Since element 1 is in the element set, its moment contributions are not considered. Since element 2 is not in the element set, its moment contributions will be considered. From the GPFORCE table, the y-moment value that is extracted for node 3 is the element 2 moment -8.871e+00.
Node 4 is attached to elements 1 and 2. Since element 1 is in the element set, its moment contributions are not considered. Since element 2 is not in the element set, its moment contributions will be considered. From the GPFORCE table, the y-moment value that is extracted for node 4 is the element 2 moment -1.024e+01.
Additionally, the cross-product of all the forces about the Y axis need to be considered. Selecting node 1 as the summation node (any node in the model can be selected) and performing rXF (all element edge lengths are 0.166) the following is obtained:

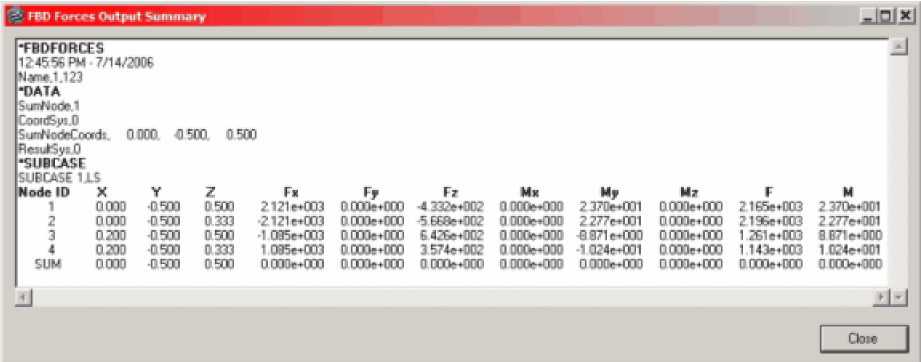
Node 1
No additional rXF contributions since it is the sum point

Node 2
-0.166*Fx + 0*Fz = -0.166*-2.121e+03 + 0*-2.121e+03 = 3.535e+02

Node 3
0*Fx + -0.166*Fz = 0*-1.085e+03 + -0.166*6.426e+02 = -1.071e+02

Node 4
-0.166*Fx + -0.166*Fz = -0.166*1.085e+03 + -0.166*3.547e+02 = -2.400e+02
These values sum to about 0.
Since there are only 4 significant digits in the GPFORCE table, the precision of the calculated moments are compromised. In the actual FBD utilities, the full result and machine precisions are used.

Z-Moment
All values are zero in this model.



FBD Forces Output Summary

*FBDFORCES
12:45:56 PM - 7/14/2006
Name,1,123
*DATA
SumNode,1
CoordSys,0
SumNodeCoords, 0.000, -0.500, 0.500
ResultSys,0
*SUBCASE
SUBCASE 1,LS

Node ID	X	Y	Z	Fx	Fy	Fz	Mx	My	Mz	F	M
1	0.000	-0.500	0.500	2.121e+003	0.000e+000	-4.332e+002	0.000e+000	2.370e+001	0.000e+000	2.165e+003	2.370e+001
2	0.000	-0.500	0.333	-2.121e+003	0.000e+000	-5.668e+002	0.000e+000	2.277e+001	0.000e+000	2.196e+003	2.277e+001
3	0.200	-0.500	0.500	-1.085e+003	0.000e+000	6.426e+002	0.000e+000	-8.871e+000	0.000e+000	1.261e+003	8.871e+000
4	0.200	-0.500	0.333	1.085e+003	0.000e+000	3.574e+002	0.000e+000	-1.024e+001	0.000e+000	1.143e+003	1.024e+001
SUM	0.000	-0.500	0.500	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000	0.000e+000

Close

Figure 1828: Output Table from the FBD Forces Utility

Element 1 (element set) with node 1 used as the summation point.

Resultant Force and Moment

The Resultant Force and Moment utility uses an element set and a node set (cross-section definition) define the values to extract from the GPFORCE table.

The cross-section definition serves several purposes.

- All nodes in the node set must be attached to one or more elements in the element set.
- All nodes in the node set that have Applied loads will be extracted and utilized in Resultant Force and Moment calculations.
- All nodes in the node set that have SPC loads will be extracted and utilized in Resultant Force and Moment calculations.
- All nodes in the node set that have MPC loads will be extracted and utilized in Resultant Force and Moment calculations.
- For all nodes in the node set, Element contributions from only those elements which are not a part of the element set of the cross-section definition will be extracted and utilized in the Resultant Force and Moment calculations.

The following example of Resultant Force and Moment extraction uses element 1 and nodes 3 & 4 from the same example as the previous FBD Forces.

Node Set

The node set contains nodes 3 and 4; the element set contains only element 1.

X-Force

Node 3 is attached to elements 1 and 2. Since element 1 is in the element set, its force contributions are not considered. Since element 2 is not in the element set, its force contributions will be considered. From the GPFORCE table, the x-force value that is extracted for node 3 is the element 2 force -1.085e+03.

Node 4 is attached to elements 1 and 2. Since element 1 is in the element set, its force contributions are not considered. Since element 2 is not in the element set, its force contributions will be considered. From the GPFORCE table, the x-force value that is extracted for node 4 is the element 2 force 1.085e+03.

These values sum to 0.

Y-Force

All values are zero in this model.

Z-Force

Node 3 is attached to elements 1 and 2. Since element 1 is in the element set, its force contributions are not considered. Since element 2 is not in the element set, its force contributions will be considered. From the GPFORCE table, the x-force value that is extracted for node 3 is the element 2 force 6.426e+02.

Node 4 is attached to elements 1 and 2. Since element 1 is in the element set, its force contributions are not considered. Since element 2 is not in the element set, its force contributions will be considered. From the GPFORCE table, the x-force value that is extracted for node 4 is the element 2 force 3.574e+02.

These values sum to 1.000e+03.

X-Moment

All values are zero in this model.

Y-Moment

Node 3 is attached to elements 1 and 2. Since element 1 is in the element set, its moment contributions are not considered. Since element 2 is not in the element set, its moment contributions will be considered. From the GPFORCE table, the y-moment value that is extracted for node 3 is the element 2 moment -8.871e+00.

Node 4 is attached to elements 1 and 2. Since element 1 is in the element set, its moment contributions are not considered. Since element 2 is not in the element set, its moment contributions will be considered. From the GPFORCE table, the y-moment value that is extracted for node 4 is the element 2 moment -1.024e+01.

Additionally, the cross-product of all the forces about the Y axis need to be considered. Selecting node 3 as the summation node (any node in the model can be selected) and performing rXF (all element edge lengths are 0.166) the following is obtained:

Node 3

No additional rXF contributions since it is the sum point

Node 4

-0.166*Fx + 0*Fz = -0.166*1.085e+03 + 0*3.547e+02 = -1.808e+02

These values sum to -2.000e+02. Since there are only four significant digits in the GPFORCE table, the precision of the calculated moments are compromised. In the actual FBD utilities, the full result and machine precisions are used.

Z-Moment

All values are zero in this model.

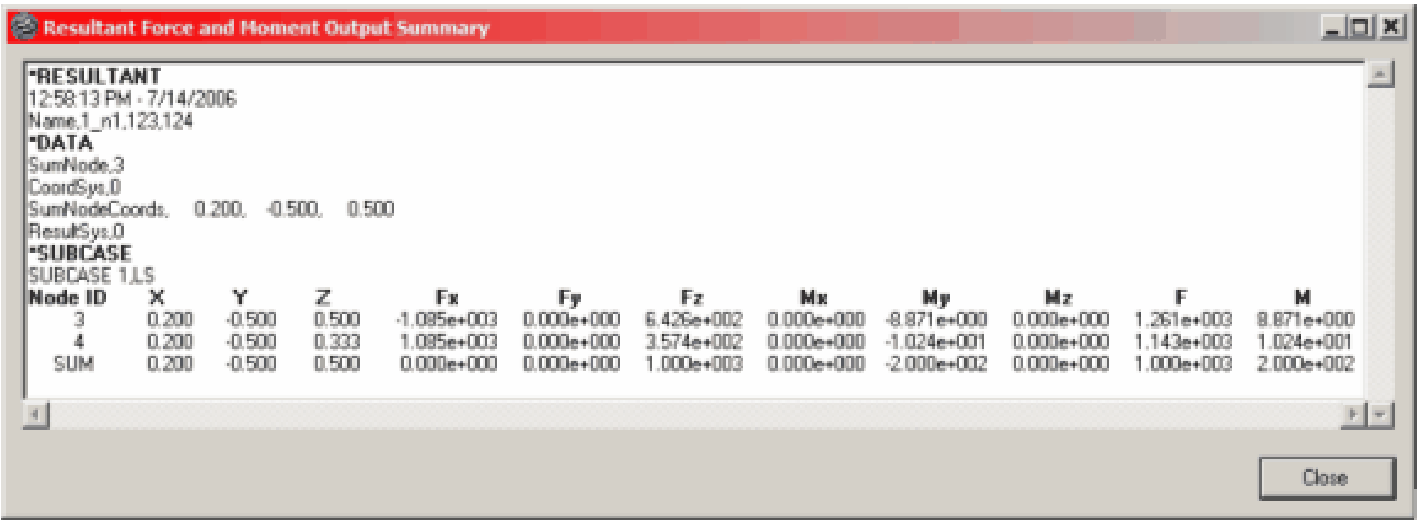


Figure 1829: Output Table from the Resultant Force and Moment Utility

For element 1 (element set) and nodes 3 and 4 (node set) with node 3 used as the summation point.

FBD Displacements

The FBD Displacements utility uses an element set and a node set to define the values to extract from the Displacement table.

The element and node sets serve several purposes:

- All nodes in the node set will have displacement and rotation values extracted.
- The element set is for visualization and breakout modeling purposes only.

Additional Information

- Recommended practice is to output GPFORCE data for the element set(s) of interest only. This procedure reduces the size of the `.op2` file and helps speed up the FBD Forces extractions. Additionally, for Nastran and OptiStruct, consider using STRESS = NONE and/or DISPLACEMENT = NONE options to further reduce the size of the `.op2` file. See the OptiStruct online Reference Guide for more information regarding STRESS and DISPLACEMENT I/O Option cards.
- MPC forces and moments are properly extracted for the following MPC constraint types:

  RBE2
  RJOINT

  RBE3
  RROD

  Rigidlink
  RBAR

- The GPFORCE and Displacement results are extracted from the `.op2` file in float point precision in binary format. This maintains the integrity of the calculations as well as enhances the performance of the utilities.

FBD Solver Interfacing

The Free Body Diagram utilities behave differently depending on the solver that you interface with in HyperMesh.

Abaqus

- Results are supported through the `.odb` output file.
- Grid point force results are requested with the following output requests:

```
*NODE OUTPUT
RF
CFD
*ELEMENT OUTPUT
NFORC
```

- Displacement results are requested with the following output request:

```
*NODE OUTPUT
U
```

- It is recommended practice to output data for only the node/element set(s) of interest. This procedure reduces the size of the solver results file and helps speed up the FBD extractions.
- Abaqus rigid elements, *Rigid bodies, *Coupling constraints, *MPC, *Fastener and *Equations do not export forces and moments. If any of these are attached to the element set of interest, all elements attached to them must be included in the element set to insure the GPF balance is correct. If they are not included, an imbalance will occur. Refer to the Abaqus documentation to determine these elements/bodies. Make sure to check the validity of all GPF results when any of these are present in the model.
- The FBD Export Manager is currently not supported for Abaqus.

ANSYS

- Results are supported through the `.rst` output file.
- Grid point force results are requested with the following output requests:

```
OUTRES,ALL,ALL
```

or

```
OUTRES,NSOL,ALL
OUTRES,RSOL,ALL
```

- Displacement results are requested with the following output requests:

```
OUTRES,ALL,ALL
```

or

`OUTRES,NSOL,ALL`

`OUTRES,RSOL,ALL`

- The FBD Export Manager is currently not supported for ANSYS.

Nastran

- Results are supported through the `.op2` and `.xdb` output files.
- Grid point force results are requested with the GPFORCE output request.
- Displacement results are requested with the DISPLACEMENT output request.
- It is recommended practice to output data for only the node set(s) of interest. This procedure reduces the size of the solver results file and helps speed up the FBD extractions. Consider using STRESS=NONE and STRAIN=NONE to further reduce the size of the results file.
- MPC forces and moments are properly extracted for the following MPC constraint types:

  RBE2

  RBE

  RigidLink

  RJOINT

  RROD

  RBAR

OptiStruct

- Results are supported through the `.op2` output file.
- Grid point force results are requested with the GPFORCE output request.
- Displacement results are requested with the DISPLACEMENT output request.
- It is recommended practice to output data for only the node set(s) of interest. This procedure reduces the size of the solver results file and helps speed up the FBD extractions. Consider using STRESS=NONE and STRAIN=NONE to further reduce the size of the results file. You may consider using the NOMODEL option on the OUTPUT,OP2 output format request.
- MPC forces and moments are properly extracted for the following MPC constraint types:

  RBE2

  RBE3

  RigidLink

  RJOINT

  RROD

  RBAR

H3D Writer

Overview of how to create and manage H3D files.

Create H3D Files from HyperMesh

Using an H3D file, you can save 3D animations from HyperMesh in the `.h3d` format for viewing with the HyperView Player.

In order to enable the option to create H3D files, you must activate the checkbox labeled **launch HV after H3D creation** in the Options panel.

HyperView Player is an Internet browser plug-in for visualizing 3D Computer Aided Engineering (CAE) models and results. Using product data in Altair's compact `.h3d` format allows you to incorporate animated images in an HTML document for presentation or engineering reports. Simulation results can be sent by e-mail or placed on the web for others to open and review.

HyperView Player is available as a free download on Altair's Web site at http://www.altair.com.

1. Select one of the following panels: Contour, Deformed, Hidden Line, Transient, or Geom Cleanup.
2. To control the display attributes for your model, specify your desired display attributes using visual options or the visual panel.
3. Click **Hyper3d** or **H3D** > **HV**.

   Two files are created. One is an H3D file, using `anim#.h3d` as the file name. The symbol # is automatically assigned to the H3D file. The other is a sample HTML file including an <EMBED> statement for the corresponding H3D file.

   H3D > **HV** loads the newly created H3D file into HyperView. You can define this option in the Options panel under modeling.
4. Review the model in a web browser.
   - Double-click the HTML file to launch a browser.
   - Click **H3D** to activate the standalone HyperView Player.

   You can customize the external HTML template, `h3d_template.html`, located in the `altair/hm/html` directory, to suit your needs.

> **Note:** In the HyperMesh Geom Cleanup panel, the Hyper3D button is displayed when you select the shaded option in the Visual options subpanel.

Embed HyperView Player Object in HTML Documentation

Arguments and example for embedding statements in an HTML document in order to view a HyperView Player graphic object.

Since the H3D file created from HyperView and HyperMesh includes scene information, the arguments in the old statements for model readers and result readers are no longer needed. The HTML statements have been simplified in this release. However, the HTML files created for HyperView Player 3.1 are still supported.

HyperView Player only supports H3D direct readers. You can create an H3D file using HyperView, HyperMesh, HyperMesh result translators, such as hmnast, hmnasto2, hmradioss, hmpam, hmansys, and hmabaqus, and OptiStruct.

> **Note:** You may need to modify your HTML files created for HyperView Player 3.1 if you were using direct readers other than `h3d.dll`, such as `adams.dll`, `gfile.dll`, `lsdyna.dll`, and `madymo.dll`, since those readers are no longer supported in HyperView Player.

To embed a HyperView Player object, the `<EMBED>` statement in HTML is used.

All arguments are case insensitive.

General Arguments for EMBED Statements	Description
`type`	Application/x-h3d
`width/height`	Measured in pixels
`SRC="URL"`	The location of the plug-in data file as indicated by its URL.

```
<EMBED         type="application/x-h3d"
            width=450 height=400
                  src="HTTP:\\www.altair.com\h3d\bumper.h3d">

<EMBED         type="application/x-h3d"
                  width=450 height=400
                  src="../bumper.h3d">
```

Figure 1830: Embedded Statement Example

Share H3D Files

Share H3D files in an HTML file or via e-mail.

Share H3D Files Using HTML File

You can use HyperView Player to share information by embedding it in an HTML file.

You can use either a relative path or a standard Uniform Resource Locator (URL) to specify the path for the H3D file in the `<EMBED>` statement. There are three different ways to define file transfer protocol: FILE, HTTP, and FTP.

Select a protocol for file transfer using the files, `anim1.html` and `anim1.h3d`, as examples.

Option	Description
Embedded mode	**FILE://** An absolute path is required for `File://` and the H3D file must reside in the specified path. When you distribute the files, you may need to modify the HTML file for the path.

```
<EMBED type="application/x-h3d" width=450 height=400
SRC="File://c:\Altair\demos\hvp\anim1.h3d">
```

Figure 1831: File:// Example

HTTP://

Others can access your public web area on the Internet. When sharing an H3D file, you can place the `anim1.html` and `anim1.h3d` files in your public HTML directory and send the link to others in the company by email. The link could be, for example, `http://www:8080/~John/anim1.html`. By doing this, you do not need to distribute the H3D file and can guarantee the path is working correctly. If you have HyperView Player installed and you click the link in the email, the model will be displayed.

```
<EMBED type="application/x-h3d" width=450 height=400
SRC="http://www:8080/~John/anim1.h3d>
```

Figure 1832: HTTP:// Example

FTP://

You can place the `anim1.html` and `anim1.h3d` files on your FTP site. If you have HyperView Player installed and you click `ftp://ftp.altair.com/pub/outgoing/HVP/anim1.html`, the model is displayed.

```
<EMBED type="application/x-h3d" width=450 height=400
SRC="FTP://ftp.altair.com/pub/outgoing/HVP/anim1.html>
```

Figure 1833: FTP:// Example

Option	Description
Relative path	Using a relative path allows you to distribute an HTML file easily. You can create a folder with the necessary HTML and h3d files for distribution.

```
<EMBED type="application/x-h3d" width=450 height=400
SRC="../anim/.h3d>
```

Figure 1834: Relative Path Example

Option	Description
Full page mode	In an HTML file, you can easily hyperlink a string to an H3D file. When you click the hyperlink, the plug-in loads a model in full-page mode.

Share H3D Files using E-Mail

You can attach an H3D file to an e-mail message.

If the recipient is using a PC, he or she can click the attachment and the H3D file will load automatically. You can also save the H3D file and at a later time double-click the file to invoke the standalone version of HyperView Player. Another available option is to drag the H3D file and drop it into a browser to view the model in a full window.

1. Open your e-mail.
2. Attach the H3D file to your e-mail.
3. Click **Send**.

H3D FAQ

Learn more about H3D writer with these FAQs.

How can I view a model with shaded display in HyperView Player?
The model display settings are recorded during H3D creation. Set up the desired model display, with such options as mesh lines and feature lines, in the Vis panel under the Performance Graphics mode in HyperMesh.

Why doesn't the legend setting in HyperView Player reflect the setting in HyperMesh?
HyperView Player only supports default settings for the legend display from HyperMesh. The specified Max and Min values in HyperMesh are not recorded in the H3D file of this version.

Can I view complex result animation in HyperView Player?
No, the H3D writer does not support complex result data.

What types of element are not supported in the H3D file?
Second order elements, rigidlinks, and RBE3s are not supported in this version. The mid-side node of a second order element is ignored when it is read into HyperView Player.

How can I change the window size for viewing HyperView Player in the browser?
You can modify the width and height values in the <EMBED> statement in the HTML file. For example, width = 600, height = 600.