Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-4605: Defining LS-DYNA Model and Load Data, Controls, and Output

HM-4605: Defining LS-DYNA Model and Load Data, Controls, and Output

In this tutorial, you will learn to:

- View LS-DYNA keywords in HyperMesh as they will appear in the exported LS-DYNA input file
- Understand part, material, and section creation and element organization
- Create sets
- Create velocities
- Understand the relation of LS-DYNA entity type to HyperMesh element and load configurations
- Create nodal single point constraints
- Create contacts with set segment ID
- Define output and termination
- Export models to LS-DYNA formatted input files

Tools/Utilities

The following tools/utilities set the foundation for settings up an LS-DYNA input deck with HyperMesh:

- LS-DYNA FE input translator
- FE output template
- LS-DYNA **Utility** menu
- LS-DYNA User Profile

Model Files

This tutorial uses the `head_start.hm`, `head_2.hm`, and `head_3.hm` files, which can be found in `<hm.zip>/interfaces/lsdyna/`. Copy the file(s) from this directory to your working directory.

Exercises

This tutorial contains the following exercises:

Exercise 1: Define Model Data for the Head and A-Pillar Impact Analysis

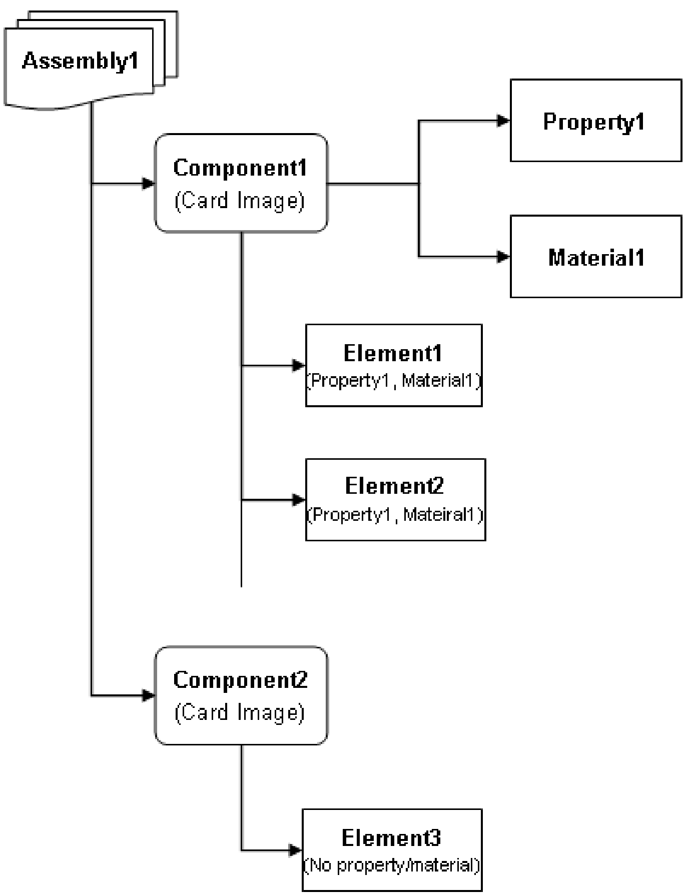
Element property and material assignment rules are based on the current user profile (solver interface).

Element Property and Material Assignment Rules for LS-DYNA Solver

- Components have no card images; typically "part" card images.
- Properties and materials are only assigned to components; properties and materials are not directly assigned to elements. Use the **Entity Editor** to assign properties and materials to components.
- Elements are assigned through the property and material assigned to the component in which they are organized into.
- If a component is not assigned to a property or material, then all of the elements within that component will not have a property or material assignment.

Property/Material Assignment Schematic

The figure below shows how the keywords *PART, *ELEMENT, *MAT, and *SECTION relate to each other.



Relation of *PART, *ELEMENT, *MAT, and *SECTION to Each Other

*ELEMENT EID PID

*PART PID SID MID

*SECTION SID

*MAT MID

A *PART shares attributes such as section properties (*SECTION) and a material model (*MAT). A group of elements (*ELEMENT) sharing common attributes generally share a common part ID (PID). The figure below shows how the keywords *PART, *ELEMENT, *MAT and *SECTION relate to each other. A unique PID assigns a material ID (MID) and a section ID (SID) to an element.

The table below shows how the *ELEMENT, *PART, *SECTION, and *MAT keywords are organized in HyperMesh.

*ELEMENT EID PID	Elements are organized into a component collector
*PART PID SID MID	Component collector's card image
*SECTION SID	Property collector with a property card image. Assign a property to a *PART by pointing to the property collector in the component collector's card image.
*MAT MID	Material collector with a material card image. Assign the material to the *PART by associating the material collector to the component collector.

Create and modify component, property, and material collectors in the **Collectors** panel, **Model** or **Solver** browser, and **Entity Editor**.

View LS-DYNA Keywords in HyperMesh

Use a HyperMesh card image to view the keywords and data lines for defined LS-DYNA entities as interpreted by the loaded template. The keywords and data lines appear in the exported LS-DYNA input file as you see them in the card images. Additionally, for some card images, you can define and edit various parameters and data items for the corresponding LS-DYNA keyword.

View card images using the **Card Editor**, which can be accessed by doing one of the following:

- From the menu bar, click ***Tools*** > ***Card Edit***.
- On the **Collectors** toolbar, click .

- In the **Model** browser or **Solver** browser, right-click on an entity and select ***Card Edit*** from the context menu.

Create *MAT

In HyperMesh, a *MAT is a material collector with a card image. To relate it to a *PART, the material collector is associated to a component collector.

Create a material collector by doing one of the following:

- In the **Model** browser or **Solver** browser, right-click and select ***Create*** > ***Material*** from the context menu.
- From the menu bar, click ***Materials*** > ***Create***.
- On the **Collectors** toolbar, click .

Update a Component's Material

Change the material assigned to a component in the **Component Collectors** panel, **update** subpanel, or in the **Entity Editor**.

- Access the **update** subpanel by clicking on the **Collectors** toolbar.
- Access the **Entity Editor** by left-clicking on a component in the **Model** or **Solver** browser.

Material Table

Use the **Material Table** to:

- View existing materials in an interactive tabular list
- Create materials
- Merge identical materials
- Search for duplicate materials
- Change the properties of existing materials

The Material Table can be accessed in the LS-DYNA **Utility** menu, **DYNA Tools** page, under **Parts**.

Create *SECTION

In HyperMesh, *SECTION is a property collector with a card image.

Create a property collector by doing one of the following:

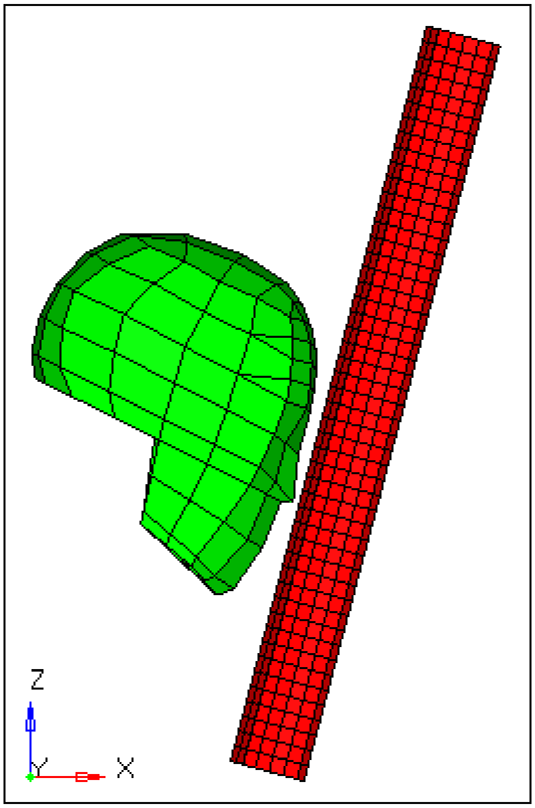
- In the **Model** browser or **Solver** browser, right-click and select ***Create*** > ***Property*** from the context menu.
- From the menu bar, click ***Properties*** > ***Create*** > ***Properties***.

- On the **Collectors** toolbar, click .

Exercise Objective and Tasks

The purpose of this exercise is to help you become familiar with defining LS-DYNA materials, sections, and parts in HyperMesh.

In this exercise you will set up model data for a LS-DYNA analysis of a hybrid III dummy head impacting an A-pillar. The head and A-pillar model is depicted below.



Head and A-pillar model

This exercise contains the following tasks:

- Define the material *MAT_ELASTIC for the A-pillar part and head part
- Define *SECTION_SHELL for the A-pillar
- Define *SECTION_SOLID for the head
- Define *PART for the A-pillar and the head

Step 1: Load the LS-DYNA user profile

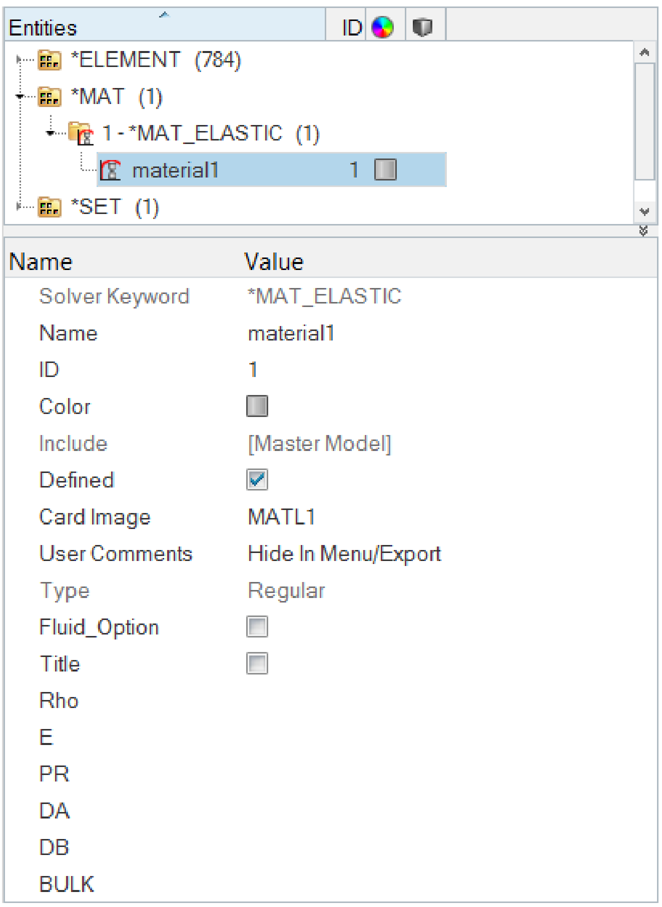
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

Step 2: Retrieve the HyperMesh file

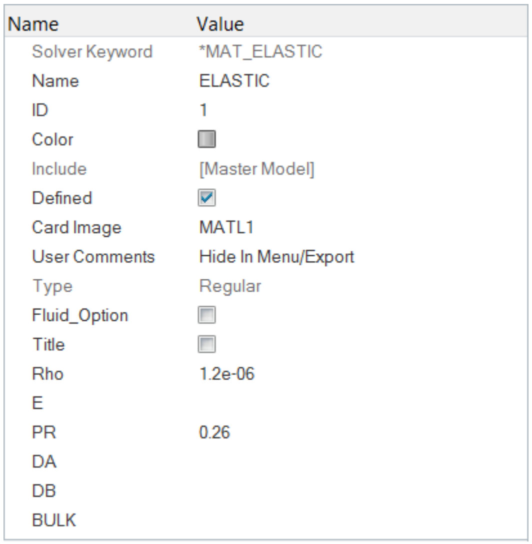
1. Open a model file by clicking ***File*** > ***Open*** > ***Model*** from the menu bar, or clicking  on the **Standard** toolbar.
2. In the **Open Model** dialog, open the `head_start.hm` file. The model appears in the graphics area.

Step 3: Define the material *MAT_ELASTIC for the A-pillar and head

1. In the **Solver** browser, right-click and select ***Create*** > ****MAT*** > ***MAT(1-50)*** > ***1 - *MAT_ELASTIC*** from the context menu. HyperMesh creates and opens a new material in the **Entity Editor**.



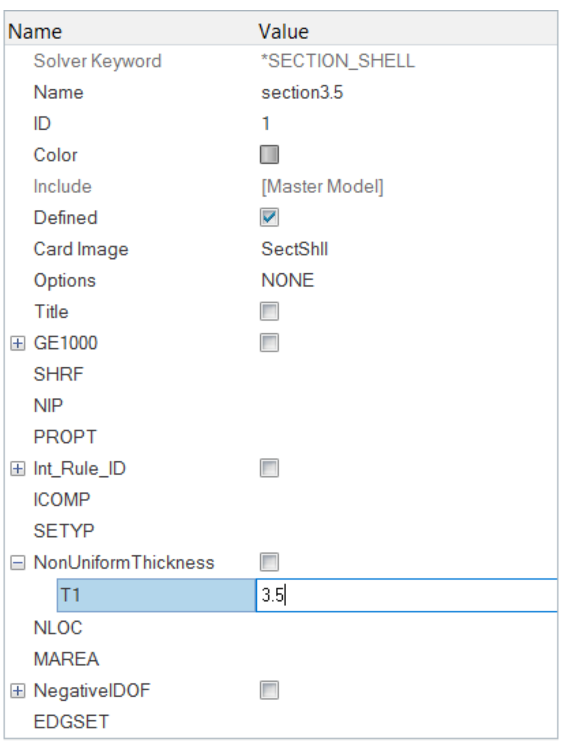
2. For **Name**, enter `ELASTIC`.
3. For **Rho** (Density), enter `1.2E-6`.
4. For **E** (Young's modulus), enter `210`.
5. For **PR** (Poisson's ratio), enter `0.26`.



Name	Value
Solver Keyword	*MAT_ELASTIC
Name	ELASTIC
ID	1
Color	
Include	[Master Model]
Defined	
Card Image	MATL1
User Comments	Hide In Menu/Export
Type	Regular
Fluid_Option	
Title	
Rho	1.2e-06
E	
PR	0.26
DA	
DB	
BULK	

Step 4: Define property (*SECTION_SHELL) with a thickness of 3.5 mm for the A-pillar

1. In the **Solver** browser, right-click and select ***Create*** > ****SECTION*** > ****SECTION_SHELL*** from the context menu. HyperMesh creates and opens a new property in the **Entity Editor**.
2. For **Name**, enter `section3.5`.
3. For **Card Image**, select ***SectShll***
4. Expand ***NonUniformThickness***, and enter `3.5` for **T1**.



Name	Value
Solver Keyword	*SECTION_SHELL
Name	section3.5
ID	1
Color	
Include	[Master Model]
Defined	
Card Image	SectShll
Options	NONE
Title	
GE1000	
SHRF	
NIP	
PROPT	
Int_Rule_ID	
ICOMP	
SETYP	
NonUniformThickness	
T1	3.5
NLOC	
MAREA	
NegativeIDOF	
EDGSET	

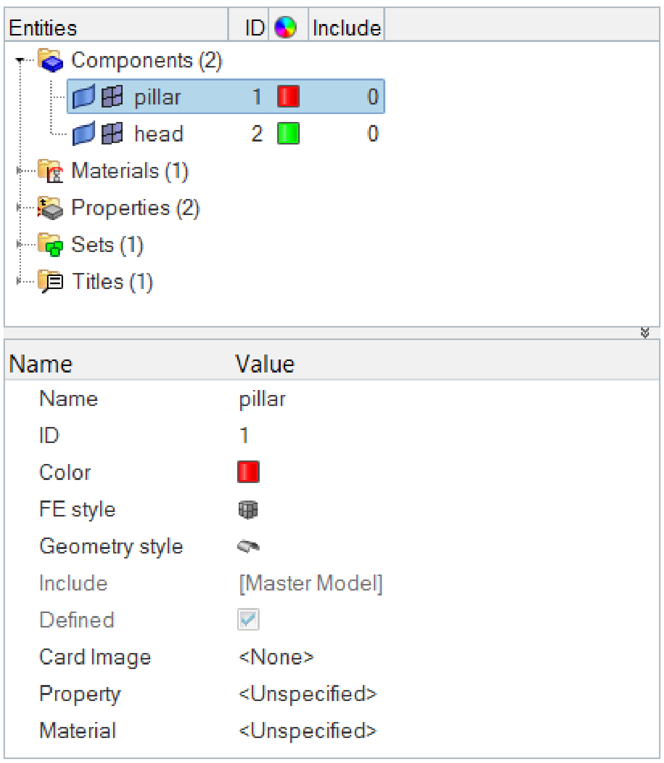
Step 5: Define *SECTION_SOLID for the head

1. In the **Solver** browser, right-click and select ***Create*** > ****SECTION*** > ****SECTION_SOLID*** from the context menu. HyperMesh creates and opens a new property in the **Entity Editor**.
2. For **Name**, enter `solid`.

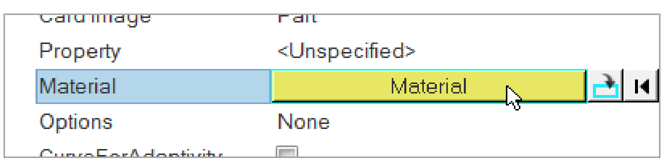
Step 6: Define *PART for the A-pillar

MAT_ELASTIC is the material collector named "**ELASTIC**"; ***SECTION_SHELL** is the property collector named "**section3.5**".

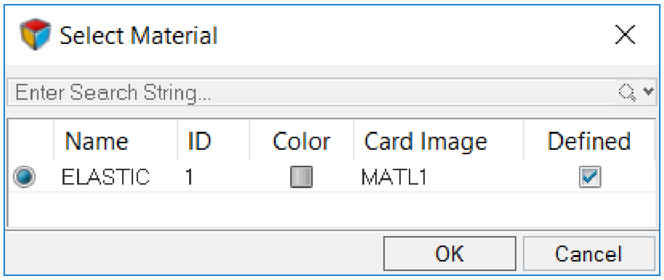
1. In the **Model** browser, **Component** folder, click ***pillar***. The **Entity Editor** opens, and displays the component's corresponding data.



2. Set **Card image** to ***Part***.
3. For **Material**, click ***Unspecified*** >> ***Material***.



4. In the **Select Material** dialog, select ***ELASTIC*** and then click ***OK***. HyperMesh assigns the material **Elastic** to the component **pillar**.

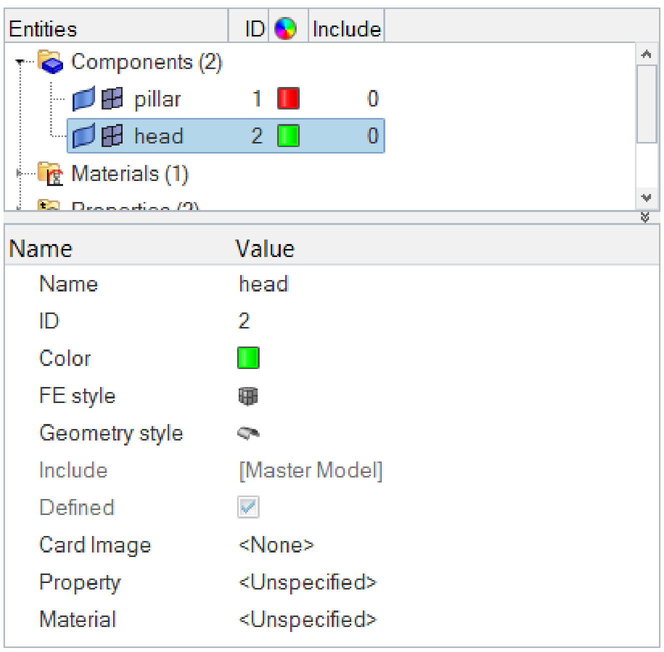


5. For **Property**, click ***Unspecified*** >> ***Property***.
6. In the **Select Property** dialog, select ***section3.5*** and then click ***OK***. HyperMesh assigns the property **section3.5** to the component **pillar**.

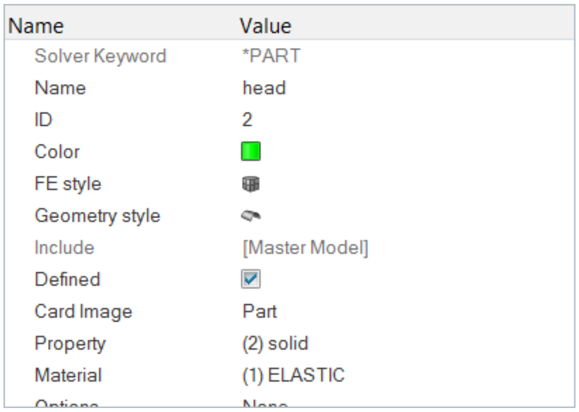
Step 7: Define *PART for the head

***MAT_ELASTIC** is the material collector named "**ELASTIC**"; ***SECTION_SOLID** is the property collector named "**solid**".

1. In the **Model** browser, **Component** folder, click ***head***. The **Entity Editor** opens, and displays the component's corresponding data.



2. Set **Card Image** to ***Part***.
3. Assign the material **ELASTIC** to the component.
4. Assign the property **solid** to the component.



Step 8 (Optional): Save your work

The exercise is complete. Save your work as a HyperMesh file named `head_2.hm`.

Exercise 2: Define Boundary Conditions and Loads for the Head and A-Pillar Impact Analysis

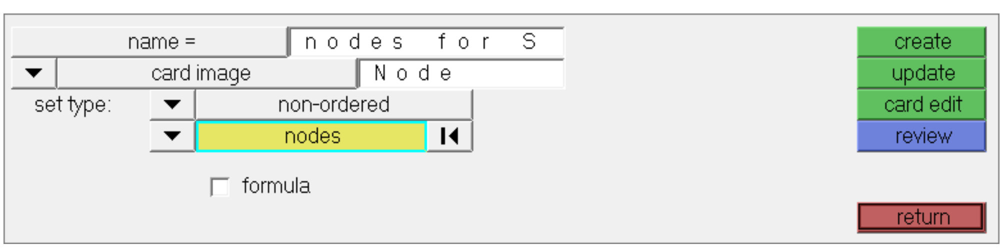
*INITIAL_VELOCITY_(Option)

LS-DYNA keywords used for defining initial velocity.

LS-DYNA keyword	Velocity applied to …	Setup in HyperMesh
*INITIAL_VELOCITY	set of nodes, *SET_NODE_LIST	Entity set of nodes, load collector with **InitialVel** card image
*INITIAL_VELOCITY_GENERATION	one *PART or set of parts, *SET_PART_LIST	Entity set of comps, load collector with **InitialVel** card image
*INITIAL_VELOCITY_NODE	individual nodes	Created from **Velocity** panel, organized in load collector with no card image

*SET

Graphically view a set's contents in the **Entity Sets** panel using the **review** function.



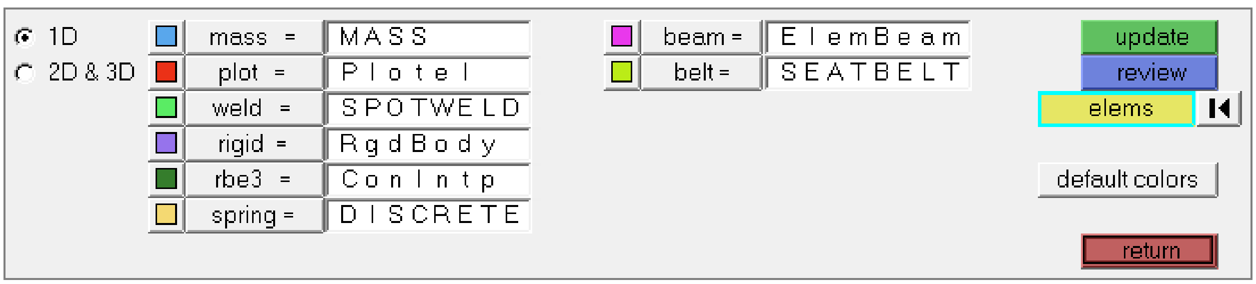
HyperMesh Entity Configurations and Types

HyperMesh elements and loads have two identifiers:

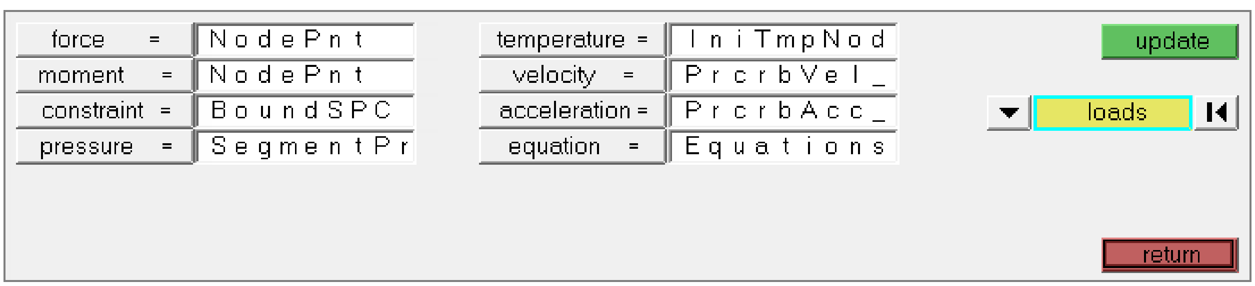
- **Configuration** - a HyperMesh core feature.
- **Type** - defined by the loaded FE output template. A configuration can support multiple types. Before creating elements or loads, select the desired type from either the Elem Types panel.

Only use the **Load Types** subpanel to directly create loads on nodes or elements. For all other cases, define loads by creating a load collector with a card image. For example, *INITIAL_VELOCITY_NODE (applied directly to nodes) is created from the **Velocities** panel, while *INITIAL_VELOCITY (applied to nodes in a set) is a load collector with the **InitialVel** card image.

View a list of element and load configurations in the **Elem Types** panel and the **Load Types** panel, respectively.



elem types panel



load types panel

Some element configurations are rigid and quad4. When you load a `dyna.key` template, the following types of the rigid configuration are available: RgdBody, ConNode, and GenWeld (*CONSTRAINED_NODAL_RIGID_BODY, *CONSTRAINED_NODE_SET, and *CONSTRAINED_GENERALIZED_WELD_SPOT).

Similarly, some load configurations are force and pressure. Types of the pressure configuration are ShellPres and SegmentPre (*LOAD_SHELL_ELEMENT and *LOAD_SEGMENT).

Most element and load configurations have their own panels. For example, rigids are created with the **Rigids** panel and constraints are created with the **Constraints** panel.

***BOUNDARY_SPC_(Option)**

LS-DYNA keywords used for defining nodal single point constraints.

LS-DYNA keyword	Constraint applied to …	Setup in HyperMesh
*BOUNDARY_SPC_NODE	individual nodes	These are constraints created from the **Constraints** panel and organized into a load collector with no card image.
*BOUNDARY_SPC_SET	a set of nodes *SET_NODE_LIST	This is an entity set of nodes referenced in a load collector's **BoundSpcSet** card image.

***CONTACT and *SET_SEGMENT**

A LS-DYNA contact is a HyperMesh group. Select groups when you want to manipulate a *CONTACT, such as delete, renumber, or display it off.

LS-DYNA Contact Master and Slave Types

LS-DYNA has multiple contact master and slave types from which to choose.

***SET_SEGMENT and Contactsurfs Panel**

Create a *SET_SEGMENT by right-clicking in the **Solver** browser and selecting ***Create*** > ****SET*** > ****SET_SEGMENT*** from the context menu. Additionally, add and remove elements from an existing *SET_SEGMENT and adjust the normal of segments without adjusting the normal of elements with the **Contactsurfs** panel.

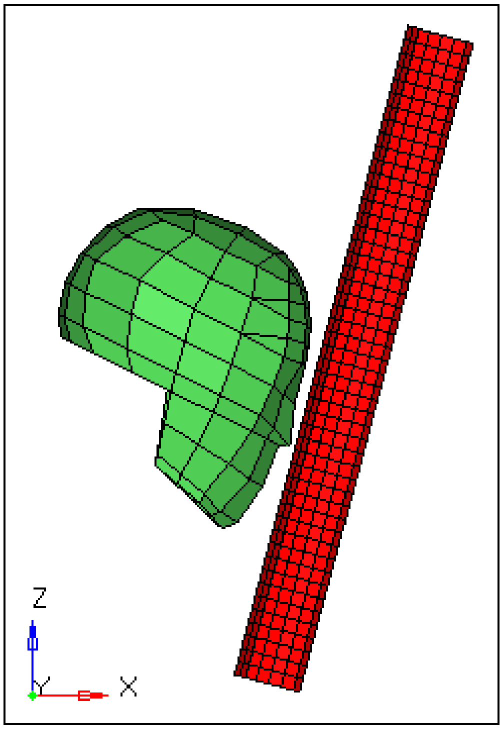
The graphical representation of a contactsurf is pyramids, one pyramid for each segment. The orientation of a pyramid represents the normal orientation of the segment.

By default, the orientation of a pyramid is the same as the normal of the element underneath.

Exercise Objective and Tasks

The purpose of this exercise is to help you become familiar with defining LS-DYNA boundary conditions, loads, and contacts in HyperMesh.

In this exercise you will set up the boundary conditions and load data for a LS-DYNA analysis of a hybrid III dummy head impacting an A-pillar. The head and A-pillar model is depicted below.



Head and A-pillar model

This exercise contains the following tasks:

- Define velocity on all nodes of the head with *INITIAL_VELOCITY
- Constrain the pillar's end nodes in all six degrees of freedom with *BOUNDARY_SPC_NODE
- Define a contact between the head and A-pillar with *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE

Step 1: Load the LS-DYNA user profile

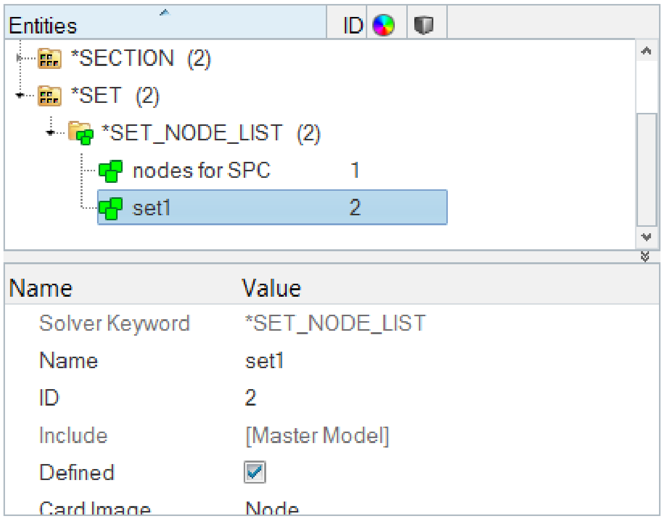
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

Step 2: Retrieve the HyperMesh file

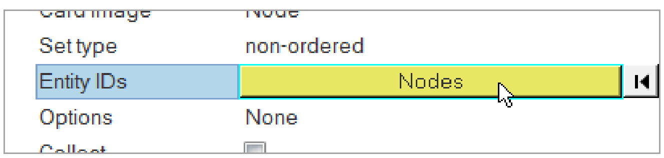
1. Open a model file by clicking ***File*** > ***Open*** > ***Model*** from the menu bar, or click  on the **Standard** toolbar.
2. In the **Open Model** dialog, open the `head_2.hm` file. The model appears in the graphics area.
3. Observe the model using various visual options available in HyperMesh (rotation, zooming, etc.).

Step 3: Create a node set, *SET_NODE_LIST, containing all the nodes in the head component

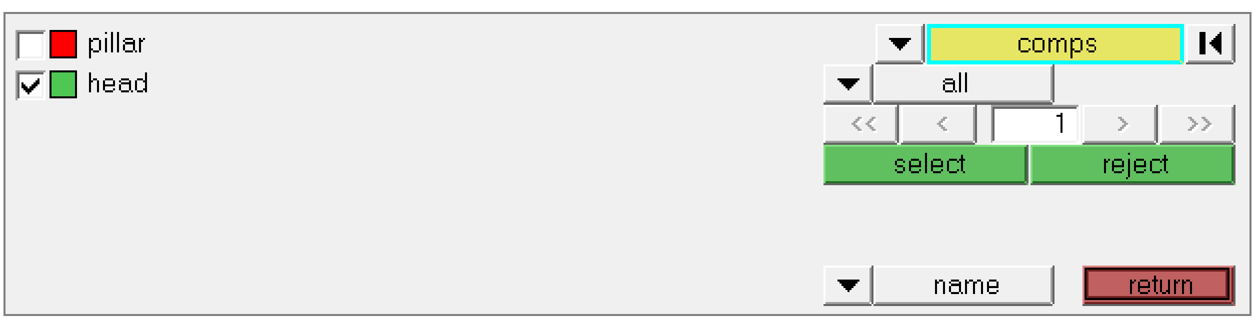
1. In the **Solver** browser, right-click and select ***Create*** > ****SET*** > ****SET_NODE*** > ****SET_NODE_LIST*** from the context menu. HyperMesh creates and opens a new set in the **Entity Editor**.



2. For **Name**, enter `Vel_Nodes`.
3. For **Entity IDs**, click ***0 Nodes*** >> ***Nodes***.



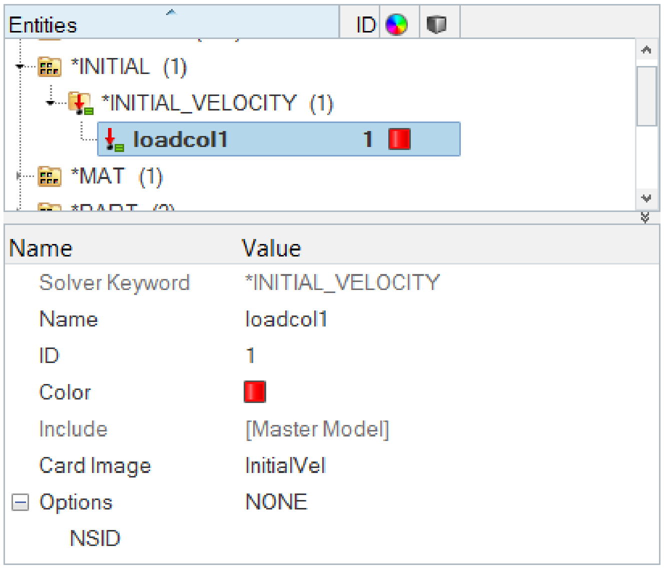
4. In the panel area, click ***nodes*** >> ***by collector***.
5. Select the component, ***head***.



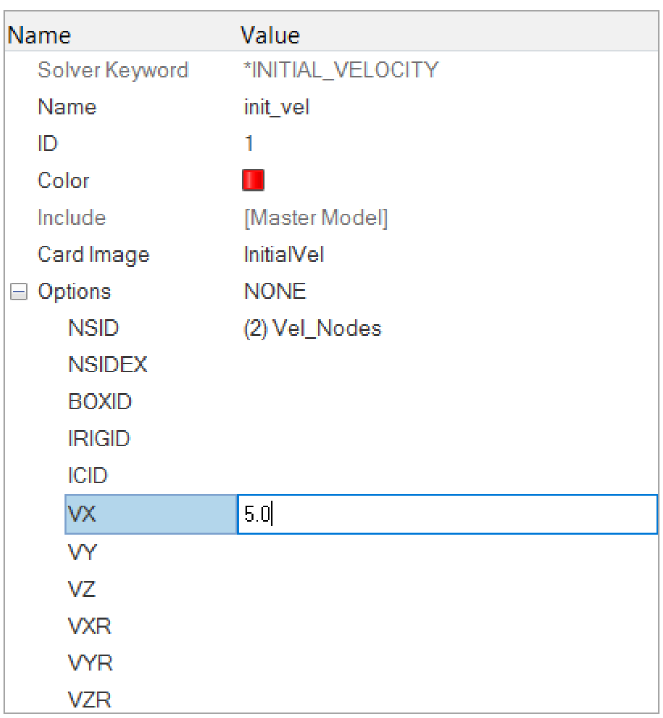
6. Click ***select***.
7. Click ***proceed***.

Step 4: Define the velocity

1. In the **Solver** browser, right-click and select ***Create*** > ****INITIAL*** >****INITIAL_VELOCITY*** from the context menu. HyperMesh creates and opens a new load collector in the **Entity Editor**.



2. For **Name**, enter `init_vel`.
3. Click ***NSID*** (Node Set ID), and then click ***Set***.
4. In the **Select Set** dialog, select ***Vel_Nodes*** and then click ***OK***.
5. For ***VX*** (x-component of mass center of velocity), enter `5`.



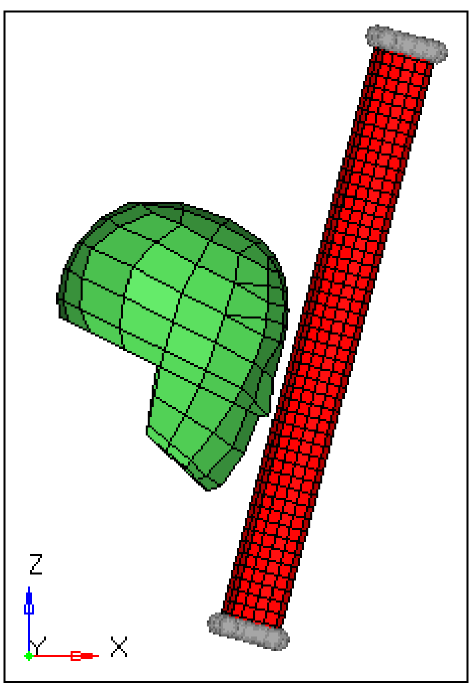
Step 5: Create a load collector for the constraints to be created

1. In the **Model** browser, right-click and select ***Create*** > ***Load Collector*** from the context menu. HyperMesh creates and opens a new load collector in the **Entity Editor**.
2. For **Name**, enter `SPC`.
3. Optional: Click the ***Color*** icon, and select a color to display the load collector.
4. Set **Card Image** to ***<None>***.

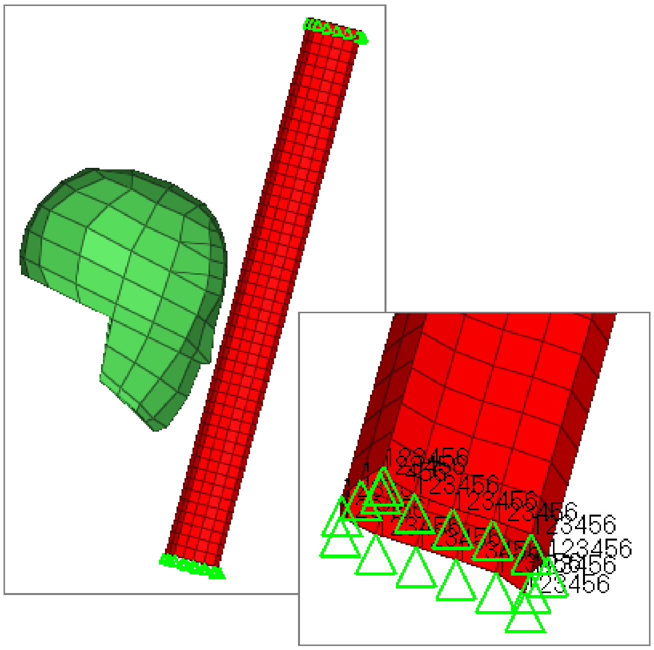
Step 6: Create constraints on the pillar's end nodes

1. In the **Solver** browser, right-click and select ***Create*** > ****BOUNDARY*** > ****BOUNDARY_SPC_NODE*** from the context menu. The **Constraints** panel opens.
2. In the panel area, set the entity selector to ***nodes***.
3. Click ***nodes*** >> ***by sets***.
4. Select the entity set, ***nodes for SPC***.

5. Click ***select***. HyperMesh selects the nodes on both ends of the pillar.



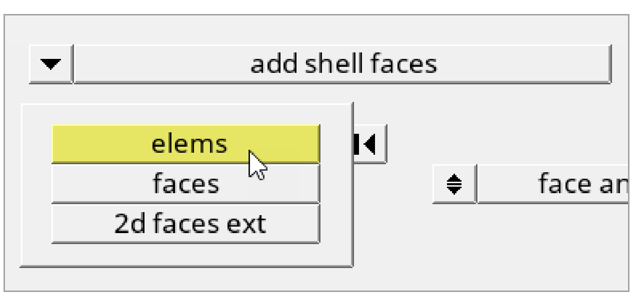
6. Verify that all six **dof** (degree of freedom) checkboxes are selected.
7. Verify that the **load type** is set to ***BoundSPC***.
8. Select the ***label constraints*** checkbox to display each constraint's label in the graphics area.
9. Click ***create***.



10. Click ***return*** to close the panel.

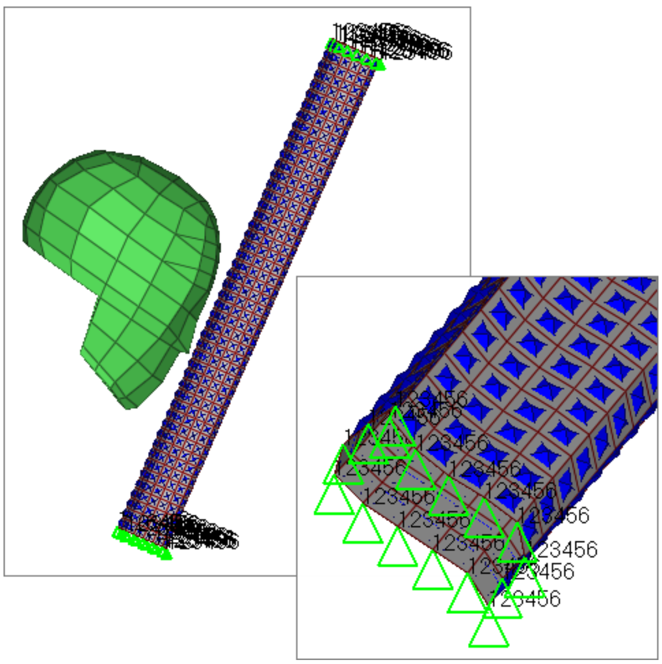
Step 7: Define a *SET_SEGMENT for the slave entities, the A-pillar elements

1. In the **Solver** browser, right-click and select ***Create*** > ****SET*** > ****SET_SEGMENT*** > ****SET_SEGMENT*** from the context menu. HyperMesh creates and opens a new contactsurf in the **Entity Editor**.
2. For **Name**, enter `pillar_slave`.
3. Optional. Click the ***Color*** icon, and select a color to display the contactsurf.
4. For ***Elements***, click ***0 Elements*** >> ***Elements***.
5. In the panel area, set the second switch to ***elems***.



6. Click ***elems*** >> ***by collector***.
7. Select the component, ***pillar***.

8. Click ***select***.
9. Click ***add***.
10. Review the contactsurf to make sure that its pyramids are pointing out of the pillar.



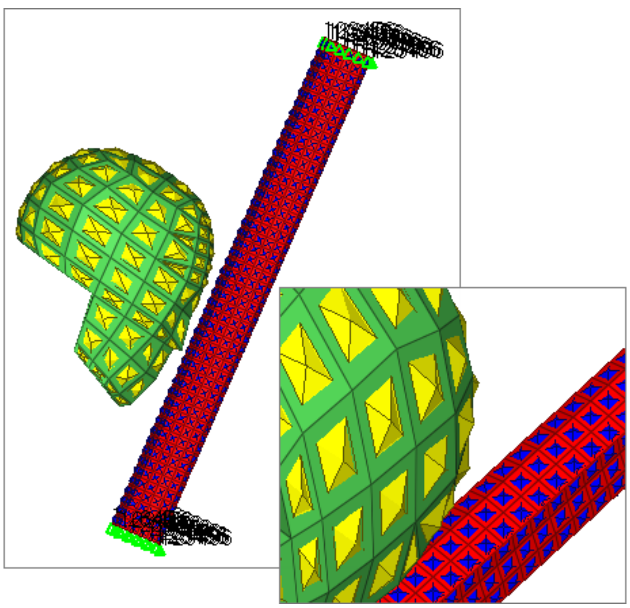
11. Click ***return*** to close the panel.

Step 8: Define a *SET_SEGMENT for the master entities, the head elements

1. In the **Solver** browser, right-click and select ***Create*** > ****SET*** > ****SET_SEGMENT*** > ****SET_SEGMENT*** from the context menu. HyperMesh creates and opens a new contactsurf in the **Entity Editor**.
2. For **Name**, enter `head_master`.
3. Optional. Click the ***Color*** icon, and select a color to display the contactsurf.
4. For ***Elements***, click ***0 Elements*** >> ***Elements***.
5. In the panel area, set the first switch to ***add solid faces***.

6. Set the second switch to ***elems***.
7. Click ***elems*** >> ***by collector***.
8. Select the component, ***head***.

9. Click ***select***.
10. Using the ***nodes*** selector, select three nodes that belong to the same face of a solid element.
11. In the **face angle** field, enter `30`.
12. Click ***add***.
13. Review the contactsurf to make sure that its pyramids are pointing out of the head.



14. Click ***return*** to close the panel.

Step 9: Create a HyperMesh group with the SurfaceToSurface card image

1. In the **Solver** browser, right-click and select ***Create*** > ****CONTACT*** > ****CONTACT_SURFACE_TO_SURFACE*** > ****CONTACT_SURFACE_TO_SURFACE*** from the context menu. HyperMesh creates and opens a new group in the **Entity Editor**.
2. For ***Name***, enter `contact`.

Step 10: Add the slave and master contactsurfs to the HyperMesh group

In this step, the **Entity Editor** should still be open for the group, **contact**.

1. In the **Entity Editor**, click ***MSID*** and set the entity selector to ***Contactsurfs***.
2. Click ***Contactsurfs***.
3. In the **Select Contactsurfs** dialog, select ***head_master*** and then click ***OK***.
4. Click ***SSID***, and set the entity selector to ***Contactsurfs***.
5. Click ***Contactsurfs***.
6. In the **Select Contactsurfs** dialog, select ***pillar_slave*** and then click ***OK***.

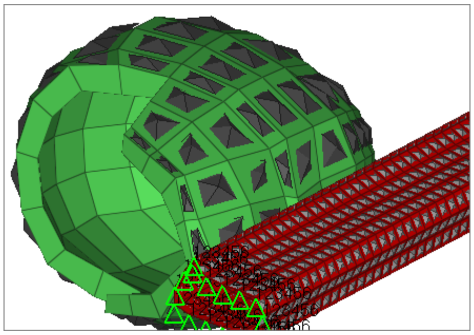
Step 11: Edit the group's card image to define the AUTOMATIC option

In this step, the **Entity Editor** should still be open for the group, **contact**.

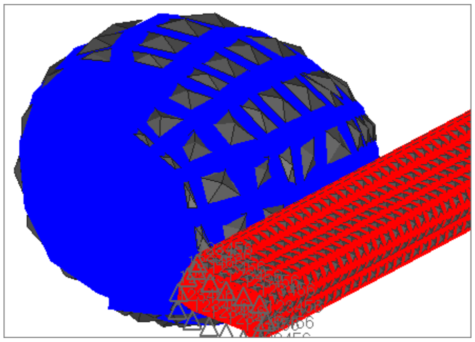
1. In the **Entity Editor**, for the first ***Options*** parameter, select ***Automatic***.

Step 12: Review the group's master and slave surfaces

1. Open the **Interfaces** panel by clicking ***interfaces*** from the ***Analysis*** page.
2. Go to the **add** subpanel.
3. Click ***name=***, and select ***contact***.



4. Click ***review***. HyperMesh temporarily displays the master and slave entities in blue and red, respectively.



5. Click ***return*** to close the panel.

Step 13 (Optional): Save your work

The exercise is complete. Save your work as a HyperMesh file named `head_3.hm`.

Exercise 3: Define Termination and Output for the Head and A-Pillar Impact Analysis

*CONTROL and *DATABASE

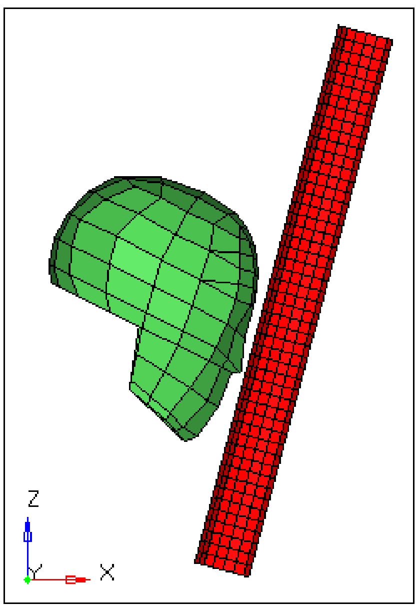
*CONTROL cards are optional, but can be used to change defaults and activate solution options, such as mass scaling, adaptive meshing, and an implicit solution. It is advisable to define *CONTROL_TERMINATION in a model to specify a job's end time.

*DATABASE cards are optional, but are necessary to obtain output files containing results.

Exercise Objective and Tasks

The purpose of this exercise is to help you become familiar with defining LS-DYNA control data and output requests in HyperMesh.

In this you will define the termination and output for a LS-DYNA analysis of a hybrid III dummy head impacting an A-pillar. The head and A-pillar model is shown in the image below.



Head and A-pillar model

This exercise contains the following tasks:

- Specify the time at which LS-DYNA is to stop the analysis with *CONTROL_TERMINATION
- Specify ASCII output with *DATABASE_(Option) cards

- Specify the output of d3plot files with *DATABASE_BINARY_D3PLOT
- Export the model to an LS-DYNA 970 formatted input file

Step 1: Load the LS-DYNA user profile

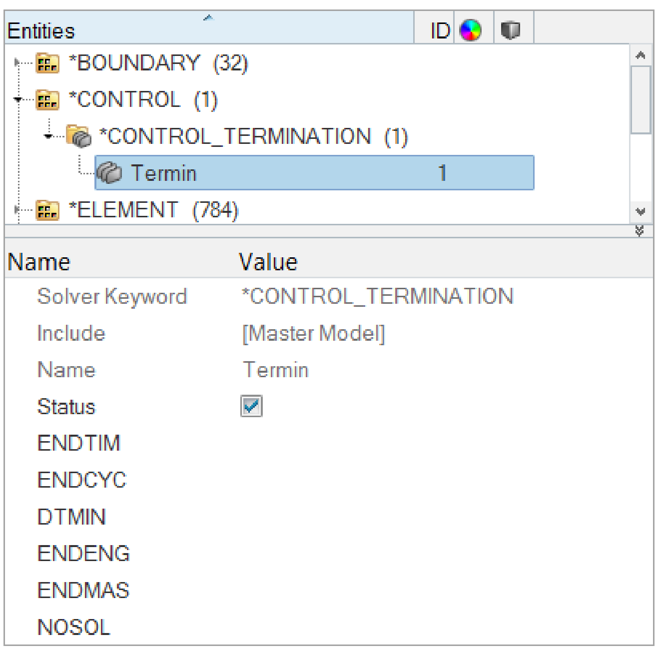
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, set the user profile to **LsDyna**.

Step 2: Retrieve the HyperMesh file

1. Open a model file by clicking ***File*** > ***Open*** > ***Model*** from the menu bar, or clicking on the **Standard** toolbar.
2. In the **Open Model** dialog, open the `head_3.hm` file. The model appears in the graphics area.

Step 3: Specify the time at which you want LS-DYNA to stop the analysis with *CONTROL_TERMINATION

1. In the **Solver** browser, right-click and select *****Create*** > *****CONTROL*** > *****CONTROL_TERMINATION*** from the context menu. HyperMesh creates and opens a new control in the **Entity Editor**.



2. For **ENDTIM** (termination time), enter `2.5`.

Step 4: Specify the output of d3plot files with *DATABASE_BINARY_D3PLOT

1. In the **Solver** browser, right-click and select ***Create*** > ****DATABASE*** > ****DATABASE_BINARY_D3PLOT*** from the context menu. HyperMesh creates and opens a new database in the **Entity Editor**.
2. For **DT** (time interval between outputs), enter `0.1`.

Step 5: Specify ASCII output with *DATABASE_(Option) cards

1. In the **Solver** browser, right-click and select ***Create*** > ****DATABASE*** > ****DATABASE_OPTION*** from the context menu. HyperMesh creates and opens a new database in the **Entity Editor**.
2. Select the ***GLSTAT*** checkbox.
3. For **DT**, enter `0.1`.

   Note: This specifies the output of global data at every 0.1 ms.
4. Select the ***MATSUM*** checkbox.
5. For **DT**, enter `0.1`.

   Note: This specifies the output of material energies every 0.1 ms.
6. Select the ***SPCFORC*** checkbox.
7. For **DT**, enter `0.1`.

   Note: This specifies the output of SPC reaction forces every 0.1 ms.

Step 6: Export the model as an LS-DYNA keyword file

1. From the menu bar, click ***File*** > ***Export*** > ***Solver Deck***.
2. In the **Export - Solver Deck** tab, set **File type** to ***LsDyna***.
3. From the **Template** list, select appropriate template.
4. In the **File** field, navigate to your working directory and save the file as `head_complete.key`.
5. Click ***Export***.

Step 7 (Optional): Submit the LS-DYNA input file to LS-DYNA 970

1. From your desktop's **Start** menu, open the **LS-DYNA Manager** program.
2. From the **solvers** menu, select **Start LS-DYNA analysis**.
3. Load the file `head_complete.key`.
4. Click ***OK*** to start the analysis.

Step 8 (Optional): Post-process the LS-DYNA results using HyperView

The exercise is complete. Save your work to a HyperMesh file.