Altair

HyperWorks

HM-4625: Assemblies using DYNA

In this tutorial, you will learn how to:

- Weld between geometry surfaces and shell elements

- Weld using a master connectors file and duplicating and reflecting connectors

- Create connectors from existing welds to create new welds of a different type

- Swap welded part

- Understand why connectors may fail to realize and how to correct the problems

Tools

The **Connectors** module can be accessed by:

- The **Connectors** menu

- The **Connector Browser** which can be opened using *View* > *Connector Browsers*

Model Files

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

- `frame_assembly_1.hm`

- `frame_assembly_2.hm`

- `frame_assembly_3.hm`

- `frame_assembly_4.hm`

- `body_side_assembly.hm`

Exercises

This tutorial contains the following exercises:
Exercise 1: Weld Between Geometry Surfaces and Shell Elements
Exercise 2: Weld Using a Master Connectors File and Duplicating and Reflecting Connectors
Exercise 3: Create Connectors from Existing Welds to Create New Welds of a Different Type
Exercise 4: Swap Welded Part
Exercise 5: Troubleshoot Failure of Connectors to Realize

The first four exercises will help you become familiar with connecting (welding) an assembly of parts, using various methods, and replacing parts with newer, similar parts and updating their affected connections. The fifth exercise will help you become familiar with troubleshooting failure of connectors to realize.

The part assembly used in the first four exercises is depicted in the image below. A very brief description of the corresponding exercises follows. (The exercises are independent of each other.)
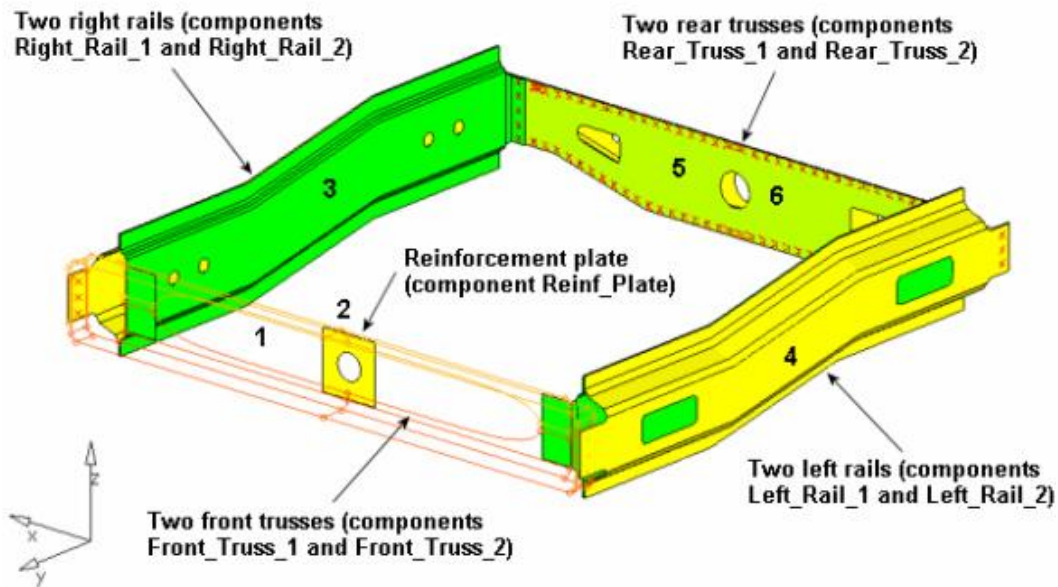

Assembly of parts to connect

Exercise 1: Weld Between Geometry Surfaces and Shell Elements

The purpose of this exercise is to become familiar with creating welds at pre-defined weld points between geometry surfaces and shell elements.

In this exercise, first weld the two front trusses depicted in the image below. To do this: 1) create connectors between their geometry surfaces at pre-defined weld points, and 2) realize the connectors into two node weld elements.
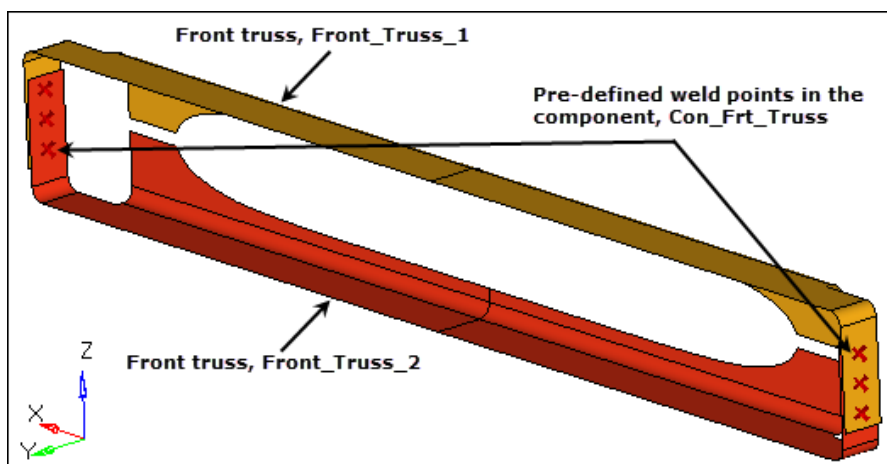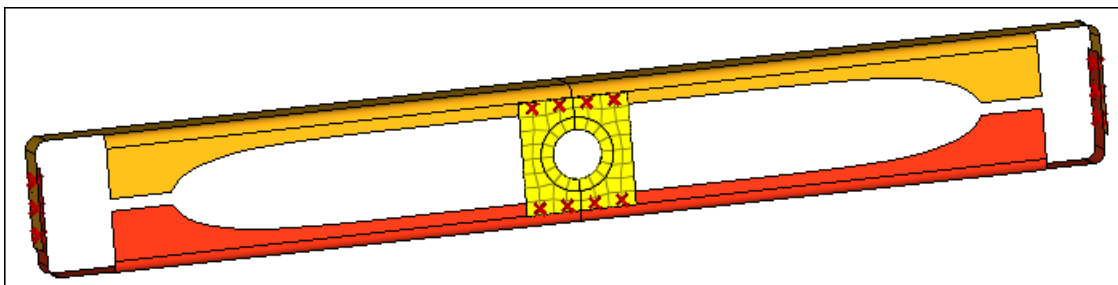

Weld the two front trusses

Second, weld the two front trusses to the reinforcement plate depicted in the image below. To do this: 1) create connectors between their shell elements at pre-defined weld points and, 2) realize the connectors into two node weld elements.
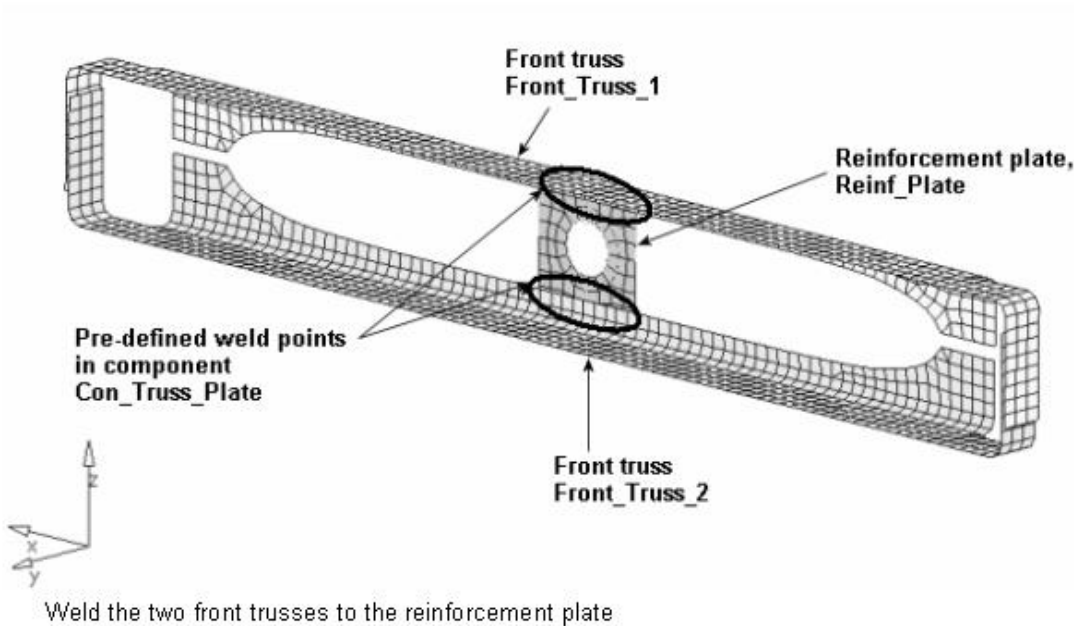
Weld the two front trusses to the reinforcement plate

Step 1: Retrieve and view the model file

1. Start HyperMesh Desktop.

2. In the **User Profile** dialog, set the user profile to **LsDyna**.

3. Open a model file by clicking **File** > **Open** > **Model** from the menu bar, or clicking on the **Standard** toolbar.

4. In the **Open Model** dialog, open the `frame_assembly_1.hm` file.

5. On the **Visualization** toolbar, click to shade your model's geometry and surface edges.

6. Observe the model using various visual options available in HyperMesh (rotation, zooming, and so on).

Step 2: Create welds between the geometry for the two front trusses at the pre-defined weld points

1. Open the connector **Spot** panel by clicking **Connectors** > **Create** > **Spots** from the menu bar.

2. In the **Model** browser, verify that the current component is **Con_Frt_Truss**.

    Hint: The current component is always boldfaced in the **Model** browser, **Component** folder.

3. Set the **location** selector to **points**.

4. Select the six pre-defined weld points by clicking **points** >> **by collectors**.

5. Select the component, **Con_Frt_Truss**.
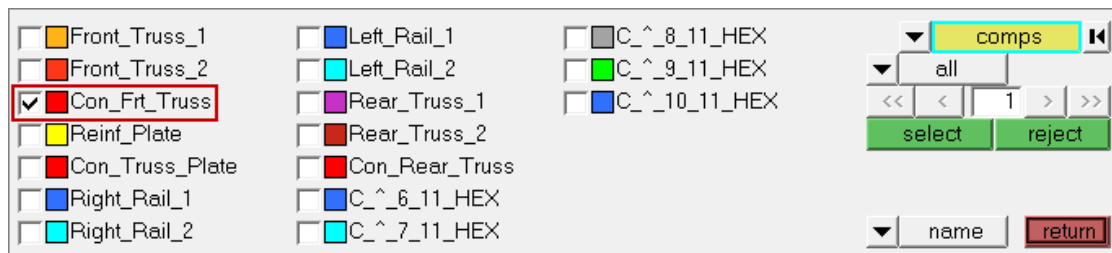
6. Click **select**.

7. Double-click **connect what: comps**.

8. Select the components, **Front_Truss_1** and **Front_Truss_2**.

9. Click **select**.

10. In the **tolerance=** field, enter 5.

11. Click **type=** and select **weld**.

12. Under the **connect what** selector, toggle from **elems** to **geom**.
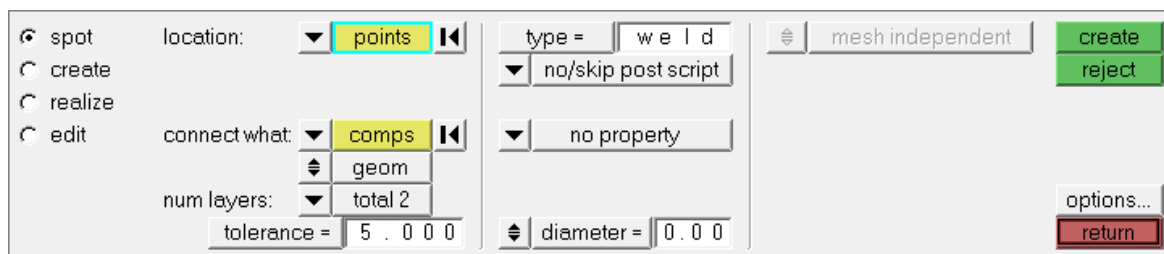
Settings for steps 2.3 through 2.12.

13. Click **create**. HyperMesh creates and realizes six spot connectors, and organizes them as geometry (not elements) in the current component collector, **Con_Frt_Truss**.

    Note: A green connector indicates that the creation of the weld entity was successful. There are three states of connectors: realized (green █), unrealized (yellow █), and failed (red █). The color of the connectors can change from yellow to green (if created manually), indicating they are realized into weld elements. If you create connectors automatically, they will be green immediately as there is no interim unrealized (yellow) state.

△ Altair
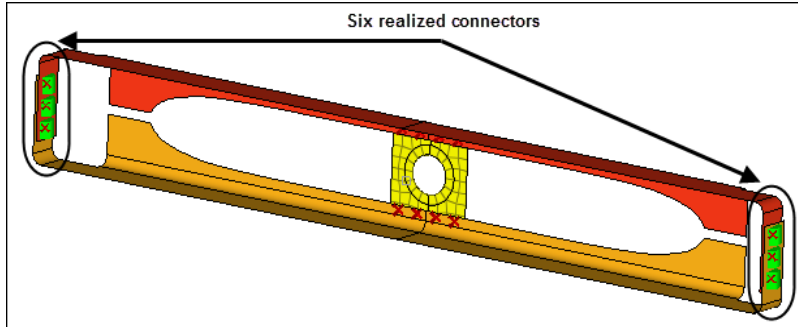
New connectors connecting the front truss components.

HyperMesh also adds fixed points to the surfaces at the ends of the weld elements to guarantee connectivity between the weld elements and the shell mesh that will be created on the surfaces.
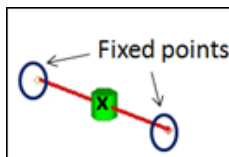

Weld element with fixed points created on the surfaces.

14. Click *return*.

Step 3: Create a shell mesh on the two front truss components

1. Open the **Automesh** panel by clicking *Mesh* > *Create* > *2D Automesh* from the menu bar, or pressing *F12*.

2. Go to the **size and bias** subpanel.

3. Click *surfs* >> *by collector*.

4. Select the components, **Front_Truss_1** and **Front_Truss_2**.

5. Click *select*.

6. In the **elem size=** field, enter 10.

7. Set the **mesh type** to *mixed*.

8. Set the **elems to surf comp**/**elems to current comp** toggle to *elems to surf comp*.

9. Switch the mesh mode from **interactive** to *automatic*.

10. Click *mesh*. HyperMesh meshes the surfaces.
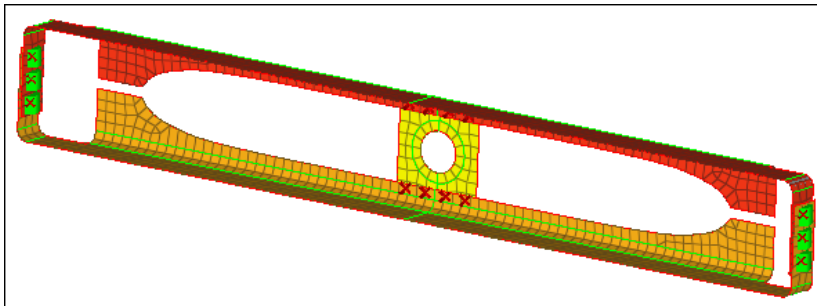
11. On the **3D View Controls** toolbar, left-click on ⌕ to zoom into the area with a connector. See how the fixed point created from the weld has ensured the mesh seeding passes through the weld.
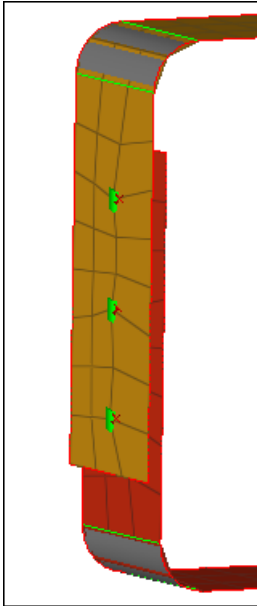

12. Click *return*.

Step 4: Create connectors between the shell mesh for the front trusses and the reinforcement plate at pre-defined points

In this step you will create and realize the connectors manually.

1. In the **Model** browser, **Component** folder, right-click on *Con_Truss_Plate* and select *Make Current* from the context menu.

2. Open the connector **Spot** panel.

3. Go to the **create** subpanel.

4. Set the **location** selector to *points*.

5. Click *points* >> *by collector*.

6. Select the component, **Con_Truss_Plate**.

7. Click *select*.

8. Set **connect when** to *now*.

9. Double-click *connect what: comps*.

10. Select the components: **Front_Truss_1**, **Front_Truss_2**, and **Reinf_Plate**.

11. Click *select*.

12. Under the **connect what** selector, toggle from **geom** to *elems*.
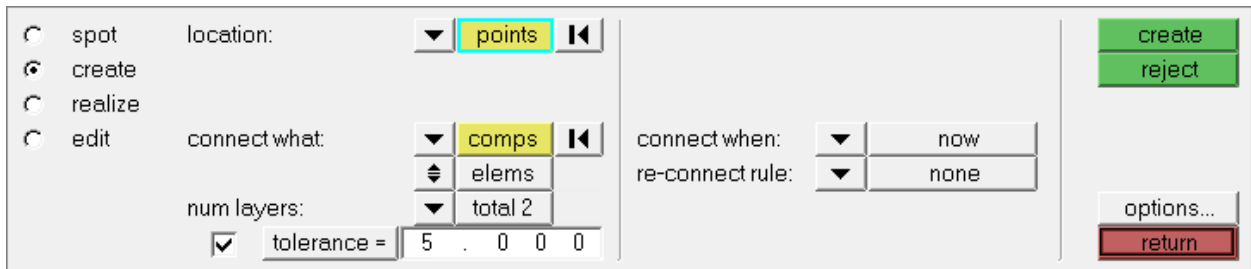
Settings for steps 4.4 through 4.12.

13. Click **create**. HyperMesh creates eight spot connectors at the selected weld points, and organizes them into the current component collector, **Con_Truss_Plate**.
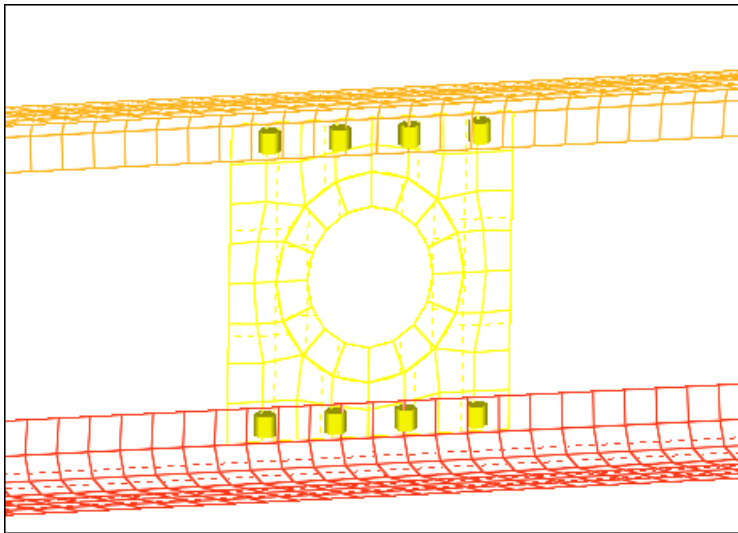

Step 5: Realize the connectors in the component Con_Truss_Plate into weld elements

1. Go to the **realize** subpanel.

2. Click **location: connectors** >> **by collector**.

3. Select the component, **Con_Truss_Plate**.

4. Click **select**.

5. Click **type=** and select **weld**.

6. In the **tolerance=** field, enter 5.

7. Set the **mesh independent**/**mesh dependent** toggle to **mesh dependent**.

   Note: When **mesh dependent** is active, HyperMesh equivalences the nodes if the realized finite element of the connector is coincident to a node of the shell mesh it is being connected to. If there are no suitable nodes present, this option partitions the mesh accordingly to ensure the mesh seeding passes through the weld point.
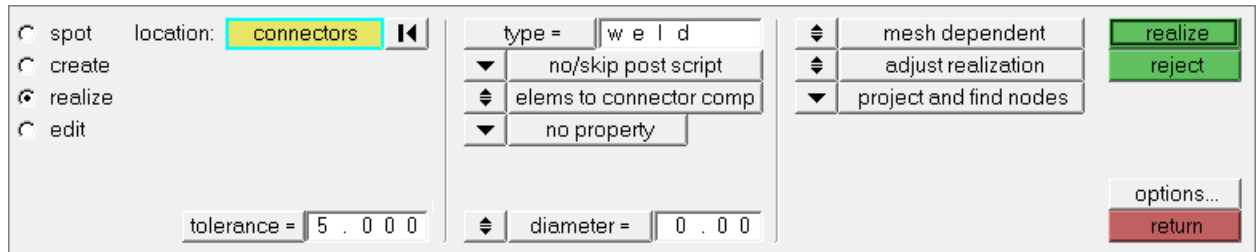
8. Click *realize*. HyperMesh realizes the selected connectors into weld elements.
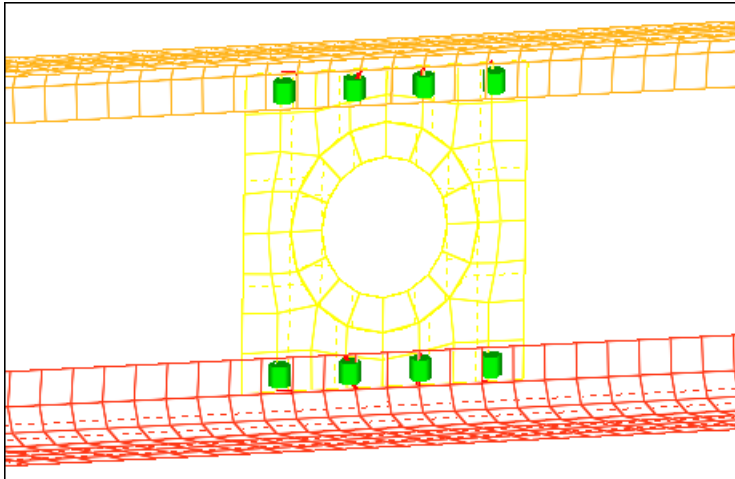
9. Click *return*.

## Step 6 (Optional): Save your work

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

Exercise 2: Weld Using a Master Connectors File and Duplicating and Reflecting Connectors

The purpose of this exercise is to become familiar with:

-   Defining weld locations in HyperMesh by importing a master connectors file.
-   Duplicating, reflecting, and updating connectors to create welds.

In this exercise, you will first weld the two right rails to each other and to the two front trusses depicted in the image below. To do this: 1) import weld point data from a master connectors file, 2) create connectors, and 3) realize the connectors into LS-DYNA 100 Mat100 (beam) welds.

△ Altair
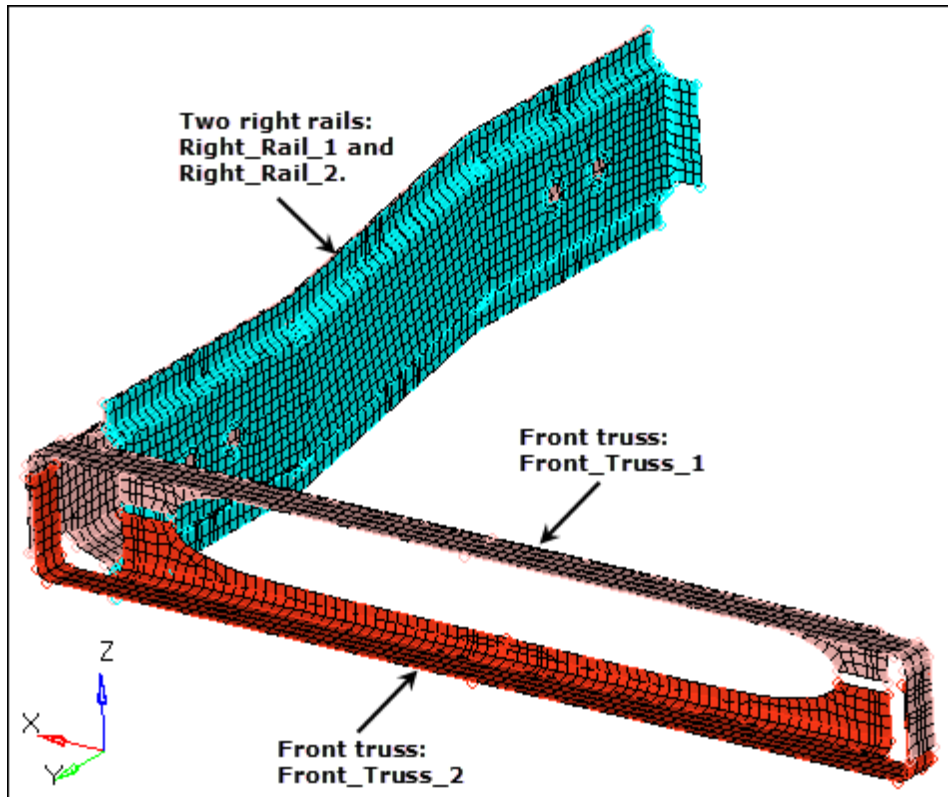
Weld the two right rails to each other and to the front trusses.

Second, you will weld the two left rails to each other and to the two front trusses depicted in the image below. To do this: 1) duplicate and reflect the connectors that were created by importing the master connectors file, 2) update the link information for the reflected connectors, and 3) realize the connectors into LS-DYNA 100 Mat100 (beam) welds.
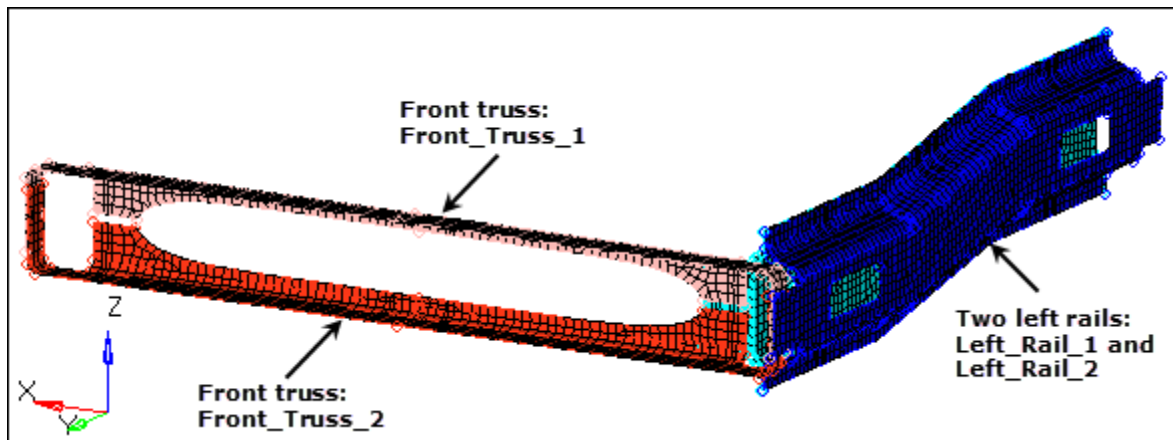

Weld the two left rails to each other and to the two front trusses.

Third, you will combine adjacent 2T connectors into 3T connectors in order to update adjacent 2T welds to 3T welds.

Step 1: Retrieve and view the model file

1. Start HyperMesh Desktop.

2. In the **User Profile** dialog, set the user profile to **LsDyna**.

3. Open a model file by clicking *File* > *Open* > *Model* from the menu bar, or clicking 📂 on the **Standard** toolbar.

4. In the **Open Model** dialog, open the `frame_assembly_2.hm` file.

5. Observe the model using various visual options available in HyperMesh (rotation, zooming, and so on).
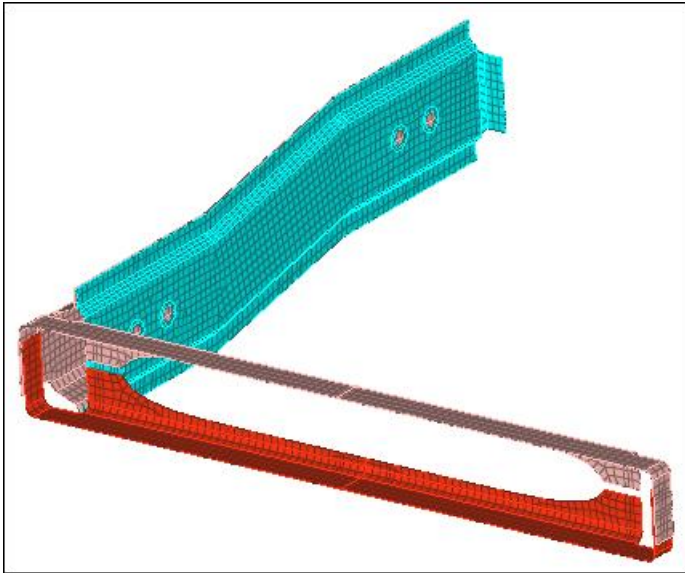
### Step 2: Create connectors to connect the right rails to each other and to the front trusses by importing a master connectors file

1. From the menu bar, click *File* > *Import* > *Connectors*.

2. Set **File type** to *Connectors*

3. In the **File** field, open the `rails_frt_truss.mwf` file.

4. Click *Import*. HyperMesh imports the connectors from the file, and organizes them into a new component, **CE_Locations**.

△ Altair
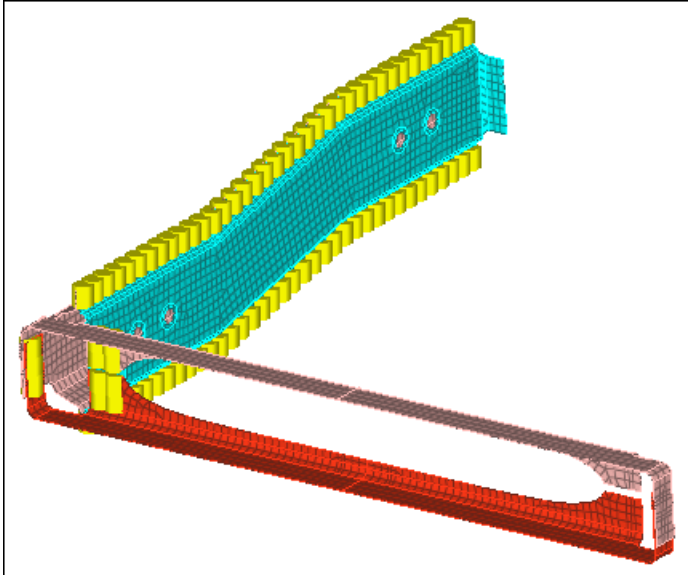
Step 3: Realize the connectors in the component, CE_Locations, into LS-DYNA 100 Mat100 (beam) welds

1. In the **Model** browser, **Component** folder, right-click on *CE_Locations* and select *Make Current* from the context menu.

2. Open the connector **Spot** panel, **realize** subpanel by clicking *Connectors > Realize > Spots* from the menu bar.

3. Click *location: connectors >> by collector*.

4. Select the component, *CE_Locations*.
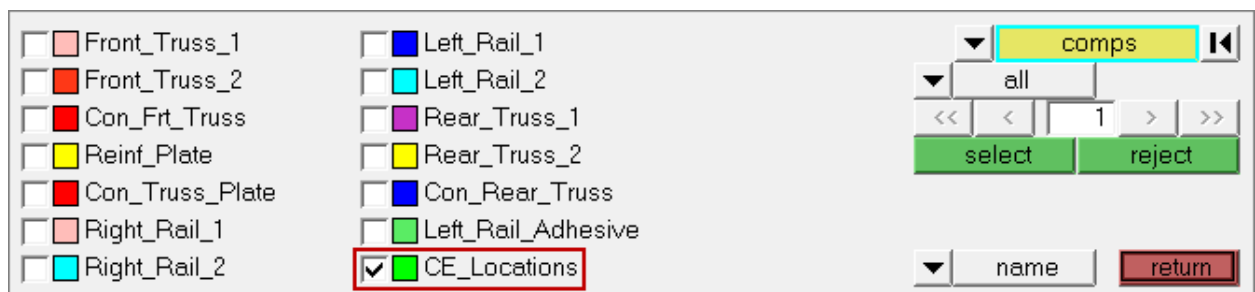
5. Click *select*.

6. In the **tolerance =** field, enter 5.

7. Set the **mesh independent**/**mesh dependent** toggle to *mesh independent*.

8. Click *type =* and select *mat100*.
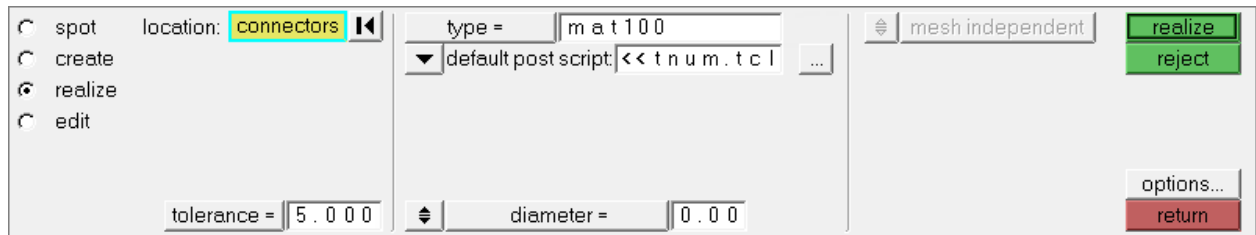
Settings for steps 3.3 through 3.8.

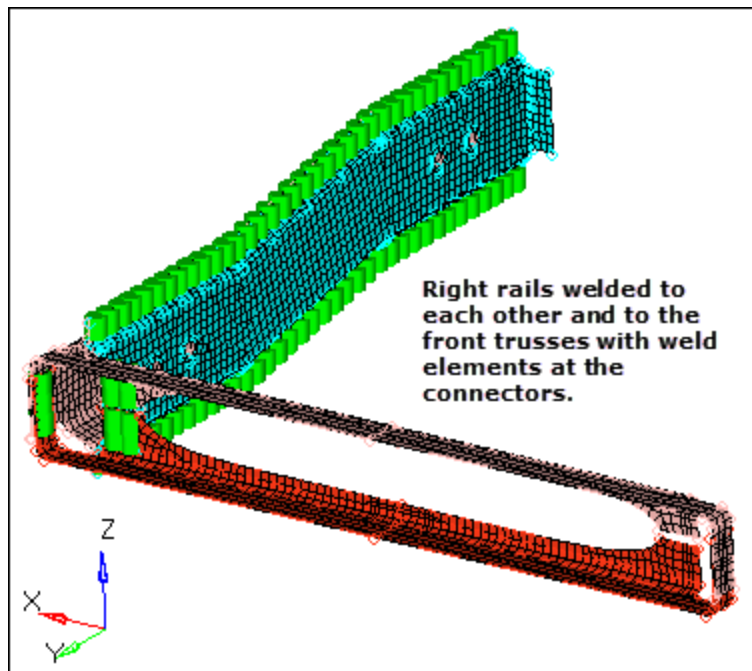9. Click *realize*. HyperMesh realizes the selected connectors into LS-DYNA 100 Mat100 (beam) welds, and organizes them into the following new component collectors:

    - C_^_1_7_BEAM_100

    - C_^_6_7_BEAM_100

    - C_^_2_7_BEAM_100

    - C_^_1_6_BEAM_100

    Note: The naming convention for these components is C_^_[id of comp 1]_[id of comp 2].

Right rails welded to each other and to the front trusses with weld elements at the connectors.

10. Click *return*.

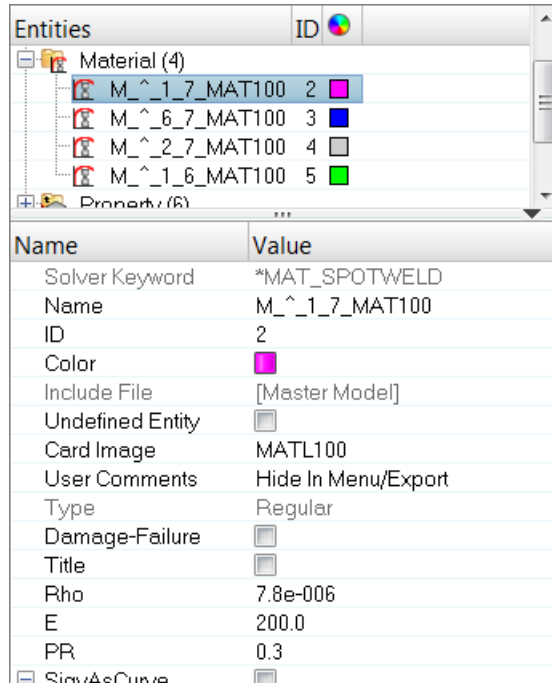## Step 4: Verify materials were automatically created when the LS-DYNA 100 Mat100 (beam) welds were created

1. In the **Model** browser, **Material** folder, review the materials with IDs 2 through 5, that were created when the LS-DYNA 100 Mat100 (beam) welds were created.

    Note: The naming convention for them is M_^_[id of comp 1]_[id of comp 2]. One material is created for every two components that are connected.

2.   Select one of the materials with IDs 2 through 5. The **Entity Editor** opens and displays
     the material's card details.

     Note: The solver keyword is set to ***MAT_SPOTWELD**, and the values are
                 automatically specified for the **density**, **Young's Modulus**, and **Poisson's
                 Ratio** parameters.

Entities	ID 🔵	
🗀 Material (4)		
🔧 M_^_1_7_MAT100	2	🟪
🔧 M_^_6_7_MAT100	3	🟦
🔧 M_^_2_7_MAT100	4	⬜
🔧 M_^_1_6_MAT100	5	🟩
🗀 Property (6)		

| Name	Value
Solver Keyword	*MAT_SPOTWELD
Name	M_^_1_7_MAT100
ID	2
Color	🟪
Include File	[Master Model]
Undefined Entity	☐
Card Image	MATL100
User Comments	Hide In Menu/Export
Type	Regular
Damage-Failure	☐
Title	☐
Rho	7.8e-006
E	200.0
PR	0.3
SigyAsCurve	☐

Step 5: Verify that properties were automatically created when the LS-DYNA 100 Mat100 (beam) welds were created

1.   In the **Model** browser, **Property** folder, review the properties with IDs 1 through 6,
     that were created when the LS-DYNA 100 Mat100 (beam) welds were created.

     Note: The naming convention for them is `P_^_[id of comp 1]_[id of comp 2]`.
                 One property is created for every two components that are connected.

2.   Select one of the properties with IDs 1 through 6. The **Entity Editor** opens and displays
   the property's card details.

     Note: The solver keyword is set to ***SECTION_BEAM**, and the default values are
                 specified for the property's parameters.

△ Altair
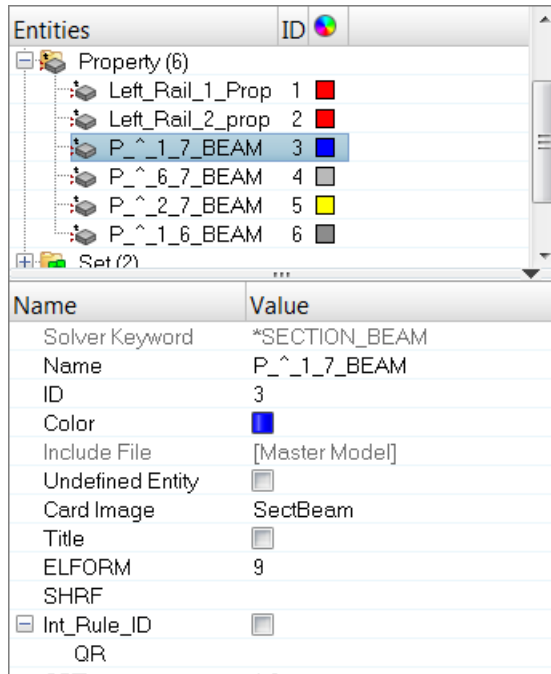
Step 6: Verify that a contact was automatically created when the LS-DYNA 100 Mat100 (beam) welds were created

1. In the **Model** browser, **Group** folder, click *C_Contact_Spotweld_1*. The **Entity Editor** opens and displays the group's card details.

    Note:　The solver keyword is set to ***CONTACT_SPOTWELD_ID**.
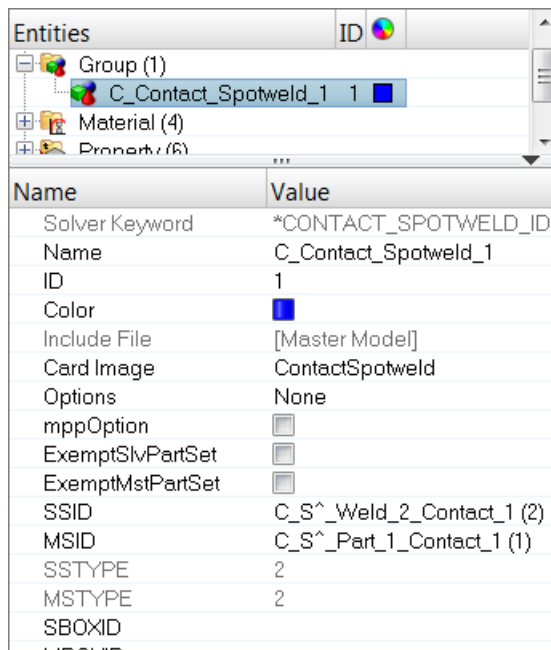
2. Open the **Interfaces** panel by clicking *BCs* > *Assign* > *Interface Entities* from the menu bar.

3. In the panel area, click **name=** and select **C_Contact_Spotweld_1**.

4. Review the contacts by clicking **review**. The contact's master elements temporarily display blue, and its slave elements temporarily display red.
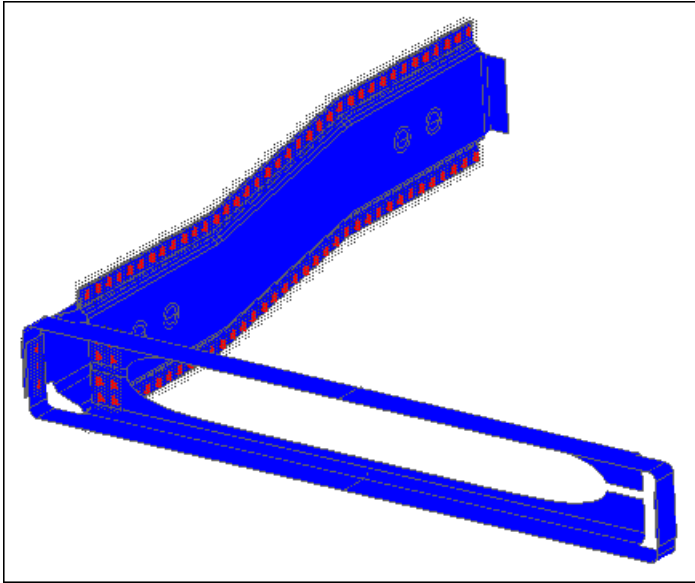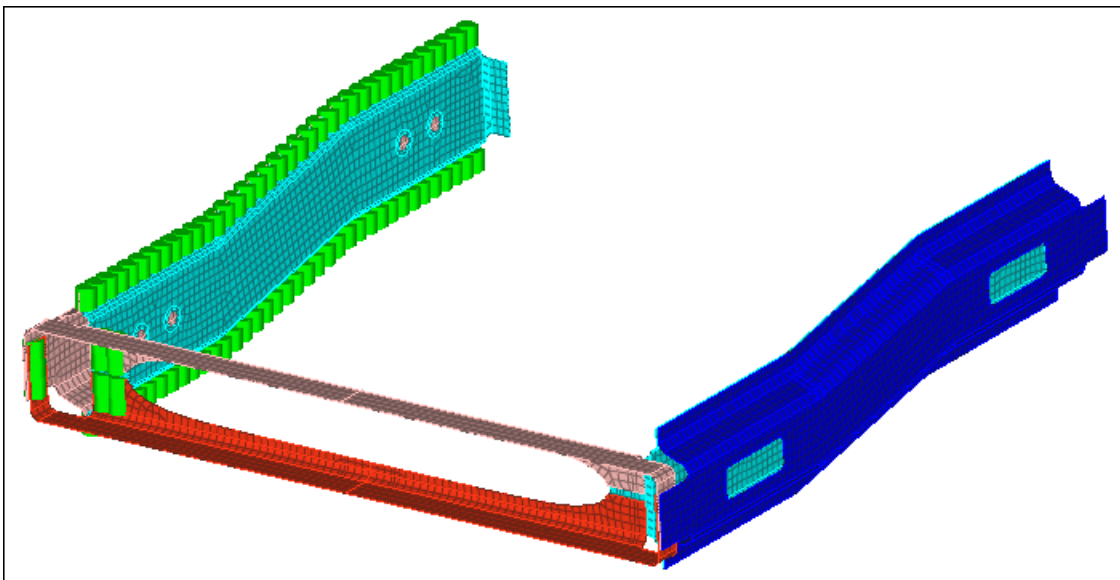

5. Click **return**.

Step 7: Display the components Left_Rail_1 and Left_Rail_2 for elements

1. In the **Model** browser, **Component** folder, right-click on **Left_Rail_1** and **Left_Rail_2** and select **Show** from the context menu. The components display in the graphics area.

   Tip: Select multiple components by pressing **CTRL** while left-clicking on the components.

Step 8: Duplicate the connectors created from the master connectors file and reflect them

1. In the **Model** browser, right-click and select **Create** > **Component** from the context menu. A new component opens in the **Entity Editor**.
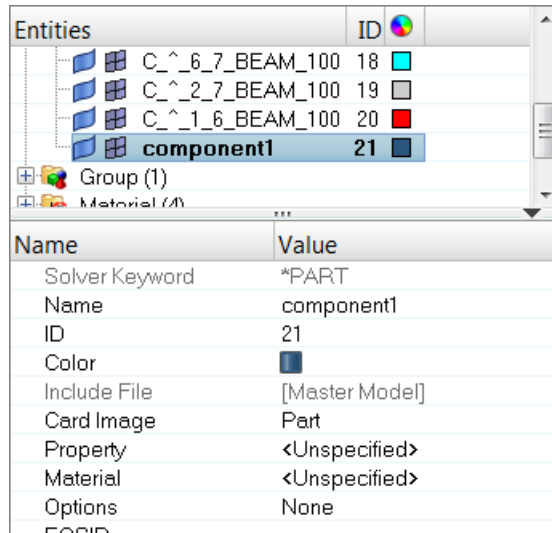

2. For **Name**, enter `CE_Locations_Dup`.

    Note: You do not need to assign any materials or properties.

3. Open the **Reflect** panel by clicking **Connectors** > **Reflect** > **Connectors** from the menu bar.

4. Click **connectors** >> **by collector**.

5. Select the component, **CE_Locations**.

6. Click **select**.

7. Click **connectors** >> **duplicate** >> **current comp**. HyperMesh duplicates the displayed connectors, are organizes them into the current component, **CE_Locations_Dup**.

8. Set the orientation selector to **x-axis**.

    Note: This is the axis normal to the plane of interest.

9. Reflect about the base node by clicking **B**.

10. Click **x =**. HyperMesh activates the **x =**, **y =**, and **z =** fields.

    Note: By default their values are `0.000`, which is the base point you want to reflect about.
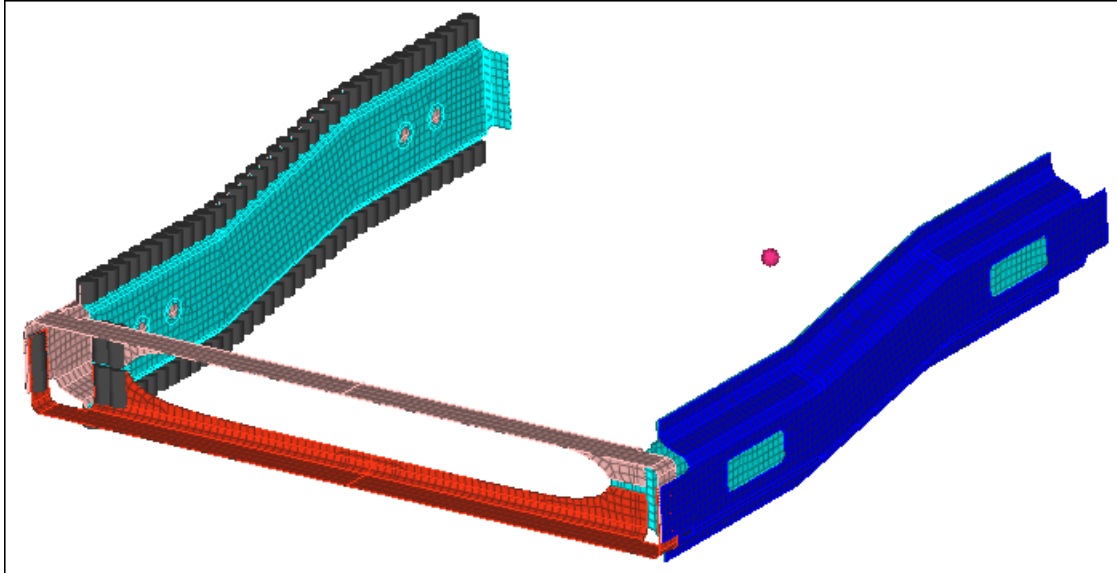
11. Click **return**.

12. Click **reflect**. HyperMesh reflects the connectors.

13. Click **return**.
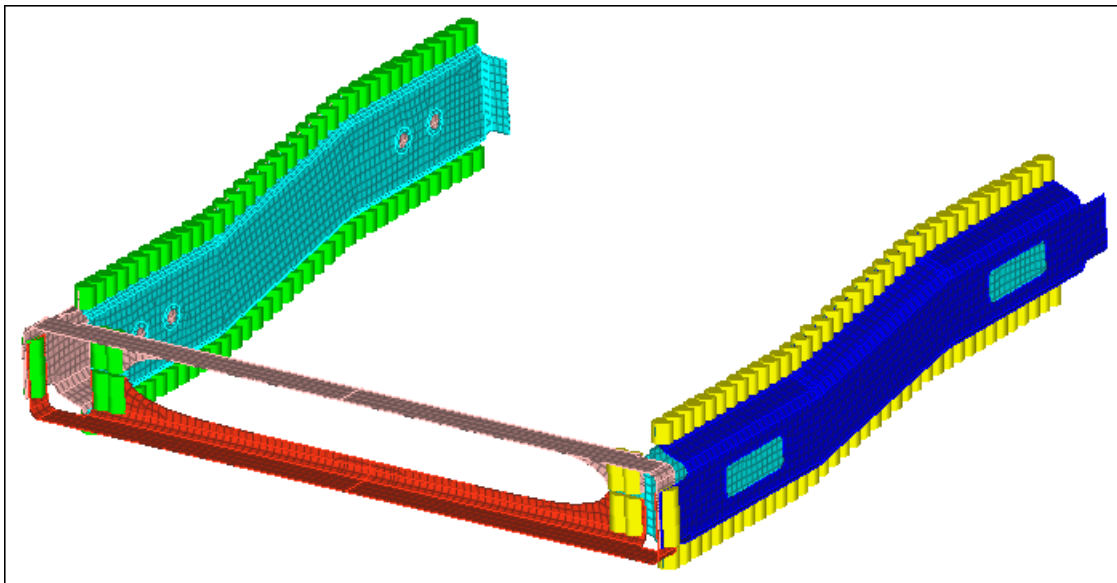
Step 9: Update the connectors for the left rails to link them to the left rail components

1. Open the **Connector** browser by clicking **View** > **Browsers** > **HyperMesh** > **Connector** from the menu bar.

2. In the **Connector Entity** browser, expand the **mat 100** folder, which contains the connectors in the model with material mat100.

> **Note**: The reflected connectors are displayed in yellow, which indicates that they are unrealized.

3. Select all of the unrealized connectors in the list.

Entities	Layer	Tolerance	Link1	Link2	Link3	RealizeTo	State
● 188	2	5.0	◆ Right_Rail_1	◆ Right_Rail_2		property script	realized
● 187	2	5.0	◆ Right_Rail_1	◆ Front_Truss_1		property script	realized
● 186	2	5.0	◆ Right_Rail_1	◆ Front_Truss_1		property script	realized
● 185	2	5.0	◆ Front_Truss_1	◆ Right_Rail_1		property script	realized
○ 270	2	5.0	◆ Front_Truss_1	◆ Right_Rail_2		property script	unrealized
○ 271	2	5.0	◆ Front_Truss_1	◆ Right_Rail_2		property script	unrealized
○ 272	2	5.0	◆ Front_Truss_1	◆ Right_Rail_2		property script	unrealized
○ 273	2	5.0	◆ Front_Truss_1	◆ Right_Rail_2		property script	unrealized
○ 274	2	5.0	◆ Front_Truss_2	◆ Right_Rail_2		property script	unrealized
○ 275	2	5.0	◆ Front_Truss_2	◆ Right_Rail_2		property script	unrealized
○ 276	2	5.0	◆ Front_Truss_2	◆ Right_Rail_2		property script	unrealized
○ 277	2	5.0	◆ Right_Rail_2	◆ Front_Truss_2		property script	unrealized
○ 278	2	5.0	◆ Right_Rail_2	◆ Right_Rail_1		property script	unrealized
○ 279	2	5.0	◆ Right_Rail_1	◆ Right_Rail_2		property script	unrealized
○ 280	2	5.0	◆ Right_Rail_1	◆ Right_Rail_2		property script	unrealized

4. In the **Entities** column, right-click on the selected connectors and select ***Update Link*** from the context menu. The **Update** window opens.

Entities	Layer	Tolerance	Link1	Link2	Link3	RealizeTo	State
○ 270			ront_Truss_1	◆ Right_Rail_2		property script	unrealized
○ 271			ront_Truss_1	◆ Right_Rail_2		property script	unrealized
○ 272			ront_Truss_1	◆ Right_Rail_2		property script	unrealized
○ 273			ront_Truss_1	◆ Right_Rail_2		property script	unrealized
○ 274			ront_Truss_2	◆ Right_Rail_2		property script	unrealized
○ 275			ront_Truss_2	◆ Right_Rail_2		property script	unrealized
○ 276			ront_Truss_2	◆ Right_Rail_2		property script	unrealized
○ 277			ight_Rail_2	◆ Front_Truss_2		property script	unrealized
○ 278			ight_Rail_2	◆ Right_Rail_1		property script	unrealized
○ 279			ight_Rail_1	◆ Right_Rail_2		property script	unrealized
○ 280			ight_Rail_1	◆ Right_Rail_2		property script	unrealized
○ 281			ight_Rail_1	◆ Right_Rail_2		property script	unrealized
○ 282			ight_Rail_1	◆ Right_Rail_2		property script	unrealized
○ 283			ight_Rail_1	◆ Right_Rail_2		property script	unrealized
○ 284			ight_Rail_1	◆ Right_Rail_2		property script	unrealized
○ 285			ight_Rail_1	◆ Right_Rail_2		property script	unrealized
○ 286			ight_Rail_2	◆ Right_Rail_1		property script	unrealized

Context menu items:
- Create ▶
- Delete ▶
- Convert ▶
- Renumber
- Update Layer
- Update Tolerance
- Update RealizeTo
- Update DiaFactor
- Update Diameter
- Add Link
- **Update Link**
- Remove Links

5. In the **Search** column, set the **Link Type** to ***comps***.

	Update ✕

Name	Search	Replace
Link Type	comps ▾	▾
Link Select	*	
Link State	⌄	⌄
Link Rule	⌄	⌄

6. In the **Search** column, click the ***Link Select*** field.

7. In the panel area, click *component*.

8. Select the component, *Right_Rail_1*.

9. Click *proceed*. The **Link Select** field displays **Right_Rail_1**.

		Update ✕
Name	Search	Replace
Link Type	comps ▾	▾
Link Select	(6) Right_Rail_1	
Link State	⌄	⌄
Link Rule	⌄	⌄

10. In the **Replace** column, set the **Link Type** to *comps*.

11. In the **Replace** column, click the *Link Select* field.

12. In the panel area, click *component*.

13. Select the component, *Left_Rail_1*.

14. Click *proceed*. The **Link Select** field displays **Left_Rail_1**.

		Update ✕
Name	Search	Replace
Link Type	comps ▾	comps ▾
Link Select	(6) Right_Rail_1	(8) Left_Rail_1
Link State	⌄	⌄
Link Rule	⌄	⌄

15. Click *Update*. HyperMesh updates the connector's links.

16. Repeat 9.5 through 9.15, except search for the **Right_Rail_2** component and replace it with the **Left_Rail_2** component.

17. Scroll through the list of unrealized connectors to make sure that none of the connectors are linked to the right rail components.

18. Close the **Update** window by clicking *X* next to **Update**.

19. Right-click on the selected connectors and select *Rerealize* from the context menu. HyperMesh realizes the connectors.
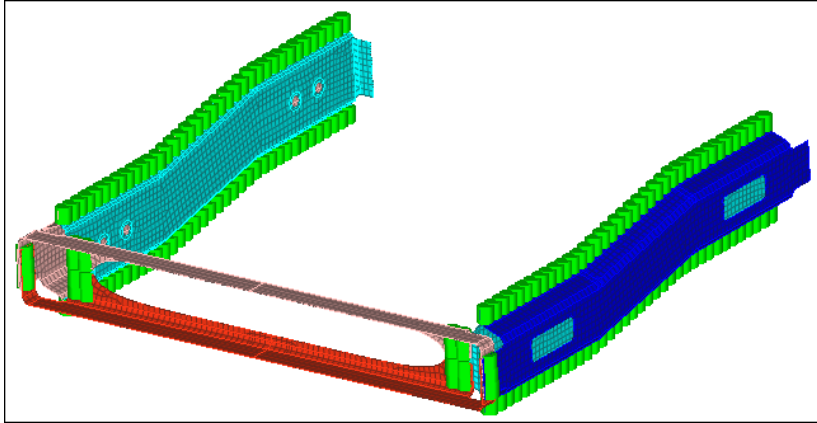
Step 10: Verify that all connectors are realized and identify the pairs of adjacent connectors

All of the connectors listed in the **Connector Entity** browser are displayed in green in the **State** column, which indicates that they are realized.

1. Zoom into one of the two areas where the front trusses are connected to the rail components.

   **Note**:   In these two areas, pairs of adjacent connectors exist.

2. On the **Visualization** toolbar, click 🖥️.

3. In the **Visualization** tab, click 🛢️ .

4. Under **Color by**, select *Layer*. HyperMesh changes the connectors color to purple because under **Layers**, **2t** (two thickness) is defined by the color purple.

    Note: This option indicates that each of these connectors link two components. Because each pair of connectors creates a series of two weld elements, you can combine each pair into a single connector, which links the three components together.
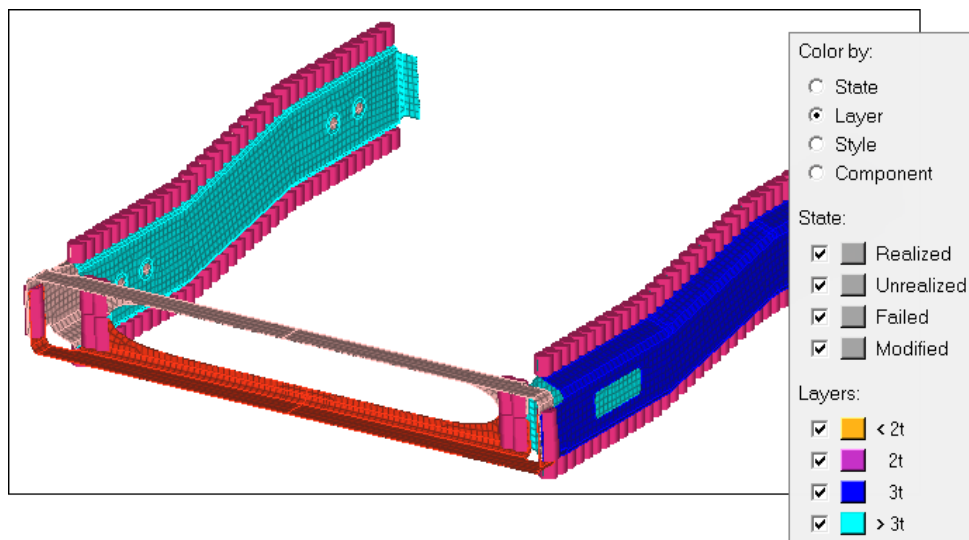
Step 11: Isolate the pairs of adjacent, 2t connectors identified in the previous step

1. In the **Model** browser, turn off the geometry display for all of the components.

2. In the **Connector** browser, **Link Entity** browser, while pressing *CTRL*, select *Front_Truss_1*, *Front_Truss_2*, *Right_Rail_1*, and *Left_Rail_1*.

3. Right-click and select *Find Between* from the context menu. HyperMesh finds and displays the connectors between the four components you selected.
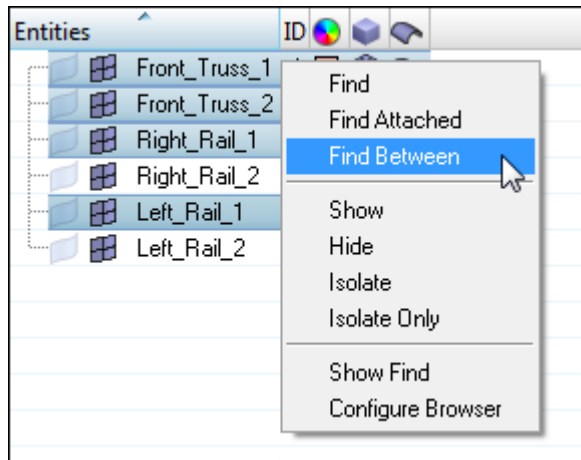
Step 12: Unrealize the selected connectors

The connectors you found in the previous step should still be selected in the **Connector Entity** browser.

1. In the **Connector Entity** browser, right-click on the selected connectors and select *Unrealize* from the context menu.

Step 13 (Optional): Save your work

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

Exercise 3: Create Connectors from Existing Welds to Create New Welds of a Different Type

The purpose of this exercise is to become familiar with absorbing existing finite element welds into connectors in order to create new finite element welds of a different type.

In this exercise, LS-DYNA 101 Mat100 (hexa) welds already connect the rear trusses to each other. You will first update the weld type to LS-DYNA 100 Mat100 (beam) welds. To do this: 1) create connectors from the existing LS-DYNA 101 Mat100 (hexa) welds, and 2) realize the connectors into LS-DYNA 100 Mat100 (beam) welds.

Existing LS-Dyna MAT100(HEXA) Welds
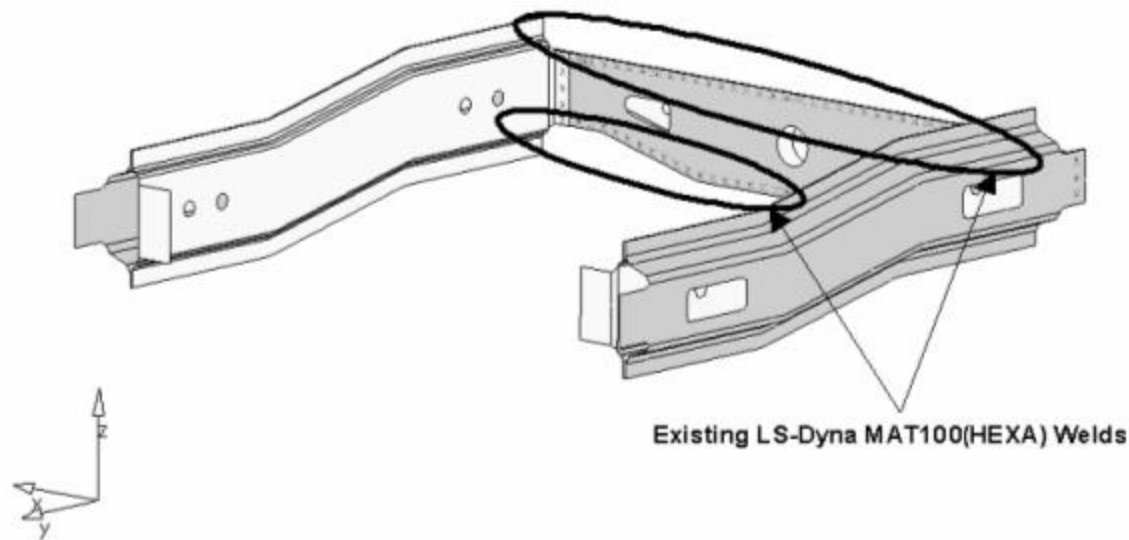
Second, you will update the existing LS-DYNA 101 MAT100 (hexa) welds to LS-DYNA 100 Mat100 (beam) welds.

Step 1: Retrieve and view the model file

1. Start HyperMesh Desktop.

2. In the **User Profile** dialog, set the user profile to **LsDyna**.

3. Open a model file by clicking *File* > *Open* > *Model* from the menu bar, or clicking on the **Standard** toolbar.

4. In the **Open Model** dialog, open the `frame_assembly_3.hm` file.

5. Observe the model using various visual options available in HyperMesh (rotation, zooming, and so on).
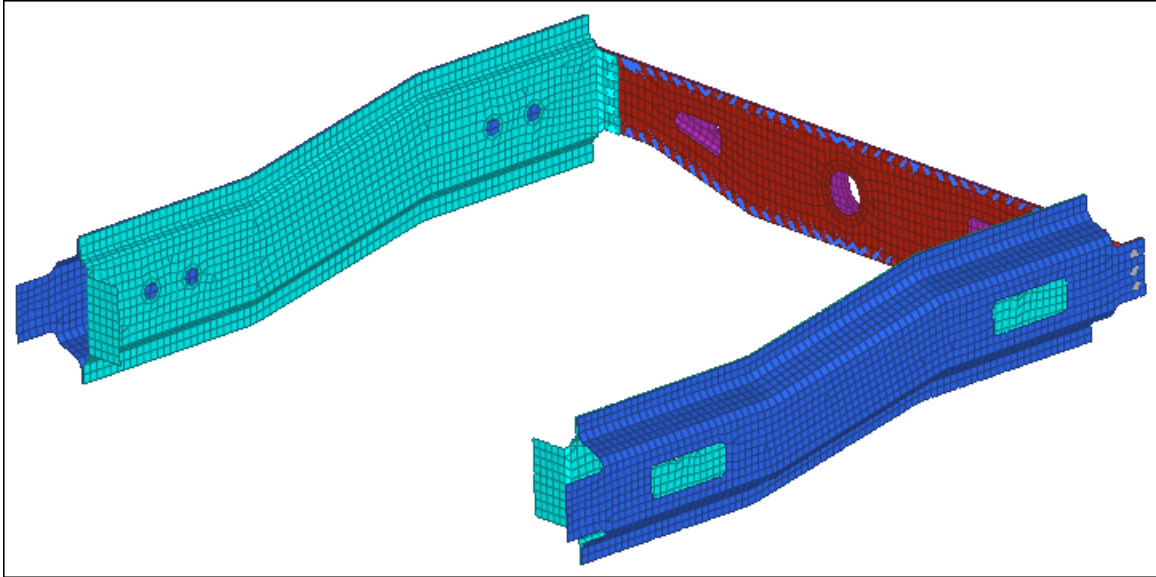
Step 2: Create connectors from existing LS-DYNA 101 Mat100 (hexa) welds

1. From the menu bar, click **Connectors** > **Fe Absorb**.

2. In the **Automated Connector Creation and FE Absorption** dialog, set **FE configs** to **custom**.

3. Set **FE type** to **dyna 101 mat 100 (hexa)**.

4. Set the **Elem filter** toggle to **select**.

5. Double-click the **Elem filter: Elements** selector.

6. In the panel area, click **elems >> by collector**.

7. Select the following components:

   - **C_^_6_11_HEX**
   - **C_^_7_11_HEX**
   - **C_^_8_11_HEX**
   - **C_^_9_11_HEX**
   - **C_^_10_11_HEX**

8. Click **select**.

9. Click **proceed** to return to the dialog.

10. Set **FE connectivity** to **Mesh independent (projs)**.

11. Select the **Move connectors to FE component** checkbox.

12. Click **Absorb**. HyperMesh absorbs the elements into connectors at the locations of the LS-DYNA 101Mat100 (hexa) welds, and organizes them into the respective components to which the LS-DYNA 101 Mat100 (hexa) welds belong.

13. Click **Close**.

Step 3: Isolate the 2t connectors between the Rear_Truss_2 component and the Right_Rail_2 and Left_Rail_2 components

1. Open the **Connector** browser by clicking **View** > **Browsers** > **HyperMesh** > **Connector** from the menu bar.

2. In the **Link Entity** browser, while pressing **CTRL**, select **Rear_Truss_2**, **Right_Rail_2**, and **Left_Rail_2**.

3. Right-click and select **Find Between** from the context menu. HyperMesh finds and displays ten connectors between the three components you selected.
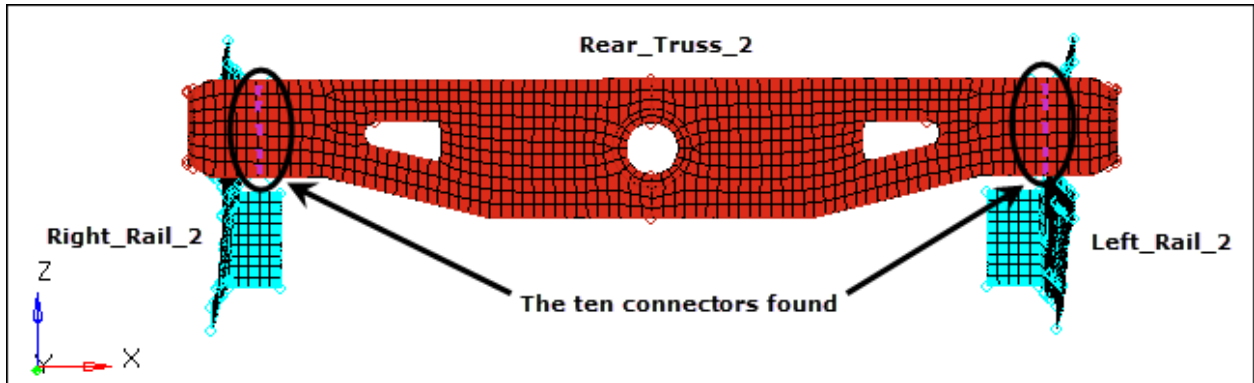
Connectors found between the Rear_Truss_2, Left_Rail_2, and Right_Rail_2 components.

Step 4: Add Rear_Truss_1 as a third link to four of the ten displayed 2t connectors

1. In the **Model** browser, **Component** folder, while pressing **CTRL**, select **Rear_Truss_2**, **Left_Rail_2** and **Right_Rail_2** .

2. Right-click on the selected components and select **Show** from the context menu.

3. Turn on the geometry display for the component **C^9_11_HEX**.

4. Open the **Add Links** panel by clicking **Connectors** > **Assign** > **Links** from the menu bar.

5. Activate the **location: connectors** selector.

6. Select the four connectors indicated in the following image.
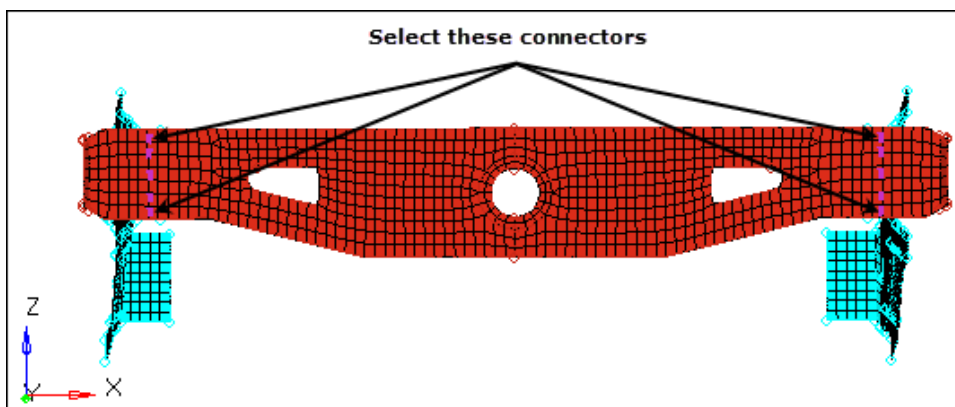
7. Set the **connect what** selector to *comps*.

8. Click *comps*.

9. Select the component, **Rear_Truss_1**.

10. Click *select*.

11. Set the **connect what** toggle to *elems*.

12. Select the checkbox next to **search tol =**.

13. In the **search tol=** filed, enter 5.

14. Set **# of layers** to *total 3*.

Settings for steps 4.5 through 4.14.

15. Click *add links*. HyperMesh updates the connector's definitions, and changes their display color to blue, as they are now 3t connections.
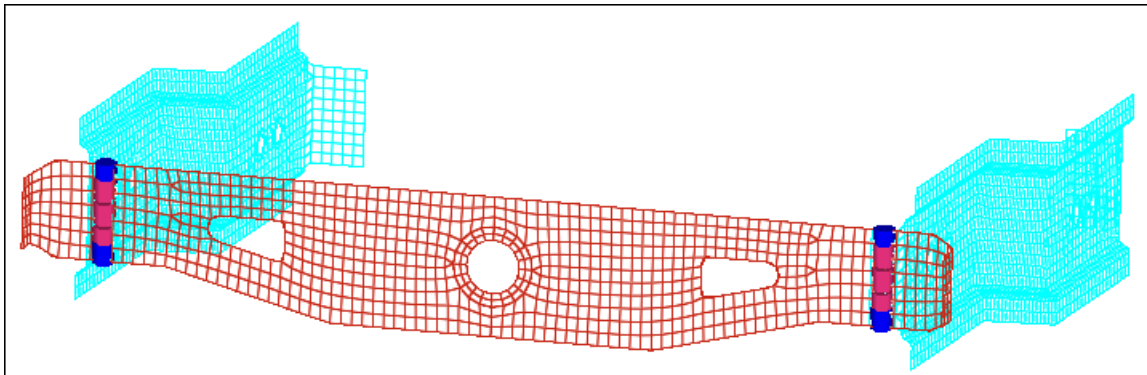
16. Click *return.*

Step 5: Unrealize the connectors for the LS-DYNA 101 Mat100 (hexa) welds

1. In the **Model** browser, turn on the geometry display for the following components only:

    - **C_^_6_11_HEX**
    - **C_^_7_11_HEX**
    - **C_^_8_11_HEX**
   - **C_^_9_11_HEX**
    - **C_^_10_11_HEX**
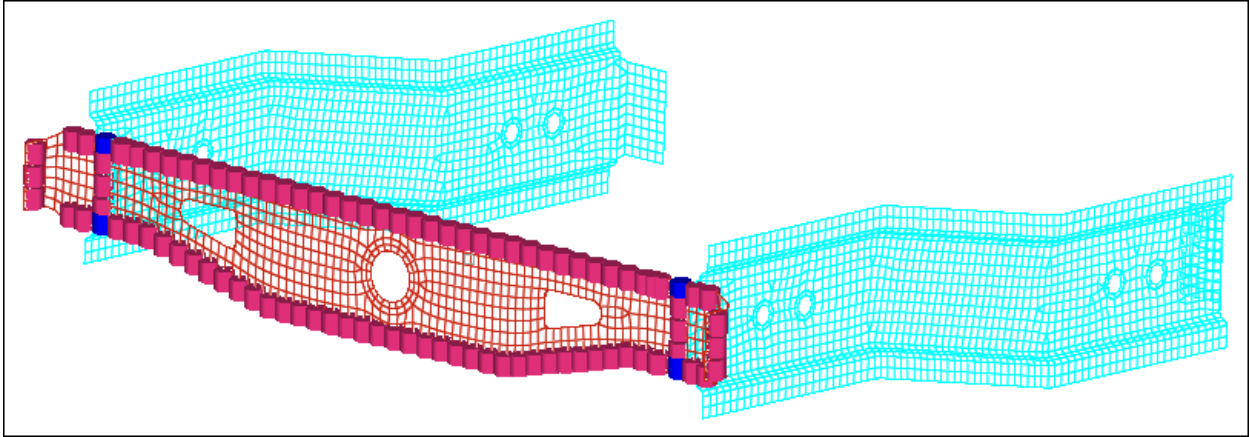
2. Open the **Unrealize** panel by clicking *Connectors* > *Unrealize* from the menu bar.

3. Click *connectors* >> *displayed*.

4. Click *unrealize*. HyperMesh unrealizes the connectors, and deletes the weld elements associated to these connectors.

5. Click *return*.

6. On the **Visualization** toolbar, click 🖥️.

7. In the **Visualization** tab, click 🟢 .

8. Under **Color by**, select *State*. HyperMesh displays the connectors in yellow, which indicates that they are not realized into FE elements.

Step 6: Realize the unrealized connectors into LS-DYNA 100 Mat100 (beam) welds

1. Open the connector **Spot** panel, **realize** subpanel by clicking *Connectors* > *Realize* > *Spots* from the menu bar.

2. Click *location: connectors* >> *displayed*.

3. Click **type=** and select *mat 100*.

4. In the **tolerance =**, enter 5.

5. Click *realize*. HyperMesh realizes the connectors.
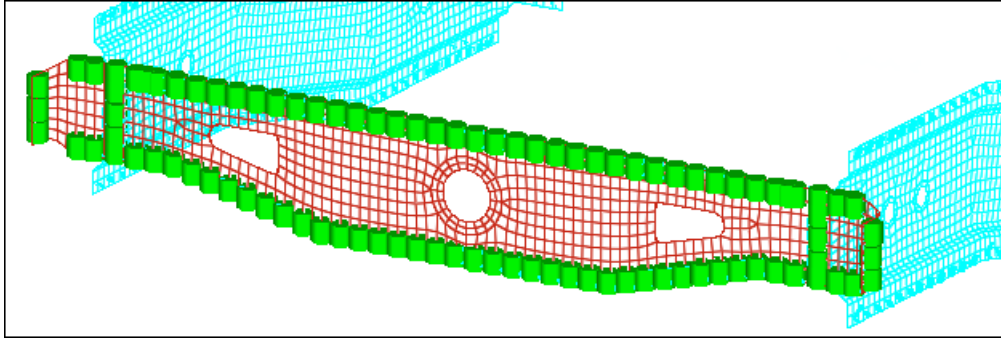
6.   Click *return.*

Step 7 (Optional): Save your work

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

Exercise 4: Swap Welded Part

The purpose of this exercise is to become familiar with swapping welded parts and updating their affected connections (welds).

In this exercise, you will replace the component **Rear_Truss_1** with a new, similar part and update its affected connections (welds). To do this: 1) update the connectors to use the "use name" rule, 2) delete the old part, 3) import the new part, and 4) realize the corresponding connectors into LS-DYNA 100 Mat100 (beam) welds.
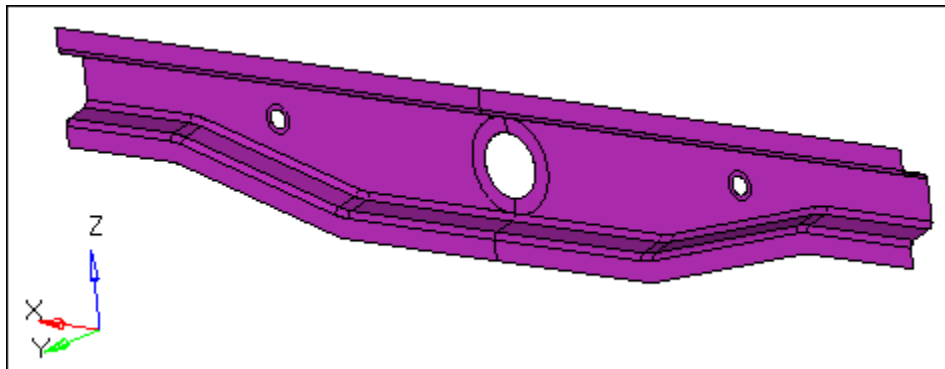

This part (Rear_Truss_1 component) will be swapped.

Step 1: Open the model file

1.   Start HyperMesh Desktop.

2.   In the **User Profile** dialog, set the user profile to **LsDyna**.

3.   Open a model file by clicking *File* > *Open* > *Model* from the menu bar, or clicking on the **Standard** toolbar.

4. In the **Open Model** dialog, open the `frame_assembly_4.hm` file.

5. Observe the model using various visual options available in HyperMesh (rotation, zooming, and so on).
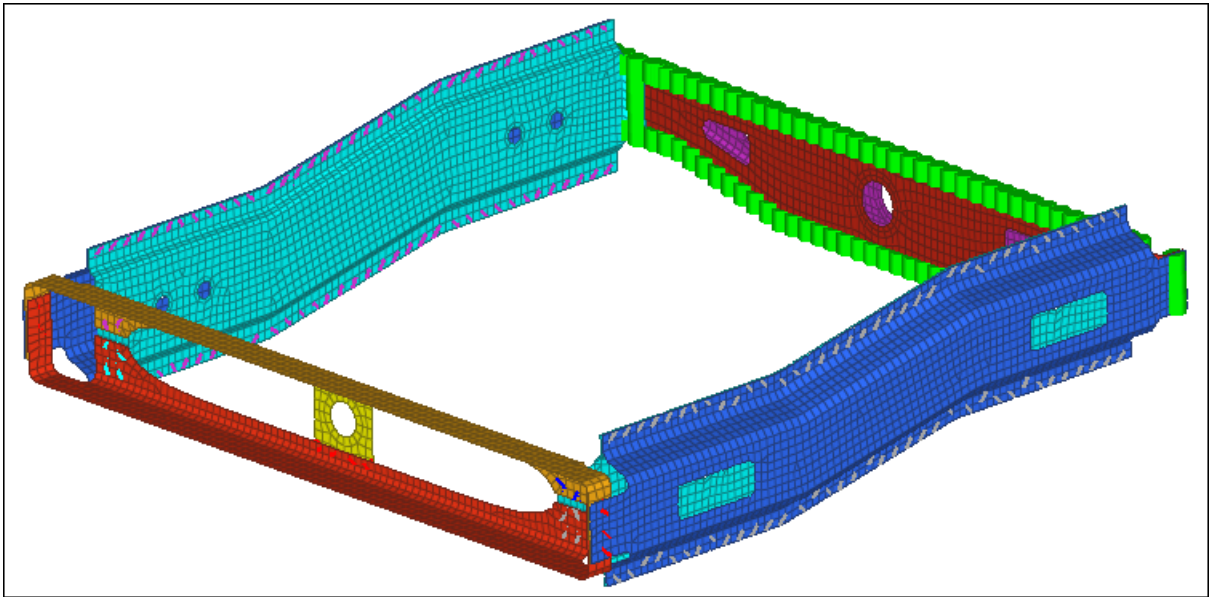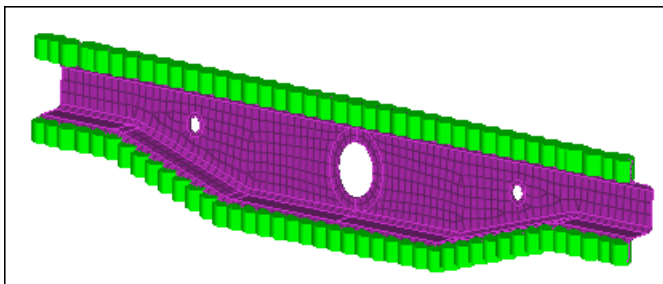

Step 2: Import the new model and update the connectors to use the rule "use name" when reconnecting parts

1. Open the **Connector** browser by clicking *View* > *Browsers* > *HyperMesh* > *Connector* from the menu bar.

2. In the **Connector Entity** browser, right-click and select *Configure Browser* from the context menu.

3. In the **Browser Configuration** dialog, click the *Options* tab.

4. Under **Local connector options**, select the *Extended information* checkbox.

   Note: This option allows you to edit connector attributes.
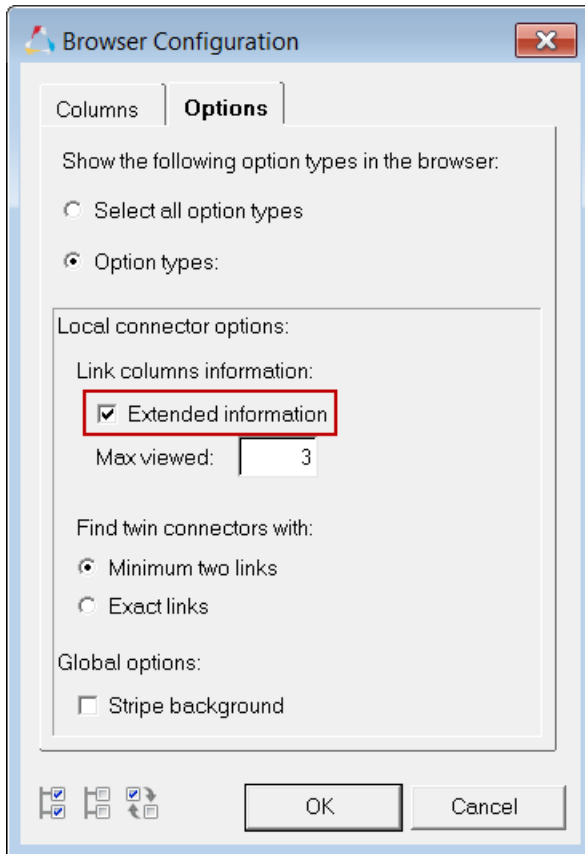
5. Click **OK**.

6. From the menu bar, click **File** > **Import** > **Model**.

7. In the **Import** tab, click [folder icon].

8. In the **Open dialog**, navigate to your working directory and open the
   `rear_truss_1_new.hm` file.

9. Click **Import**.

10. In the **Link Entity** browser, right-click on **Rear_Truss_1** and select **Find** from the
    context menu. HyperMesh isolates the component **Rear_Truss_1** and the connectors
    attached to it in the graphics area. Also, HyperMesh highlights the corresponding
    connectors in the **Connector Entity** browser.

11. In the **Connector Entity** browser, right-click on the selected connectors and select
    Update Link from the context menu.

12. In the **Update** window, **Search** column, click the **Link Select** field.

13. In the panel area, click **component**.

14. Select the component, **Rear_Truss_1**.

15. Click **proceed**. The **Link Select** field displays **Rear_Truss_1**.

16. In the **Search** column, set **Link Rule** to **none**.

17. In the **Replace** column, click the **Link Select** field.

18. In the panel area, click **component**.

19. Select the component, **Rear_Truss_1.1**.

20. Click **proceed**. The **Link Select** field displays **Rear_Truss_1.1**.

	Update ×	
Name	Search	Replace
Link Type	comps ▼	▼
Link Select	(10) Rear_Truss_1	(13) Rear_Truss_1.1
Link State	▼	▼
Link Rule	none ▼	▼

21. In the **Replace** column, set **Link Rule** to *use-name*.

	Update ×	
Name	Search	Replace
Link Type	comps ▼	▼
Link Select	(10) Rear_Truss_1	(13) Rear_Truss_1.1
Link State	▼	▼
Link Rule	none ▼	use-name ▼

22. Click *Update*. HyperMesh updates the connector links.

23. Close the **Update** window by clicking *X* next to **Update**.

Step 3: Delete the component Rear_Truss_1

1. Open the **Delete** panel by pressing *F2*.

2. Set the entity selector to *comps*.

3. Click *comps*.

4. Select the component, *Rear_Truss_1*.

5. Click *select*.

6. Click *delete entity*. The display color of the connectors associated to the deleted component changes from green (realized state) to yellow (unrealized state), because the deleted component was one of the connectors' links. Also, HyperMesh deletes the existing finite element welds.
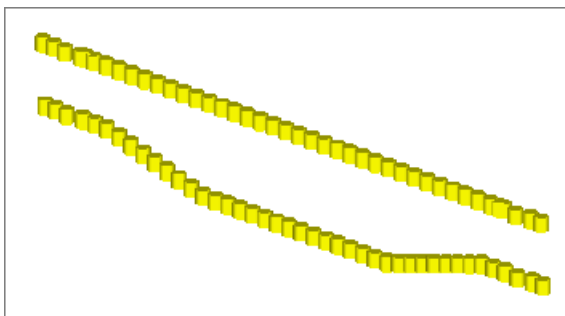
7. Click *return*.

Step 4: Realize the unrealized connectors

1. Open the connector **Spot** panel, **realize** subpanel by clicking ***Connectors* > *Realize* > *Spots*** from the menu bar.

2. Click ***location: connectors* >> *displayed***.

3. Click **type=** and select ***mat100***.

4. In the **tolerance =**, enter 5.

5. Click ***realize***. HyperMesh realizes the connectors.

6. Click ***return***.

Step 5 (Optional): Save your work

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

Exercise 5: Troubleshoot Failure of Connectors to Realize

The purpose of this exercise is to become familiar with troubleshooting the failure of connectors to realize. Specifically, this exercise will help you identify two common issues: 1) small projection tolerance and 2) missing link definitions.

In this exercise, you will realize connectors to weld parts of a vehicle door frame. The model is depicted below.
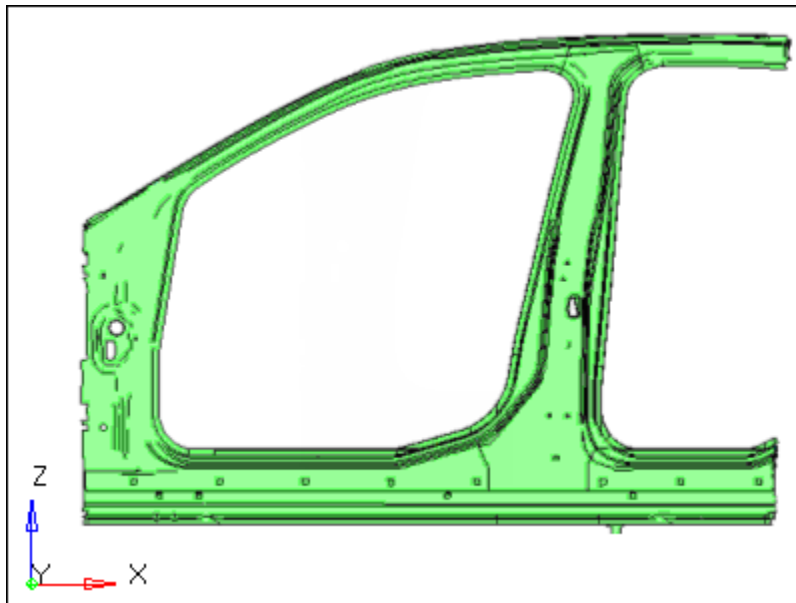

Vehicle door frame model

Step 1: Retrieve the HyperMesh file

1. Start HyperMesh Desktop.

2. In the **User Profile** dialog, set the user profile to **LsDyna**.

3. Open a model file by clicking *File* > *Open* > *Model* from the menu bar, or clickin 📁 on the **Standard** toolbar.

4. In the **Open Model** dialog, open the `body_side_assembly.hm` file.

5. Observe the model using various visual options available in HyperMesh (rotation, zooming, and so on).
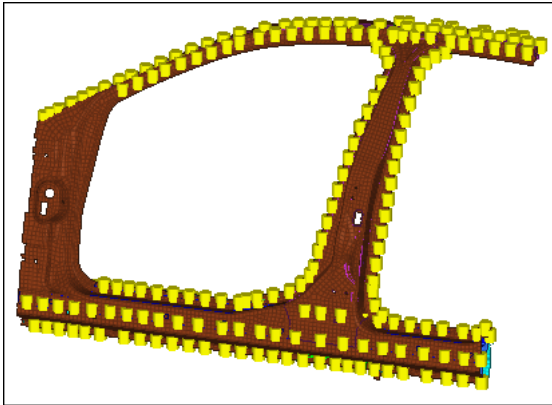
Step 2: Realize all the connectors using a projection tolerance of 1.0

1. Open the connector **Spot** panel, **realize** subpanel by clicking *Connectors* > *Realize* > *Spots* from the menu bar.

2. Click *location: connectors* >> *all*.

3. Click **type =** and select *mat100*

4. In the **tolerance =**, enter `1.0`

5. Click *realize*. HyperMesh realizes the selected connectors into LS-DYNA 100 Mat100 (beam) welds.

   Note: The **Status** bar reads, "*257 connectors realized (9 failed)*".

Entities	Layer	Tolerance	Link1	Link2	Link3	State
⊟ 🟩 mat100 (266)						
🔴 96	2	1.0	🔹 15 none elems	🔹 14 none elems		failed: Link entity 1 (comp id 15)
🔴 223	2	1.0	🔹 19 none elems	🔹 15 none elems		failed: Link entity 1 (comp id 19)
🔴 261	2	1.0	🔹 47 none elems	🔹 9 none elems		failed: Link entity 1 (comp id 47)
🔴 262	2	1.0	🔹 47 none ele			
🔴 155	2	1.0	🔹 49 none ele			
🔴 152	3	1.0	🔹 9 none eler			
🔴 153	3	1.0	🔹 9 none eler			
🔴 154	3	1.0	🔹 9 none eler			
🔴 156	3	1.0	🔹 9 none eler			
🟢 9	2	1.0	🔹 9 none eler			

△ Altair

6.   Click *return*.

Step 3: Review the information table listing the connectors that failed to realize and determine the reasons for failure

1.   On the **Visualization** toolbar, click 🖥.

2.   In the **Visualization** tab, click 🛢 .

3.   Under **State**, clear the **Realized** checkbox. The display of the realized (green) connectors turns off.
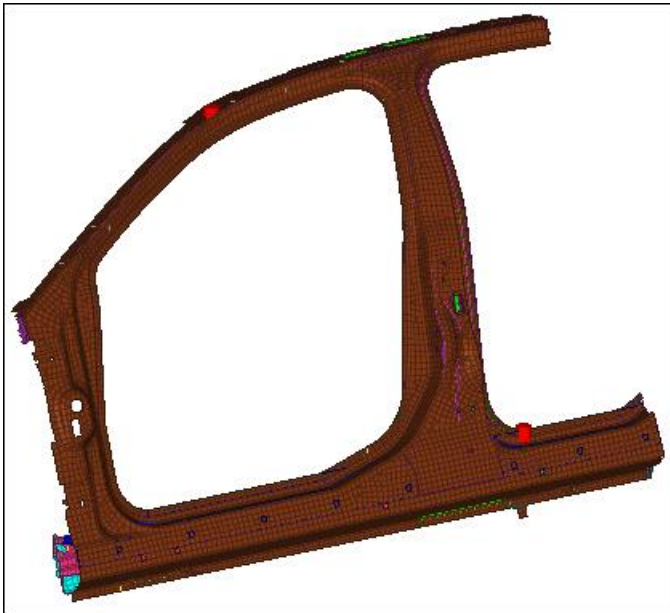
4.   Open the **Connector** browser by clicking *View* > *Browsers* > *HyperMesh* > *Connector* from the menu bar.

5.   In the **Connector Entity** browser, review the five connectors (with the IDs: 96, 155, 223, 261, and 262) that are red.

     Note: In the **Layer** column, 2 layers are specified, and in the **Link1** and **Link2** columns, a link is defined. Because the number of layers and links that were defined match, a possible cause for the connectors not realizing is a small projection tolerance.

6.   Review the four connectors with the IDs 152, 153, 154, and 156.

     Note: In the **Layer** column, 3 layers are specified. In the **Link1** column, one link is defined; in the **Link2** and **Link3** columns, no links are defined. Because the numbers of layers and links do not match, the likely cause for the connectors not realizing is undefined link definitions.

△ Altair

Step 4: Realize the failed connectors using a larger projection tolerance

1. In the **Connector Entity** browser, right-click and select **Create** > **Spot** from the context menu.

2. Click *location: connectors >> displayed*.

3. Click **type =** and select *mat100*

4. In the **tolerance =**, enter 2.0

5. Click *realize*. HyperMesh realizes the selected connectors into LS-DYNA 100 Mat100 (beam) welds.

    Note: The **Status** bar reads, "*5 connectors realized (4 failed)*".

Step 5: Define the missing second link for the failed connectors

1. In the **Connector Entity** browser, select the four red, unrealized connectors.

    Tip: Select multiple connectors by pressing **CTRL** while left-clicking on connectors.

2. Right-click on the selected connectors and select **Add Link** from the context menu.

3. In the **Add Link** window, set **Link Type** to **Components**.

4. Set **Links** to **Select**.

5. Under **Links**, for **Entities**, click **0 Component** >> **comps**.

Name	Value
Add Links	To existing
⊞ Layers	Keep previous
⊞ Tolerance	Keep previous
Link Type	Components
⊟ Links	Select
Entities	comps
⊞ Advanced	

6. In the panel area, click **comps**.

7. Select the component, **Comp2**.

8. Click **select**.

9. Click **proceed**.

10. At the top of the **Add Links** window, click **Add**.

11. In the **confirm** dialog, click **Add Link**. HyperMesh adds the specified link entity to the connector.

Step 6: Define the missing third link for the failed connectors
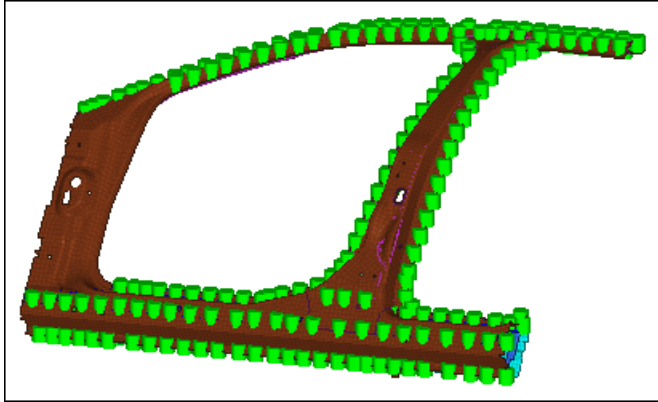
1. Repeat Step 5 above, except select **Comp10** in step 5.7.

Step 7: Realize the failed connectors

In this step you should still be in the **Spot** panel, **realize** subpanel.

1. Click **location: connectors** >> **displayed**.

2. Click **type =** and select **mat100**

3. In the **tolerance =**, enter `4.0`

4. Click **realize**. HyperMesh realizes the selected connectors into LS-DYNA 100 Mat100 (beam) welds.

5. On the **Visualization** toolbar, click 🖥.

6. In the **Visualization** tab, click 🟢 .

7. Under **Color by**, select **State**.

8. Under **State**, select the **Realized** checkbox. HyperMesh displays all of the connectors, which are green.

    Note: All of the connectors are realized into FE elements.

Step 8 (Optional): Save your work

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