



Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-3400: Creating Connectors

altairhyperworks.com

HM-3400: Creating Connectors

In this tutorial, you will learn how to:

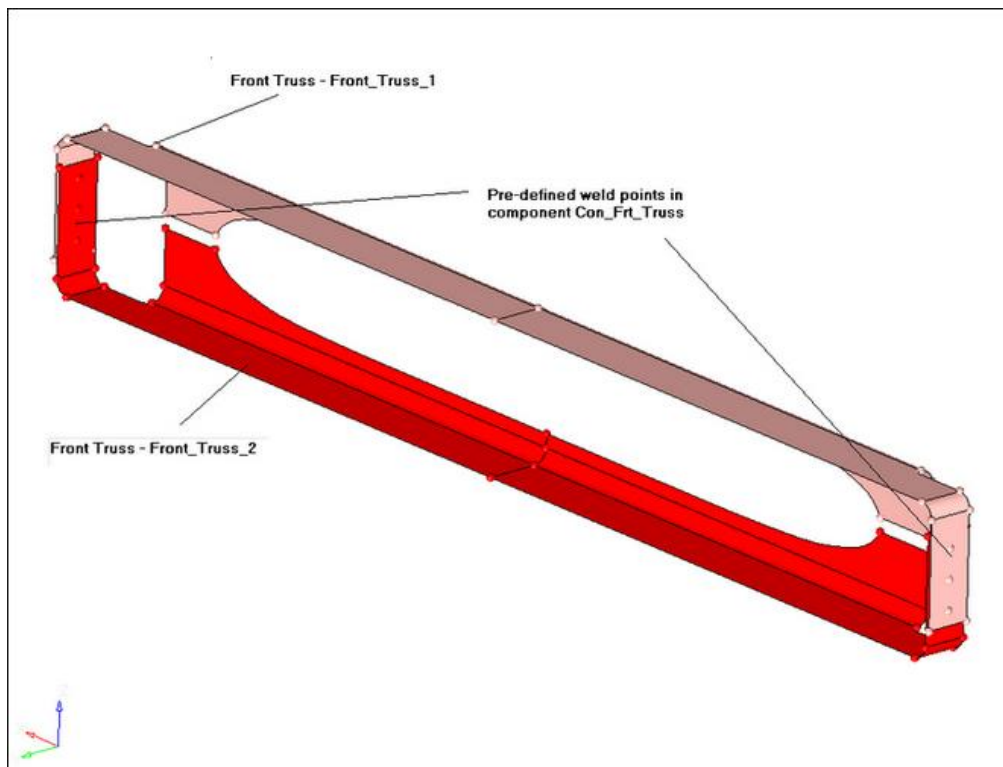
- Weld the two front trusses to each other by creating connectors at pre-defined weld points
- Weld the two front trusses to the reinforcement plate by creating connectors between shell elements
- Weld the right rails to each other and to the front trusses by creating connectors from a master connectors file
- Update weld type of NASTRAN/OPTISTRUCT ACM (area contact method) welds, which already connect the rear trusses to each other, by first creating connectors from these welds and then realizing the connectors into two-noded weld elements.

Model Files



This exercise uses the `frame_assembly.hm` file, which can be found in the `hm.zip` file. Copy the file(s) from this directory to your working directory.

Exercise



Weld the two front trusses by creating connectors between geometry surfaces at pre-defined weld points.



Step 1: Retrieve and view the model file.

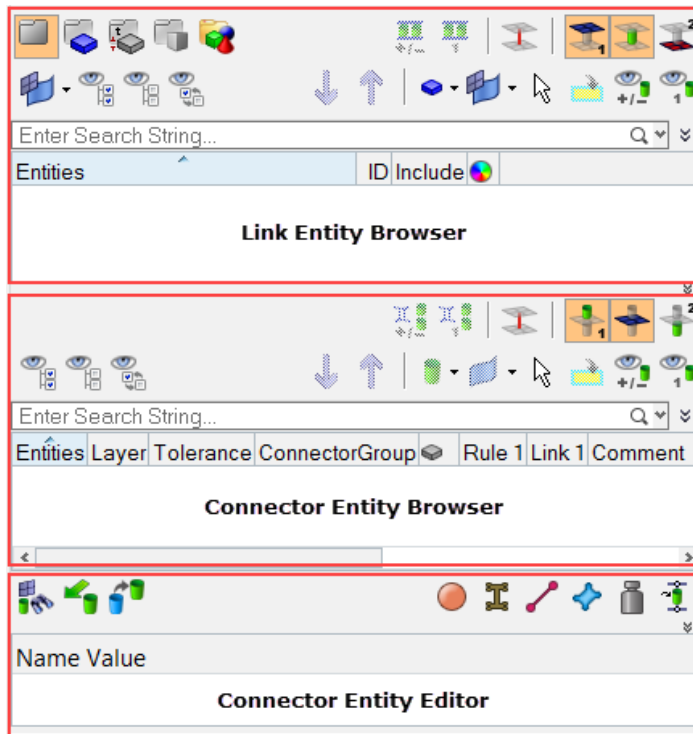
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, select **OptiStruct**.
3. Click **OK**.
4. Open a model file by clicking **File > Open > Model** from the menu bar, or click  on the **Standard** toolbar.
5. In the **Open Model** dialog, open the `frame_assembly.hm` file. A model appears in the graphics area.
6. On the **Visualization** toolbar, click  to shade your model's geometry and surface edges.
7. Observe the model using various visualization options available in HyperMesh (rotation, zooming, and so on).

Step 2: Display only the assembly *assem_1* for elements and geometry.

1. In the **Model** browser, click  (**Model View**).
2. Expand the **Assembly Hierarchy** folder and sub-folders.
3. Set the entity selection to  (**Elements and Geometry**).
Note: This options turns on/off both elements and geometry when you perform right-click operations in the **Model** browser.
4. Right-click on *assem_1* and select **Isolate** from the context menu. HyperMesh only displays the components that are in the *assem_1* assembly.

Step 3: Load the Connector Browser.

1. Open the **Connectors** browser by clicking **View > Browsers > HyperMesh > Connector** from the menu bar.
2. Review the layout of the **Connector** browser. Currently there are no components or connectors listed because there are no connectors in the model.
Note: Use the **Connector** browser to view and manage connectors. The top portion of the browser is referred to as the **Link Entity** browser, and it displays information about linked entities. The middle portion is referred to as the **Connector Entity** browser, and it contains a list of the connectors in your model. The bottom portion of the browser is referred to as the **Connector Entity Editor**, and it displays attributes assigned to the connector(s) selected in the Connector Entity browser. HyperMesh groups the connectors based on their connection type.



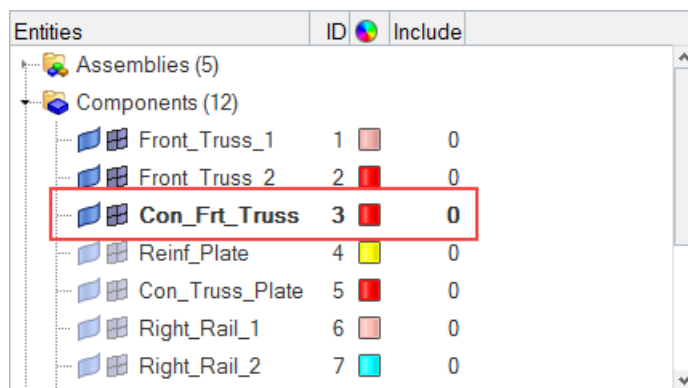
Step 4: Create welds between the geometry for the two front trusses at the pre-defined weld points.

Connectors can be created automatically or manually. The automatic approach creates and realizes connectors automatically. The manual approach allows you to create and realize connectors manually. Realization is the process in which the connector creates the weld entity.

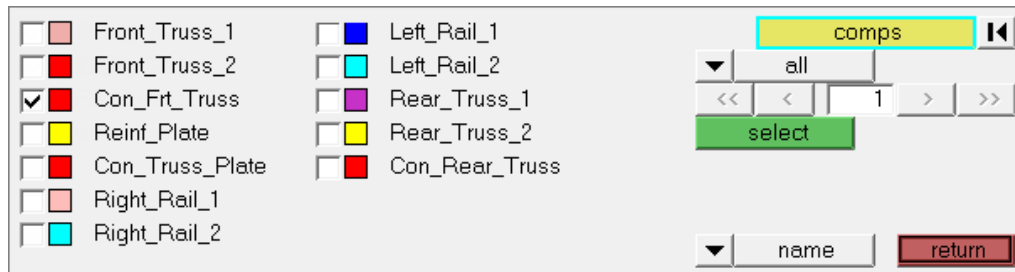
Use the **Spot**, **Bolt**, **Seam**, and **Area** panels to create connectors automatically within the **Connector** browser, and use the **create** and **realize** subpanels to create connectors manually.

1. Verify that the current component is **Con_Frt_Truss**.

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



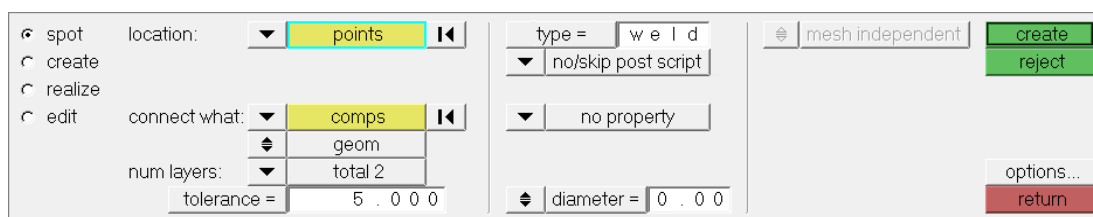
2. Open the **Spot** panel by right-clicking in the **Connector Entity** browser and selecting **Create > Spot** from the context menu.
3. Go to the **spot** subpanel.
4. Set the **location** selector to **points**.
5. Select the six pre-defined weld points by clicking **points >> by collector**.
6. Select the component **Con_Frt_Truss**.



7. Click **select**.
8. Set the **connect what** selector to **comps**.
9. Click **comps**.
10. Select the components **Front_Truss_1** and **Front_Truss_2**.
11. Click **select**.
12. In the **tolerance =** field, enter 5.

Note: The connector will connect any selected entities within this distance of itself.

13. Click **type=** and select **weld**.
14. Under **connect what**, switch the toggle from **elems** to **geom**.

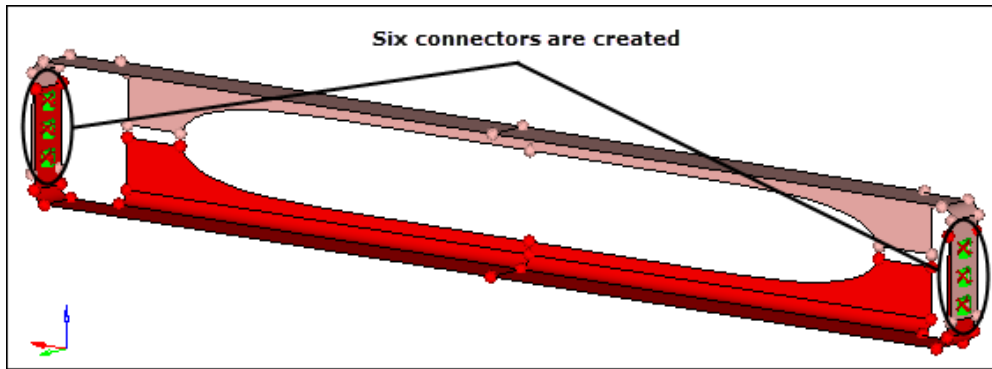


Spot panel settings for steps 4.4 through 4.14.

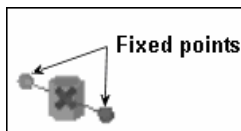
15. Click **create**. HyperMesh automatically creates and realizes six connectors (**Status** bar reads, "6 spot connectors created, 6 realized."), and organizes them as geometry (not elements) in the current component collector, **Con_Frt_Truss**.

Note: Green connectors indicate that the creation of the weld entity was successful.

There are four states of connectors: realized (green ■), unrealized (yellow ■), failed (red ■), and modified (■). If connectors were created manually, the color of the connectors changes from yellow to green, which indicates that they are realized into weld elements. As mentioned above, if you create connectors automatically they will be green immediately as there is no interim unrealized (yellow) state.



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

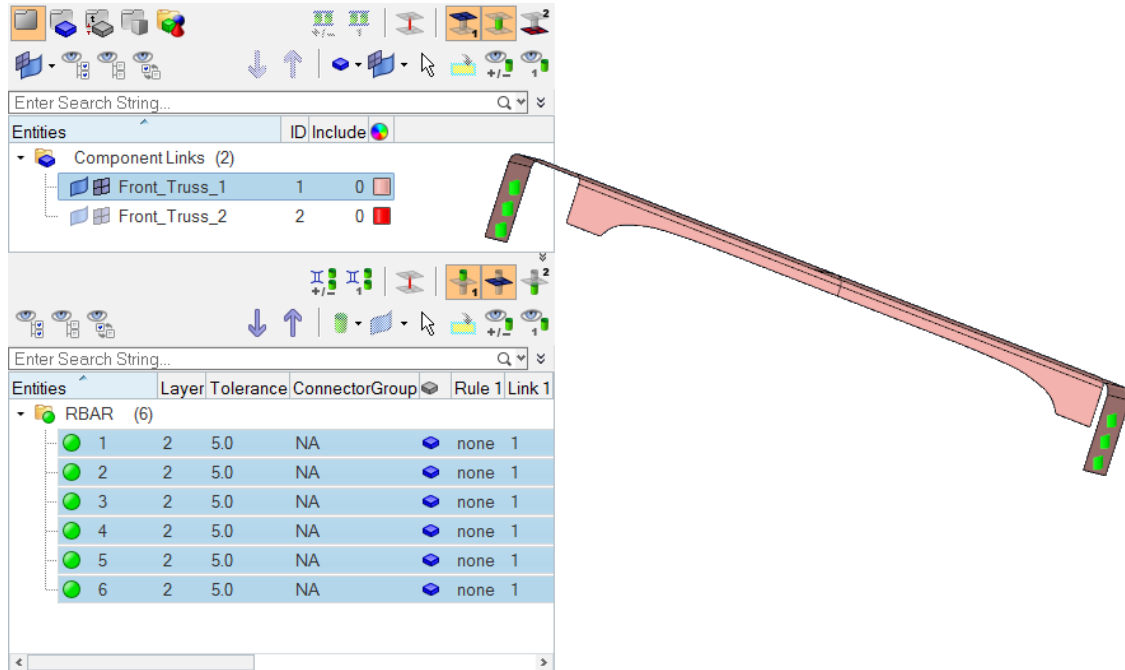
16. Click **return**.

Step 5: Review the Connector Browser.

1. In the **Connector Entity** browser, expand the **RBAR** folder.

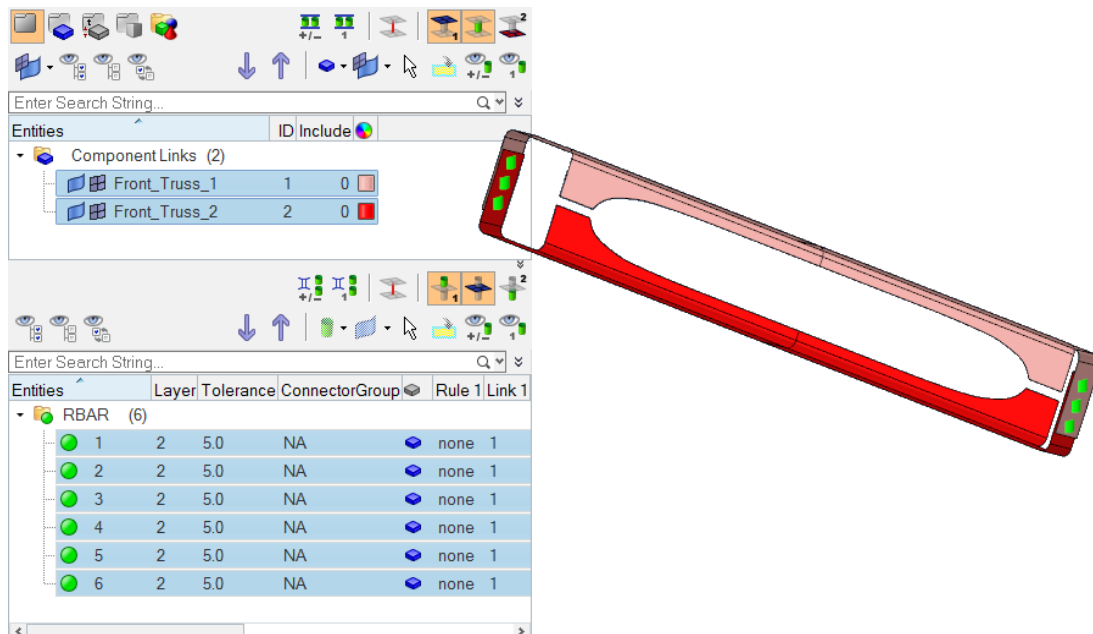
Note: The **RBAR** folder contains the six connectors that you just created. HyperMesh grouped all of them under **RBAR** because that is the type of connector created. Notice the IDs of the connectors in the **Entities** column, the **Links** of the connectors, and the **State** of the connectors. You may need to increase the size of the tab area to see the **State** column.

- In the **Link Entity** browser, right-click on **Front_Truss_1** and select **Find** from the context menu. HyperMesh isolates the component in the graphics area and highlights the six connectors in the **Connector Entity** browser to indicate that these connectors have **Front_Truss_1** as a link.



- Right-click on **Front_Truss_1** and select **Find Attached** from the context menu. HyperMesh finds the components that are attached to **Front_Truss_1** through the connectors.

Note: Front_Truss_1 and Front_Truss_2 are now both highlighted in the **Link Entity** browser, which indicates that they are displayed in the graphics area.




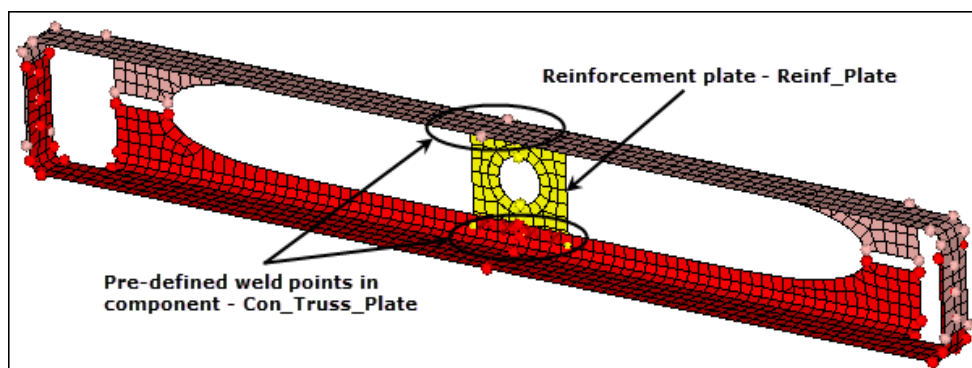
Step 6: Create a shell mesh on the two front truss components.

1. Open the **AutoMesh** panel by pressing **F12**.
2. Go to the **size and bias** subpanel.
3. Set the mesh mode to **automatic**.
Note: It may currently be set to **interactive**.
4. Set the entity selector to **surfs**.
5. Click **surfs** >> **displayed**.
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. Click **mesh**. HyperMesh meshes the surfaces.
10. Zoom into the area with a connector and note how the fixed point created from the weld has ensured that the mesh seeding passes through the weld.
11. Click **return**.

Step 7: Display only the assembly **assem_2** for elements and geometry.

In this step you will display the reinforcement plate that needs to be welded to the two front trusses.

1. In the **Model** browser, set the entity selection to  (**Elements and Geometry**).
2. Right-click on **assem_2** and select **Isolate** from the context menu.



Weld the two front trusses to the reinforcement plate.

Step 8: Create connectors between the shell mesh of the front trusses and the reinforcement plate at pre-defined points.

In this step you will manually create connectors between the shell elements of the front trusses and reinforcement plate at pre-defined weld points.

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

2. Open the **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 the **connect what** selector to **comps**.
9. Click **comps**.
10. Select the components: **Front_Truss_1**, **Front_Truss_2**, and **Reinf_Plate**.
11. Click **select**.
12. Under **connect what**, switch the toggle from **geom** to **elems**.
13. Set the **num layers** to **total 2**.
14. Set **connect when** to **now**.
15. Click **create**. HyperMesh creates eight spot connectors with comp links (**Status** bar reads "8 spot connectors created with comps links.") at the selected weld points, and organizes them into the current component collector, **Con_Truss_Plate**.

Note: In the **Connector Entity** browser, these eight connectors are currently grouped as **undefined**.

16. In the **Connector Entity** browser, expand the **undefined** folder.

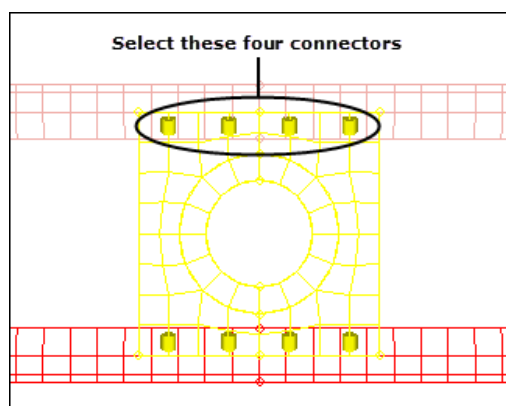
Note: The connectors in this folder are colored yellow, which indicates that they are unrealized.

Step 9: Realize the connectors in the component Con_Truss_Plate into weld elements.

In this step you will realize the undefined, unrealized connectors and assign them a connector type.

1. In the **Connector Entity** browser, select the following unrealized connectors: 7, 8, 11, and 12.

Hint: These connectors are displayed along the top of the Reinf_Plate component.



2. In the **Entity Editor**:

- Set **Config Name** to *weld*.
- For **Tolerance**, enter 7.
- Set **Connectivity** to *mesh dependent*.
- Set **Adjust Option** to *adjust realization*.
- Set **Adjust Realization** to *project and find nodes*

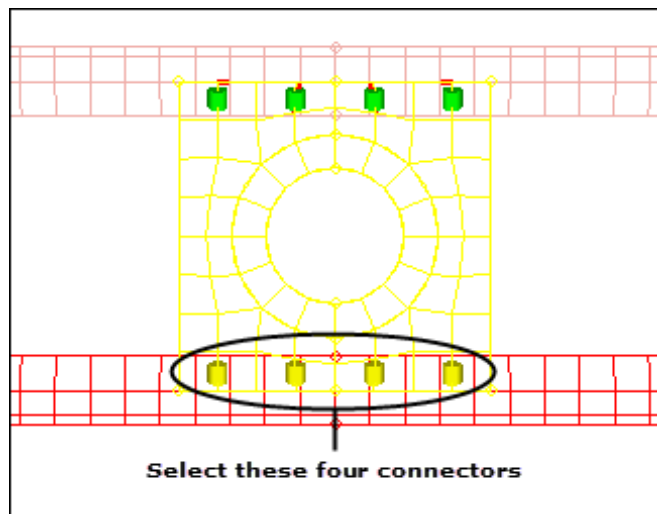
Note: When this option is active, the nodes will be equivalenced if the realized finite element of the connector is coincident to a node of the shell mesh it is being connected to. If there are no suitable nodes present, the closest node of the element the projection is landing is connected.

3. In the **Connector Entity** browser, right-click on the selected connectors and select **Rerealize** from the context menu. The selected connectors realize into the weld elements.

Note: The mesh has not been remeshed to connect the two components.

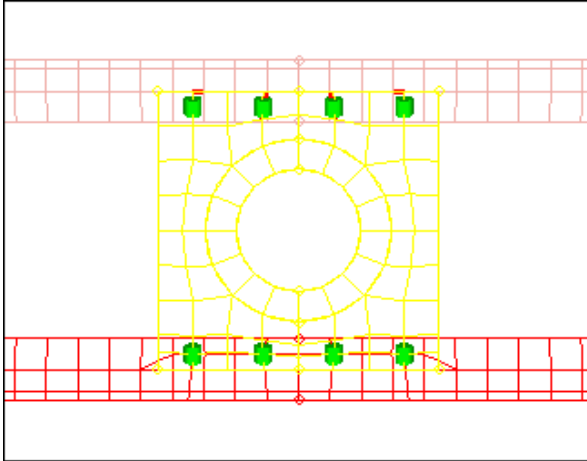
4. In the **Connector Entity** browser, select the four remaining unrealized connectors (9, 10, 13, 14).

Hint: These connectors are displayed along the bottom of the Reinf_Plate component.

5. In the **Entity Editor**:



- Set **Config Name** to *weld*.
- For **Tolerance**, enter 7.
- Set **Connectivity** to *mesh dependent*.
- Set **Adjust Option** to *adjust mesh*.
- Set **Adjust mesh** to *remesh*.

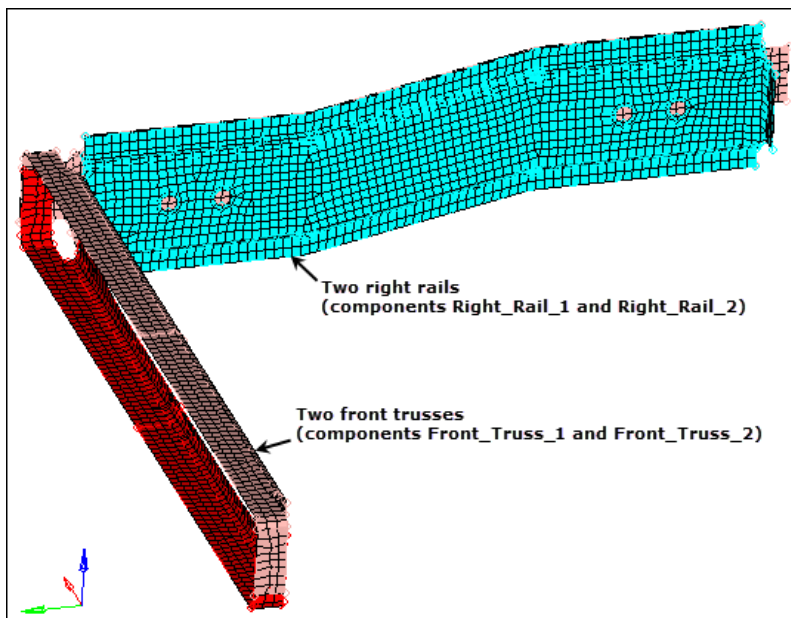
6. In the **Connector Entity** browser, right-click on the unrealized connectors and select **Rerealize** from the context menu. HyperMesh realizes the selected connectors, and remeshes the mesh to connect the two components.



The two front trusses welded to the reinforcement plate with weld elements at the connectors


Step 10: Display only the assembly *assem_3* for elements and geometry.

1. In the **Model** browser, set the entity selection to  (**Elements and Geometry**).
2. On the **Visualization** toolbar, click  to shade your model's elements and mesh lines.
3. Right-click on *assem_3* and select **Isolate** from the context menu.

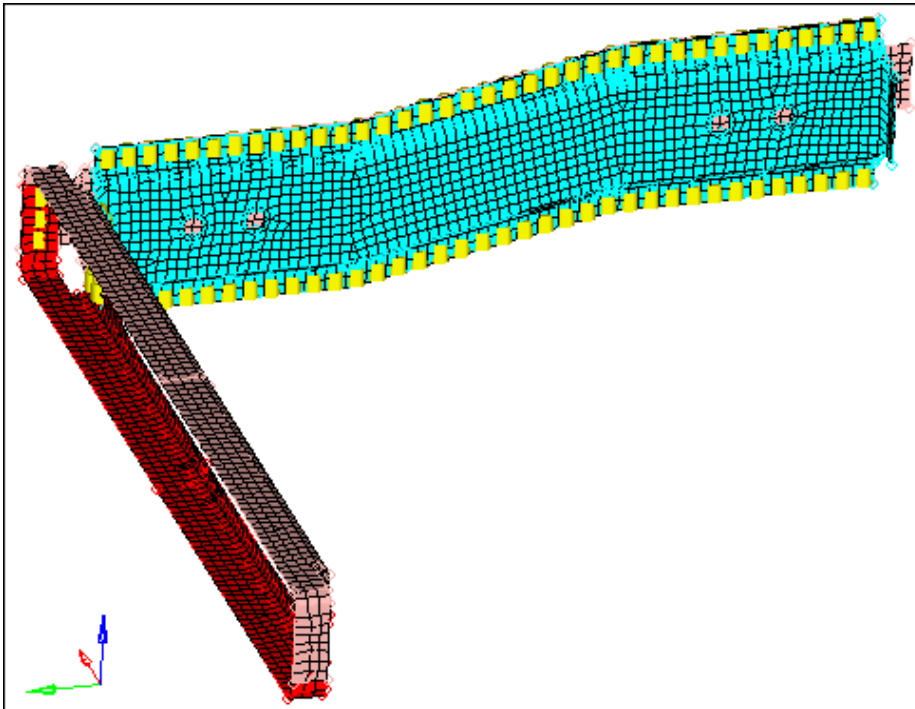


Weld the two right rails to each other and to the two front trusses by creating connectors from a master connectors file.

Step 11: 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. In the **Import** tab, click  in the **File** field.
3. In the **Select connector file** dialog, open the `rails_frt_truss.mwf` file.
4. Click **Import**. HyperMesh imports connectors and organizes them into a new component, **CE_Locations**.

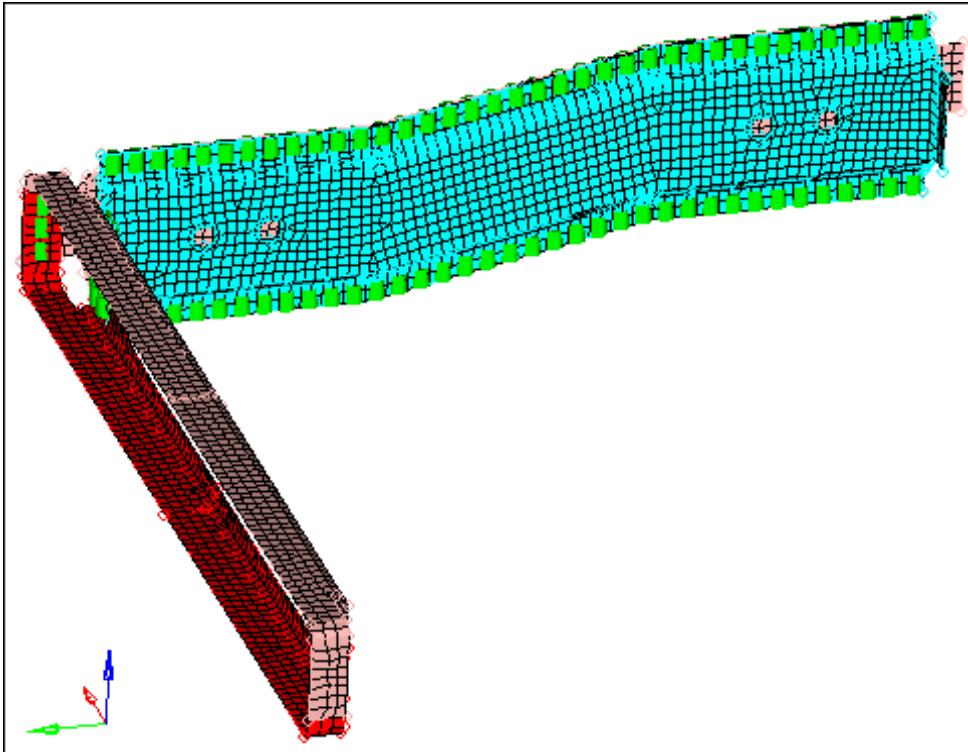
Note: It will take a few seconds for HyperMesh to import the connectors.



Step 12: Realize the connectors in the component **CE_Locations** into weld elements.


1. In the **Model** browser, **Component** folder, right-click on **CE_Locations** and select **Make Current** from the context menu.
2. In the **Connector Entity** browser, click the **undefined** folder. The **Entity Editor** opens and displays the undefined connector's corresponding data.
3. In the **Entity Editor**:
 - Set **Config Name** to **Weld**.
 - For **Tolerance**, enter 7.
 - Set **Connectivity** to **mesh dependent**.
 - Set **Adjust Option** to **adjust realization**.
 - Set **Adjust Realization** to **project and find nodes**.

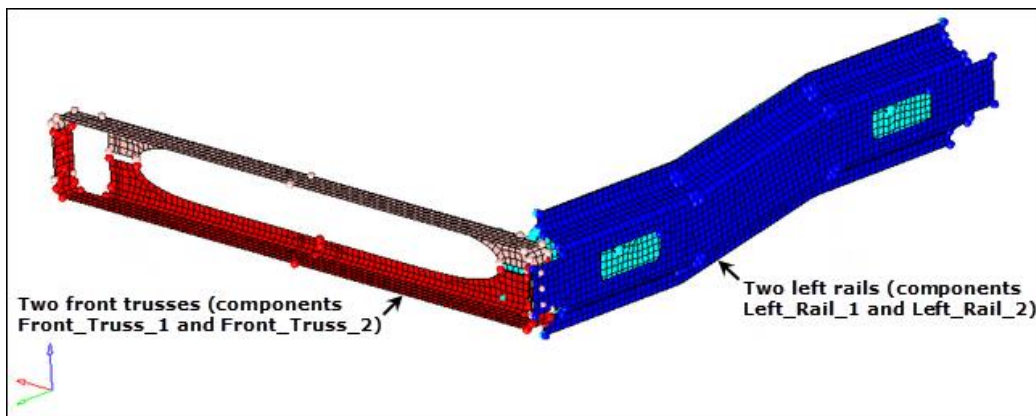
4. In the **Connector Entity** browser, right-click on the selected connectors and select **Rerealize** from the context menu.



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

Step 13: Display only the assembly *assem_4* for elements and geometry.

1. In the **Model** browser, set the entity selection to  (**Elements and Geometry**).
2. Right-click on *assem_4* and select **Isolate** from the context menu.




Weld to the two front trusses by duplicating and reflecting selected connectors created from the master connectors file

Step 14: Create a new component collector to hold new connectors.

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

Note: HyperMesh makes this new component, the current component.

2. For **Name**, enter CE_Locations_Dup.
3. Click the **Color** icon, and select a color.

Name	Value
Name	CE_Locations_Dup
ID	14
Color	
Include File	[Master Model]
Property	<Unspecified>
Material	<Unspecified>

Step 15: Duplicate the connectors created from the master connectors file and reflect them.

1. In the **Model** browser, **Component** folder, right-click on **CE_Locations** and select **Show** from the context menu.
2. Open the **Reflect** panel by clicking **Connectors** > **Reflect** > **Connectors** from the menu bar.
3. Set the entity selector to **connectors**.
4. Click **connectors** >> **by collector**.
5. Select **CE_Locations**.
6. Click **select**.
7. Click **connectors** >> **duplicate** >> **current comp**. HyperMesh duplicates the displays the connectors, and 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. Specify a **base** node to reflect about by double-clicking **B**.
10. Click **x=**. HyperMesh activates the **x=**, **y=**, and **z=** fields.

Note: Their values are all 0.000 by default, which is the base point you want to reflect about.

11. Return to the **reflect** panel by clicking **return**.
12. Click **reflect**. HyperMesh reflects the connectors.
13. Click **return**.

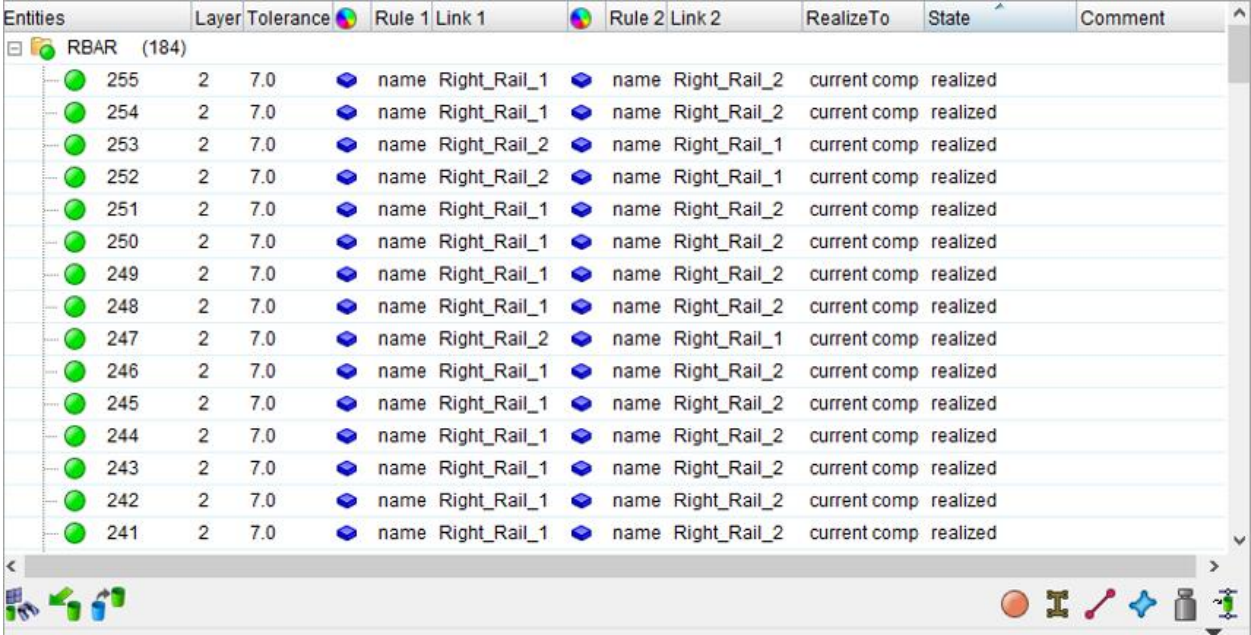
Note: The connectors are yellow, which indicates that they are unrealized.

Step 16: Update the connectors for the left rails to link them to the left rail components.

1. In the **Connector Entity** browser, expand the **RBAR** folder.
2. Sort the connectors by their state by clicking **State**. HyperMesh organizes all of the realized connectors at the top of the list.
Tip: You may need to increase the size of the tab area to see the **State** column.
3. Click **State** again. HyperMesh organizes all of the unrealized connectors at the top of the list.
4. In the **Link1** and **Link2** columns, review the the unrealized connectors.

Note: Some of the connectors are linked to the components, **Right_Rail_1** and **Right_Rail_2**. This data is from the `rails_frt_truss.mwf` file that you imported. These links need to be updated to reflect the components, **Left_Rail_1** and **Left_Rail_2**.

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



Entities	Layer	Tolerance	Rule 1	Link 1	Rule 2	Link 2	RealizeTo	State	Comment
RBAR (184)									
255	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	
254	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	
253	2	7.0	name	Right_Rail_2	name	Right_Rail_1	current comp	realized	
252	2	7.0	name	Right_Rail_2	name	Right_Rail_1	current comp	realized	
251	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	
250	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	
249	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	
248	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	
247	2	7.0	name	Right_Rail_2	name	Right_Rail_1	current comp	realized	
246	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	
245	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	
244	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	
243	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	
242	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	
241	2	7.0	name	Right_Rail_1	name	Right_Rail_2	current comp	realized	

6. In the **Entities** column, right-click on the selected connectors and select **Update Link** from the context menu.
7. In the **Update** window, **Search** column, set the **Link Type** to **comps**.
8. In the **Search** column, click the **Link Select** field.
9. In the panel area, click **component**.
10. Select the component, **Right_Rail_1**.
11. Click **proceed**. HyperMesh inserts **Right_Rail_1** into the **Link Select** field.
12. In the **Replace** column, set the **Link Type** to **comps**.
13. In the **Replace** column, click the **Link Select** field.

14. In the panel area, click **component**.
15. Select the component, **Left_Rail_1**.
16. Click **proceed**. HyperMesh inserts **Left_Rail_1** into the **Link Select** field.

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

17. Click **Update**. HyperMesh updates the connectors' links.
18. Repeat 16.7 through 16.17, except search for the **Right_Rail_2** component and replace it with the **Left_Rail_2** component.
19. Scroll through the list of unrealized connectors to make sure that none of the connectors are linked to the right rail components.
20. Close the **Update** window by clicking **X** next to **Update**.


Step 17: Realize the connectors in the component **CE_Locations_Dup** into weld elements.


1. In the **Connector Entity** browser, select all of on the unrealized connectors, which are organized in the **CE_Locations_Dup** component. The **Entity Editor** opens and displays selected connector's common attributes.
2. In the **Entity Editor**:
 - Set **Config Name** to **weld**.
 - For **Tolerance**, enter 7.
 - Set **Connectivity** to **mesh dependent**.
3. In the **Connector Entity** browser, right-click on the selected unrealized connectors and select **Rerealize** from the context menu. HyperMesh realizes the selected connectors into weld elements.

Step 18: Verify that all connectors are realized and identify the pairs of adjacent connectors.

1. In the **Connector Entity** browser, expand the **RBAR** folder.
2. From the **State** column, verify that all of the connectors are realized.
3. Zoom into one of the two areas where the front trusses are connected to the rail components.

Note: At these two areas, there are pairs of adjacent connectors.

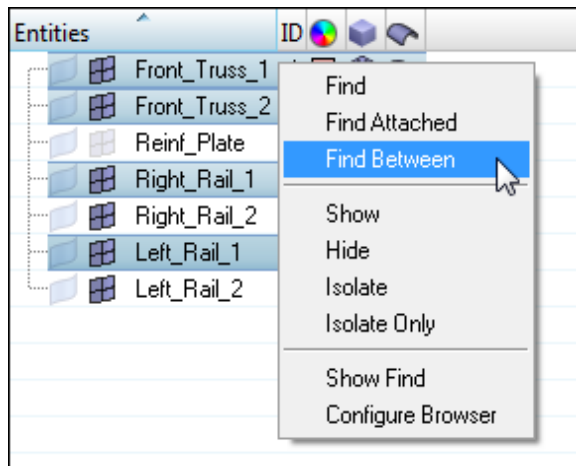
4. On the **Visualization** toolbar, click  (**Visualization Options**).

5. In the **Visualization** tab, click  (**Connectors**).
6. Under **Color by**, select **Layer**. HyperMesh changes the connectors color to purple because under **Layers, 2t** 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 19: 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 **Link Entity** browser, 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 12 connectors between the four components you selected.



Step 20: Unrealize the displayed connectors.

1. Open the **Unrealize** panel by clicking **Connectors > Unrealize** from the menu bar.
2. Click **connectors >> displayed**.

Note: The **Status** bar reads "12connectors added by 'displayed'. Total selected 12."



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

Step 21: Combine the pairs of adjacent 2t connectors into 3t connectors.


1. Open the **Connector Quality** panel by clicking **Connectors > Check > Connector Quality** from the menu bar.

2. Go to the **connectors (unrealized)** subpanel.
3. Click **connectors >> displayed**.
4. In the **tolerance =** field, enter 7.
5. Click **preview combine**. HyperMesh finds 12 connectors that need to be combined.
Note: The **status** bar reads, "12 connector(s) found that need to be combined."
6. Click **combine**. HyperMesh combines the connectors, and displays them in a dark blue color to indicate that they have three layers.
Note: The **status** bar reads, "6 connectors deleted."
7. Optional: If the connectors are not visible, right-click in the **Link Entity** browser and select **Find Between**.
8. Click **return**.

Step 22: Realize the 3t connectors in the component *Con_Frt_Truss* into weld elements.

1. In the **Model** browser, **Component** folder, right-click on **Con_Frt_Truss** and select **Make Current** from the context menu.
2. In the **Connector Entity** browser, select the 6 unrealized connectors. The **Entity Editor** opens and displays the selected connector's common attributes.
3. In the **Entity Editor**:
 - Set **Config Name** to **weld**.
 - For **Tolerance**, enter 10.
 - Set **Connectivity** to **mesh dependent**.
4. In the **Connector Entity** browser, right-click on the selected connectors and select **Rerealize** from the context menu. HyperMesh realizes the connectors.
5. Verify that there are now three links for the six connectors you just updated.
6. On the **Visualization** toolbar, click  (**Visualization Options**).
7. In the **Visualization** tab, click  (**Connectors**).
8. Under **Color by**, select **State**.

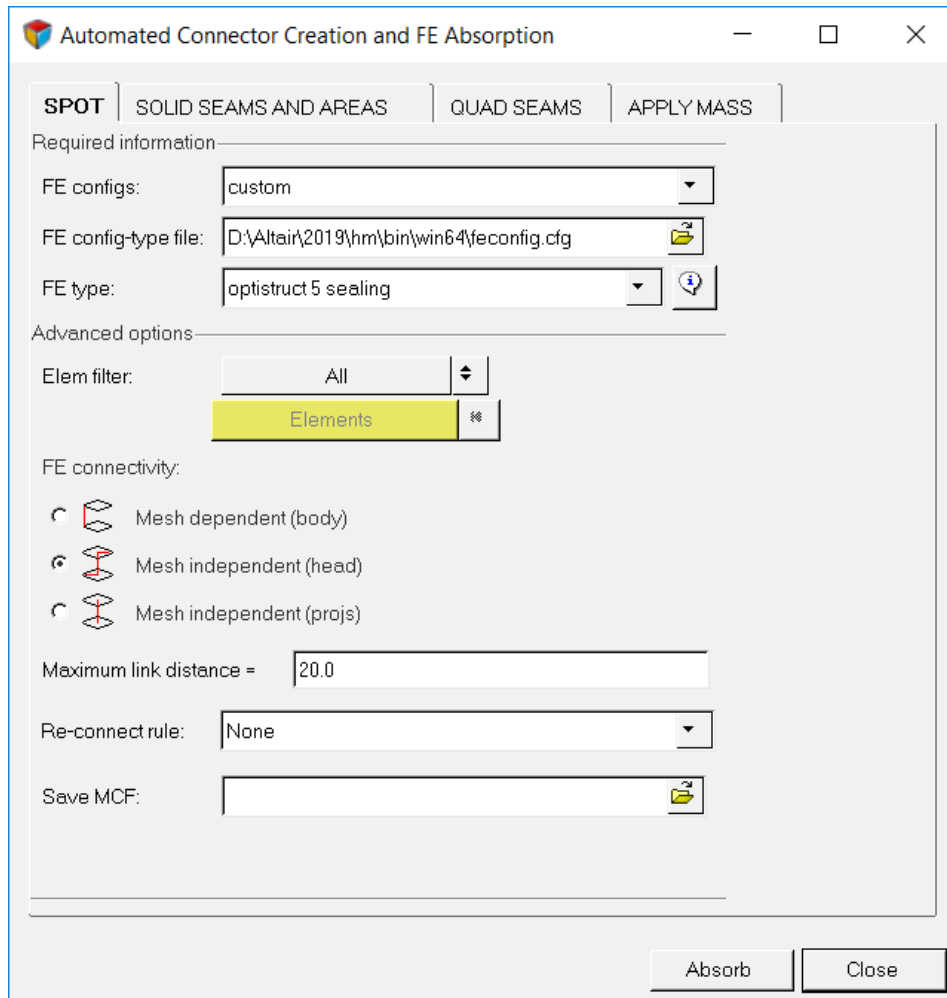
Step 23: Display only the assembly *assem_5* for elements and geometry.

1. In the **Model** browser, set the entity selection to  (**Elements and Geometry**).
2. Right-click on **assem_5** and select **Isolate** from the context menu.

Step 24: Create connectors from existing ACM welds.

In this step you will use the **Fe Absorb** tool to obtain connectors from the existing ACM welds (elements) in the component, **Con_Rear_Truss**.

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 **optistruct 5 sealing**.
4. Toggle the **Elem filter** from **All** to **Select**.
5. Double-click **Elements**.
6. In the panel area, click **elems >> by collector**.
7. Select the component, **Con_Rear_Truss**.
8. Click **select**.
9. Click **proceed**.
10. Select the **Move connectors to FE component** checkbox.
11. Click **Absorb**. HyperMesh absorbs the elements into realized, 2t connectors at the locations of the ACM welds, and organizes them into the **Con_Rear_Truss** component, the same component to which the ACMs belong.



12. Click **Close**.