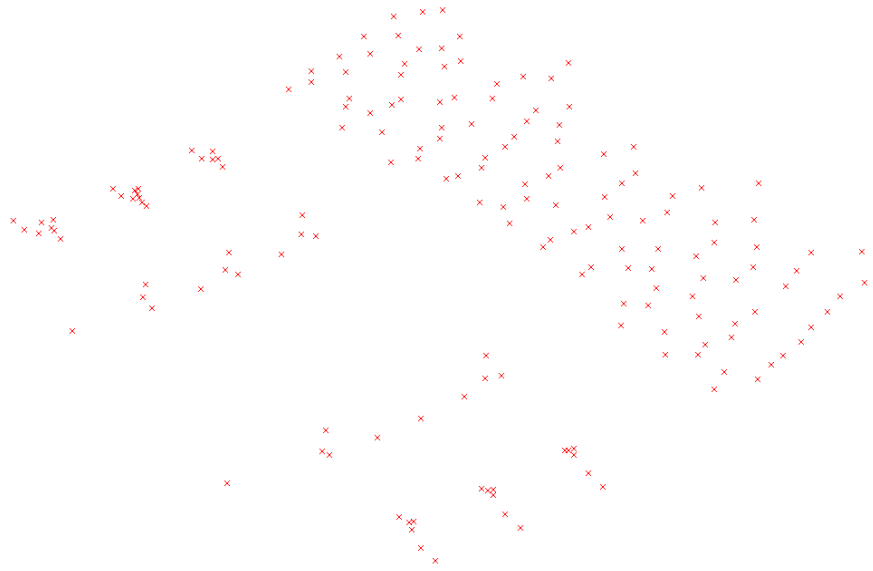
Exercise 8a - Creating Connectors on an Airplane Fuselage

This exercise will cover the basics behind the creation and visualization of connector entities, which will be used to create a variety of weld definitions. It will also demonstrate the strength of connectors when having to update the model with a new part.

Step 1: Load the model `8a-CompositeSkin_Connectors_Start.hm` and set the user profile to OptiStruct.

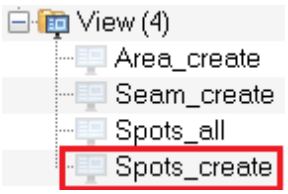
Step 2: Review the points that will be used to make connectors

1. Using the **Model Browser**, isolate the component **Spot connector points**.



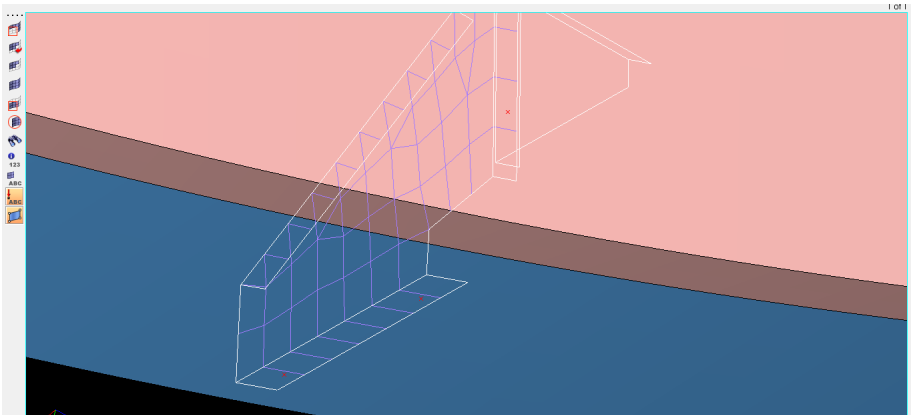
Step 3: Create a spot connector.

1. From the **Model Browser** find the predefined View entities and click on the ***Spots_create*** icon. This will snap the model to a predefined view where we will define some spot connectors.



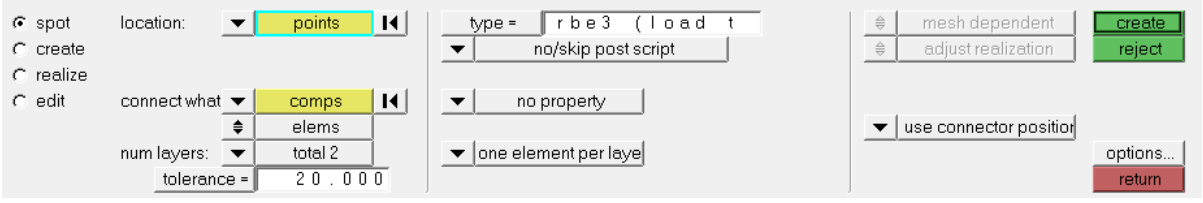
2. Make sure that the geometry icon is turned on next to the **Spot connector points** component.

3. Within the **Model Browser**, select the ***Component View***,  .
4. Locate the component **t2.5** (the components are listed alphabetically) and left click on the shaded mesh icon,  , and select the wireframe mesh icon,  .



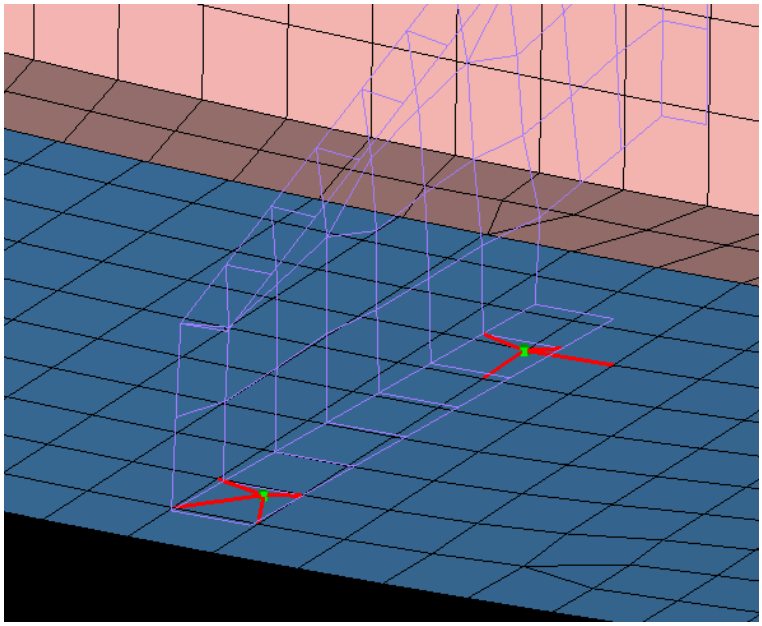
This should allow you to visualize the red points under this mesh, which designate the location of the connectors we want to make.

5. Now from the menu bar, select ***Connectors > Create > Spots.***
6. Navigate to the **Spot connector points** component in the **Model Browser**, right click on it, and then select ***Make Current***. Making this component current will cause any newly created elements to be placed in this collector.
7. Now, within the **Spots** panel, change the **location** to ***points*** and then select the two red points on the screen.
8. For **connect what**, click on ***comps*** and select the components ***t2.5*** and ***compositeSkin*** (these are the blue and purple components in the graphics area).
9. Set the **tolerance** to `20`, the **type** to ***rbe3 (load transfer)***, and then select the ***use connector position for center*** option.



10. When you are done with your selection click ***create*** to both create a connector and realize it into its FE representation.

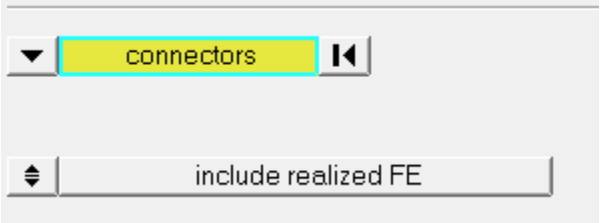
    You should now see the newly created connectors with their associated FE.



Step 4: Create all of the spot connectors at once.

Now that you walked through the process of making a spot weld connector, we are going to delete those connectors and then realize all of the spot connectors in this model at once.

1. Open the **Delete** panel and select all of the connectors in the model. Note that within the **Delete** panel it gives you an option for connectors. Choose the ***include realized FE*** option to delete the FE that was realized from these connectors.

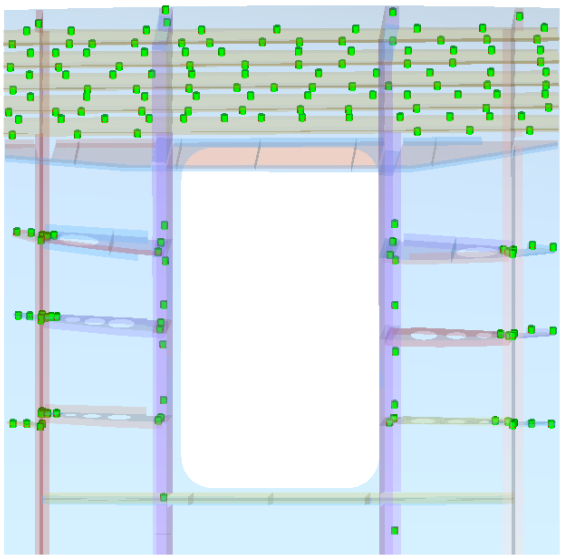


2. Click the ***f*** key to zoom out and fit the model in the viewing area.
3. Now navigate back to the **Spot Create & Realize** panel.
4. For **location**, select ***points >> by collector***.
5. Select the ***Spot connector points*** component.

   This will select all of the points in that component.
6. Then select all of the components in the model under the **connect what** field. This is done easily by click on ***comps*** and then clicking on ***comps*** in the upper right corner of the panel (shown below) and selecting ***all***:

7. Like before set the **tolerance** to `20`, the **type** to ***rbe3 (load transfer)***, and then select ***the use connector position for center*** option.
8. Then click **create**.

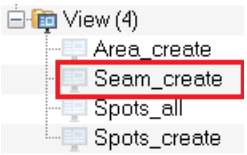
   This operation should have created and realized 176 connectors.



Step 5: Create a seam connector from FE entities.

Now that we created spot welds, we are going to move on to seam connectors. These types of connectors are used to represent welds/connections that are continuous along a line. A MIG weld is a good example of where you might want to use a seam connector.

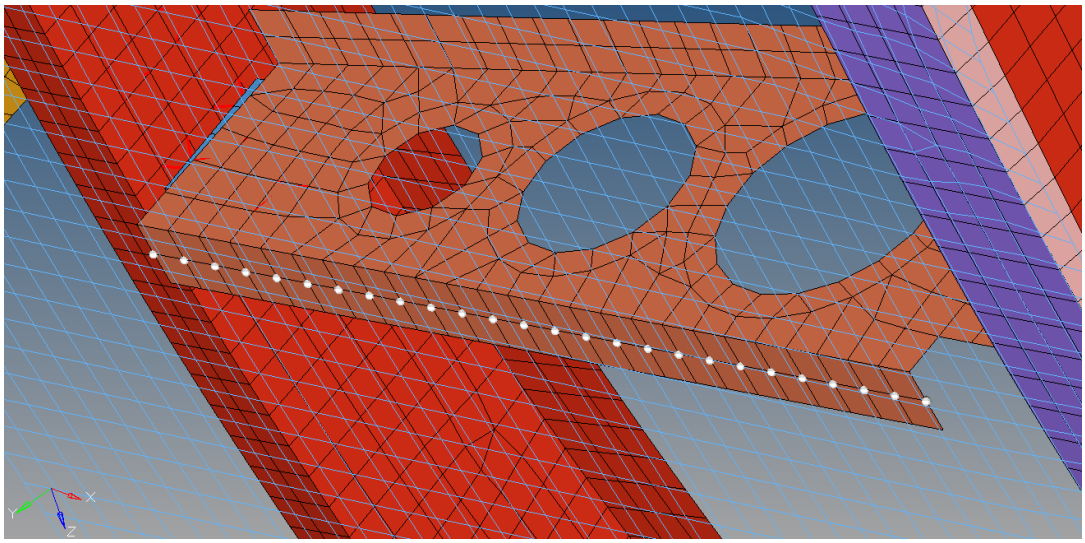
1. From the **Model Browser** find the predefined **View** entities and click on the **Seam_create** icon. This will snap the model to a predefined view where we will define some spot connectors.



2. Make the **Seam connector** component the current component and verify that the geometry and elements display are on.

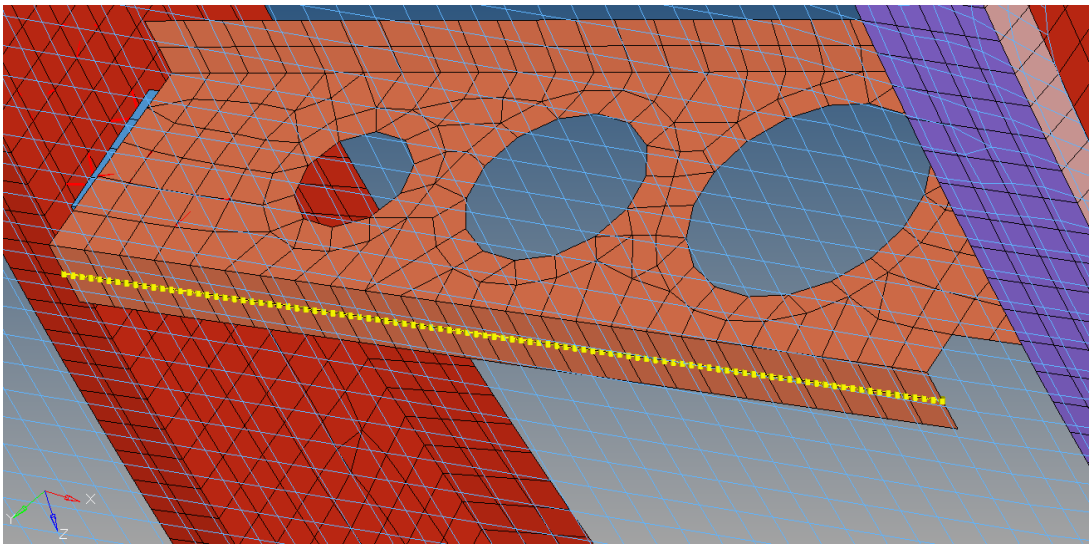
3. Within the **Model Browser**, select the ***Component View***,  .
4. Locate the component **compositeSkin** and left click on the shaded mesh icon,  , and select the wireframe mesh icon,  .
5. Now from the menu bar select ***Connectors > Create > Connectors > Seams***. This time we will just make the connector and not realize it into FE.
6. For **location**, change the selection option to ***nodelist***.
7. Now select the nodes down the center of the **t3:0** component as shown in the screenshot below.

   Hint: use the **by path** option to select the nodes faster. Be sure not to select any nodes on the **compositeSkin**.



8. For **spacing**, input a value of `5`.
9. For **connect what**, select the two components to connect (***t3:0*** and ***compositeSkin***), and then set the **tolerance** to `20`.
10. Then click ***create***.

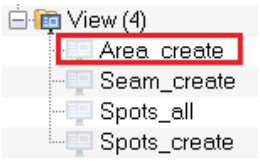
    You'll notice that a yellow seam connector was created. The color yellow indicates that it has not yet been realized into its representative FE.



Step 6: Create an area connector from elements.

Now we will make a connector from elements in the model. An area connector represents an area where a connection is needed; an adhesive connection is a good example of this.

1. From the **Model Browser**, within the **Model View**, find the predefined **View** entities and click on the **Area_create** icon. This will snap the model to a predefined view where we will define some spot connectors.

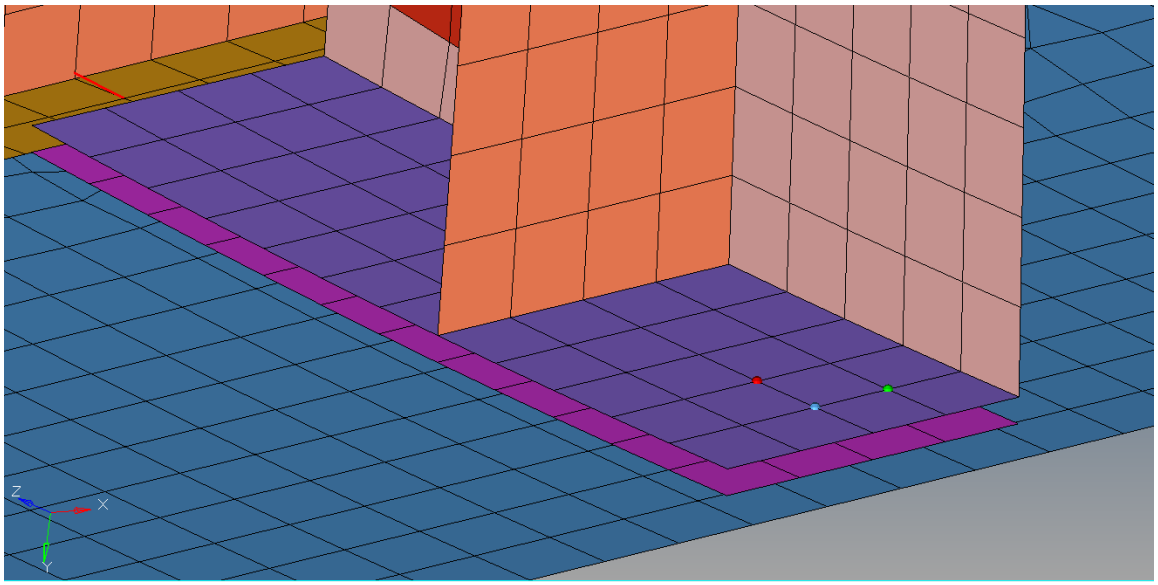


2. Make the **Adhesive connector** component current and turn on the display of geometry.
3. Now navigate to the **Translate** panel from ***Mesh > Translate > Elements***.
4. Select a few elements in the purple **t8:0** component and then select ***elems>>by face***.

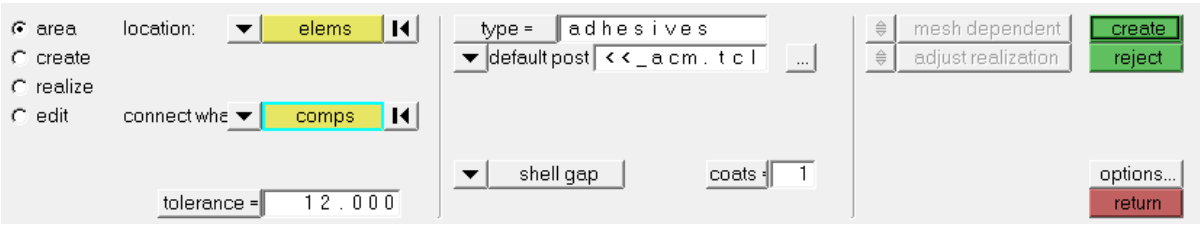
   This should select all of the elements that share the same face.
5. Now select the ***elems*** button again and select the ***duplicate*** advanced option.
6. Put these duplicate elements into the ***current component***.

   This operation makes duplicate elements of the original selected elements.
7. Now use the ***N1, N2, N3*** buttons to define a vector to translate these elements downward. These buttons follows the right hand rule.
8. Set the **magnitude** to `6` and then select ***translate +.***

   When you are down you should have an extra group of elements located in between the two components of interest, see the screenshot below.

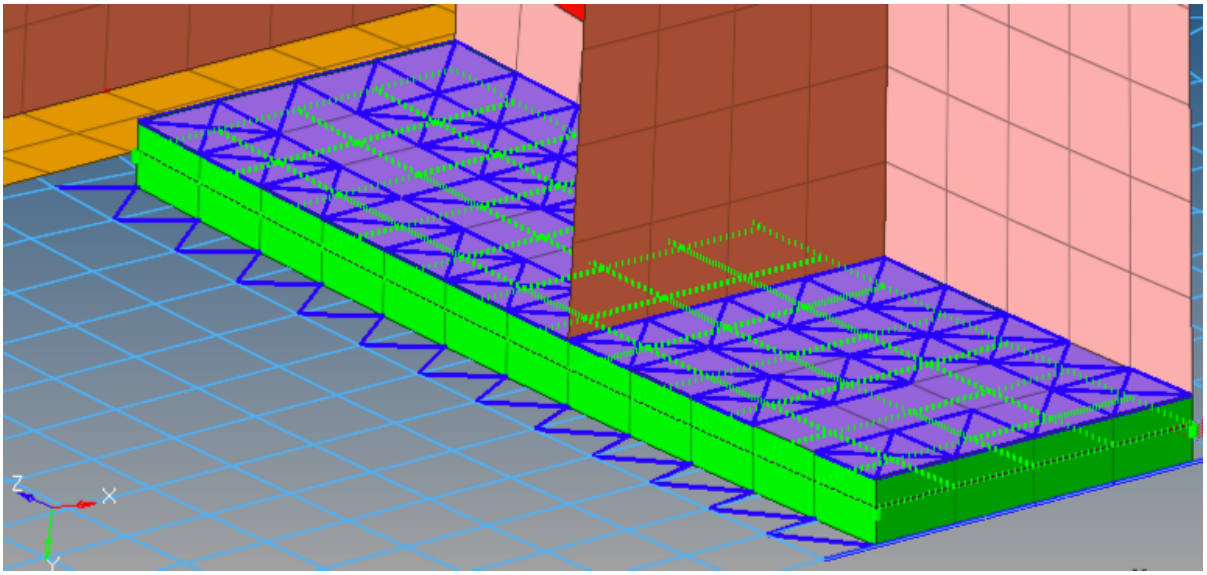


9. Now from the menu bar select ***Connectors > Create > Areas***.
10. For **location**, select elements ***by collector***, and select the newly created elements we just translated which are in the ***Adhesive connector*** component.
11. For **connect what**, select the two adjacent components (***compositeSkin*** and ***t8:0***).
12. Set the **tolerance** to `12`, and then pick the **type** as ***adhesives***.



13. Then click ***create*** to create both the connector and the FE.

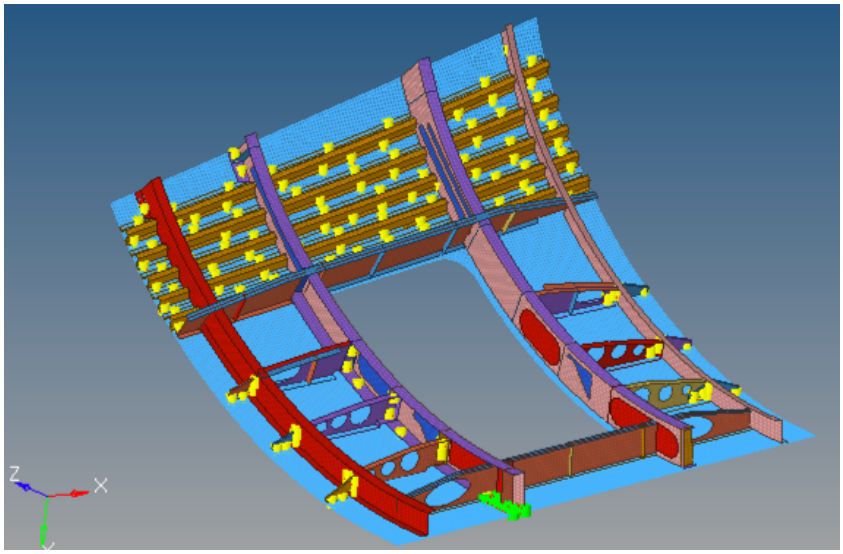
    The final adhesive should look like this:



Step 7: Replace the stringers with an updated design.

A lot of the power of connectors is in part replacement. Without connectors, when a design update is provided, the FE engineer might have to make dozens or even hundreds of modifications to the connected locations, which is very tedious. In the next section of this exercise, we are going to replace a part in our model with a new part that has both an updated mesh and geometry. Without connectors, this would be a very arduous process.

1. From the menu bar select ***Connectors > Unrealize***.
2. Now select ***connectors >> by collector*** and then select the **Spot connector points** component, and click ***unrealize***.



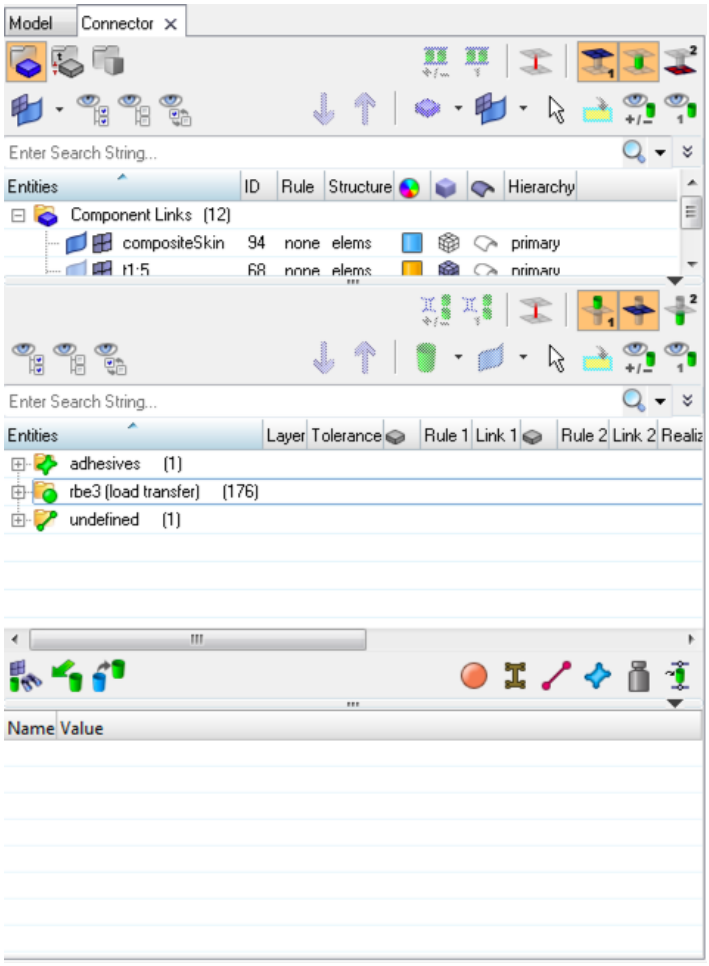
All of the FE for the spot welds has now been deleted, and only the connector entities remain. The connectors should turn from green to yellow.

3. Select ***Import Model***, browse for the file `8a-NewPart.fem`, and import the model.

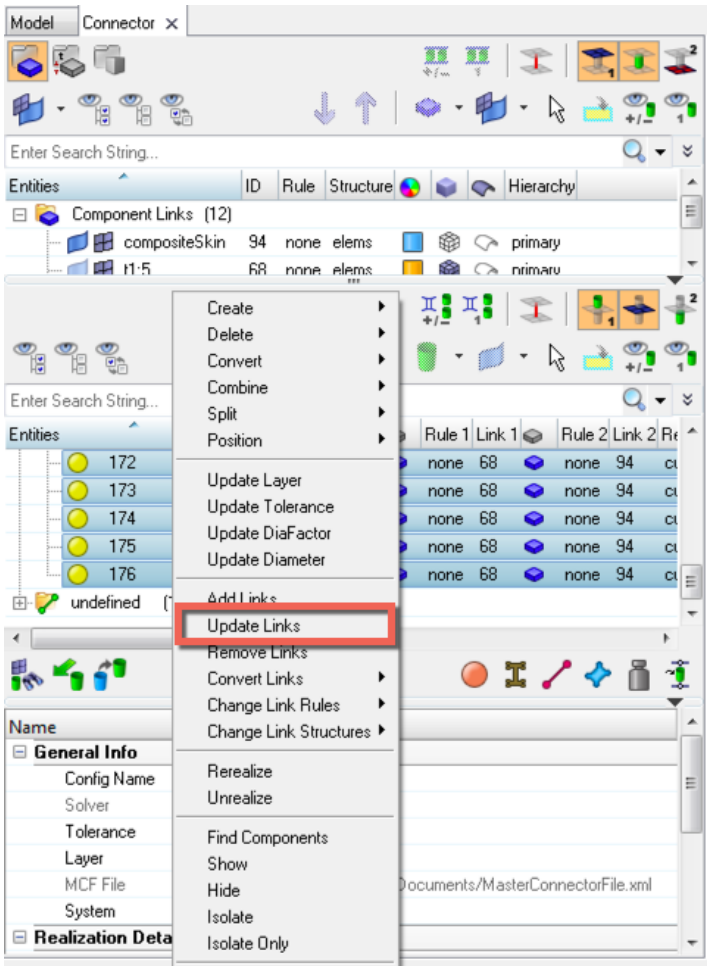
This is the new part (component **t1.5.1**) we are going to be replacing the old part (component **t1.5**) with.

4. From the menu bar select ***View > Browsers > HyperMesh > Connector.***

   This will open the Connector Browser in the tab area.



5. In the Connector Browser, expand the **rbe3 (load transfer)** folder.
6. Using the shift button, select all of the connectors in the folder.
7. Right click on one and select ***Update Links***.



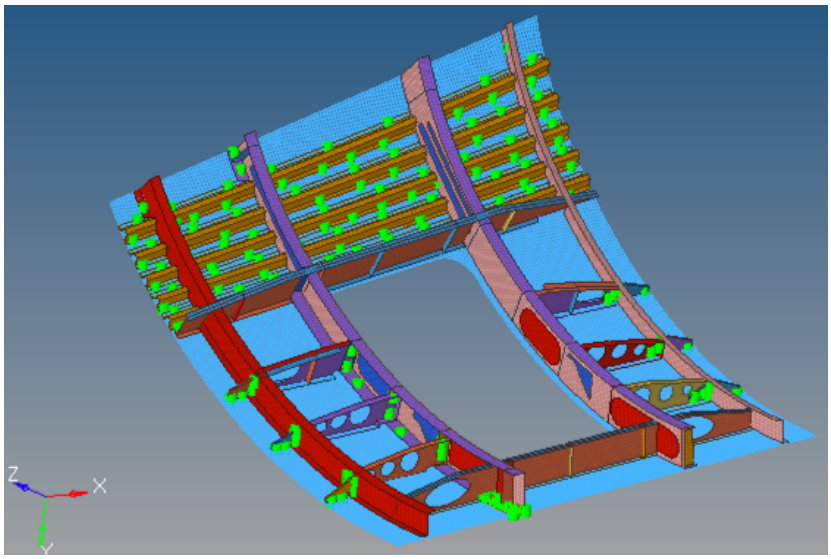
8. In the table, set the **Link Type** for **Search** and **Replace** to ***comps***.
9. Then set **Link Select** for **Search** to the component ***t1.5*** and then **Link Select** for **Replace** to the component ***t1.5.1***.

Name	Search	Replace
Link Type	comps	comps
Link Select	(68) t1:5	(102) t1:5.1
Link State		
Link Rule		

10. Click ***Update*** just above the table.

    This updates the selected connectors which have a link set to the component t1.5 so that they now have a link to the new part, component t1.5.1.

11. While the connectors are still selected in the Connector Browser, right click on them and select ***Rerealize***. If they are no longer selected, simply select all the rbe3 (load transfer) connectors again.



12. Use the **Delete** panel to delete the old component ***t1.5*.**
13. Save the model.