Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-3430: Part Replacement Through Connectors

HM-3430: Part Replacement Through Connectors

In this tutorial, you will learn how to:

- Replace the rear truss component, **Rear_Truss_1**, with a new, similar part and then update the affected connectors.
- Export the connector information
- Export the FE deck and view the connector information in the deck

After the modeling of the assembly is complete, a design change might be made to any one of the parts. When this occurs, you must replace the current part(s) in the model with the new, similar one(s) and update the affected connections (welds).

Model Files

This exercise uses the `frame_assembly_3.hm` file, which can be found in the `hm.zip` file. Copy the file(s) from this directory to your working directory.

Exercise

A new part is needed in the assembly. In this tutorial you will learn how to delete the original component, import a new part, and update the connections. You will also export the connector information to a single file, and then export the entire FE input deck and observe how the connector information is preserved.

Step 1: Retrieve and view the model file.

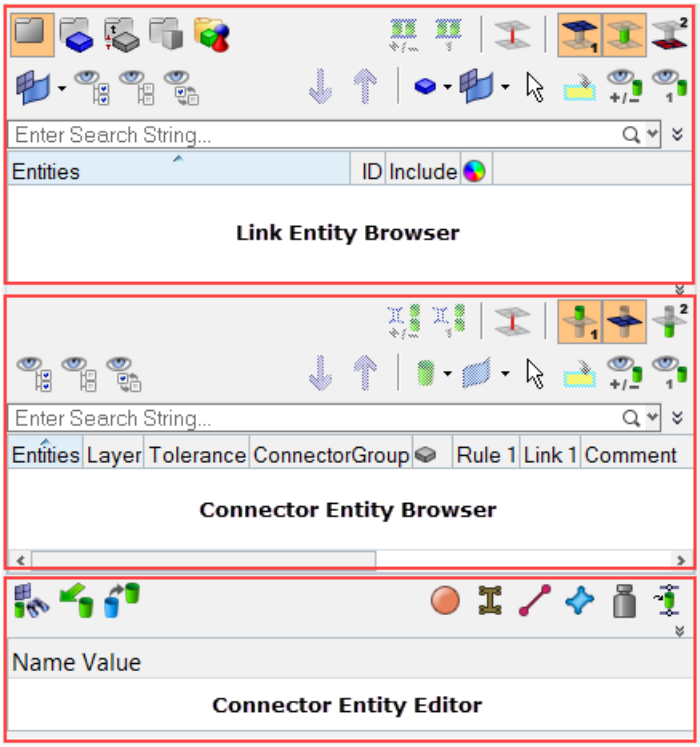
1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, select **OptiStruct**.
3. Click ***OK***.
4. Open a model file by clicking ***File*** > ***Open*** > ***Model*** from the menu bar, or clicking  on the **Standard** toolbar.
5. In the **Open Model** dialog, open the `frame_assembly_3.hm` file. A model appears in the graphics area.
6. Observe the model using various visualization options available in HyperMesh (rotation, zooming, and so on).

Step 2: Load the Connector Browser.

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

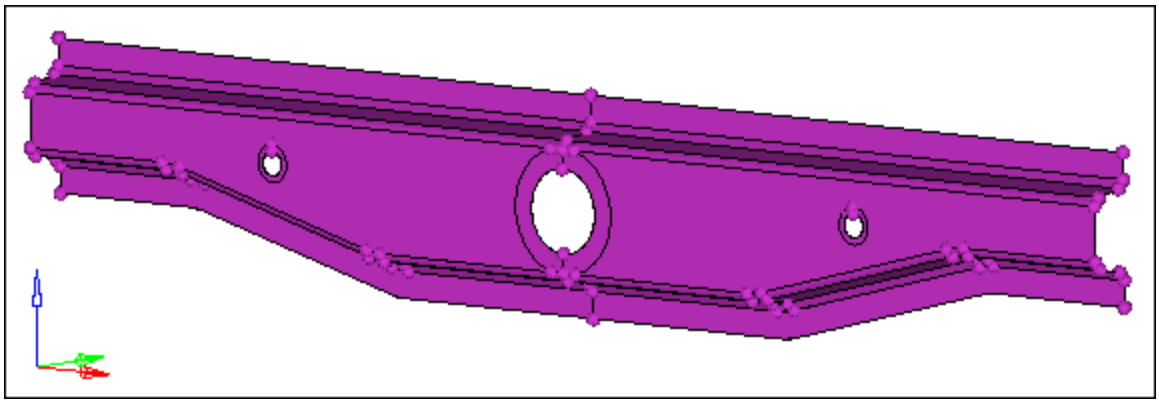
2. Review the layout of the **Connector** browser. Currently there are no components or connectors listed because there are no connectors in the model.

   Note: You can use the **Connector** browser to view and manage the connectors in your model. The top portion of the browser is referred to as the **Link Entity** browser, and it displays information about the linked entities in your model. 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.

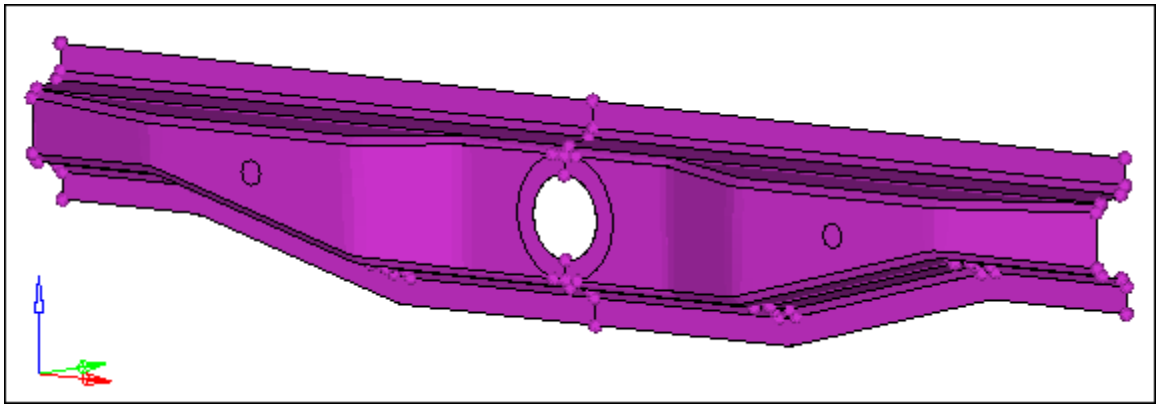


Step 3: Import rear_truss_1_new.hm to set up the link update.

1. In the **Model** browser, **Component** folder, right-click on ***Rear_Truss_1*** and select ***Isolate*** from the context menu.

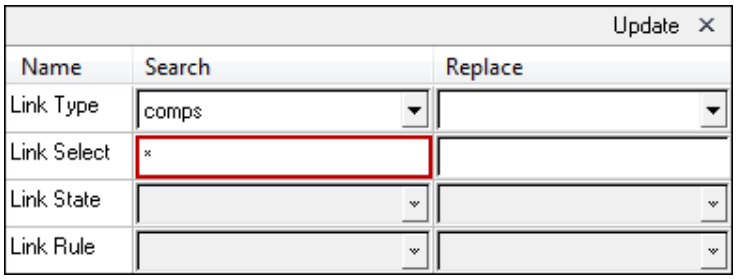


2. From the menu bar, click ***File*** > ***Import*** > ***Model***.
3. Under **File selection**, click .
4. In the **Open** dialog, open the file, `rear_truss_1_new.hm`.
5. Click ***Import***. HyperMesh imports **rear_truss_1_new** on top of **rear_truss_1**.

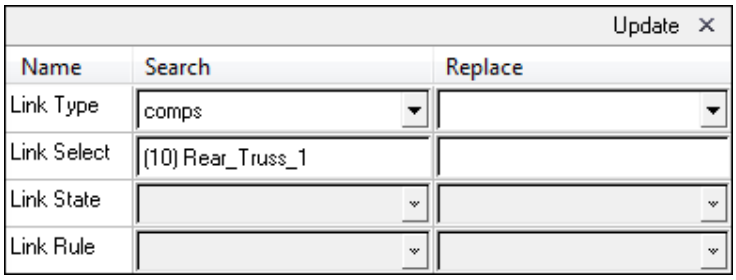


Step 4: Using the Connector Browser, update the connector links to the new component.

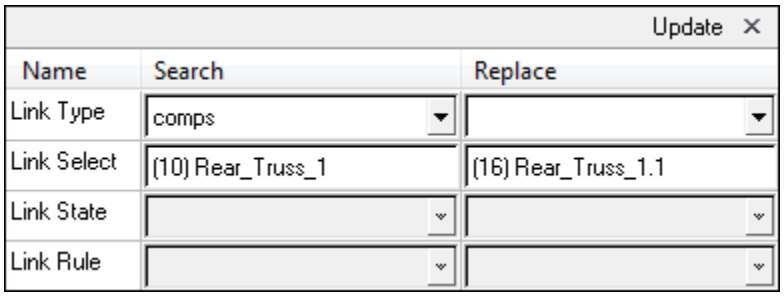
1. In the **Connector** browser, **Link Entity** browser, right-click on ***Rear_Truss_1*** and select ***Find Attached*** from the context menu.
2. In the **Connector Entity** browser, right-click on any of the highlighted connector names and select ***Update Link*** from the context menu.
3. In the **Update** window, click the ***Link Select*** field in the **Search** column.



4. In the panel area, click ***component***.
5. Select the component, ***Rear_Truss_1***.
6. Click ***proceed***. HyperMesh inserts **Rear_Truss_1** into the **Link Select** field.



7. In the **Replace** column, click the **Link Select** field.
8. In the panel area, click ***component***.
9. Select ***Rear_Truss_1.1***.
10. Click ***proceed***. HyperMesh inserts **Rear_Truss_1.1** in the **Link Select** field.



11. Click ***Update***. HyperMesh updates the connector links.
12. Close the **Update** window by clicking ***X*** next to **Update**.

Step 5: Realize the connectors in the component *Con_Rear_Truss*.

1. In the **Model** browser, **Component** folder, right-click on ***Con_Rear_Truss*** and select ***Make Current*** from the context menu.
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 **realize** subpanel.
4. Click ***connectors*** >> ***displayed***.
5. Click **type=** and select ***weld***.
6. In the **tolerance =** field, enter 10.
7. Set the **mesh dependent/mesh independent** toggle to ***mesh dependent***.
8. Under **mesh dependent**, set the **adjust realization/adjust mesh** toggle to **adjust realization**.
9. Click ***realize***. HyperMesh realizes the connectors.
10. Return to the main menu by clicking ***return***.

Step 6: Save the connector information to an XML file.

1. At the bottom of the **Connector** browser, click  (***Export connectors -XML***).
2. In the **Export to file** dialog, navigate to the location where you would like to save the XML file and click ***Save***.
3. In a text editor, open the XML file.
4. Inspect the file and observe how the connector information has been saved.

   Note: In the future, you can use the XML file to import connectors.

Step 7: Export the finite element deck and observe how the connector information is preserved.

1. From the menu bar, click ***File*** > ***Export*** > ***Solver Deck***.
2. From the **File type** list, select ***OptiStruct***.
3. From the **Template** list, select ***standard format***.
4. In the **File** field, click .
5. In the **Save OptiStruct file** dialog, select a name and location for the file to be saved to.

   Note: Be sure to use the `.fem` extension.
6. To view additional export options, click next to **Export Options**.
7. Select the **Include connectors** check box.
8. Click ***Export***.
9. In a text editor, open the `.fem` file you just saved.
10. Scroll to the very bottom. This is where all of the connector information has been saved. The information has been saved as comment cards so that when you run the analysis, the connector information is not read. When you import the input deck back into HyperMesh, the connector information is read.