Altair HyperWorks

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-3440: Model Build and Assembly

altairhyperworks.com

HM-3440: Model Build and Assembly

In this tutorial, you will learn how to:

- Import a PLMXML file.

- Load and create a Common representation and NVH representation for modal analysis

- Import connector parts, add connector representations, and realize connectors

- Edit part attributes using the Entity Editor to reflect design changes

- Activate configurations

- Import and Export an assembly as a Solver Deck

- Renumber entities using the ID-Manager

- Check for errors using the Model Checker

Model Files

This exercise uses the files located in the `HM-3440` folder, which can be found in the `hm.zip` file. Copy the file(s) from this directory to your working directory.

Exercise

Step 1: Start HyperMesh Desktop and Load the OptiStruct User Profile

1. Start HyperMesh Desktop.

2. In the **User Profile** dialog, select **OptiStruct**.

3. Click **OK**.

Step 2: Import the PLMXML File

1. Open the **Import - BOM** tab by clicking **File** > **Import** > **BOM** from the menu bar.

2. In the **File** field, open the `BOM_input.xml` file.

3. Click **Import**. Part assemblies and parts are imported into the session.

| Part Assemblies/Parts	🌈	UID	Revision	Representation	Active
▾ 🗃 **Model**	🟪				
⊢ 🗂 Underbody_000425_Safety	🟥	Underbody	A		☑
⊢ 🗂 Connectors_000481	🟥	Connectors	A		☑
⊢ 🗂 Center_Rails_000482	🟦	Center_Rails	A		☑
⊢ 🗐 Center_Rail_Connectors_var1_000484	🟦	Center_Rail_Connectors_var1	A.-		☑
⊢ 🗐 Center_Rail_Connectors_var2_000485	🟧	Center_Rail_Connectors_var2	A.-		☑
⊢ 🗂 Longitudinal_Rails_000497	🟫	Longitudinal_Rails	A		☑
⊢ 🗐 Longitudinal_Rails_Connectors_000486	🟦	Longitudinal_Rails_Connectors	A.-		☑
⊢ 🗂 Frame_Assembly_000495	🟦	Frame_Assembly	A		☑
⊢ 🗂 LeftRail_A_000433_Safety	🟪	LeftRail_A	A		☑
⊢ 🗐 LeftInner_A_000434_Safety	🟥	LeftInner_A	A.-		☑
⊢ 🗐 LeftOuter_A_000435_Safety	🟪	LeftOuter_A	A.-		☑
⊢ 🗂 RightRail_A_000436_Safety	🟦	RightRail_A	A		☑
⊢ 🗐 RightInner_A_000437_Safety	🟧	RightInner_A	A.-		☑
⊢ 🗐 RightOuter_A_000438_Safety	🟥	RightOuter_A	A.-		☑
⊢ 🗂 CenterRail_A_000426_Safety	🟪	CenterRail_A	A		☑
⊢ 🗂 CenterRail_var1_A_000427_Safety	🟧	CenterRail_var1_A	A		☑
⊢ 🗐 CenterInner_A_000428_Safety	🟫	CenterInner_A	A.-		☑
⊢ 🗐 CenterOuter_A_000429_Safety	🟩	CenterOuter_A	A.-		☑
⊢ 🗂 CenterRail_var2_A_000430_Safety	🟩	CenterRail_var2_A	A		☑
⊢ 🗐 CenterInner_var2_A_000431_Safety	🟨	CenterInner_var2_A	A.-		☑
⊢ 🗐 CenterOuter_var2_A_000432_Safety	🟫	CenterOuter_var2_A	A.-		☑

Step 3: Load and Create a Common Representation

Steps 3.4 - 3.6 below are optional as the Common representation can be created without loading the CAD into the session. Since the Common representation forms the basis of subsequent discipline specific mesh representations, its creation is a prerequisite for the next steps.

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

2. In the **Part** browser, right-click and on one of the column headers and enable the following columns from the context menu: PDM PID, PDM MID, PDM Material, PDM Thickness, and PDM MeshFlag.

   These columns show the PDM metadata that is parsed upon importation of the PLMXML file. This information is also shown in the **Entity Editor**, **PDM Data** pane.
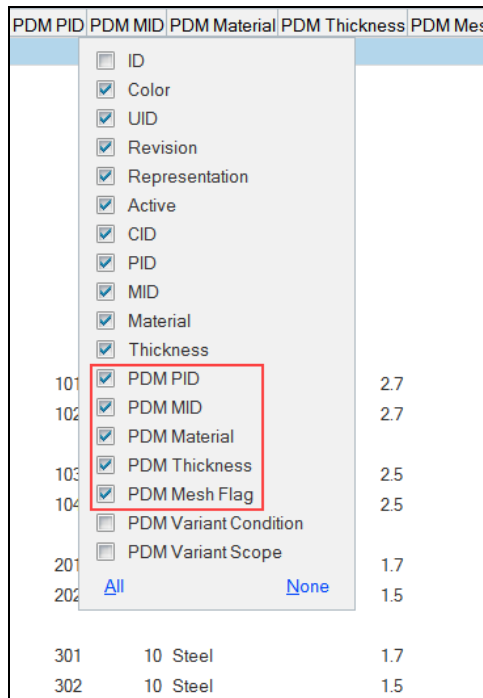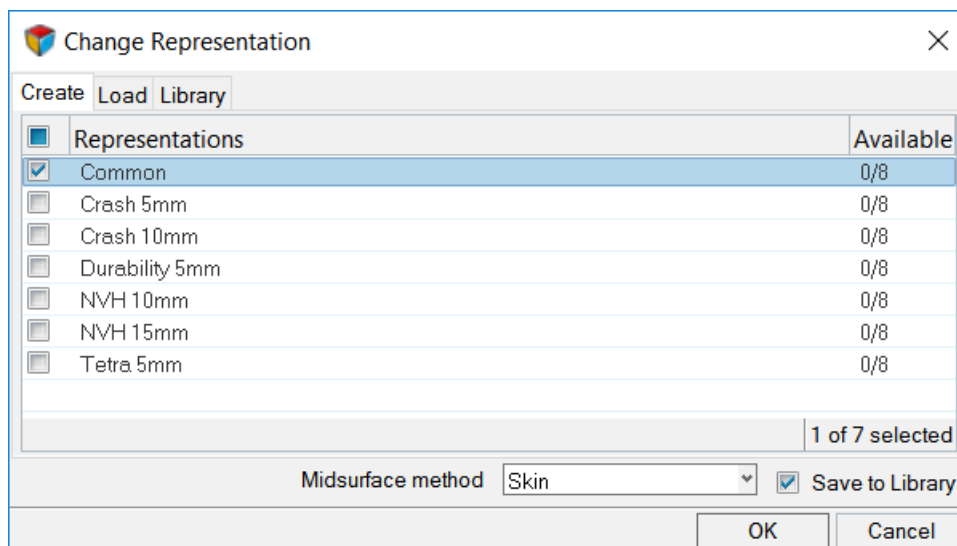

3. Right-click on the Frame_Assembly_000495 part assembly and select **Representations** > **Load** > **from Session** from the context menu.

4. In the **Change Representations** dialog, **Load** tab, select the **CAD** representation.
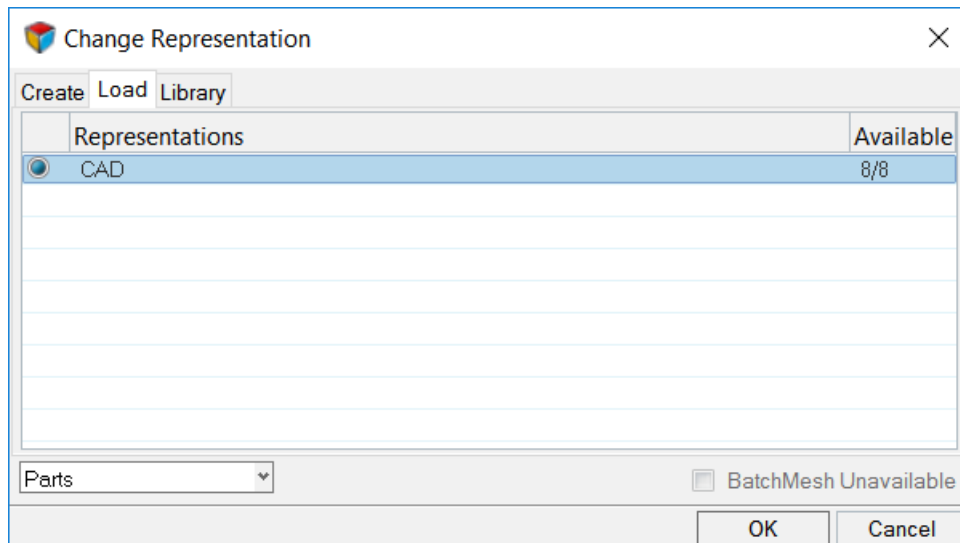
5. Click **OK**. All available CAD representations are imported into the session.
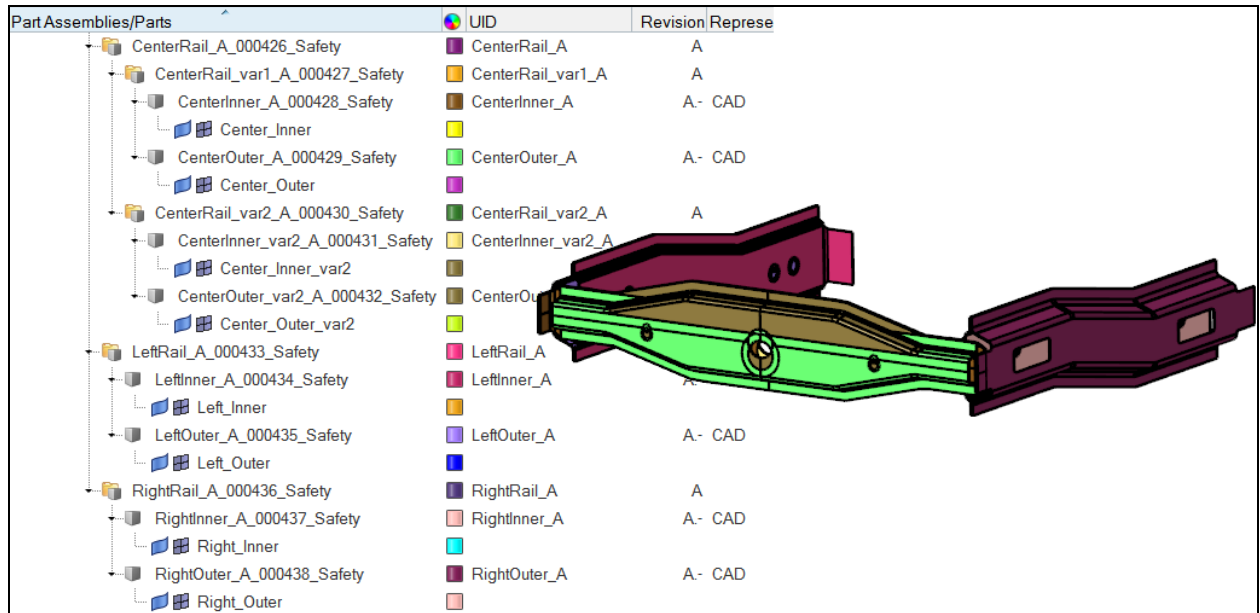
6. Right-click on the Frame_Assembly_000495 part assembly and select **Representations** > **Create** from the context menu.

7. In the **Change Representations** dialog, **Create** tab, select **Common (0/8)**.

8. Click **OK**. Available CAD representations are sent to the BatchMesher for processing.

   Note: In the case of sheet metal parts, the BatchMesher extracts the midsurface from the solid CAD representation. You can choose a Midsurface method, include Skin, Offset, Offset + Planes, Offset + Planes + Sweeps.
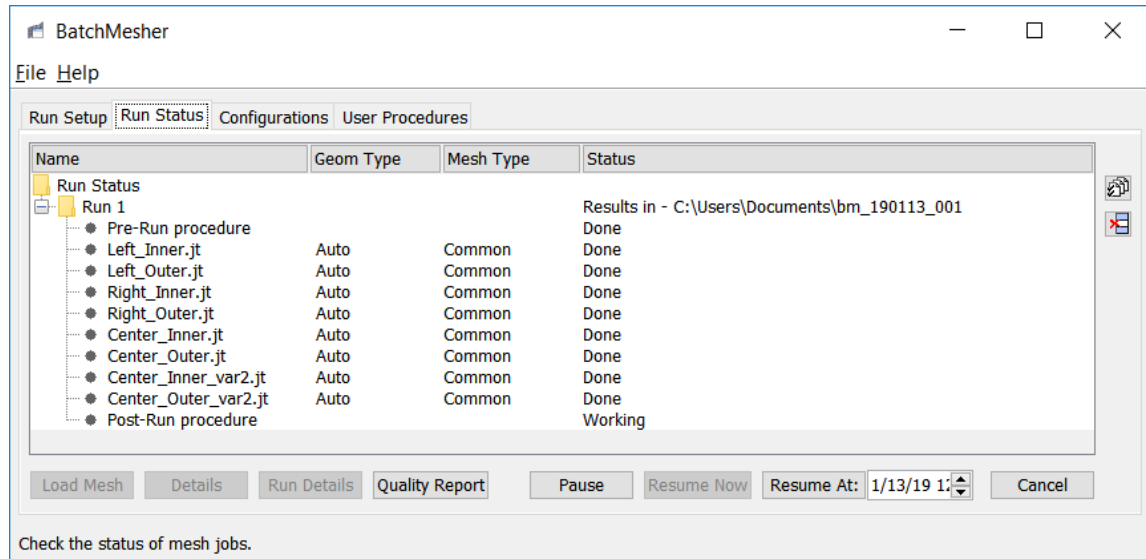
9. In the **BatchMesh** dialog, click **Yes** to load the new representations for the eight parts.

10. Repeat steps 3.7 - 3.10 to create NVH10 and NVH15 representations.

Step 6: Add Connector Representations

1. In the **Part** browser, right-click on the Connectors_000481 part assembly and select **Representations** > **Load** > **from Session** from the context menu.

2. In the **Change Representation** dialog, **Load** tab, select the **Connector** representation.
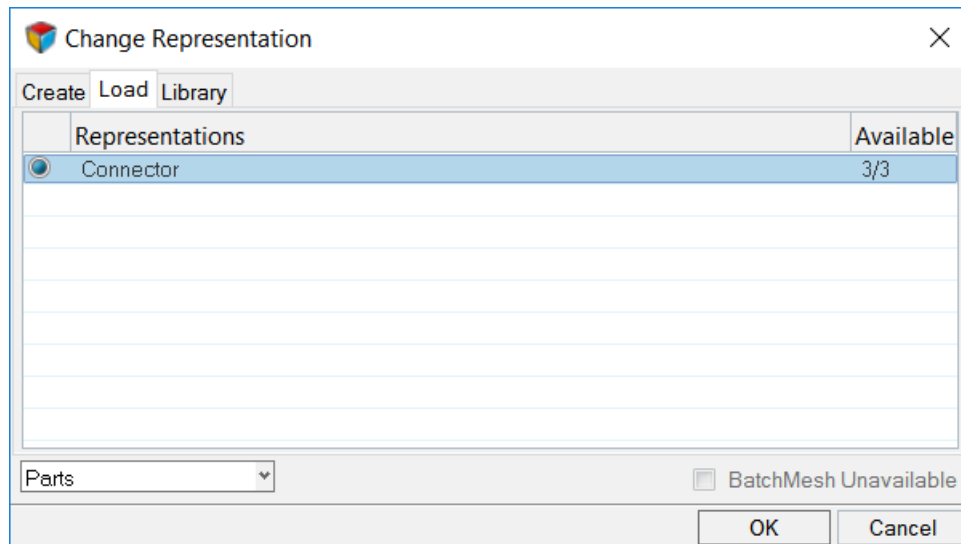
3. Click **OK**. Connectors are loaded, and the Representation column shows the Connector representation loaded.
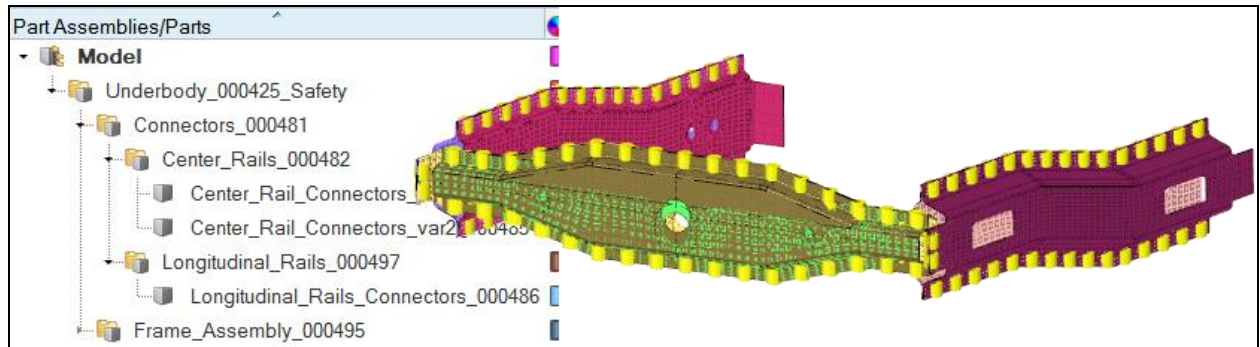
Step 7: Design change - modify Center Rail part attributes

1. In the **Part** browser note the PID of parts CenterInner_var2_A_000431_Safety and CenterOuter_var2_A_000432_Safety.

2. In the **Model** browser, switch the view mode to **Properties**.

3. Select PID 103 and 104. The **EntityEditor** opens and displays the two properties common corresponding attributes.


4. In the **Entity Editor**, **T** field, enter `3.0`.

5. In the **Part** browser, notice the attribute modification you made is updated in the **Thickness** columns. Save these changes to ensure that they are available if the current representation is unloaded.

Step 8: Create Part Sets

Create part sets Var1 and Var2 for Variant 1 and Variant 2 in the Part Set view.

1. In the **Part** browser, enable the **Part Set** view.
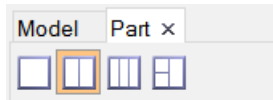
2. Create two part sets.

   a. In the **Part Set** view, right-click and select **Create** > **Part Set** from the context menu.

   b. Name the part sets **Var1** and **Var2**.

3. Group common and unique parts by dragging-and-dropping parts from the Part view onto the part set.

   Var 1:

   - Center_Rail_Connectors_var1_000484
   - CenterInner_A_000428_Safety
   - CenterOuter_A_000429_Safety

   Var2:

   - Center_Rail_Connectors_var2_000485
   - CenterInner_var2_A_000431_Safety
   - CenterOuter_var2_A_000432_Safety
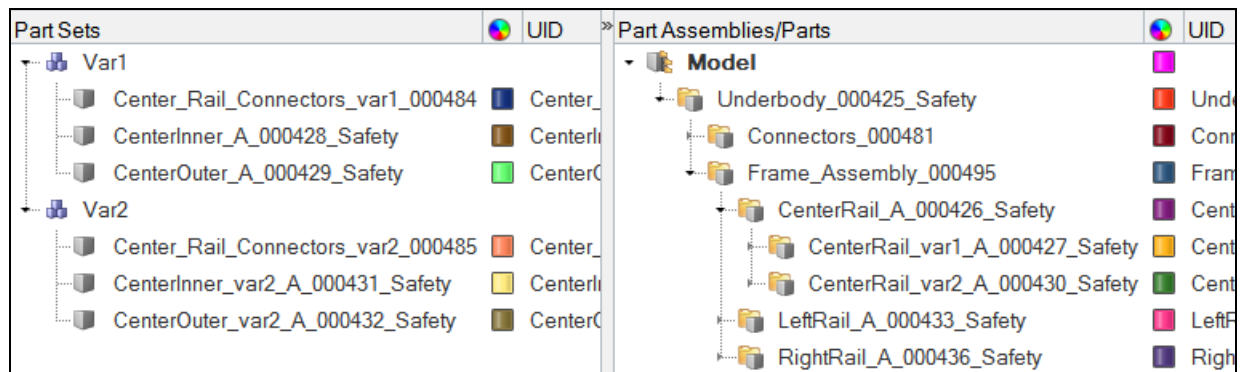
Step 9: Create Configurations

Create configurations Var1 and Var2 for part sets Var1 and Var2 in the Configuration view.

1. In the **Part** browser, enable the **Configuration** view.

2. Create two configurations.
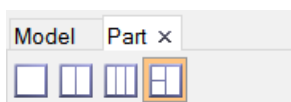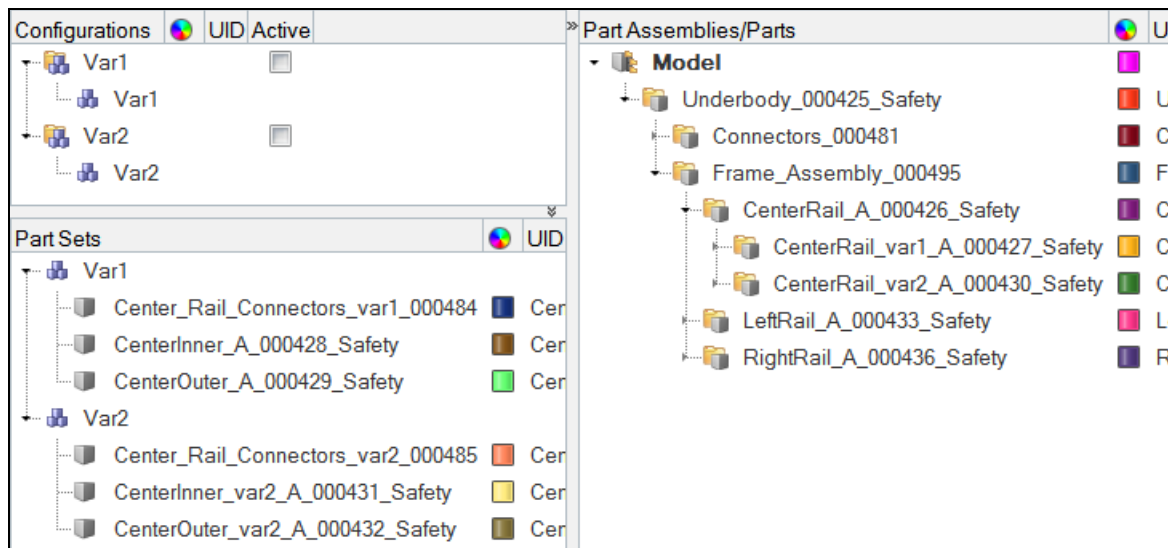
△ Altair

3. In the **Part Set** view, right-click and select **Create** > **Configuration** from the context menu.

4. Name the configurations **Var1** and **Var2**.

5. Group part sets that are unique by dragging-and-dropping part sets from the Part Set view onto the configuration.

   a. Group the Var1 part set into the Var 1 configuration.

   b. Group the Var2 part set into the Var 2 configuration.

Step 10: Activate the Var 2 Configuration

1. In the **Part** browser's **Configuration** view, **Active** column, enable the checkbox for **Var 2**.

   All of the parts, part assemblies, components, and part sets not associated with Var 2 become inactive.
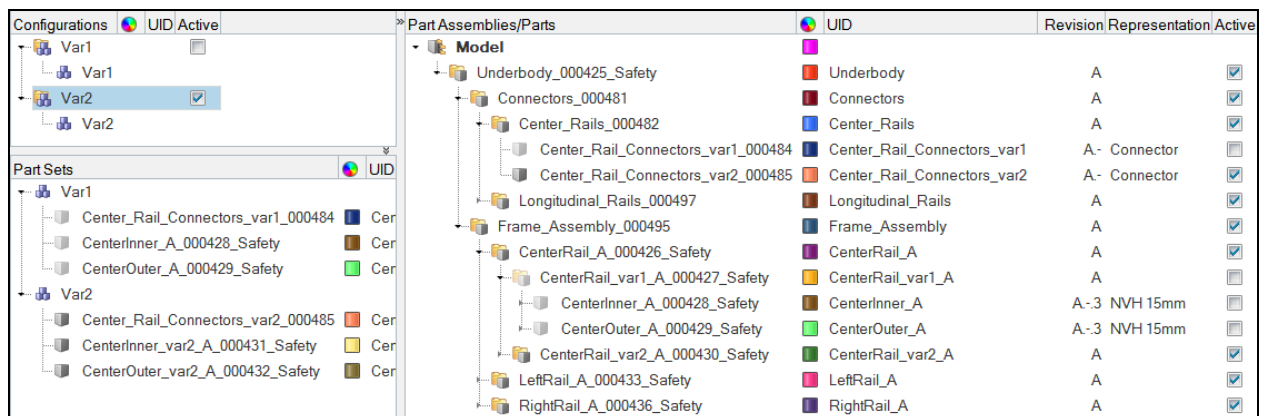
Step 11: Export Assembly as a Solver Deck

1. Open the **Export – Solver Deck** tab by clicking *File* > *Export* > *Solver Deck* from the menu bar.

2. In the **File** field, enter `frame_var2_model.fem`.

3. Under **Export Options**:

    *   Set **Export** to *Custom* to ensure that inactive parts are not written to the solver deck.

    *   Under **Comments**, select the *Part Assemblies/Parts* checkbox.

4. Click *Export*.

5. Save the model as `frame_assembly.hm`.

Step 12: Import Assembly Solver Deck

1. Start a new HyperMesh Desktop session.

2. Open the **Import – Solver Deck** tab by clicking *File* > *Import* > *Solver Deck* from the menu bar.

3. In the **File** field, locate the `frame_var2_model.fem` solver deck.

4. Click *Import*.

5. In the **Part** browser, verify that the BOM was imported correctly.

    Note: All part assembly and part metadata in the original model should be present, with the exception of the Representation name.

Step 13: Import Realizations

1. Open the **Import - Model** tab.

2. Click [folder icon], then open the `Realizations.hm` file.

3. The Spotweld component, in the `Realizations.hm` file, references a material that has the same ID as a material that already exists in the current session, therefore under **Entity Management**, set **Materials** to *Keep Existing Attributes*.

Name	Value
Solver Keyword	PLSOLID
Name	Spotwelds
ID	500
Color	[green]
Include	[Master Model]
Defined	☑
Card Image	PLSOLID
Material	(10) Steel
User Comments	Hide In Menu/Export

4.   Click *Import*. Connector parts are imported.

Step 14: Realize Connectors

1.   In the **Part** browser, right-click on the Spotwelds component and select *Make Current* from the context menu.

2.   In the **Part** browser, right-click on the Longitudinal_Rail_Connectors and select *Hide* from the context menu. The display of the component's connectors is turned off in the graphics area enabling you to get a better visual of each component's connectors.

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

4.   In the **Connector Entity** browser, select the *acm (shell gap)* connector folder.

5.   In the **Entity Editor**, set **Property Script** to *no/skip post script*.

   Note: Connector links are defined via Parts to ensure that connectors realize even if you, accidentally, renumber all of the entities in the model.

6.   In the **Connector Entity** browser, select all of the connectors.

7.   Right-click on the selected connectors and select *Rerealize* from the context menu.

8.   In the **Part** browser, right-click on the Spotwelds part and select *Isolate Only* from the context menu. Verify that the realized FE resides in the Spotwelds component.

Step 15: Renumber Nodes and Elements

1.   Open the **ID-Manager** by clicking *Tools* > *ID-Manager* from the context menu.

2.   In the **ID-Manager**, select *Components*, *Properties*, and *Materials* then right-click and select *Exclude* from the context menu.

3. For the Master Model, enter `1,000,000` in the **Min** field and `1,500,000` in the **Max** field.

Entities	ID	Excluded	Min	Max	#Overflow	Min Reserved	Max Re
⊟ 🔗 **Master Model**	**0**		**1000000**	1500000	**24682**		
🟦 Nodes			1000000	1500000	12671		
🟦 Elements			1000000	1500000	12011		
🟦 Components		✖					
🟦 Properties		✖					
🟦 Materials		✖					

4. Correct ID overflow by right-clicking on the Master Model and selecting ***Correct*** >
   Overflow from the context menu.

Step 16: Run the Model Checker

1. Open the **Model Checker** by clicking ***Tools*** > ***Model Checker*** > ***OptiStruct***.

2. In the **Model Checker**, right-click and select ***Run*** from the context menu.

3. Verify that the model is error free.

Step 17: Export the Solver Deck

1. From the menu bar, click ***File*** > ***Export*** > ***Solver Deck***.

2. In the **Export – Solver Deck** tab, **File** field, enter the file name
   `frame_var2_assembled.fem`.

3. Under **Export options**, **Comments**, select the ***Part Assemblies/Parts*** checkbox.

4. Click ***Export***.

△ Altair