Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-4010: Formatting Model for Analysis

altairhyperworks.com

HM-4010: Formatting Model for Analysis

In this tutorial, you will learn how to:

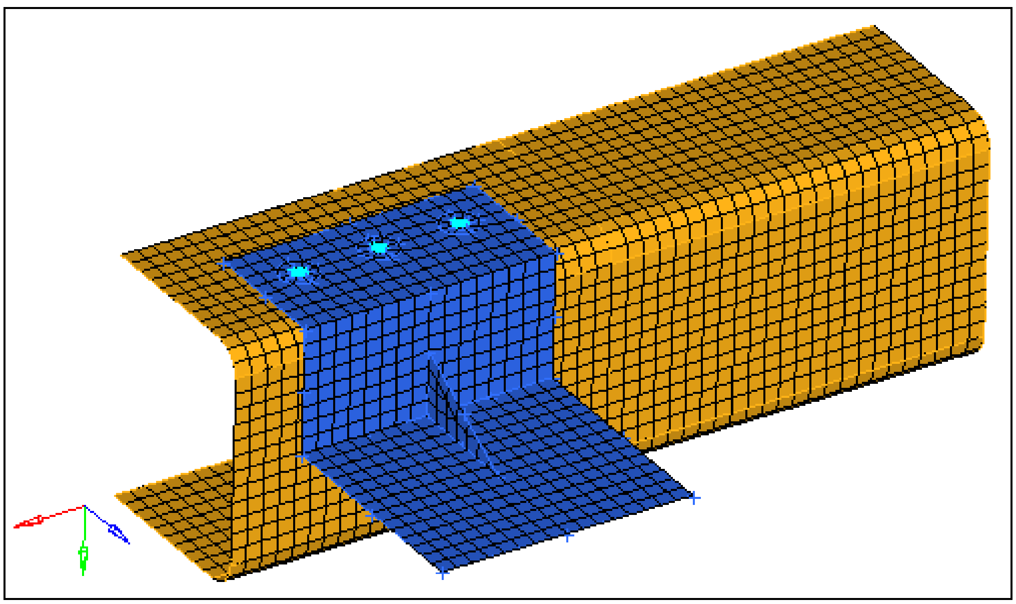
- Create a solver input file by using a template
- Review entities in HyperMesh to see how they will appear in the solver input file
- Define materials and properties
- Select solver element types for HyperMesh element configurations

The purpose of using a finite element (FE) pre-processor is to create a model that can be run by a solver. HyperMesh interfaces with many FE solvers and all of them have unique input file formats. HyperMesh has a unique template(s) for each solver it supports. A template contains solver specific formatting instructions, which HyperMesh uses to create an input file for that solver.

Model Files

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

The model contains the bracket and channel assembly in the following image.



Exercise: OptiStruct Linear Statics Setup for a Shell Assembly

Step 1: Load the OptiStruct user profile.

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, select **OptiStruct**.
3. Click ***OK***.

Step 2: Retrieve and view the file, `channel_brkt_assem_analysis.hm`.

1. Open a model file by clicking ***File*** > ***Open*** > ***Model*** from the menu bar, or clicking on the **Standard** toolbar.
2. In the **Open Model** dialog, open the `channel_brkt_assem_analysis.hm` file. A model appears in the graphics area.

Step 3: Review a bracket element to identify what type of OptiStruct element it is and to see how it will be formatted in the OptiStruct input file.

1. Open the **Card Edit** panel by clicking on the **Collectors** toolbar.
2. Set the entity selector to ***elems***.
3. In the graphics area, select an element on the **bracket** component.

   Note: The **bracket** component is blue.
4. Click ***edit***. The **Card Image** opens, and indicates that the selected element is an OptiStruct CQUAD4 or CTRIA3, depending on whether you selected a quad or tria element.

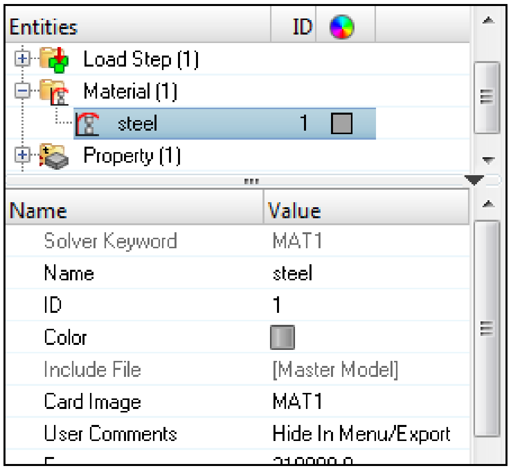
   Note: EID is the element's ID, PID is the ID of the element's property, and G(X) is the grid (node) ID that makes up the element. Options specific to the CQUAD4 or CTRIA3 appear in the menu panel area.
5. Close the **Card Image** by clicking ***return***.
6. Exit the **Card Edit** panel by clicking ***return***.

Step 4: Review and edit the existing steel material's card image by accessing the card editor from the Model browser.

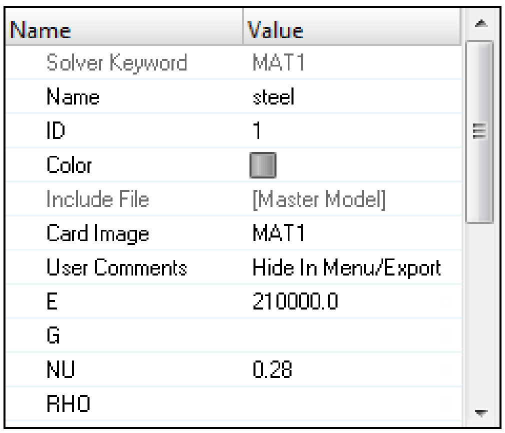
This material is defined for the channel.

1. In the **Model** browser, **Material** folder, click ***steel***. The **Entity Editor** opens and displays the material's corresponding data.

   Note: The card image indicates the material is of OptiStruct type MAT1.



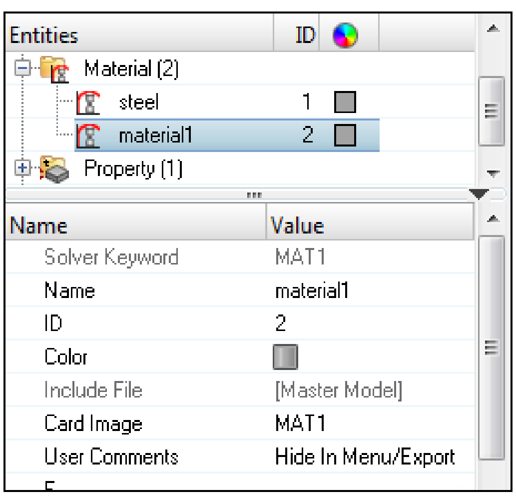
2. In the **Entity Editor**, **NU** (Poisson's Ratio) field, change the value from `0.3` to `0.28`.



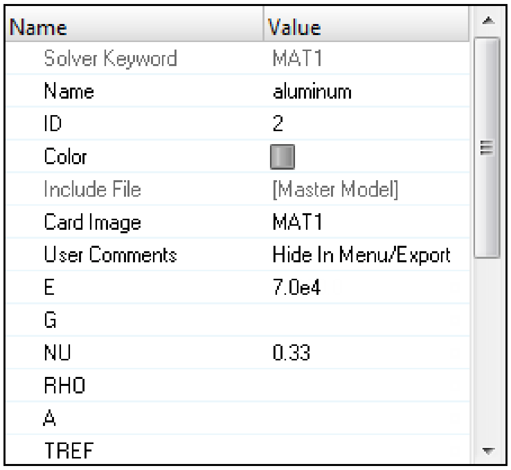
Step 5: Define a material collector named aluminum for the bracket.

This material is defined for the channel.

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

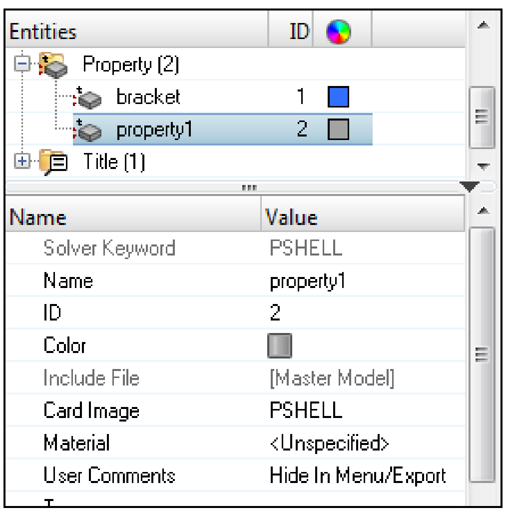


2. For **Name**, enter `aluminum`.
3. Set **Card Image** to ***MAT1***.
4. For **E** (Young's Modulus), enter `7.0e4`.
5. For **NU** (Poisson's Ratio), enter `0.33`.

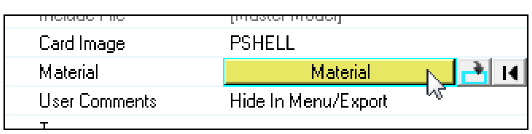


Step 6: Define a property collector (PSHELL card image) that will be assigned to the channel component collector.

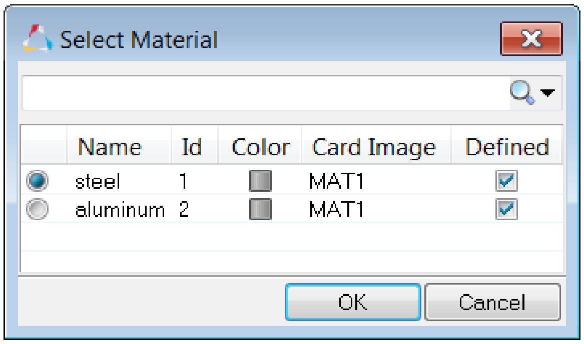
1. In the **Model** browser, right-click and select ***Create*** > ***Property*** from the context menu. HyperMesh creates and opens a property in the **Entity Editor**.



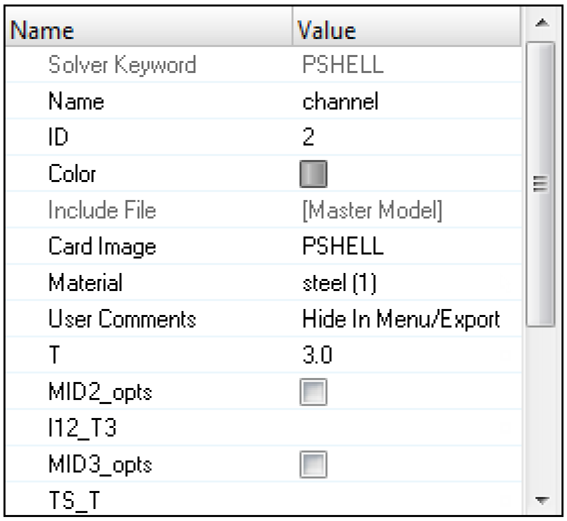
2. For **Name**, enter `channel`.
3. Set **Card Image** to ***PSHELL***.
4. For **Material**, click ***Unspecified*** >> ***Material***.



5. In the **Select Material** dialog, select ***steel*** and then click ***OK***. HyperMesh assigns the material.

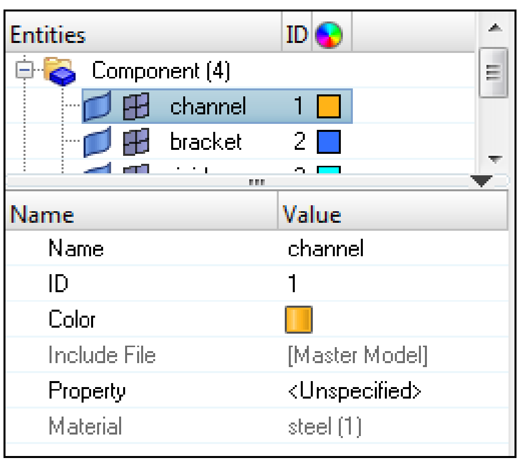


6. For **T** (thickness), enter `3.0`.

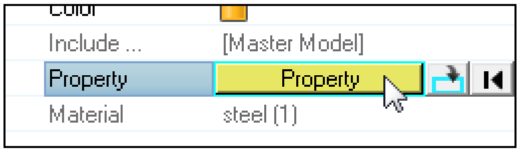


Step 7: Assign the *channel* property to the channel component.

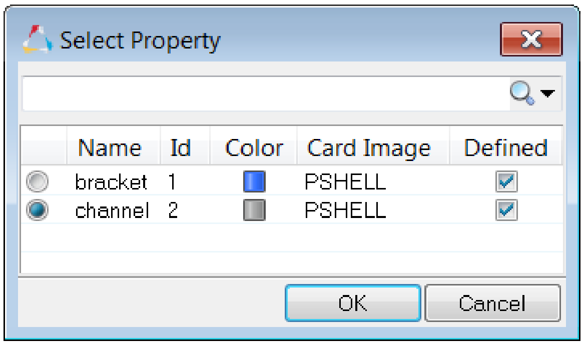
1. In the **Model** browser, **Component** folder, click ***channel***. The **Entity Editor** opens and displays the component's corresponding data.



2. For **Property**, click ***Unspecified*** >> ***Property***.

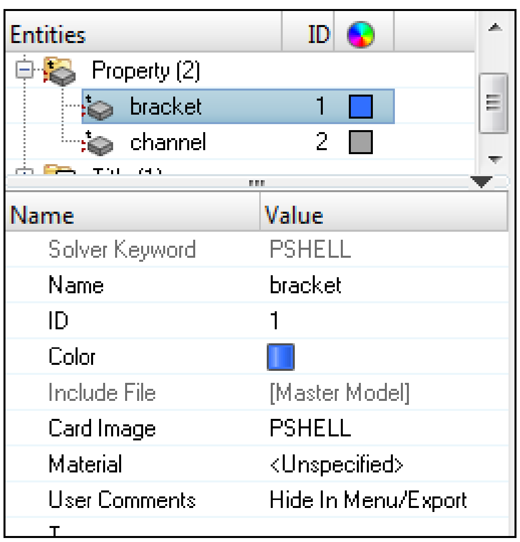


3. In the **Select Property** dialog, select ***channel*** and then click ***OK***. HyperMesh assigns the property **channel** to the component **channel**.



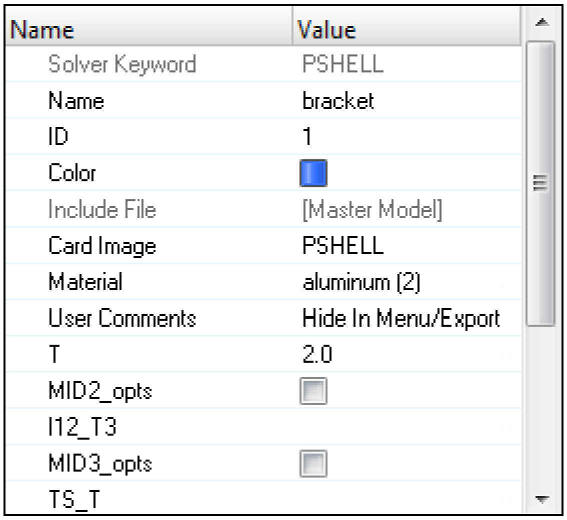
Step 8: Update the bracket property to have a PSHELL card image, a thickness of 2.0, and the aluminum material.

1. In the **Model** browser, **Property** folder, click ***bracket***. The **Entity Editor** opens and displays the properties' corresponding data.



2. Set **Card Image** to ***PSHELL***.
3. For **Material**, click ***Unspecified*** >> ***Material***.
4. In the **Select Material** dialog, select ***aluminum*** and then click ***OK***. HyperMesh assigns the material **aluminum** to the property **bracket**.

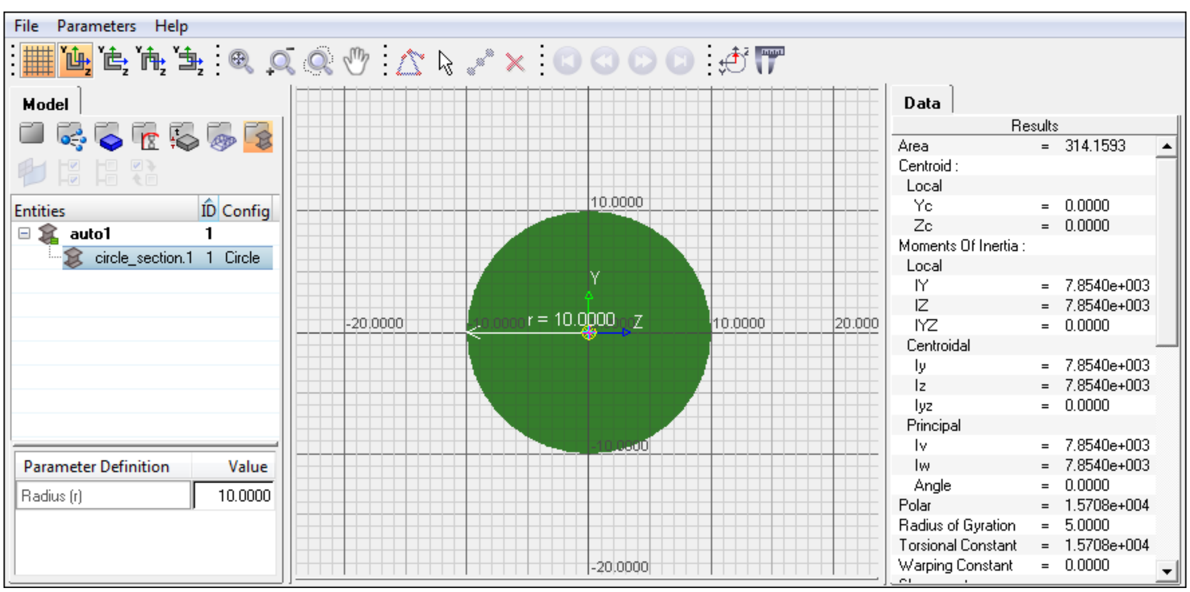
5. For **T** (thickness), enter `2.0`.



Step 9: Calculate the section properties for the bar elements (OptiStruct CBEAM) by using HyperBeam.

1. Open the **HyperBeam** panel by clicking ***Properties*** > ***HyperBeam*** from the menu bar.
2. Go to the ***standard section*** subpanel.
3. Set the **standard section library** to **HYPERBEAM**.
4. Set the **standard section type** to ***solid circle***.
5. Click ***create***. HyperMesh invokes the HyperBeam module.

   Note: The solid, green circle represents the cross section. Under the local coordinate system you should see the number, 10.0000, which is the circle's radius.

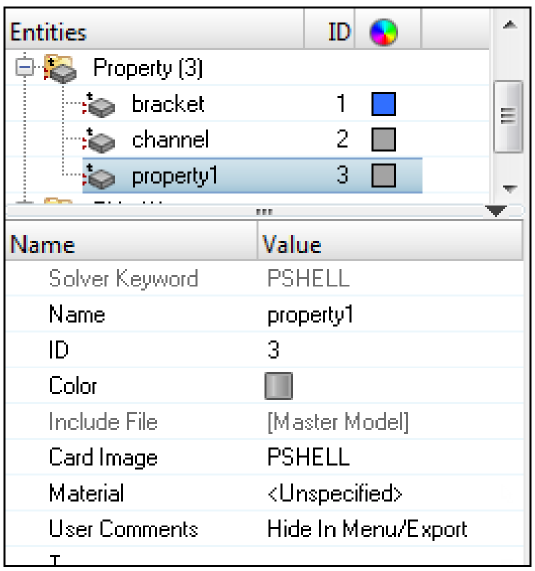


HyperBeam module with the standard solid circle section

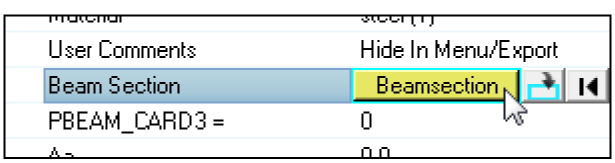
6. Under **Parameter Definition**, click the ***Value*** field next to **Radius (r)** and change the value from 10 to `3`. HyperMesh updates the values in the **Data** pane to reflect the circle's new diameter.
7. In the **Model** browser, right-click on ***circle_section.1*** and select ***Rename*** from the context menu.
8. In the editable field, rename the section `6mm_Beam_Sect`.
9. Close the HyperBeam module and return to your HyperMesh session by clicking ***File*** > ***Exit*** from the menu bar.
10. Return to the main menu by clicking ***return***.

Step 10: Create a property collector named *bars_prop* for the bar elements (OptiStruct).

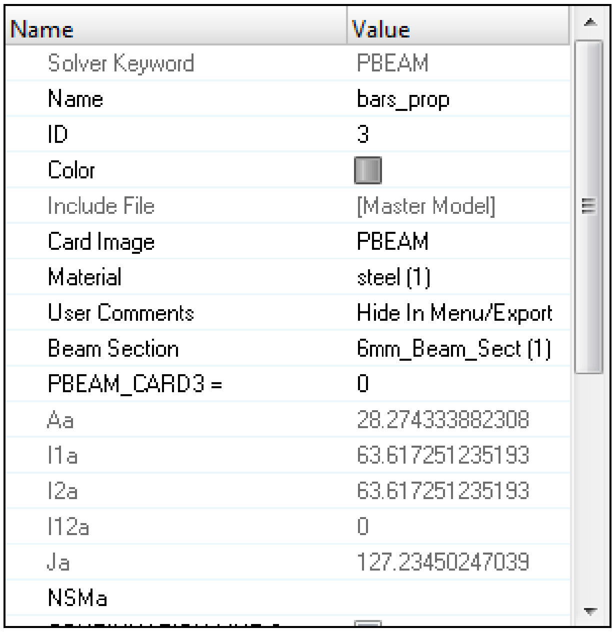
1. In the **Model** browser, right-click and select ***Create*** > ***Property*** from the context menu. HyperMesh creates and opens a property in the **Entity Editor**.



2. For **Name**, enter `bars_prop`.
3. Set **Card Image** to ***PBEAM***.
4. For **Material**, click ***Unspecified*** >> ***Material***.
5. In the **Select Material** dialog, select ***steel*** and then click ***OK***. HyperMesh assigns the material **steel** to the property **bars_prop**.
6. For **Beam Section**, click ***Unspecified*** >> ***Beamsection***.

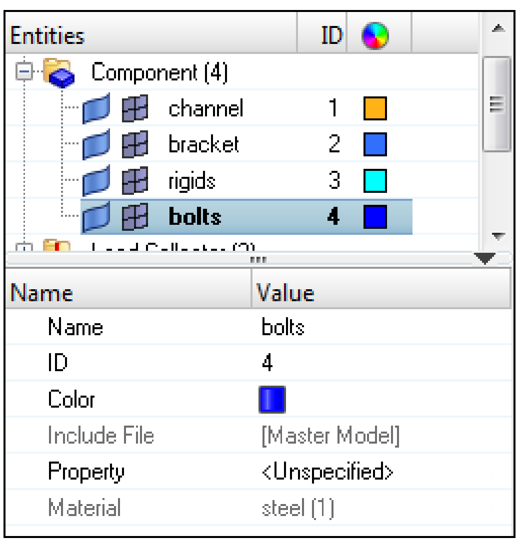


7. In the **Select Beam Section** dialog, select ***6mm_Beam_Sect*** and then click ***OK***. HyperMesh assigns the beam section, and populates the parameter fields in the PBEAM card with the data in the **6mm_Beam_Sect** beam section.



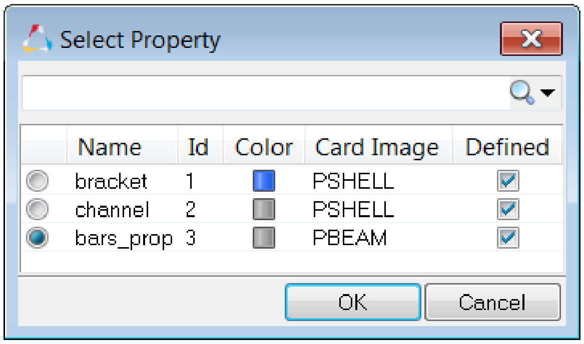
Step 11: Update the CBEAM elements in the bolts component to use the PBEAM Property.

1. In the **Model** browser, **Component** folder, click ***bolts***. The **Entity Editor** opens and displays the component's corresponding data.



2. For **Property**, click ***Unspecified*** >> ***Property***.

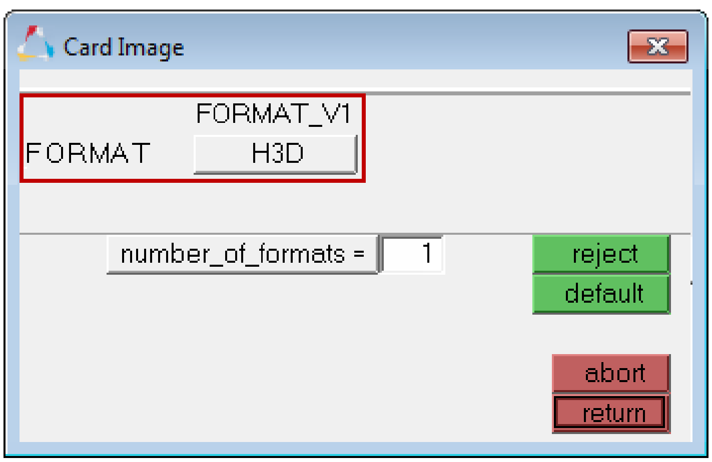
3. In the **Select Property** dialog, select ***bars_prop*** and then click ***OK***. HyperMesh assigns the property **bars_prop** to the component **bolts**.



Step 12: Define a H3D file to be output from OptiStruct by using the *control cards* panel.

1. Open the **Control Cards** panel by clicking ***Setup*** > ***Create*** > ***Control Cards*** from the menu bar.
2. In the **Card Image**, select the control card ***FORMAT***.

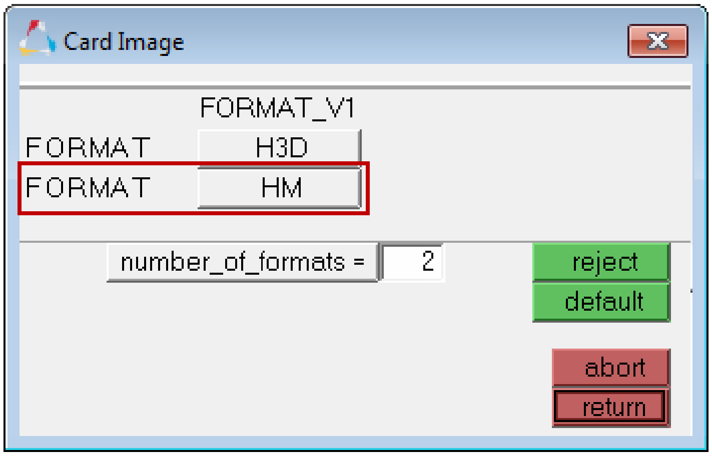
   Note: In the card image, the **FORMAT** line is set to **H3D**. This specifies OptiStruct to output results to a Hyper3D (H3D) file, which can be viewed in the HyperView Player. A HTML report file will be output and the H3D file will be embedded in it.



3. In the **number_of_formats =** field, enter `2`. A second FORMAT line appears in the card image.

4. In the second **FORMAT** line, click ***H3D*** and then select ***HM***.

   Note: This option specifies OptiStruct to output the results to a HyperMesh binary results file, allowing the results to be post-processed within HyperMesh.



5. Exit to the **Control Cards** panel by clicking ***return***.

   Note: The **FORMAT** button is now green, which indicates that the card will be exported to the OptiStruct input file.

6. Return to the main menu by clicking ***return***.

Step 13: Export the model to an OptiStruct input file.

1. From the menu bar, click ***File*** > ***Export*** > ***Solver Deck***.
2. In the **File** field, click .
3. In the **Select OptiStruct file** dialog, navigate to your working directory and save the file as `channel_brkt_assem_loading.fem`.
4. Click ***Export***. HyperMesh exports the model as an OptiStruct `.fem` input file for the solver specified by the current user profile.

Step 14: Review the contents of the file channel_brkt_assem_loading.fem.

1. In any text editor (Notepad, Wordpad, Vi, etc.), open the file `channel_brkt_assem_loading.fem`.
2. Near the top of the file, note the following:
   - The line FORMAT HM, which you specified in HyperMesh
   - The load step (OptiStruct SUBCASE) named **pressing_step** which you defined in HyperMesh
   - Under the load step, the load collector ids (OptiStruct load and constraint set identification numbers)

```
FORMAT H3D
FORMAT HM
$$------------------------------------------------------------------------------$
$$                        Case Control Cards                                    $
$$------------------------------------------------------------------------------$
$
$HMNAME LOADSTEP               1"pressing_step"       1
$
SUBCASE       1
  SPC =        2
  LOAD =        1
```

3. Search for "FORCE."
4. Note the load set identification number for each force (OptiStruct FORCE). It is either 1 or 2 as shown below. These numbers correspond to the numbers under the load steps in the file.

```
$  FORCE Data
$
FORCE           1    2587       01.0     0.0     5.0     0.0
FORCE           1    2571       01.0     0.0     5.0     0.0
```

5. Search for "SPC" (HyperMesh constraint).
6. Note the constraint set identification number for each constraint (OptiStruct SPC). It is 2 as shown below, which lists a few of the constraints. This number corresponds to the number under the load steps in the file.

```
$$  SPC Data
$$
SPC             2      59  1234560.0
SPC             2     698  1234560.0
SPC             2     699  1234560.0
SPC             2     700  1234560.0
SPC             2     701  1234560.0
SPC             2     702  1234560.0
```

7. Search for the load collector name "pressing_load."
8. Note the load collectors, **pressing_load** and **constraints**. Also, note their collector ID and color ID. When the model is imported into HyperMesh, the loads are organized into these load collectors and have these IDs and colors.

```
$$------------------------------------------------------------------------------$
$$ HyperMesh Commands for loadcollectors name and color information $
$$------------------------------------------------------------------------------$
$HMNAME LOADCOL               1"pressing_load"
$HWCOLOR LOADCOL              1       5
$$
$HMNAME LOADCOL               2"constraints"
$HWCOLOR LOADCOL              2      49
```

9. Close the file `channel_brkt_assem_loading.fem`.

Step 15 (Optional): Save your work.

With the exercise completed, you can save the model as a HyperMesh file, if desired.