Altair HyperMesh 2019 Tutorials

HM-4000: Setting up Loading Conditions

HM-4000: Setting up Loading Conditions

In this tutorial, you will learn how to:

- Create constraints (OPTISTRUCT SPC) on the channel's geometry lines

- Create a force (OPTISTRUCT FORCE) on the bracket to simulate a pressing load on it

- Define a load step (OPTISTRUCT SUBCASE)

- Export the model to an OptiStruct input file

- Submit the OptiStruct input file to OptiStruct

- Review the resulting HTML report file

The purpose of using a finite element (FE) pre-processor is to create a model, which can be run by a solver. A finite element solver can solve for responses of parts to loading conditions on them. The loads can be in the form of boundary constraints, forces, pressures, temperatures, and so on.

In this tutorial, you will gain an understanding of the basic concepts for creating a solver input file by using a template. More specifically, you will learn how to define loading conditions on a model, specify solver specific controls, and submit an input file to a solver from HyperMesh.

Model Files

This exercise uses the `channel_brkt_assem_loading.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.
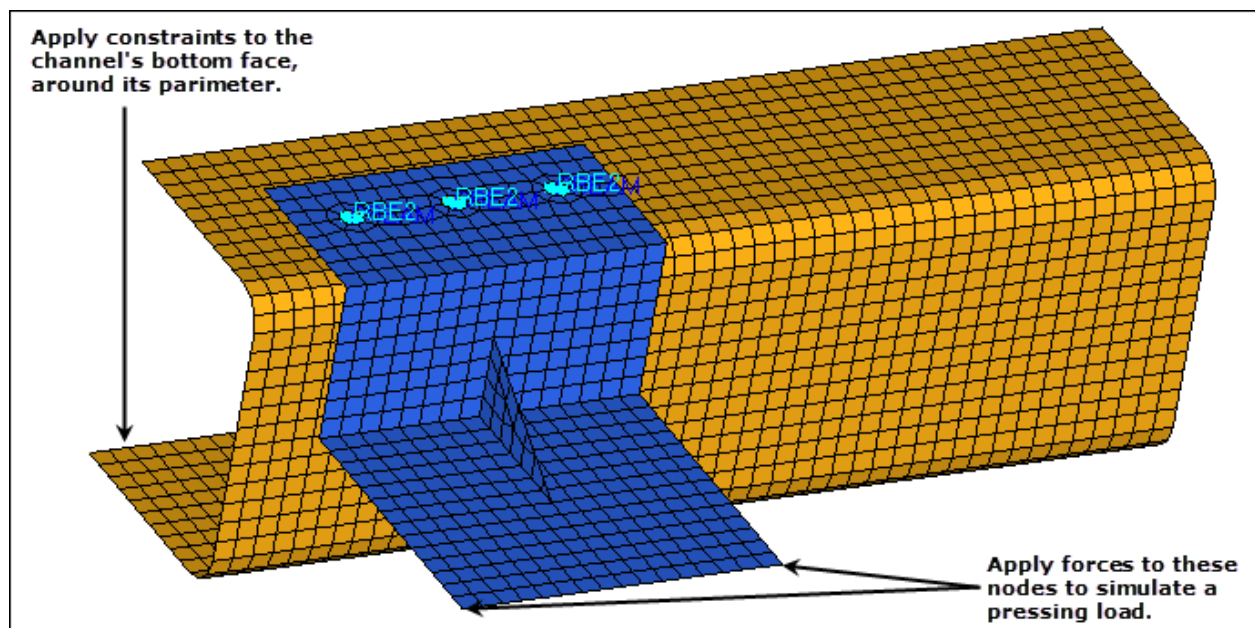
Exercise: Setting up Loading Conditions

### 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 HyperMesh model file, channel_brkt_assem_loading.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_loading.hm` file. A model appears in the graphics area.

Step 3: Create two load collectors named *pressing_load* and *constraints*.

1.   In the **Model** browser, right-click and select *Create* > *Load Collector* from the context menu. HyperMesh creates and opens a load collector in the **Entity Editor**.
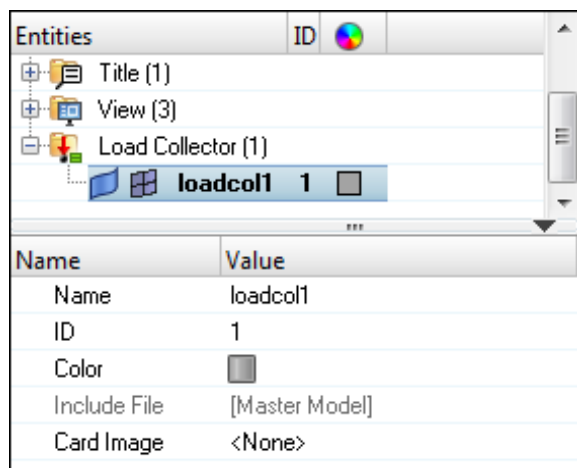
2.   In the **Entity Editor**:

   •   For **Name**, enter `pressing_load`.

   •   Click the **Color** icon, and select a color for the load collector.

   •   Set **Card Image** to *<None>*.

Name	Value
Name	pressing_load
ID	1
Color	◻
Include File	[Master Model]
Card Image	<None> ▼
	<None> ▲
	ACSRCE
	CAALOAD ≡
	CDSMETH
	CMSMETH
	DLOAD
	DTI
	EIGC
	EIGRA
	EIGRL ▼

3. Repeat steps 3.1 and 3.2 to create second load collector labeled `constraints`.

Step 4: Apply constraints (OPTISTRUCT SPC) to the channel's line geometry.

1. In the **Model** browser, **View** folder, right-click on ***View2*** and select ***Show*** from the context menu.

    Note: By selecting this view, HyperMesh sets the component's and load collector's displays back to what they were when the view was saved. The load collectors that you created in step 3 are now turned off because they did not exist when the view was saved. You will need to turn these back on to see the display of the BCs when you create them in the next steps.

2. In the **Load Collector** folder, click 🔲 next to **pressing_load** and **constraints** to turn on the display of their geometry.

3. In the **Component** folder, click 🔲 next to **channel** to turn on the display of its geometry.

4. Open the **Constraints** panel by clicking ***BCs*** > ***Create*** > ***Constraints*** from the menu bar.

5. Go to the ***create*** subpanel.

6. Set the entity selector to ***lines***.

△ Altair

7. Select the six lines on the perimeter of the channel's bottom surface as indicated in the following image.
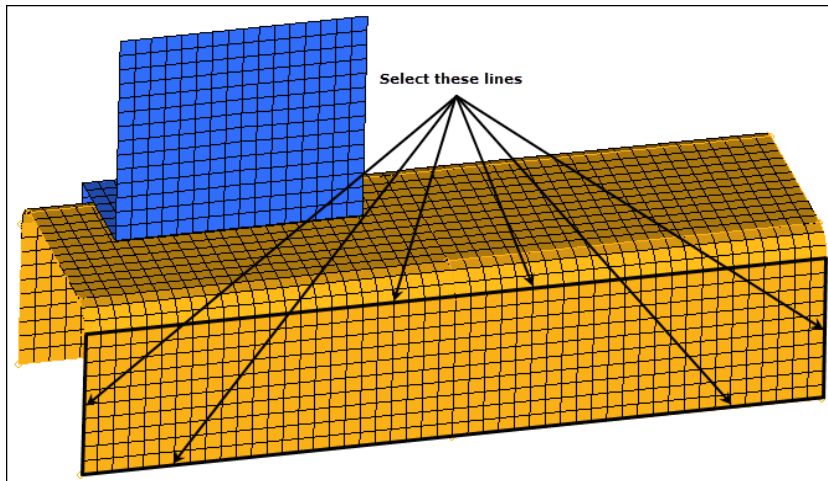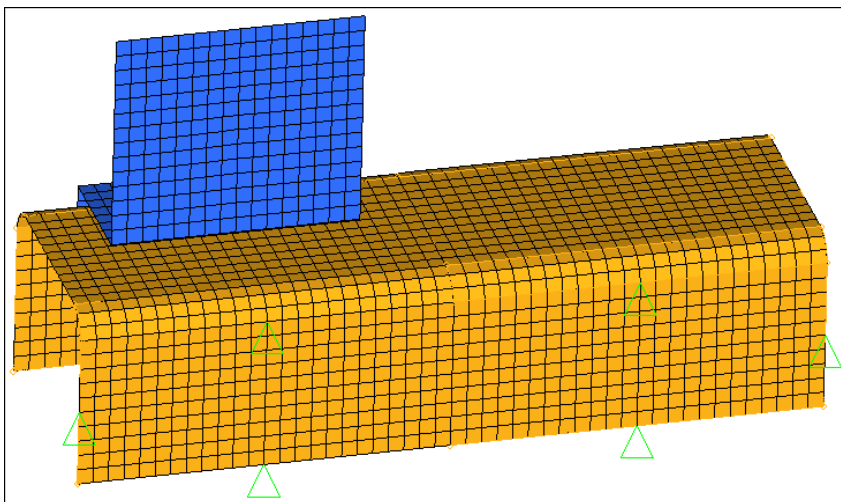

8. Verify that all six **dofs** (degrees of freedom) are selected.

    Note: For an OptiStruct linear static analysis, dof 1, 2, and 3 represent translations in the global x-, y-, and z-directions, respectively. Dof 4, 5, and 6 represent rotations about the global x-, y- and z-axis, respectively.
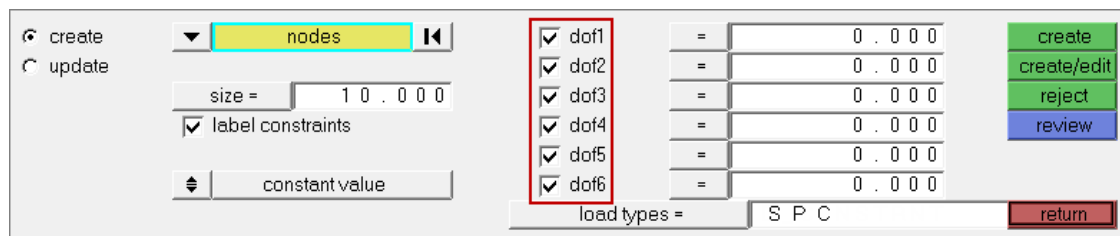
9. Click **load types =** and select **SPC**.

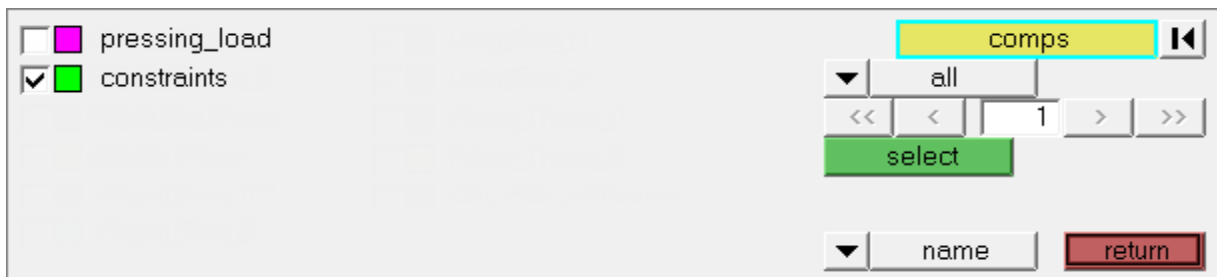10. Click **create**. Hypermesh creates constraints on the lines.

11. In the **size =** field, enter 5. HyperMesh reduces the display size of the constraints.

12. Select the **label constraints** checkbox. HyperMesh displays a label for each constraint.

    Note: The labels identify what dofs are assigned to the constraints.

13. Exit the main menu by clicking **return**.

Step 5: Map the constraints (OPTISTRUCT SPC) on the geometry lines to the channel nodes associated to the lines.

1. Open the **loads on geometry** panel by clicking **BCs** > **Loads on Geometry** from the menu bar.

2. Click **loadcols.**

3. Select the load collector, **constraints**.

4. Click **select**.

5. Click **map loads**. HyperMesh creates a constraint at each node associated to the geometry lines.
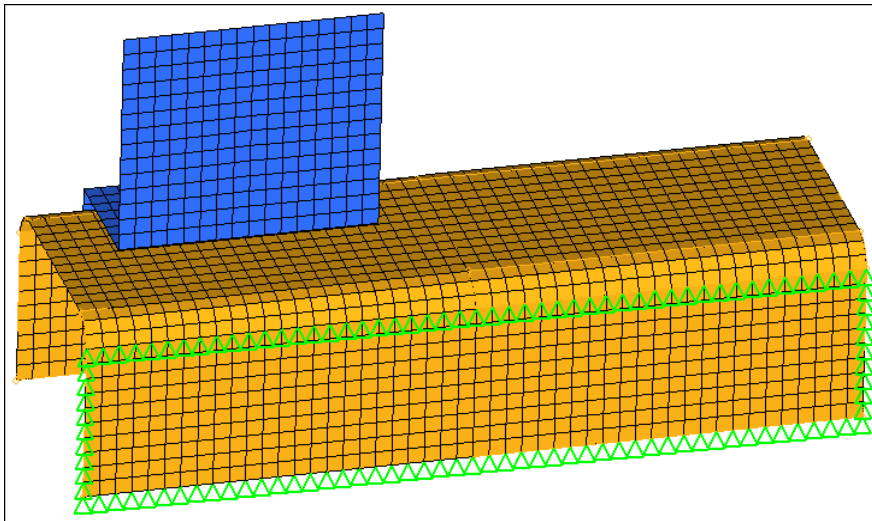
6. Click **return**.

7. In the **Model** browser, **Component** folder, turn off the display of geometry for all component collectors.

Step 6: Prepare to create forces (OPTISTRUCT FORCE) on the bracket for the pressing load case.

1. In the **Model** browser, **View** folder, right-click on *View3* and select *Show* from the context menu.

2. In the **Load Collector** folder, right-click on *pressing_load* and select *Make Current* from the context menu.

   Note: The **pressing_load** load collector is now the current load collector, and any loads created will be placed in this collector.

3. Right-click on *pressing_load* and select *Show* from the context menu.

Step 7: Create two forces (OPTISTRUCT FORCE) on the bracket for the pressing load case.

1. Open the **forces** panel by clicking *BCs* > *Create* > *Forces* from the menu bar.

2. Go to the **create** subpanel.

3. Set the entity selector to *nodes*.

4. Select the two nodes indicated in the following image.
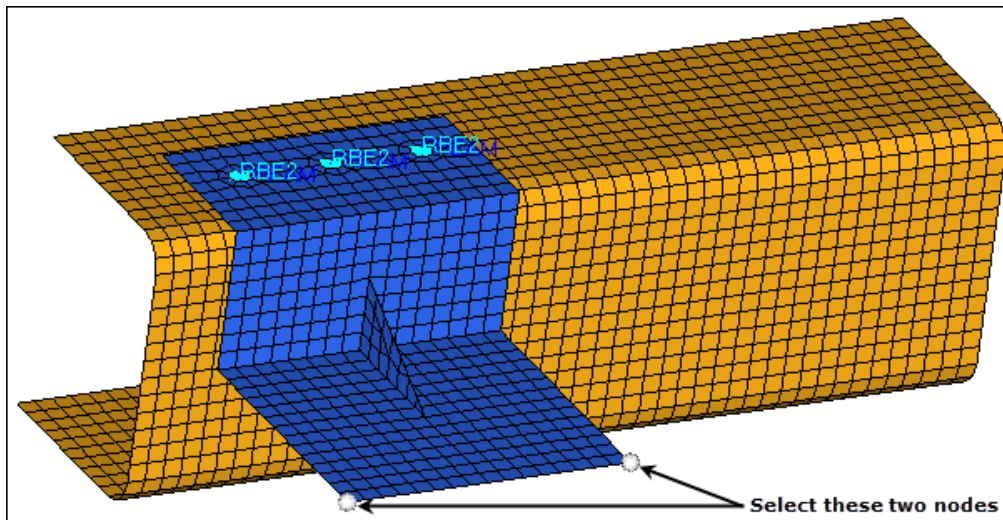
5. In the **magnitude =** field, enter 5.

6. Set the orientation selector to *y-axis*.

7. Click **load types =** and select *FORCE*.

8. Click *create*. HyperMesh creates two forces.

9. In the **magnitude % =** field, enter 200.0. HyperMesh increases the display size of the forces.

10. Select the **label loads** checkbox. Each force displays the label **FORCE = 5.00e+00**.
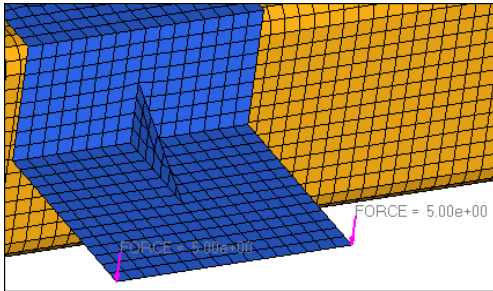
The two forces created for the pressing load case

11. Click *return*.

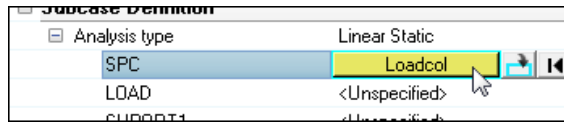## Step 8: Define the load step for the pressing load case.

1. Create a **Load Step** by right-clicking in the **Model** browser and selecting *Create* > *Load Step* from the context menu. HyperMesh creates and opens a load step in the **Entity Editor**.

2. In the **Entity Editor**:

   • For **Name**, enter `pressing_step`.

   • Set **Analysis type** to *Linear Static*.

- For **SPC**, click ***Unspecified*** >> ***Loadcol***.

- In the **Select Loadcol** dialog, select ***constraints*** and then click ***OK***.
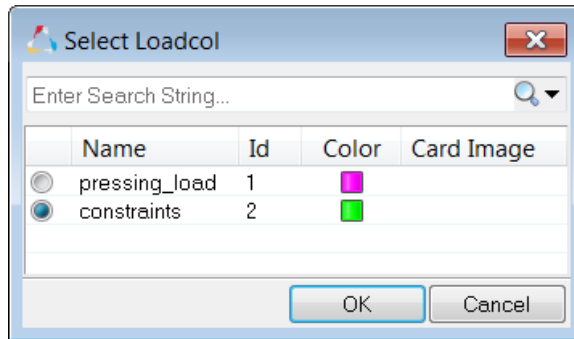
- For **LOAD**, click ***Unspecified*** >> ***Loadcol***.
- In the **Select Loadcol** dialog, select ***pressing_load*** and then click ***OK***.

Step 9: Display and mask the load step (the load collectors defined in the load step).

1. In the **Model** browser, **Load Step** folder, right-click on ***pressing_step*** and select ***Hide*** from the context menu. HyperMesh hides the **pressing_load** and **constraints** load collector

2. Right-click on ***pressing_step*** again and select ***Show*** from the context menu.