Altair

HyperWorks

Altair HyperMesh 2019 Tutorials

HM-4410: Setting Up a Model in ANSYS

HM-4410: Setting Up a Model in ANSYS

In this tutorial, you will learn how to:

- Load the ANSYS user profile
- Retrieve the HyperMesh model files for this tutorial
- Add an element type
- Apply the real constants for the elements of the model
- Apply the material properties for elements of the model
- Update each component with respective element type
- Update each component with respective real constants
- Update each component with respective material properties

The model setup includes: setting up of element type, real constants, material properties and component structure in HyperMesh for ANSYS.

Model Files

This exercise uses the `chapter2_1.hm` and `chapter2_2.hm` files, which can be found in `<hm.zip>/interfaces/ansys/`. Copy the file(s) from this directory to your working directory.

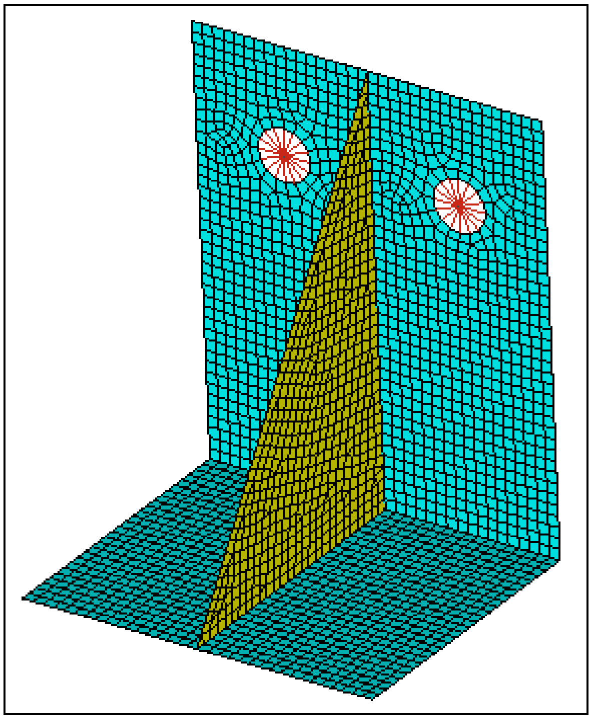
Exercise 1

Step 1: Load the ANSYS user profile

1. Start HyperMesh Desktop.
2. In the **User Profile** dialog, select **Ansys**.

Step 2: Retrieve the model file

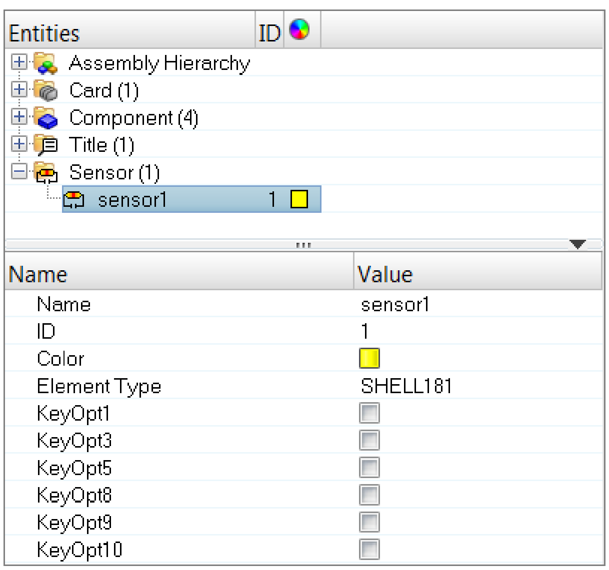
1. Open a model file by clicking ***File*** > ***Open*** > ***Model*** from the menu bar, or clicking  on the **Standard** toolbar.
2. In the **Open Model** dialog, open the `chapter2_1.hm` file.
3. If your model's elements and mesh lines are not shaded, click  on the **Visualization** toolbar.



Step 3: Add an element type

1. In the **Model** browser, right-click and select ***Create*** > ***Sensor*** from the context menu. HyperMesh creates and opens a sensor (Et Type) in the **Entity Editor**.

   Note: The **Entity Editor** displays the new Sensor's (Et Type) card details.

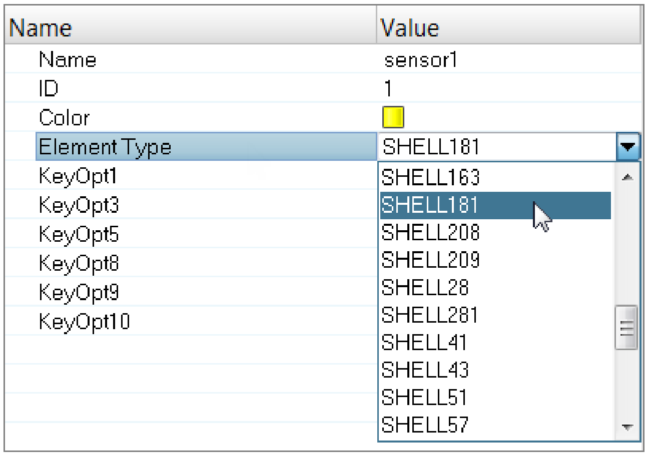


2. For **Name**, enter a new name for the Et Type.
3. Optional: For **ID**, enter a new ID for the Et Type.

   Note: By default, HyperMesh sets the ID to 1. If you create a new Et Type, HyperMesh will set the ID to n+1.

4. For **Element Type**, select a new element type.

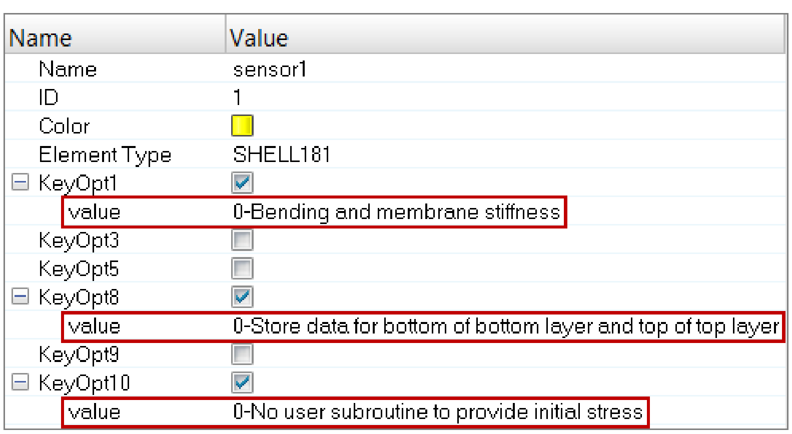
Note: By default, HyperMesh set the **Element Type** to **SHELL181**. The elements in this model are of type SHELL181, therefore you do not need to change the element type for this tutorial.



SHELL181

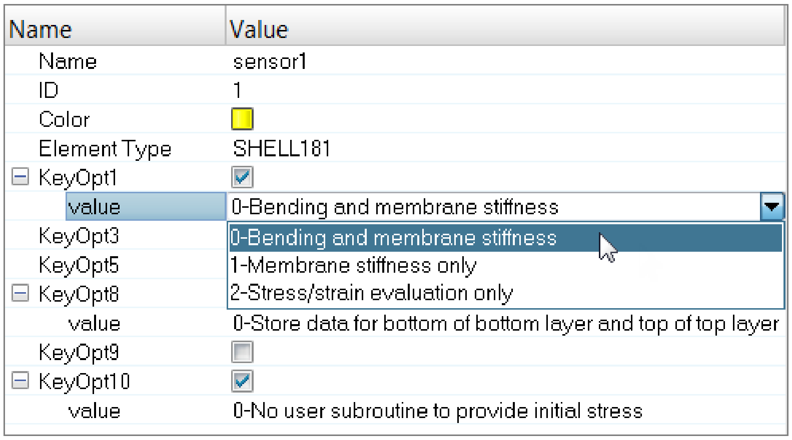
- Suitable for analyzing thin to moderately-thick shell structures. It is a 4-node element with six degrees of freedom at each node: translations in the x, y, and z direction; rotations about the x, y, and z axes (if the membrane option is used, the element has translational degrees of freedom only). The degenerate triangular option should only be used as filler elements in mesh generation.
- Well-suited for linear, large rotation, and/or large strain nonlinear applications. Change in shell thickness is accounted for nonlinear analysis. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for follower (load stiffness) effects of distributed pressures.
- May be used for layered applications for modeling laminated composite shells or sandwich construction. The accuracy in modeling composite shells is governed by the first order shear deformation theory.

5. Set the element stiffness (**KeyOpt1**), integration (**KeyOpt3**), layer data storage (**KeyOpt8**), thickness (**KeyOpt9**), and/or initial stress (**KeyOpt10**) options by selecting their corresponding checkboxes in the **Value** column. A value appears below each KeyOpt you selected.

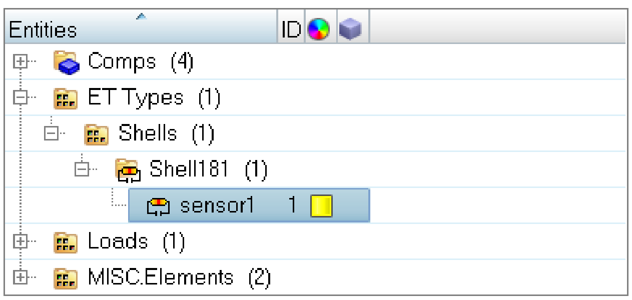


6. For each **KeyOpt** you selected, assign a **value**.

   Note: For this tutorial, use the default value assigned to each KeyOpt.

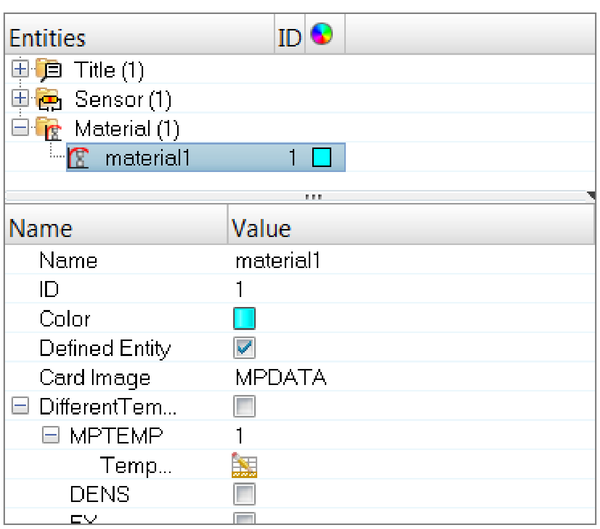


7. Open the **Solver** browser by clicking ***View*** > ***Browsers*** > ***HyperMesh*** > ***Solver*** from the menu bar.
8. Review the ET Type you just created.



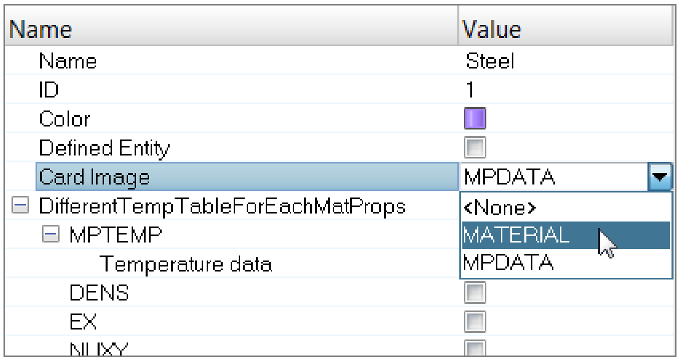
Step 4: Define material properties

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**.

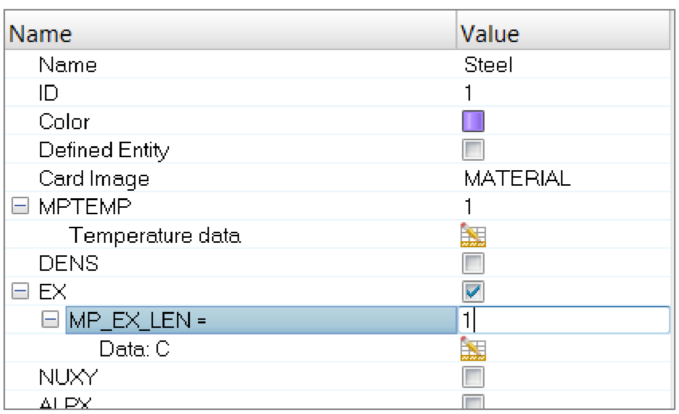


2. For **Name**, enter `Steel`.
3. Optional: For **ID**, enter a new ID.

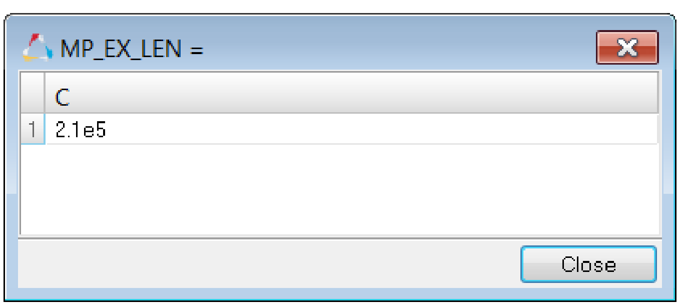
   Note: By default, HyperMesh sets the ID to 1. If you create a new material, HyperMesh will set the ID to n+1.
4. Click the ***Color*** icon, and select a color.
5. Set **Card Image** to ***MATERIAL***.



6. Select the ***EX*** (Elastic moduli) checkbox.
7. For **MP_EX_LEN** (Number of Elastic moduli to input), enter `1`.



8. Under **MP_EX_LEN=**, next to **Data: C**, click .
9. In the **MP_EX_LEN=** dialog, enter `2.1e5`.

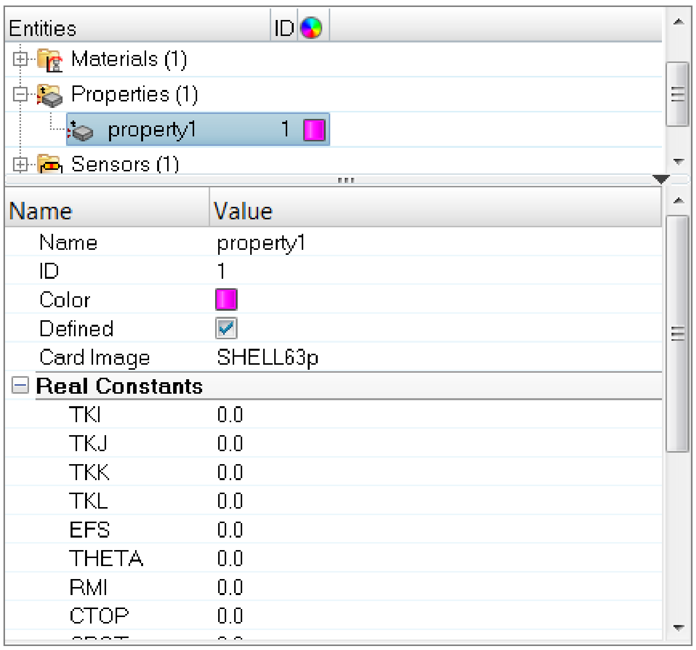


10. Click ***Close***.

11. Select the ***NUXY*** (Minor Poisson's ratio) checkbox.
12. For **MP_NUXY_LEN** (Number of Minor Poisson's ratio to input), enter `1`.
13. Under **MP_NUXY_LEN=**, next to **Data: C**, click .
14. In the **MP_NUXY_LEN=** dialog, enter `0.3`.
15. Click ***Close***.
16. Go to the **Solver** browser and review the material you just created.

Step 5: Create the section card for the shell elements in the model

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**.



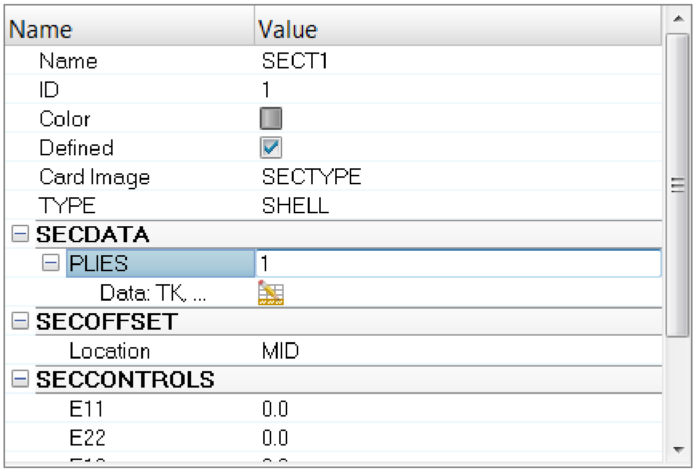
2. For **Name**, enter `SECT1`.
3. Optional: For **ID**, enter a new ID.

   Ansys sections are supported under property, with a separate ID pool. Sections are created by setting the card image to SECTYPE. All cards created with the SECTYPE card image are organized under one ID pool. Properties created with a card image other than SECTYPE are organized under another ID pool. If you want to use the same IDs in these two pools, enable the **allow duplicate IDs** option in **Preferences** > **Meshing Options**. In earlier versions of HyperMesh (14.0 or before), Ansys sections were supported under beam section collectors.

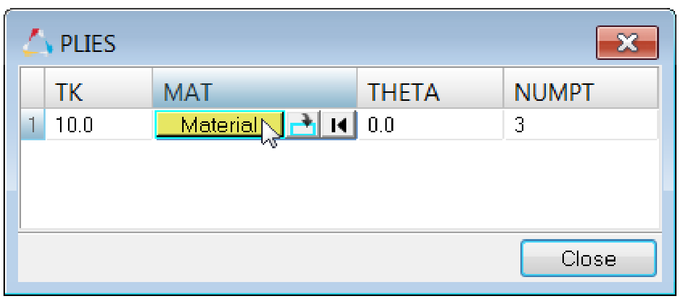
   Note: By default, HyperMesh sets the ID to 1. If you create another new beam section collector, HyperMesh will set the ID to n+1.

4. Click the ***Color*** icon, and select a new color.
5. Set **Card Image** to ***SECTYPE***.
6. Set **TYPE** to ***SHELL***.

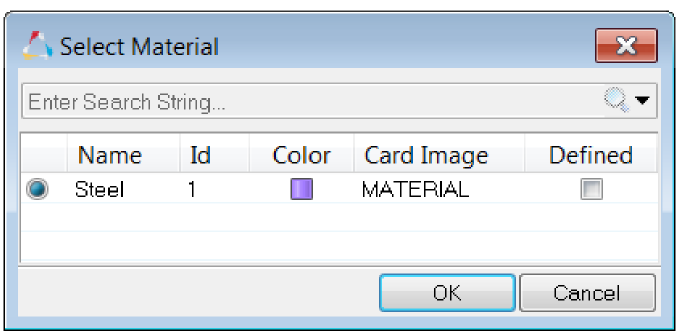
7. Under **SECDATA**, for **PLIES**, enter `1`.



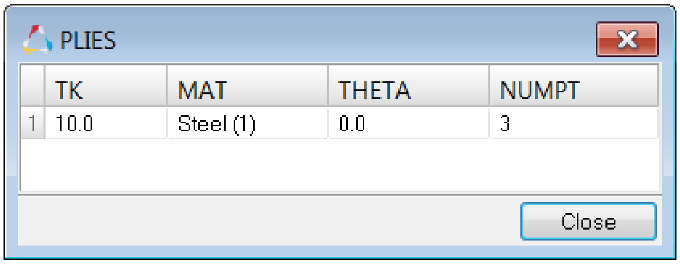
8. Under **PLIES**, next to **Data: TK**, click .
9. In the **Plies** dialog, enter `10` for **TK** (ply thickness).
10. For **MAT**, click ***Unspecified*** >> ***Material***.



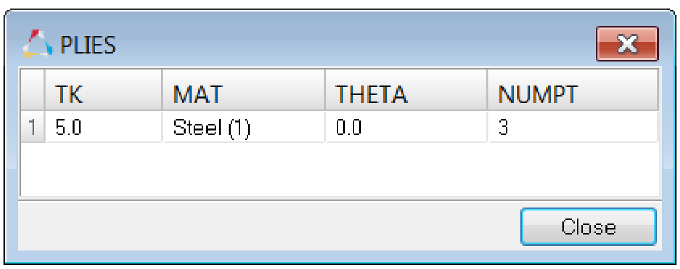
11. In the **Select Material** dialog, select ***Steel*** and then click ***OK***.



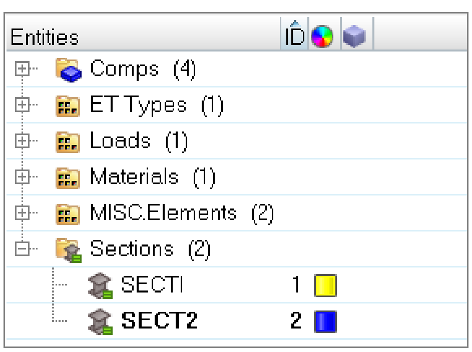
12. For **THETA** (ply angle), keep the default value `0.0`.
13. For **NUMPT** (Integration points through ply thickness), enter `3.0`.
14. Click ***Close***.



15. In the **Model** browser, **Property** folder, right-click on ***SECT1*** and select ***Duplicate*** from the context menu. HyperMesh creates a new property using the same card data as **SECT1**, except a different **Name** and **ID** are specified.
16. For **Name**, enter `SECT2`.
17. Click the **Color** icon, and select a new color.
18. Under **PLIES**, next to **Data: TK**, click .
19. In the **PLIES** dialog, change the value for **TK** from `10` to `5`.
20. Leave **MAT**, **THETA**, and **NUMPT** unchanged.
21. Click ***Close***.

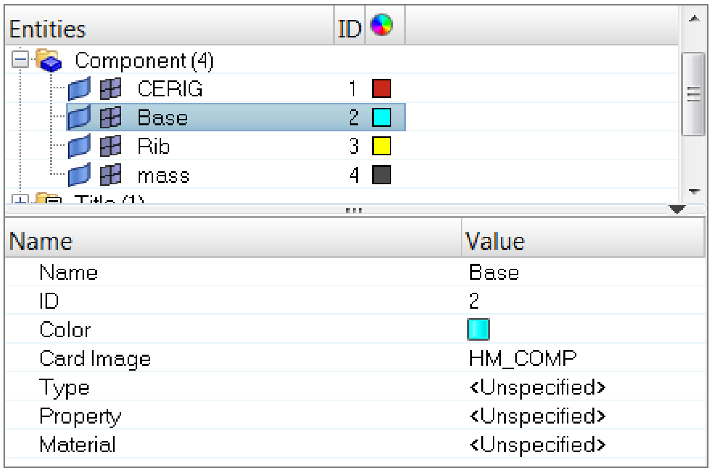


22. Go to the **Solver** browser and review the two sections you just created.

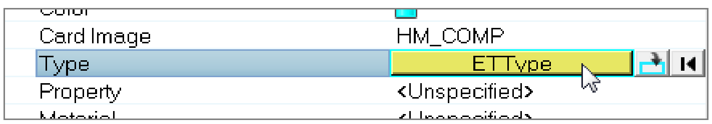


Step 6: Update each component with the respective element type, property, material, and section information

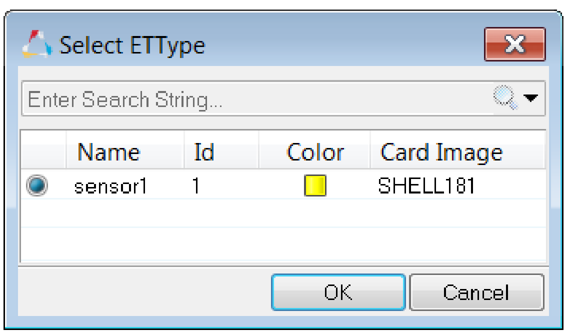
1. In the **Model** browser, **Component** folder, click ***Base***. The **Entity Editor** opens and displays the component's corresponding data.



2. For **Type**, click ***Unspecified*** >> ***ETType***.

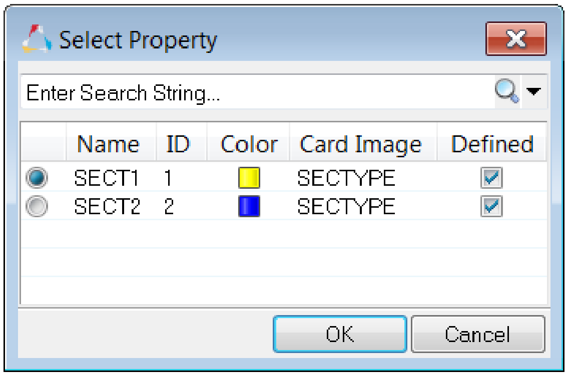


3. In the **Select ETType** dialog, select ***sensor1*** (SHELL181) and then click ***OK***.



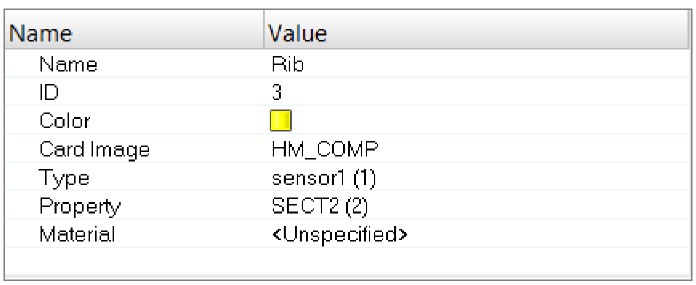
4. For **Property**, click ***Unspecified*** >> ***Property***.
5. In the **Select Property** dialog, select ***SECT1*** and then click ***OK***.

   Note: You do not have to assign a **Property** or **Material** to this component, because this information is already defined in **SECT1**.



6. In the **Model** browser, **Component** folder, click ***Rib***. The **Entity Editor** opens and displays the component's corresponding data.
7. For **Type**, click ***Unspecified*** >> ***ETType***.
8. In the **Select ETType** dialog, select ***sensor1*** (SHELL181) and then click ***OK***.
9. For **Property**, click ***Unspecified*** >> ***Property***.
10. In the **Select Property** dialog, select ***SECT2*** and then click ***OK***.

    Note: You do not have to assign a **Property** or **Material** to this component, because this information is already defined in **SECT2**.



Name	Value
Name	Rib
ID	3
Color	
Card Image	HM_COMP
Type	sensor1 (1)
Property	SECT2 (2)
Material	<Unspecified>

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

    Note: The **mass** component does not currently have a type, property, or material attached to it.

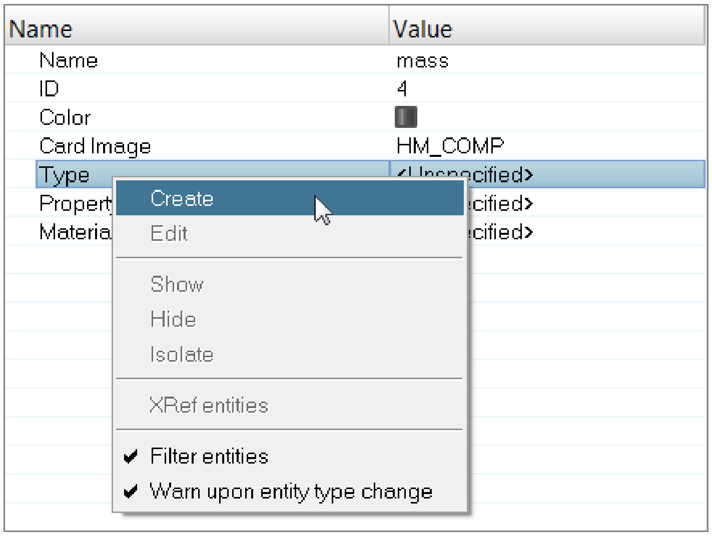
12. For **Type**, click ***Unspecified*** >> ***ETType***. You need to attach the element type **MASS21** to the **mass** component. This element type is not available in the **Select ETType** dialog because it does not exist in the model, therefore you need to create and attach it to the component.

    MASS21

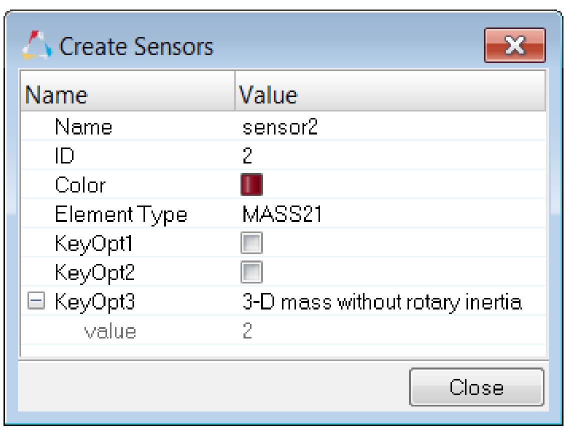
    - A point element that can have up to six degrees of freedom: translations in the nodal x, y, and z directions; rotations about the nodal x, y, and z axes. A different mass and rotary inertia may be assigned to each coordinate direction.
    - Defined by a single node, concentrate mass components (Force*Time^2/Length) in the element coordinate directions and rotary inertias (Force*Length* Time^2) about the element coordinate axes. The element coordinate system may be initially parallel to the global Cartesian coordinate system or to the nodal coordinate system (KEYOPT(2)). The coordinate system rotates with the nodal coordinate rotations

during a large deflection analysis. Options are available to exclude the rotary inertia effects and to reduce the element to a 2-D capability (KEYOPT(3)). If the element requires only the mass input, it is assumed to act in all appropriate coordinate directions

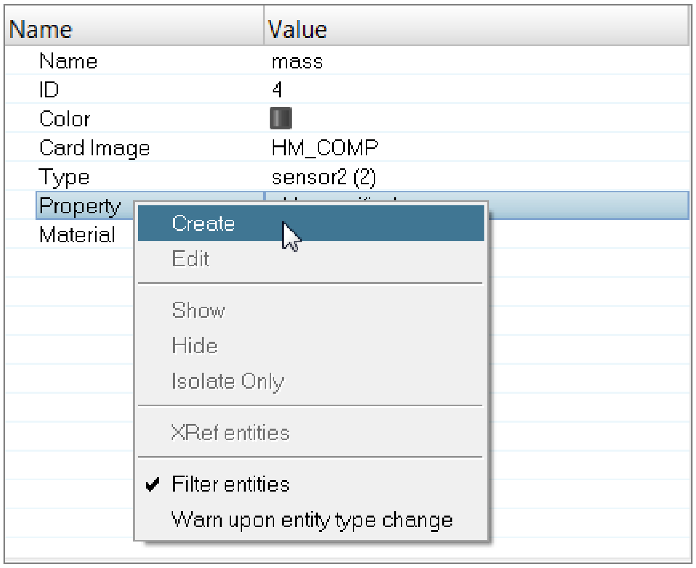
13. Click ***Cancel***.
14. Right-click on ***Type*** and select ***Create*** from the context menu. The **Create Sensors** dialog opens.



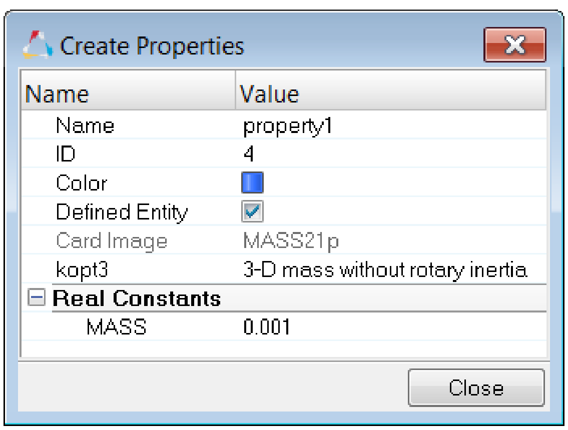
15. Set **Element Type** to ***MASS21***.
16. Set **KeyOpt3** to ***3-D mass without rotary inertia***.
17. Click ***Close***. HyperMesh creates and attaches the new sensor to the **mass** component.



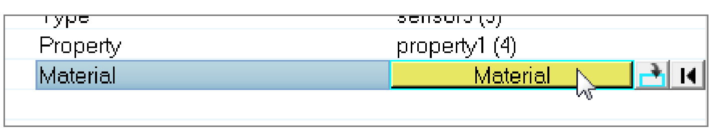
18. Create a property card that associates a small mass to the mass elements by right-clicking on ***Property*** and selecting ***Create*** from the context menu.



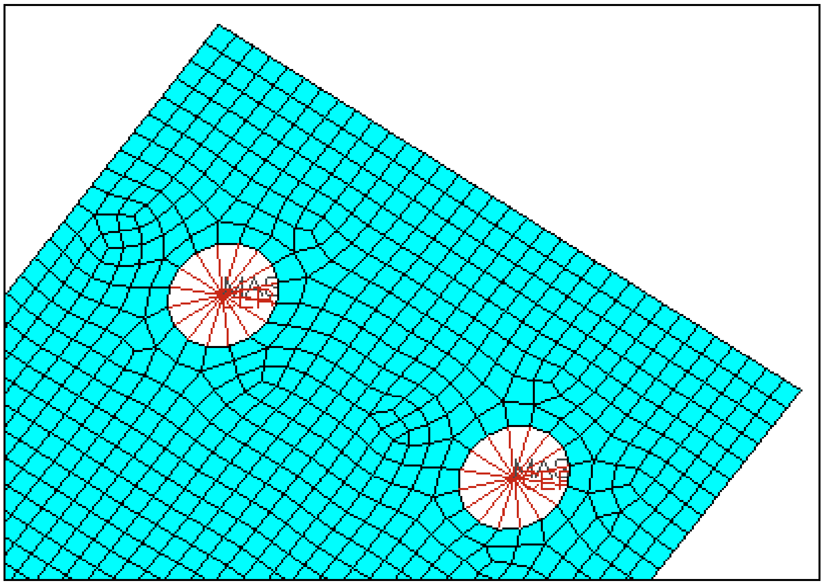
19. In the **Create Properties** dialog, the **Card Image** is automatically set to **MASS21p** because the element type attached to the **mass** component is **MASS21**.
20. Set **KeyOpt3** to ***3-D mass without rotary inertia***.
21. Under **Real Constants**, enter `0.001` for **MASS**.
22. Click ***Close***. HyperMesh creates and attaches the new property to the **mass** component.



23. For **Material**, click ***Unspecified*** >> ***Material***.



24. In the **Select Material** dialog, select ***Steel*** and then click ***OK***.
25. The component **CERIG** contains ANSYS rigid elements. These elements define the rigid region and do not require an element type, property, or material, therefore you do not have to assign a card to this component.



Step 7: Save your model

1. From the menu bar, click ***File*** > ***Save As*** > ***Model***.
2. In the **Save Model As** dialog, navigate to your working directory and save the file.
3. To apply boundary conditions and create load steps for your model, proceed to **Exercise 2**.

Exercise 2

Introduction to ANSYS Load Steps

This exercise introduces the concept of ANSYS load steps in HyperMesh. In HyperMesh, you need to have each load or constraints in a separate load collector (load cols). With the help of these load collectors, you can create multiple load steps depending on the requirement. The combination of loads with constraints, form a load step. If you have created load steps in your model, the exported `*.cdb` file will have all of the load step information. This `*.cdb` file when imported into ANSYS, automatically creates the `*.so` files in the working directory which can be used later if needed.

In this tutorial, you will learn how to:

- Load the ANSYS user profile.
- Retrieve the HyperMesh model file for this tutorial.
- Create constraint load collectors.
- Apply the constraints to the model.
- Apply the force on mass elements with force1 load collector.
- Apply the force on mass elements with force2 load collector.
- Apply the force on mass elements with force3 load collector.

- Create multiple load steps.
- Add /SOLU & LSSOLVE in control cards
- Export the deck to ANSYS *.cdb format

Optional - Step 1: Load the ANSYS user profile

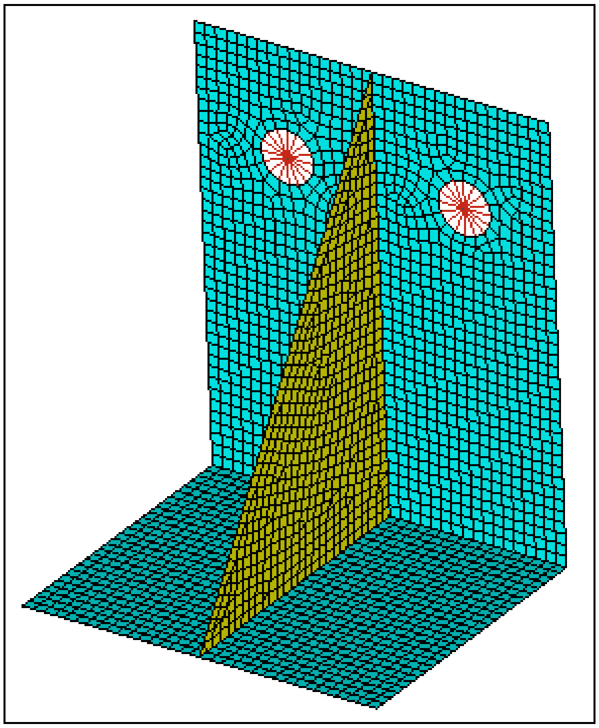
You only need to perform this step, if you did not complete Exercise 1.

1. Start HyperMesh.
2. In the **User Profile** dialog, set the user profile to **Ansys**.

Optional - Step 2: Retrieve the HyperMesh model file

You only need to perform this step, if you did not complete Exercise 1.

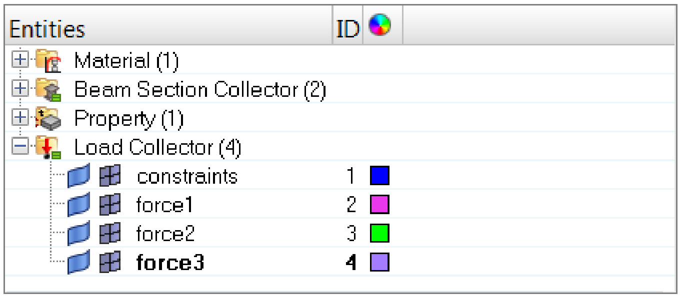
1. From the menu bar, click ***File*** > ***Open*** > ***Model***.
2. In the **Open Model** dialog, open the chapter2_2.hm file.
3. If you model's elements and mesh lines are not shaded, click  on the **Visualization** toolbar.



Step 3: Create a constraints load collector

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. For **Name**, enter constraints.

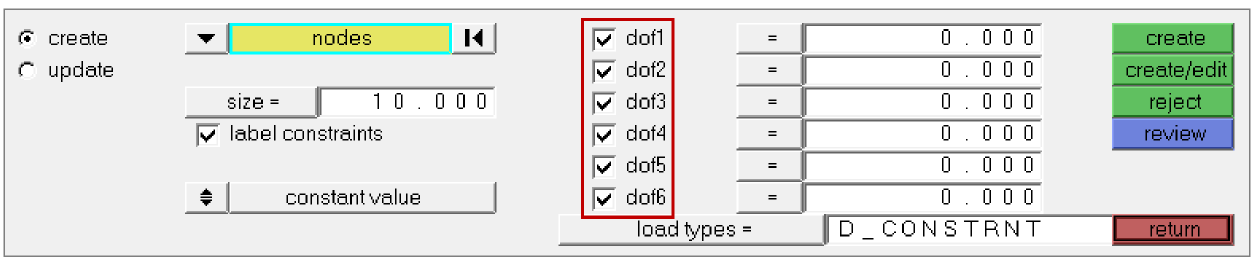
3. Click the ***Color*** icon, and select a new color for the load collector.
4. Create three more load collectors labeled **force1**, **force2**, and **force3**.



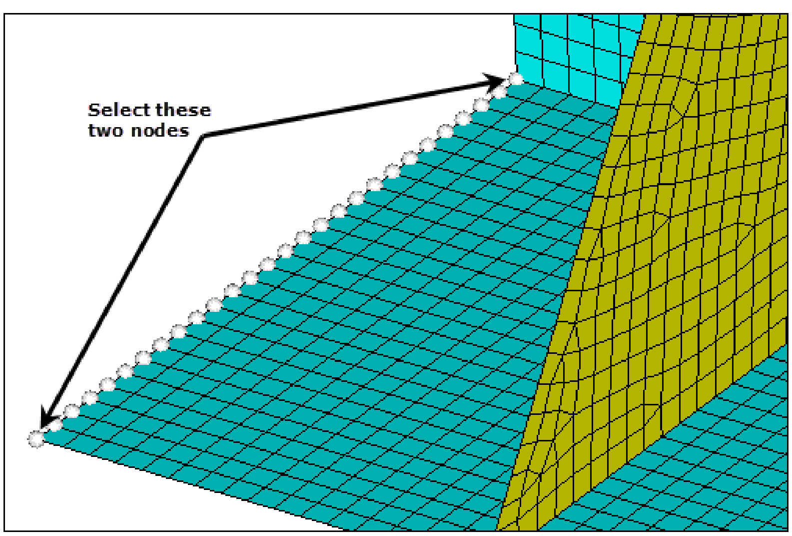
Step 4: Apply the constraints to the model

1. In the **Model** browser, **Load Collector** folder, right-click on ***constraints*** and select ***Make Current*** from the context menu.

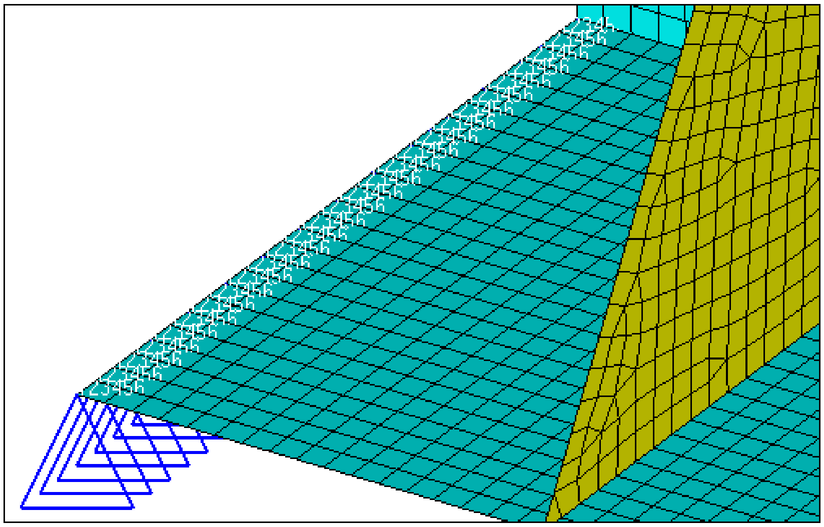
   Note: When new loads are created, Hypermesh will place them in this collector.
2. Open the **Constraints** panel by clicking ***BCs*** > ***Create*** > ***Constraints*** from the menu bar.
3. Select all of the ***dof*** (degree of freedom) checkboxes.



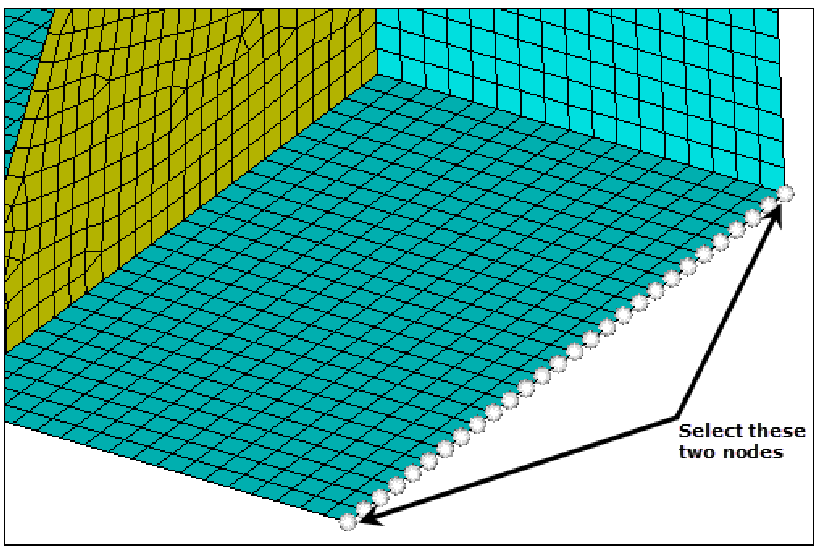
4. Click ***nodes*** >> ***by path***.
5. Select a starting node and an end node on the left side of the model as indicated in the following image.



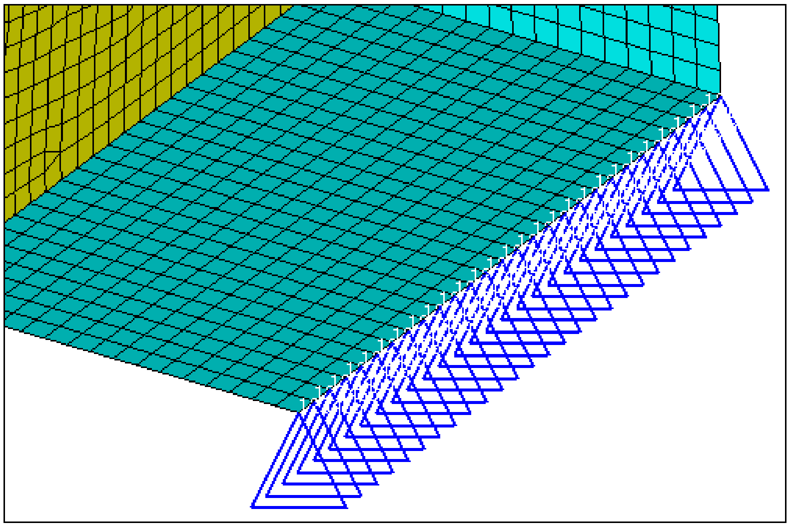
6. Click ***create***.



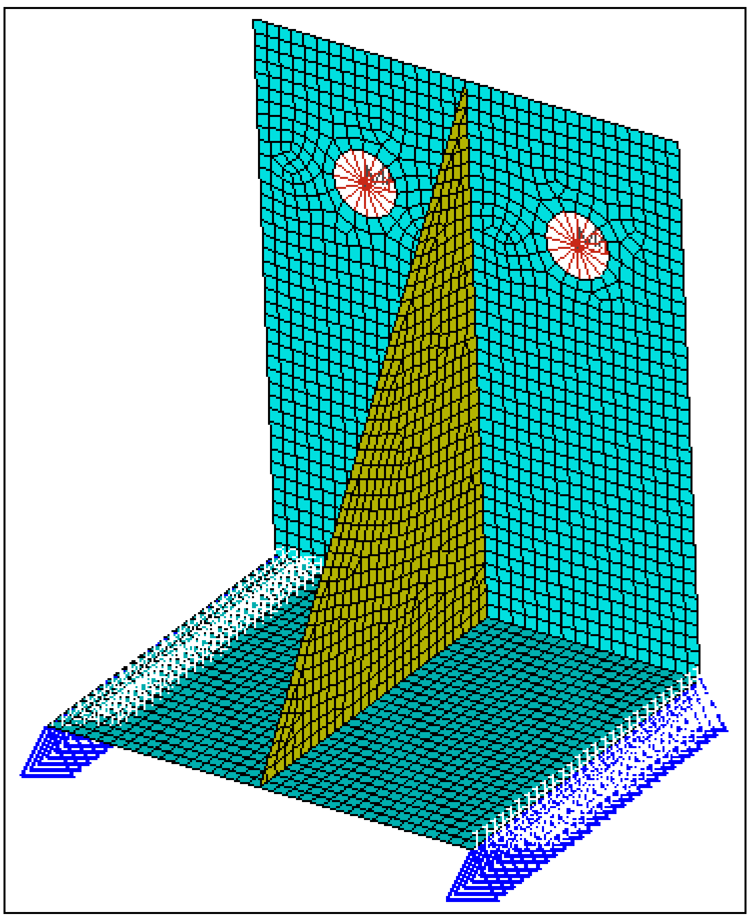
7. Repeat steps 4.4 and 4.5 to select a starting node and an end node on the right side of the model as indicated in the following image.



8. Click ***create***.

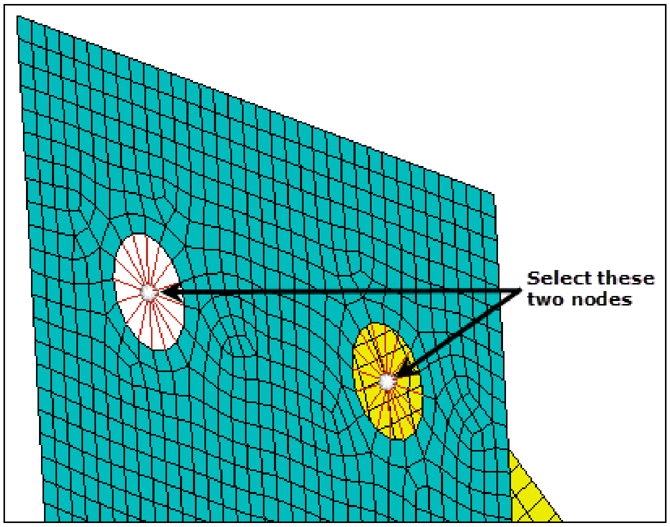


9. Click ***return*** to exit the **Constraints** panel.

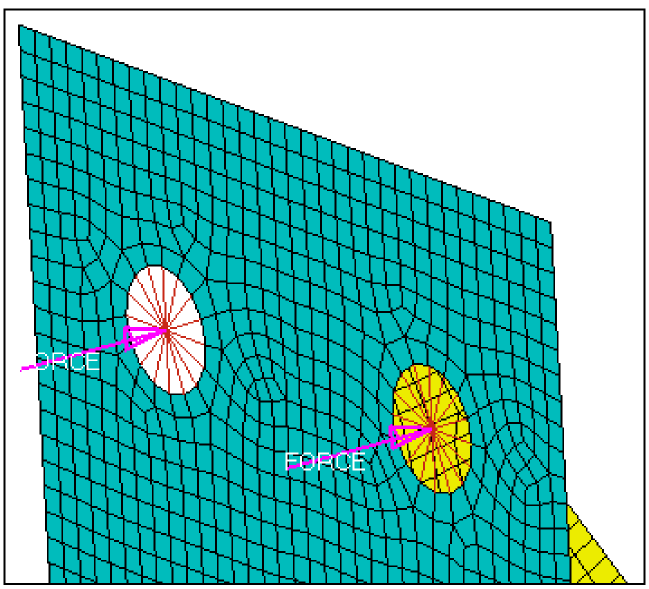


Step 5: Apply the force on mass elements with the force1 load collector

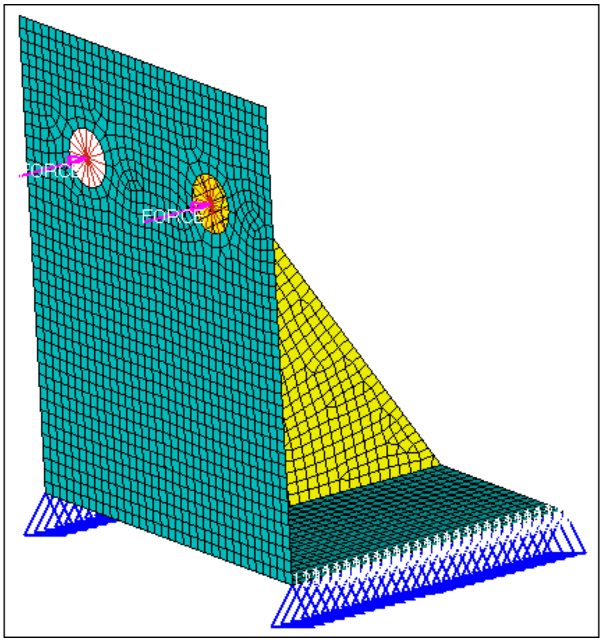
1. In the **Model** browser, **Load Collector** folder, right-click on ***force1*** and select ***Make Current*** from the context menu.
2. Open the **Forces** panel by clicking ***BCs*** > ***Create*** > ***Forces*** from the menu bar.
3. Verify that the entity selector is set to ***nodes***.
4. Select the two nodes in the center of the two bolt holes as indicated in the following image.



5. In the **magnitude=** field, enter `500`.
6. Set the orientation selector to **z-axis** for the direction of application of the force.
7. In the **uniform size=** field, enter `20`.
8. Click ***create***.

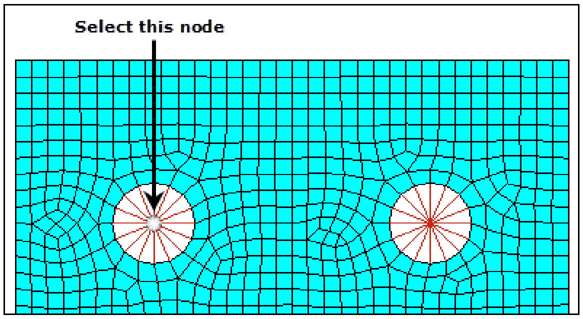


9. Click ***return*** to exit the **Forces** panel.

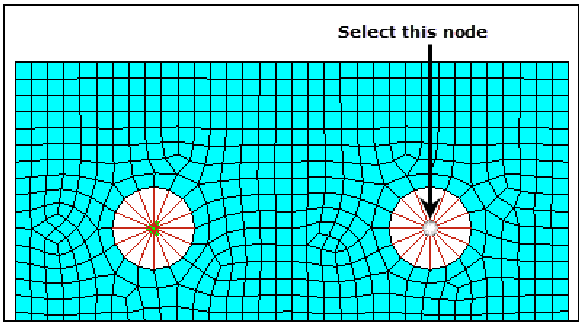


Step 6: Apply the force on mass elements with the force2 load collector

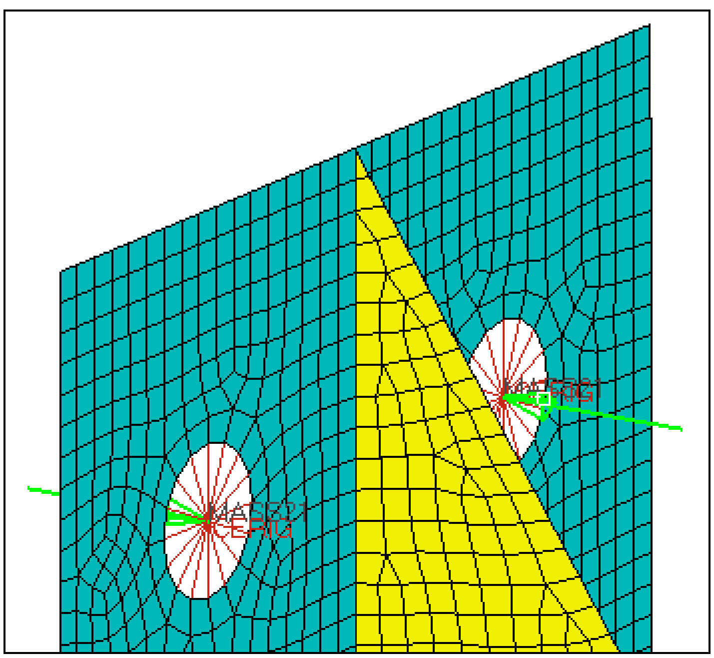
1. In the **Model** browser, **Load Collector** folder, right-click on ***force2*** and select ***Make Current*** from the context menu.
2. For better visualization, press ***F5*** to open the **Mask** panel.
3. Set the entity selector to ***loads***.
4. Select the two forces you created in step 5.8.
5. Click ***mask***.
6. Click ***return***.
7. Open the **Forces** panel.
8. Verify that the entity selector is set to ***nodes***.
9. On the **Standard Views** toolbar, click .
10. Select the left side node in the center of the bolt hole as indicated in the following image.



11. In the **magnitude=** field, enter `500`.
12. Set the orientation selector to **z-axis** for the direction of application of the force.
13. Click ***create***.
14. Select the right side node in the center of the bolt hole as indicated in the following image.



15. In the **magnitude=** field, enter `-500`.
16. Set the orientation selector to **z-axis** for the direction of application of the force.
17. Click ***create***.

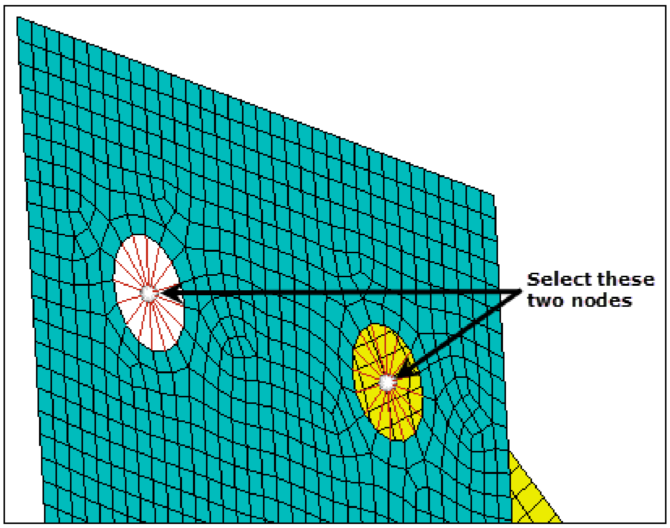


18. Click ***return*** to exit the **Forces** panel.

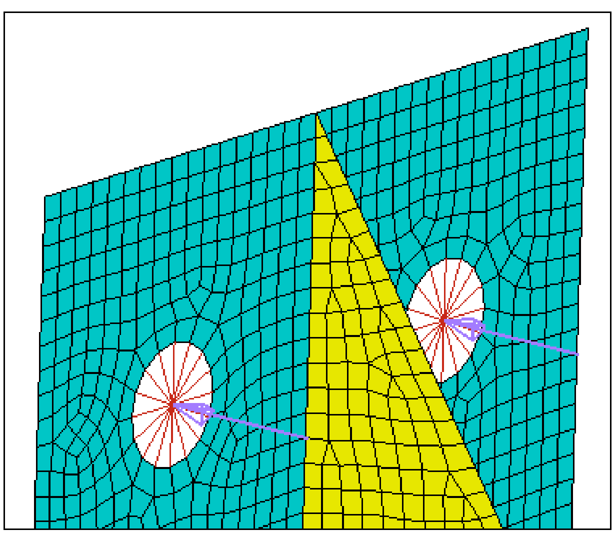
Step 7: Apply the force on mass elements with the force3 load collector

1. In the **Model** browser, **Load Collector** folder, right-click on ***force3*** and select ***Make Current*** from the context menu.

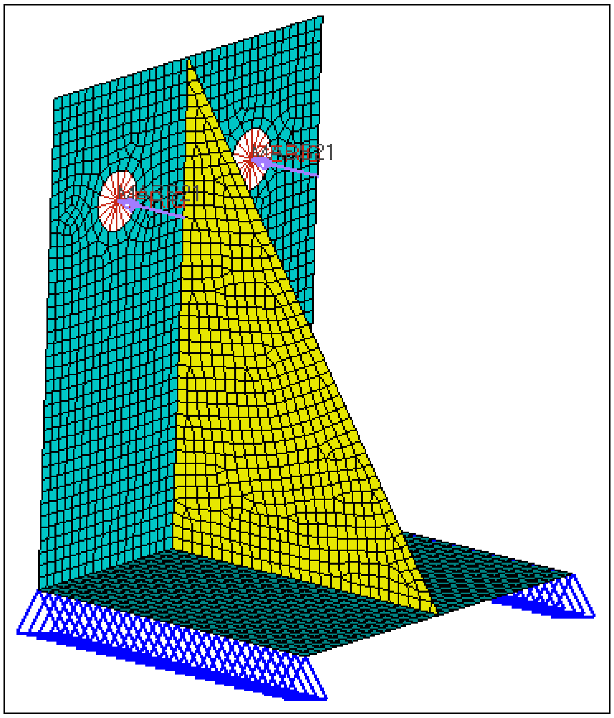
2. Open the **Mask** panel.
3. Verify that the entity selector is set to **loads**.
4. Select the two forces you created in steps 6.13 and 6.17.
5. Click ***mask***.
6. Click ***return***.
7. Open the ***Forces*** panel.
8. Verify that the entity selector is set to ***nodes***.
9. Select the two nodes in the center of the two bolt holes as indicated in the following image.



10. In the **magnitude=** field, enter `-500`.
11. Set the orientation selector to **z-axis** for the direction of application of the force.
12. Click ***create***.

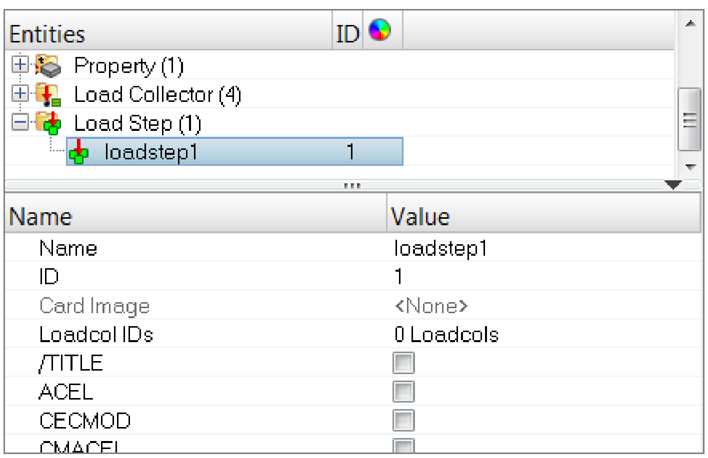


13. Click ***return*** to exit the **Forces** panel.

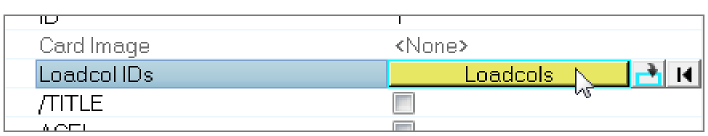


Step 8: Create multiple load steps

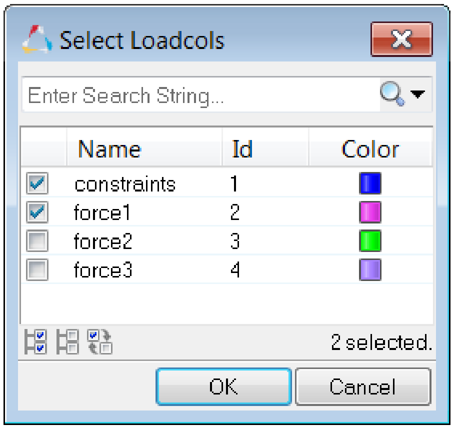
1. In the **Model** browser, right-click and select ***Create*** > ***Load Step*** from the context menu. HyperMesh creates and opens a load step in the **Entity Editor**.



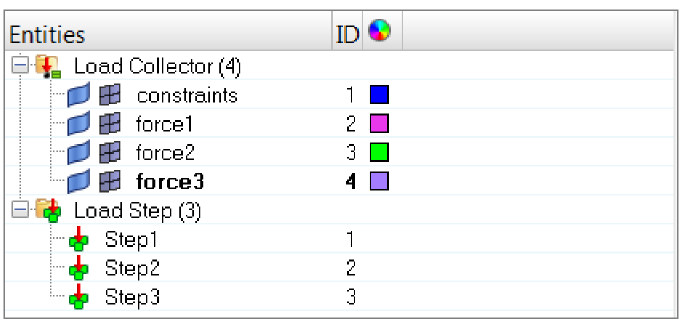
2. For **Name**, enter `Step1`.
3. For **Loadcol IDs**, click ***0 Loadcols*** >> ***Loadcols***.



4. In the **Select Loadcols** dialog, select ***constraints*** and ***force1***.



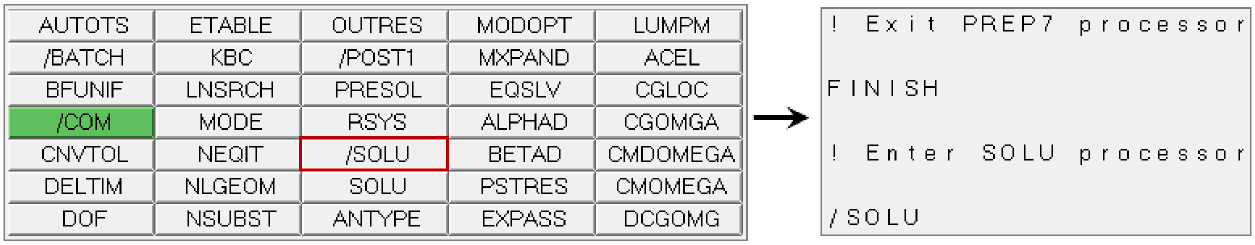
5. Click ***OK***.
6. Create a second load step labeled **Step2**, and assign it the load collectors **constraint** and **force2**.
7. Create a third load step labeled **Step3**, and assign it the load collectors **constraint** and **force3**.
8. In the **Model** browser, review the **Load Collectors** and **Load Steps** you created.



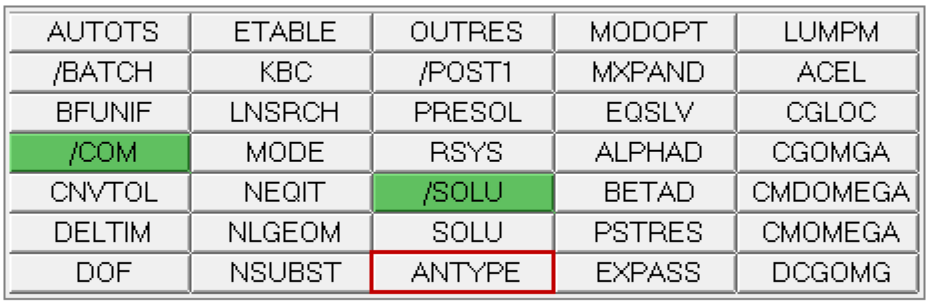
9. Open the **Solver** browser by clicking ***View*** > ***Browsers*** > ***HyperMesh*** > ***Solver*** from the menu bar.
10. Review the **Load Collectors** and **Load Steps** you created.

Step 9: Add /SOLU, ANTYPE, and LSSOLVE in the control cards

1. Open the **Control Cards** panel by clicking ***Setup*** > ***Create*** > ***Control Cards*** from the menu bar.
2. In the **Card Image**, click **/SOLU** to exit the **PREP7 preprocessor** and enter the **SOLU preprocessor**.



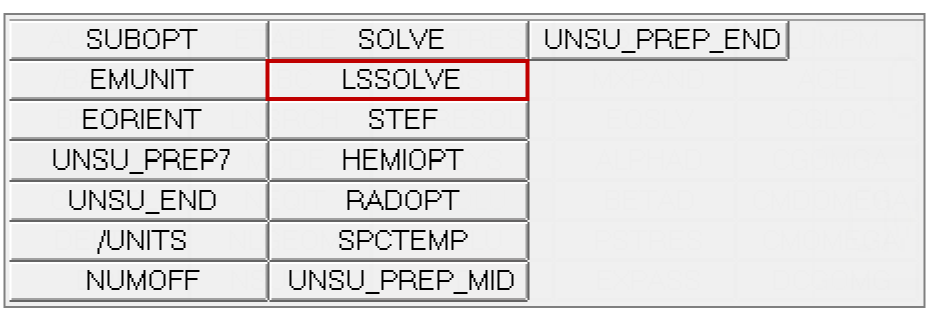
3. Click ***return***.
4. Because you are solving the model for static analysis, click ***ANTYPE***.



5. Set **type** to ***STATIC*** and **status** to ***NEW***.

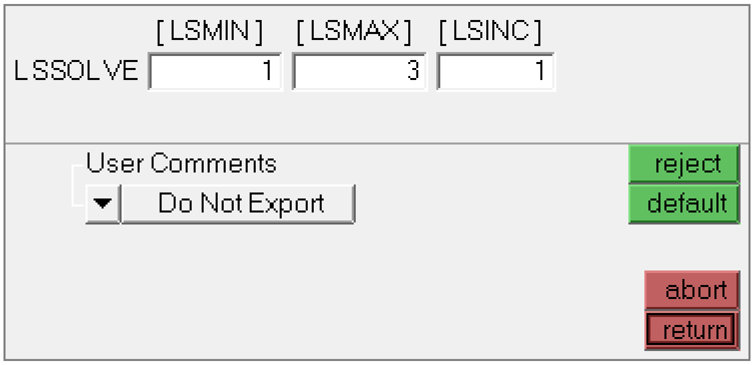
6. Click ***return***.
7. Click ***LSSOLVE***.

   Tip: If you do not see the **LSSOLVE** control card, click ***next***.



8. Set the minimum number of load steps by entering `1` in the **LSMIN** field.
9. Set the maximum number of load steps by entering `3` in the **LSMAX** field.

10. Set the load step increment by entering `1` in the **LSINC** field.



This card image commands the solver to solve all three load steps.

11. Click ***return*** to exit the card image.
12. Click ***return*** to exit the **Control Cards** panel.

Step 10: Export the deck to ANSYS `*.cdb` format.

1. Open the **Export** tab by clicking ***File*** > ***Export*** > ***Solver Deck*** from the menu bar.
2. Set **File type** to **Ansys**.

   Note: If you are in the **ANSYS** user profile, HyperMesh automatically sets the **File type** to Ansys and loads ANSYS as the default **Template**.
3. In the **File** field, navigate to your working directory and save the file as `4410_export.cdb`.
4. Click ***Export***.