Altair

HyperWorks

**HM-4450: Introduction to HyperBeam**

In this tutorial, you will learn how to:

- Assign a standard beam section using the **Section** panel.
- Assign the arbitrary beam section using the **HyperBeam** panel.
- Couple degrees of freedom.

The steps involved include:

- Loading the ANSYS user profile
- Retrieving the HyperMesh model files for this tutorial
- Adding element types
- Creating collectors
- Creating beam elements
- Coupling DOF – Rigid elements creation
- Creating standard circular beam sections
- Creating arbitrary beam sections
- Updating component collectors with respective beam section collectors

The following images are a pictorial representation of the original model. The model can be simplified in such a way that you can extract represent the entire model with a set of shell elements representing the plate part of the model and two lines indicating the beam.
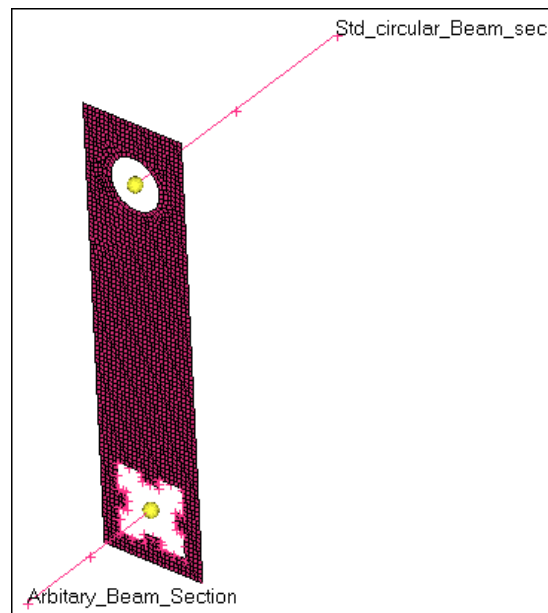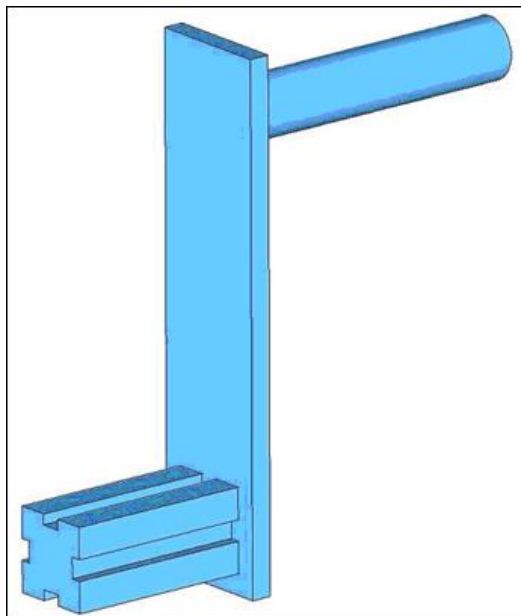
Model Files

This exercise uses the `chapter3.hm` file, which can be found in `<hm.zip>/interfaces/ansys/`. Copy the file(s) from this directory to your working directory.

Exercise

Step 1: Load the ANSYS User Profile

1. Start HyperMesh Desktop.

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

Step 2: Retrieve the HyperMesh Model File

1. From the menu bar, click *File* > *Open* > *Model*.

2. In the **Open Model** dialog, open the `chapter3.hm` file. The model shown above displays in the graphics area.

    This model contains a plate collector containing shell elements. The plate component is updated with the necessary element type, real constant, and material properties.


3. If your model's elements and mesh lines are not shaded, click 🧊 on the **Visualization** toolbar.

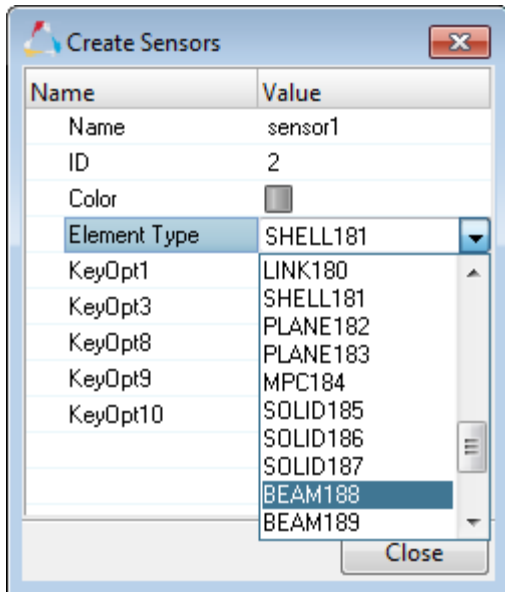### Step 3: Create Collectors and Attach Element Types and Materials to Them

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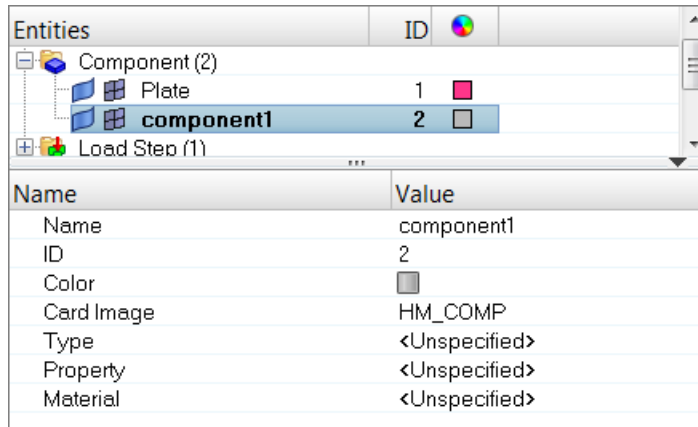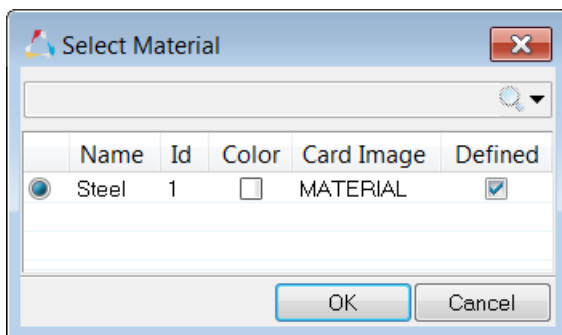

2. For **Name**, enter `beam_std`.

3. Click the *Color* icon, and select a color.

4. Right-click on *Type* and select *Create* from the context menu. The **Create Sensors** dialog opens.
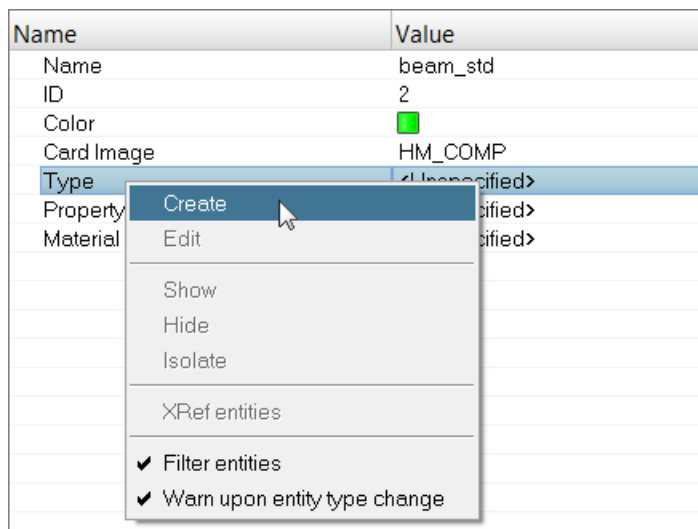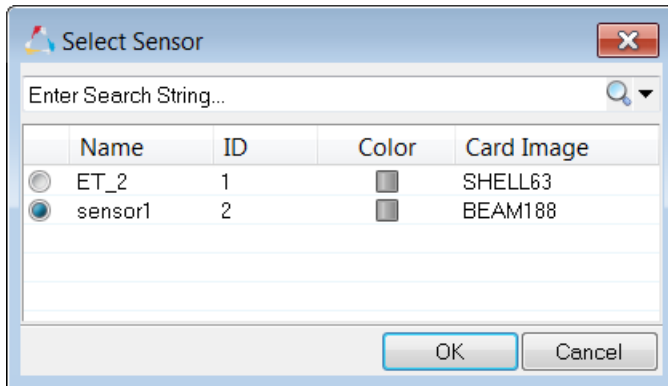

5. Set **Element Type** to *BEAM188*.

△ Altair

6. Click *Close*.

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

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

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

10. For **Name**, enter `beam_asec`.

11. Click the *Color* icon, and select a new color.
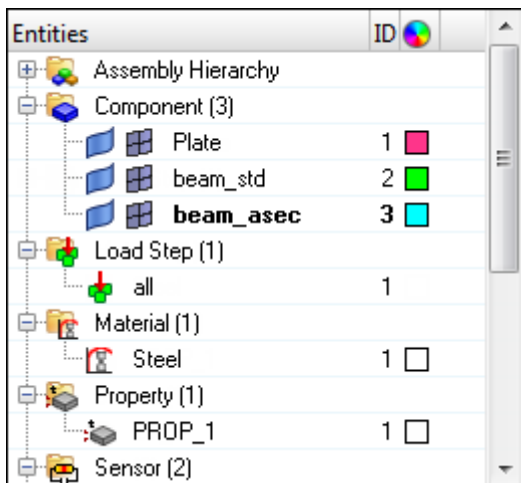
12. For **Type**, click *Unspecified* >> *Sensor*.
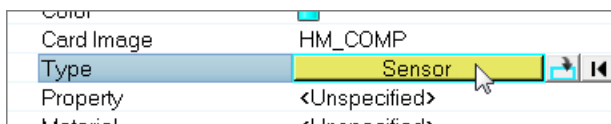
13. In the **Select Sensor** dialog, select **sensor1** (BEAM188) and then click *OK*.

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

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

Step 4: Create a Beam Element

In the **Model** browser, notice that the current component is **beam_asec**.

1. Opens the **Bars** panel by clicking **Mesh** > **Create** > **1D Elements** > **Bars** from the menu bar.

2. Go to the **bar2** subpanel.

3. Set **orientation** to **plane**.

4. Set **Beam Y Axis** to **parallel to XY**.
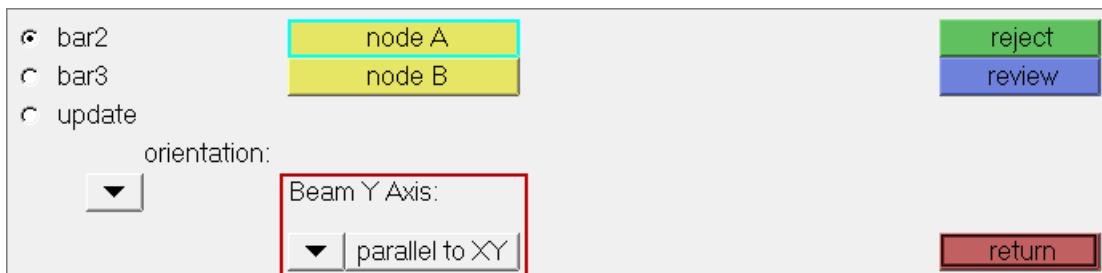
5. Using the **node A** and **node B** selectors, select the two nodes that form the ends of the **Arbitary_Beam_Section** line as indicated in the following image. HyperMesh creates a **BEAM188** element.

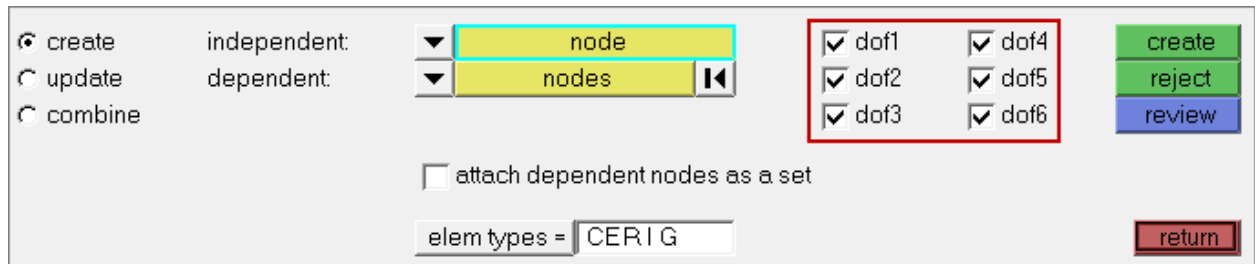   **Note**:   **Arbitary_Beam_Section** is shown as a tag in the graphics area.
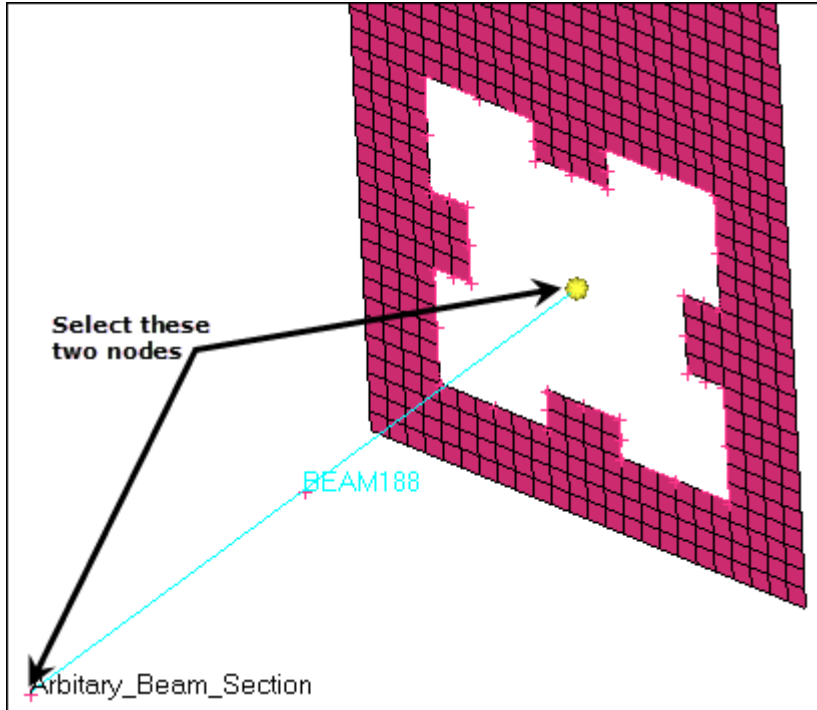
6. In the **Model** browser, **Component** folder, right-click on *beam_std* and select *Make Current* from the context menu.

7. Using the **node A** and **node B** selectors, select the two nodes that form the ends of the **Std_circular_Beam_sec** line as indicated in the following image. HyperMesh creates a **BEAM188** element.
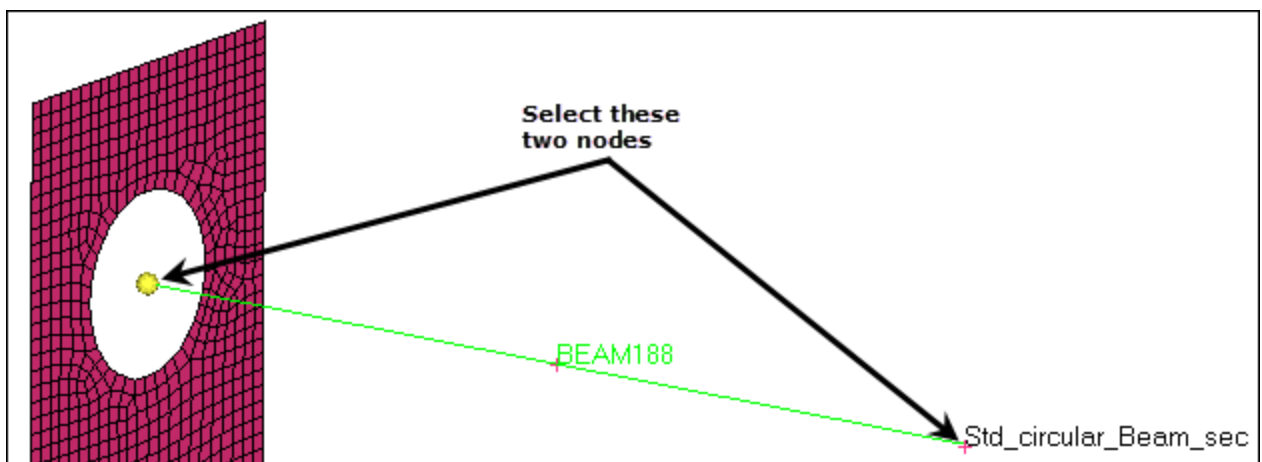

8. Click *return* to exit the **bar2** panel.

Step 5: Create Coupled DOF - Rigid Elements

1. Open the **Rigids** panel by clicking **Mesh** > **Create** > **1D Elements** > **Rigids** from the menu bar.

2. Set the **dependent** selector to **multiple nodes**.
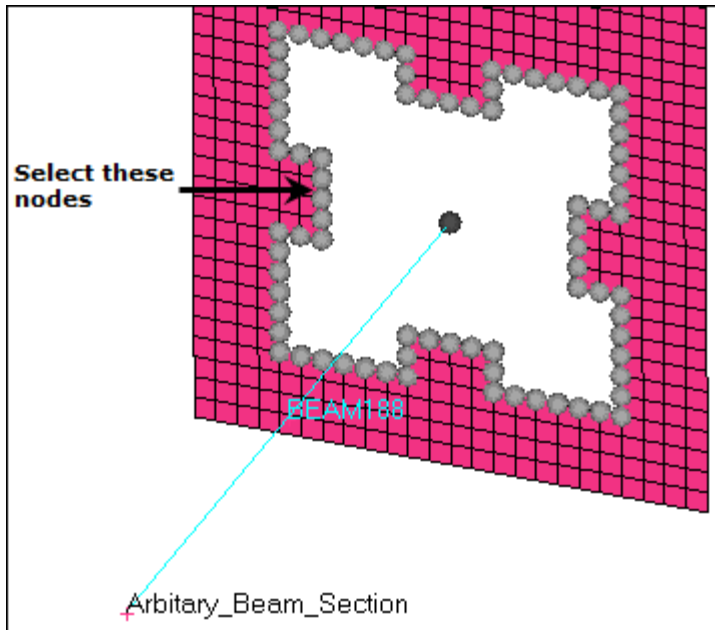
3. Select all of the **dof** checkboxes.

4. Set **elem types** to **CERIG**.

5. In the **Model** browser, **Component** folder, right-click on **beam_asec** and select **Make Current** from the context menu.

6. Activate the **independent** selector set to **node**.

7. On **Arbitary_Beam_Section**, select the independent node at the end of the beam element as indicated in the following image.
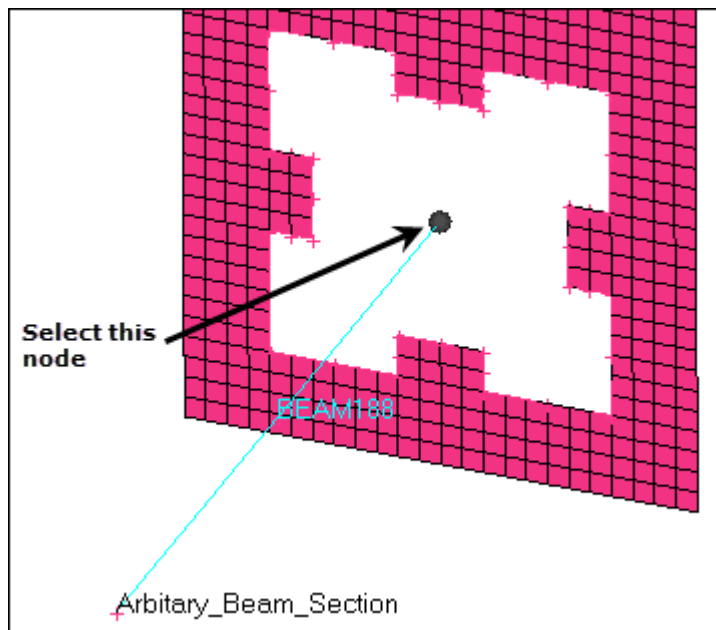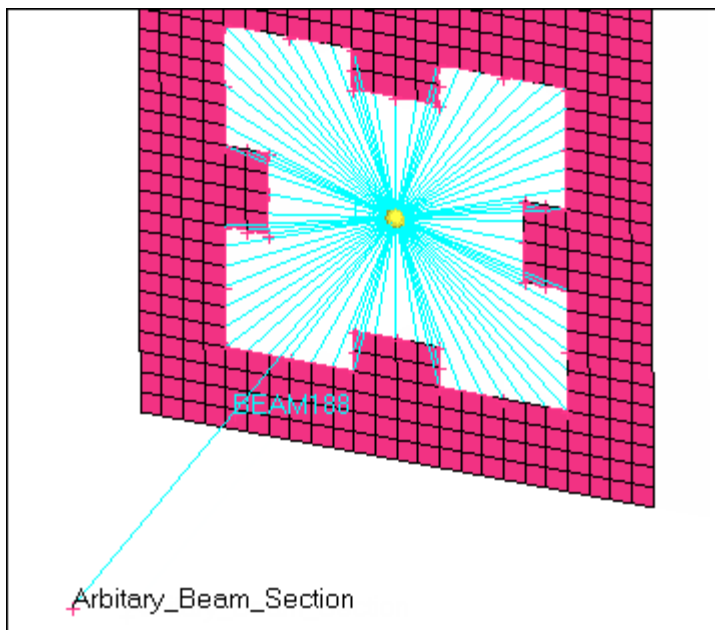

8. Click **dependent: nodes** >> **by path**.

9. On the **plate** component, select the dependent nodes indicated in the following image.
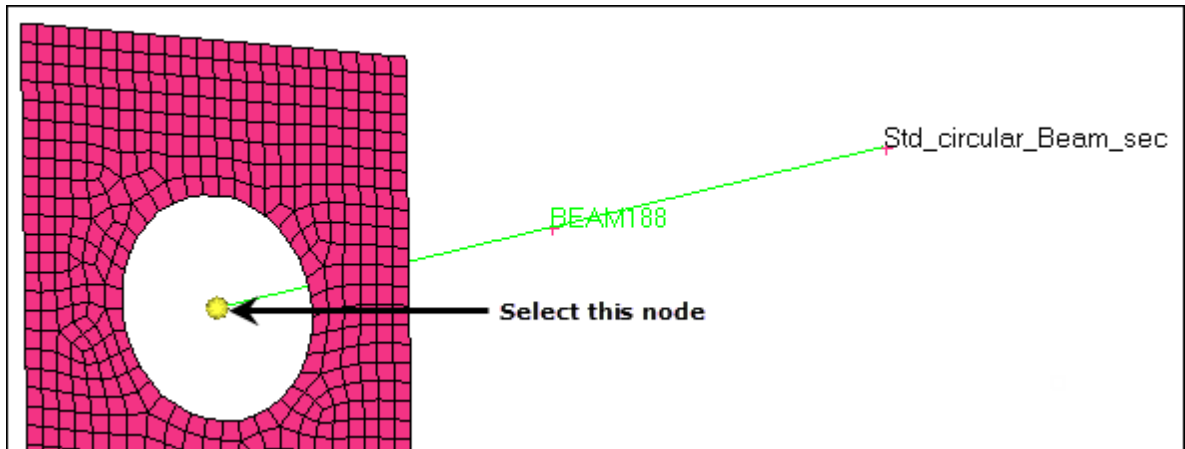
△ Altair

10. Click **create**.

11. In the **Model** browser, **Component** folder, right-click on **beam_std** and select **Make Current** from the context menu.

12. Activate the **independent** selector set to **node**.

13. On **Std_circular_Beam_sec**, select the independent node at the end of the beam element as indicated in the following image.
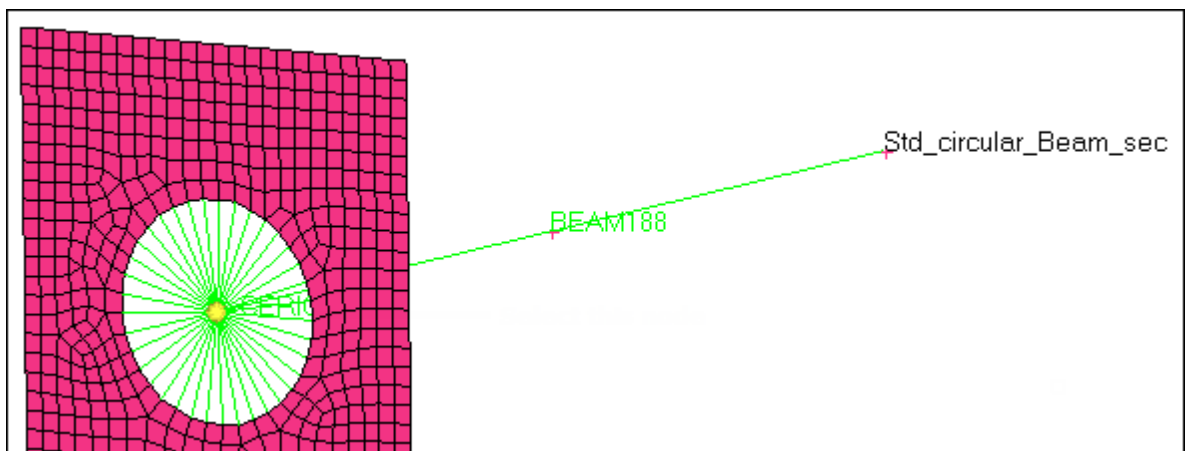
△ Altair

14. Click **dependent: nodes** >> *by path*.

15. On the **plate** component, select the dependent nodes indicated in the following image.
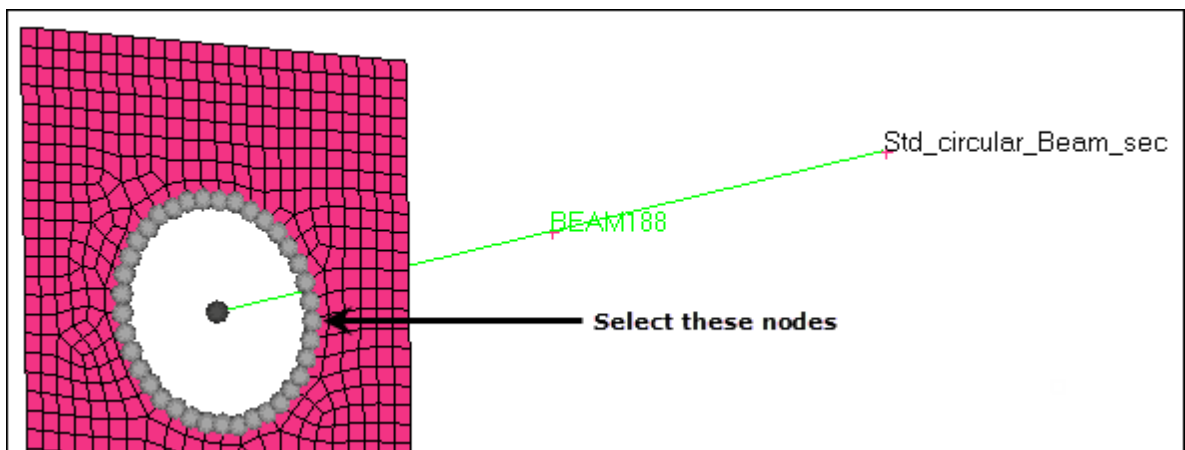
16. Click *create*.

17. Click *return* to exit the **Rigids** panel.

Step 6: Create and Attach a Standard Circle Property to a Component

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

2. Right-click on *property* and select *Create* from the context menu. The **Create Properties** dialog opens.

3. For **Name**, enter `Circular_Sec`.

4. Set **SUBTYPE** to *CSOLID*.

5. Under **SECDATA**, enter `15` for **R**.

6. Click *Close*. HyperMesh assigns the property **Circular_Sec** to the component **beam_std**.

Step 7: Create an Arbitrary Property

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 `arbi_sec`.

3. Set **Card Image** to *SECTYPE*.

4. In the **Model** browser, **Component** folder, click 🔲 next to **beam_std**, **beam_asec**, and **Plate** to turn off the display of their elements.

   Note: You should only see geometric entities in the graphics area.

5. Open the **HyperBeam** panel by clicking *Properties* > *HyperBeam* from the menu bar.

6. Go to the **solid section** subpanel.

7. Set the entity selector to *lines*.

8. Select all of the lines (displayed in gray in the following image) that form the **Arbitrary_Beam_Section**.
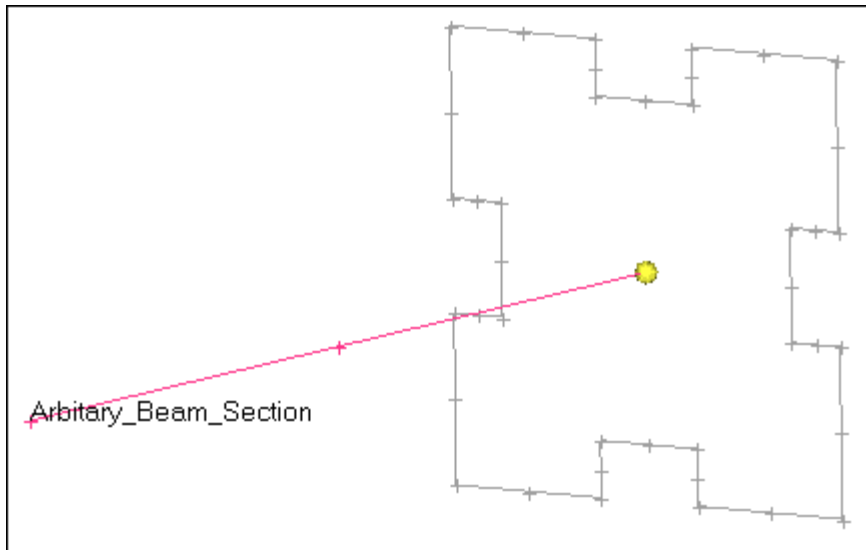
9. Set **section base node** to *centroid*.
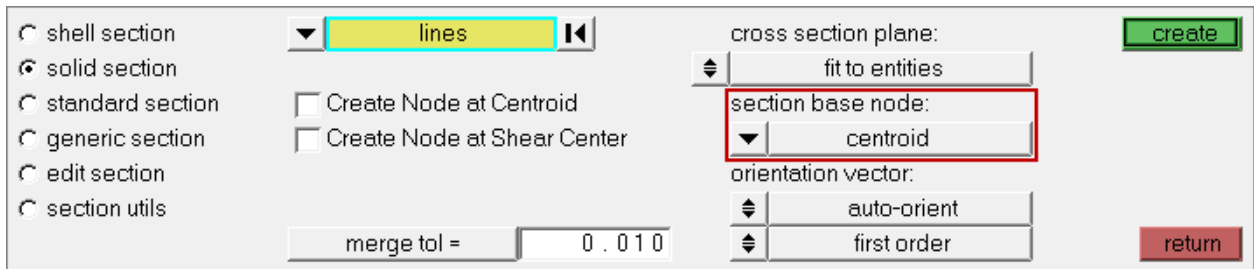
10. Click *create*. The **HyperBeam Data Model** dialog opens, meshes the area enclosed by the selected lines with quadrilateral elements, and calculates the properties using these elements.
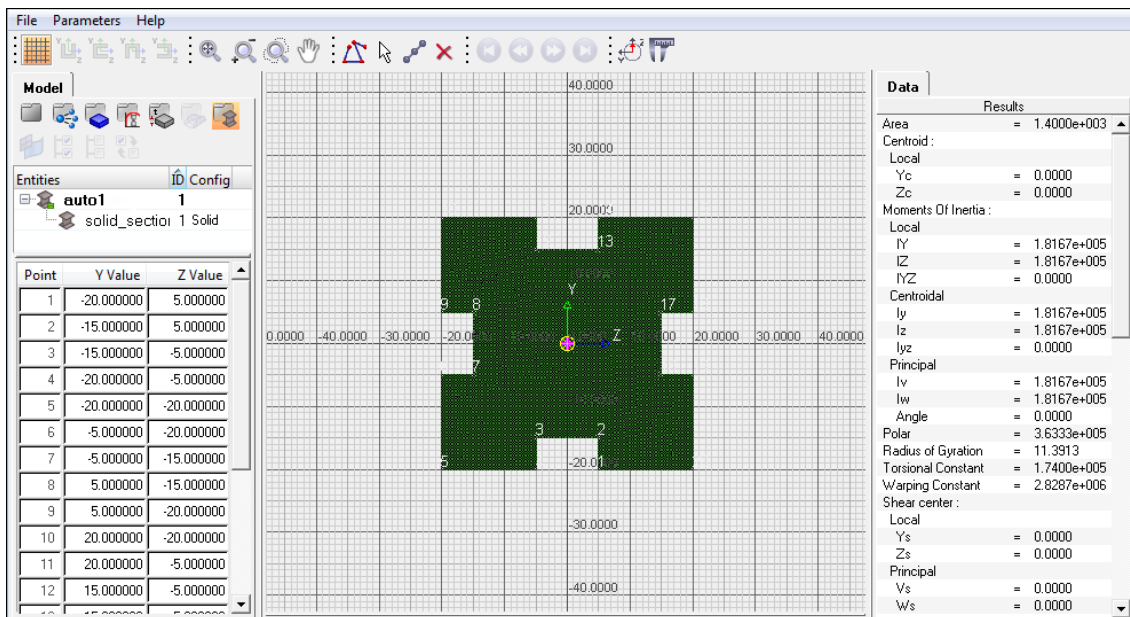
11. Return to **HyperMesh** by clicking *File* > *Exit* from the menu bar.

12. Click *return* to exit the **HyperBeam** panel.
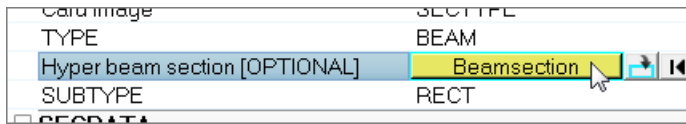
13. In the **Model** browser, **Properties** folder, click *arbi_sec*. The **Entity Editor** opens and displays the beam section's corresponding data.

14. For **Hyper beam section [OPTIONAL]**, click *Unspecified* >> *Beamsection*.

Card image	SECTTPL		
TYPE	BEAM		
Hyper beam section [OPTIONAL]	Beamsection		
SUBTYPE	RECT		

15. In the **Select Beamsection** dialog, select *solid_section.0* and then click *OK*.

Step 8: Update the Component Collector with the Beam Section

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

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

3. In the **Select Property** dialog, select *arbi_sec* and then click *OK*. HyperMesh attaches this property to the component.

4. In the **Model** browser, click 🗹 to display all of the entities in the graphics area.
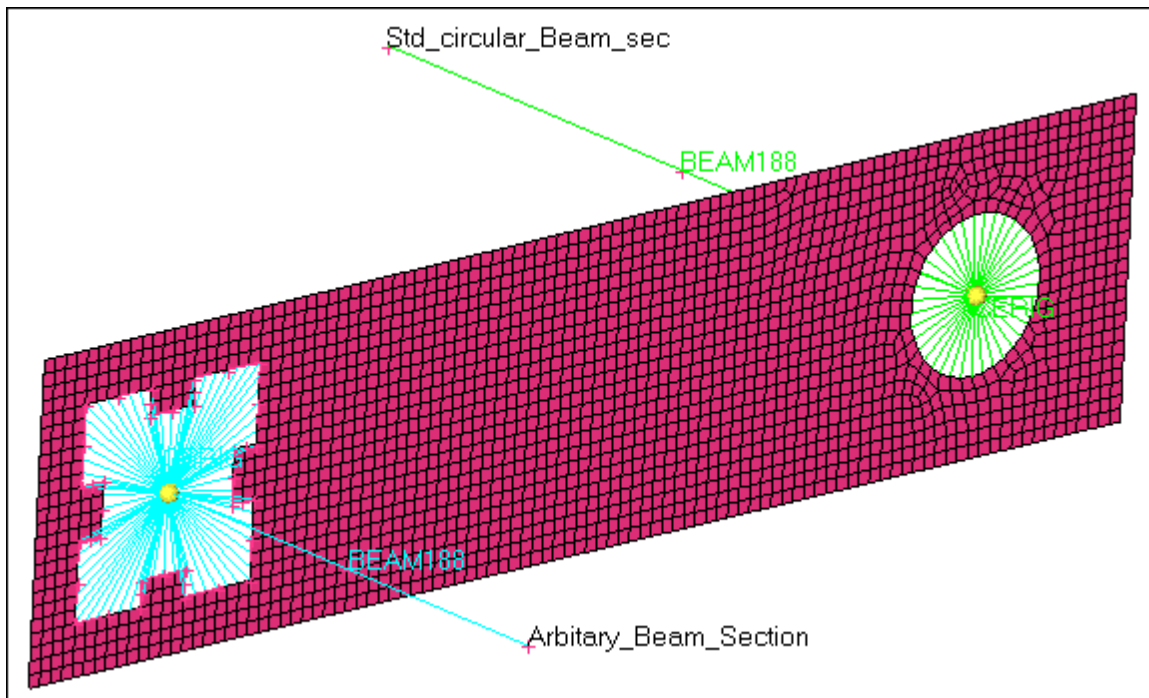
Step 9: Save you Work

1. From the menu bar, click *File* > *Save As* > *Model*.

2. In the **Save Model As** dialog, navigate to your working directory and save your file.

Altair

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
    `4450_export.cdb.`

4. Click *Export*.