



Altair

HyperWorks

HM-3210: Creating a Hex-Penta Mesh using Surfaces

In this tutorial, you will learn how to:

- Create solids using different functions
- Check and fix improper model connectivity

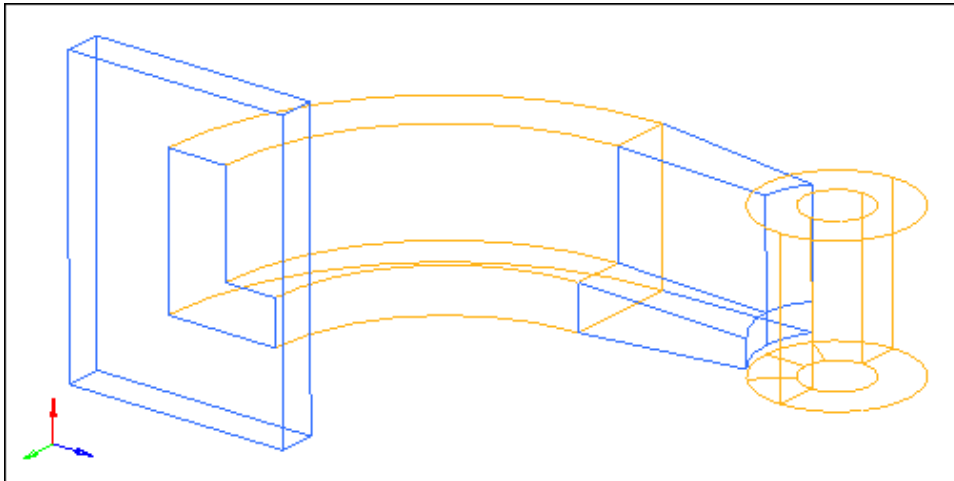
For some analyses, it is desirable to use a mesh of hexahedral and pentahedral elements. This is especially true for parts which have a large thickness compared to the element size being used, or for parts that have many features and/or changes in thickness. Castings or forgings are good examples.

Model Files

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

Exercise: Creating a Hex-Penta Mesh using Surfaces


The objective of this exercise is to introduce you to a number of HyperMesh functions that are used to create hexa-penta meshes. The `arm_bracket.hm` model is organized into four IGES layers, consisting of 1) the base, 2) the first section of the arm, with a constant cross section and curvature, 3) the second section of the arm, with a tapered cross section, and 4) the boss.

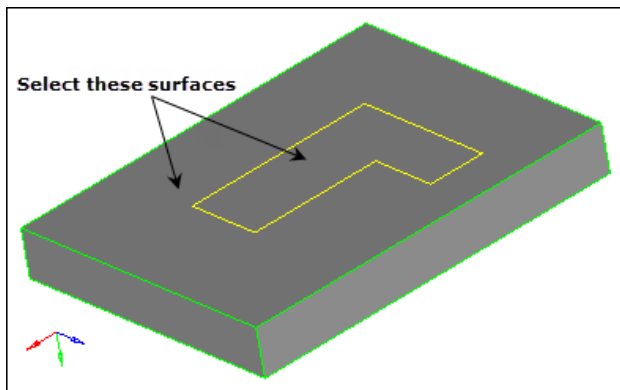


Step 1: Retrieve and review model file.

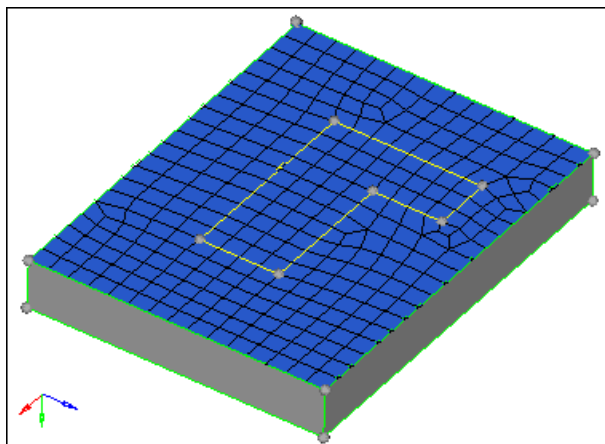
1. Start HyperMesh Desktop.
2. From the menu bar, click **File > Open > Model**.
3. In the **Open Model** dialog, open the `arm_bracket.hm` model file.

Step 2: Mesh the top surface of the base, including the L-shaped surface.

1. In the **Model** browser, **Component** folder, right-click on **base** and select **Make Current** from the context menu.
2. Right-click on **base** again and select **Isolate** from the from the context menu. HyperMesh hides all of the components except for **base**.
3. Open the **AutoMesh** panel by clicking **Mesh > Create > 2D AutoMesh** from the menu bar.
4. Go to the **size and bias** subpanel.
5. Shade the model's geometry and surface edges by clicking  on the **Visualization** toolbar.
6. Verify that the entity selector is set to **surfs**.
7. Select the surfaces on the top of the base, including the L-shaped surface at the intersection of the base and the arm.



8. Set the meshing mode to **automatic**.
9. In the **element size** field, enter 10.
10. Set the **mesh type** to **quads only**.
11. Click **mesh**. HyperMesh meshes the selected surface.

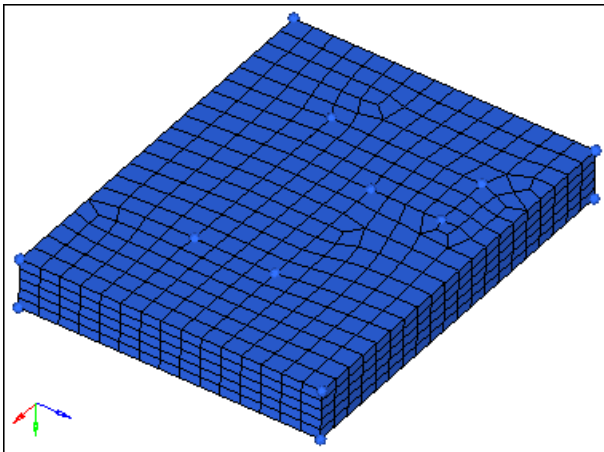


Resulting quad mesh on base surfaces

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

Step 3: Create layers of hex elements for the base.

1. Open the **Elem Offset** panel by clicking **Mesh > Create > 3D Elements > Element Offset** from the menu bar.
2. Go to the **solid layers** subpanel.
3. Click **elems to offset: elems >> displayed**. HyperMesh selects the elements on the base.
4. In the **number of layers =** field, enter 5.
5. In the **total thickness =** field, enter 25.
6. Click **offset+**. HyperMesh creates the hexa mesh.




Hex mesh on base

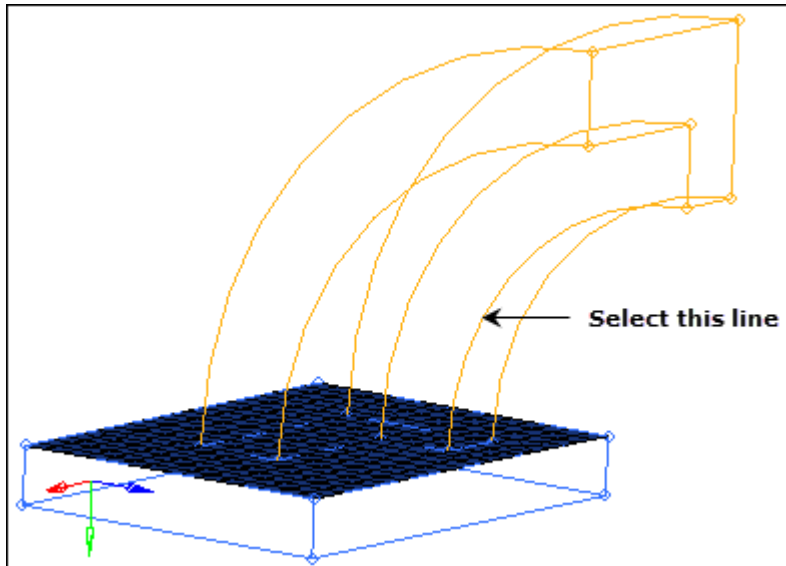
Step 4: Prepare the display for meshing the arm's curved segment.

1. In the **Model** browser, **Component** folder, right-click on **arm_curve** and select **Show** from the context menu.
2. Open the **Mask** panel by pressing **F5**.
3. Click **elems >> by config**.
4. Click **config=** and select the **hex8** configuration.
5. Click **select entities**. HyperMesh selects all of the elements with a configuration of hex8 in the model.
6. Click **mask**. HyperMesh masks the elements.
7. Return to the main menu by clicking **return**.

Step 5: Create a node at the center of the arm radius.

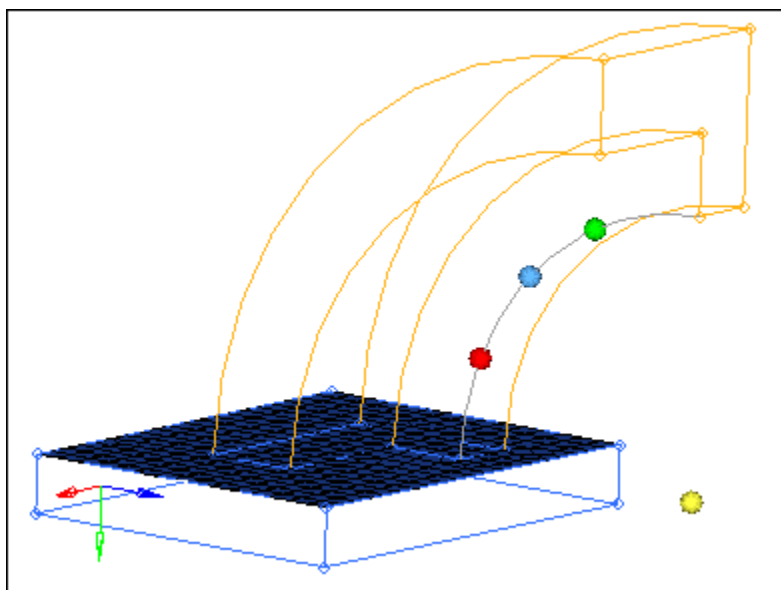
The first segment of the arm can be meshed using the **Spin** panel. This requires a node to be selected as the center point of rotation. In this step, you will use the **distance** panel, **3 nodes** subpanel to create a node that will be used as the center point of rotation.

1. Open the **Distance** panel by pressing **F4**.
2. Go to the **three nodes** subpanel.
3. Verify that the **N1** node selector is active.
4. While pressing the left mouse button, move it over the curved line as indicated in the following image, and then release it when the cursor changes to . HyperMesh highlights the line.



5. Click three locations along the selected line. The active selector advances from **N1** to **N2** to **N3**, and HyperMesh create the temporary nodes on the selected curved line of the arm.
6. Click **circle center** . HyperMesh creates the node at the center.

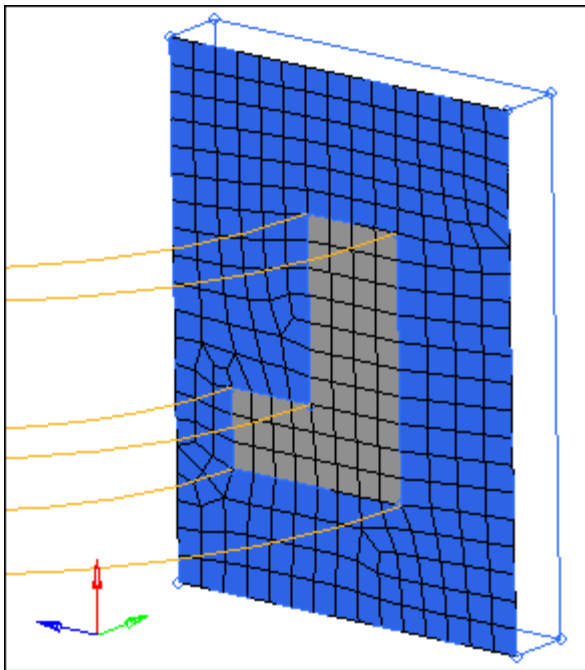
Note: You will use this node in the next step when you mesh the arm.



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

Step 6: Create hexa elements in the curved portion of the arm using spin.

1. In the **Model** browser, **Component** folder, right-click on **arm_curve** and select **Make Current** from the context menu.
2. Open the **Spin** panel by clicking **Mesh > Create > 3D Elements > Spin** from the menu bar.
3. Go to the **spin elems** subpanel.
4. Click **2d elems: elems >> by window**.
5. Select the plate elements within the L-shaped cross section of the arm as indicated in the following image.

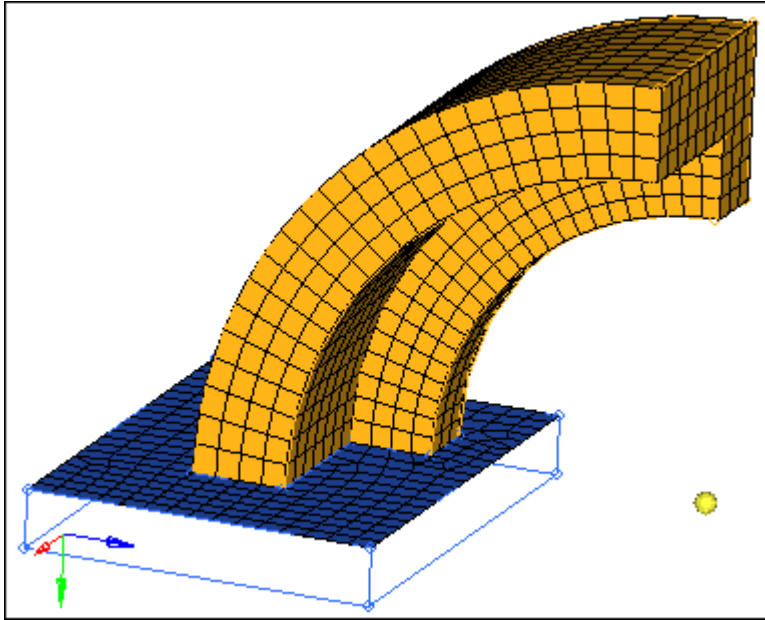


6. Click **select entities**.
7. In the **angle =** field, enter 90 degrees.
8. Set the **orientation vector** to **x-axis** (Y-Z plane).
9. Select the center node you created in step 5 for the base node (**B**).
10. In the **on spin =** field, enter 24.

Note: This option determines how many layers of hex elements HyperMesh creates when the plate elements are spun.

11. Click **spin -**.

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




spin panel results

Step 7: Create faces on the hex elements.

- Open the **Faces** panel by clicking **Mesh > Check > Components > Faces** from the menu bar.
- Set the entity selector to **comps**.
- Click **comps >> arm_curve**.
- Click **select**.
- Click **find faces**. HyperMesh creates 2D shell elements on the free faces of every 3D solid element in the component, and places them in a new component named **^faces**.

Note: The **^faces** component is created with its visualization set to wireframe, therefore you will not be able to see the new elements right away if the **arm_curve** component is displayed and in shaded mode.

- Shade the model's elements and mesh lines by clicking  on the **Visualization** toolbar. The graphics area now displays the elements in the **^faces** component.

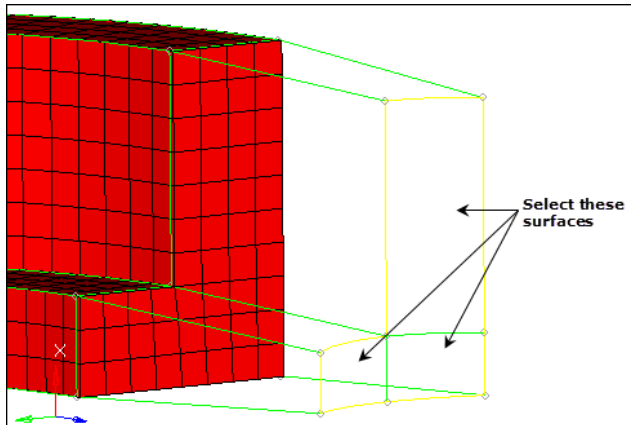
Step 8: Prepare the display for meshing the second arm segment.

- From the **Model** browser, turn on the display of the **arm_straight** and **^faces** components.

Step 9: Mesh the L-shaped set of surfaces between the arm_straight and boss components.

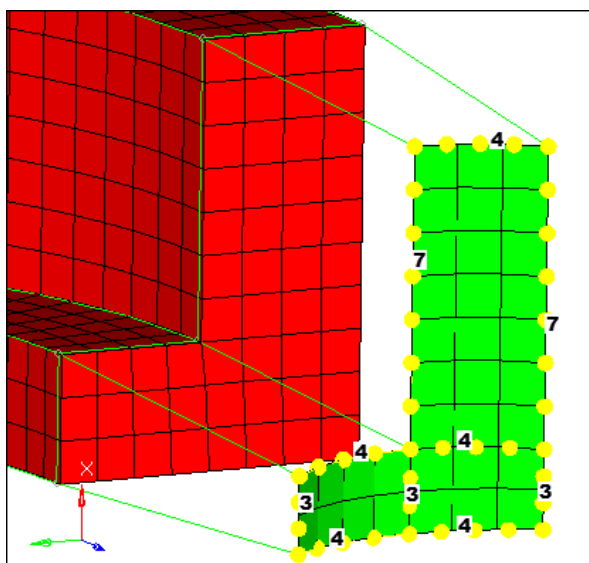
1. In the **Model** browser, **Component** folder, right-click on **arm_straight** and select **Make Current** from the context menu.
2. Open the **Automesh** panel.
3. Select the three surfaces lying on the intersection between the **arm_straight** and **boss** components as indicated in the following image.

Note: These surfaces are all in the **arm_straight** component.



4. Set the meshing mode to **interactive**.
5. Click **mesh**. The meshing module opens.
6. Go to the **density** subpanel.
7. Adjust the density of each edge to obtain a mesh that matches the following image.

Note: This mesh pattern matches the mesh pattern at the intersection of the two arm segments. This is necessary for the next step.



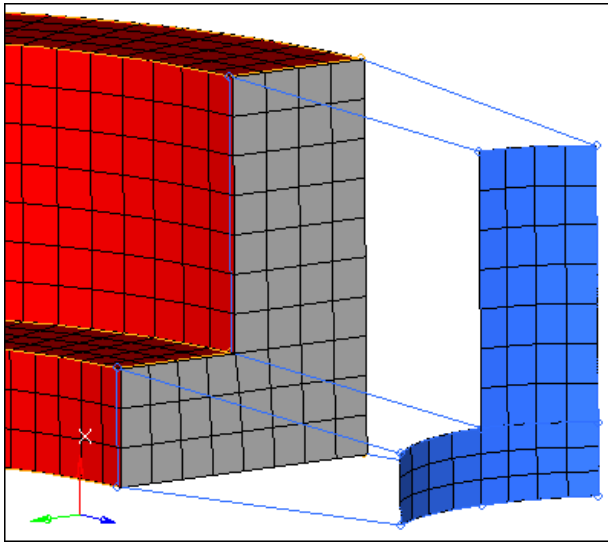
Densities to correspond to the mesh on the end face

8. Click **mesh**. HyperMesh updates the mesh density.
9. Create the elements and go back to the main menu by clicking **return** twice.

Step 10: Use linear solid to build the mesh between the two sets of shell elements.

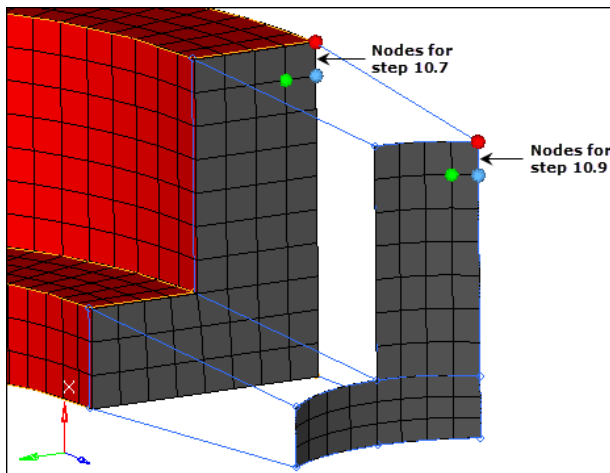
1. Open the **Linear Solid** panel by clicking **Mesh > Create > 3D Elements > Linear 3D** from the menu bar.
2. Activate the **from: elems** selector.
3. Select the **^faces** elements lying on the intersection between the first and second arm segments as indicated in the following image.

Tip: Quickly select all of the necessary elements by selecting one of the elements and then clicking **from: elems >> by face**.



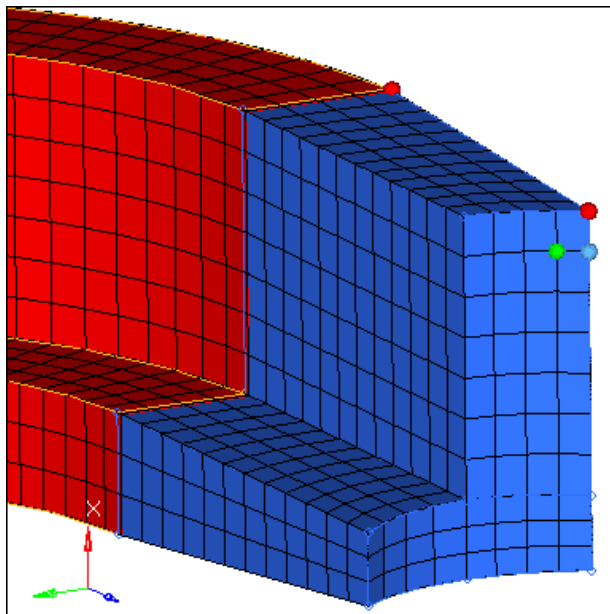
4. Activate the **to: elems** selector.
5. Select the shell elements, between the arm and boss, that you created using the **Automesh** panel in step 9.
6. Activate the **from: alignment: N1** selector.
7. Select three nodes on one of the "from elements" that you selected in step 10.3 as indicated in the following image.
8. Activate the **to: alignment: N1** selector.

9. Select three nodes on the "to element" that corresponds to the "from element" with the three "from nodes" as indicated in the following image.



Example selection for alignment nodes

10. In the **density =** field, enter 12.
11. Click **solids**. HyperMesh creates the mesh.

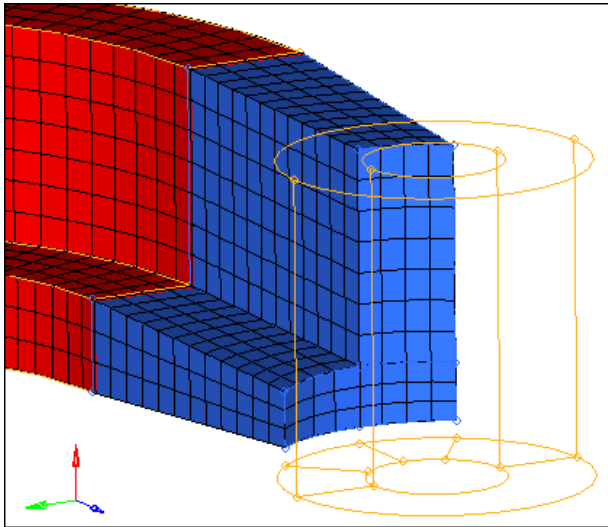


Linear solid mesh

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

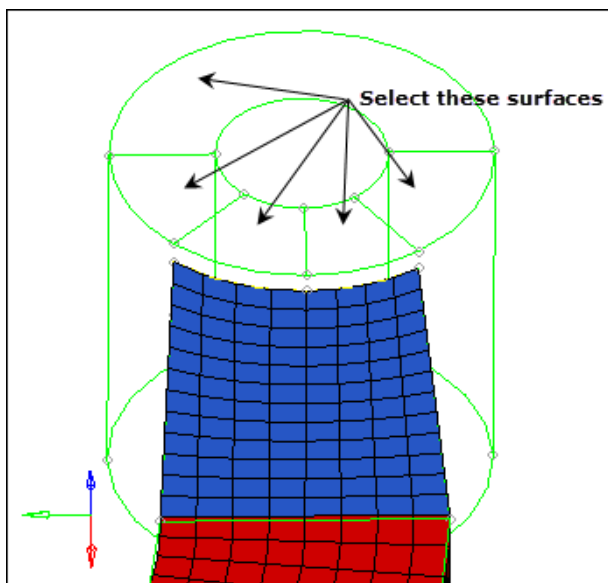
Step 11: Prepare the display for meshing the boss.

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



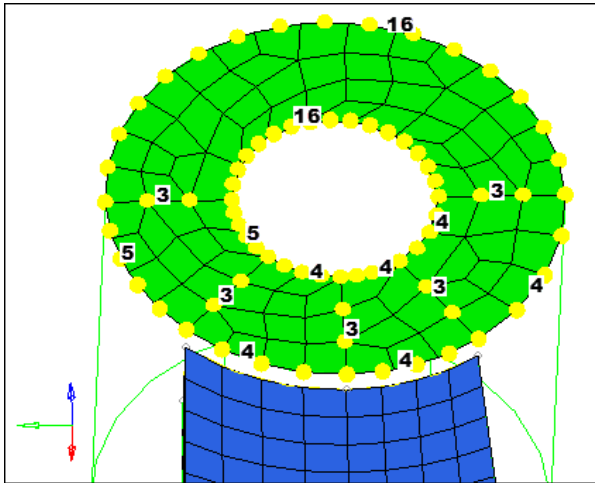
Step 12: Create a shell mesh on the bottom of the boss.

1. In the **Model** browser, **Component** folder, right-click on **boss** and select **Make Current** from the context menu.
2. Open the **Automesh** panel.
3. Select the five surfaces on the bottom face of the boss as indicated in the following image.



4. Click **mesh**. The meshing module opens.

- Adjust the density of each edge to obtain a mesh that matches the following image.

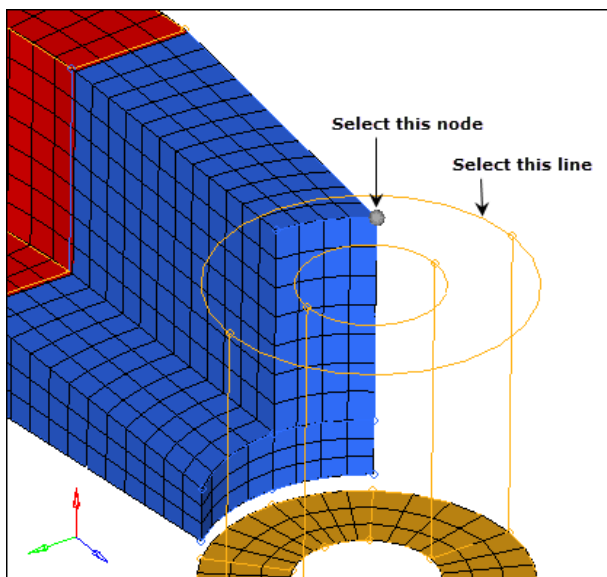


Mesh densities on the bottom of the boss

- Click **mesh**. HyperMesh updates the mesh density.
- Return to the main menu by clicking return **twice**.

Step 13: Project a node to the bottom face of the boss.

- Open the **Project** panel by clicking **Mesh > Project > Nodes** from the menu bar.
- Go to the **to line** subpanel.
- Select the node on the rightmost top vertex as indicated in the following image.
- Click **nodes >> duplicate**.
- Activate the **to line: line list** selector.
- Select the line on the boss' top face as indicated in the following image.

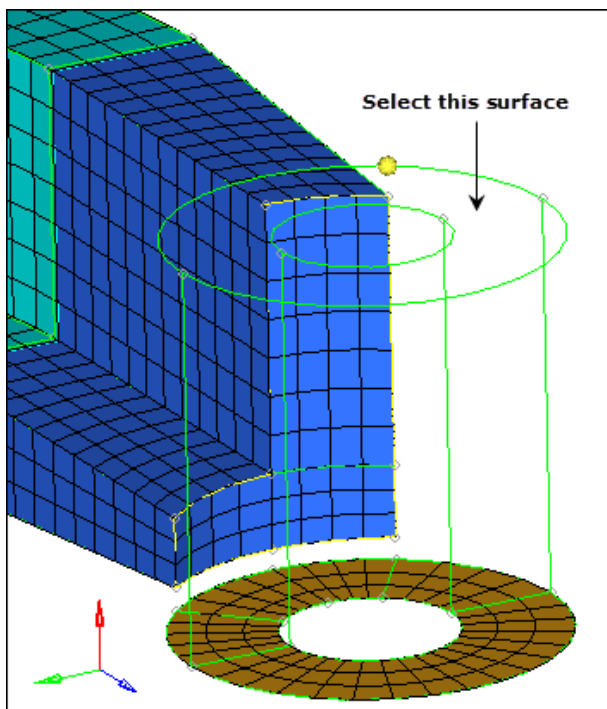


Projecting a node to a line

7. Set the **along vector** to **x- axis**.
8. Click **project**. The node projects to the line.
9. Return to the main menu by clicking **return**.

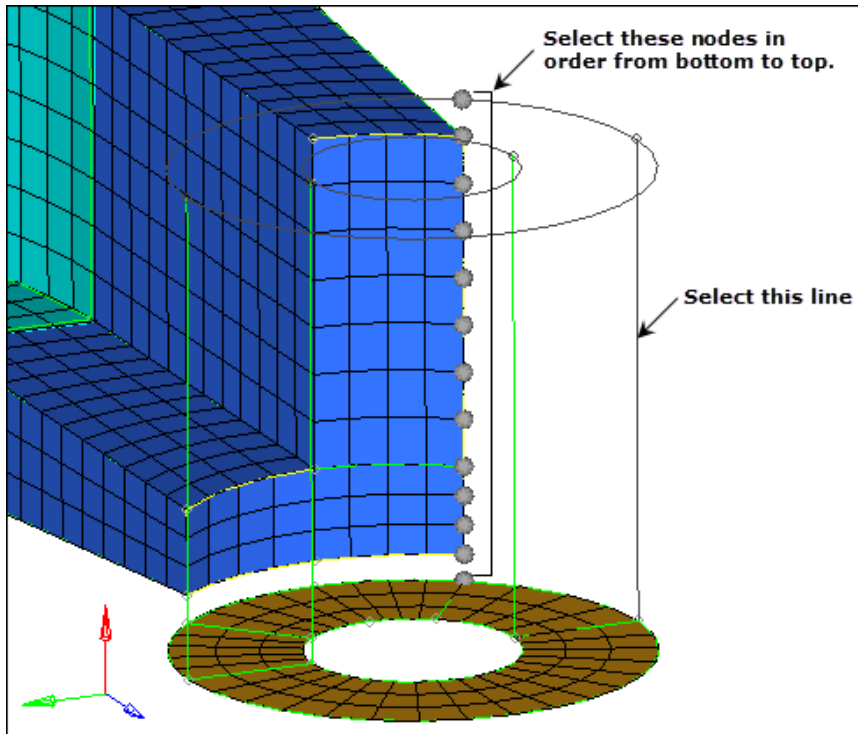
Step 14: Generate hexas for the boss using the Solid Map panel.

1. Open the **Solid Map** panel by clicking **Mesh > Create > Solid Map Mesh** from the menu bar.
2. Go to the **general** subpanel.
3. Click the **source geom** switch and select **none**.
4. Activate the **dest geom: surf** selector.
5. Select the top surface of the boss as indicated in the following image.



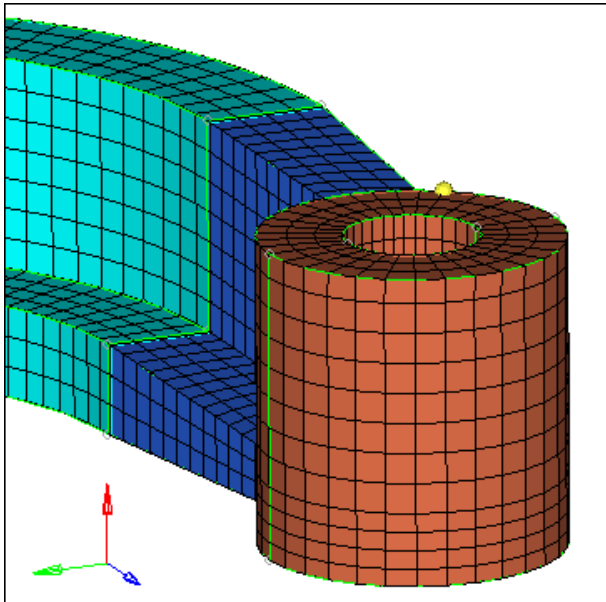
6. Click the **along geom** switch and select **mixed**.
7. Activate the **along geom: mixed lines** selector.
8. Select the line indicated in the following image.
9. Activate the **along geom: mixed node path** selector.
10. Select the 13 nodes as indicated in the following image, to define the exact location of the solid element layers.

Note: A total of 13 nodes should be selected, starting at the boss mesh, and then using all of the nodes along the edge of the **arm_straight** component, ending with the node projected to the top of the boss.



Along nodes for solid map

11. Click **elems to drag: elems >> by collector >> boss.**
12. Click **select.**
13. Click **mesh.** HyperMesh creates the elements and completes the mesh on this part.



Completed mesh of the arm bracket

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

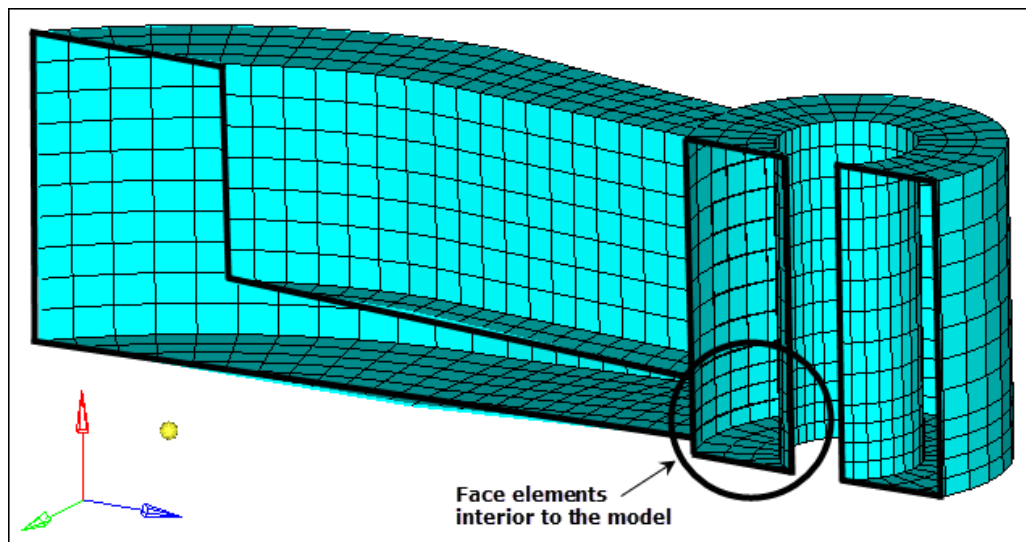
Step 15 (optional): Check the connectivity of the model.

1. Go to the **Faces** panel.
2. Click **comps**. HyperMesh displays a list of components.
3. Select every component from the list, or select **comps >> all**.
4. Click **select**.
5. Click **find faces**.
6. From the **Model** browser, turn off the geometry display of all components.
7. Turn off the element display of all components except **^faces**.
8. Exit the the panel by clicking **return**.
9. From the **Post** page, select **hidden line**.
10. Select the **xz plane** and **trim plane** checkboxes.
11. Click **show plot**. HyperMesh displays the faces with a plane cutting the model in half.

Note: This allows you to review the interior of the model.

12. Click near the cutting plane, hold the left mouse button down, and move your mouse back and forth. The cutting plane moves through the model, allowing you to see if any face elements exist on the interior of the model.


Note: You should see that there are face elements interior to the model, between the boss and arm. You need to perform some corrections on the connectivity.



Hidden line view of faces

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

Step 16 (Optional): Correct the connectivity of the model.

1. In the **Model** browser, display the elements for all of the components except for **^faces**.
2. Display the model's elements as transparent by clicking  on the **Visualization** toolbar.
3. Go to the **Faces** panel.
4. Set the entity selector to **elems**.
5. Click **elems >> displayed**.
6. Click **preview equiv**. HyperMesh highlights coincident nodes on the intersection between the arm and the boss.
7. Specify a slightly larger value in the **tolerance =** field, and then click **preview equiv**. HyperMesh identifies more more coincident nodes on the intersection.
8. Repeat step 16.7 until all 60 coincident nodes have been found.
9. Click **equivalence**. HyperMesh replaces the nodes to the location of the lowest node ID.
10. Switch all the components to the **shaded** visual mode.

Step 17 (Optional): Recheck the connectivity of the model.

Repeat Step 15 to make sure the model is now equivalenced. If you find errors, repeat Step 16.

Step 18 (Optional): Save your work.

Now that the 3D solid mesh has been completed, it is a good time to save your model.

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