Altair

HyperWorks

**HM-3110: Meshing without Surfaces**

In this tutorial, you will learn about:

- The basic concepts of surfaceless meshing and how to mesh a bracket.
- Scale (Uniform scaling)
- 2D mesh by using spline, line drag, and skin
- Ruled mesh

Surfaceless meshing is defined as the creation of mesh using points, lines, and nodes rather than surfaces. Some parts may have missing surfaces and some parts may not have any surfaces at all and are instead defined by line data. Either way, a mesh still must be created. HyperMesh has a number of panels that you can use to create a mesh based on geometry rather than surfaces.

Model Files

This exercise uses the `bracket.hm` file, which can be found in the `hm.zip` file. Copy the file(s) from this directory to your working directory. The model consists of only line data; no surfaces are present.
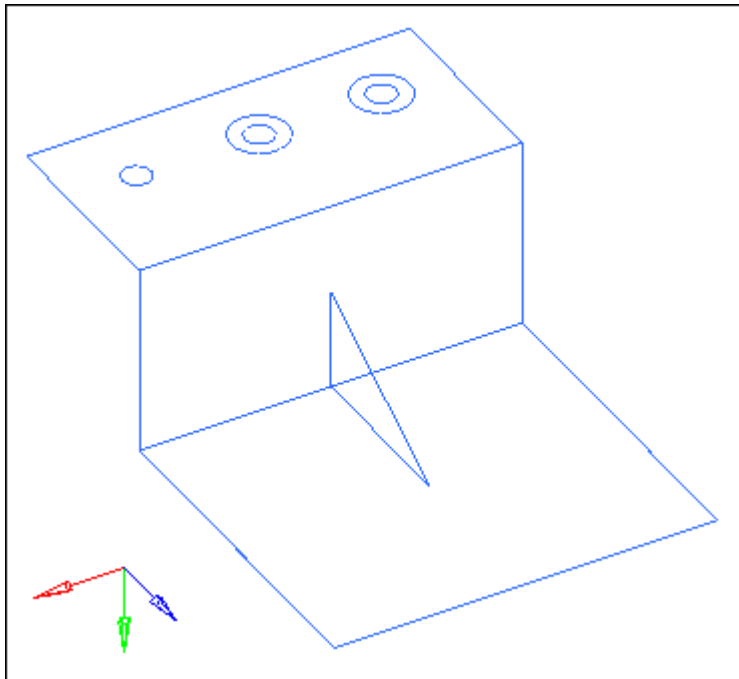
Exercise: Meshing a Bracket

Step 1: Retrieve and view the model file.

1. Start HyperMesh Desktop.

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

3. In the **Open Model** dialog, open the `bracket.hm` model file.

4. Observe the model using the different visual options available in HyperMesh (rotation, zooming, etc.).

### Step 2: Create a concentric circle around a hole on the top face using the *scale* panel.

There are three circles on the upper region of the bracket, which represent three holes in the bracket. Two of the holes have concentric circles around them. This configuration allows you to create a radial mesh pattern around the holes. The following steps will show you how to create a concentric circle around the third hole.

1. To open the **Scale** panel, click *Geometry* > *Scale* > *Lines* from the menu bar, or select *Scale* from the **Tool** page.

2. Click *uniform* and enter `2.0` in the **scale=** field.

3. To open the **Distance** panel, press *F4*.

4. Go to the **three nodes** subpanel.

5. Verify that the **N1** node selector is active.

6. Press and hold your left mouse button, move it over the circle representing the hole as indicated in the following image, and then release it when the cursor changes to a square with a dot in the center ⊡. HyperMesh highlights the circle.

△ Altair
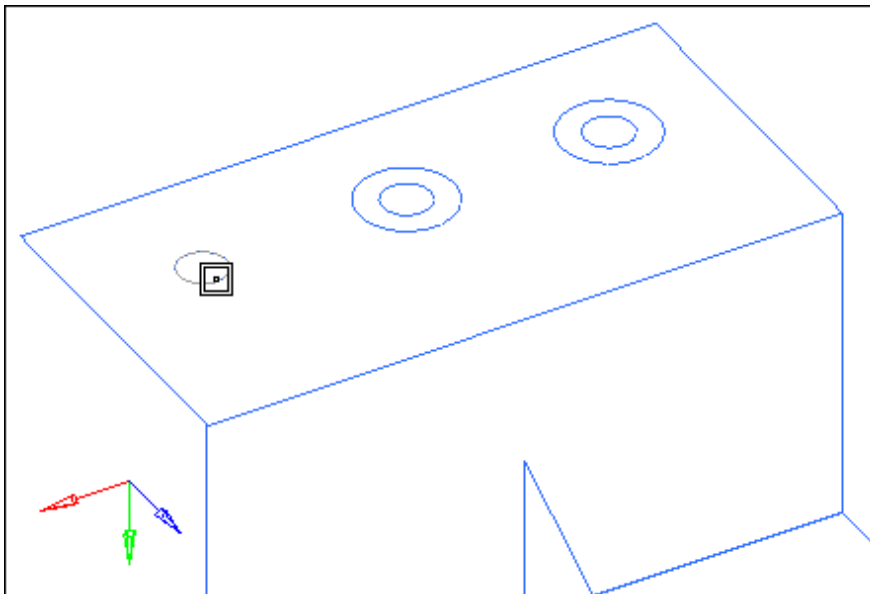
7. Left-click on the highlighted circle. HyperMesh creates a node for **N1**.

8. Left-click twice more at different locations on the circle to create nodes **N2** and **N3**.

9. Click *circle center*. Hypermesh creates a node at the circle's center.

   Note: This node will be selected as the origin node when the circle is duplicated and scaled.
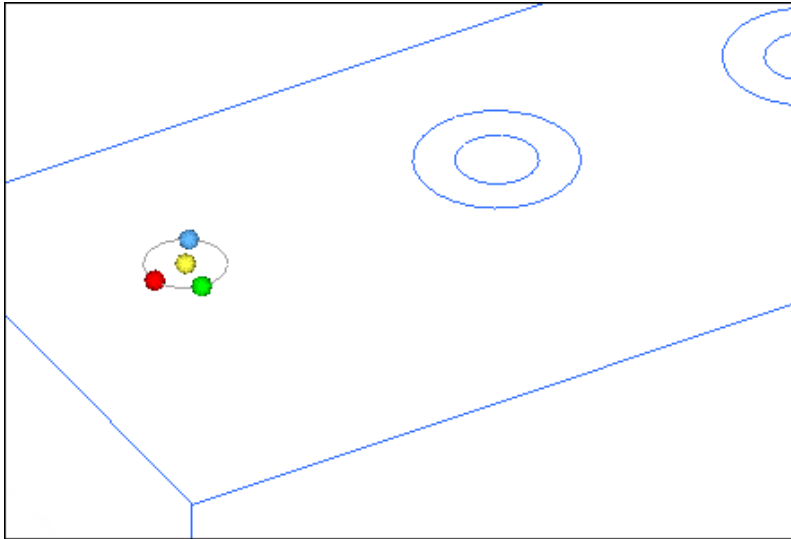
10. To go back to the **Scale** panel, click *return*.

11. Set the entity selector to *lines*.

12. Select the circle with the node in its center.

13. Click *lines >> duplicate >> original comp*.

14. Activate the *origin: node* selector.

15. Select the temporary node you created in the circle's center.

16.   Click *scale +*. HyperMesh creates a new circle, which is concentric with the original.
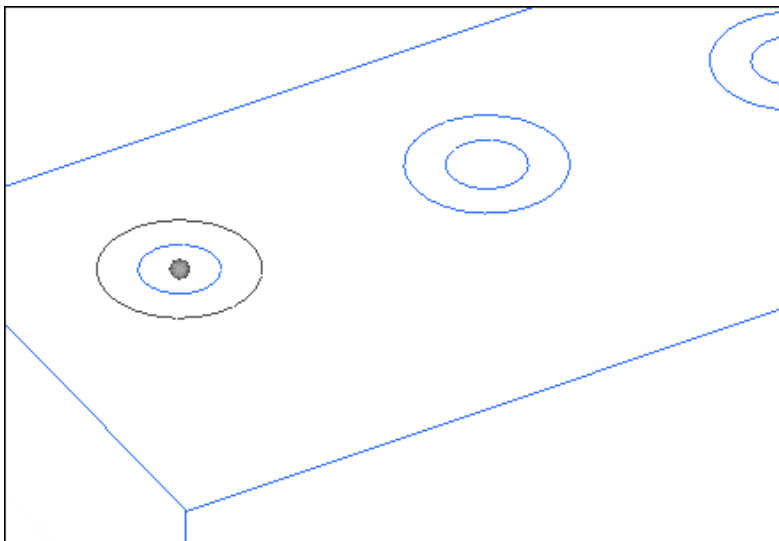
17. To exit the panel, click ***return***.

Step 3: Create a radial mesh between each of the concentric circles using the *spline* panel.

1. To open the **Spline** panel, click ***Mesh*** > ***Create*** > ***2D Elements*** > ***Spline*** from the menu bar, or select ***Spline*** from the **2D** page.

2. Verify that the entity selector is set to ***lines***.

3. Select all six circular lines.
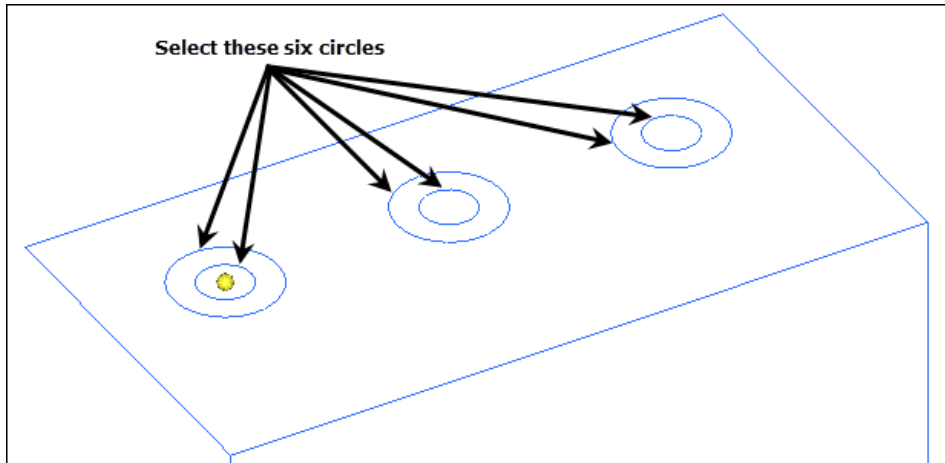
4. Switch from **mesh, keep surf** to ***mesh***, ***dele surf***.

   Note: This option creates surfaces based on the selected entities, uses the surfaces to create a mesh, and then deletes the surfaces.

5. Clear the **keep tangency** check box.

6. Click ***create***. The meshing module opens, and element edge density numbers appear on the selected lines.

   Note: The numbers on a pair of concentric circular lines must be identical in order to achieve a radial mesh.

7. In the **density** subpanel, enter 8 in the **elem density=** field.

8. Click ***set all to***. HyperMesh changes all of the circular line's element edge densities to 8.

△ Altair

9. Click *mesh*. HyperMesh updates the preview mesh based on the change.
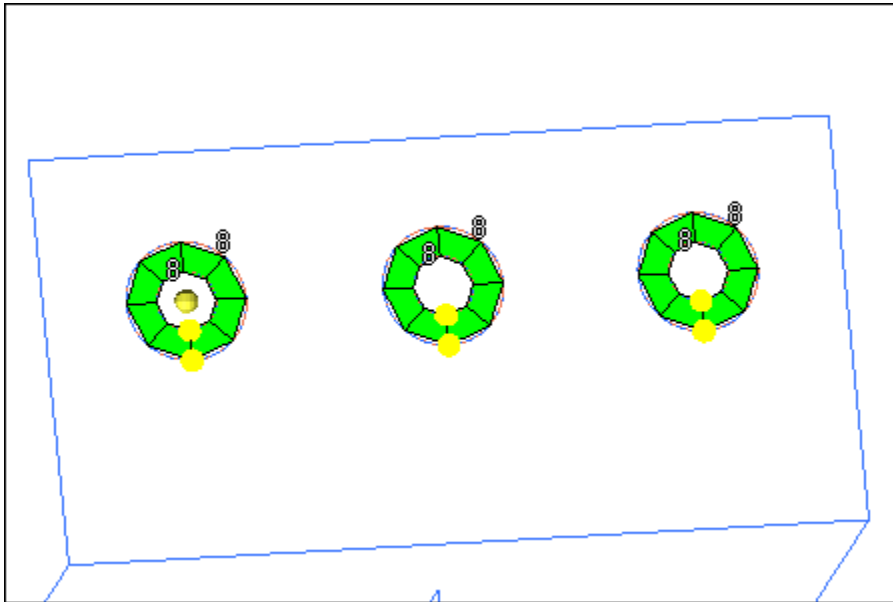
10. To accept the mesh and go back to the **Spline** panel, click *return*.

## Step 4: Mesh the rest of the top face using the *spline* panel.

1. With the **entity type** set to *lines*, select the four lines defining the perimeter of the top face and the three circular lines defining the outside perimeter of the three radial meshes.
2. Click *create*. The meshing module opens.
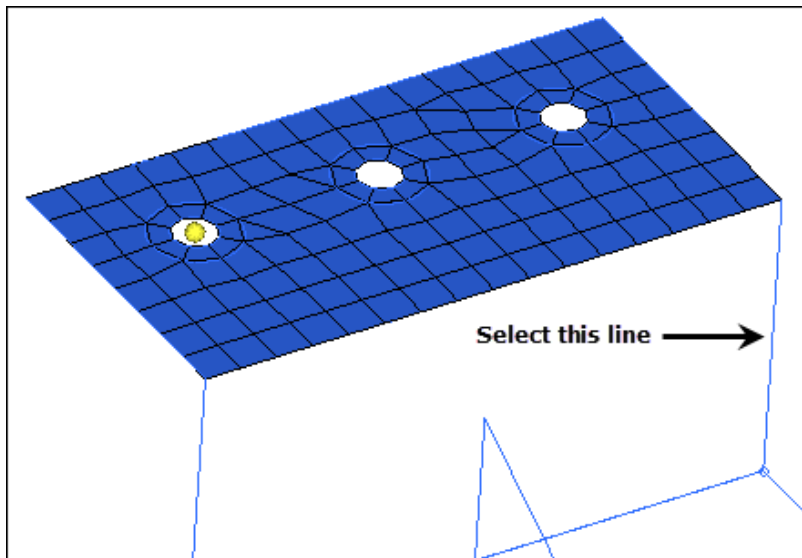3. To accept the mesh and go back to the main menu, click *return* twice.
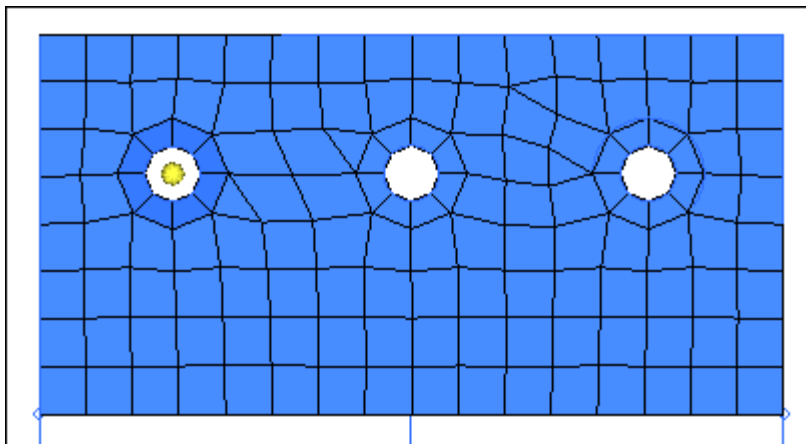
Final mesh on the top face of the bracket

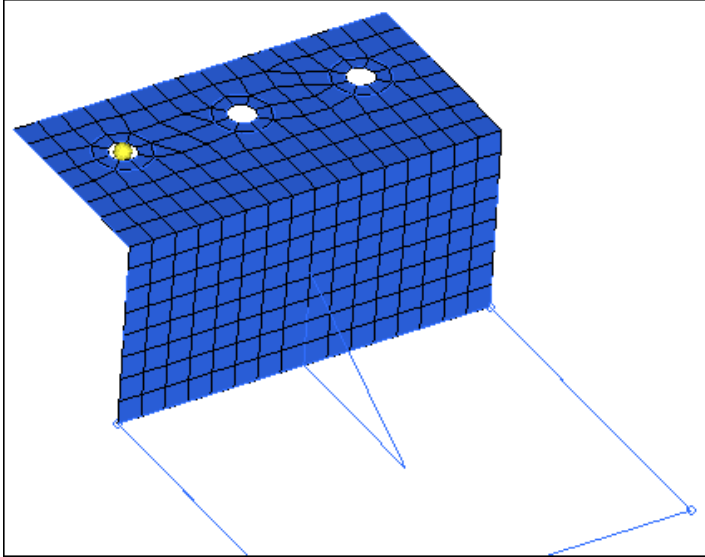### Step 5: Mesh the back face of the bracket using the *line drag* panel.

1. To open the **Line Drag** panel, click *Mesh* > *Create* > *2D Elements* > *Line Drag* from the menu bar, or click *Line Drag* from the **2D** page.

2. Go to the **drag geoms** subpanel.

3. Set the **drag** selector to *line list*.

4. Select the line that is on the perimeter of the existing mesh and adjacent to the bracket's back face as indicated in the following image.

Select this line
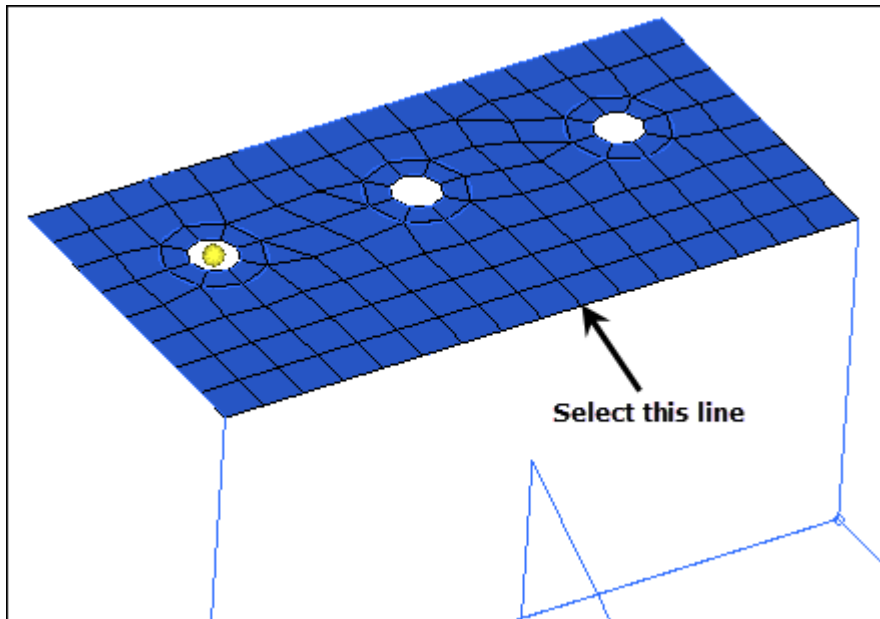
5. Activate the ***along: line list*** selector.
6. Select one of the two lines defining the back face that are perpendicular to the selected line to drag as indicated in the following image.


Select this line ⟶

△ Altair

7. Leave the toggle set to **use default vector**.

8. Leave the creation method set to **mesh, w/o surf**.

9. Click ***drag***. The meshing module opens.

10. To accept the mesh and return to the main menu, click ***return*** twice.

Mesh of top and back faces

Step 6: Mesh the bottom face of the bracket using the *ruled* panel.

1. To open the **Ruled** panel, click ***Mesh*** > ***Create*** > ***2D Elements*** > ***Ruled*** from the menu bar, or select ***Ruled*** from the **2D** page.

2. Verify that the top entity selector is set to ***node list***.

3. Click ***node list*** >> ***by path***. The entity selector changes to ***node path***.

4. Select the end nodes located on the back face edge that borders the bottom face, as indicated in the following image. HyperMesh selects all of the nodes between the two selected nodes.


Select these nodes
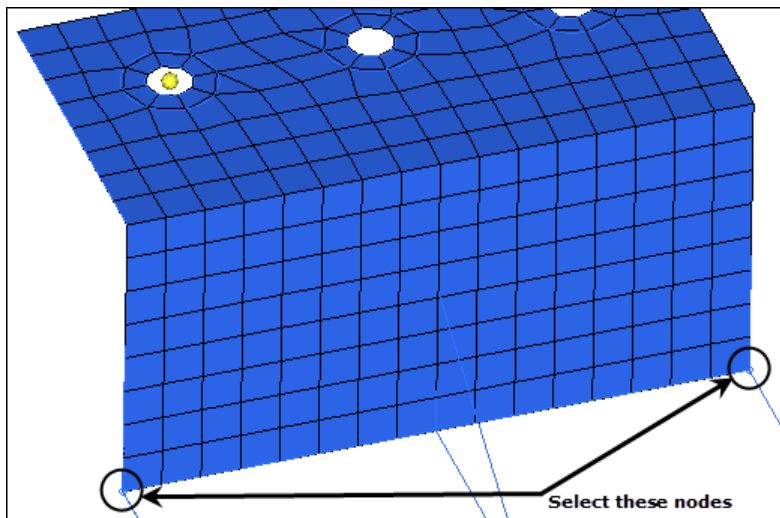
5. Click **node path** >> **show node order**. HyperMesh highlights and numbers the nodes to show the order in which they have been selected.

6. Set the bottom entity selector to **line list**.

7. Select the line defining the opposite edge of the bottom face as indicated in the following image.
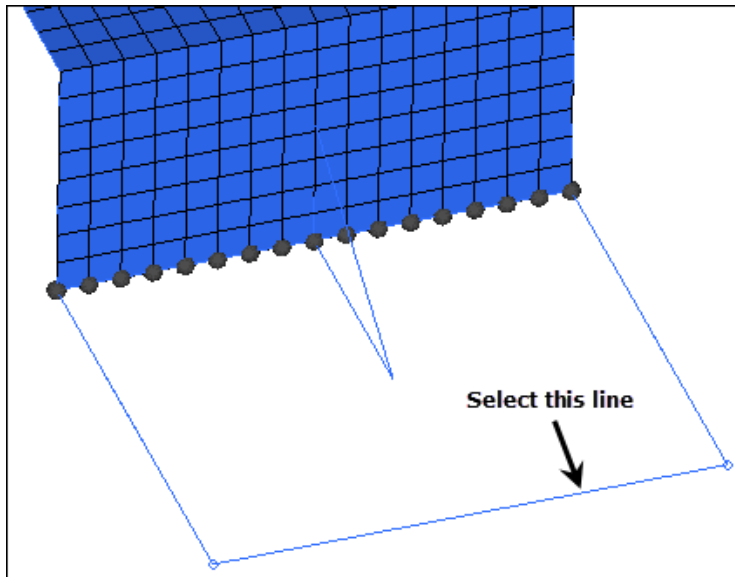
8. Switch the creation method from **mesh, keep surf** to **mesh, w/o surf**.

9. Select the **auto reverse** check box.

   Note: When elements are generated, the edges used to create them can be ordered in different directions. The order of the edges is determined by the order in which the nodes are selected or the direction of the selected line(s). If the direction is different for each selection, then a mesh that crosses itself, similar to a bow tie, will be created. To prevent this, the **auto reverse** option ensures elements are generated with a similar order on each side of the mesh.
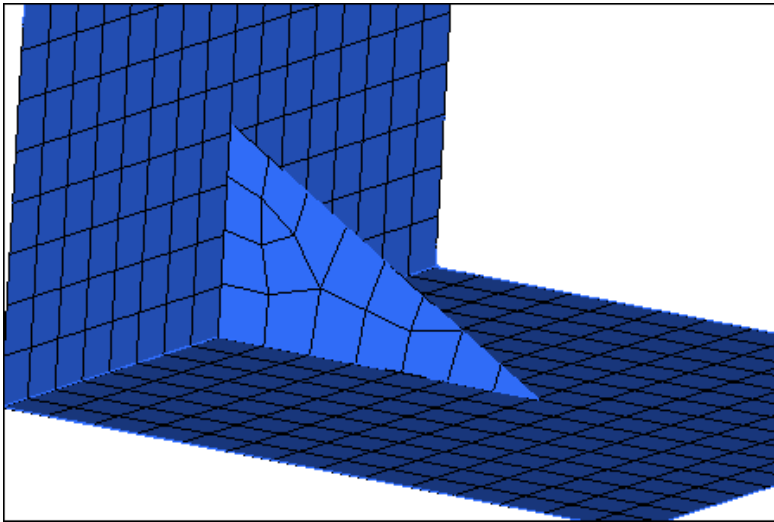
10. Click **create**. The meshing module opens.

11. To accept the mesh and return to the main menu, click **return** twice.
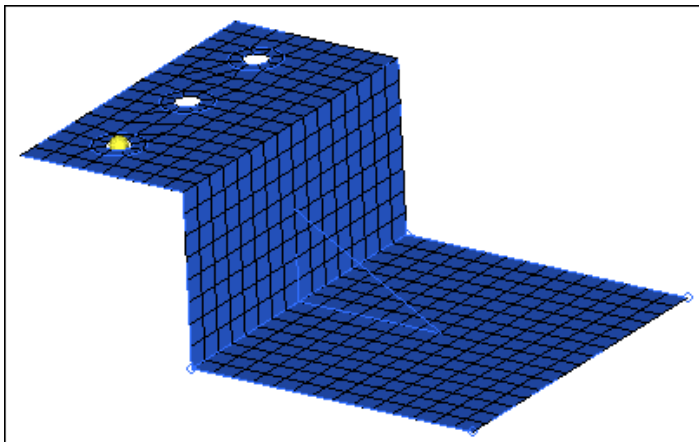
Mesh of top, back, and bottom faces of bracket

Step 7: Mesh the rib using the *skin* panel.

1. To open the **Skin** panel, click ***Mesh*** > ***Create*** > ***2D Elements*** > ***Skin*** from the menu bar, or select ***Skin*** from the **2D** page.

2. With the ***line list*** selector active, select any two of the three lines defining the rib.

3. Switch the creation method from **mesh, keep surf** to ***mesh, dele surf***.

4. Leave the toggle set to **auto reverse**.

5. Click ***create***. The meshing module opens.

6. To accept the mesh and return to the main menu, click ***return*** twice.

Mesh of rib