



Altair

---

**HyperWorks**

## HM-2030: Refining Topology to Achieve a Quality Mesh

In this tutorial, you will learn how to:

- Mesh the part to determine poor element quality
- Suppress small edges
- Split surfaces
- Remove interior fixed points
- Replace closely placed fixed points
- Create final mesh

Topological details of the geometry may affect the quality of the mesh created from the surfaces. Some of these details may not reflect any major feature of the part's shape, and can be removed without concern. When modifying the topology affects the shape of the surfaces, a compromise must be made between the part shape and the element quality necessary for the analysis. Other times, adding topological features that do not change the shape of the part may actually help create a better quality mesh.

### Model Files

---

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

### Exercise: Refining Topology to Achieve a Quality Mesh

---

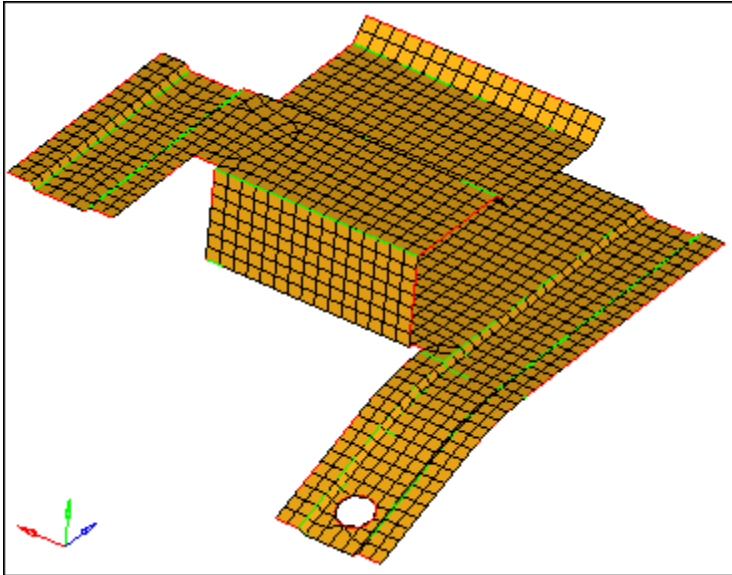
#### Step 1: Open the model file, `clip_refine.hm`.

1. From the menu bar, click **File > Open > Model**.
2. In the **Open Model** dialog, open the `clip_refine.hm` model file.
3. Take a few moments to observe the model using the different visual options available in HyperMesh (rotation, zooming, etc.).

#### Step 2: Create a preliminary mesh.

1. To open the **Automesh** panel, click **Mesh > Create > 2D AutoMesh** from the menu bar, or press **F12**.
2. Set the entity selector to **surfs**.
3. Go to the **size and bias** subpanel.
4. In the **element size =** field, type `2.5`.
5. From the **mesh type** list, select **mixed**.
6. Switch the meshing mode from **interactive** to **automatic**.

7. Click **surfs >> displayed**.
8. Click **mesh**. HyperMesh meshes the surfaces.




Initial mesh on the defeatured clip model

### Step 3: Review the mesh quality.

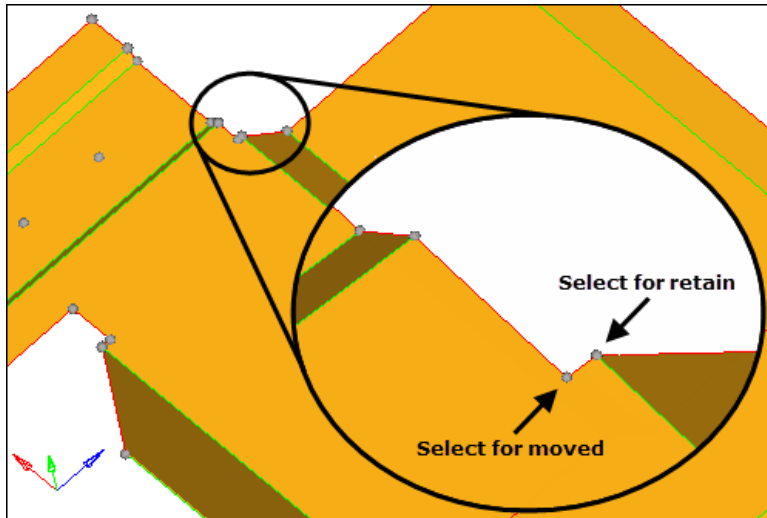
1. Take a minute to rotate, zoom, and pan the model to review the mesh that was created. Pay attention to the locations where the mesh was not created in rows and columns of quads.
2. To open the **Check Elements** panel, click **Mesh > Check > Elements > Check Elements** from the menu bar, or press **F10**.
3. Go to the **2-d** subpanel.
4. In the **length <** field, enter 1.
5. To evaluate the minimum length, click **length**. HyperMesh highlights the elements that failed the check.
6. To exit the panel, click **return**.
7. In the **Model** browser, **Component** folder, click  next to **Middle Surface** to turn off the display of it's elements.

### Step 4: Remove short edges by combining fixed points.

1. To open the **Replace** panel, click **Geometry > Edit > Fixed Points > Replace** from the menu bar.
2. Verify that the **moved points** selector is active.
3. Select the **lower fixed** point as indicated in the following image.

Tip: If there are no visible fixed points on your model, verify that  is selected on the **Display** toolbar.

4. Activate the **retained point** selector.
5. Select the **upper fixed** point as indicated in the following image.
6. Click **replace**.



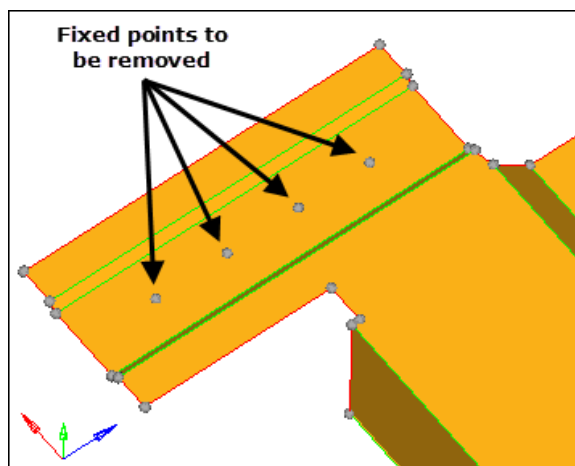
Selecting fixed points to be combined

### Step 5: Remove the fixed points interior to all surfaces.

You should still be in the **Points** panel.

1. Go to the **suppress** subpanel
2. Under **at cursor**, verify that the **point** selector is active.
3. Select the four fixed points as illustrated in the following image. HyperMesh deletes each fixed point when you select it.

These fixed points are left over from a defeaturing operation where small holes (pinholes) were removed. They could remain without greatly sacrificing the element quality, given the element size used for the mesh, but the mesh would be better without them.

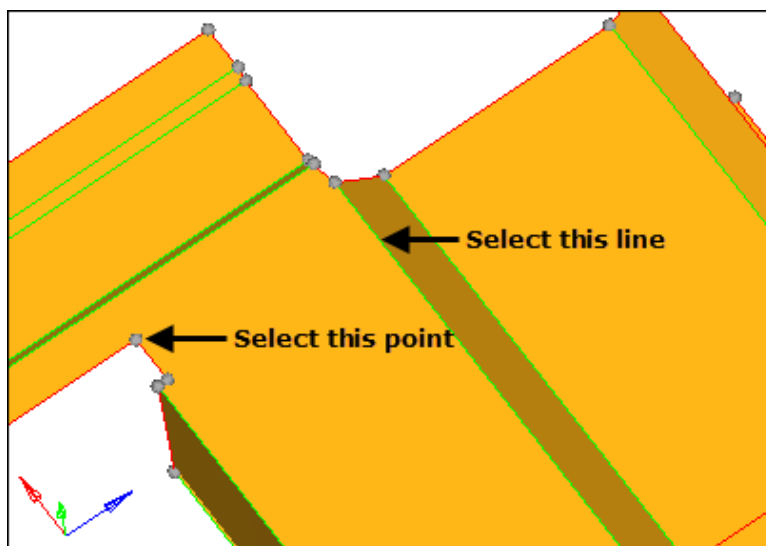


Fixed points to be removed

4. To exit the panel, click **return**.

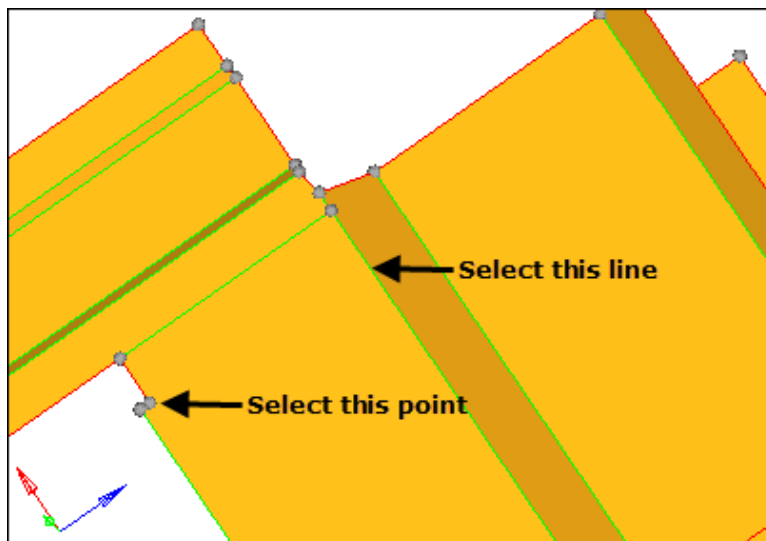
### Step 6: Add edges to the surfaces to control the mesh pattern.

1. To open the **Trim with Nodes** panel, click **Geometry > Edit > Surfaces > Trim with Nodes** from the menu bar.
2. Under **node normal to edge**, click **node**.
3. Zoom into the area illustrated in the following image and select the indicated fixed point.
4. With the **lines** selector now active, select the line shown in the image below. HyperMesh creates an edge from the location of the fixed point perpendicular to the line.



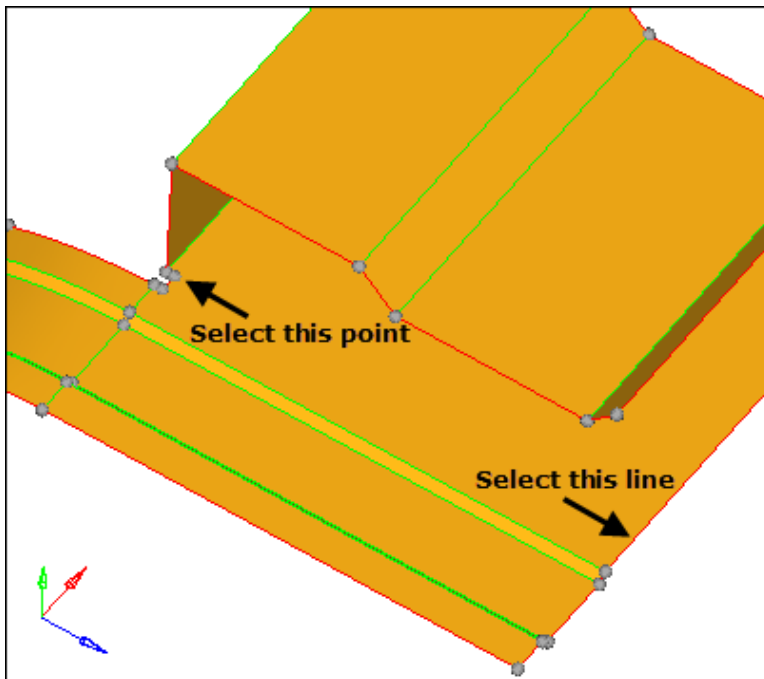
Select fixed point and line to split the surface.

5. Repeat steps 6.2 through 6.4 for the point and line illustrated in the following image.



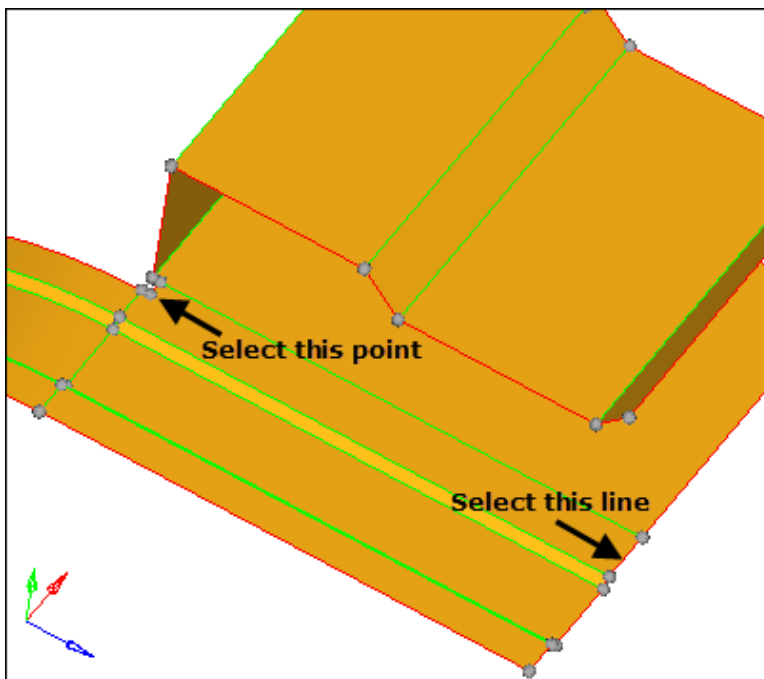
Select fixed point and line to split the surface.

- Repeat steps 6.2 through 6.4 for the point and line illustrated in the following image.



Select fixed point and line to split the surface.

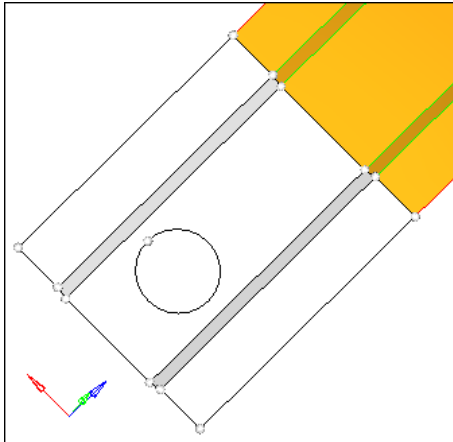
- Repeat steps 6.2 through 6.4 for the point and line illustrated in the following image.





Select fixed point and line to split the surface

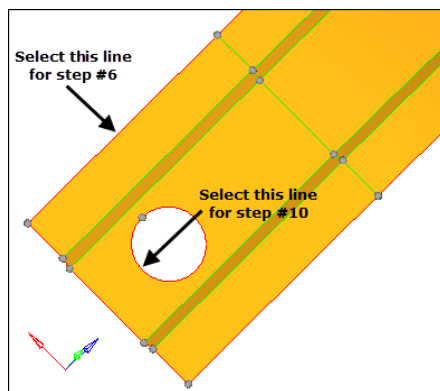
## Step 7: Add edges to the surfaces to control the mesh pattern.

1. Go to the **trim with surfs/planes** subpanel.
2. In the **with plane** column, verify that the **surfs** selector is active.
3. Select the five surfaces indicated in the following image.



Surfaces to be selected for splitting

4. Verify that the direction selector is set to **N1, N2, N3**.
5. Click **N1** to make the selector active.
6. Press and hold your left mouse button, move it over the edge as indicated in the following figure, and then release it when the cursor changes to a square with a dot in the center .
7. Click two points anywhere along the edge. Do not click a third. Hypermesh places nodes on the line for **N1** and **N2**.
8. To open the **Distance** panel, press **F4**.
9. Go to the **three nodes** subpanel.
10. Press and hold your left mouse button, move it over the edge of 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 .



Select fixed point and line to split the surface

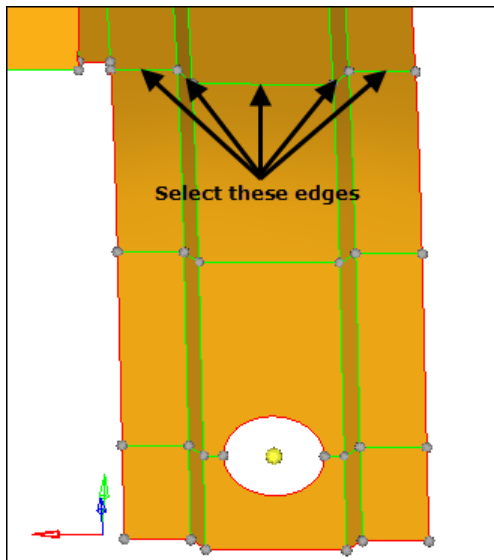
11. Click three points anywhere along the edge. HyperMesh places temporary nodes on the line representing **N1**, **N2**, and **N3**.

Note: The technique you used to create nodes to select where none existed before can be used in any place where nodes need to be selected but do not exist in the model. You can create nodes in this manner on lines, surfaces and elements. For more details, see the HyperMesh online help. Pick the index and type, Picking Nodes on Geometry or Elements.

12. Click **circle center**. HyperMesh creates a node at the center of the hole.
13. To exit the **Surface Edit** panel, click **return**.
14. Activate the **B** selector.
15. Select the node that was just created at the center of the hole.
16. Click **trim**. HyperMesh trims the surfaces through the center of the hole.
17. To exit the panel, click **return**.

### Step 8: Suppress shared edges causing a small edge.

1. To go to the **(Un)suppress** panel, click **Geometry > Edit > Surface Edges > (Un)Suppress** from the menu bar.
2. Select the five lines illustrated in the following image.




Surface edges to suppress by toggling

3. Click **suppress**. HyperMesh suppresses each line, which is indicated by a blue, dashed line.

### Step 9: Remesh the part.

In this step, you will remesh the surfaces of the part, using the **automatic** mode, a **size** of 2.5, and the **mixed mesh** type.



1. In the **Model** browser, **Component** folder, click  next to **Middle Surface** to display it's elements.
2. To open the **Automesh** panel, click **Mesh > Create > 2D AutoMesh** from the menu bar, or press **F12**.
3. Verify that **elem size =** is set to 2.5 and the **mesh type** is set to **mixed**.
4. To select all of the displayed surfaces, click **surfs >> displayed**.
5. Click **mesh**.

### Step 10: Review the mesh quality.

1. Take a minute to rotate, zoom, and pan the model to review the mesh that was created. The mesh consists completely of rows and columns of quads.
2. To open the **Check Elements** panel, click **Mesh > Check > Elements > Check Elements** from the menu bar, or press **F10**.
3. Go to the **2-d** subpanel.
4. In the **length <** field, enter 1.
5. To evaluate the minimum length, click **length**. HyperMesh highlights only two elements that failed the check. Both of these elements failed the check because of the shape of the part. These elements are not too small compared to the global element size, therefore you can leave them as is.
6. To open the **Automesh** panel, click **Mesh > Create > 2D AutoMesh** from the menu bar, or press **F12**.
7. Go to the **QI optimize** subpanel.
8. Verify that **elem size =** is set to 2.5 and the **mesh type** is set to **mixed**.
9. Click **edit criteria**.
10. In the **Criteria File Editor dialog**, enter 2.500 in the **Target element size** field.
11. Click **Apply**.
12. Click **OK**.
13. To select all of the displayed surfaces, select **surfs >> displayed**.
14. Click **mesh**. HyperMesh replaces the old mesh with a new mesh.
15. If there is a message saying, "There is a conflict between the user requested element size and quality criteria ideal element size," click the **Recompute quality criteria** button using a size of 2.5.
16. To access the **Quality Index** panel, click **Mesh > Check > Elements > Quality Index** from the menu bar.
17. Go to **page 1**.
18. Verify that the **comp. QI** is 0.01. This low value indicates that the mesh is good quality. The higher the number, the lower the mesh quality.

**Step 11 (Optional): Save your work.**

The part is now meshed and ready to be set up for an analysis. Save the model, if desired.

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