Altair

HyperWorks

HM-2020: Simplifying Geometry

In this tutorial, you will learn how to:

- Mesh the clip, review the mesh quality, and determine the features to be simplified
- Remove surface fillets
- Remove edge fillets
- Remove pinholes

This exercise involves changing the shape of a part in order to simplify the geometry. Certain details of the shape, such as small holes or blends, may simply not be necessary for the analysis being performed. When these details are removed, the analysis can run more efficiently. Additionally, mesh quality is often improved as well. Changing the geometry to match the desired shape can also allow a mesh to be created more quickly.

Model Files

This exercise uses the `clip_defeature.hm` file, which can be found in the `hm.zip` file. Copy the file(s) from this directory to your working directory. This model file contains geometry that has been midsurfaced. Surfaces have been created on the mid-plane of the part.

Exercise

In this exercise, you will mesh the model using an element size of 2.5. You can assume a simple structural analysis will be run on the part, and thus does not require much detail. There are unnecessary features in this model that can also be removed.

△ Altair
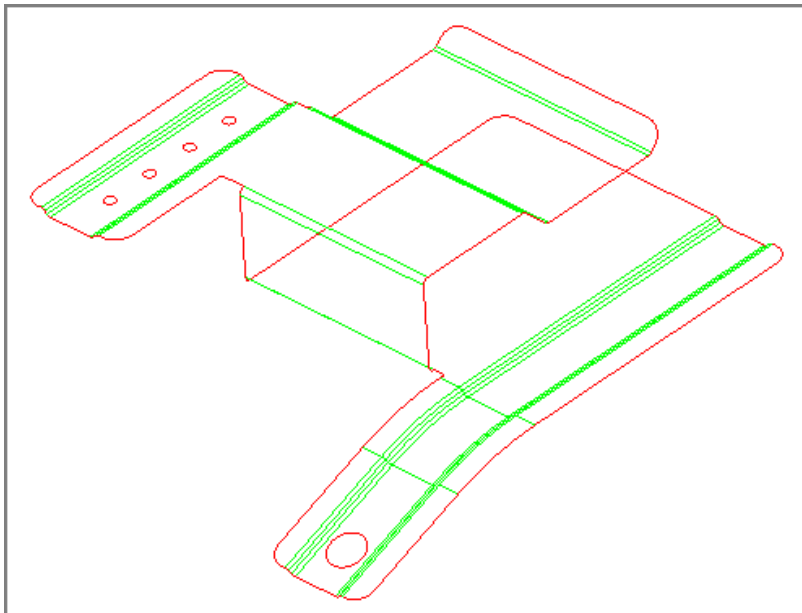
Step 1: Open and view the model file, `clip_defeature.hm.`

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

2. In the **Open Model** dialog, open the `clip_defeature.hm` model file.

Step 2: To easily work with the midsurface, turn off the display of the lvl10 component.

If the **lvl10** component is displayed, it needs to be turned off so that you can easily work on the midsurface geometry.

1. In the **Model** browser, **Component** folder, click ▢ next to **lvl10** to turn off its geometry display.

Step 3: Mesh the clip to view mesh quality before defeaturing.

1. To shade the model's geometry and surface edges, click 🔧 on the **Visualization** toolbar.

2. To open the **AutoMesh** panel, click *Mesh* > *Create* > *2D AutoMesh* from the menu bar, or press *F12*.

3. Set the entity selector to **surfs**.

4. In the **element size =** field, type `2.5`.

5. From the **mesh type** list, select *mixed*.

6. Toggle the meshing mode from **interactive** to **automatic**.

7. Verify that the **elems to surf comp** toggle is set.

8. To select all of the displayed surfaces, click *surfs >> displayed*.

9. Click *mesh*. HyperMesh generates the mesh preview.
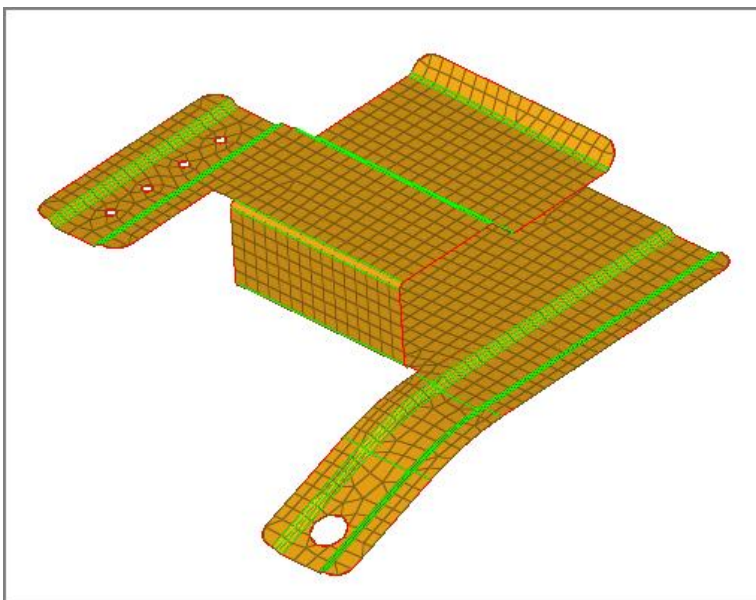

Initial mesh on the clip model

△ Altair

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

Step 4 (Optional): Review the quality of the mesh.

In this step, you will review the quality of the mesh that was created. Pay attention to the areas that have an irregular, poor quality mesh. An irregular, poor quality mesh generally contains rows and columns of quads that are not neat. You will use the **Check Elements** panel to evaluate the minimum length check of the elements.

1. To open the **Check Elements** panel, click ***Mesh* > *Check* > *Elements* > *Check Elements*** from the menu bar, or press ***F10***.

2. Go to the **2-d** subpanel.

3. In the **length <** field, enter `1`.

4. To evaluate the minimum length, click ***length***. Many of the elements that failed the length test are located around the fillets of this model.

   Tip: For better visualization of element quality, you may need to display the geometry in wire frame mode by clicking ⌣ on the **Visualization** toolbar.
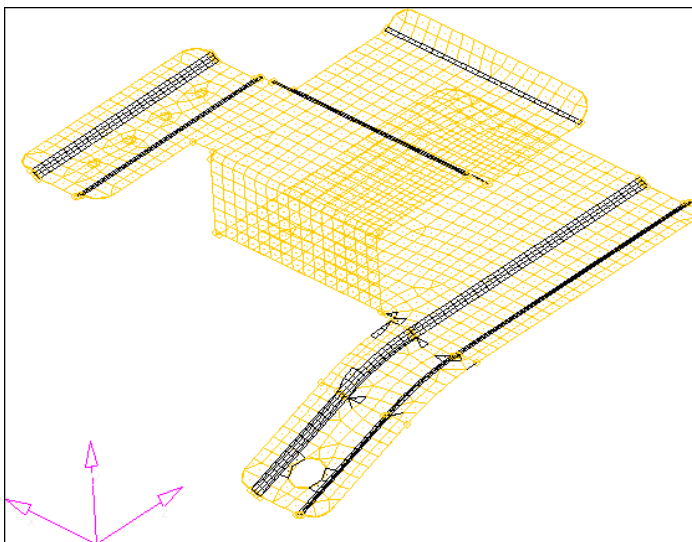

Elements failing the length check

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

Step 5: Remove the four small pinholes.

Pinholes are closed free edge loops within a surface. Pinholes do not need to be circular.

1. To open the **Defeature** panel, click ***Geometry* > *Defeature* > *Pinholes*** from the menu bar.

2. In the **diameter <** field, enter `3.0`.

3. Select ***surfs >> all***.

4. Click *find*. HyperMesh identifies all of the pinholes that have a diameter of 3 or less in the model, and places a white xP symbol in their centers.
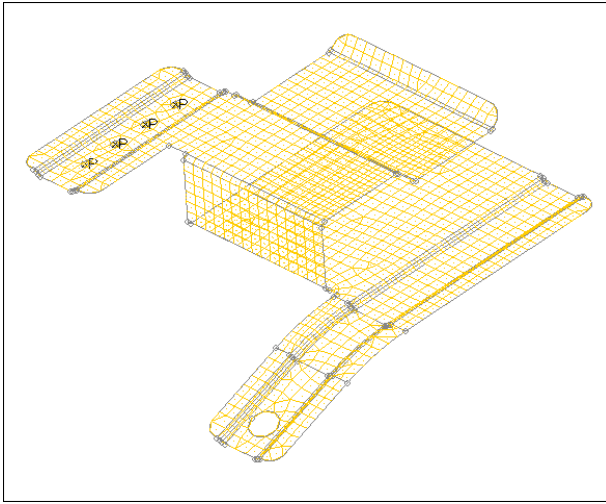

Pinholes identified using a 3 mm diameter

5. Click *delete*. HyperMesh removes the selected pinholes in the model and replaces them with fixed points located at the center of the original pinholes. The mesh also updates according to the changes in the geometry.

Step 6: Remove all surface fillets in the clip.

1. Go to the **surf fillets** subpanel.

2. If the model's geometry and surface edges are not shaded, click ⬚ on the **Visualization** toolbar.

3. Under **find fillets in selected**, click *surfs* >> *displayed*.

4. In the **min radius** field, type 2.0.

5. Click *find*. HyperMesh identifies all of the surface fillets with a radius of 2 or greater in the model.
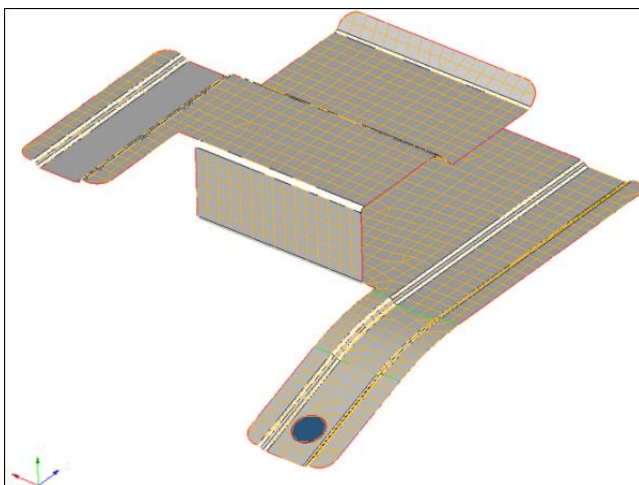
Surface fillets identified for removal

△ **Altair**

6. Click *remove*.

Step 7: Automatically identify and remove rounded corners of surfaces.

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

1. Go to the **edge fillets** subpanel.
2. Click *surfs >> displayed*.
3. In the **min radius** field, type `1.0`.
4. To find all of the fillets in the model, set the bottom switch to *all*.
5. Click *find*. HyperMesh identifies all of the edge fillets that meet the filter criteria, and displays a F symbol where they are located in the model along with radial lines.
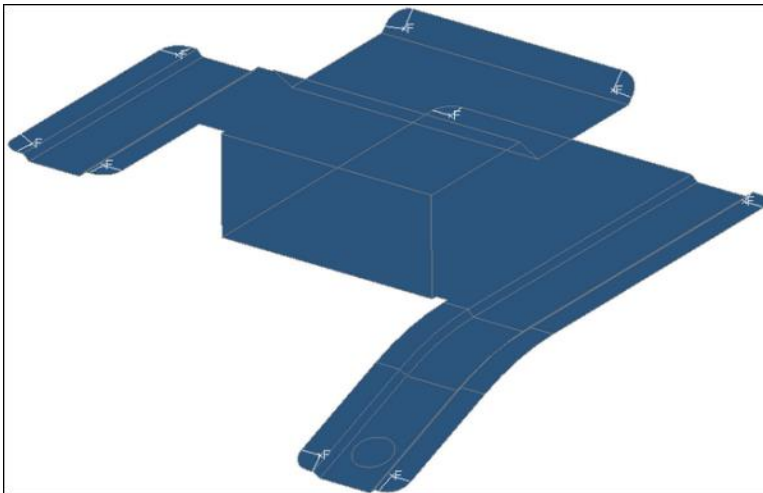
Edge fillets identified for removal

6. With the **fillets** entity selector active, right-click on one of the **F fillet** markers on the screen to deselect the fillet.
7. Click *remove*. HyperMesh deletes all of the selected edge fillets except for the one which was deselected in step 6.

Step 8 (Optional): Save your work.

Now that the model has been simplified, it is a good time to save the model.

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

Summary

The model is now represented in a much simpler form that suits the analysis that will be performed. Holes, surface fillets, and edge fillets were removed that were considered too small to be captured by the desired element size of 2.5.