Altair

HyperWorks

HM-2000: Importing and Repairing CAD

In this tutorial, you will:

- Delete untrimmed surfaces
- Close missing surfaces
- Set the cleanup tolerance
- Equivalence free edges
- Delete duplicate surfaces

The benefits of importing and repairing CAD are:

- Correcting any errors in the geometry from import
- Creating the simplified part needed for the analysis
- Meshing a part all at once
- Ensuring proper connectivity of mesh
- Obtaining a desirable mesh pattern and quality

Model Files

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

Exercise: Importing and Repairing CAD Geometry Data

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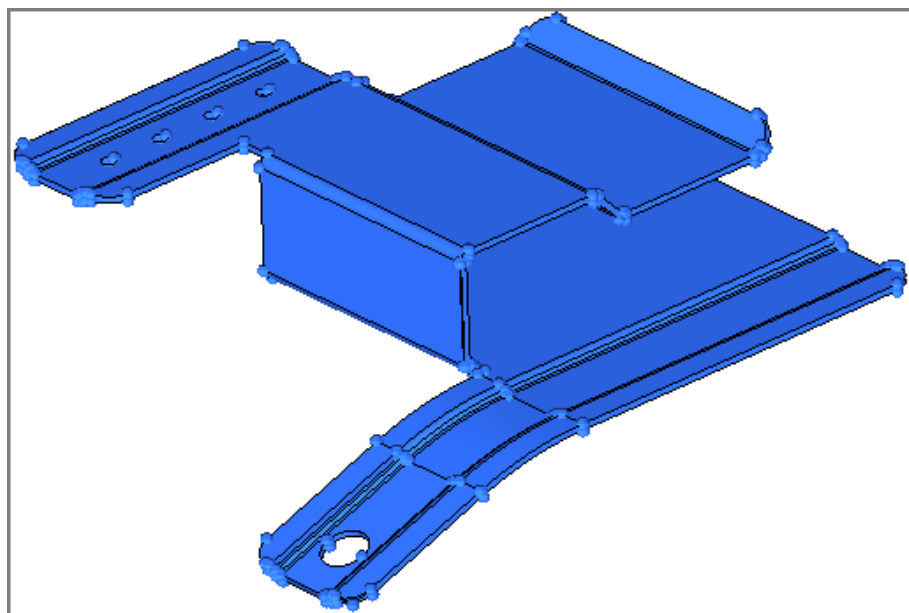
Step 1: Open and view the model file, `clip_repair.hm`.

1. Start HyperMesh Desktop.

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

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

Step 2: View the model in topology display toolbar and shaded mode to evaluate its integrity.

1. Observe where the model has incorrect connectivity and missing or duplicate surfaces.

2. To open the **Auto Geometry Cleanup** panel, click *Geometry* > *Autocleanup* from the menu bar. The surface edges are now colored according to their topology status.

   Note: This occurs because **Geometry Color** is set to ⬛ Auto ▼.

3. To display the model's geometry in wire frame mode, click ⌒ on the **Visualization** toolbar.

   Note: The **Visualization** toolbar contains icons that control the display of the surfaces and surface edges. Surfaces can be shaded with or without edges or wireframe. Right-click the icons to access the drop-down menu for additional options. Place your mouse over the cursor to view a description of the button's functionality.

4. To open the **Visualization** browser and access the **Topology** options, click 🖥.

   Note: The **Topology** options control the display of the surfaces and surface edges. Surfaces can be shaded or wireframe. The check boxes within the **Visualization** browser turn the display of the different edge types and fixed points (surface vertices) on or off.

5. Select only the **Free** check box. The graphics area displays only the free edges.

6. Observe the free (red) edges and make note of where they are. Free edges show where there is incorrect connectivity or gaps.

7. Observe the locations where there are closed loops of free edges. These are locations that probably have missing surfaces.
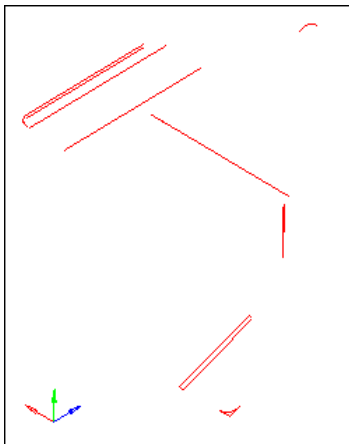
Free edges indicating surface discontinuities of the clip geometry

8. Select only the ***Non-manifold*** check box.

9. Observe the non-manifold edges and make note of where they are. Non-manifold edges show where there are more than two surfaces sharing an edge, which might indicate incorrect connectivity. For this part, there are yellow edges completely surrounding two areas. This indicates that there are probably duplicate surfaces in these locations.

10. Select all of the check boxes.

11. Close the **Visualization** browser.

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

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

14. To locate any errors in the geometry, rotate, zoom, and pan.

15. Note the areas to be worked on:

    •   A surface that overhangs a round corner
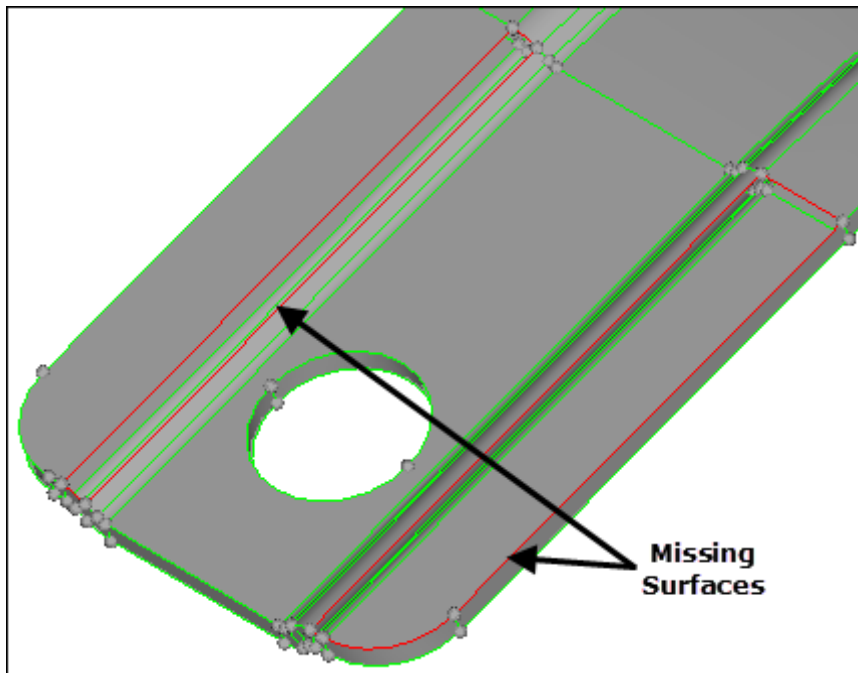
    •   A missing surface
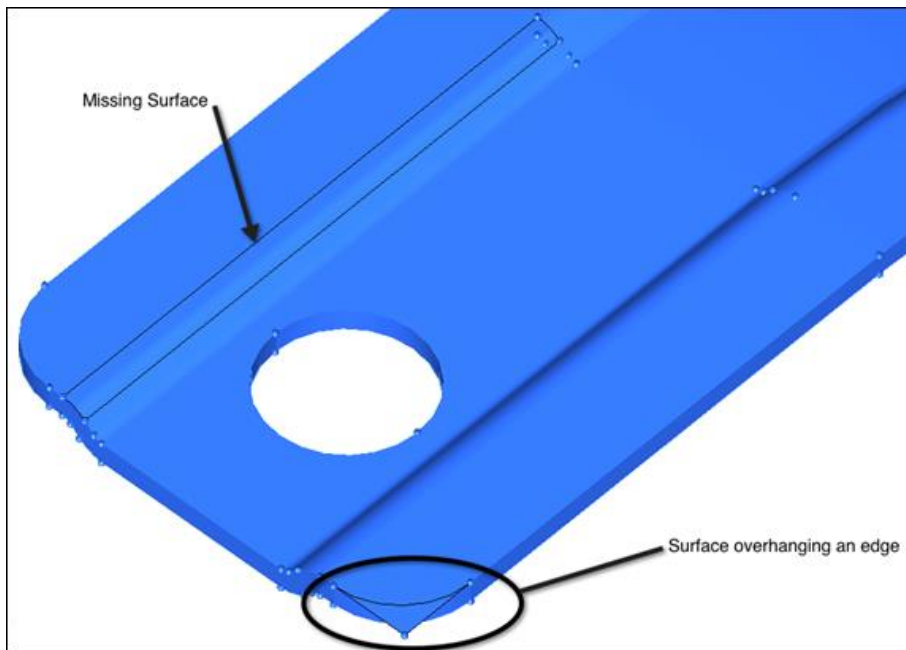
Surface overhanging an edge and a missing surface

Step 3: Delete the surface that overhangs the round corner.

1. To open the **Delete** panel, click ***Geometry*** > ***Delete*** > ***Surfaces*** from the menu bar, or press ***F2***.

2. Optional: If you opened the **Delete** panel by pressing ***F2***, set the entity selector to **surfs**.

3. Select the overhanging surface shown in the previous image.

4. Click ***delete entity***. HyperMesh deletes the selected entities.

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

△ Altair

Step 4: Create surfaces to fill large gaps in the model.

1. To open the panel from which you can create a surface, click *Geometry* > *Create* > *Surfaces* > *Spline/Filler* from the menu bar.

2. Clear the **Keep tangency** check box.

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

4. Verify the *Auto create (free edges only)* check box is selected. The **Auto create** option simplifies the selection of the lines bounding the missing surface. Once a line is selected, HyperMesh selects the remaining free edges that form a closed loop, and then creates the filler surface.

5. Zoom into the area indicated in the following image.

Area of missing surfaces

6. Select one of the red lines bounding one of the gaps (missing surfaces) shown in the previous image. HyperMesh creates a filler surface to close the hole.

7. Repeat step 4.6 to create a filler surface in the other gap.

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

## Step 5: Set the global geometry cleanup tolerance to .01.

1. Press *O*. The **Options** panel opens.

2. Go to the **geometry** subpanel.

3. In the **cleanup tol =** field, type `0.01` to stitch the surfaces with a gap less than 0.01

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

Step 6: Combine multiple free edge pairs at one time with the equivalence tool.

1. From the menu bar, click *Geometry* > *Edit* > *Surface Edges* > *Equivalence*.

2. Select the **equiv free edges only** check box.

3. Select *surfs >> all*.

4. Verify that the **cleanup tol=** is set to `0.01`. This is the global cleanup tolerance that you specified in the **Options** panel.

5. Click *equivalence*. HyperMesh combines any free edge pairs within the specified cleanup tolerance. Most of the red free edges are combined into green shared edges. The few remaining are caused by gaps larger than the cleanup tolerance.

Step 7: Combine free edge pairs, one pair at a time, using the toggle.

1. Go to the **toggle** subpanel.

2. In the **cleanup tol =** field, type `0.1`.

3. Click one of the free edges shown in the following image. When you select the edge, it will change from red to green, indicating that the free edge pair has been equivalenced.
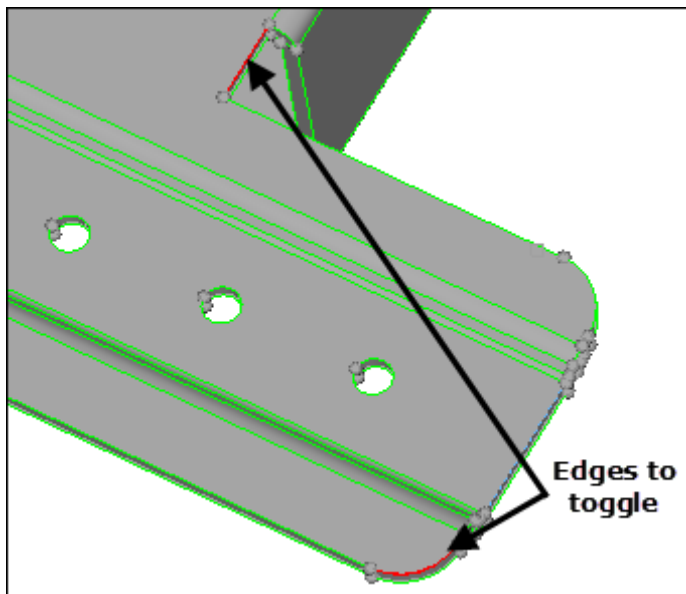

Area where free edges need to be toggled

4. Use toggle to equivalence the other edges shown in the previous image.

Step 8: Combine the remaining free edge pair using *replace*.

1. Go to the **replace** subpanel.

2. In the **Model** browser, **View** folder, right-click on *View2* and select *Show* from the context menu. The graphics area displays two edges to retain and remove for replacement.
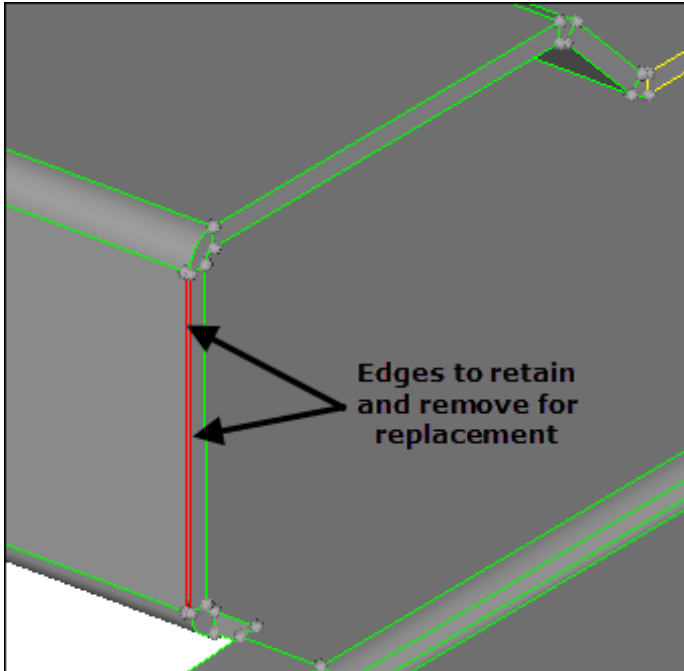
3. With the **moved edge line** selector active, click the leftmost free edge.

4. With the **retained edge line** selector now active, select the rightmost red edge.

5. In the **cleanup tol =** field, enter `0.1`.

6. Click *replace*. HyperMesh posts a message similar to, "Gap = (.200018). Do you still wish to replace?".
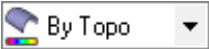
7. To close the gap, click *Yes*.
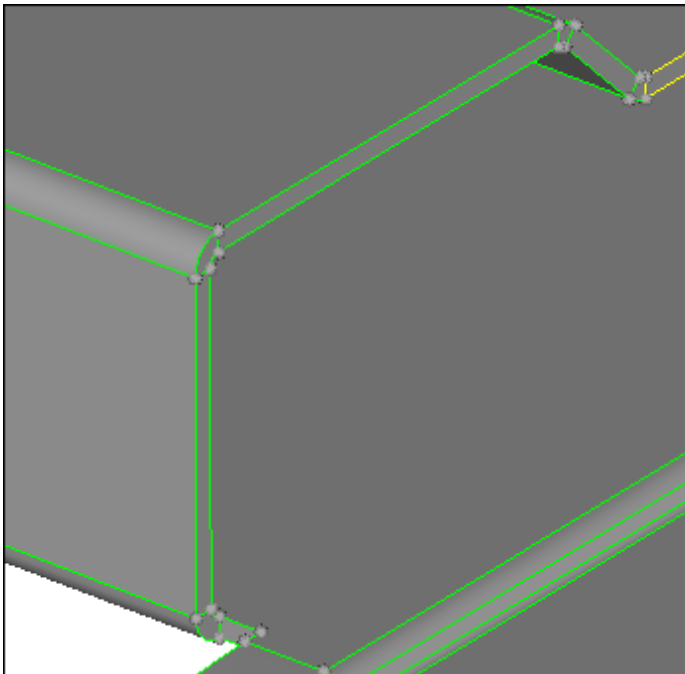
8. To exit the **replace** subpanel, click *return*.

Step 9: Find and delete all duplicate surfaces.

1. From the menu bar, click *Geometry* > *Defeature* > *Duplicates*.

2. Click *surfs* >> *displayed*.

3. In the **cleanup tol =** field, type `0.01`.

4. Click *find*. The status bar displays the following message, "2 duplicated surfaces were found."

5. To remove duplicate surfaces, click *delete*.

Step 10: Observe the model again to identify any remaining free edges, or missing or duplicate surfaces.

1. On the **Visualization** toolbar, change the geometry color mode to and click to shade the model's geometry and surface edges.

2. Observe the model again to identify the remaining free edges and missing or duplicate surfaces.

    Note: All of the edges in the model should be displayed as green shared edges, indicating that you have a completely enclosed thin solid part.

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

Step 11 (Optional): Save your work.

With the cleanup operations completed, now is a good time to save your work.

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