Exercise 2c - Generating a Midsurface

This exercise will cover the basic aspects of geometry repair and preparation for meshing. It will cover repairing problems with the geometry, midsurfacing and defeaturing.
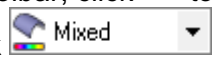
The surfaces in this model have no connectivity errors.

This could be because the file was imported without errors or because the errors were corrected using HyperMesh. In this case, errors in the topology were repaired in the previous exercise (missing surfaces are re-created, duplicate surfaces are deleted, gaps are closed, and so on).

For this exercise, you can use the Midsurface panel to generate a midsurface.

Step 1: Retrieve and view the model file.

This exercise uses the model `02c-clip_midsurface.hm`.

1. On the **Visualization** toolbar, click to shade the model's geometry and surface edges, and click to change the geometry color mode to mixed. The surfaces displayed in the graphics area represent a solid part.

Note: These visualization techniques will be necessary for viewing the newly created midsurface.
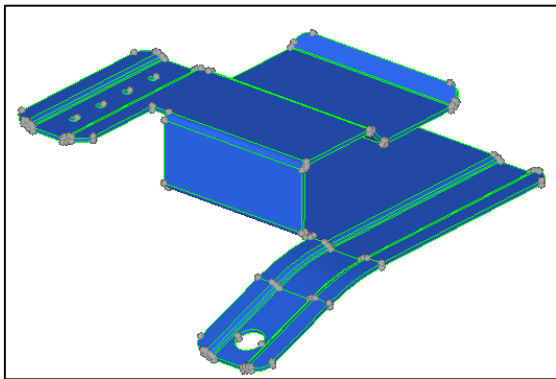

Step 2: Generate a midsurface from Midsurfaces panel.

1. From the menu bar, click *Geometry > Create > Midsurfaces > Auto*.
2. Verify that the **closed solid** option is selected, and the entity selector is active and set to **surfs**.
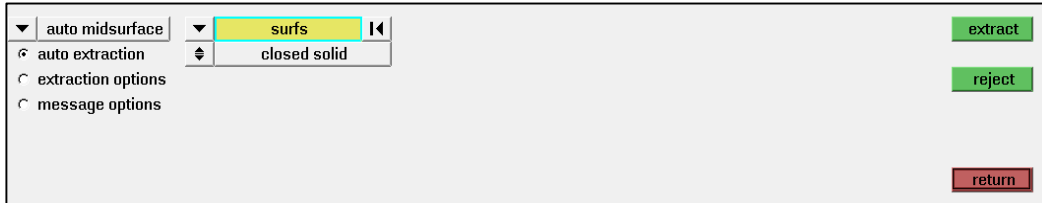
3. Select one surface.
4. To start the midsurface generation, click *extract*. HyperMesh creates the midsurface, and organizes the surfaces into a new **Middle Surface** component. When HyperMesh creates the midsurface, transparency is turned on for all of the other components in the model except the new **Middle Surface** component.

Step 3: Review the part's midsurface.

1. In the **Model** browser, only display the **Middle Surface** component. The graphics area displays the midsurfaces generated for the solid sections of the model using the **auto-midsurface** panel.
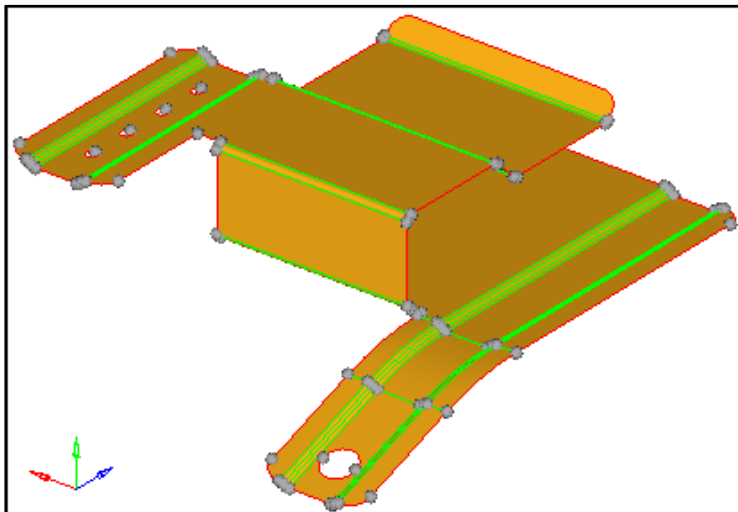
2. In the **Model** browser, turn the geometry for the **lvl10** component back on

3. To open the **Transparency** panel, click on the **Visualization** toolbar.
4. With the **comps** selector active, select a line or surface of the **lvl10** component. HyperMesh selects the entire component because the entity selector is set to **comps**.

   Tip: You may need to zoom in on the model to select a valid entity.

5. Under transparency, click several times. The surfaces in the **lvl10** component become more and more transparent.
6. **Optional:** Drag the transparency slider back and forth to control the level of transparency.
7. To visualize the midsurface, rotate, zoom, and pan.
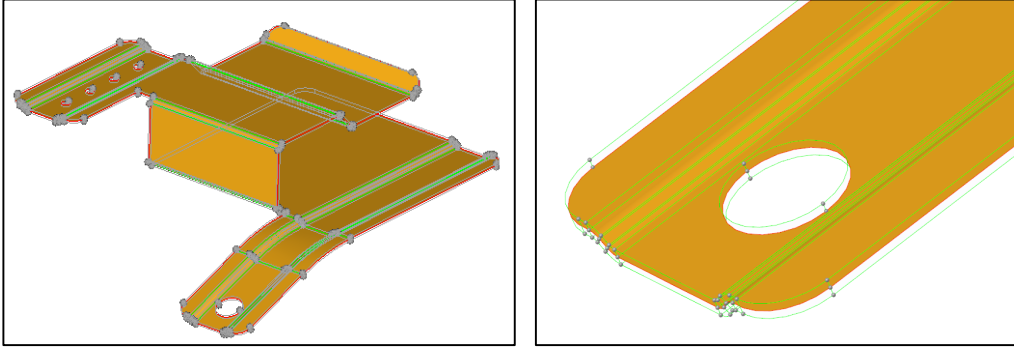
Step 4 (Optional): Save your work.

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

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

Summary

You have now created surfaces on the mid-plane of the part. These surfaces can now be meshed or further modifications can be made to their topology, depending on the requirements of the analysis.