Altair

HyperWorks

HM-2015: Auto-Midsurfacing with Advanced Extraction Options

In this tutorial, you will learn how to:

- Use the **offset+planes+sweeps** option when midsurfacing.
- Manually correct gaps in an auto-generated midsurface using the **plates edit** function.

In this tutorial, you will be using CAD geometry data for a box with thin ribs inside of it. Because the geometry consists of thin planar sections, it is assumed that it will be modeled for FEA as shell elements. The elements will be created on the mid-planes of each section.
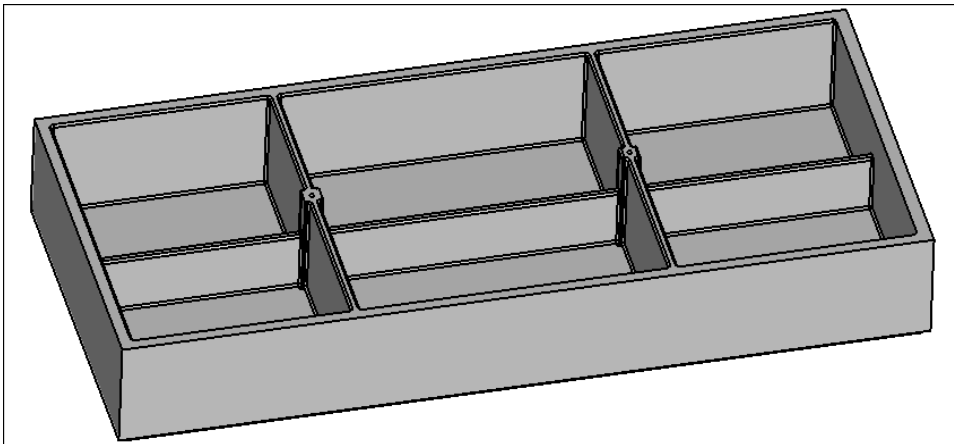
Model Files

This exercise uses the `Insert_planes.hm` file, which can be found in the `hm.zip` file. Copy the file(s) from this directory to your working directory.
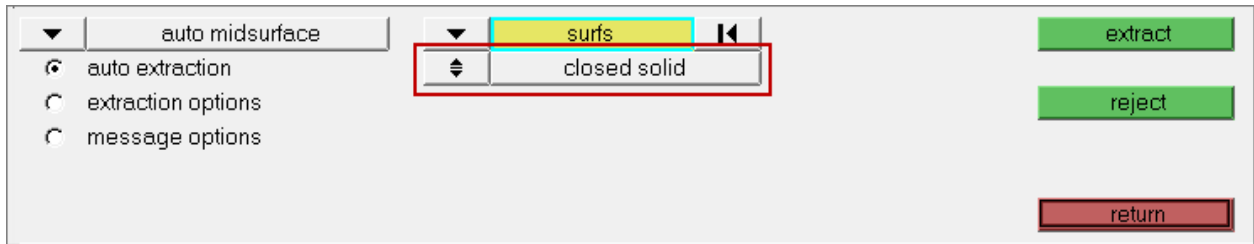
Exercise: Generating a Midsurface

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 `Insert_planes.hm` model file.

4. From the menu bar, click *Geometry* > *Create* > *Midsurfaces* > *Auto*.
5. In the **Auto Midsurface** panel, click the toggle and select **closed solid**.

6. Select any surface.

7. Click *extraction options*.
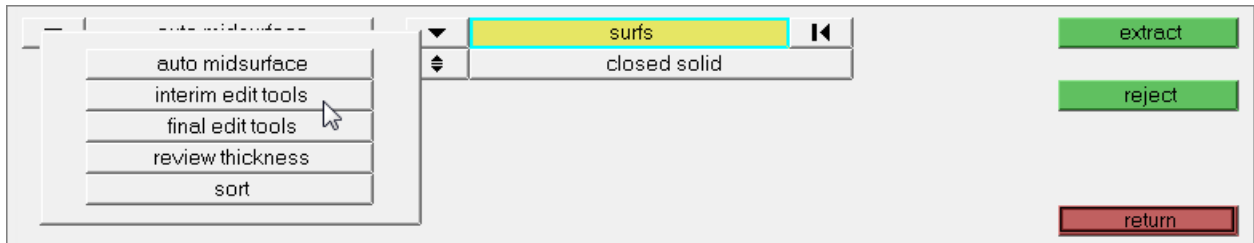
8. From the drop-down menu, select *offset+planes+sweeps*.
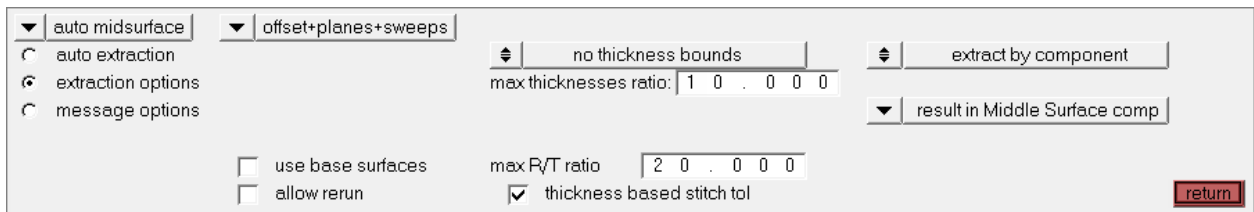
9. Go to the *auto extraction* subpanel.

10. Click *extract*.

11. Shade the model's geometry and surface edges by clicking 🔵 on the **Visualization** toolbar.

12. Review the generated midsurface by hiding the **Body.1** component in the **Model** browser. Some of the plates do not properly cross.
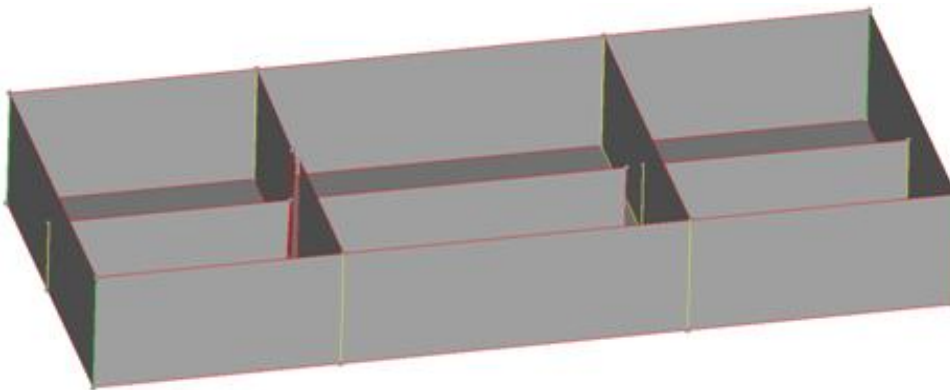
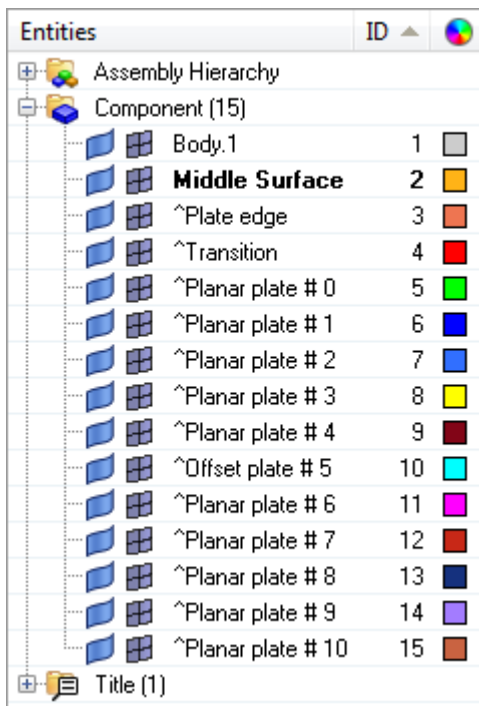## Step 2: Use Plates Edit to Resolve Midsurface Gaps

1. In **Model** browser, display the component **Body.1**.

2. On the **Visualization** toolbar, set the geometry display mode to 🔵 Mixed ▾.

3. From the panel area, select the ***interim edit tools*** from the drop-down.

|		
auto midsurface	surfs	extract
interim edit tools	closed solid	
final edit tools		reject
review thickness		
sort		return

4. From the **edit plates** subpanel, click ***show/edit all***. HyperMesh populates the **Model** browser with plates that were detected by the tool.

   Note: If you have not yet extracted the middle surface using either the **offset+planes** or **offset+planes+sweeps** options, then the model will not have any plate information yet. Plate components will not be populated in this situation.

Entities	ID ▲	🌈
⊞ 🗃 Assembly Hierarchy		
⊟ 🗃 Component (15)		
📄 ⊞ Body.1	1	⬜
📄 ⊞ **Middle Surface**	2	🟧
📄 ⊞ ^Plate edge	3	🟧
📄 ⊞ ^Transition	4	🟥
📄 ⊞ ^Planar plate # 0	5	🟩
📄 ⊞ ^Planar plate # 1	6	🟦
📄 ⊞ ^Planar plate # 2	7	⬜
📄 ⊞ ^Planar plate # 3	8	🟨
📄 ⊞ ^Planar plate # 4	9	🟫
📄 ⊞ ^Offset plate # 5	10	🟦
📄 ⊞ ^Planar plate # 6	11	🟪
📄 ⊞ ^Planar plate # 7	12	🟥
📄 ⊞ ^Planar plate # 8	13	🟦
📄 ⊞ ^Planar plate # 9	14	🟪
📄 ⊞ ^Planar plate # 10	15	🟧
⊞ 📄 Title (1)		

5. In the **Model** browser, hide the components **Body.1** and **Middle Surface**.

6. Verify that the **full plate surfs** selector is active.

7. Select the green face. HyperMesh selects all of the plates in the **^Planar plate #0** component.

8. Hide all of the plates in the **^Planar plate #0** component by right-clicking on the green face.

9. Hide the three remaining exterior sides. HyperMesh hides the components **^Planar plate #2**, **^Planar plate #3**, and **^Planar plate #4**.
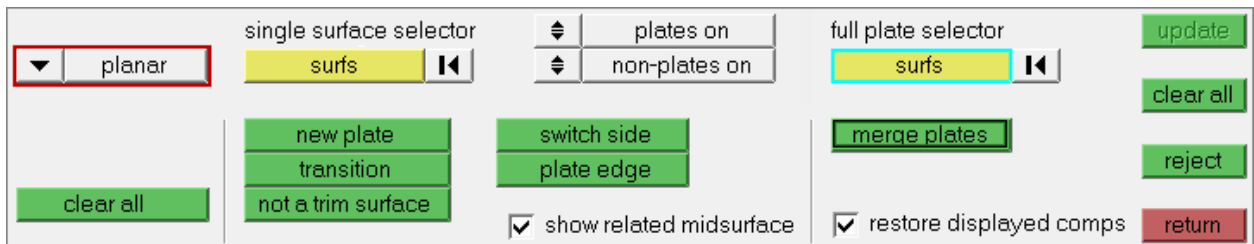
△ Altair

10. Select any face from the long interior rib, as shown in the following image. This rib was split into three groups by the algorithm, and needs to be reunited into one component.
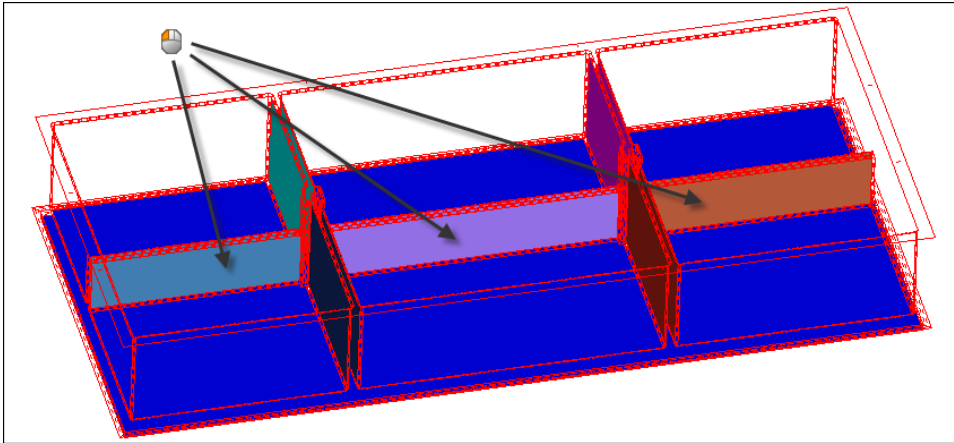
11. Select the two remaining plates from the long interior rib.
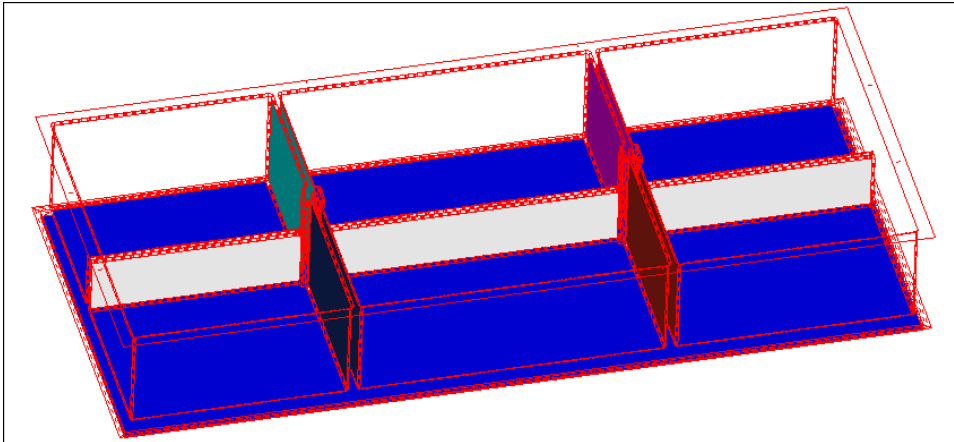
12. Merge the three plates into a single planar plate by setting the plate type to *planar*.

13. Click *merge plates* or middle-click in the graphics area. HyperMesh combines the three plates into a single component and displays them in the same color.
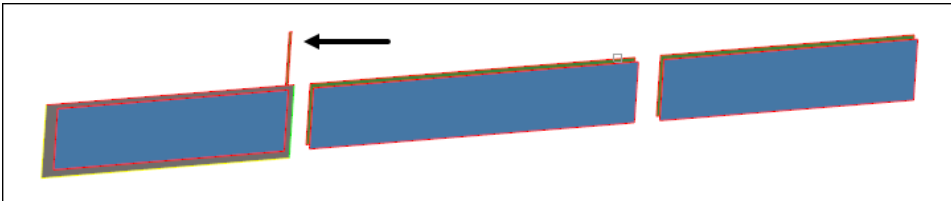
14. Using the **full plate selector**, select the newly created plate.
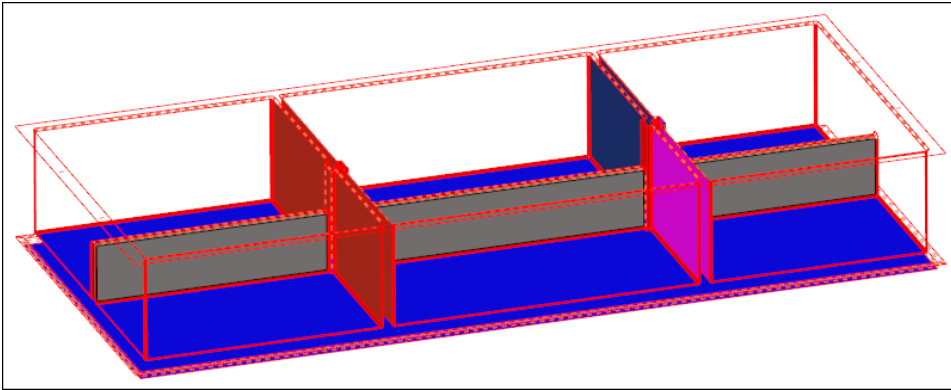
15. Click the ***single surface selector***. The selected plate is isolated, and one side is colored blue and the other side is colored green.
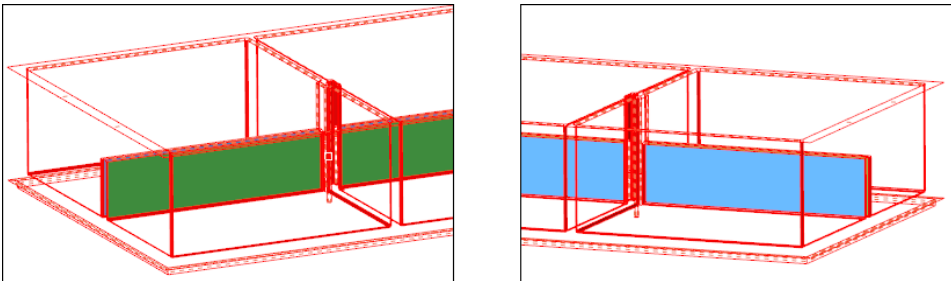

16. Review the isolated plate by toggling non-plates off/non-plates on to ***non-plates off***.

17. Notice the long narrow surface that displays with the left-most plate. Select it with the **single plate selector**, and set it to ***plate edge***.

18. Click the ***full plate selector*** to recover the view you had before isolating and reviewing the plate, and hide the plate sides shown in blue and green.

19. Merge the two remaining internal ribs.

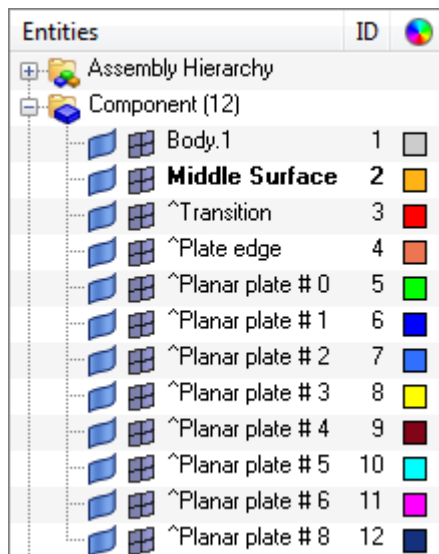20. Click ***update***.

21. Click ***return***.

22. Review the generated misdurface by hiding the **Body.1** component in the **Model** browser. The plates are closer together, but they are still not the full length of the rib due to the holes that trim the plates.
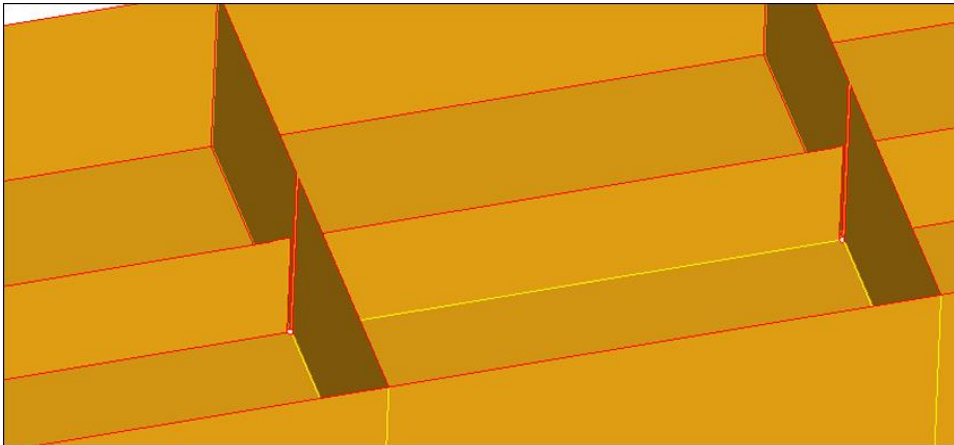
Step 3: Use Plates Edit a Second Time to Resolve Remaining Gaps

In this step, you will need to tell the auto-midsurface algorithm not to trim the plates where the holes are.

1. In the *interim edit tools* panel, **edit plates** subpanel, click *show/edit all*. HyperMesh populates less plate components in the **Model** browser because some plates were merged in the previous steps.
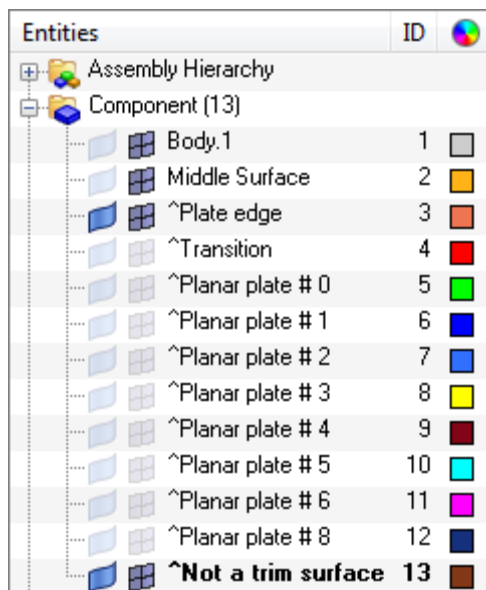

2. Hide all of the components except ^**Plate edge**.

3. Using the **single surface surfs** selector, select all four internal surfaces of the two holes.

   **Note**: Each hole has two internal surfaces.
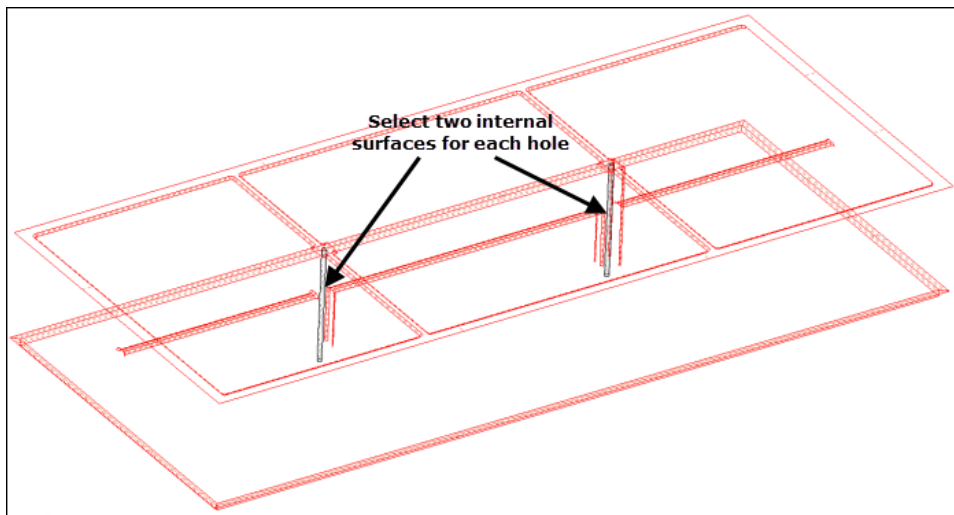
4. Click **not a trim surface**. HyperMesh organizes the selected surfaces into a new component labeled ^**Not a trim surface**.

5. Click **update**.
6. Click **return**.

7. Review the generated misdurface by hiding the **Body.1** component in the **Model** browser. There is now a yellow edge where the plates meet, which indicates that the plates are intersected correctly. It would have been possible to reorganize the plates and create the **Not a trim surface** component at the same time.
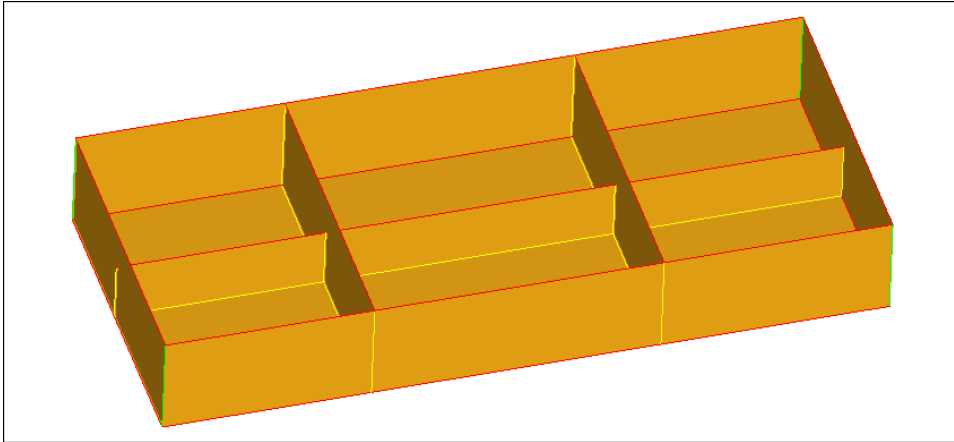
Step 4 (Optional): Save Your Work

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

Summary

The model now contains surfaces on the mid-plane of the part. You used **insert planes** and **plates edit** to ensure that there were no erroneous gaps in the generated midsurfaces. You can now mesh these surfaces, or further modify their topology, depending on the requirements of the analysis.