

Exercise 4a - 2D Shell Meshing and Topology Refinement

Overview of exercise strategy:

1. Evaluate the integrity of the mesh using **Quality Index** tool and plan your mesh editing strategy.
2. Edit the midsurface to refine/improve mesh using **Quick Edit**, **Edge Edit**, and **Point Edit** panels.
3. Mesh editing using the **Elem Cleanup**, **Place Node**, **Edit Element**, etc panels.
4. Evaluate the integrity of the midsurface, and cleanup using the **Quick Edit**, **Extend Surface**, **Replace Edge**, etc panels.
5. Simplify the geometry by deleting unneeded surface fillets, edge fillets, and pinholes.

Step 1: Open the model file 4a-Surface_Meshing.hm, and isolate *Rib 2 Midsurface*.

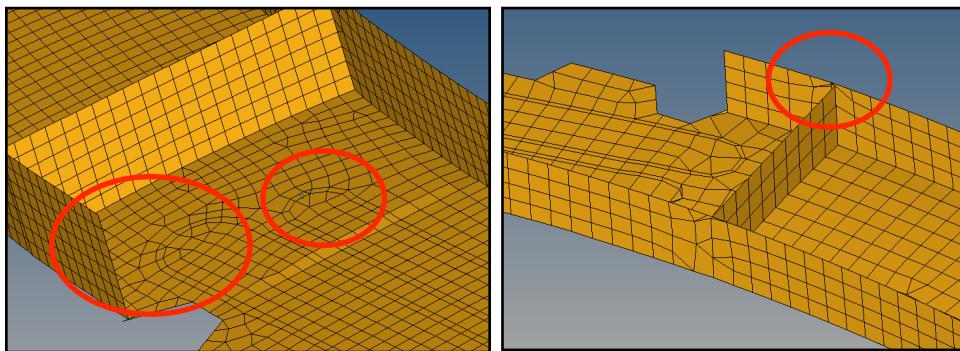
Step 2: Automatic 2D Meshing

1. Go to **Mesh > Create > 2D AutoMesh** to open the **Automesh** panel.



2. Select an **element size** of 20. Set all of the options to match the picture above.
3. Select all of the surfaces on the **Rib 2 Midsurface** component.
4. Click **mesh** and then select **return** to accept the mesh.
5. Click **return** to exit the **Automesh** panel.
6. Review the mesh. Overall, the mesh looks pretty good, but closer examination of areas around the part reveals that there are some extremely poor quality elements.

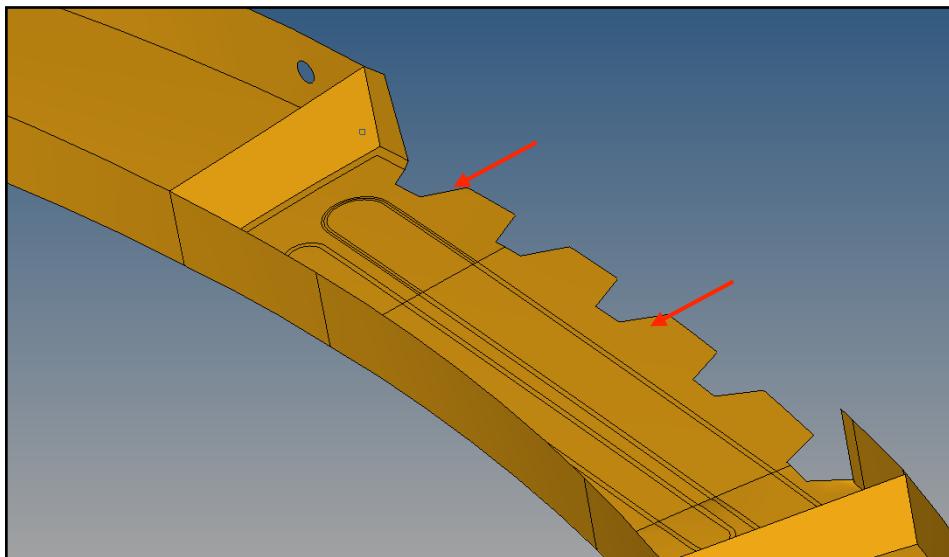




7. Manipulate the part using the Pan, Rotate and Zoom functions and identify areas of poor mesh formation.

As discussed in the lecture, HyperMesh always maintains all edges in the model except those that are suppressed.

Turning the mesh visualization off shows the surface edge lines and reveals that there are many features within the model that interfere with mesh quality. This is very often the case in geometric models imported from CAD. Topology refinement is used to fix those areas to improve the quality of the generated mesh.



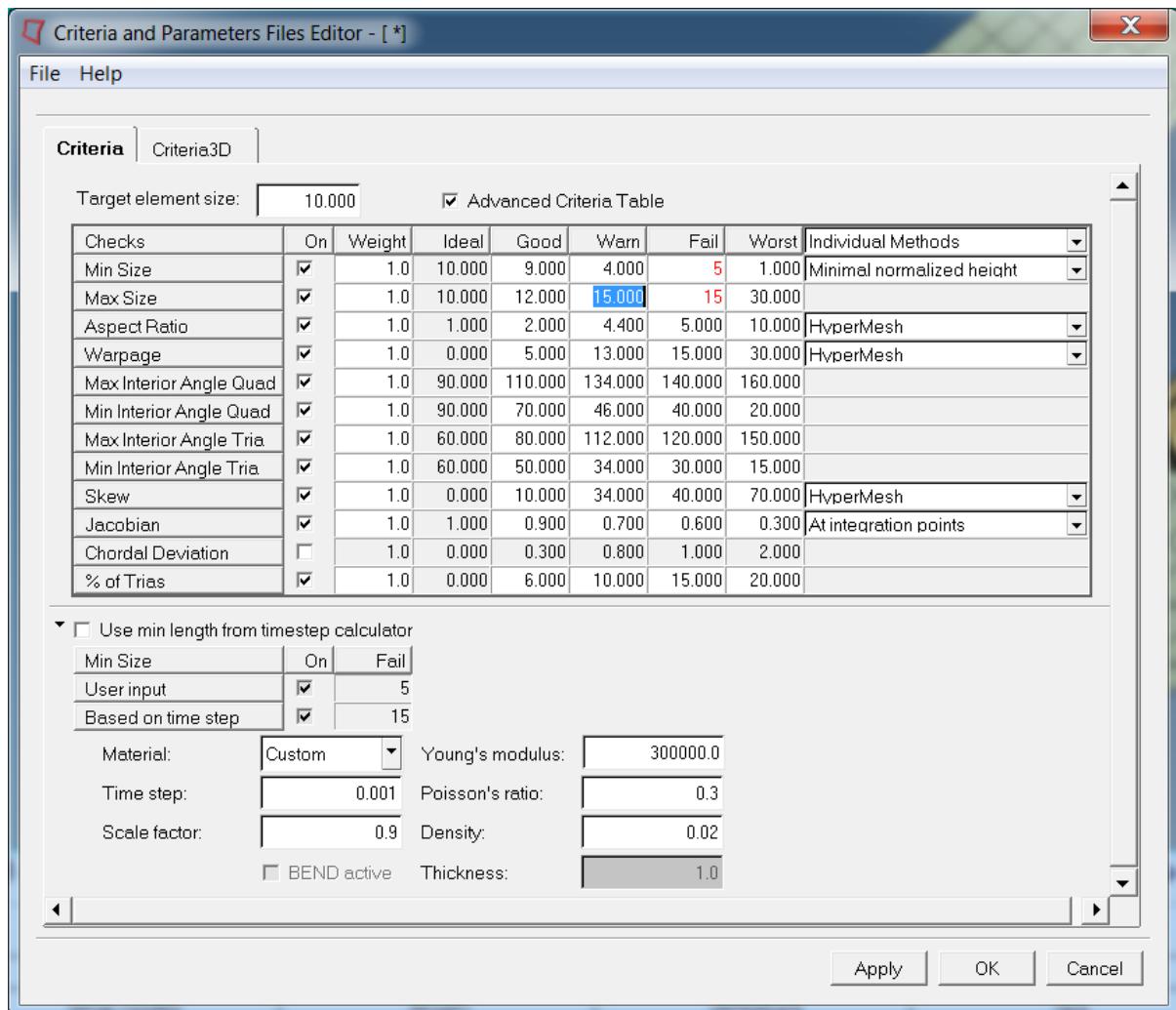
Step 3: Topology Refinement

The goal of topology refinement is to manipulate the geometry to remove or alter geometric features that cause poor element quality. HyperMesh has many tools, both automatic and manual, to assist in this process.

1. Enter the **Autocleanup** panel by selecting **Geometry > Autocleanup** from the menu bar.
2. Click the **edit parameters** button.
3. Enter 10 for the **Target element size**.

4. Leave the **Geometry cleanup** option checked and deselect all other options.
5. Click **OK**.
6. Click the **edit criteria** button.
7. Enter 10 for the **target element size**,
8. Check the **Advanced Criteria Table** option.
9. Under the **Fail column**, enter 5 for **Min Size** and 15 for the **Max Size**. Adjust the **Good** and **Warn** columns appropriately.

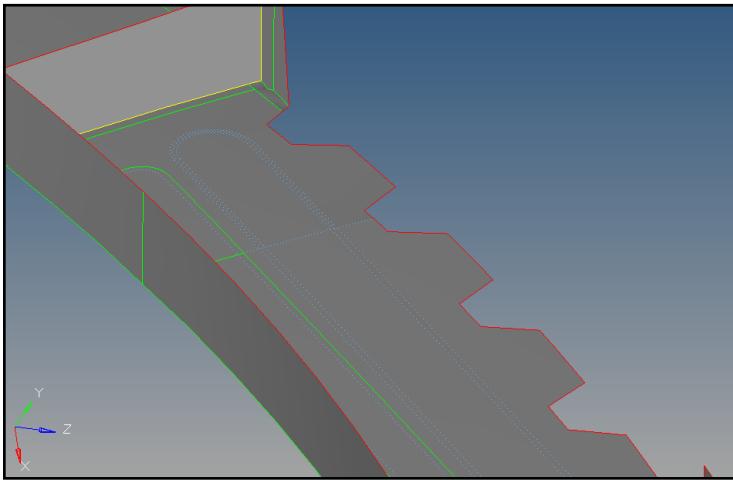
NOTE: If a value is unacceptable, it will show up in red. When adjusted to an appropriate value, it will automatically change to black.



10. Click **OK**.

11. Select all the surfaces and click ***autocleanup***.

HyperMesh has suppressed edges that it felt would not allow elements to meet the criteria to be created.



12. Click ***return*** to exit the **Autocleanup** panel.

13. Review the part again having a look at the new mesh that was remeshed automatically during the topology modification. The mesh was recreated because of the **Meshing Options** already defined (**Preferences > Meshing Options > topology revision: > advanced remesh**).

You will see that the quality of the mesh has improved drastically. There are still some areas within the mesh that need to be improved.

14. Using the **Quick Edit** panel, improve the quality of the mesh using the following functions:

- Toggle Line (suppress unneeded lines)
- Adjust Density (change number of nodes along an edge)
- Add/Remove Point
- Split Surface
- Add a washer around a hole using the **Quick Edit** panel.

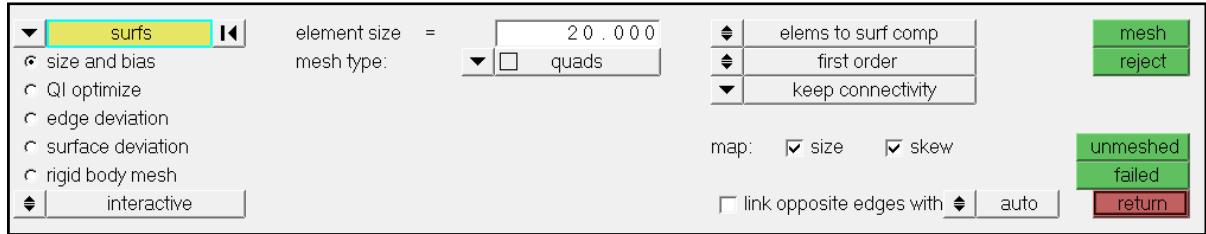
15. Use the element editing tools to achieve a Quality Index as low as possible utilizing the following tools:

- Split Element (**F6**)
- Combine Element (**F6**)
- Quality Index Cleanup Tools (**Mesh > Cleanup Elms > Quality Index**)

Step 4: Interactive 2D Meshing

While automatic meshing is quick and the overall mesh quality is good, HyperMesh allows the user to interactively manipulate a preview mesh, controlling various settings before the mesh is finalized.

1. **Delete** the existing elements and enter the **Automesh** panel.
2. Change the toggle from **automatic** to **interactive**.



3. Select the surfaces.
4. Click **mesh** to enter into interactive mesh editing mode.

HyperMesh now enters the **Size and Bias** Interactive mesh preview mode. The green mesh that is shown is only a preview mesh. Changes can be made to the various settings in the panel and the results of those changes can be visualized by clicking the **mesh** button again. The mesh will not be finalized until the **return** button is clicked.

5. From this screen try the following functions and see the effect they have on the mesh:
 - a. Adjust the edge densities by clicking on the **node number** and then click **mesh**.
 - b. Recalculate the entire model to have an element size of **10.0** by entering the number in **elem size =**, and then clicking **recalc all**, and then click **mesh**.
 - c. Change the **mesh style** so that the **element type** is all **trias** and then all **R-Trias** (remember to click **set all**), and then click **mesh**.
 - d. Alter the biasing on a few edges and determine the difference between **linear**, **exponential** and **bell curve biasing**, and then click **mesh**.
 - e. Recalculate the mesh to have **quad elements** with a size of **5.0** on all surfaces and then click **mesh**.
6. Click **return** to save the mesh.

(Optional): Save your work by clicking **File > Save As > Model**.