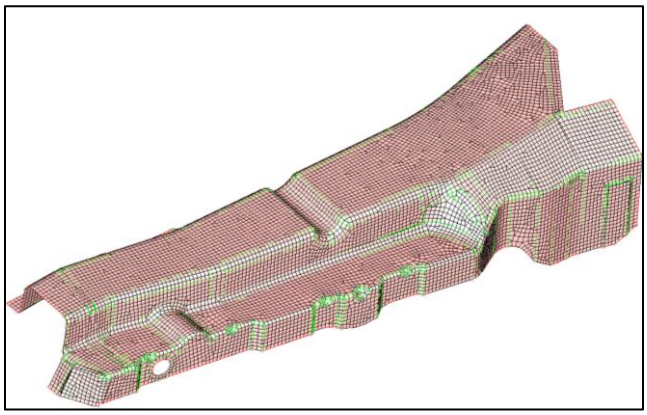
Exercise 4a - 2D Shell Meshing and Topology Refinement

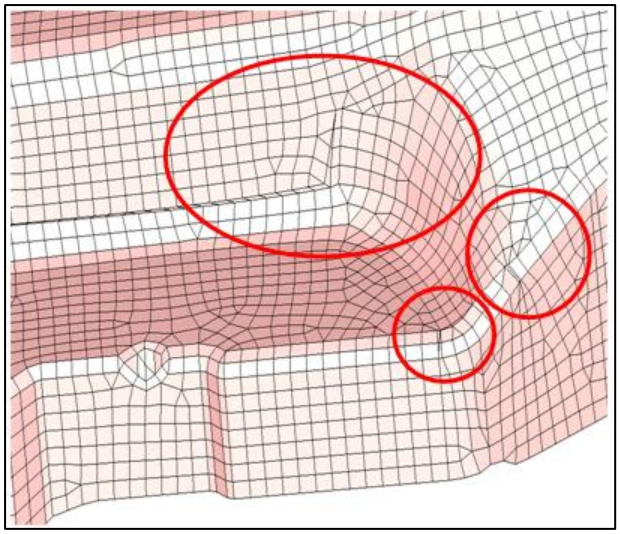
Step 1: Load the model: `04a-2D-MESH.hm`.

Step 2: Automatic 2D Meshing

1. From menu bar, click ***Mesh > Create > 2D AutoMesh*** to open the **Automesh** panel and select the displayed surfaces.

2. Mesh the part with an **element size** of 5. Set all of the options to match the picture above.
3. Review the mesh. Overall the mesh looks pretty good but closer examination of areas around the part reveals there are some extremely poor quality elements.

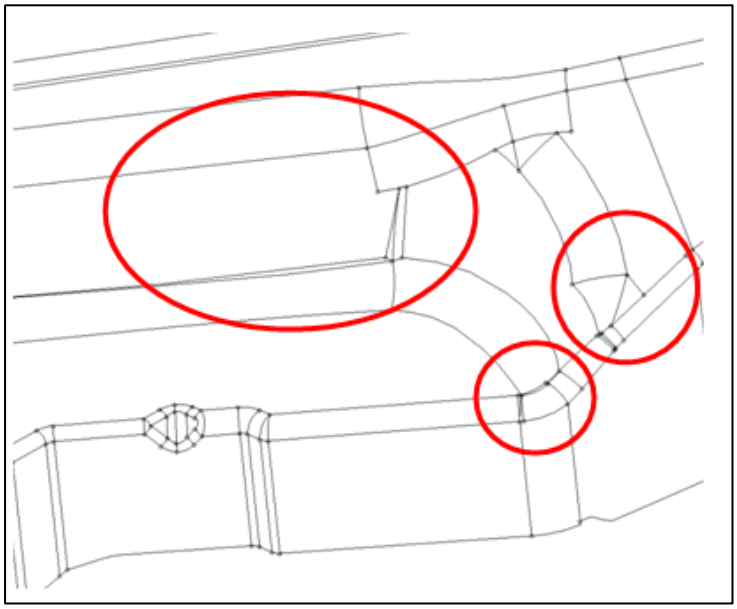




4. Manipulate the part using the Pan, Rotate and Zoom functions and identify areas of poor mesh formation.

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

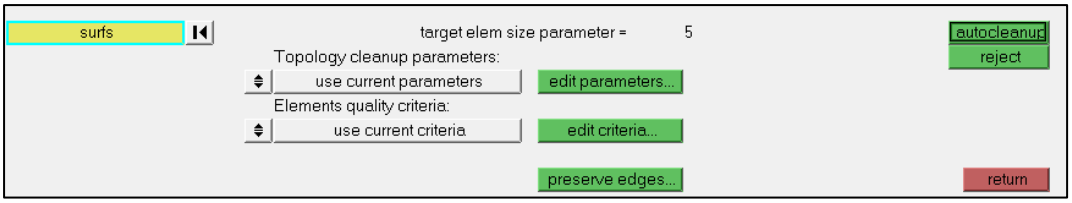
   Turning the mesh visualization off shows the surface edge lines and reveals that there are many features within the model that interfere with mesh quality. This is very often the case in geometric models imported from CAD. Topology refinement is used to fix those areas to improve the quality of the generated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



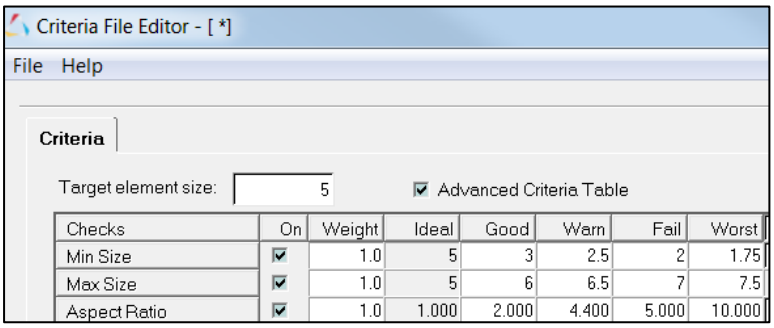
Step 3: Topology Refinement

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 ***Geometry > Autocleanup*** panel.
2. From the **Autocleanup** panel select the **edit parameters** button.

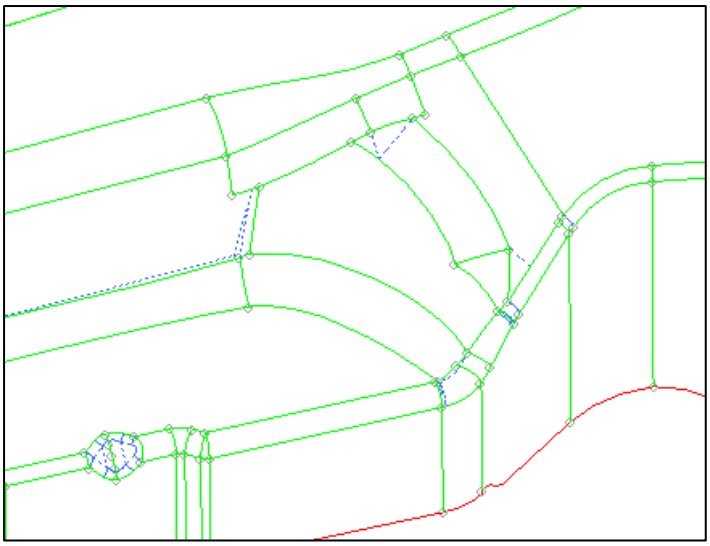


3. Enter 5 for the Target element size.
4. Leave the **Geometry cleanup** option checked and deselect all other options.
5. Click ***OK***.
6. Select the **edit criteria** button and check **Advanced Criteria Table** option.
7. Enter 5 for the **Target element size**, 3 for the ***Min Size > Good*** and 6 for the ***Max Size > Good.***



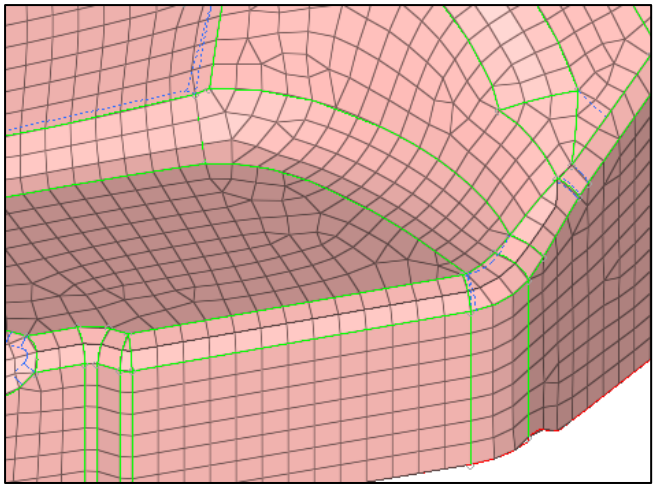
8. Click ***OK***.
9. Select all the surfaces and click **autocleanup**.

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



10. Review the part again having a look at the new mesh that was remeshed automatically during the topology modification due to the **Meshing Options** already defined (***Preferences > Meshing Options > topology revision: > advanced remesh***).

    You will see that the quality of the mesh has improved drastically.



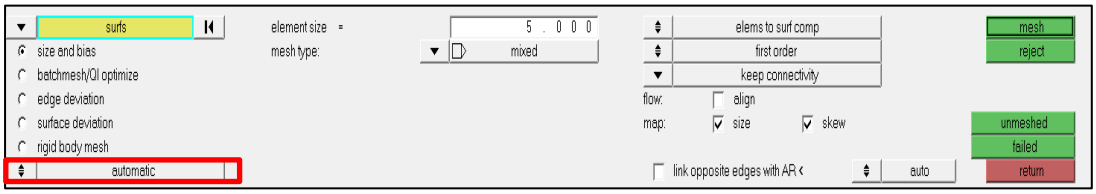
There are still some issues with the mesh though.

11. Using the ***Mesh > Quick edit (F11)*** panel improve the quality of the mesh using the following functions:
    - toggle edge
    - adjust/set density
    - add/remove point
    - split surf-node; split surf-line;

Step 4: Interactive 2D Meshing

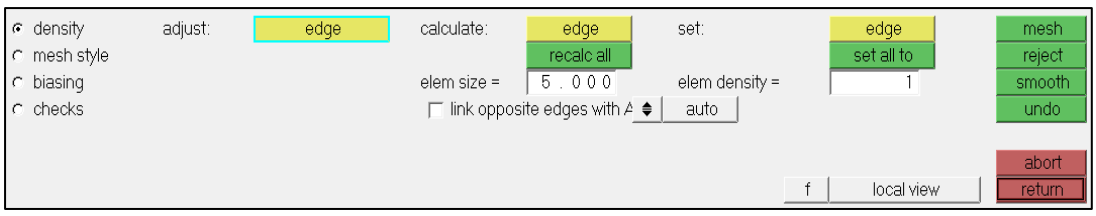
While automatic meshing is quick and the overall mesh quality is good, HyperMesh allows the user to interactively manipulate a preview mesh, controlling various settings before the mesh is finalized.

1. Delete created displayed elements and click the ***Mesh > Create > 2D AutoMesh (F12)*** panel.

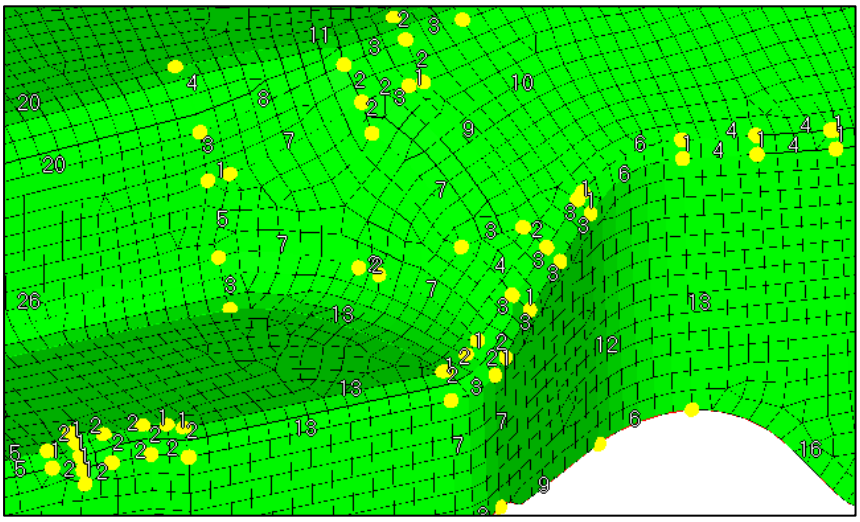


2. Change the toggle from **automatic** to **interactive.**
3. Select the surfaces in the **Standard** collector.
4. Click on **mesh** to create 2D shell elements.

   HyperMesh now enters in a secondary panel.



The green mesh that is shown is only a preview mesh and to see how changes affect it click the green **mesh** button. The mesh will not be finalized until the **return** button is clicked.

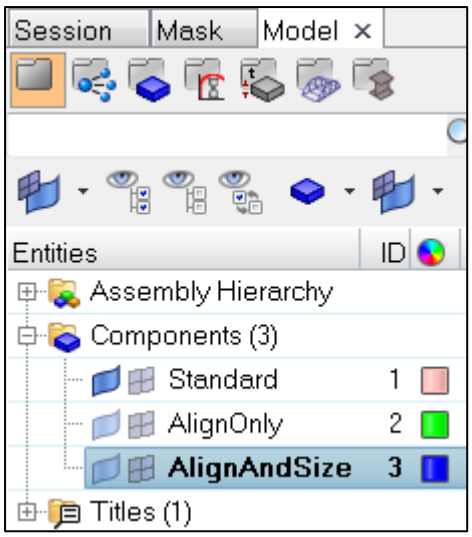


5. From this screen try the following functions and see the effect they have on the mesh. After each interactive change, click on **mesh** to update your green mesh to see effects.
    - Adjust the edge densities.
    - Recalculate the entire model to have a 6 mm element size.
    - Change the mesh style so that the element type is all trias and then all R-Trias.
    - Alter the biasing on edges and determine the difference between linear, exponential and bell curve biasing.
    - Recalculate the mesh to have 5 mm quad elements on all surfaces.
6. Click on **return** to save the mesh.

Step 5: Model Organization

As this exercise will demonstrate the differences between meshing options, multiple components will be necessary to separate the various meshes.

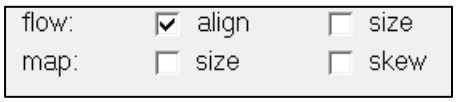
1. Create two new components called **AlignOnly** and **AlignAndSize** and make them distinct colors.
2. Organize a copy of all the surfaces into both collectors.



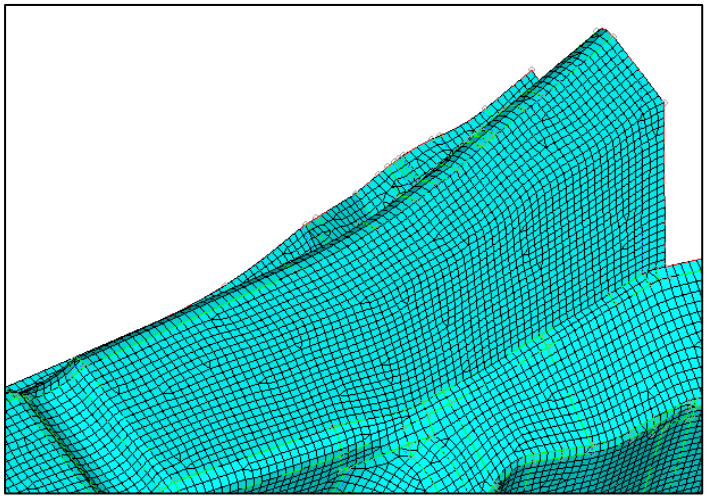
Step 6: Meshing Options

There are a few options in the **2D AutoMesh** panel which can have a profound effect on the mesh created. This section will explore these options.

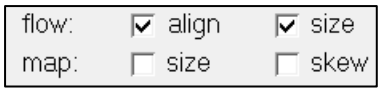
1. Make the **AlignOnly** collector **current** and the only collector visible.
2. From the ***2D AutoMesh*** panel, **size and bias** sub panel, next to **flow** select the **align** option. Leave the two **size** boxes un-selected.



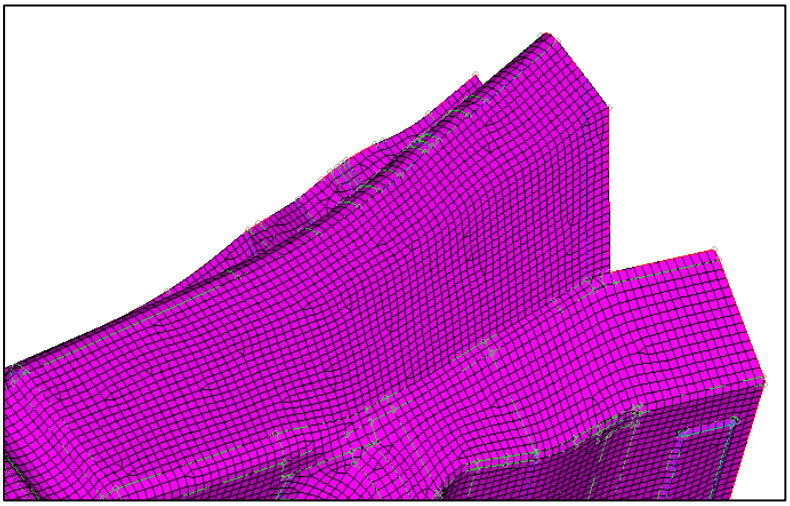
3. **Select** the surfaces in the **AlignOnly** collector and click on **mesh**.



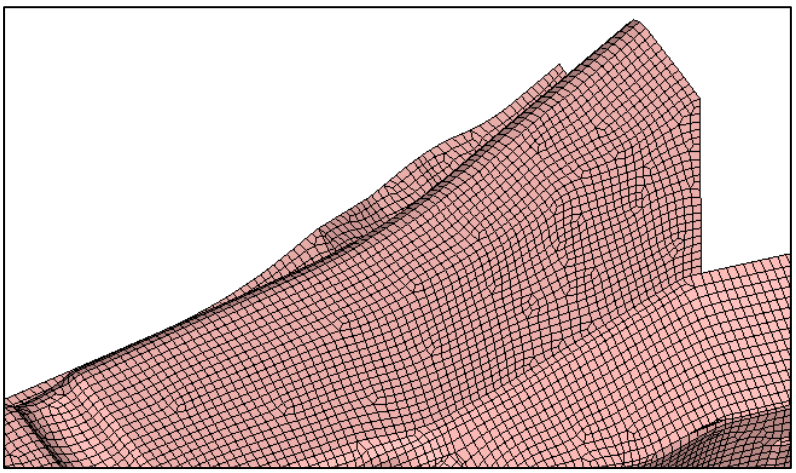
4. Make the **AlignAndSize** collector **current** and the only collector visible.
5. From the ***2D AutoMesh*** panel, **size and bias** sub panel, next to **flow** select the ***align*** and ***size*** options.



6. **Select** the surfaces in the **AlignAndSize** collector and click on **mesh.**



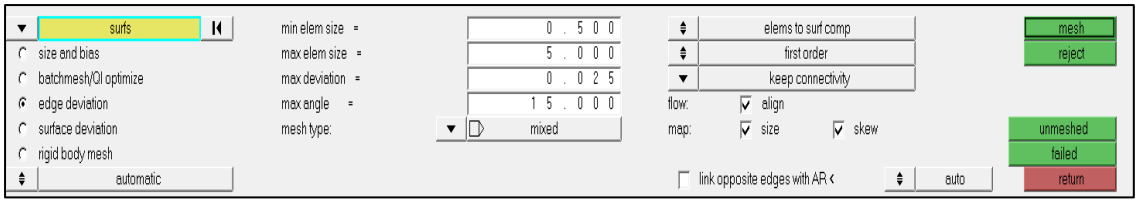
7. Utilizing the isolate function in the **Model Browser** to see the results of the different meshing options.



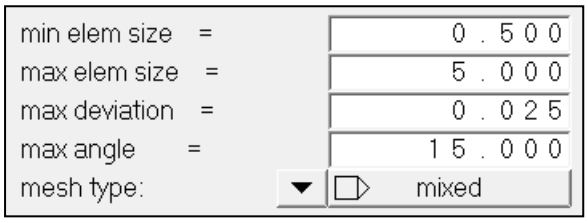
Note that the non-aligned standard mesh tends to be more orthogonal following the direction of the cardinal axis. The Align option allows the mesh to flow with the contours of the part and the addition of the size option controls the size of the elements more and results in less trias.

Step 7: Edge and Surface Deviation

1. Create 2 more component collectors called **EdgeDev** and **SurfDev** and make them distinct colors.
2. Organize a copy of all the surfaces into both collectors.
3. **Make Current** the **EdgeDev** collector and the only collector visible.
4. In the the **2D AutoMesh** panel, select the **edge deviation** subpanel.

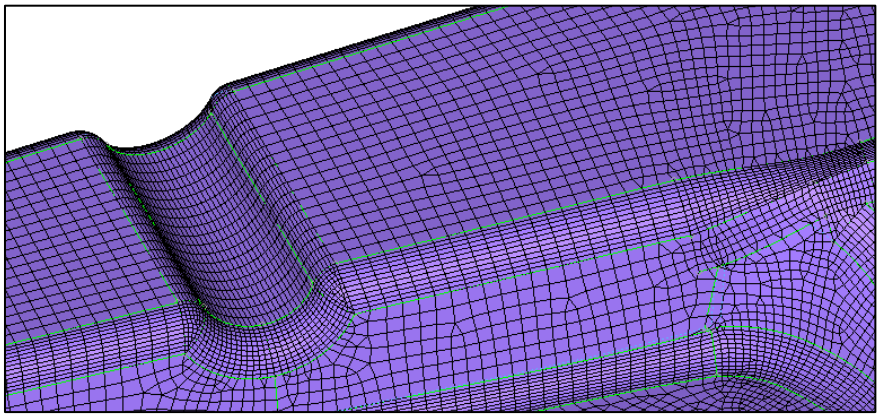


5. Set the values as follows:

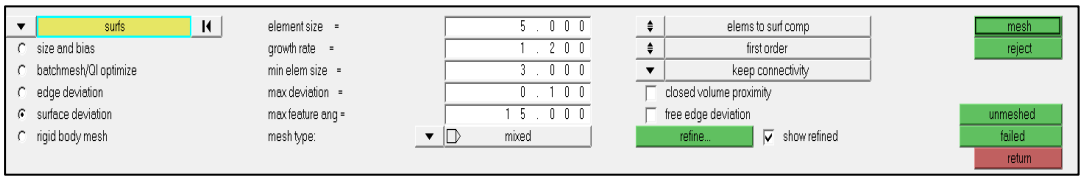


6. Mesh the part and if in interactive mode, finalize the mesh.

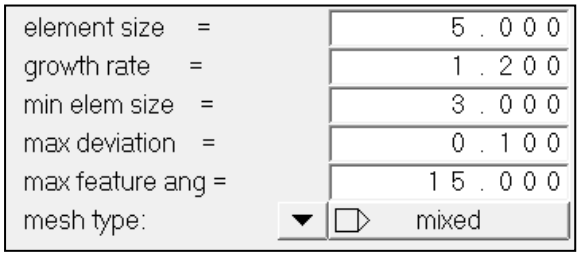
   Note how the mesh size varies depending on surfaces curvature. At rounded edges and around holes the mesh size drops down to the minimum element size to capture the curvature. Areas of no curvature are meshed at the largest element size.



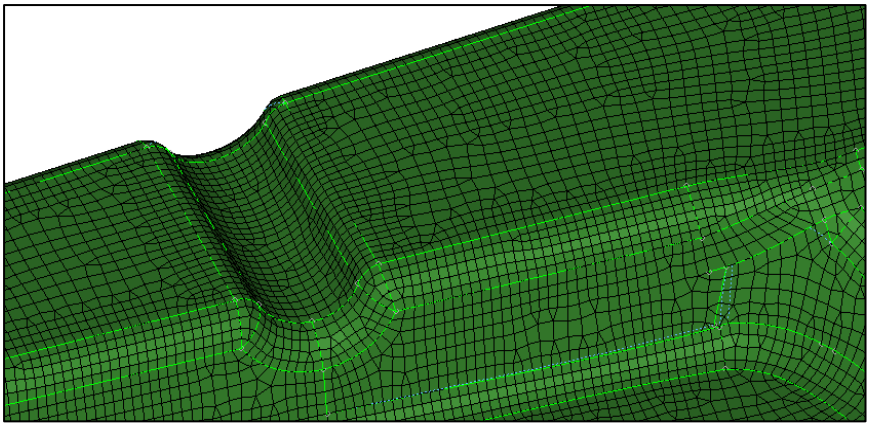
7. **Make Current** the **SurfDev** collector and the only collector visible.
8. Select the **surface deviation** sub panel.



9. Set the values as follows:



10. Mesh the part. Note how now the mesh size is dependent on and varies with the curvature of the surfaces. Fillets between and areas of high surface curvature are captured with smaller elements but large flat areas are of a higher element size.



11. Experiment in these two sub panels and determine how the interactive mesh controls can be used to enhance the feature capturing abilities of these meshing styles.