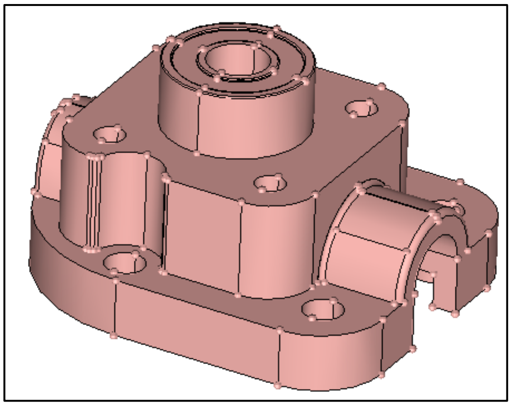
Exercise 6a - Tetra Meshing

Step 1: Load the model

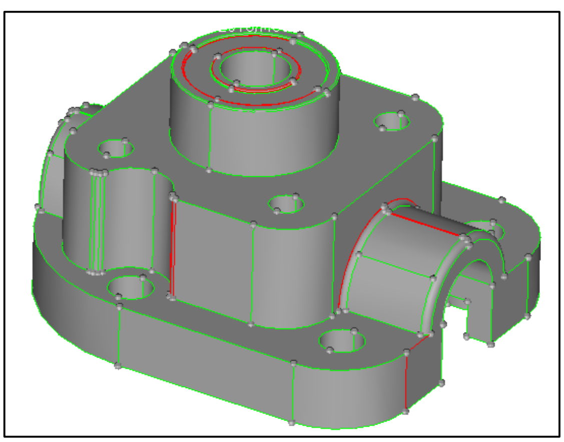
1. Load the model `06a-VOLUME-TETRA-MESH.hm`



Step 2: Attempt to TetraMesh the part

1. From the menu bar, click **Mesh > Create > *Tetra Mesh*** to create a 3D Tetramesh.
2. Select the ***Volume tetra*** sub panel.
3. Change the **enclosed volume** switch to ***surfs.***
4. Attempt to select a surface on the model. (**<u>Note:</u>** You will not be able to.)

   NOTE: With a properly enclosed model, the **Volume tetra** sub-panel will automatically select the entire volume and allow a mesh to be created. With the model now in a topological display mode, you will note there are many issues with the topology of the model. Only a fully enclosed volume can be properly tetrameshed, so we need to fix the model.



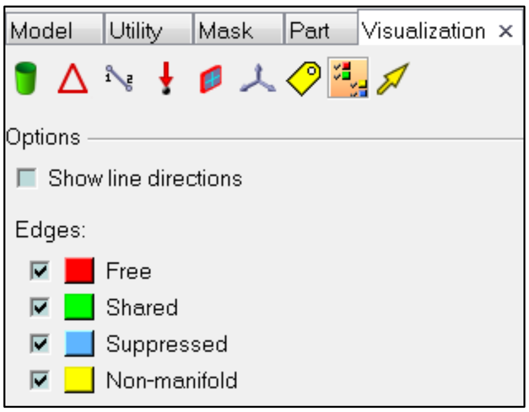
Step 3: Fix the geometry topology.

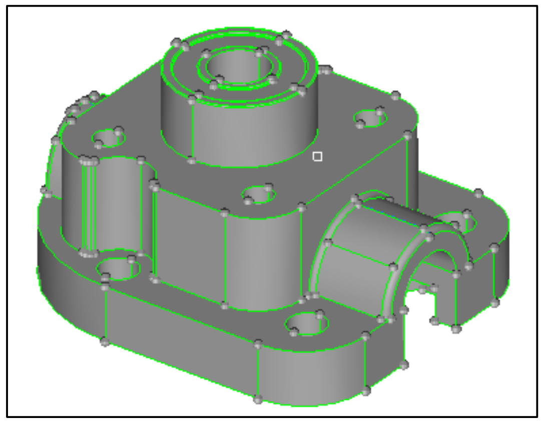
1. Using the **Geometry** menu in the menu bar, use the geometry cleanup tools to ensure a fully enclosed volume.

   <u>Hints:</u> Equivalence and Toggle will solve most of the problems. Some issues require filler surfaces and point replacement. Remember that topology visualization can assist in finding problems.

   The main tool to use is **Geometry > *Quick Edit***

2. Check the Topology with the following tool, verify if you still have free edges and if you now a closed volume of surfaces. **Select** the ***Visualization Options*** icon  and verify edges.

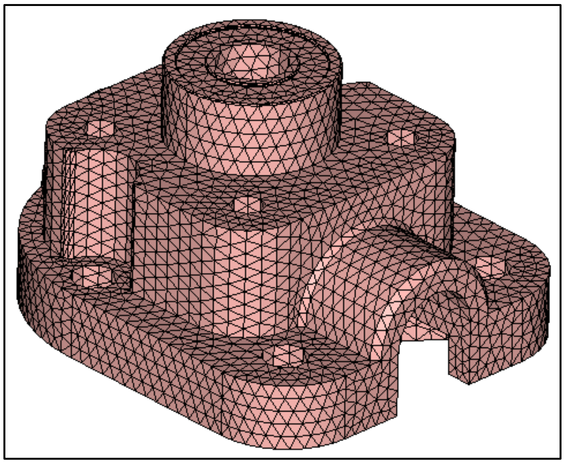




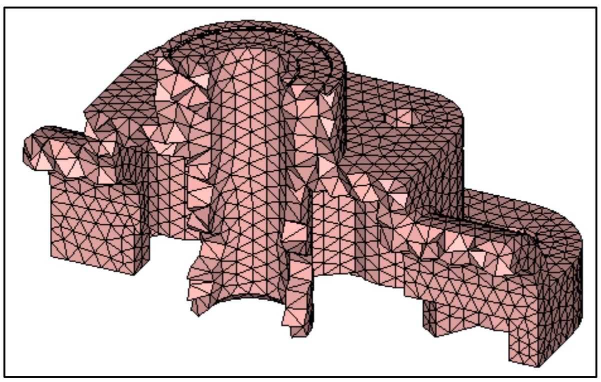
Step 4: TetraMeshing

With a properly enclosed volume you can now create the TetraMesh

1. From the menu bar, click **Mesh > Create > *Tetra Mesh***.
2. Select the ***Volume tetra*** sub panel
3. Change the **enclosed volume** switch to ***surfs***.
4. Select a surface on the model. HyperMesh will automatically select all of the surfaces that enclose the volume. If this fails, there are still errors in the volume and need to be corrected using the geometry cleanup tools.
5. Leave all the default values and enter `4` into the **element size=** field.
6. Click on ***mesh*** to mesh the part. The part should now look similar to this:



7. Mask  half part to see the Tetrahedral Element structure.



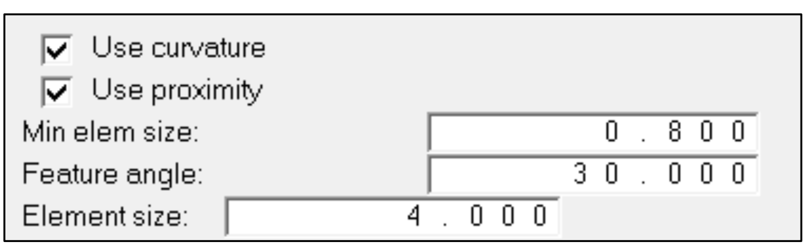
8. Now **delete** the mesh.

   From menu bar, click **Mesh > Delete > *Elements***
   Click ***elems >> displayed*** and click **delete entity**.

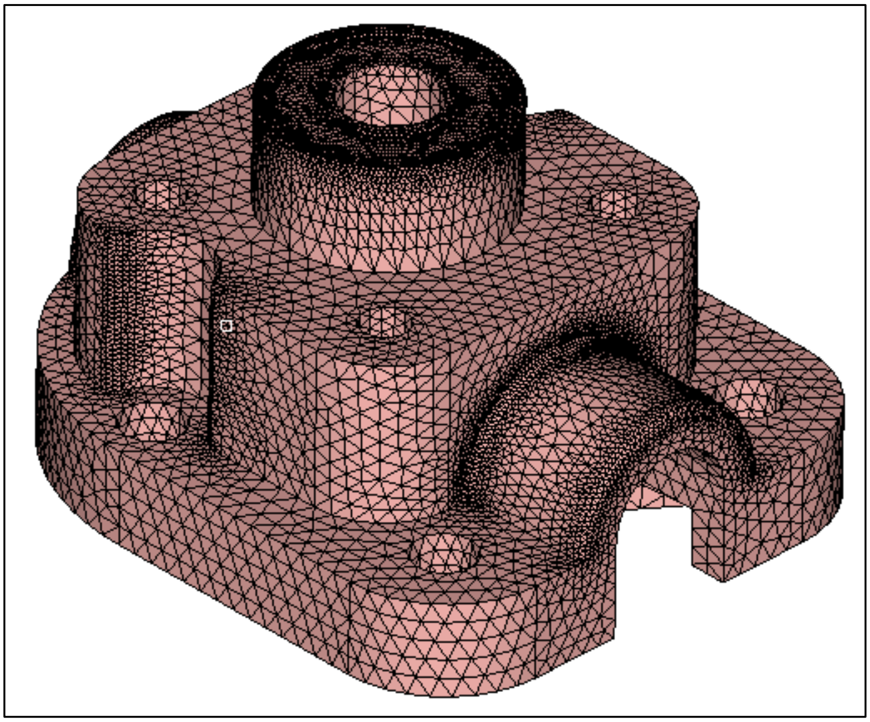
Step 5: Using Proximity and Curvature Options

Proximity and Curvature options can provide a mesh that adheres closer to the geometry in areas of curvature or small cross sections.

1. From the **Volume tetra** subpanel, select the part and select the **Use proximity** and **Use curvature** options
2. Set the following fields to the values shown:

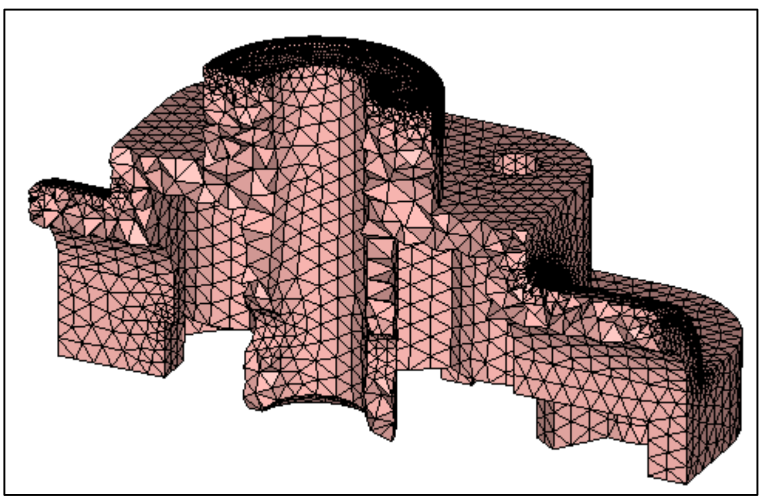


3. Click on ***mesh*** to mesh the part.



Note the areas of curvature have a smaller mesh size to better capture the geometric curvature.

4. (Optional): Mask  half part to see the Tetrahedral Element internal structure.



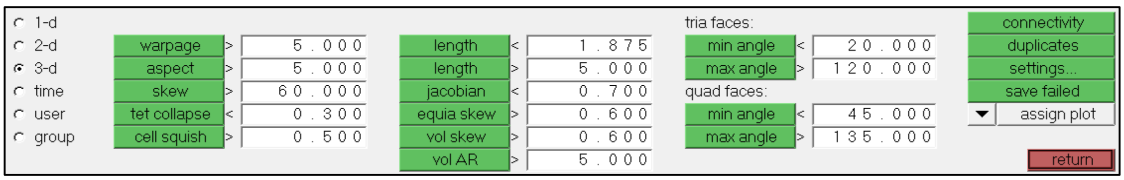
Step 6: Check and Improve the mesh quality.

To improve the overall Tetrahedral Element quality, we will check the **tet collapse** value of the elements.

Tetra elements whose collapse value falls below the value specified are highlighted when the tetra collapse function is selected. These elements remain highlighted until the **Check Elems** panel is exited.

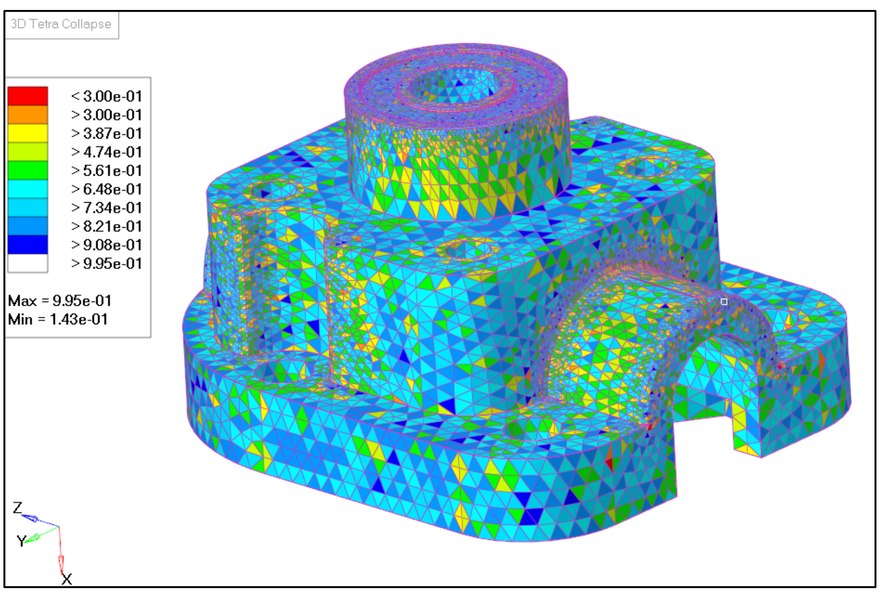
HyperMesh calculates tetra collapse by the following procedure. At each of the four nodes of the tetra, the distance from the node to the opposite side of the element is divided by the square root of the area of the opposite side. The minimum value found is normalized by dividing it by 1.24, and then reported. As the tetra collapses, this value approaches 0.0. For a perfect tetra, this value is 1.0.

1. Find the **Mesh > Check > Elements > *Check Elements*** option from the menu bar.
2. Select the ***3-d*** sub-panel.
3. Enter `0.3` into the **tet collapse<** field and click the ***tet collapse*** button.



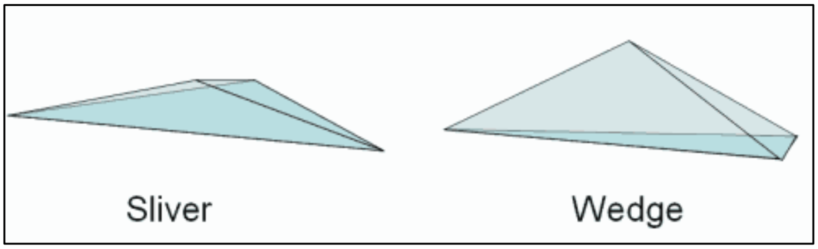
Note the number of failed elements in the dialog bar; the value should be around 80 elements.

4. **Save** the failed elements by selecting ***save failed***.
5. **Select** the switch standard and choose the option ***assign plot*** assign plot, click on ***tet collapse*** to view a contour map of **3D Tetra Collapse.**



6. In order to improve Tetrahedral Element quality, you can use the following tool from menu bar **Mesh > Check > Elements > *Tetra Mesh Optimization***

   Use this tool to modify an existing tetramesh, either by moving nodes or remeshing, to meet required parameters. One function is to remove sliver elements--tetrahedral elements which are so flattened that all of their nodes are very close to planar. If the element's Aspect Ratio (the ratio of its maximum length to its minimum length) is high, the element is a sliver; otherwise, it is a wedge.

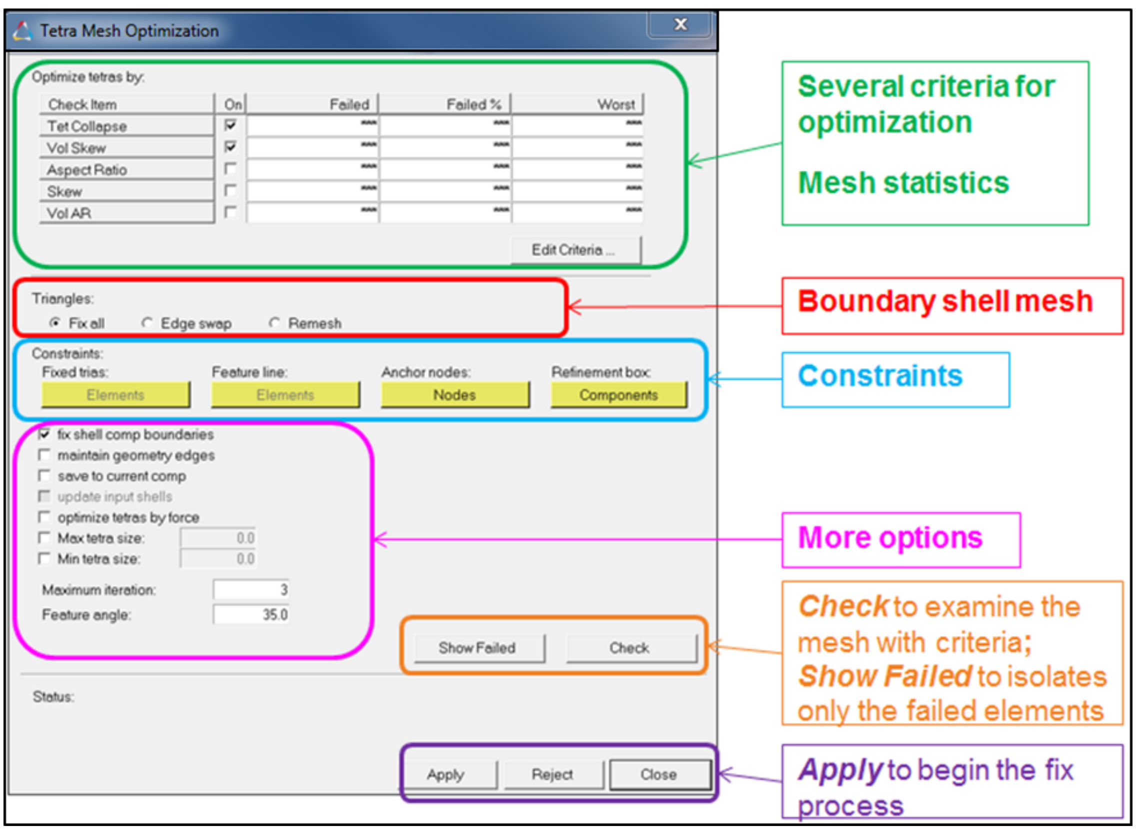


This sliver is nearly flat in the horizontal plane, while this wedge is nearly flat in the vertical plane.

When you click ***Tetra Mesh Optimization***, you will first be prompted with a temporary panel to select a set of elements to fix.

7. Select **elems >>*displayed*** and click on ***proceed***.

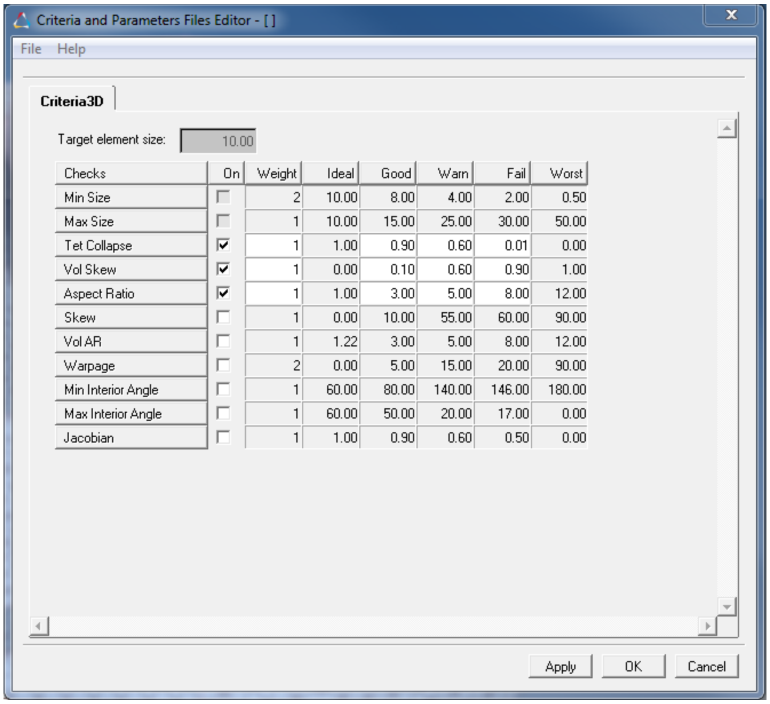
8. A **Tetra Mesh Optimization** window opens which contains the tools and settings for fixing slivers and wedges. The utility also has the ability to constrain trias, feature lines, nodes or elements within a refinement box.



There are many criteria that you can consider in fixing such elements, each of which is drawn from the **Edit Criteria…**

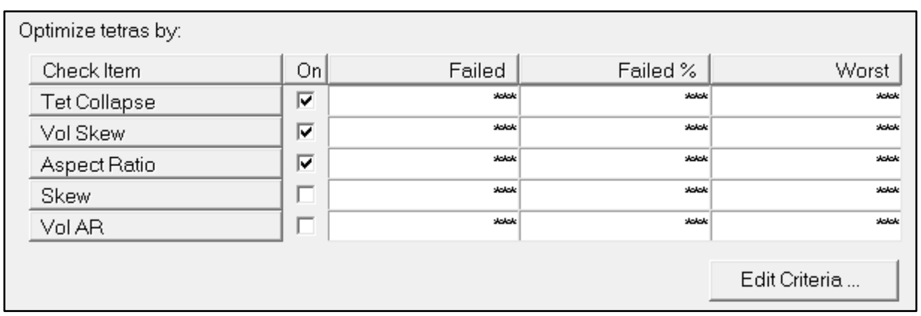
9. Click on **Edit Criteria…,** this will open the **Criteria File Editor** to change the element quality requirements.

10. Select ***Tetra Collapse, Vol Skew*** and ***Aspect Ratio***, as shown below.



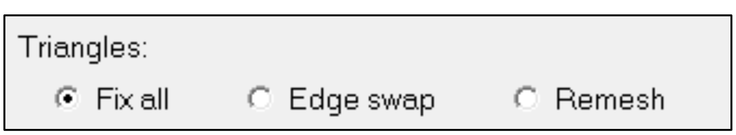
Checks	On	Weight	Ideal	Good	Warn	Fail	Worst
Min Size	☐	2	10.00	8.00	4.00	2.00	0.50
Max Size	☐	1	10.00	15.00	25.00	30.00	50.00
Tet Collapse	☑	1	1.00	0.90	0.60	0.01	0.00
Vol Skew	☑	1	0.00	0.10	0.60	0.90	1.00
Aspect Ratio	☑	1	1.00	3.00	5.00	8.00	12.00
Skew	☐	1	0.00	10.00	55.00	60.00	90.00
Vol AR	☐	1	1.22	3.00	5.00	8.00	12.00
Warpage	☐	2	0.00	5.00	15.00	20.00	90.00
Min Interior Angle	☐	1	60.00	80.00	140.00	146.00	180.00
Max Interior Angle	☐	1	60.00	50.00	20.00	17.00	0.00
Jacobian	☐	1	1.00	0.90	0.60	0.50	0.00

11. Click on ***Apply*** and ***OK***.
12. You're again in the **Tetra Mesh Optimization** window.
13. The 3 previous criteria are selected in the **Optimize tetras by**: section.



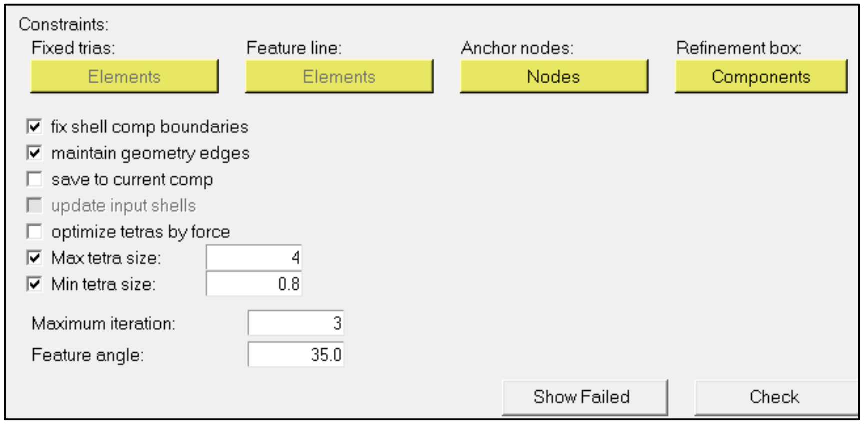
Check Item	On	Failed	Failed %	Worst
Tet Collapse	☑	***	***	***
Vol Skew	☑	***	***	***
Aspect Ratio	☑	***	***	***
Skew	☐	***	***	***
Vol AR	☐	***	***	***

14. In the **Triangles:** section, select the following, as shown also in the picture below:
    - ***Fix all*** option.

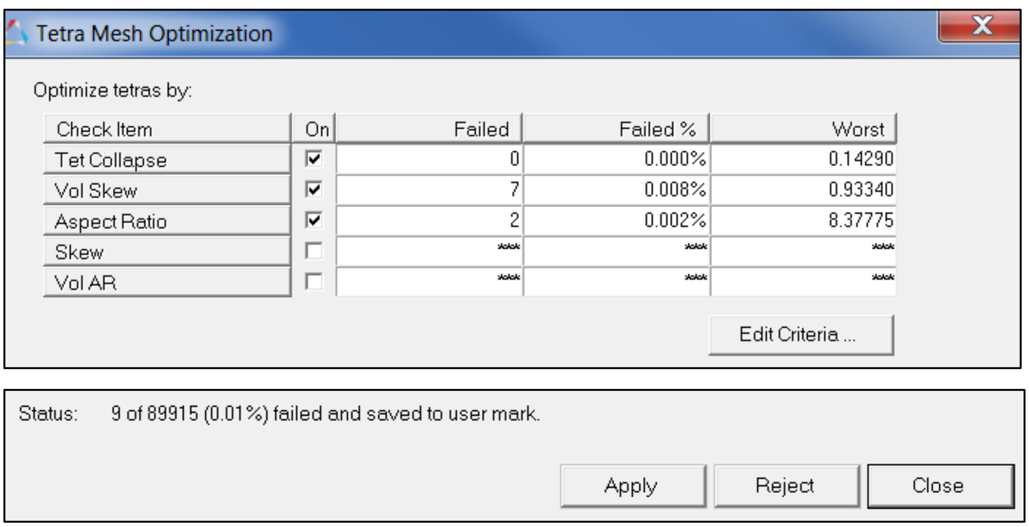


15. In the **Constraints:** section, select the following, as shown also in the picture below:
    - ***fix shell comp boundaries*** option.
    - ***maintain geometry edges*** option.
    - ***Max tetra size,*** enter `4`.
    - ***Min tetra size,*** enter `0.8`.

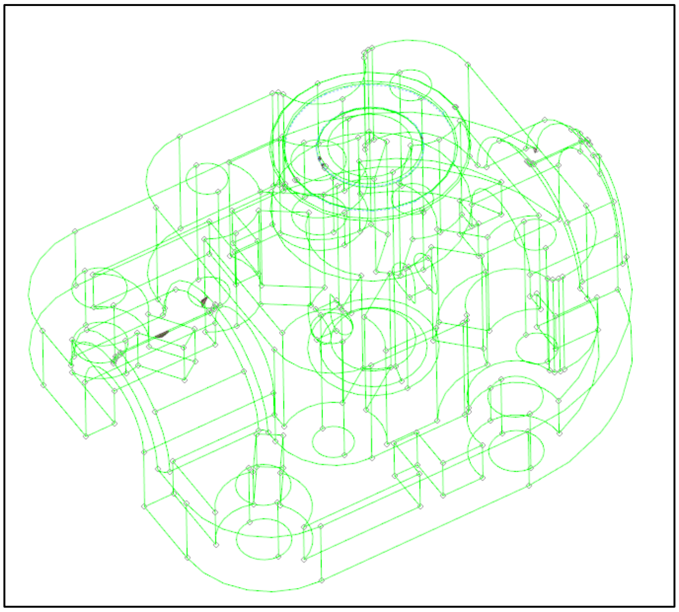
- Leave the other options with default values.



16. Click on ***Check*** button, to examine the mesh and count the number of bad elements, according to the criteria supplied (Jacobian, Volume Skew, etc.) The results display in the ***Status:*** area.



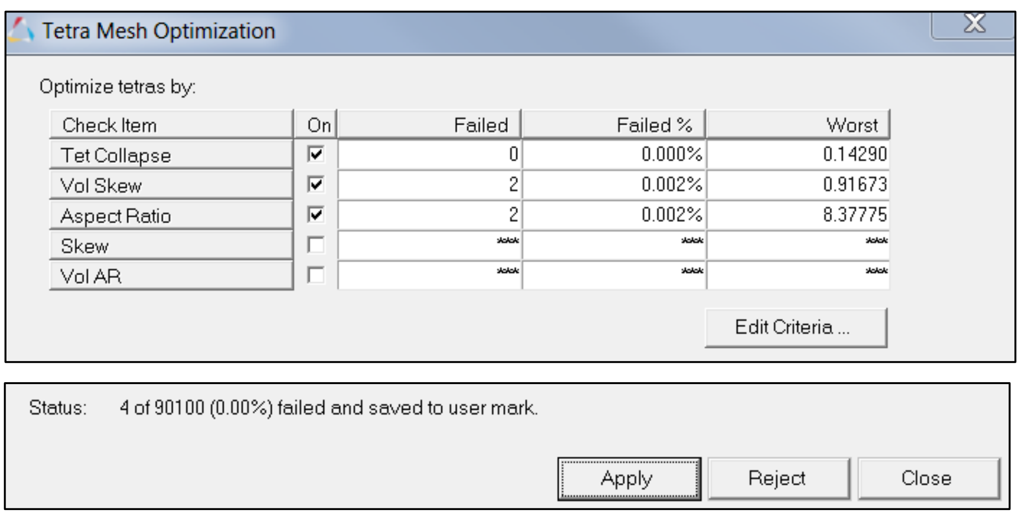
17. Click on ***Show Failed*** to isolates only the failed elements in the graphics area.



18. Click on ***Apply*** to begin the fix process. The mesh is scanned and the program will try to fix as many elements as it can in accordance with the specified settings and criteria. You can abort the fix attempt early by clicking holding down the right-mouse button.

    Note that there can be a significant delay before HyperMesh finishes its current fix attempts and stops processing.

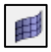
    The results are shown next:



19. If the results of the fixes are acceptable, click on ***Close*** to exit from **Tetra Mesh Optimization** utility

20. If the results of the fixes are unacceptable, click ***Reject*** to revert the mesh to its pre-fixed state.

    NOTE: You can only undo one fix operation this way--you cannot "back up" more than one step!

21. Click on  to unmask all elements.
22. Refer to ***Online help*** to get more details.
23. **Delete** the mesh.

    From menu bar, click **Mesh > Delete > *Elements***

    Click ***elems >> displayed*** and click **delete entity**.
24. Go to **Step** number **8**.

[OPTIONAL] Step 7 (from Step6-point5): Other methods to check and improve the mesh quality

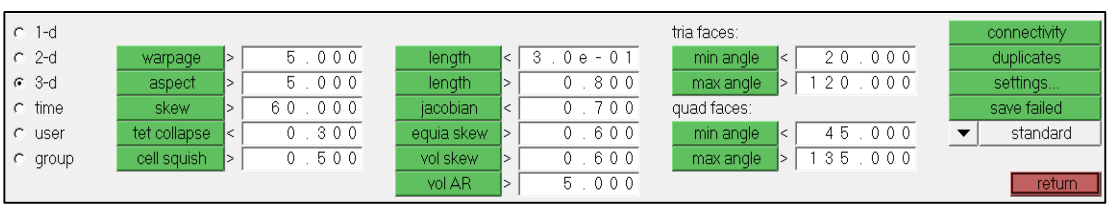
1. Use **Geometry Cleanup** tools and **Tetra remesh** functions to try to achieve the best possible mesh. Experiment with different techniques and discover the results.

   To improve the overall Tetrahedral Element quality, we will check the **tet collapse** value of the elements.

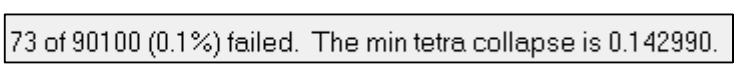
   Tetra elements whose collapse value falls below the value specified are highlighted when the tetra collapse function is selected. These elements remain highlighted until the **Check Elems** panel is exited.

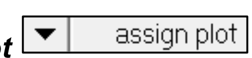
   HyperMesh calculates tetra collapse by the following procedure. At each of the four nodes of the tetra, the distance from the node to the opposite side of the element is divided by the square root of the area of the opposite side. The minimum value found is normalized by dividing it by 1.24, and then reported. As the tetra collapses, this value approaches 0.0.

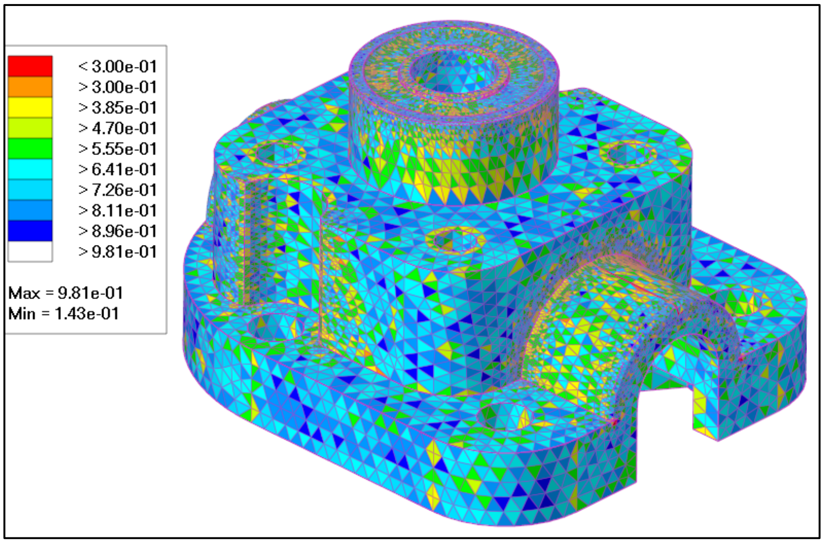
   For a perfect tetra, this value is 1.0.
2. Go to **Mesh > Check > Elements > Check Elements**.
3. Select the ***3-d*** sub panel.
4. Enter `0.3` into the **tet collapse<** field and click the ***tet collapse*** button.

   Note the number of failed elements in the dialog bar. The value should be around 100 elements.

5. **Save** the failed elements by selecting ***save failed***.
6. **Select** the switch  and choose the option ***assign plot***  , click on ***tet collapse*** to view a contour map of **3D Tetra Collapse.**



7. Isolate the failed elements

   Failed elements can be isolated on the screen anytime using the following procedure.

   A. Go to the **mask** function.

   B. Click the ***elems*** button.

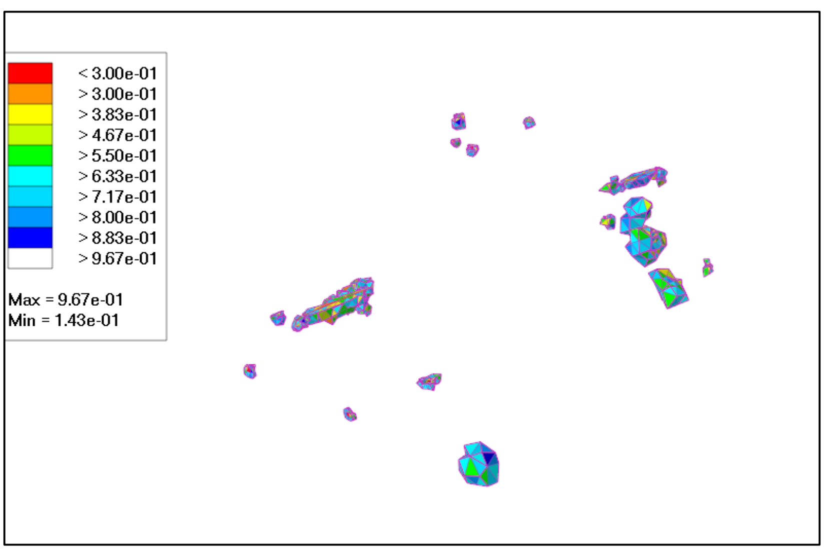
   C. Select ***retrieve***.

   D. Click the ***elems*** button again.

   E. Select ***reverse***.

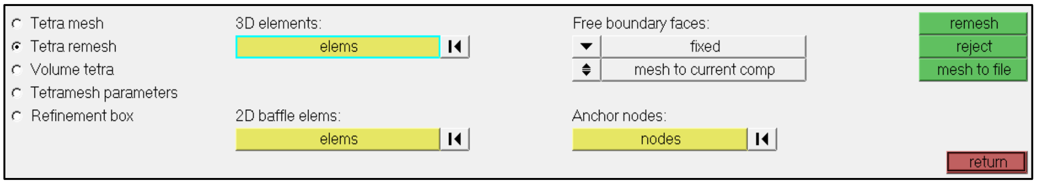
   F. **mask** the elements.

8. Using the **unmask adjacent** button  twice to retrieve two layers of elements surrounding the failed elements.



9. In the **tetramesh** panel select the ***Tetra remesh*** subpanel.

10. Select the displayed elements and **remesh** them.



11. Check the **tet collapse** again and note the number has dropped.
12. **Delete** the mesh.

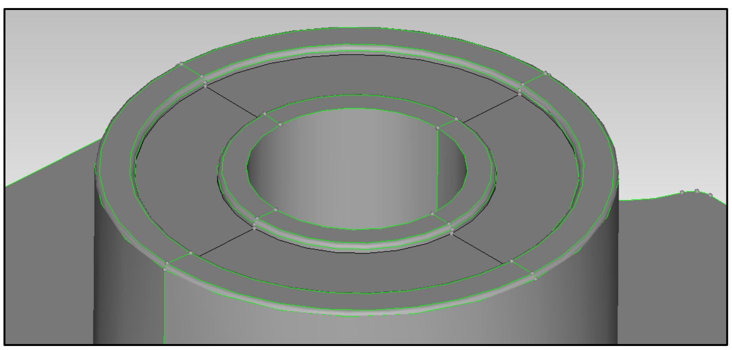
    From menu bar, click **Mesh > Delete > *Elements.***
    Click ***elems >> displayed*** and click **delete entity**.

Step 8: Defining Mesh Patterns

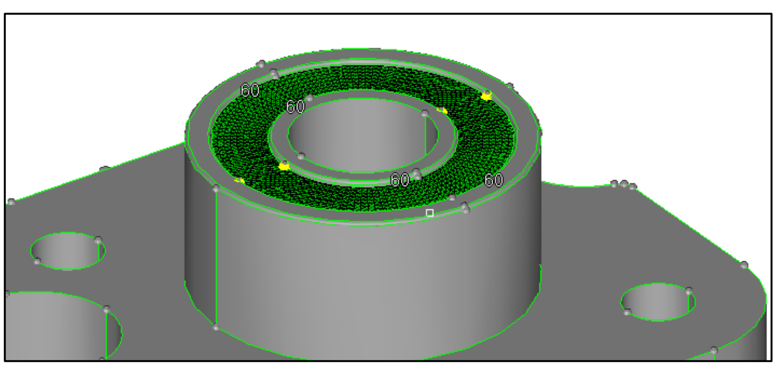
In instances where the user needs to define a specific mesh pattern for surfaces or features, the volume tetra function can incorporate that pattern into the created tetra mesh.

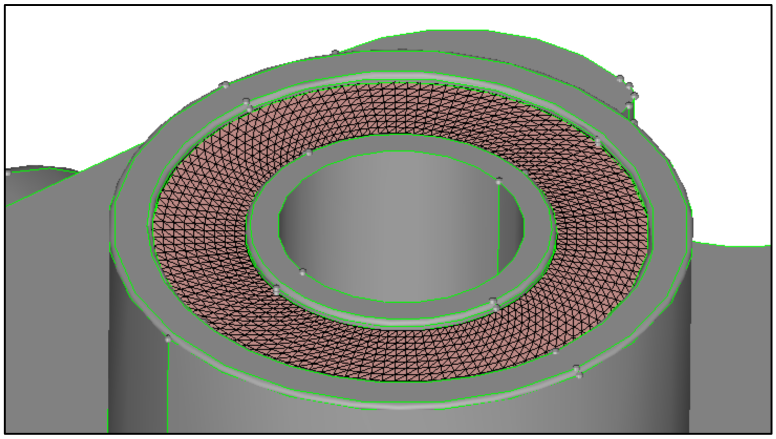
1. **Mesh** the flat ring area with an element size of `1` and type of R-Tria.

   From menu bar, click **Mesh > Create > *2D AutoMesh***.



Set all edges to 60 elements. The resulting mesh pattern should look similar to the one below.





2. Create a new volume tetra mesh, this time selecting the **match existing mesh** option. Make sure to set the tetra element size back to `4`.

Note the Tetra Mesh has incorporated the defined mesh pattern.