Solid Map Meshing

In solid meshing, the ability to be meshed is referred to as mappability. Mappability is directional and can be likened to putting a surface mesh on one face of the solid, then extending that mesh along a vector through the solid volume.

The ability to control the mesh pattern of a solid mesh by placing a shell mesh on the surface has been available in previous versions. To achieve this, though, the user had to put the shell mesh on the surface before the solid map function was performed and had to do it for each desired face. The solid map panel automatically places a shell mesh on the source faces and then enters into a mesh adjustment panel similar to the one in the automesh panel:

This allows for control over the mesh density and style using tools that work the same as in the automesh panel.
Exercise 6a - 3D Solid Meshing with Hexas and Pentas

This exercise will demonstrate a method for splitting a solid and then use the solid map function to create Hexa/Penta Solid elements. It is important to note that this is simply one way of splitting this solid. As with any solid geometry there are often many ways of obtaining a fully mappable solid and while some are better than others, there is rarely a “right” way of doing it. Experience is the key with this function; so experiment with different techniques for solid splitting and observe the results you get.

Step 1: Import the model
1. Locate and import the file 6a-STAND-SOLID-MAP.prt

   This model is in a ProE .prt format.

Step 2: Defeaturing

Small fillets make the geometry substantially more difficult to split into mappable regions and result in a far more complex solid mesh. In many cases, these fillets are for manufacturing purposes and can be eliminated from the geometry.

1. **Defeature** all of the small internal surface fillets.

   **HINT:** Setting the search values to be **0.5>5.5** will select all of the fillets needed. This range will also result in the fillet shown in the picture below to be selected (fillet in the red circle area). This fillet must be removed (**mouse >Right Click**) from the selected fillets, before to proceed with “remove”, as defeating it would cause a sharp point that would act as a severe stress concentration area.
Step 3: The first split

There is no set method for splitting a solid and often the first cut is the hardest, as picking the location to begin can be confusing. Often it is easiest to find areas that look to be close to being mappable. Many regions are only one cut away from becoming mappable and these frequently are the best place to start. In the case of this model, these areas are the flat "feet". One cut will separate them from the rest of the solid and they will immediately become mappable.

1. Turn on **Mappable** visualization:

2. In the **solid edit** panel select the **trim with plane/surf** subpanel.

3. Select the solid and using the **N1 N2 N3** option, define a plane on the flat area as shown in the picture below.

4. Trim the solid and the result will be a mappable region on the “foot”.

**HyperWorks 13.0**

Proprietary Information of Altair Engineering, Inc.
5. Repeat this trim on the other side of the part.

**Step 4: Splitting out further mappable regions.**

With the first splits done, now we can look to what is remaining and determine how these regions can be made mappable. It is often easiest to visualize this by masking the areas already split into mappable regions, thus showing only the areas of the part that remain to be split.

1. **Mask** the two mappable solids that were created in Step 3.

2. From the **trim with plane/surf** subpanel, select the solid and define a plane on the flat recessed area.

3. **Trim** the solid.
4. Repeat on the other side.

The solid is now in three distinct regions; the two outer regions being mappable and the central region which is still un-mappable.

5. **Mask** the two newly created mappable solids.

**Step 5: The last trims.**

With the thin slice of the part remaining, it is now important to determine which feature(s) is (are) causing this solid to remain non-mappable.

Remember that the rules state that a mappable solid can have multiple source faces but only ONE destination face.

The surfaces that make up the face of the pocket that was on the complete solid (highlighted in white in the picture below) occur on both sides of the remaining solid. This means there are multiple surfaces on both sides of the solid and thus violate the mappable rules.

In instances where specific regions prevent a solid from mapping, trimming those regions out can result in a mappable solid.
1. Select the **trim with lines** subpanel.

2. From the **with sweep lines** column, pick the remaining solid.

3. For the **sweep lines**, select the outline of one of the surface shown in white above.

4. As this model is aligned with the Global Axis, select the **sweep to: option to be by a vector >> z-axis**, select the **sweep all** option, and then **trim** the solid.
5. Repeat this process for the other side.
   This will result in a fully mappable solid.

6. **Save** the model.

### Step 6: Solid Meshing

With a fully mappable solid, the solid meshing tools can now be used to create the 3D elements.

1. Enter the **solid map** panel and select the **multi solids** subpanel.

2. Set the options shown below and mesh the solids.

---

**HyperWorks 13.0**
The interactive multi solid meshing will allow for 2D mesh customization prior to the creation of the 3D mesh. HyperMesh will show the order in which each solid is to be meshed and will indicate the direction in which the mesh will be extruded.

Additionally the panel now allows the user to alter the 2D mesh that will be used as the pattern to extrude the 3D elements. A panel similar to that used in interactive shell meshing is opened and the pattern mesh is displayed on the solids.

Using procedures identical to 2D meshing, edge densities can be adjusted, element sizes can be re-calculated, mesh styles can be changed and other meshing options can be altered. Clicking the mesh button will show the solid mesh but the mesh will not be finalized until the return button is clicked so further changes can be made.

3. Use the edge density, master face style and options sub panels to make changes to the mesh and see their outcome on the 3D mesh, proceed to mesh the solids. When happy with the 3D mesh, return from the function and save the part.
Exercise 6b- Tetra Meshing

Step 1: Load the model
1. Load the model 6b-VOLUME-TETRA-MESH.hm

Step 2: Attempt to TetraMesh the part
1. From the menu bar, enter the Tetra Mesh panel to create a 3D Tetramesh.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Connectors</th>
<th>Materials</th>
<th>Properties</th>
<th>Bin</th>
</tr>
</thead>
<tbody>
<tr>
<td>Create</td>
<td>Line Mesh</td>
<td>2D AutoMesh</td>
<td>F12</td>
<td></td>
</tr>
<tr>
<td>Edit</td>
<td>MidSurf Mesh</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Assign</td>
<td>Tetra Mesh</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Delete</td>
<td>CFD Tetramesh</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

2. Select the Volume tetra sub panel.
3. Change the enclosed volume switch to surfs.
4. Attempt to select a surface on the model. (Note: You will not be able to.)

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.
   
   **Hints:** 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, enter the panel to create Tetramesh.

   ![TetraMeshing Panel](image)

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 40 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.

**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 109 elements.

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

HyperWorks 13.0

Proprietary Information of Altair Engineering, Inc.
6. In order to improve Tetrahedral Element quality, there are a couple of tools you can use. First let's use the following tool from the 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.

![Tetra Mesh Optimization Window](image)

- Several criteria for optimization
- Mesh statistics
- Boundary shell mesh
- Constraints
- More options

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.

---

HyperWorks 13.0

Proprietary Information of Altair Engineering, Inc.
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.

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 40
- **Min tetra size**, enter 8
- Leave the other options with default values

![Constraints section](image)

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.

![Mesh optimization results](image)

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

---

**HyperWorks 13.0**

Proprietary Information of Altair Engineering, Inc.
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 below:

![Optimize tetras by](image)

In general, when you use this too, if the results of the fixes are acceptable, we click on **Close** to exit from **Tetra Mesh Optimization** utility. If the results are unacceptable, we click **Reject** to revert the mesh to its pre-foxed state. Please **Note**: You can only undo one fix operation this way – you cannot “back up” more than one step.

19. For this example, we are going to select **Reject** so we can use some of the other tools.

**Step 7 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 109 elements.

5. **Save** the failed elements by selecting **save failed**.
6. Select the switch **assign plot** 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 ![unmask](image) 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.

**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 10 and type of R-Tria.

   ![Mesh panel](image)

   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 40.

3. Note the Tetra Mesh has incorporated the defined mesh pattern.