
Tutorial 3.

Zonal Hybrid Mesh Generation

Introduction

There are many cases in which you may use hexahedral cells to mesh one part of your geometry, but complexities in another part of the geometry require that it be meshed with tetrahedral cells. In such cases, you can use the usual preprocessor to create the mixed triangular surface mesh and the hexahedral volume mesh, and then use TGrid to complete the hybrid mesh generation.

This tutorial demonstrates the mesh generation procedure for a hybrid mesh, starting from a hexahedral volume mesh and a triangular boundary mesh. This tutorial demonstrates how to do the following:

1. Read the mesh files and display the boundary mesh.
2. Merge the free nodes on the two pieces of the mesh (hexahedral volume mesh and triangular boundary mesh).
3. Create pyramids as a transition between the hexahedral and tetrahedral mesh using the Auto Mesh procedure.
4. Build prisms from the bottom of the tetrahedral region.
5. Check the quality of the entire volume mesh.
6. Merge the multiple cell zones into a single cell zone.
7. Create a non-conformal interface as a transition between the hexahedral and tetrahedral mesh using the Auto Mesh procedure.

Prerequisites

This tutorial assumes that you have little experience with TGrid, but that you are familiar with the graphical user interface.

Preparation

1. Download `zonal-hybrid.zip` from the [FLUENT User Services Center](#) to your working directory. This file can be found from the Documentation link on the TGrid product page.

OR

Copy `zonal-hybrid.zip` from the TGrid documentation CD to your working directory.

- For UNIX systems, insert the CD into your CD-ROM drive and go to the following directory:

```
cdrom/tgrid5.0/help/tutfiles/
```

where, *cdrom* must be replaced by the name of your CD-ROM drive.

- For Windows systems, insert the CD into your CD-ROM drive and go to the following folder:

```
cdrom:\tgrid5.0\help\tutfiles
```

where, *cdrom* must be replaced by the name of your CD-ROM drive (e.g., E).

2. Unzip `zonal-hybrid.zip`.

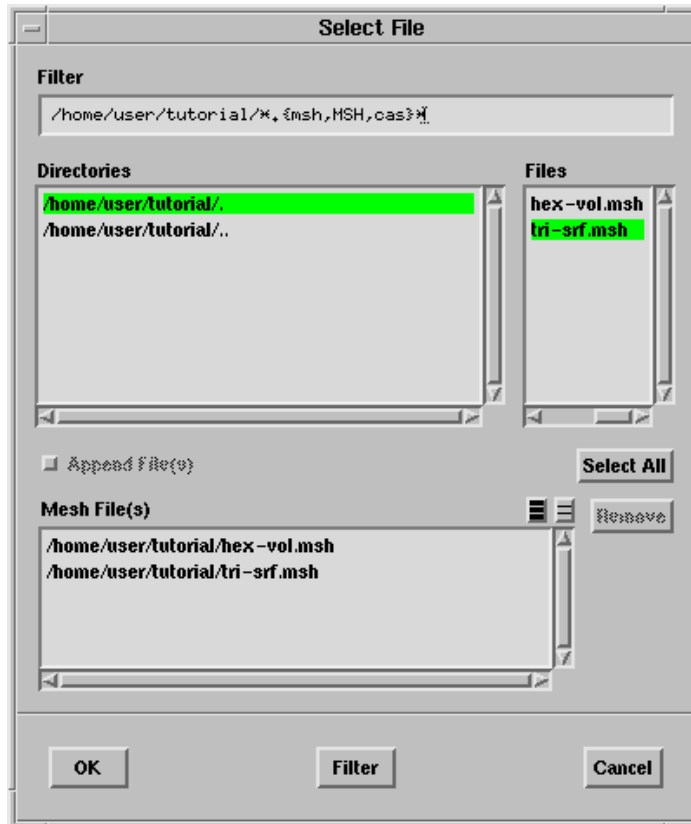
The files, hex-vol.msh and tri-srf.msh can be found in the zonal-hybrid folder created on unzipping the file.

3. Start the 3D (3d) version of TGrid.

Step 1: Read and Display the Mesh

1. Read the two mesh files.

File → Read → Mesh...



- (a) Select `hex-vol.msh` in the Files list.

The file will be added to the list of Mesh File(s) in the Select File dialog box.

- (b) Select `tri-srf.msh` in the Files list.

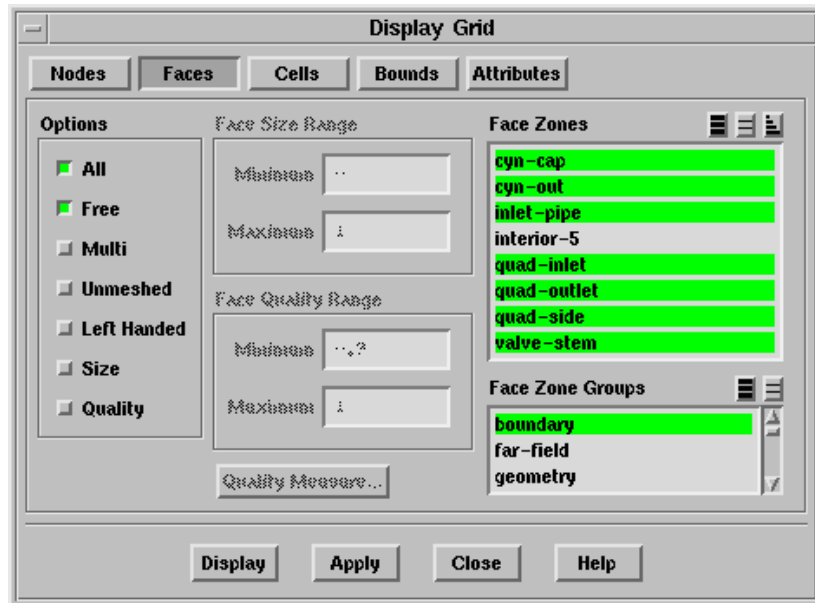
This file will also be added to the Mesh File(s) list.

- (c) Click OK.

TGrid will read both files and append them, but you will need to merge the shared nodes (i.e., the boundary nodes located along the circle where the triangular surface mesh and the quadrilateral surface mesh meet) so that the two meshes can be treated as a single unit.

2. Display the boundary mesh (Figure 3.1).

Display → Grid...



- (a) Select **boundary** in the Face Zone Groups selection list to select all the boundary zones in the Face Zones selection list.
- (b) Make sure **Free** is enabled (in addition to the default, **All**) in the Options group box.

This option allows you to see the nodes shared by the triangular and quadrilateral surface meshes. The nodes are free because, though both surface meshes have nodes at the same location, the two sets of nodes are not aware of one another. You will merge these nodes so that the two meshes can be treated as a unit.

- (c) Click the **Attributes** tab and disable **Filled** in the Options group box.
- (d) Click **Display**.
- (e) Zoom in to focus on the free nodes (Figure 3.2).

In Figure 3.2, the triangular faces that use the free nodes on the boundary between the tri and quad face zones are colored.

- (f) Close the Display Grid panel.

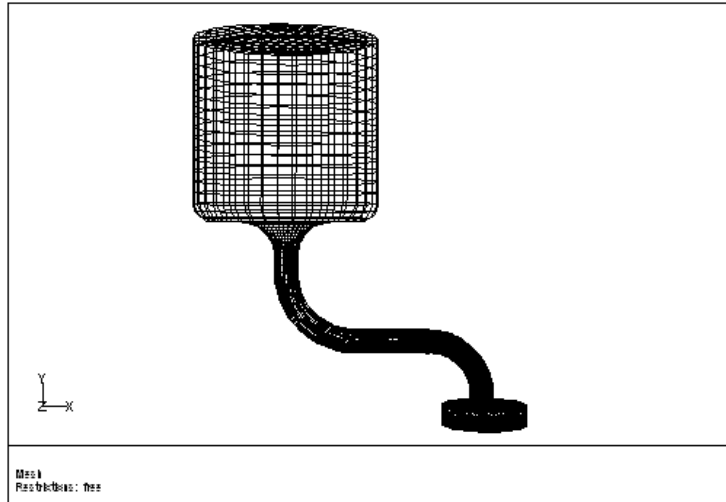


Figure 3.1: Boundary Mesh for the Valve Port

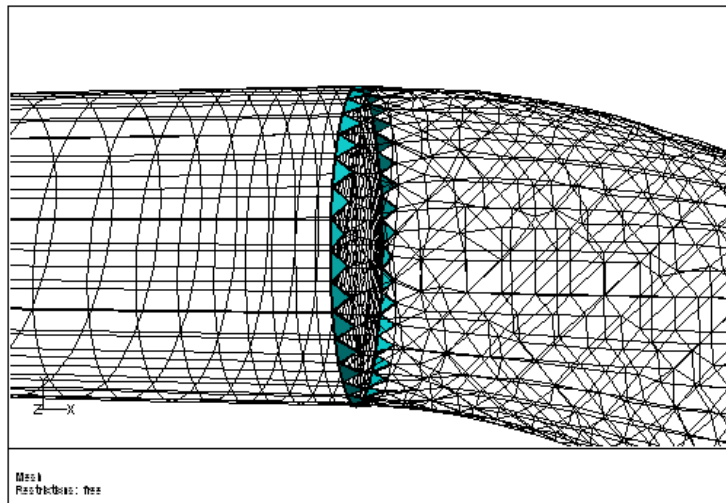
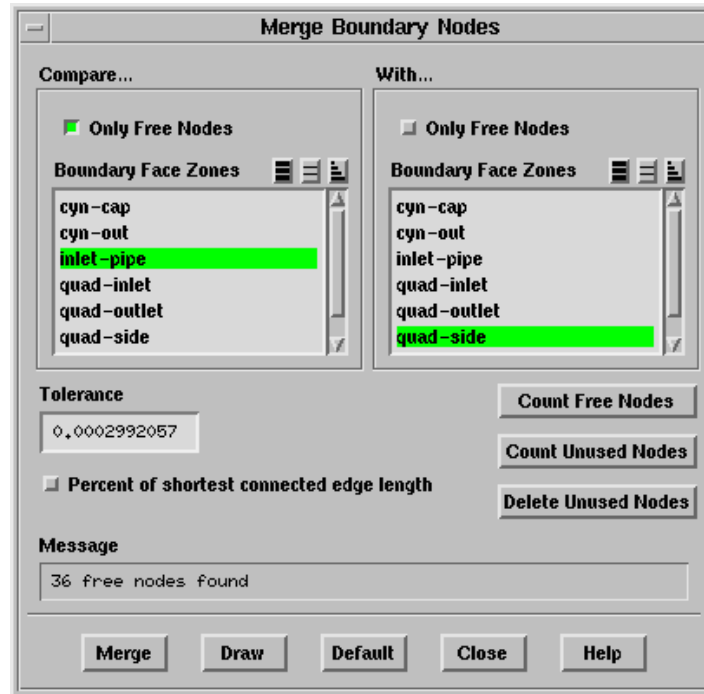


Figure 3.2: Free Nodes at the Intersection of the Tri and Quad Boundary Meshes

Step 2: Merge the Free Nodes on the Tri/Quad Border

In this step, you will merge the free nodes on the border between the triangular and quadrilateral face zones.

Boundary → Merge Nodes...



1. Select only inlet-pipe in the Boundary Face Zones selection list in the Compare... group box.

This is the triangular face zone that connects to the quadrilateral face zone for the side of the hexahedral region.

2. Disable Only Free Nodes and select only quad-side in the Boundary Face Zones selection list in the With... group box.

This is the external face zone of the hexahedral mesh that connects to the triangular face zone of the inlet pipe.

Note: *Disabling Only Free Nodes allows you to compare the free nodes on inlet-pipe (the triangular face zone) with all the nodes on quad-side (the quadrilateral face zone). This is necessary because the nodes in question are not free on the quadrilateral face zone. They are used by the side of the hexahedral region (quad-side) as well as the cap on the hexahedral region (quad-outlet). The nodes on the triangular face zone are free because each is used by only one face.*

After you merge the free nodes, the nodes of the triangular face will be connected to quad-outlet and quad-side.

3. Click Count Free Nodes.

TGrid will report the number of free nodes in the Message box.

4. Click Merge to merge the free nodes.

When the number of merged nodes is reported, not all of the free nodes were merged. This implies that some of the nodes differ from their counterparts by a distance greater than the specified Tolerance. Increase the Tolerance by a factor of 10 and try the merge operation again.

5. Enter 0.002992057 for Tolerance.

6. Click Merge.

The remaining nodes should now be merged.

7. Click Count Free Nodes again to ensure that all the free nodes have been merged.

8. Close the Merge Boundary Nodes panel.

9. Save the mesh file.

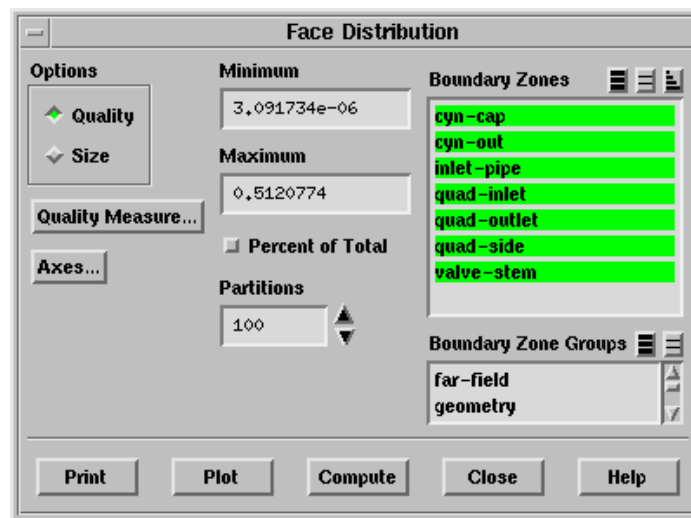
File → Write → Mesh...

(a) Enter hex-tri-merged.msh for Mesh File.

(b) Click OK to save the mesh.

Step 3: Check the Skewness Distribution of the Boundary Mesh

Display → Plot → Face Distribution...



1. Select all the zones in the **Boundary Zones** selection list.
2. Click **Compute**.
3. Click **Plot** (Figure 3.3).

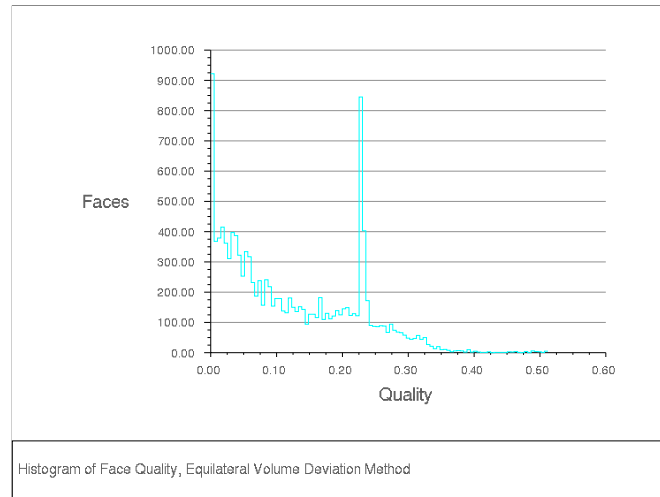


Figure 3.3: Boundary Mesh Skewness Distribution

You can change the Minimum and Maximum values to display the number of faces between two specific skewness values. It is a good practice to display the upper end of the skewness range (e.g., between 0.8 and 1.0). As a rule of thumb, the maximum boundary face skewness should be below 0.75.

For details on methods for improving the face skewness, see Tutorial 2.

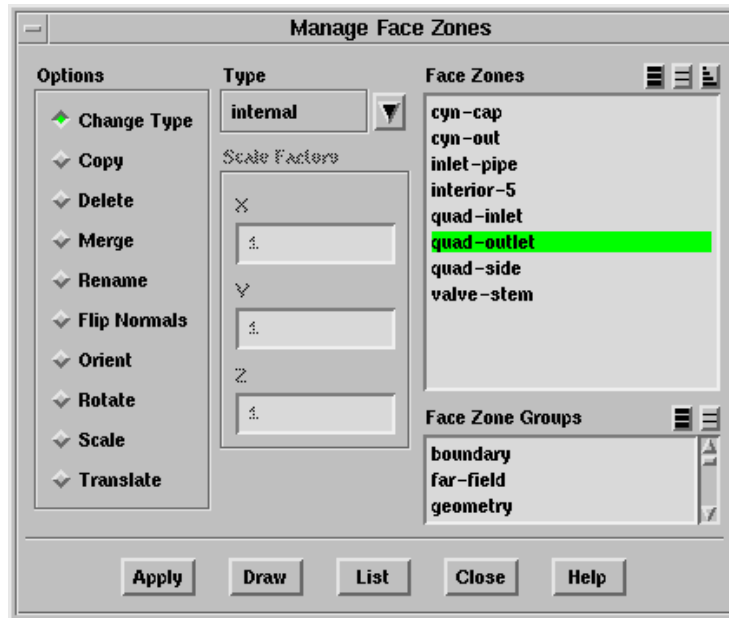
Step 4: Generate the Tetrahedral Mesh Using Pyramids to Transition Between the Hexahedral and Tetrahedral Mesh

In this step, you will use the Auto Mesh procedure in TGrid and use pyramids to transition between the quadrilateral and triangular boundary mesh.

1. Change the boundary type of **quad-outlet**.

When the surface mesh and the hexahedral mesh were created in the preprocessor, quad-outlet was given the type wall because there were cells on only one side of the surface. When you generate the tetrahedral mesh with pyramids on the other side, this boundary will simply be an interior boundary between fluid cells.

Boundary → Manage...



- (a) Select `quad-outlet` in the Face Zones selection list and click List.

The current zone type and other information will be reported in the console.

- (b) Retain the selection of Change Type in the Options list and select `internal` in the Type drop-down list.



It is recommended that you select `internal` instead of `interior` for the boundary type. If you clear the mesh in TGrid, all interior zones will be removed, but the internal zones will be retained. When you read the completed mesh into the solver, the internal zones will automatically be converted to interior type.

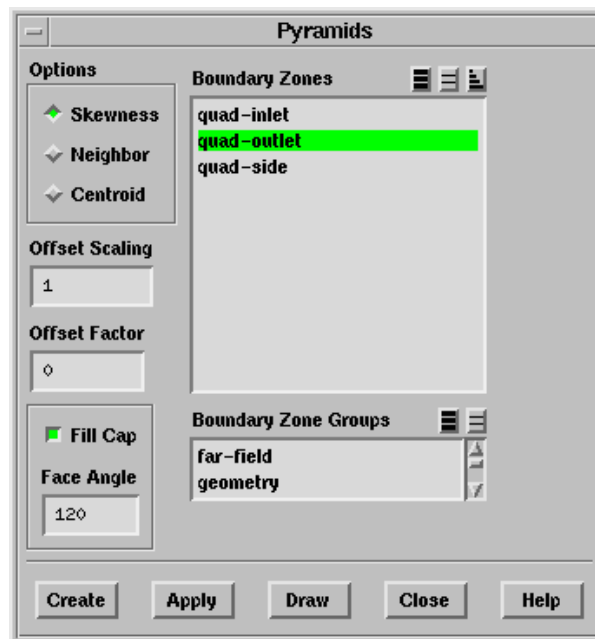
- (c) Click Apply and close the Manage Face Zones panel.

2. Set the meshing parameters.

Mesh → Auto Mesh...

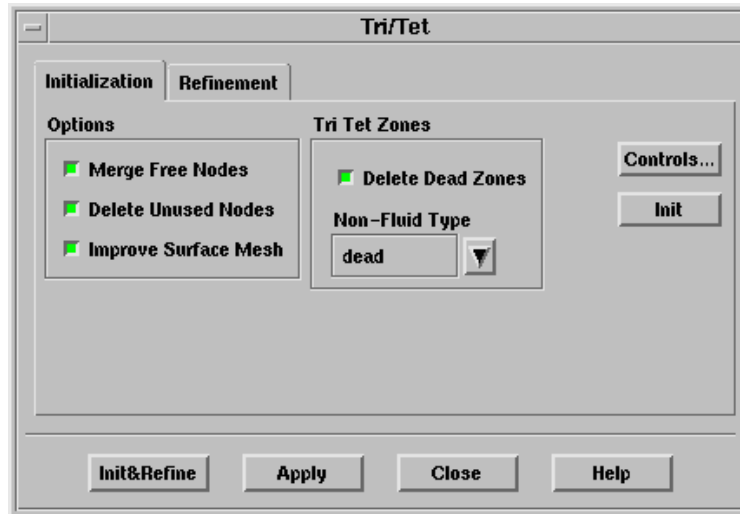


- (a) Retain the selection of Pyramids in the Quad Tet Transition list and click the Set... button to open the Pyramids panel.

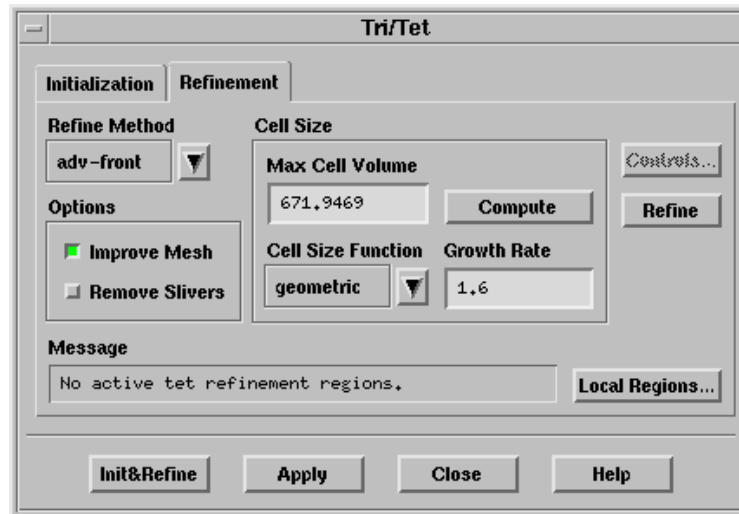


- i. Select quad-outlet in the Boundary Zones selection list.

- ii. Retain the selection of Skewness in the Options list.
 - iii. Enable Fill Cap and click Apply.
 - iv. Close the Pyramids panel.
- (b) Retain the selection of Tri/Tet in the Volume Fill list and click the Set... button to open the Tri/Tet panel.



- i. Enable Delete Dead Zones in the Tri Tet Zones group box in the Initialization tab.



- ii. Retain the default settings in the Refinement tab.
 - iii. Click Apply and close the Tri/Tet panel.
- (c) Click Apply in the Auto Mesh panel.

- (d) Preserve the existing hexahedral mesh.

```
> /mesh/tritet/preserve-cell-zone <Enter>
()
Cell Zones(1) [()] fluid* <Enter>
Cell Zones(2) [()] <Enter>
```

- (e) Click **Mesh** in the Auto Mesh panel.

The maximum and average skewness values reported at the end of the meshing are approximately 0.816 and 0.343, respectively.

- (f) Close the Auto Mesh panel.

3. Display the pyramid cap.

Display → Grid...

- (a) Select **quad-outlet** and **quad-outlet-pyramid-cap-#** in the Face Zones selection list in the Faces tab.
- (b) Disable **Free** in the Options group box.
- (c) Click the **Attributes** tab and enable **Filled** and **Lights** in the Options group box.
- (d) Click **Display** and manipulate the display to obtain the view shown in Figure 3.4.

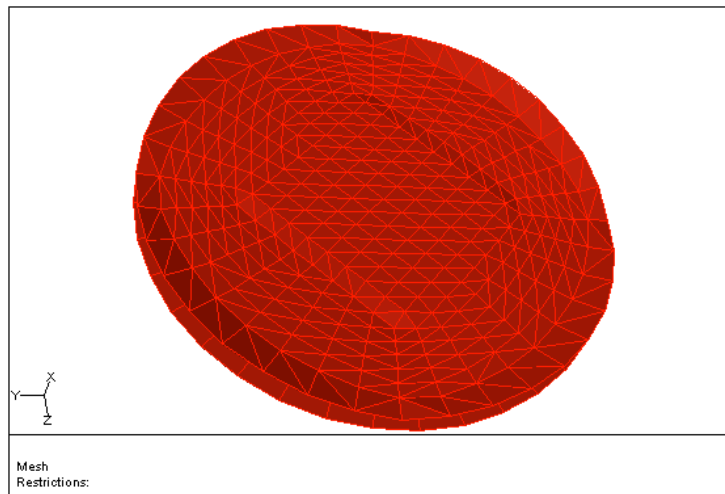


Figure 3.4: Filled Pyramid Cap

If the pyramids were created without using the Fill Cap option, the pyramid cap would look like what is shown in Figure 3.5.

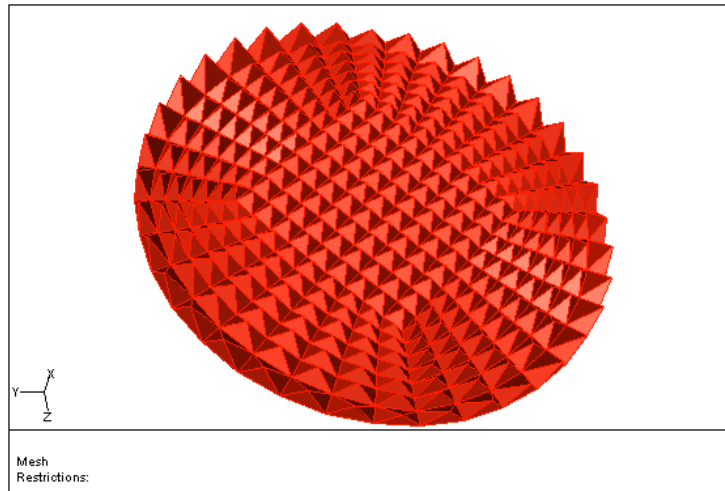


Figure 3.5: Pyramid Cap Without the Fill Cap Option

4. Examine the transition between the hexahedral and tetrahedral mesh.

Display → Grid...

- (a) Deselect all the previous selections in the Face Zones selection list in the Faces tab.
- (b) Click the Cells tab and select all the zones in the Cell Zones selection list.
- (c) Enable All in the Options group box.
- (d) Click the Attributes tab and enter 0.4 for Shrink Factor.
- (e) Click the Colors... button to open the Grid Colors panel.
 - i. Select Color by ID in the Options list.
 - ii. Close the Grid Colors panel.
- (f) Click Display and zoom in close to the boundary between the hexahedral and tetrahedral mesh (Figure 3.6).

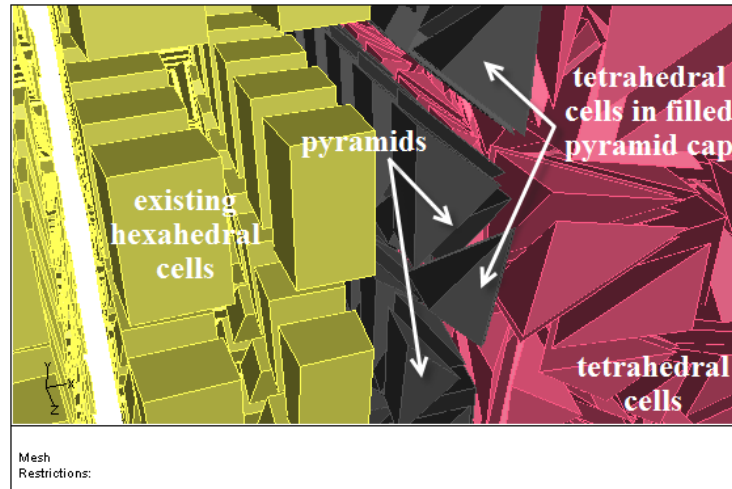
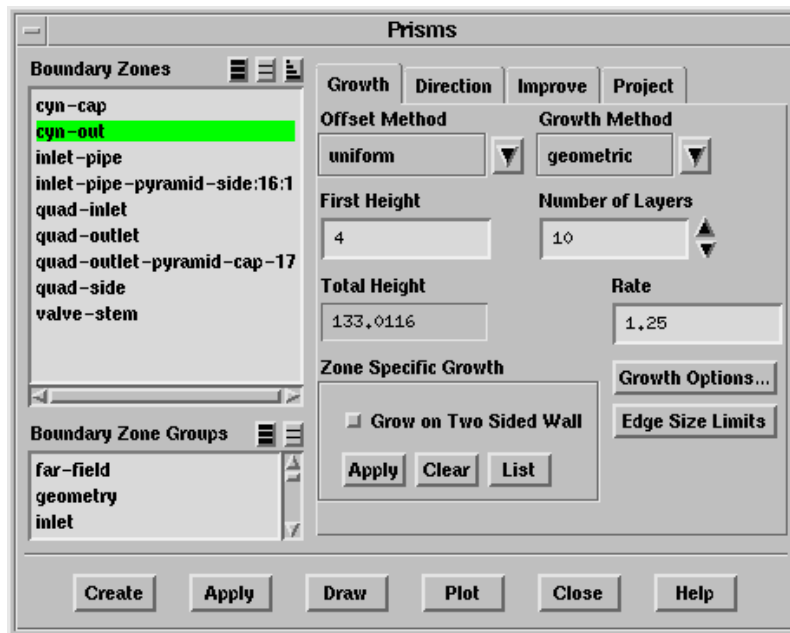


Figure 3.6: Pyramid Transition Between the Hexahedral and Tetrahedral Mesh

Step 5: Extend the Mesh Using Prisms

Mesh → Prisms...



1. Select cyn-out in the Boundary Zones selection list.

This is currently the bottom of the cylinder. You will extend the cylinder by building prisms from this triangular boundary. You can click Draw to display the zone. Make sure the Shrink Factor is set to 0 in the Attributes tab and the All option is disabled in the Cells tab of the Display Grid panel before clicking Draw.

2. Set the parameters controlling prism growth.

(a) Retain the selection of uniform in the Offset Method drop-down list and select geometric in the Growth Method drop-down list, respectively.

(b) Enter 4 for First Height and 1.25 for Rate, respectively.

This means that the first prism layer will have a height of 4, the second a height of 5 (4×1.25), and so on.

(c) Enter 10 for Number of Layers.

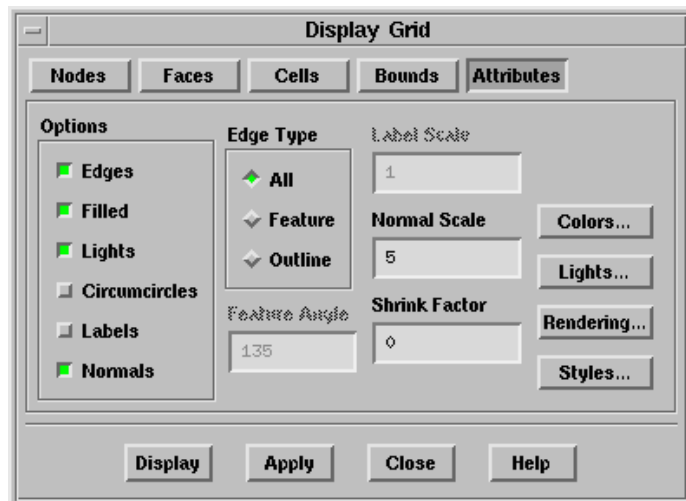
The Total Height added by the prisms is slightly more than 133.

3. Check that the face normals are pointing the right way.

The normal direction for the face zone determines which side of the zone the prisms are built on. To extend the domain down from the current cylinder bottom, you need to ensure that the normals on the cyn-out zone are pointing down.

(a) Enable the display of normals.

Display → Grid...



i. Click the Attributes tab and enable Normals in the Options group box.

ii. Enter 5 for Normal Scale.

Larger normals are easier to see in the grid display.

(b) Click the Faces tab and deselect the previous selections in the Face Zones selection list.

(c) Select only cyn-cap and cyn-out in the Face Zones selection list.

(d) Click Display, zoom out, and rotate the display to see the bottom of the cylinder (Figure 3.7).

In Figure 3.7, the normals are not pointing out from the bottom of the cylinder. Since they need to point out (i.e., down), you need to flip them.

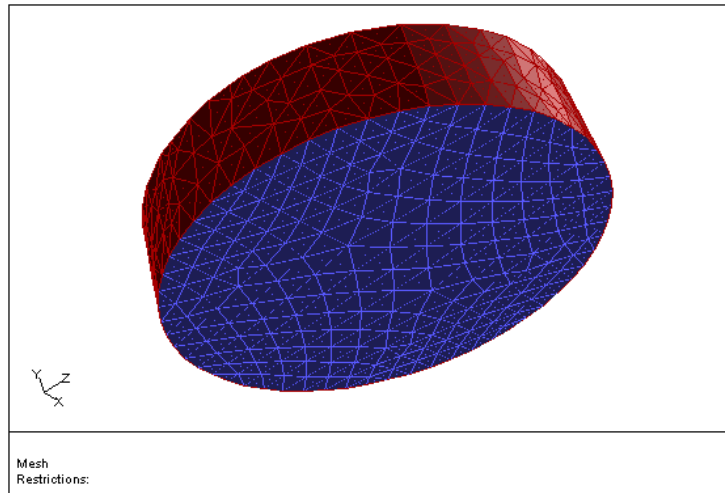


Figure 3.7: Cylinder Normals in Wrong Direction

- (e) Click the **Direction** tab in the **Prisms** panel (to access the direction parameters), and then click **Flip Normals**.
- (f) Click **Display** and close the **Display Grid** panel.

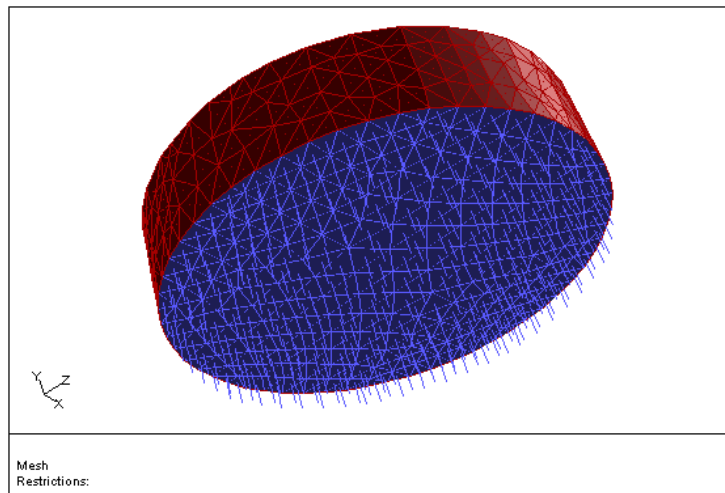
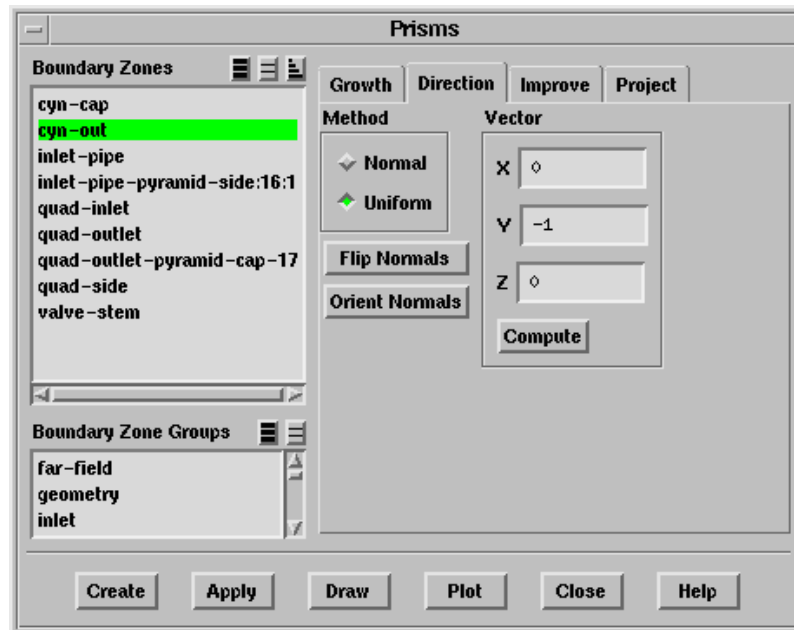


Figure 3.8: Cylinder with Normals in Correct Direction

In Figure 3.8, the normals are pointing in the correct direction. The prisms built will extend the cylinder below its current bottom.

- Specify the growth direction for the prisms.



- Select Uniform in the Method list.

The Uniform method is recommended when you are simply extruding to form a straight-sided prism region. You can use the default Normal method when growing prisms in more complicated regions.

- Click Compute in the Vector group box to update the normal direction vector for the cyn-out zone.

- Click Apply to save the prism parameters.
- Save an intermediate mesh file (temp.msh).

File → Write → Mesh...

It is a good practice to save the prism settings to a mesh file before generating prisms. If for any reason you are dissatisfied with the prisms, you can read the mesh file back in, modify the parameters, and try again.

- Click Create.

TGrid will create the layers of prisms, and summarize the new zones that have been created:

```
Prism Layer Summary:

3920 wedge cells created in new zone prism-cells-#.

9128 quadrilateral interior faces created in new zone interior-#.

392 boundary faces created in new zone prism-cap-#.

560 quadrilateral boundary faces created in new zone prism-side-#.

1521 interior nodes created in new zone node-#.

729 boundary nodes created in new zone boundary-node-#.
```

where, # denotes the respective zone IDs. The exact number may differ on different platforms.

The face and cell zones of interest are as follows:

interior-# contains the wedge prism cells.

prism-cap-# is the new bottom of the cylinder (with triangular faces).

prism-side-# contains the quadrilateral boundary faces on the outside of the cylinder.

8. Close the Prisms panel.
9. Display the new boundaries of the cylinder (Figure 3.9).

Display → Grid...

- (a) Select *cyn-cap*, *prism-cap-#*, and *prism-side-#* in the Face Zones selection list.
- (b) Click Display.

Make sure the Normals option has been disabled in the Attributes tab of the Display Grid panel.

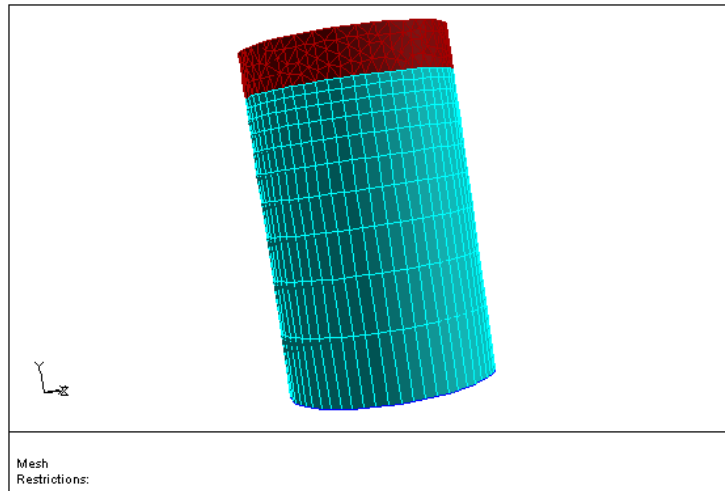


Figure 3.9: Cylinder Extended Using Prisms

10. Change the zone types for the zone you built the prisms from (**cyn-out**) and the new cap face (**prism-cap-#**).

*By default, the caps of the prism cells are wall zones. In this tutorial, the cap faces represent the outlet of the domain. Also, the zone you built the prisms from, **cyn-out**, is currently a wall zone. It should be an interior boundary between fluid cells.*

- (a) Change the zone type for **cyn-out**.

Boundary → Manage...

- i. Select **cyn-out** in the Face Zones list and click List.

The current zone type and other information will be reported in the console.

- ii. Select **internal** in the Type drop-down list.



It is recommended that you select **internal** instead of **interior** for the boundary type. If you clear the mesh in TGrid, all interior zones will be removed, but the internal zones will be retained. When you read the completed mesh into the solver, the internal zones will automatically be converted to interior type.

- iii. Click **Apply**.

- (b) Change the zone type for **prism-cap-#**.

- i. Select **prism-cap-#** in the Face Zones list.
- ii. Select **pressure-outlet** in the Type list.
- iii. Click **Apply**.

If required, you can change the zone names using the Rename option in the Manage Face Zones panel.

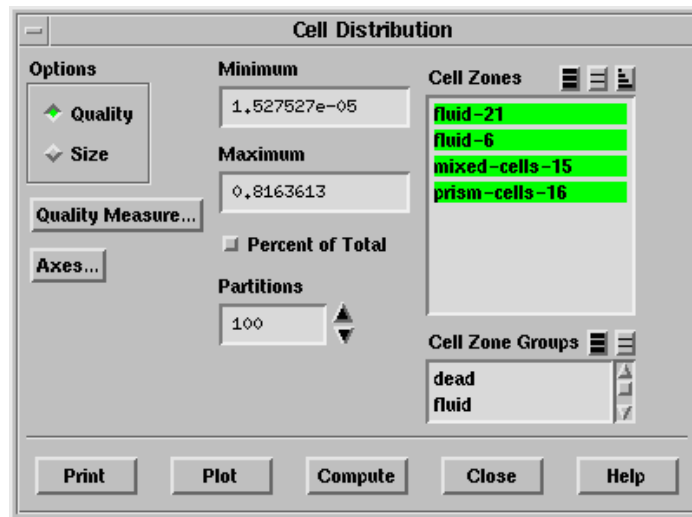
- (c) Close the Manage Face Zones panel.

Step 6: Check and Save the Volume Mesh

1. Check the skewness of the entire volume mesh.

- (a) Plot the cell skewness distribution (Figure 3.10).

Display → Plot → Cell Distribution...



- i. Select all the zones in the Cell Zones selection list.
- ii. Click Compute.
- iii. Click Plot.
- iv. Close the Cell Distribution panel.

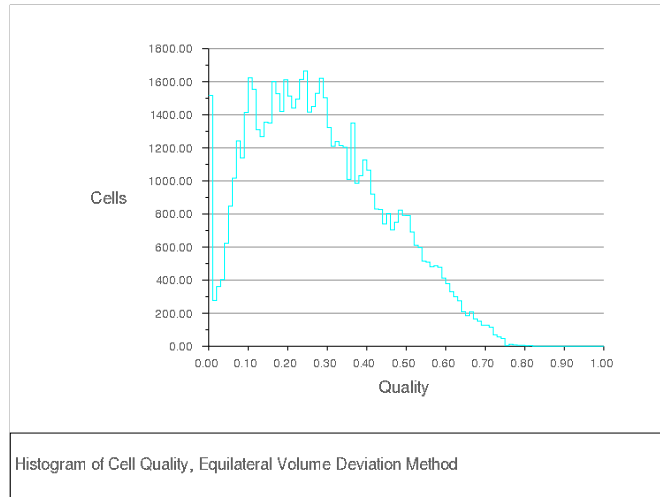
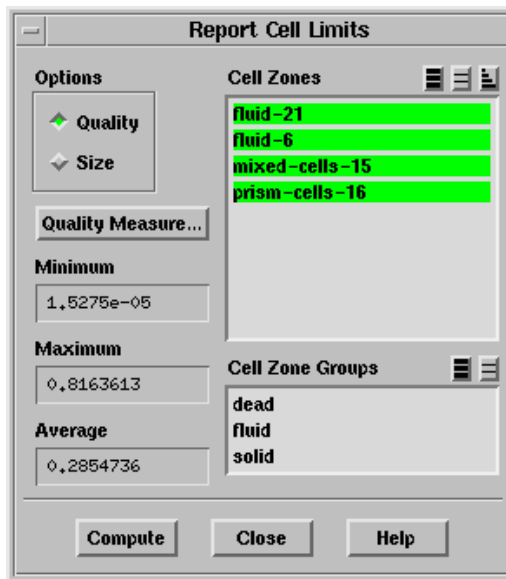


Figure 3.10: Cell Skewness Distribution

(b) Report the worst cell skewness.

Report → Cell Limits...

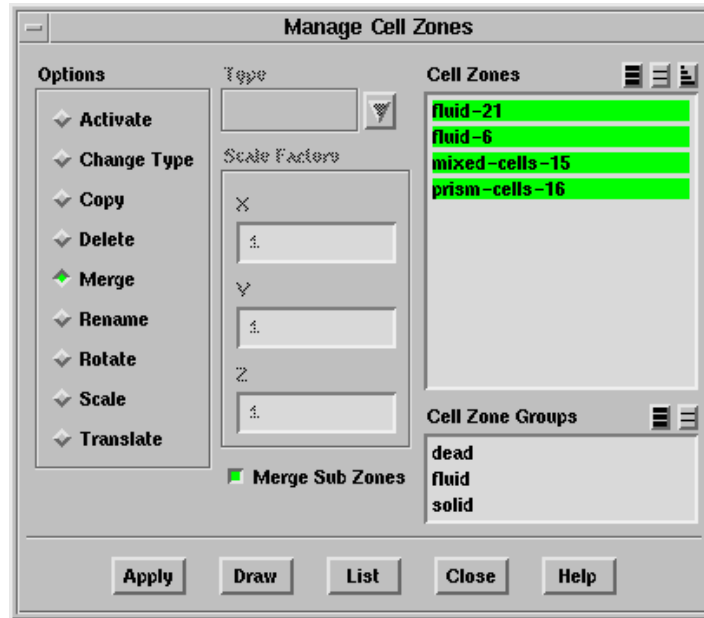


- i. Select all the zones in the Cell Zones selection list.
- ii. Click Compute.
- iii. Close the Report Cell Limits panel.

2. Merge the four cell zones.

In this problem, the hexahedral, pyramid, prism, and tetrahedral cells are all part of the same fluid region. Hence, there is no need to retain four separate cell zones. In this step, you will merge the cell zones before saving the final volume mesh.

Mesh → Manage...



- Select all the zones in the Cell Zones selection list.
- Select Merge in the Options list.
- Enable Merge Sub Zones.

When the Merge Sub Zones option is enabled, TGrid will merge the face zones associated with the cell zones, where appropriate.

- Click Apply.

You will see the four Cell Zones merge into a single zone in the Manage Cell Zones panel. The face zones that were merged together while merging the cell zones will be reported in the console.

- Close the Manage Cell Zones panel.

3. Check the volume mesh.

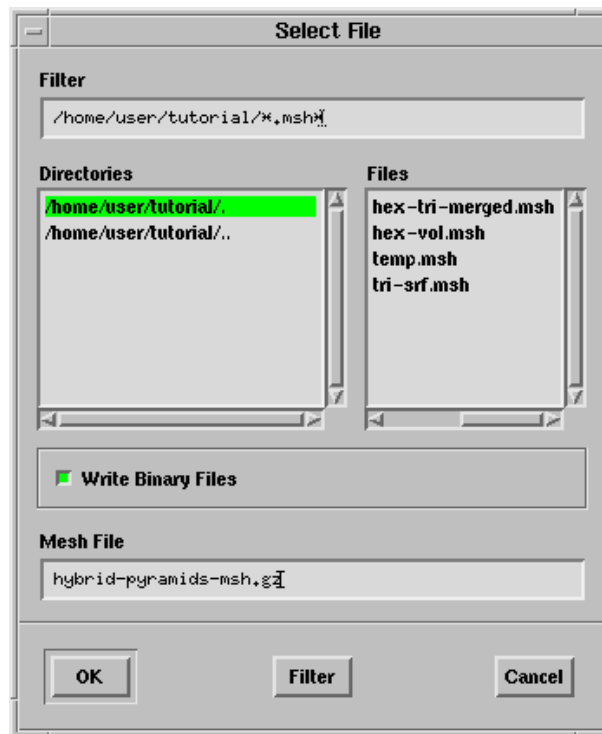
Before saving the mesh file, check it to ensure that it has no negative cell volumes or left-handed faces.

Mesh → Check

The printed results of the check show no problems, so the mesh can be used in the solver.

4. Save the mesh.

File → Write → Mesh...



- (a) Enter hybrid-pyramids.msh.gz for Mesh File.
- (b) Click OK to save the volume mesh.

Step 7: Generate the Tetrahedral Mesh Using a Non-Conformal Transition Between the Hexahedral and Tetrahedral Mesh

In this step, you will use the Auto Mesh procedure in TGrid and specify a non-conformal transition between the quadrilateral and triangular boundary mesh. The retriangulation methods available are as follows:

- *Quad-Split (recommended for low aspect ratio quads)*
- *Prism (recommended for high aspect ratio quads)*
- *Remesh (recommended for high aspect ratio quads)*

In this case, the quads are of a relatively low aspect ratio, hence, you will use the Quad-Split option. The use of alternative retriangulation options is demonstrated in Tutorial 4.

Note: *The steps in this section are similar to those described in previous sections, and hence are less explicit.*

1. Read the mesh file saved after merging the free nodes (`hex-tri-merged.msh`).

File → **Read** → Mesh...

2. Change the type of the quad-outlet zone to internal.

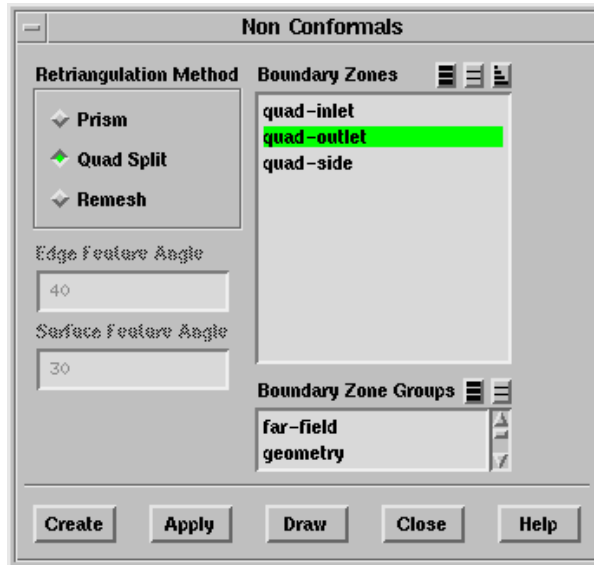
Boundary → Manage...

3. Set the meshing parameters.

Mesh → Auto Mesh...



- (a) Select Non Conformals in the Quad Tet Transition list and click the Set... button to open the Non Conformals panel.
 - i. Select quad-outlet in the Boundary Zones selection list.
 - ii. Select Quad Split in the Retriangulation Method list.



- iii. Click **Apply** and close the **Non Conformals** panel.
- (b) Retain the selection of **Tri/Tet** in the **Volume Fill** list and click the **Set...** button to open the **Tri/Tet** panel.
 - i. Enable **Delete Dead Zones** in the **Tri Tet Zones** group box in the **Initialization** tab.
 - ii. Retain the default settings in the **Refinement** tab and click **Apply**.
 - iii. Close the **Tri/Tet** panel.
- (c) Click **Apply** in the **Auto Mesh** panel.
- (d) Preserve the existing hexahedral mesh.

```

> /mesh/tritet/preserve-cell-zone <Enter>
()
Cell Zones(1) [()] fluid* <Enter>
Cell Zones(2) [()] <Enter>
    
```

- (e) Click **Mesh** in the **Auto Mesh** panel.

The maximum and average skewness values reported at the end of the meshing are approximately 0.886 and 0.344, respectively.
 - (f) Close the **Auto Mesh** panel.
4. Examine the transition between the hexhedral and tetrahedral mesh.

Display → **Grid...**

- (a) Make sure that any previous selections in the **Face Zones** selection list are deselected and select **quad-outlet-intf:#**.

- (b) Disable **Free** in the **Options** group box in the **Faces** tab.
- (c) Click the **Cells** tab and select all the zones in the **Cell Zones** selection list.
- (d) Enable **All** in the **Options** group box.
- (e) Click the **Attributes** tab and enable **Filled** and **Lights** in the **Options** group box.
- (f) Enter 0.4 for **Shrink Factor**.
- (g) Click the **Colors...** button to open the **Grid Colors** panel.
 - i. Select **Color by ID** in the **Options** list.
 - ii. Close the **Grid Colors** panel.
- (h) Click **Display** and zoom in close to the boundary between the hexahedral and tetrahedral mesh.

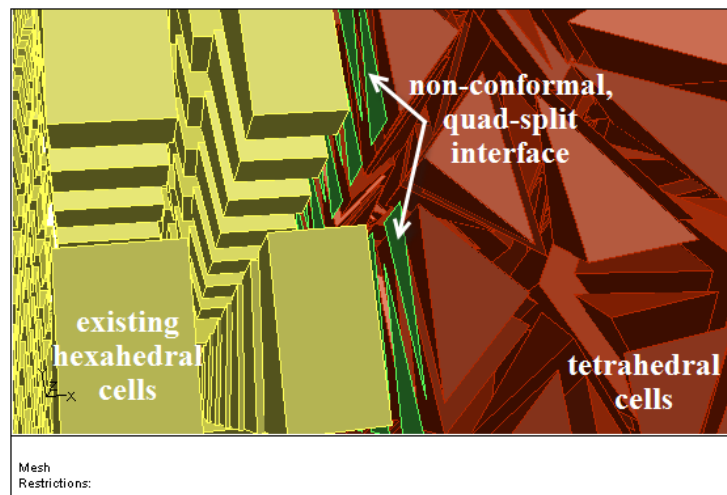


Figure 3.11: Non Conformal Transition Between Hexahedral and Tetrahedral Mesh

Extra: *If required, you may extend the mesh using prisms as described in **Step 5**. Change the type for the appropriate boundaries, as required.*

5. Check the skewness of the entire volume mesh.
 - (a) Plot the cell skewness distribution (Figure 3.12).

→ → Cell Distribution...

- (b) Report the worst cell skewness.

→ Cell Limits...

The worst cell skewness is approximately 0.886.

6. Merge the cell zones.

→ Manage...

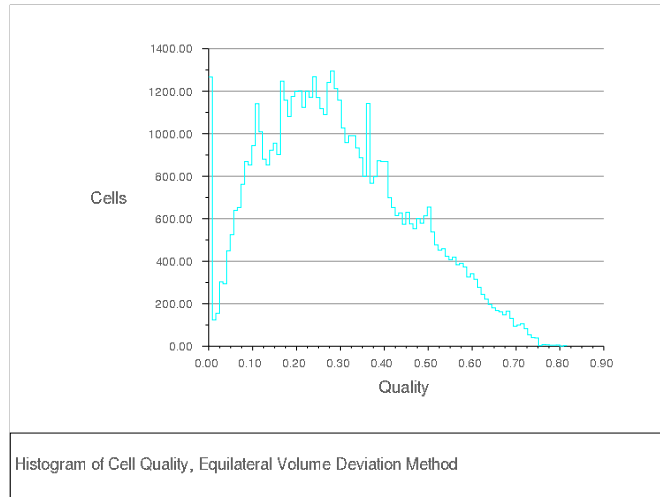


Figure 3.12: Cell Skewness Distribution

7. Check the volume mesh.

Mesh → Check

8. Save the mesh (hybrid-nonconformal.msh.gz).

File → **Write** → Mesh...

9. Exit TGrid.

File → Exit

Summary

This tutorial demonstrated the creation of a hybrid mesh starting from a hexahedral volume mesh and a triangular boundary mesh. The tutorial described the procedure to create the tetrahedral mesh with a transition layer of pyramid cells, while preserving the existing hexahedral mesh. It also described the extending of the mesh by building layers of prism cells from the bottom of the tetrahedral portion of the mesh. Finally you merged all the cell zones into a single fluid cell zone for convenience. The tutorial also described the procedure to create a non-conformal transition between the hexahedral and tetrahedral mesh.

