

### Introduction

Hexcore meshing is a hybrid meshing scheme which generates Cartesian cells inside the core of the domain and tetrahedral cells close to the boundaries. Hanging-node refinements on the Cartesian cells allow efficient cell size transition from the boundary to the interior of the domain. Thus, a hexcore mesh can significantly reduce the cell count compared with a fully tetrahedral mesh. The hexcore meshing scheme is applicable to all volumes but is useful for volumes with large internal regions and few internal boundaries. It is fully automated and compatible with prism (boundary layer) generation.

Hexcore meshes are more useful in applications with large open spaces. One such application from the automotive industry is explained here.

This tutorial demonstrates how to do the following:

1. Read and display the mesh.
2. Check and improve the skewness of the surface mesh.
3. Repair the boundary mesh by merging the nodes and splitting the edges of the geometry.
4. Generate the tetrahedral mesh.
5. Generate the hexcore mesh.
6. Examine the effect of buffer layers on the hexcore mesh.
7. Generate the hexcore mesh in conjunction with boundary layer (prism) meshing and local refinement.

### Prerequisites

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

### Preparation

1. Download `hexcore.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 `hexcore.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 `hexcore.zip`.

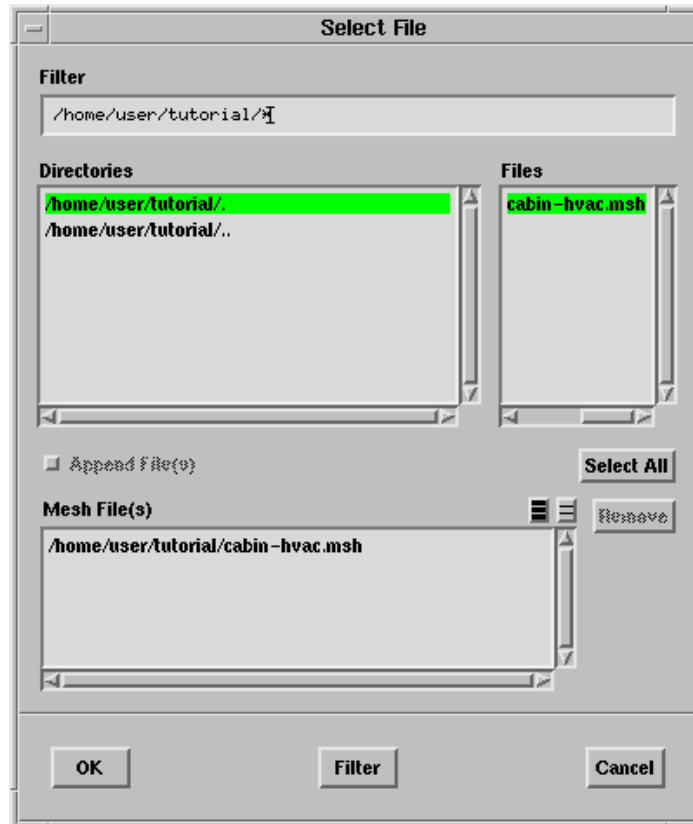
*The file, `cabin-hvac.msh` can be found in the `hexcore` folder created on unzipping the file.*

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

## Step 1: Read and Display the Mesh

1. Read the mesh.

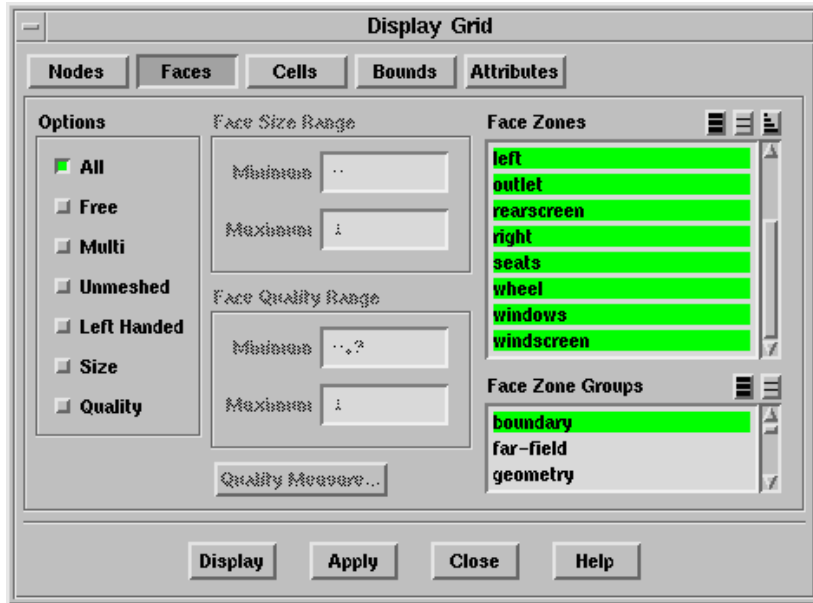
File → Read → Mesh...



- (a) Select `cabin-hvac.msh` in the Files list.
- (b) Click OK.

2. Display the mesh (Figure 5.1).

**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) Click **Display**.

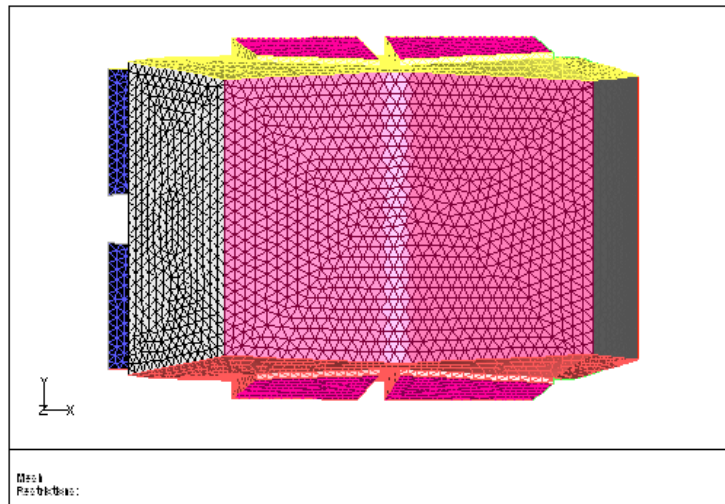
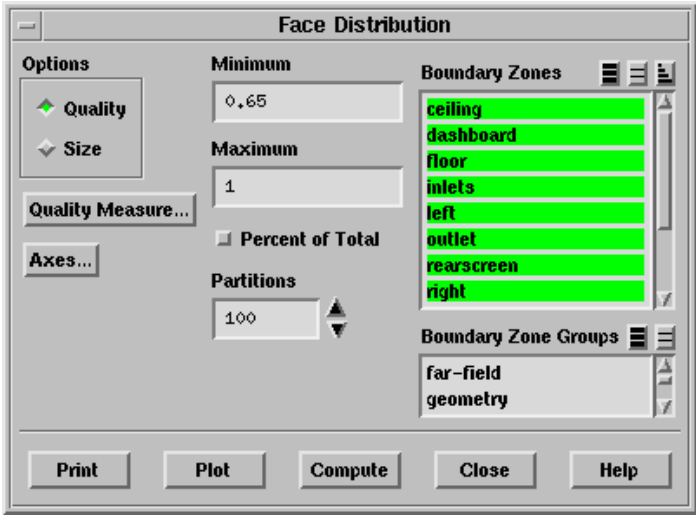
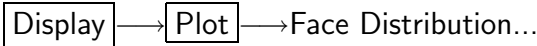


Figure 5.1: Grid Display

- (c) Close the Display Grid panel.

### Step 2: Check and Improve the Skewness of the Surface Mesh

- 1. Display the surfaces with higher skewness.
  - (a) Find the number of faces with a skewness greater than 0.65.



- i. Select all the zones in the Boundary Zones selection list.
- ii. Enter 0.65 for Minimum.
- iii. Click Plot (Figure 5.2).

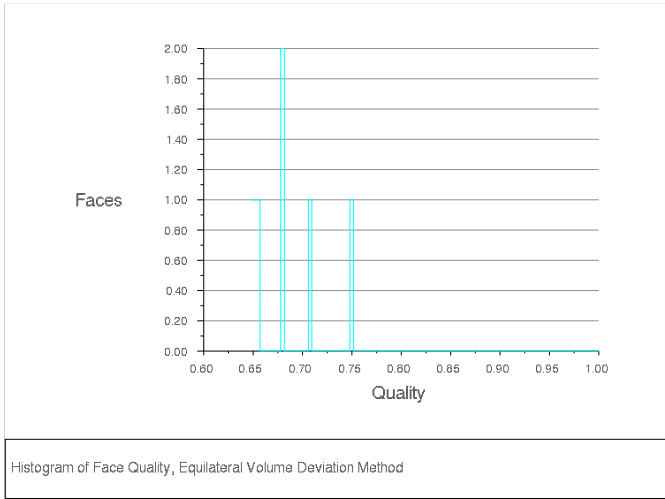
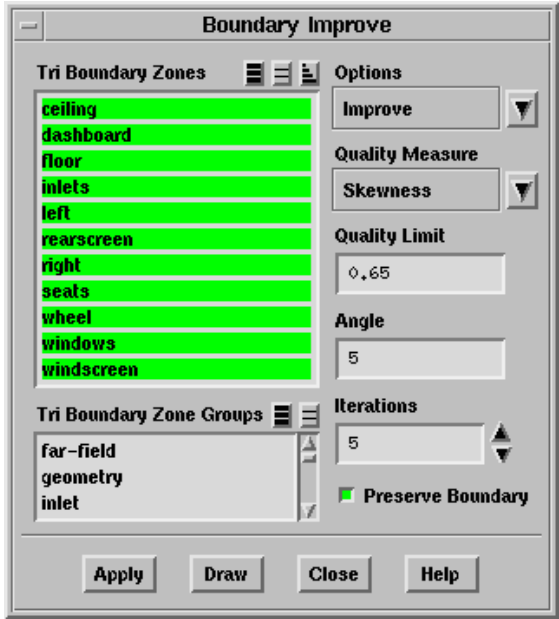


Figure 5.2: Skewness Distribution Above 0.65

- iv. Close the Face Distribution panel.

### Step 3: Repair the Boundary Mesh

1. Modify the faces having skewness greater than 0.65.



- (a) Select all the zones in the Tri Boundary Zones selection list.
- (b) Retain the selection of Skewness in the Quality Measure drop-down list and enter 0.65 for Quality Limit.
- (c) Enter 5 for Angle.
- (d) Retain the value of 5 for Iterations and click Apply.

*Specifying a lower value for Angle will reduce the modifications made to the boundary and better maintain the geometry.*

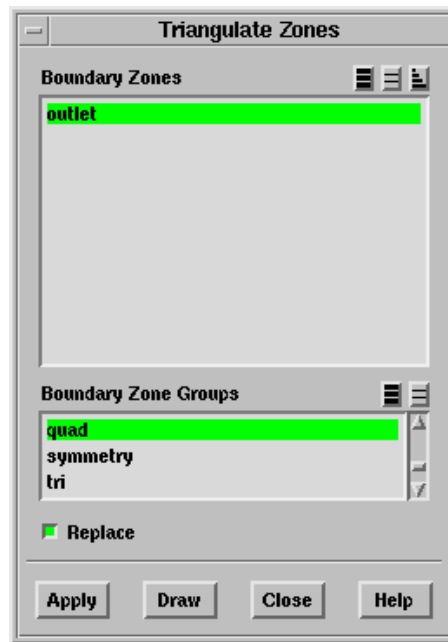
*TGrid will fix the 6 faces above the specified maximum quality (0.65) and report that the current maximum quality is approximately 0.649.*

2. Convert the quad face zones into tri face zones.



- (a) Select quad in the Boundary Zone Groups selection list.

*The outlet zone will be automatically selected in the Boundary Zones selection list.*



(b) Click Draw (Figure 5.3).

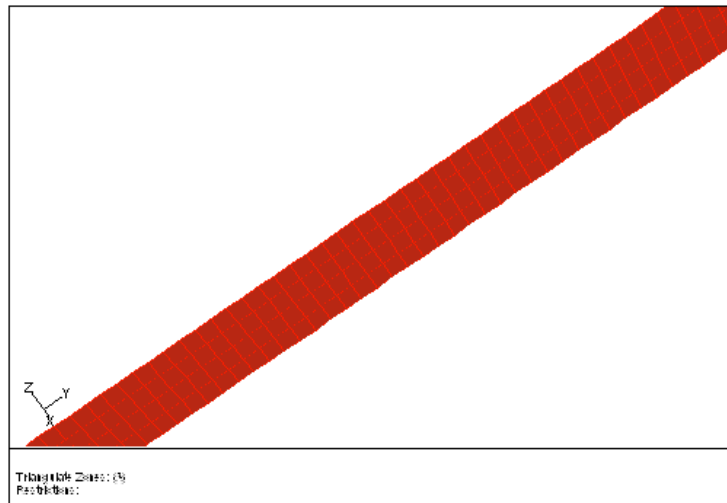


Figure 5.3: Quad Faces in the outlet Zone

(c) Retain the selection of Replace and click Apply.

*The quad faces will be split into triangles. TGrid will report that 1200 triangular faces were created in the new zone, outlet:#.*

(d) Close the Triangulate Zones panel.

**Extra:** *You can also create a non-conformal interface to allow quad faces during the hexcore mesh initialization. In this case, TGrid will automatically create the triangular mesh on all quad faces available in the domain. All the surfaces*

*having quad elements will be copied and remeshed with triangular cells. The free nodes of the triangular mesh will be merged with the original surface mesh.*

3. Check the quality of the surface mesh.
  - (a) Select all the zones in the **Tri Boundary Zones** selection list in the **Boundary Improve** panel.
  - (b) Retain the selection of **Skewness** in the **Quality Measure** drop-down list and **0.65** for **Quality Limit**.
  - (c) Click **Apply**.

*The maximum skewness reported is still less than 0.65, hence the mesh is acceptable.*
  - (d) Close the **Boundary Improve** panel.

4. Save the mesh file.

**File** → **Write** → Mesh...

### Step 4: Generate the Tetrahedral Mesh

1. Generate a tetrahedral mesh with the default settings.

**Mesh** → Tri/Tet...

- (a) Enable **Delete Dead Zones** in the **Tri Tet Zones** group box.
- (b) Retain the default settings for the remaining parameters and click **Init&Refine**.

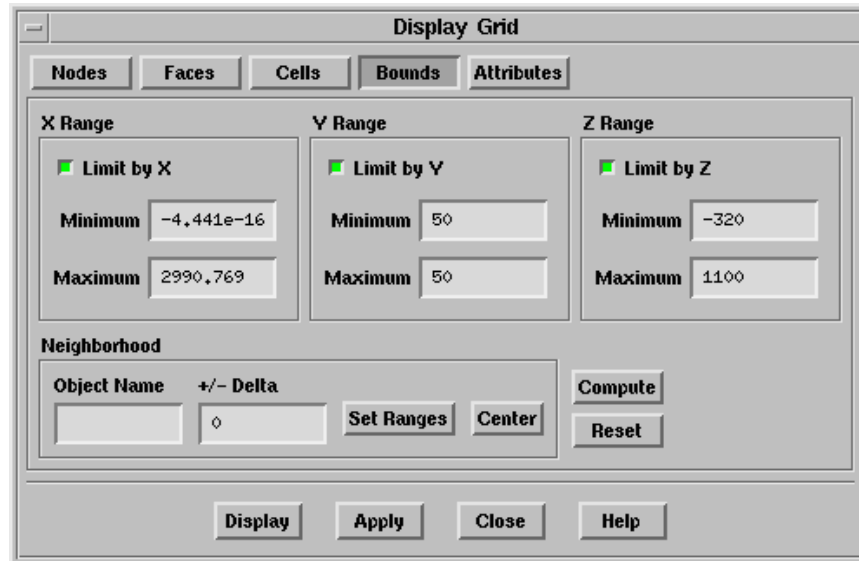
*The number of cells generated is approximately 204763 while the maximum skewness reported is around 0.856. The exact number may vary slightly on different platforms.*
- (c) Close the **Tri/Tet** panel.

2. Display a slide of cells at  $y = 50$ .

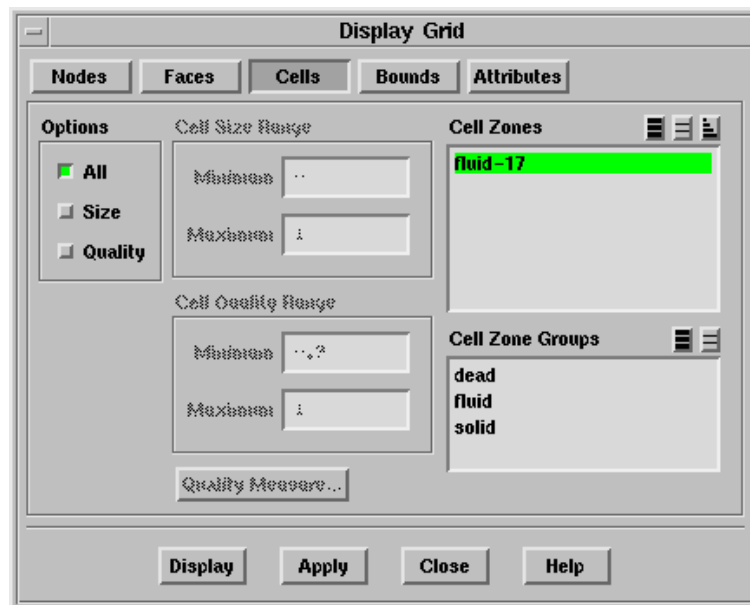
**Display** → Grid...

- (a) Retain the selection of all the boundary zones in the **Face Zones** selection list.
- (b) Click the **Bounds** tab and click **Compute**.





- (c) Enter 50 for Minimum and Maximum in the Y Range group box.  
 (d) Click the Cells tab and enable All in the Options group box.



- (e) Select the fluid zone (fluid-#) in the Cell Zones selection list and click Display.  
 (f) Display the top view (Figure 5.4).

**Display** → Views...

- i. Select top in the Views list.
  - ii. Click Apply and close the Views panel.
- (g) Similarly, display the cells for a value of  $y = -525$  (Figure 5.5).

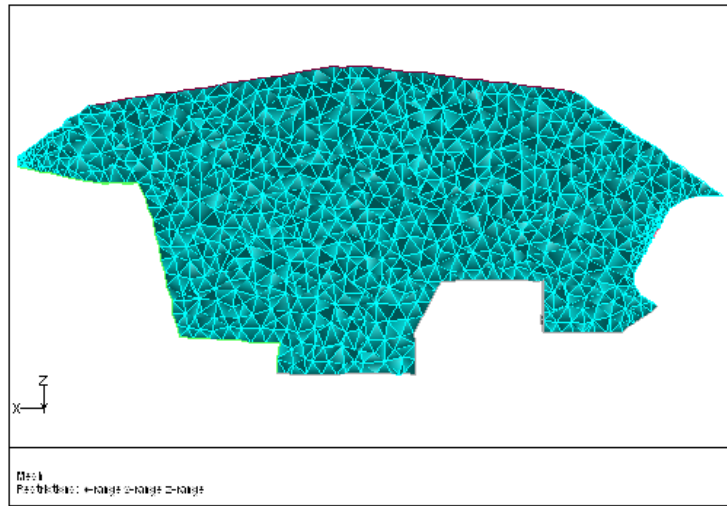


Figure 5.4: Slide of Cells at  $y = 50$  for the Tetrahedral Mesh

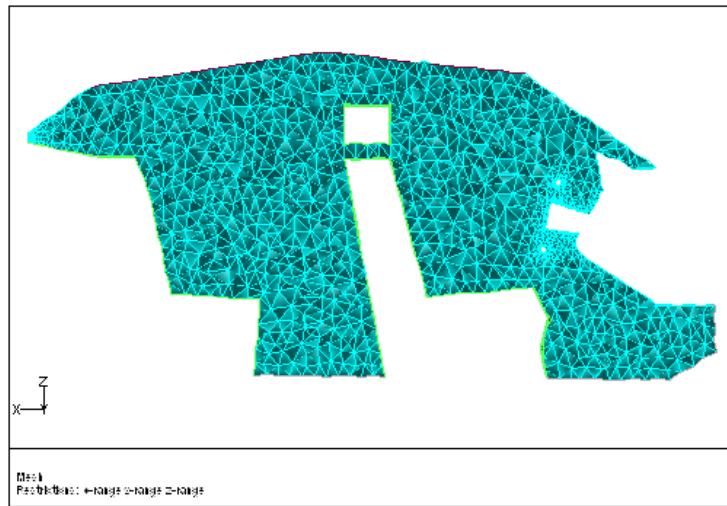


Figure 5.5: Slide of Cells at  $y = -525$  for the Tetrahedral Mesh

3. Clear the mesh.

**Mesh** → Clear

## Step 5: Generate the Hexcore Mesh

1. Generate the hexcore mesh using the default parameters.

**Mesh** → Hexcore...

- (a) Enable Delete Dead Zones in the Zones group box.
- (b) Retain the default settings for the remaining parameters and click Create.

**Note:** *The maximum skewness reported at the end of the refinement process is not the final value. Some slivers can be removed.*

- (c) Display the parent face.
  - i. Click Reset in the Bounds tab.
  - ii. Click the Faces tab and select parent-face-# in the Face Zones selection list.
  - iii. Click Display (Figure 5.6).

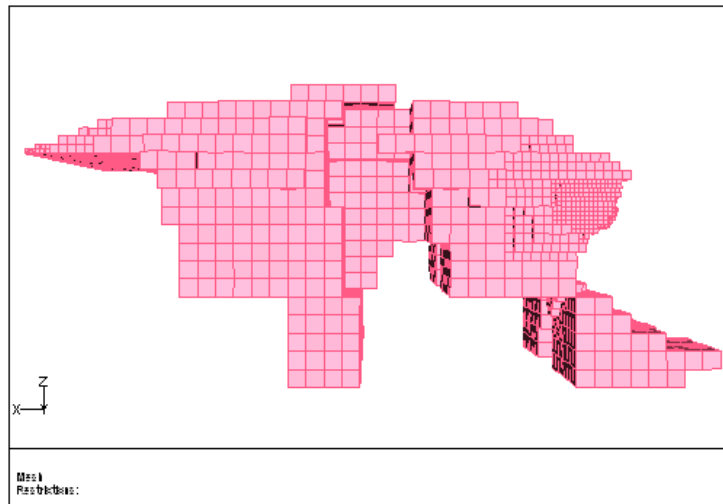


Figure 5.6: Parent Face for the Hexcore Mesh

- (d) Display a slide of cells at  $y = 50$  and  $y = -525$  (Figures 5.7 and 5.8).

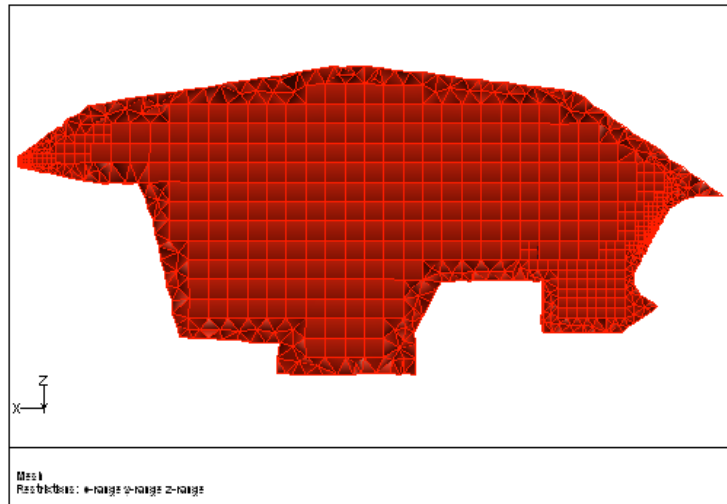


Figure 5.7: Slide of Cells at  $y = 50$  for the Hexcore Mesh With Buffer Layers = 1

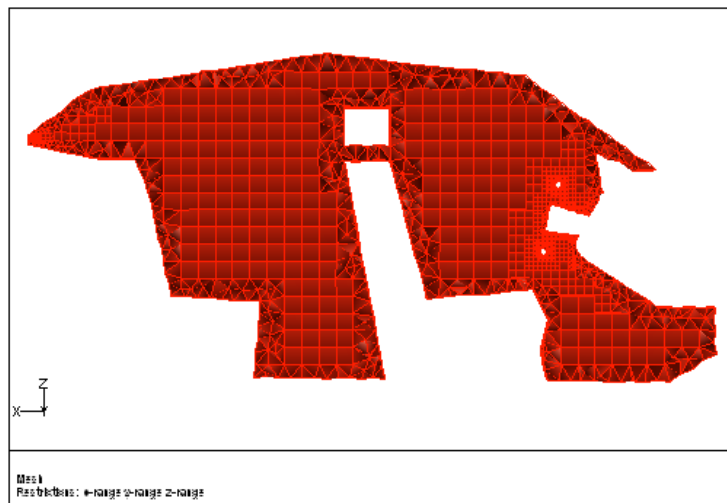


Figure 5.8: Slide of Cells at  $y = -525$  for the Hexcore Mesh With Buffer Layers = 1

2. Check the number of cells generated.

**Report** → Mesh Size...

- (a) Click Update.

*The number of cells is approximately 185810. The exact number may vary slightly on different platforms.*

- (b) Close the Report Mesh Size panel.

3. Check the maximum skewness.

**Report** → Cell Limits...

- (a) Select the fluid zone in the Cell Zones selection list and click Compute.

*The maximum skewness reported is around 0.884.*

- (b) Close the Report Cell Limits panel.

## Step 6: Examine the Effect of the Buffer Layers on the Hexcore Mesh

*Buffer Layers specifies the number of extra layers of hex cells of a particular size before subdivision. Increasing the number of buffer layers will significantly increase the number of cells.*

*The default setting for Buffer Layer is 1. You will now examine the effect of different buffer layer settings on the mesh.*

1. Clear the mesh.
2. Generate the hexcore mesh with Buffer Layers set to 0.

**Mesh** → Hexcore...

- (a) Click Compute to determine the value for Max Cell Length based on the current mesh.

- (b) Set Buffer Layers to 0 and click Apply.

*A Warning dialog box will appear, warning you that a buffer layer of zero may result in poor mesh quality.*

- (c) Click OK in the Warning dialog box.

- (d) Click Create.

3. Display the slide of cells at  $y = 50$  and  $y = -525$  (Figures 5.9 and 5.10).

4. Check the maximum skewness and the number of cells.

*For buffer layers = 0, the number of cells is 165035 and the maximum skewness is 0.922. The exact values may vary on different platforms.*

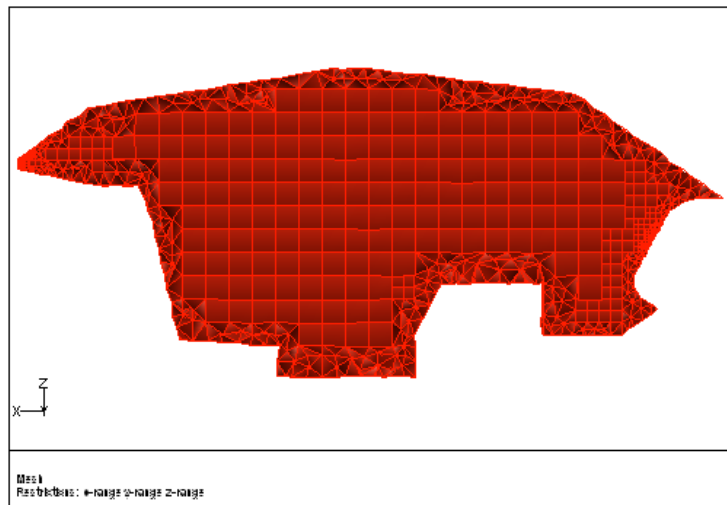


Figure 5.9: Slide of Cells at  $y = 50$  for the Hexcore Mesh With Buffer Layers = 0

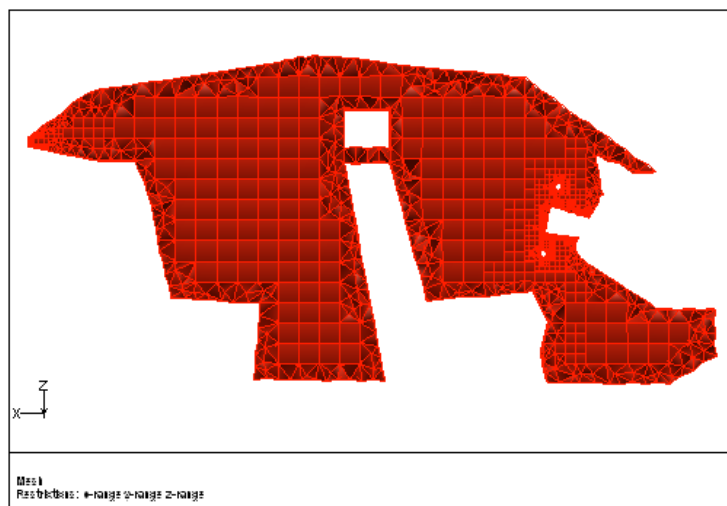


Figure 5.10: Slide of Cells at  $y = -525$  for the Hexcore Mesh With Buffer Layers = 0

5. Clear the mesh.
6. Set Buffer Layers to 2 and generate the hexcore mesh.
7. Display the slide of cells at  $y = 50$  and  $y = -525$  (Figures 5.11 and 5.12).

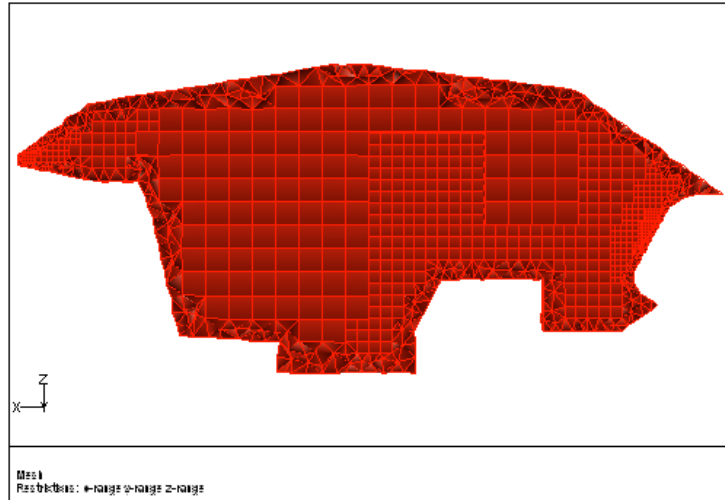


Figure 5.11: Slide of Cells at  $y = 50$  for the Hexcore Mesh With Buffer Layers = 2

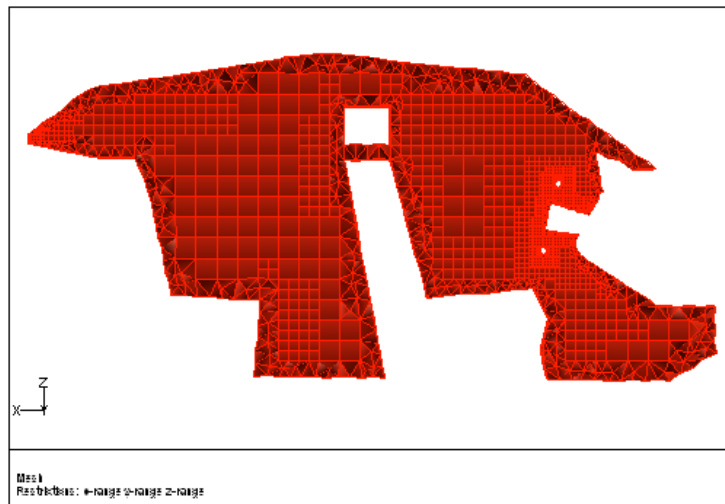


Figure 5.12: Slide of Cells at  $y = -525$  for the Hexcore Mesh With Buffer Layers = 2

8. Check the maximum skewness and the number of cells.

*For buffer layers = 2, the number of cells is 225134 and the maximum skewness is 0.894. The exact values may vary on different platforms.*

*Hence, the number of buffer layers has a strong impact on the number of cells.*

## Step 7: Automatically Generate the Hexcore Mesh with Prism Layers and a Local Refinement Region

1. Clear the mesh.
2. Verify that the normals on the surfaces you want to create prisms from are pointing in the right direction.

**Display** → Grid...

- (a) Click Reset in the Bounds section of the Display Grid panel.
- (b) Select ceiling, outlet:#, rearscreen, and windscreen in the Face Zones selection list.
- (c) Click the Attributes tab.
- (d) Enable Normals in the Options group box and enter 10 for Normal Scale.  
*Larger normals are easier to see in the grid display.*
- (e) Click Display.

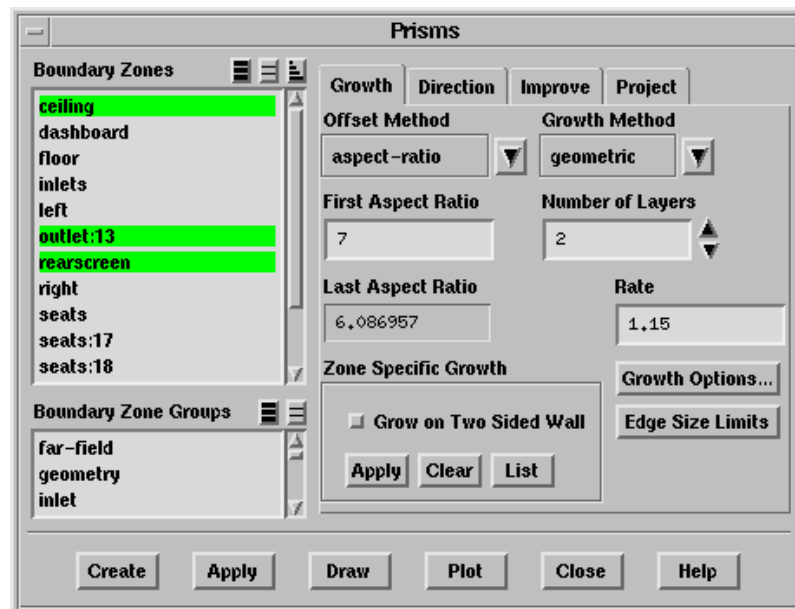
*If you zoom in, the normals on the selected zones point inward.*

3. Define the meshing parameters.

**Mesh** → Auto Mesh...

*The Prisms option is greyed out as no prism parameters have been set.*

- (a) Define the parameters for growing prism layers.
  - i. Click the Set... button in the Boundary Layer Mesh group box to open the Prisms panel.





- ii. Select **ceiling**, **outlet:#**, **rearscreen**, and **windscreen** in the **Boundary Zones** selection list.
- iii. Select **aspect-ratio** in the **Offset Method** drop-down list and enter **7** for **First Aspect Ratio**.
- iv. Select **geometric** in the **Growth Method** drop-down list and set the **Number of Layers** to **2**.
- v. Enter **1.15** for **Rate**.
- vi. Click **Apply** in the **Zone Specific Growth** group box.



It is necessary to apply the prism growth parameters on specific zones for TGrid to retain the growth parameters in memory. The **Prisms** option in the **Auto Mesh** panel will be visible only after applying zone-specific growth.

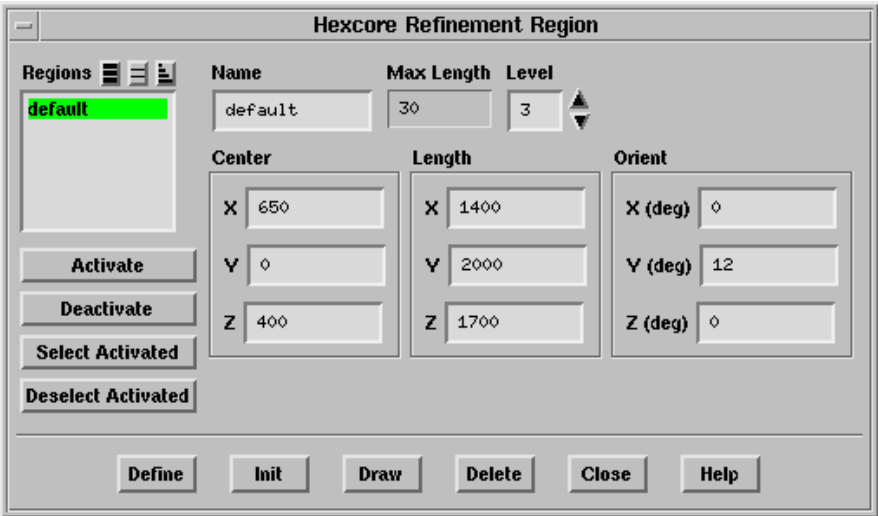
- vii. Close the **Prisms** panel.
  - viii. Enable **Prisms** in the **Boundary Layer Mesh** group box in the **Auto Mesh** panel.
- (b) Display all the surfaces except **right** using the **Display Grid** panel.  
*Make sure the display of normals is disabled in the Attributes tab.*
- (c) Define the parameters for hexcore meshing.
- i. Select **Hexcore** in the **Volume Fill** list in the **Auto Mesh** panel and click the **Set...** button to open the **Hexcore** panel.
  - ii. Compute the average length of the faces on the **ceiling** zone.
    - A. Zoom in to the **ceiling** zone in the graphics display.
    - B. Use the hot-key **Ctrl + N** to enable the selection of nodes.
    - C. Select two adjacent nodes using the right mouse button.
    - D. Use the hot-key **Ctrl + D** to calculate the distance between the selected nodes.  
*The distance between the nodes is computed to be approximately 75.*
  - iii. Enter **120** for **Max Cell Length**.  
*This is the maximum length for the hexahedral cells in the volume mesh.*
  - iv. Set **Buffer Layers** to **1**.



- v. Enable Delete Dead Zones in the Zones group box.
- vi. Click the Tet Controls... button to open the Tri/Tet panel.
  - A. Retain the settings in the Tri/Tet panel.
  - B. Close the Tri/Tet panel.
- vii. Define and visualize the refinement region.
 

*The hex cells will be refined in the local region defined using the Hexcore Refinement Region panel.*

  - A. Click the Local Regions... button to open the Hexcore Refinement Region panel.
  - B. Retain default for Name.
  - C. Set Level to 3.



D. Enter the following parameters:

	Center	Length	Orient
X	650	1400	0
Y	0	2000	12
Z	400	1700	0

E. Click Define.

F. Click Draw to visualize the refinement region (Figure 5.13).

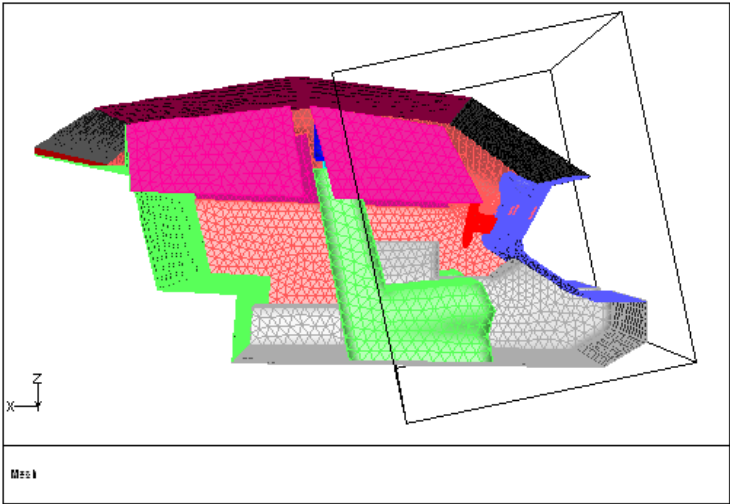


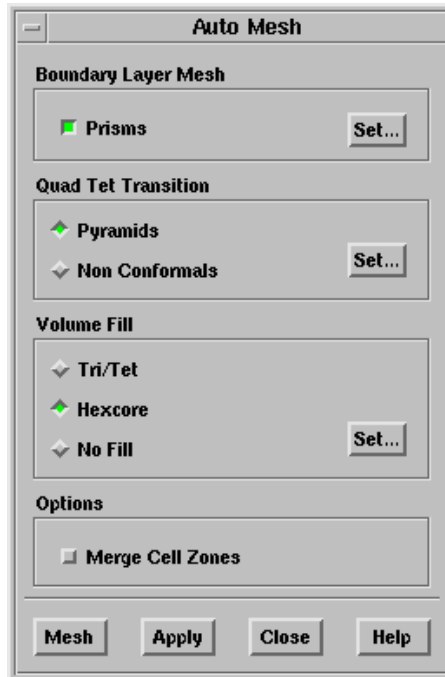
Figure 5.13: Refinement Region

G. Retain the selection of default in the Regions selection list and click Activate.

H. Close the Hexcore Refinement Region panel.

TGrid will report (in the Message field in the Hexcore panel) that there is one active hexcore refinement region.

- (d) Click Apply and close the Hexcore panel.



4. Click Mesh in the Auto Mesh panel to generate the mesh.
5. Close the Auto Mesh panel.
6. Separate the hex cells from the tet cells for better visualization.

```
> /mesh/separate/separate-cell-by-shape
Cell Zone [ ] fluid*

Moved 198071 cells from fluid-# to fluid-#:#
Moved 43952 triangular faces from interior-# to new zone interior-#:#.
Moved 164084 mixed faces from interior-# to new zone interior-#:#.
Moved 60 quadrilateral faces from dashboard to new zone dashboard:#.
Moved 300 quadrilateral faces from seats to new zone seats:#.
Moved 122 quadrilateral faces from left to new zone left:#.
Moved 122 quadrilateral faces from right to new zone right:#.

Interior face zone interior-#:# changed to boundary face zone.
6 nodes moved to existing boundary node zone boundary-node-#
```

Here, # denotes the respective zone IDs. The exact ID may vary on different platforms.

7. Examine the mesh by zone type.

**Mesh** → Manage...

- (a) Select all the zones in the Cell Zones selection list and click List.

*You can also draw the cell zones to identify the respective zones. There are 11728 prism cells (prism-cells-#), 198071 tetrahedral cells (fluid-#), and 57515 hexahedral cells (fluid-#:#).*

8. Check the cell count and the maximum skewness.

*The number of cells is 267314 and the maximum skewness is 0.884. The exact number may vary on different platforms.*

9. Display the slide of cells at  $y = 50$  and  $y = -525$  (Figures 5.14 and 5.15).

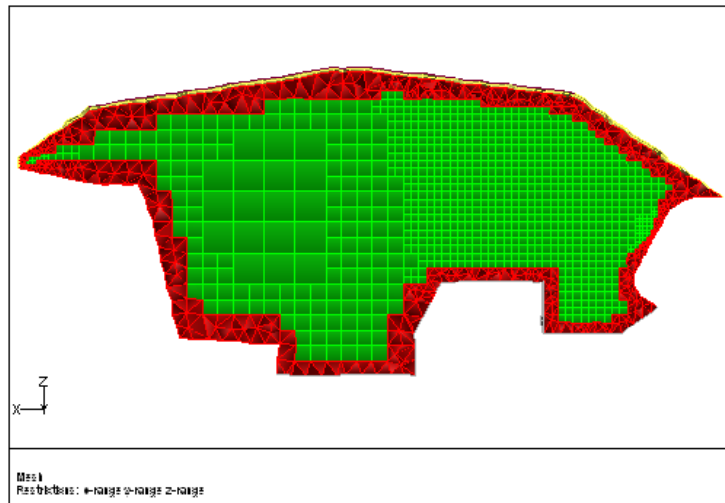


Figure 5.14: Slide of Cells at  $y = 50$

10. Check the size of the largest hex cell.

*You should get a value close to 120.*



Figure 5.15: Slide of Cells at  $y = -525$

11. Save the mesh file.

File → Write → Mesh...

12. Exit TGrid.

File → Exit

## Summary

This tutorial demonstrated the generation of a hexcore mesh using both the manual and the automatic mesh generation procedure. It also examined the effect of the buffer layers specified on the generated mesh. The use of local refinement regions was also demonstrated.