# Chapter 4.

# **Sample Session**

This chapter describes a sample session to illustrate the basic procedure for examining and creating the mesh in TGrid.

This sample session consists of the following steps:

- Section 4.1: Preparation
- Section 4.2: Reading the Boundary Mesh
- Section 4.3: Examining the Boundary Mesh
- Section 4.4: Generating the Interior Mesh
- Section 4.5: Displaying the Mesh
- Section 4.6: Reporting the Mesh Statistics
- Section 4.7: Checking the Mesh
- Section 4.8: Saving the Mesh and Exiting TGrid

In this session, you will generate a triangular volume mesh for a simple wedge-shaped geometry. The boundary mesh file is provided with **TGrid** on the CD.

### 4.1 Preparation

- 1. Copy wedge.zip from the TGrid documentation CD to your working directory.
  - For UNIX systems, locate the file by inserting the CD into your CD-ROM drive and going to the following directory:

#### cdrom/tgrid5.0/help/tutfiles/

Replace *cdrom* by the name of your CD-ROM drive.

• For Windows systems, locate the file by inserting the CD into your CD-ROM drive and going to the following directory:

cdrom:\tgrid5.0\help\tutfiles\

Replace *cdrom* by the name of your CD-ROM drive (e.g., E).

Alternatively, download wedge.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.

2. Unzip wedge.zip.

The file, wedge.msh can be found in the wedge folder created on unzipping the file.

3. Start the 2D version of TGrid (see Section 2.2: Starting TGrid).

#### 4.2 Reading the Boundary Mesh

1. Read in the mesh file.

 $\mathsf{File} \longrightarrow \mathsf{Read} \longrightarrow \mathsf{Boundary} \ \mathsf{Mesh}...$ 

The Select File dialog will open.

- Select File	
Filter	
/home/user/tutorial/#.{msh.MSH.cas}#	
Directories	Files
/home/user/tutorial/.	🔺 wedge.msh 🔺
/home/user/tutorial/	X X X
⊐ Appand File(v)	Select All
Mesh File(s)	B E Roseve
/home/user/tutorial/wedge.msh	
OK Filter	Cancel

Figure 4.2.1: Reading the Mesh File

The Filter and Directories that appear in the dialog box depend on the location of working directory.

2. Select wedge.msh in the Files list.

The file wedge.msh will be added to the Mesh File(s) list.

3. Click OK.

TGrid will read the mesh file and it's console shows the progress as the file is read.

#### 4.3 Examining the Boundary Mesh

After reading the mesh file, you can check the boundary mesh for topological problems such as free and multi-connected nodes and faces. All these operations can be performed in the Merge Boundary Nodes panel. You can display the grid using the Display Grid panel.

Boundary  $\longrightarrow$  Merge Nodes...

- Merge Bo	undary Nodes
Compare	With
Only Free Nodes	Only Free Nodes
Boundary Face Zones 📑 📃 🔛	Boundary Face Zones 🔳 🗐 🔛
pressure-inlet-3	pressure-inlet-3
Tolerance 5,26315e-05 Percent of shortest connected e	Count Free Nodes Count Unused Nodes Delete Unused Nodes
Message	
0 free nodes found	
Merge Draw Def	ault Close Help

Figure 4.3.1: The Merge Boundary Nodes Panel

1. Click Count Free Nodes to see if any free nodes exist.

**TGrid** reports (in the **Message** field) that none were found. You can click **Merge** to merge the free nodes, if any exist.

- 2. Click Count Unused Nodes to see if any unused nodes exist. You can click Delete Unused Nodes to delete unused nodes, if any exist.
- 3. Close the Merge Boundary Nodes panel.

For a 3D surface mesh, you can use the Boundary Improve panel (also opened from the Boundary menu) to check the boundary mesh quality and make improvements.

4. Display the boundary mesh.

 $\mathsf{Display} \longrightarrow \mathsf{Grid}...$ 

	υιοριαγ	Gina
Nodes Face	es Cells Bounds	Attributes
Options	Face Size Babge	Face Zones 📃 📃 🗎
F All	Missiona	<pre>pressure-inlet-3 wall-2</pre>
⊐ Free ⊐ Multi	Maximum	
Unmeshed	Face Quality Badge	
□ Left Handed □ Size	Minimum?	
🖬 Quality	Махіончы і	boundary
	Quality Measure	far-field geometry
	Displau Applu	Close Help

Figure 4.3.2: The Display Grid Panel

The Display Grid panel has capabilities for displaying free, multi-connected, and unmeshed faces. It is useful to display unmeshed faces when the initial mesh generation process fails and you want to locate the problem areas easily.

- (a) Select all the boundary zones in the Face Zones selection list.
- (b) Click Display (see Figure 4.3.3).

The boundaries are color coded according to their type.

(c) Close the Display Grid panel.



Figure 4.3.3: Display of Wedge Boundary Faces

# 4.4 Generating the Interior Mesh

When you are satisfied with the boundary mesh, generate the interior mesh. For most cases, the automatic mesh generation procedure will create an good-quality mesh.

Mesh  $\longrightarrow$  Auto Mesh...

- Auto Mesh	
Boundary Layer Mesh	
🖬 Prisero	Set
Quad Tet Tradsilied	
<ul> <li>Permeide</li> <li>Vee Conferente</li> </ul>	Set
Volume Fill	
◆ Tri/Tet ↓ Hexcere	
∻ No Fill	Set
Options	
Merge Cell Zones	
Mesh Apply Close	Help

Figure 4.4.1: The Auto Mesh Panel

1. Click Mesh in the Auto Mesh panel.

A Working dialog box will appear during the mesh generation. You can click Cancel in the Working dialog box to stop the mesh generation. TGrid will report the progress of the mesh generation in the console (Figure 4.4.2).

F	- TGrid [2D]	
	<u>File Boundary Mesh Display Report</u>	<u>H</u> elp
	Number of Cells Number of Nodes Nodes/Sec 495 211 4 Number of Cells Number of Nodes Nodes/Sec 629 278 67 smooth cells with skewness > 0.75 for 10 iterations. 0 faces swapped.	
	<pre>moved 0 nodes, most improvement 0 629 cells, min skew 4.112162e-15, max skew 0.72758774, avg skew 0.15876303. swap cells with skewness &gt; 0.2 for 5 iterations. smooth cells with skewness &gt; 0.75 for 5 iterations. 0 faces swapped. smooth cells with skewness &gt; 0.3 for 3 iterations. moved 0 nodes, most improvement 0 moved 178 nodes, rms 0.114148 moved 123 nodes, rms 0.0327698</pre>	
	629 cells, min skew 5.2864892e-06, max skew 0.48732538, avg skew 0.1173402. Done. [	V
F	4	

Figure 4.4.2: Text Output from Auto Mesh

2. Close the Auto Mesh panel.

This example describes the generation of the mesh using the default triangular mesh generation parameters. You can modify the mesh generation parameters in the Tri/Tet panel (see Section 11.5: The Tri/Tet Panel), if required.

# 4.5 Displaying the Mesh

You can display the mesh using the Display Grid panel (Figure 4.5.1).

 $\mathsf{Display} \longrightarrow \mathsf{Grid}...$ 

- 1. Click the **Cells** tab at the top of the panel.
- 2. Select fluid-8 in the Cell Zones selection list.

Alternatively, you can select fluid in the Cell Zone Groups list to select the fluid zone in the Cell Zones selection list.

-		Di	splay Gri	d
Nodes	Faces	Cells	Bounds	Attributes
Options All Size Quality	Cell Size R Missionan Maximum	ange 		Cell Zones 📑 🖃 dead-8 Auid-7
	Cell Occility Missionan Maximum Quality Ma	2000/2 1 1 1		Cell Zone Groups 📄 🗐 dead fluid solid
	Display	Apply	Clos	e Help

Figure 4.5.1: The Display Grid Panel—Cells Tab

- 3. Enable All in the **Options** group box.
- 4. Click Display (see Figure 4.5.2).



Figure 4.5.2: Grid Display

5. Close the Display Grid panel.

The Cell Zones list has a dead zone. When TGrid initializes the mesh, it automatically makes the cell zone with the largest volume (or the cell zone that contains a boundary zone with type pressure-outlet) the active fluid zone.

TGrid treats the rest of the cell zones (non-fluid zones) as dead zones. In this case, there is a dead zone for the region inside the wedge (see Figure 4.5.3). Since the cells in the dead zone are not in the computational domain, they can be ignored.

When you read the mesh into the solver, the dead cell zones will be ignored. If you have a large mesh with many cells in a dead zone, you can delete the dead zones using the Manage Cell Zones panel. This reduces the memory requirements. You can zoom in on the wedge using the middle mouse button to create a zoom box around it (see Section 14.9: Controlling the Mouse Buttons for details). The resulting display is shown in Figure 4.5.3.



Figure 4.5.3: Grid—Zoomed View

# 4.6 Reporting the Mesh Statistics

It is important to examine the grid statistics in the Report menu. Looking at tetrahedral cells does not provide much useful information about the mesh. Hence, you need to rely primarily on grid statistics.

1. Open the Report Boundary Cell Limits panel.

Report →Boundary Cell Limits...

(a) Click Compute.

The maximum cell skewness of cells having 1 boundary face is 0.47. A skewness of zero indicates an equilateral cell. You can click the Quality Measure... button to open the Quality Measure panel (see Section 15.5.2: The Quality Measure Panel) where you can report additional measures of quality (aspect ratio and

- Report Boundary Cell Limits		
Boundary	Number	Minimum
	1	3,902823e-05
Faces	Maximum	
		0,474659
Quality Meas	ure	
Compute	Clos	e Help

Figure 4.6.1: The Report Boundary Cell Limits Panel

change in size) by selecting them. The default measure of skewness is sufficient for analyzing the quality of triangular and tetrahedral cells.

- (b) Close the Report Boundary Cell Limits panel.
- 2. Open the Report Cell Limits panel.

Report  $\longrightarrow$  Cell Limits...

- Re	port Cell Limits
Options	Cell Zones 📃 🗏 🔛
<ul><li>◆ Quality</li><li>↓ Size</li></ul>	dead-8 Nuid-7
Quality Measure	
Minimum	
5,2865e-06	
Maximum	Cell Zone Groups 📑 🗐
0,4873254	dead
Average	fluid
0,1173402	<b>3011</b> 0
Compute	Close Help

Figure 4.6.2: The Report Cell Limits Panel

You can use this panel to determine the best (minimum), worst (maximum), and average cell quality (skewness, by default).

(a) Select Quality in the Options list.

- (b) Select the fluid-7 zone in the Cell Zones selection list.
- (c) Click Compute.

The worst cell has a skewness of 0.488. The average skewness is approximately 0.12. These values are typical for 2D problems and are suitable for obtaining a solution using FLUENT.

- (d) Close the Report Cell Limits panel.
- 3. Open the Report Mesh Size panel.

Report  $\longrightarrow$  Mesh Size...

You can use this panel to check the number of nodes, faces, and cells. This panel separates the number of nodes, faces, and cells into two groups—boundary and interior.

(a) Click Update.

The number of cells is 664 (see Figure 4.6.3).

- Report Mesh Size		
	Boundary	Interior
Nodes	73	278
Faces	73	941
Cells		664
Report Number Meshed		
Upda	te Close	Help

Figure 4.6.3: The Report Mesh Size Panel

- (b) Close the Report Mesh Size panel.
- 4. Plot the histogram.

Use the Cell Distribution panel to plot a histogram of the distribution of cell quality (skewness, by default) to get an overall feel of the quality of the mesh.

Display  $\longrightarrow$  Plot  $\longrightarrow$  Cell Distribution...

- (a) Select Quality in the Options list.
- (b) Select fluid-7 in the Cell Zones selection list.
- (c) Enter 0.5 for Maximum and 50 for Partitions.
- (d) Enable Percent of Total.

-	Cell Distribu	ition
Options Quality Size Quality Measure Axes	Minimum  Maximum  An aximum  An a	Cell Zones E E
Print	Plot Compute	fluid 7

(e) Click Plot.

TGrid will plot the histogram shown in Figure 4.6.4.



Figure 4.6.4: Histogram of Cell Quality (Skewness)

(f) Close the Cell Distribution panel.

### 4.7 Checking the Mesh

Before saving the mesh file, check it to make sure it has no negative cell volumes or left-handed faces. See Section 13.11: Checking the Mesh for details.

 $\mathsf{Mesh} \longrightarrow \mathsf{Check}$ 

```
Domain extents.
  x-coordinate: min = -1.000000e+01, max = 1.000000e+01.
  y-coordinate: min = -1.000000e+01, max = 1.000000e+01.
Volume statistics.
  minimum volume: 1.246947e-03.
  maximum volume: 2.172626e+00.
    total volume: 3.999559e+02.
Face area statistics.
   minimum face area: 1.022265e-02.
   maximum face area: 2.254692e+00.
   average face area: 9.797752e-01.
Checking number of nodes per cell.
Checking number of faces per cell.
Checking cell faces.
Checking face handedness.
Checking periodic face pairs.
Checking face children.
Checking face zone boundary conditions.
Checking for invalid node coordinates.
Done.
```

The printed results of the check do not show any problem, hence, the mesh is valid for use in FLUENT.

#### 4.8 Saving the Mesh and Exiting TGrid

 $\mathsf{File} \longrightarrow \mathsf{Write} \longrightarrow \mathsf{Mesh...}$ 

- 1. Enter the file name (wedge-final.msh) in the Mesh File field.
- 2. Click OK to write the file.

The default file name is the same as the input file name.

	ect file
Filter	
/home/user/tutorial/*.msł	าพู้
Directories	Files
/home/user/tutorial/ /home/user/tutorial/	wedge.msh
📕 Write Binary Files	
Mesh File	
wedge-final.msH	
OK Fi	Iter Cancel

Figure 4.8.1: Writing the Mesh File

3. End the TGrid session by selecting Exit in the File menu. File  $\longrightarrow$  Exit

Alternatively, you can type /exit in the TGrid console.