

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.

`File` → `Read` → Boundary Mesh...

*The Select File dialog will open.*

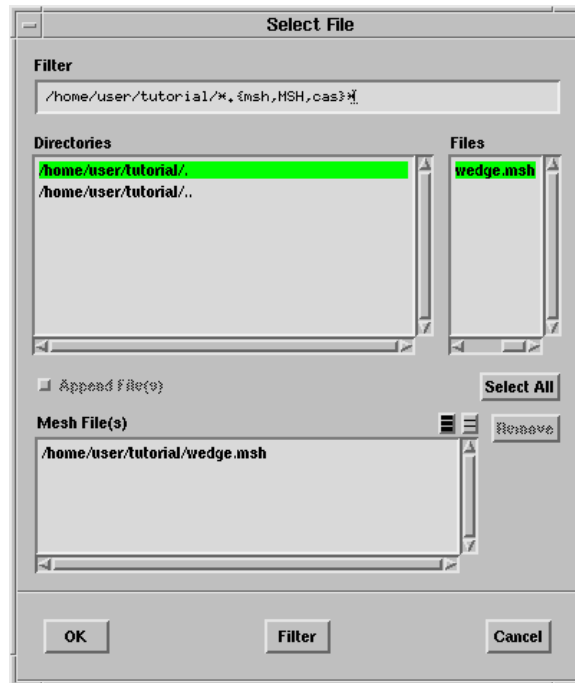


Figure 4.2.1: Reading the Mesh File

*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 its 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 → Merge Nodes...

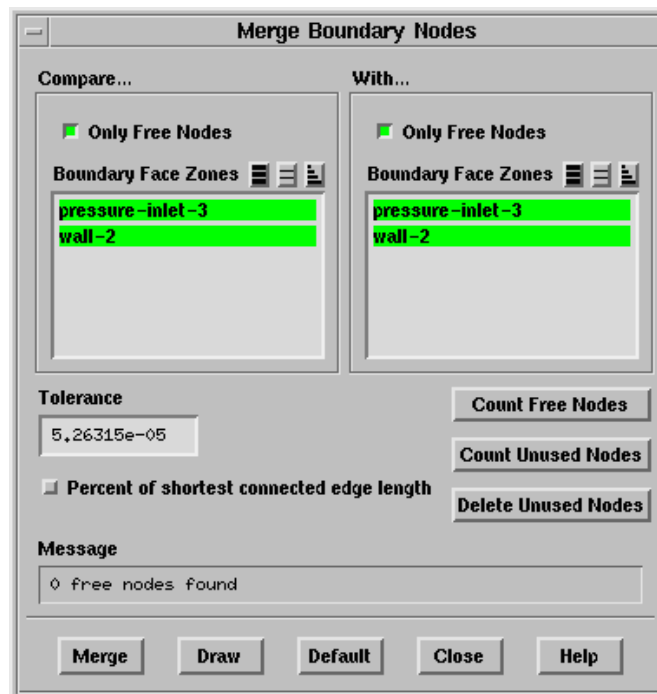


Figure 4.3.1: The Merge Boundary Nodes Panel

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.

Display → Grid...

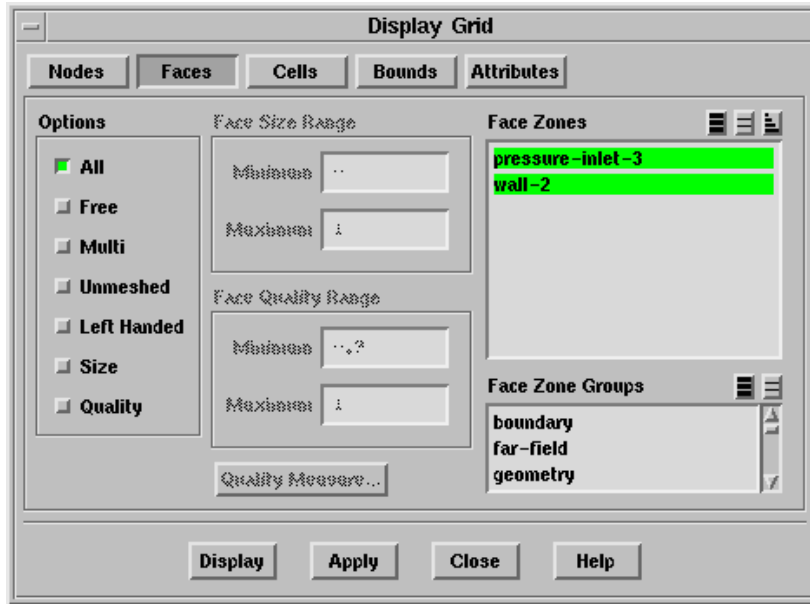


Figure 4.3.2: The Display Grid Panel

*The Display Grid panel has capabilities for displaying free, multi-connected, and un-meshed 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.

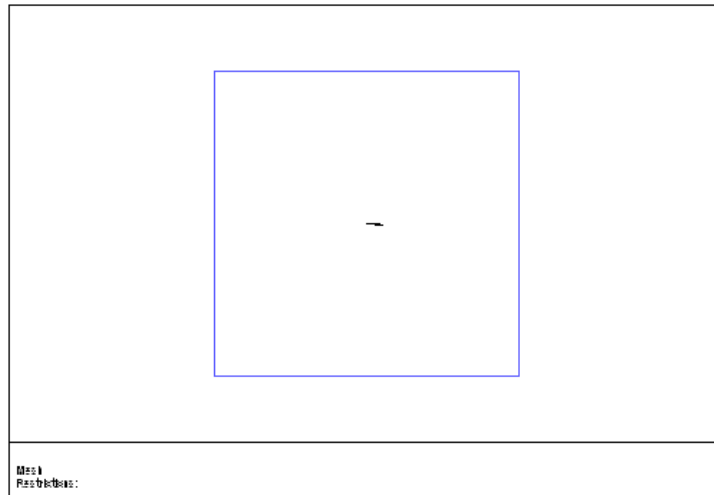


Figure 4.3.3: Display of Wedge Boundary Faces

## 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** → Auto Mesh...



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).

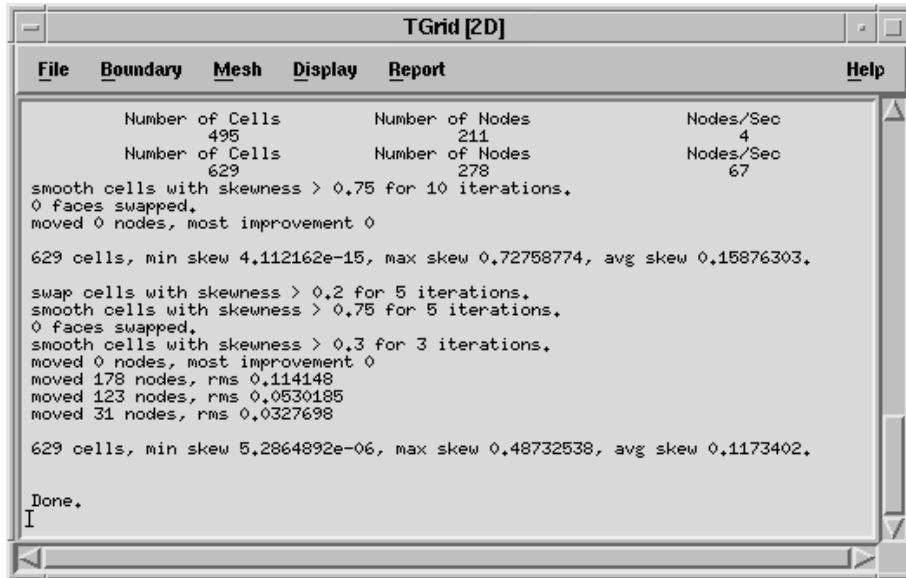


Figure 4.4.2: Text Output from Auto Mesh

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).

Display → 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.*

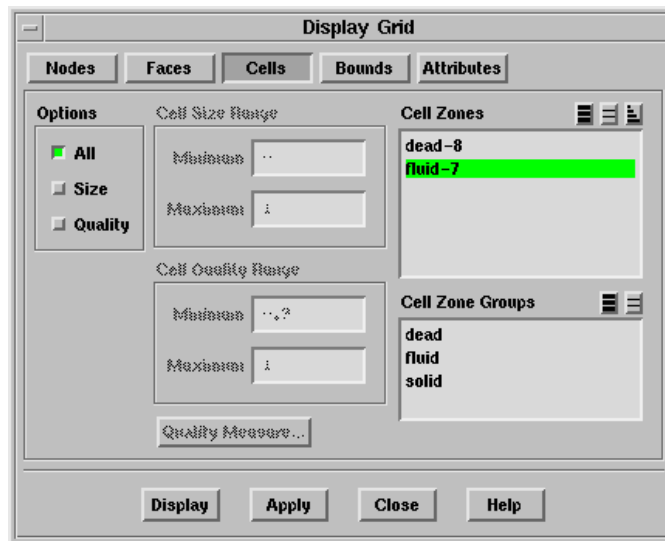


Figure 4.5.1: The Display Grid Panel—Cells Tab

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

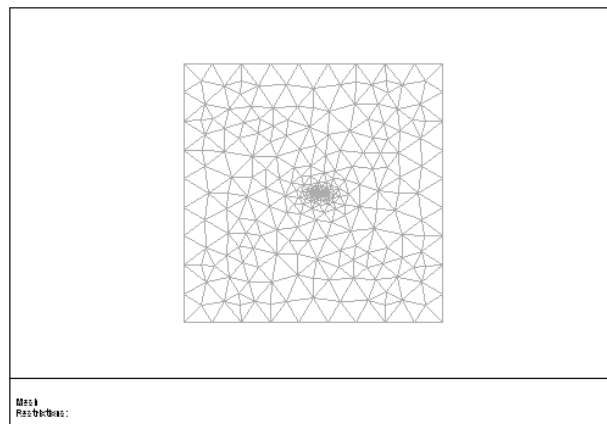


Figure 4.5.2: Grid Display

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.

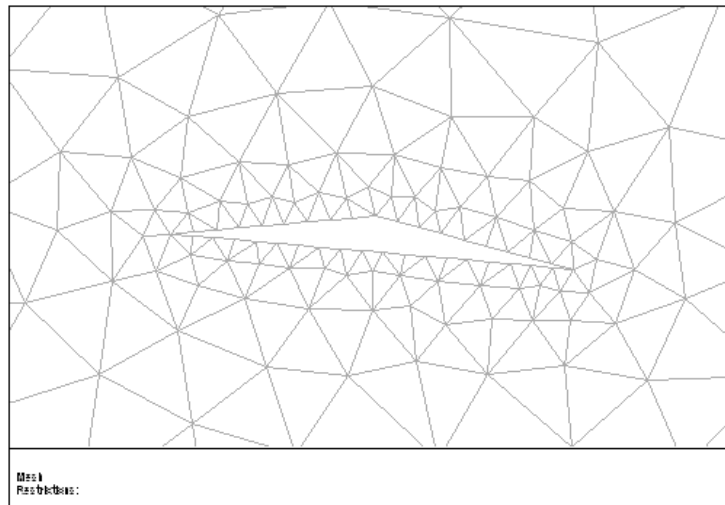


Figure 4.5.3: Grid—Zoomed View

## 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*



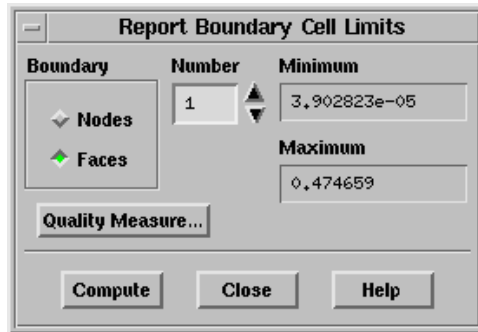


Figure 4.6.1: The Report Boundary Cell Limits Panel

*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 → Cell Limits...

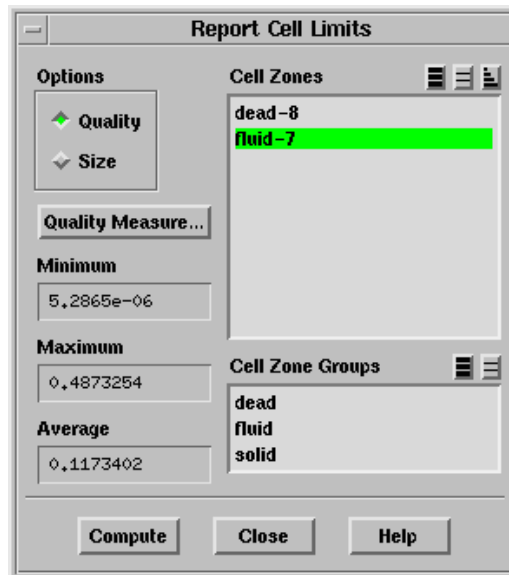


Figure 4.6.2: The Report Cell Limits Panel

*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** → 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).*

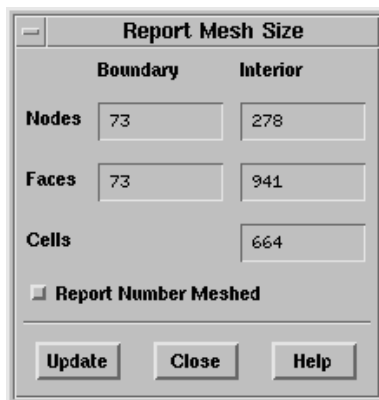


Figure 4.6.3: The Report Mesh Size Panel

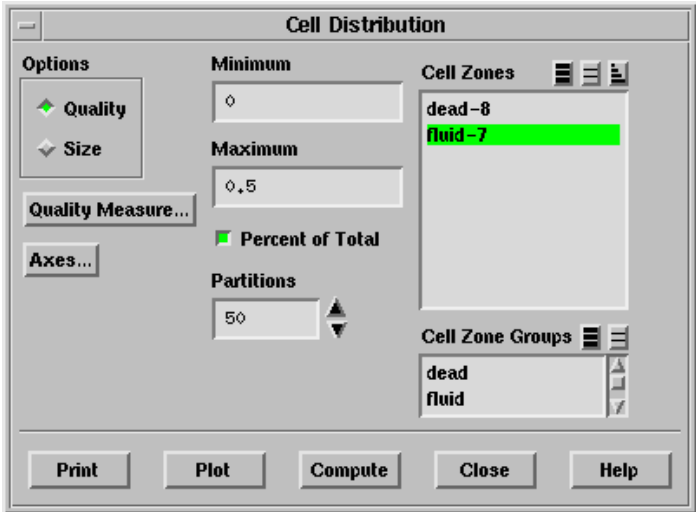
- (b) Close the Report Mesh Size panel.

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** → **Plot** → 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.



- (e) Click Plot.  
TGrid will plot the histogram shown in Figure 4.6.4.

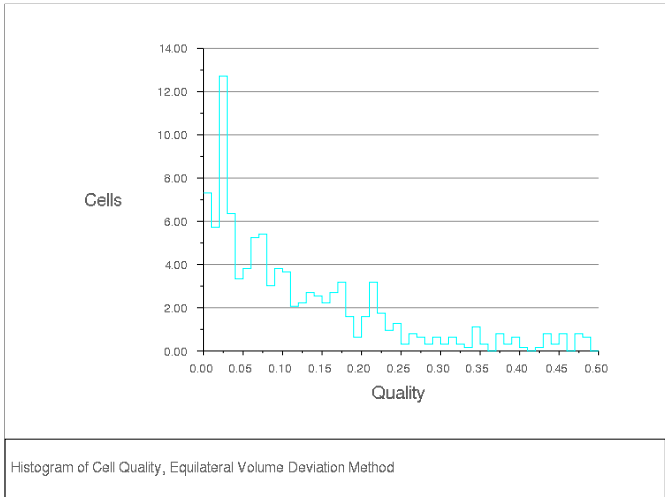


Figure 4.6.4: Histogram of Cell Quality (Skewness)

- (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.

Mesh → 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

File → Write → 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.*

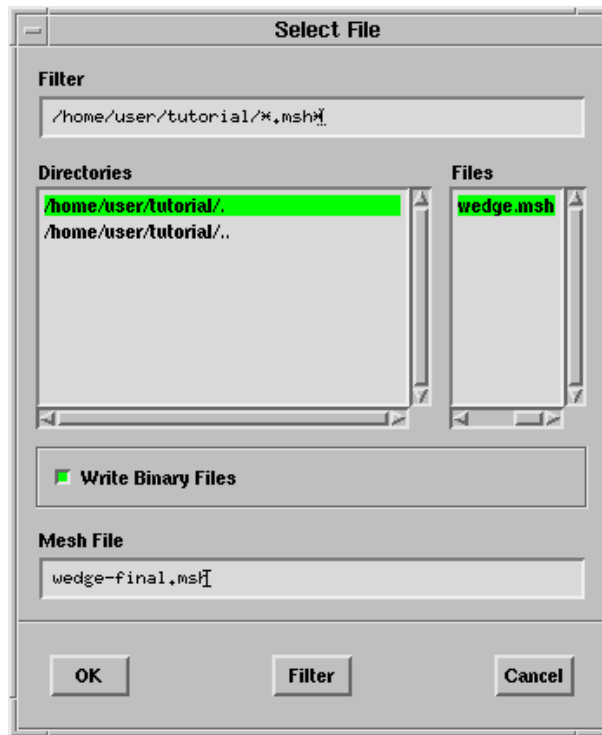


Figure 4.8.1: Writing the Mesh File

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

**File** → Exit

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

