

Fluid Dynamics

Structural Mechanics

Electromagnetics

Systems and Multiphysics

## ANSYS FLUENT Meshing Tutorials

---



ANSYS, Inc.  
Southpointe  
275 Technology Drive  
Canonsburg, PA 15317  
[ansysinfo@ansys.com](mailto:ansysinfo@ansys.com)  
<http://www.ansys.com>  
(T) 724-746-3304  
(F) 724-514-9494

Release 14.5  
October 2012

ANSYS, Inc. is  
certified to ISO  
9001:2008.

---

## **Copyright and Trademark Information**

© 2012 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, ANSYS Workbench, Ansoft, AUTODYN, EKM, Engineering Knowledge Manager, CFX, FLUENT, HFSS and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners.

## **Disclaimer Notice**

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. is certified to ISO 9001:2008.

## **U.S. Government Rights**

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

## **Third-Party Software**

See the [legal information](#) in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, please contact ANSYS, Inc.

Published in the U.S.A.

---

# Table of Contents

<b>1. Using This Manual .....</b>	1
1.1.What's In This Manual .....	1
1.2.Where to Find the Files Used in the Tutorials .....	2
1.3.Typographical Conventions Used In This Manual .....	2
<b>2. Repairing a Boundary Mesh .....</b>	3
2.1.Prerequisites .....	3
2.2.Preparation .....	3
2.3.Starting ANSYS FLUENT in Meshing Mode .....	4
2.4.Read and Display the Boundary Mesh .....	5
2.5.Check for Free and Unused Nodes .....	8
2.6.Repair the Boundary Mesh .....	9
2.7.Use the Rezoning Feature .....	13
2.8.Improve the Boundary Mesh .....	15
2.9.Check the Skewness Distribution of the Boundary Mesh .....	15
2.10.Further Improve the Boundary Mesh .....	16
2.11.Generate a Multiple Region Volume Mesh .....	29
2.12.Check the Volume Mesh Quality .....	31
2.13.Check and Save the Volume Mesh .....	32
2.14.Summary .....	33
<b>3. Tetrahedral Mesh Generation .....</b>	35
3.1.Prerequisites .....	35
3.2.Preparation .....	35
3.3.Read and Display the Boundary Mesh .....	35
3.4.Generate the Mesh using the Skewness-Based Refinement Method .....	38
3.5.Generate the Mesh using the Skewness-Based Refinement Method and a Size Function .....	43
3.6.Generate the Mesh using the Advancing Front Refinement Method and a Size Function .....	45
3.7.Examine the Effect of the Growth Factor .....	47
3.8.Generate a Local Refinement in the Wake of the Car .....	51
3.9.Check and Save the Volume Mesh .....	55
3.10.Summary .....	55
<b>4. Zonal Hybrid Mesh Generation .....</b>	57
4.1.Prerequisites .....	57
4.2.Preparation .....	57
4.3.Generate the Tetrahedral Mesh Using Pyramids to Transition Between the Hexahedral and Tetrahedral Mesh .....	58
4.4.Generate the Tetrahedral Mesh Using a Non-Conformal Transition Between the Hexahedral and Tetrahedral Mesh .....	78
4.5.Summary .....	82
<b>5. Viscous Hybrid Mesh Generation .....</b>	83
5.1.Prerequisites .....	83
5.2.Preparation .....	83
5.3.Generate the Mesh Using the Allow Shrinkage Option and Manual Tetrahedral Meshing .....	84
5.4.Generate the Mesh Using the Allow Ignore Option and Automatic Meshing .....	100
5.5.Summary .....	106
<b>6. Hexcore Mesh Generation .....</b>	107
6.1.Prerequisites .....	107
6.2.Preparation .....	107
6.3.Read and Display the Mesh .....	108
6.4.Check the Skewness of the Surface Mesh .....	109
6.5.Improve the Boundary Mesh .....	110

6.6. Generate the Hexcore Mesh .....	113
6.7. Examine the Effect of the Buffer Layers on the Hexcore Mesh .....	115
6.8. Automatically Generate the Hexcore Mesh with Prism Layers and a Local Refinement Region .....	118
6.9. Summary .....	125
<b>7. Generating the Hexcore Mesh to Domain Boundaries .....</b>	<b>127</b>
7.1. Prerequisites .....	127
7.2. Preparation .....	127
7.3. Manually Generate the Hexcore Mesh to the Boundaries .....	127
7.4. Automatically Generate the Hexcore Mesh to the Boundaries with Prism Layers .....	133
7.5. Generate the Hexcore Mesh to the Boundaries with Prism Layers Using TUI Commands .....	146
7.6. Summary .....	147
<b>8. Using the Boundary Wrapper .....</b>	<b>149</b>
8.1. Prerequisites .....	149
8.2. Preparation .....	149
8.3. Read and Display the Mesh .....	150
8.4. Perform Pre-Wrapping Operations to Close Holes in the Geometry .....	152
8.5. Initialize the Surface Wrapper .....	162
8.6. Examine the Region to be Wrapped .....	163
8.7. Refine the Main Region .....	169
8.8. Close Small Holes Automatically .....	171
8.9. Wrap the Main Region .....	174
8.10. Capture Features .....	175
8.11. Post Wrapping Operations .....	179
8.12. Create the Tunnel .....	190
8.13. Generate the Volume Mesh .....	191
8.14. Improve the Volume Mesh .....	193
8.15. Separate the Tunnel Inlet and Outlet .....	194
8.16. Summary .....	195
<b>9. CutCell Mesh Generation .....</b>	<b>197</b>
9.1. Prerequisites .....	197
9.2. Preparation .....	197
9.3. Import the CAD Geometry .....	197
9.4. Create Capping Surfaces for the Inlet and Outlets .....	201
9.5. Set Up Size Functions .....	207
9.6. Generate the CutCell Mesh .....	209
9.7. Post CutCell Meshing Cleanup Operations .....	215
9.8. Generating Prisms for the CutCell Mesh .....	217
9.9. Summary .....	222
<b>10. Object Based Mesh Generation .....</b>	<b>225</b>
10.1. Prerequisites .....	225
10.2. Preparation .....	225
10.3. Starting ANSYS FLUENT in Meshing Mode .....	226
10.4. Import the CAD Geometry .....	227
10.5. Prepare the Geometry .....	228
10.6. Sewing Objects .....	234
10.7. Generate the Volume Mesh .....	237
10.8. Transfer the Mesh to Solution Mode .....	242
10.9. Solution Setup .....	242
10.10. Summary .....	260
<b>11. Cavity Remeshing .....</b>	<b>261</b>
11.1. Prerequisites .....	261
11.2. Preparation .....	261

11.3. Cavity Remeshing For a Tetrahedral Mesh .....	261
11.4. Cavity Remeshing For a Hybrid Mesh (Tetrahedra and Prisms) Having a Single Fluid Zone .....	271
11.5. Cavity Remeshing For a Hybrid Mesh (Tetrahedra and Prisms) Having Multiple Fluid Zones .....	282
11.6. Cavity Remeshing For a Hexcore Mesh .....	285
11.7. Summary .....	289



---

# Chapter 1: Using This Manual

---

This preface is divided into the following sections:

- 1.1.What's In This Manual
- 1.2.Where to Find the Files Used in the Tutorials
- 1.3.Typographical Conventions Used In This Manual

## 1.1.What's In This Manual

This Tutorial Guide contains a few tutorials that teach you how to use the meshing mode in ANSYS FLUENT for different types of problems. Each tutorial contains instructions for performing tasks related to the features demonstrated in the tutorial.

- [Repairing a Boundary Mesh \(p. 3\)](#) is a detailed tutorial designed to introduce the beginner to the meshing mode in ANSYS FLUENT. This tutorial provides explicit instructions for all steps in the tutorial.  
The remaining tutorials assume that you have read or solved [Repairing a Boundary Mesh \(p. 3\)](#), and that you are already familiar with the interface. Some steps will not be shown explicitly in these tutorials.
- [Tetrahedral Mesh Generation \(p. 35\)](#) demonstrates the mesh generation procedure for a problem that has multiple regions.
- [Zonal Hybrid Mesh Generation \(p. 57\)](#) demonstrates the mesh generation procedure for a hybrid mesh, starting from a hexahedral volume mesh and a triangular boundary mesh.
- [Viscous Hybrid Mesh Generation \(p. 83\)](#) demonstrates the mesh generation procedure for a viscous hybrid mesh, starting from a triangular boundary mesh for a sedan car body.
- [Hexcore Mesh Generation \(p. 107\)](#) explains an application from the automotive industry, demonstrating how the hexcore mesh can significantly reduce the cell count compared with a fully tetrahedral mesh.
- [Generating the Hexcore Mesh to Domain Boundaries \(p. 127\)](#) demonstrates the creation of a hexcore mesh up to the domain boundaries for a sedan car.
- [Using the Boundary Wrapper \(p. 149\)](#) demonstrates the use of the boundary wrapper to repair an existing geometry. It also describes the procedure to improve the wrapper surface quality.
- [CutCell Mesh Generation \(p. 197\)](#) demonstrates the procedure for generating a CutCell mesh.
- [Object Based Mesh Generation \(p. 225\)](#) demonstrates the procedure for generating the volume mesh for a mixer pipe based on objects from the imported geometry in the meshing mode in ANSYS FLUENT. It also demonstrates the transfer of the mesh to the solution mode, the set up and solution of the CFD problem, and visualizing the results.
- [Cavity Remeshing \(p. 261\)](#) demonstrates the procedure for replacing an entity in the existing mesh with another by creating a cavity and remeshing it.

## 1.2. Where to Find the Files Used in the Tutorials

Each of the tutorials uses existing mesh or geometry files. The **Preparation** step of each tutorial will indicate the necessary files. You will find the appropriate files on the ANSYS Customer Portal.

## 1.3. Typographical Conventions Used In This Manual

Several typographical conventions are used in the text of the tutorials to facilitate your learning process.

- Different type styles are used to indicate graphical user interface menu items and text interface menu items (e.g., **Display Grid** dialog box, display/grid command).
- The text interface type style is also used when illustrating exactly what appears on the screen or exactly what you need to type in the text field in a dialog box.
- Instructions for performing each step in a tutorial will appear in standard type.
- A mini flow chart is used to guide you through the navigation pane, which leads you to a specific task page or dialog box. For example,



indicates that **Size Functions...** is selected in the **Create** group box in the **Mesh Generation** task page.

or



indicates that **Models** is selected in the navigation pane, which then opens the corresponding task page. In the **Models** task page, **Viscous** is selected from the list. Clicking the **Edit...** button opens the **Viscous Model** dialog box.

A mini flow chart is also used to indicate the menu selections that lead you to a specific dialog box. For example,



indicates that the **Grid...** menu item can be selected from the **Display** pull-down menu.

---

## Chapter 2: Repairing a Boundary Mesh

---

ANSYS FLUENT offers several tools for mesh repair in meshing mode. While there is no right or wrong way to repair a mesh, the goal is to improve the quality of the mesh with each mesh repair operation. This tutorial demonstrates the use of some mesh repair tools to find and fix known deficiencies in an existing boundary mesh.

This tutorial demonstrates how to do the following:

- Read the mesh file and display the boundary mesh.
- Check for free and unused nodes.
- Repair the boundary mesh by recreating missing faces.
- Use the rezoning feature.
- Improve the boundary mesh.
- Check the skewness of the boundary faces.
- Further improve the boundary mesh.
- Generate a multiple region volume mesh.
- Check the quality of the entire volume mesh.
- Check and save the volume mesh.

### 2.1. Prerequisites

This tutorial assumes that you have little experience with the meshing mode in ANSYS FLUENT, but are familiar with the graphical user interface.

### 2.2. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

To prepare for running this tutorial:

1. Download the tutorial input file (`mesh-repair.zip`) for the tutorial.
2. Unzip `mesh-repair.zip`.

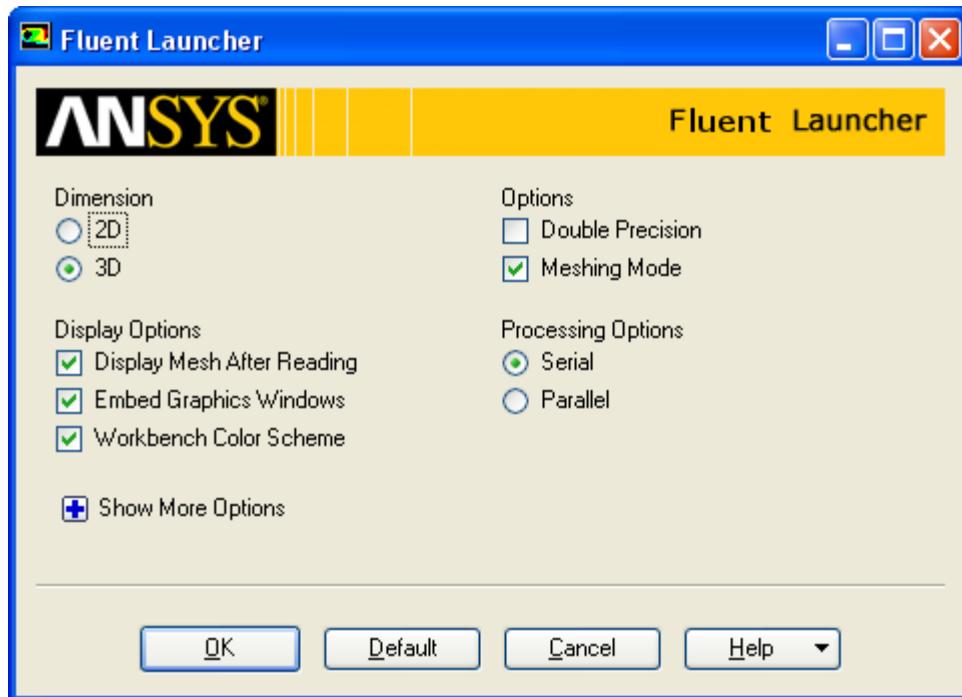
The file `problem-surf.msh` can be found in the `mesh-repair` folder created on unzipping the file.

## 2.3. Starting ANSYS FLUENT in Meshing Mode

1. Open the FLUENT Launcher by clicking the Windows **Start** menu, then selecting **FLUENT 14.5** in the **Fluid Dynamics** sub-menu of the **ANSYS 14.5** program group.

**Start → All Programs → ANSYS 14.5 → Fluid Dynamics → FLUENT 14.5**

2. Select the appropriate start up options.



- a. Ensure that **3D** is selected in the **Dimension** list.
- b. Enable **Meshing Mode** under **Options**.
- c. Retain the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options.

---

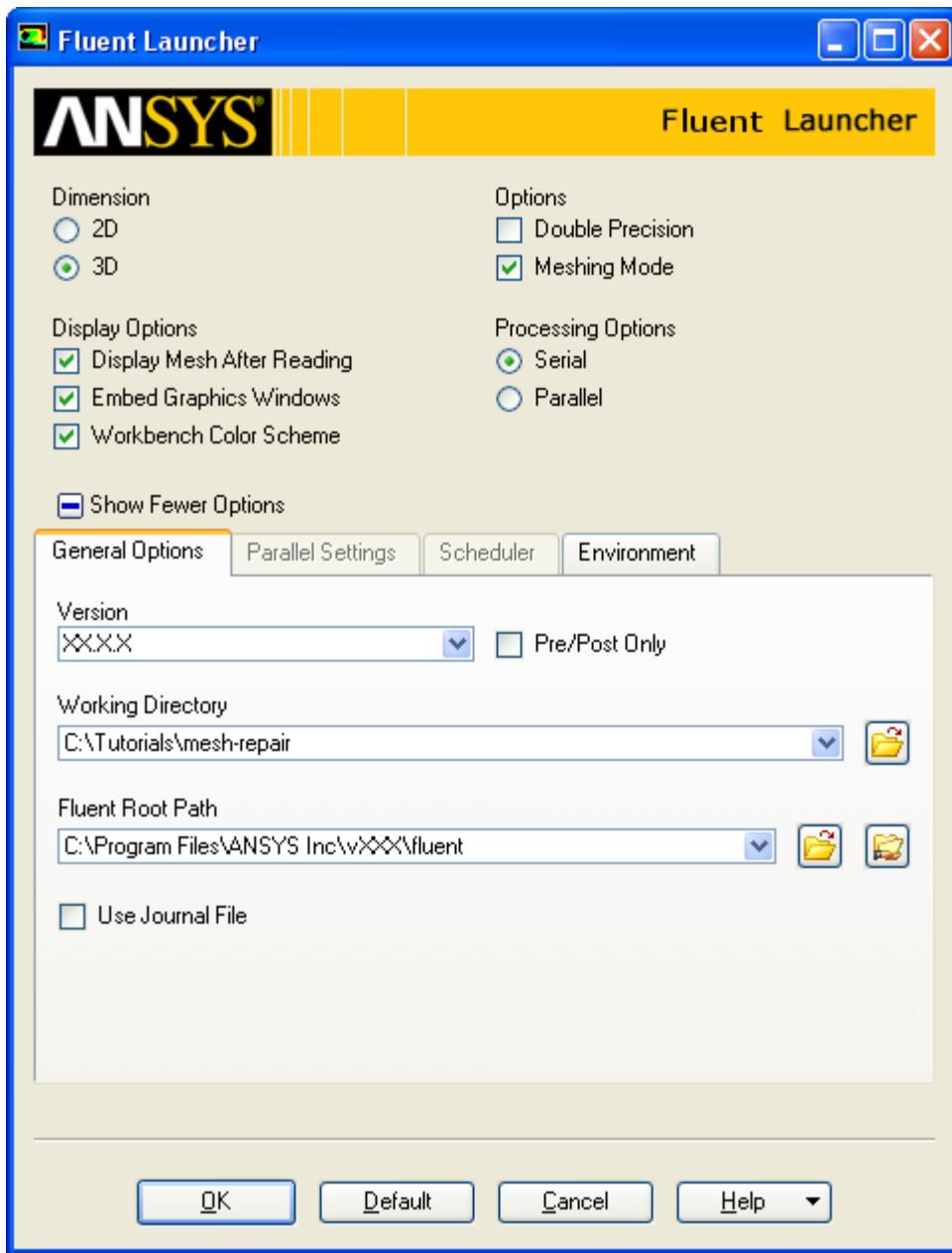
### Note

The selected preferences will be retained for future sessions.

---

3. Set the path to the working directory.

- a. Click the **Show More Options** button.



- b. Enter the path to the working directory by double-clicking the **Working Directory** text box and typing.

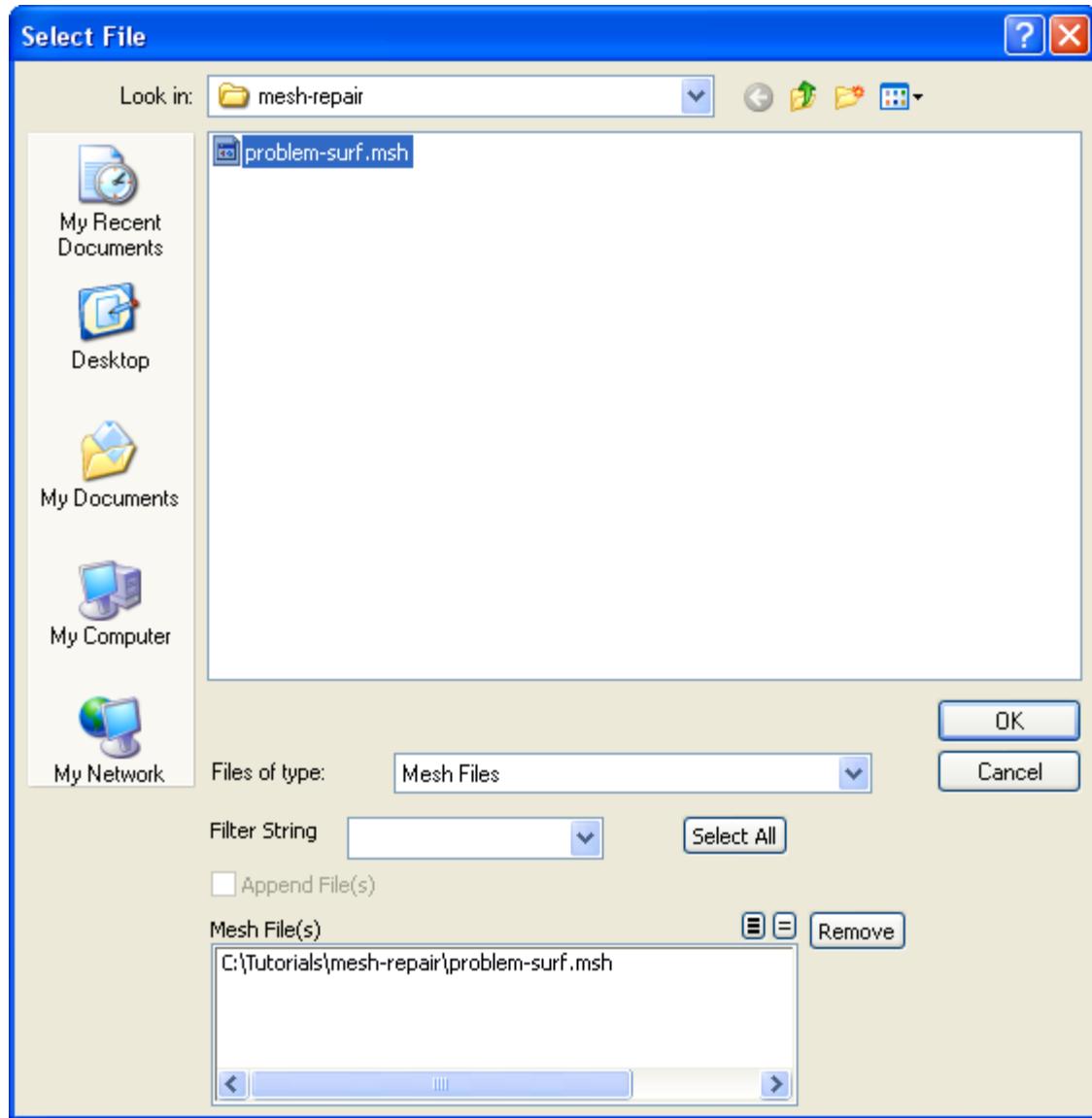
Alternatively, you can click the browse button (  ) next to the **Working Directory** text box and browse to the directory, using the **Browse For Folder** dialog box.

4. Click **OK** to start ANSYS FLUENT in meshing mode.

## 2.4. Read and Display the Boundary Mesh

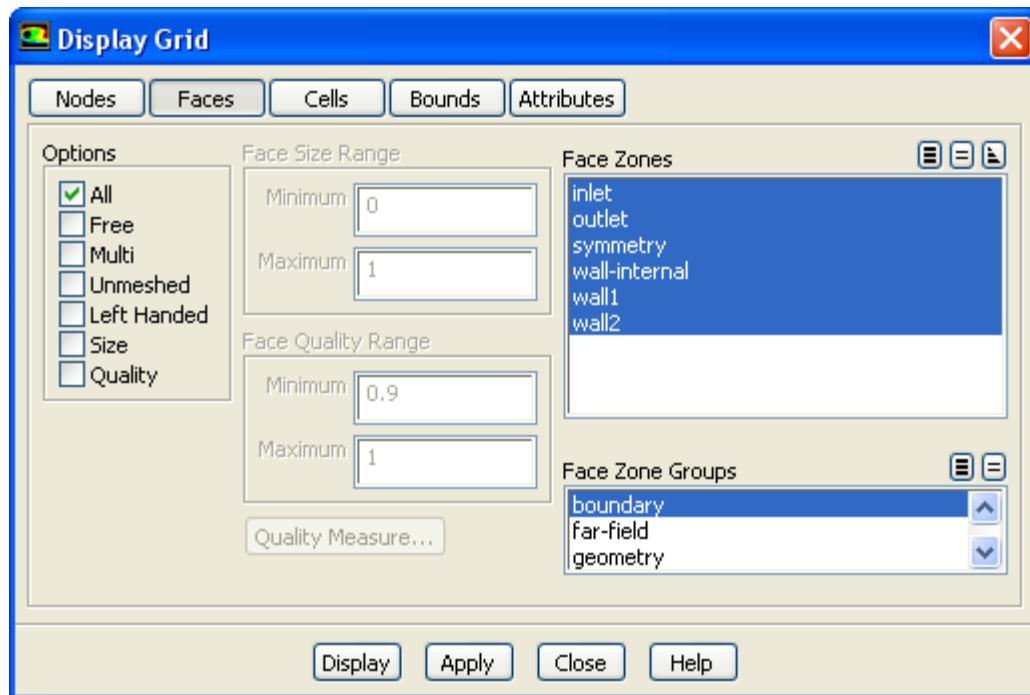
1. Read in the boundary mesh file (`problem-surf.msh`).

**File → Read → Boundary Mesh...**



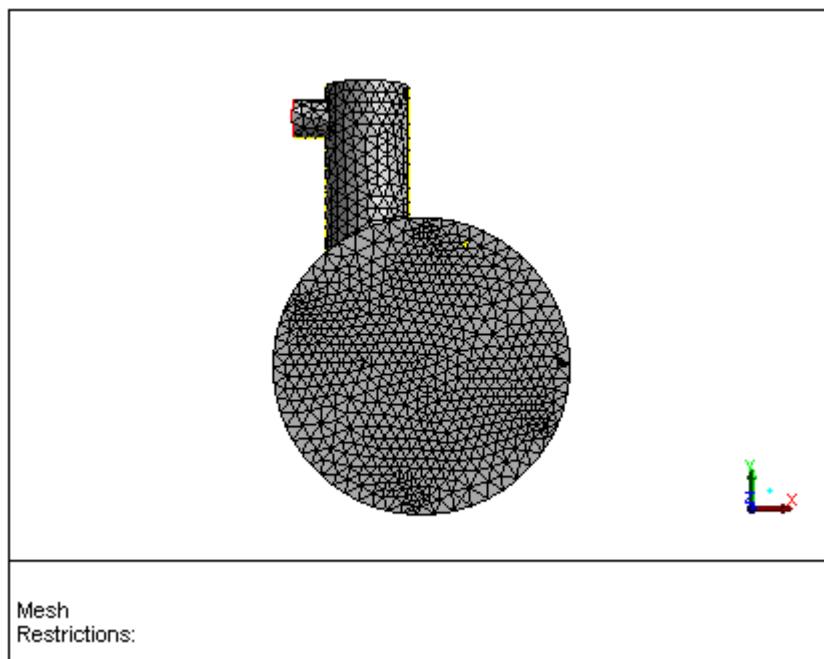
- Select **problem-surf.msh** and click **OK**.
2. Display the boundary mesh.

**Display → Grid...**



- Select **boundary** in the **Face Zone Groups** selection list.
- Click the **Attributes** tab and enable **Filled** and **Lights**.
- Click **Display** (Figure 2.1: Boundary Mesh (p. 7)).

**Figure 2.1: Boundary Mesh**

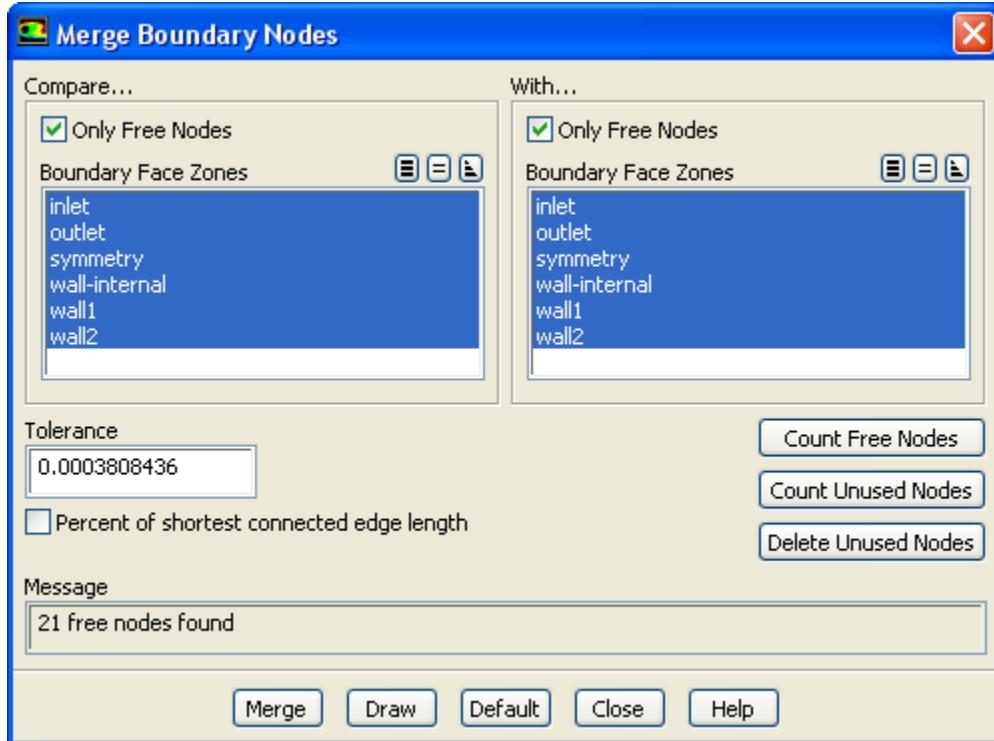


- Close the **Display Grid** dialog box.

## 2.5. Check for Free and Unused Nodes

After reading the boundary mesh, you will check for topological problems such as free and multiply-connected nodes and faces.

**Boundary → Merge Nodes...**



1. Click **Count Free Nodes**.

The number of free nodes is reported in the **Message** box.

---

### Note

Here, the free nodes are due to seven missing faces in the surface mesh. [Repair the Boundary Mesh \(p. 9\)](#) demonstrates the use of the mesh repair tools to recreate the missing faces.

---

2. Click **Count Unused Nodes**.

The number of unused nodes is reported in the **Message** box. If there are unused nodes, click **Delete Unused Nodes** to remove them.

3. Close the **Merge Boundary Nodes** dialog box.

## 2.6. Repair the Boundary Mesh

This section demonstrates repairing the boundary mesh by recreating the missing faces.

### Note

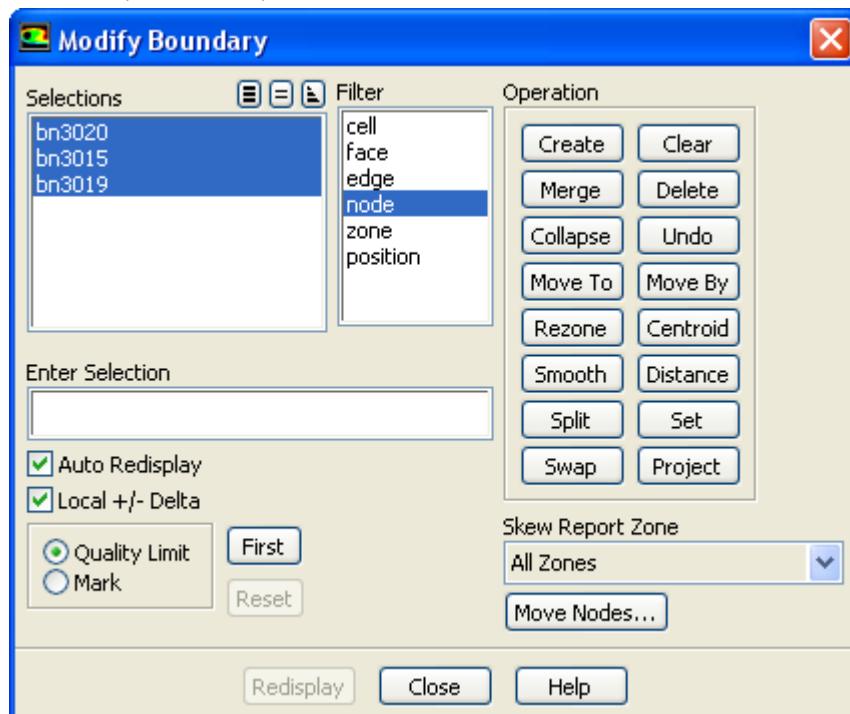
A number of the repair operations have associated keyboard shortcuts which allow you to perform these operations repetitively without using the dialog boxes. Refer to the appendix on Shortcut Keys in the [Meshing User's Guide](#) for details. You can also click in the graphics window and use the hot-key, **Ctrl-H**, to obtain a list of the hot keys available.

1. Zoom in to one of the missing faces ([Figure 2.2: Recreating the Missing Face \(p. 10\)](#)).

The faces surrounding the missing face can be highlighted to enable easy identification of the missing face. Enable **Free** in the **Options** group box in the **Display Grid** dialog box to highlight the faces surrounding the missing face.

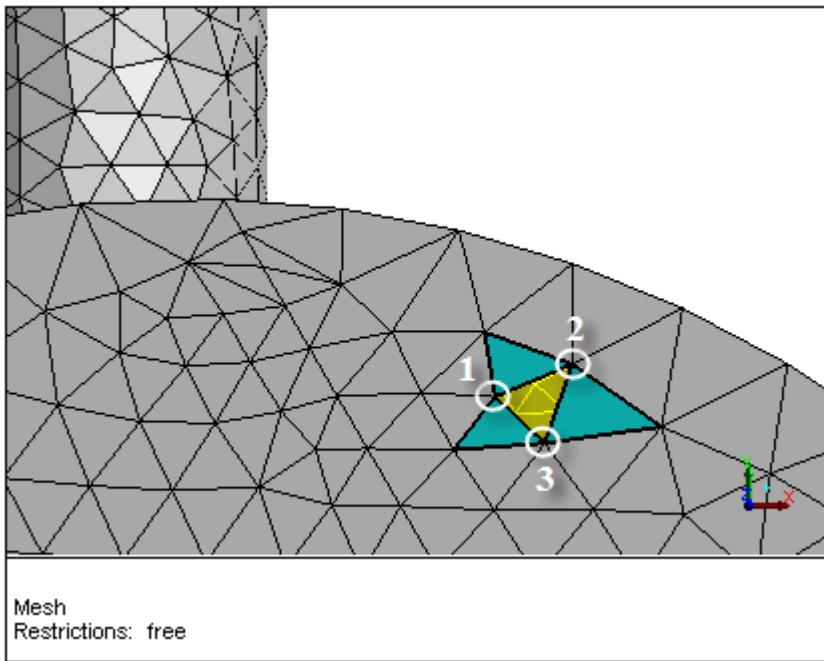
2. Recreate the missing face.

### Boundary → Modify...



- Select **node** in the **Filter** list.
- Select the three nodes surrounding the missing face using the right mouse button (see [Figure 2.2: Recreating the Missing Face \(p. 10\)](#)).

**Figure 2.2: Recreating the Missing Face**



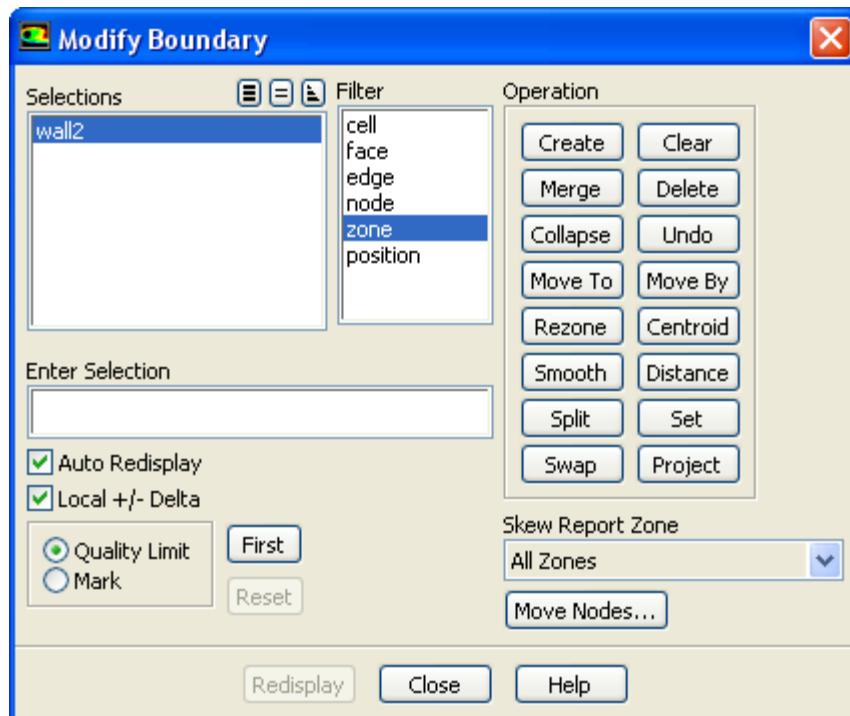
---

**Tip**

If you select the wrong node, click on it again with the right mouse button to remove it from the **Selections** list.

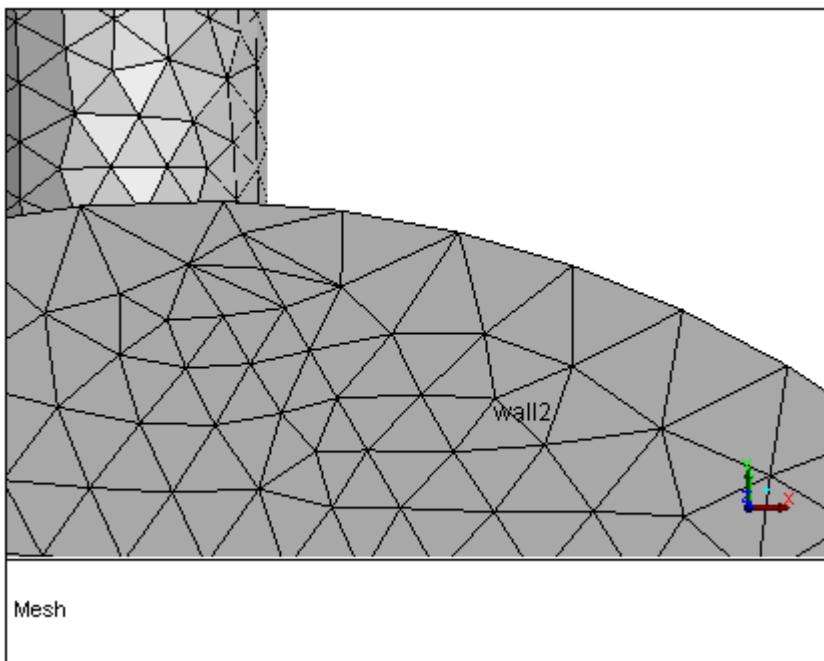
---

- c. Click **Create** in the **Operation** group box when the correct nodes are selected.  
The missing face will be recreated.
3. Check if the new face is in the correct boundary zone.



- a. Select **zone** in the **Filter** list.
- b. Select the face just created using the right mouse button.

The zone name will be displayed in the graphics window ([Figure 2.3: Verifying the Zone of the New Face \(p. 12\)](#)).

**Figure 2.3: Verifying the Zone of the New Face****Note**

The face is placed in the same zone as the majority of the nodes that comprise the face. If two out of the three selected nodes are in the **symmetry** zone, then the face created is placed in the **symmetry** zone. In this example, the three nodes selected are in the **wall2** zone, hence the face created is also placed in the **wall2** zone.

- c. If the face is in the wrong zone, use the **Rezone** option in the **Operation** group box to move the face to the appropriate zone (see [Use the Rezoning Feature \(p. 13\)](#)).
  - 4. Similarly, recreate the other missing faces.
  - 5. Verify that all missing faces have been recreated.
- Display the boundary mesh with only **Free** enabled in the **Options** group box in the **Display Grid** dialog box to ensure that no free faces exist.
- 6. Save an intermediate mesh file (`ttemp.msh`).

**File → Write → Mesh...**

**Warning**

It is not always possible to undo an operation. Hence, it is recommended that you save the mesh periodically when modifying the boundary mesh.

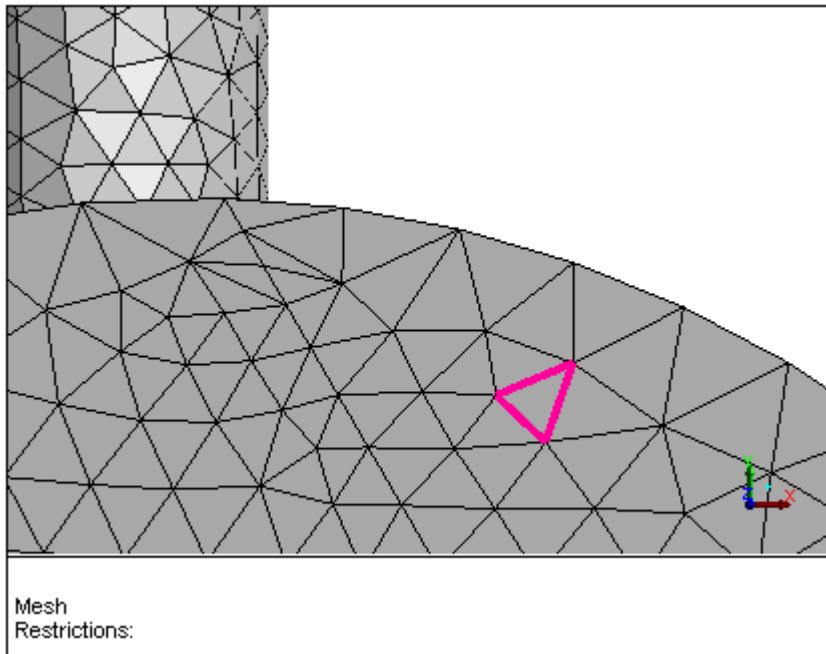
## 2.7. Use the Rezoning Feature

This section illustrates the use of the **Rezone** option to move a face from one zone to another. First, you will move the face from the **wall2** boundary to the **symmetry** boundary. When this is done, you will move the selected face back to the **wall2** zone.

### Boundary → Modify...

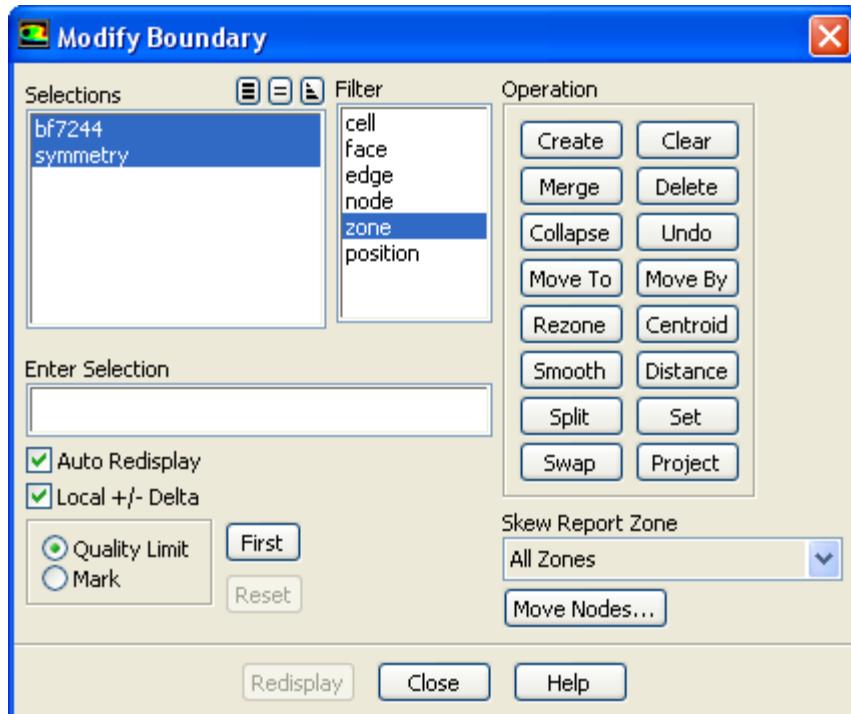
1. Select **face** in the **Filter** list.
2. Select the face to be rezoned using the right mouse button (Figure 2.4: Face Selected to be Rezoned (p. 13)).

**Figure 2.4: Face Selected to be Rezoned**



3. Select **zone** in the **Filter** list.
4. Select the zone where you want to move the face using the right mouse button (**symmetry**).

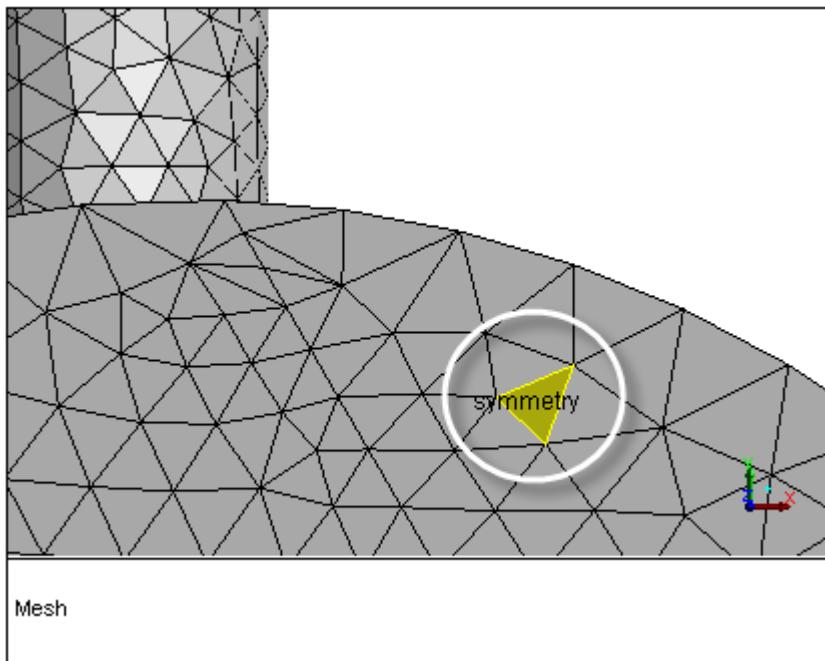
After selecting the **symmetry** zone the **Selections** list in the **Modify Boundary** dialog box will show the face identification number and the zone to which you want to move it.



5. Click **Rezone** in the **Operation** group box.

The selected face will be moved to the **symmetry** zone (Figure 2.5: Face Rezoned to Symmetry Boundary (p. 14)).

**Figure 2.5: Face Rezoned to Symmetry Boundary**



### Warning

This section was included only to demonstrate the use of the **Rezone** option. Move the selected face back to the **wall2** zone using the **Rezone** operation.

- Close the **Modify Boundary** dialog box.

## 2.8. Improve the Boundary Mesh

**Boundary** → **Mesh** → **Improve...**



- Select all the zones in the **Tri Boundary Zones** selection list.
- Select **Swap** in the **Options** drop-down list.
- Click **Skew** to check if the maximum face skewness is below 0.9.  
The maximum face skewness is approximately 0.992.
- Click **Check** to check for Delaunay violations in the boundary mesh.  
The violations will be reported in the console.
- Retain the default values of 10 and 0.9 for **Max Angle** and **Max Skew**, respectively.
- Click **Apply** until zero modifications are reported in the console.
- Click **Skew** to verify that the maximum face skewness is below 0.9.
- Close the **Boundary Improve** dialog box.

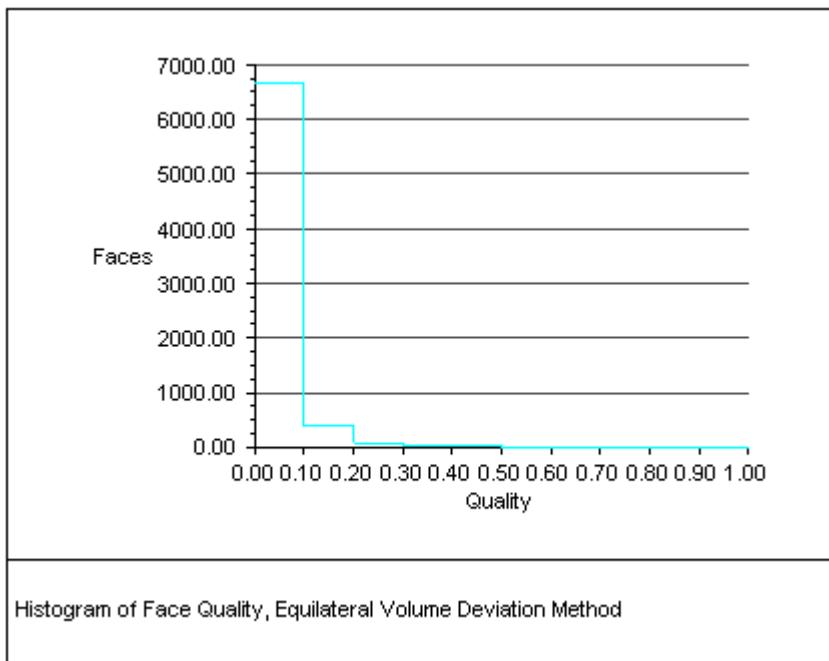
## 2.9. Check the Skewness Distribution of the Boundary Mesh

**Display** → **Plot** → **Face Distribution...**

- Select all the zones in the **Boundary Zones** selection list.
- Enter 10 for **Partitions**.

3. Click **Plot** (Figure 2.6: Histogram Plot of Face Skewness (p. 16)).

**Figure 2.6: Histogram Plot of Face Skewness**



4. Click **Print**.

The histogram information will be printed by decades in the console. There are zero faces with a skewness greater than 0.9, four faces with a skewness greater than 0.8, two faces with a skewness greater than 0.7, and 11 faces with a skewness greater than 0.6.

5. Close the **Face Distribution** dialog box.

#### Extra

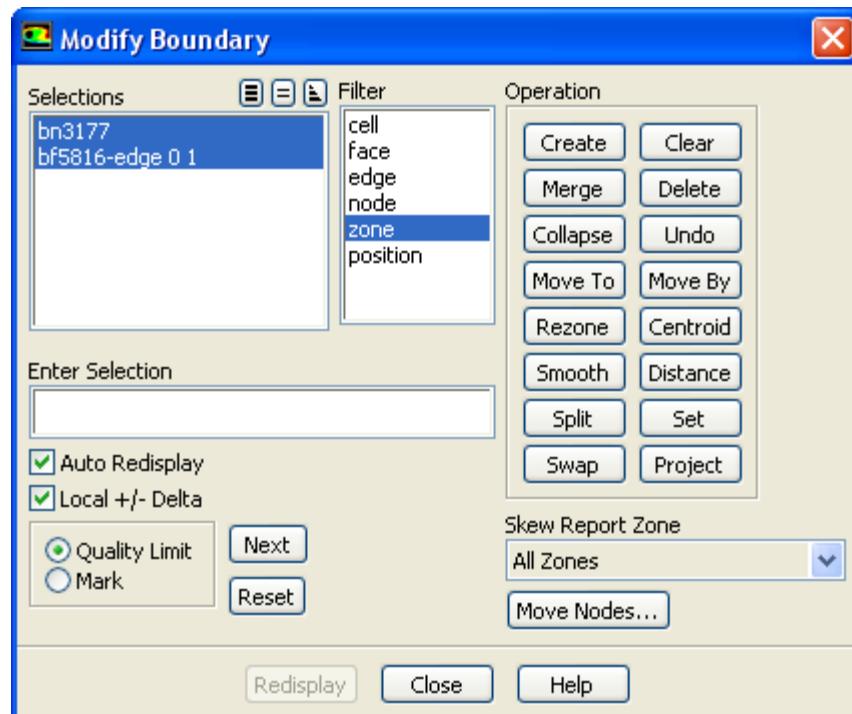
This tutorial also aims at reducing the maximum face skewness below 0.6. This tutorial exposes you to some of the mesh repair tools. Then, it is up to you to try and get the maximum face skewness below 0.6.

## 2.10. Further Improve the Boundary Mesh

This section demonstrates further boundary mesh improvements by merging and smoothing nodes, swapping and splitting edges, and splitting faces.

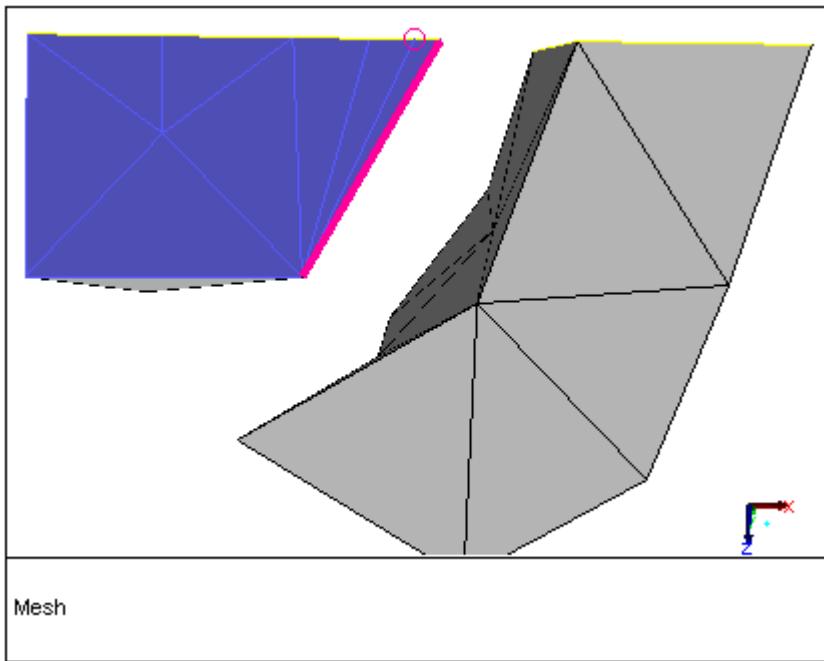
1. Modify the mesh by merging nodes.

**Boundary** → **Modify...**



- a. Retain the selection of **Quality Limit** and click **First**.

The display shows the face having the greatest skewness ([Figure 2.7: Face with the Greatest Skewness \(p. 18\)](#)). You will merge the highlighted node with the corner node to repair the skewed face.

**Figure 2.7: Face with the Greatest Skewness**

---

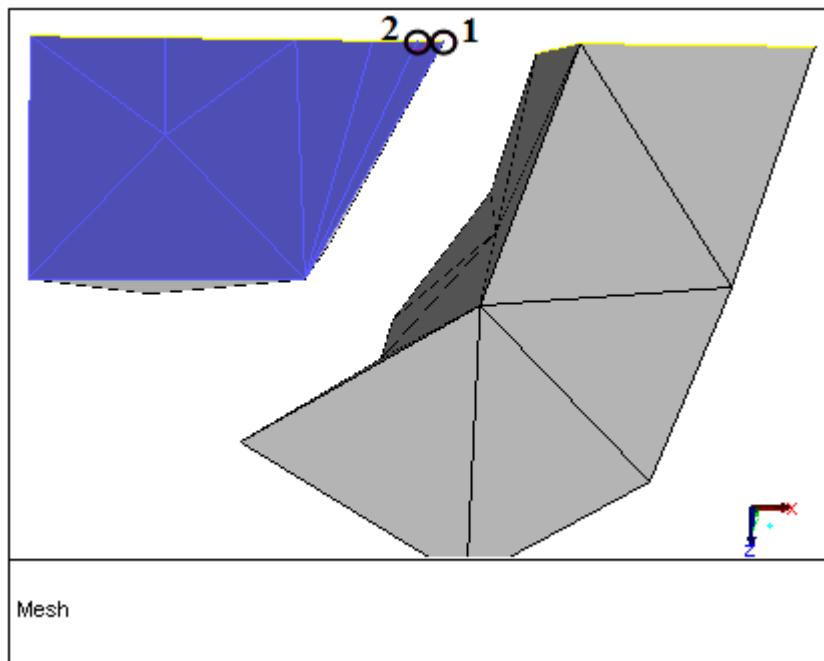
**Tip**

When merging nodes, the first node selected is the one that remains after the merging operation.

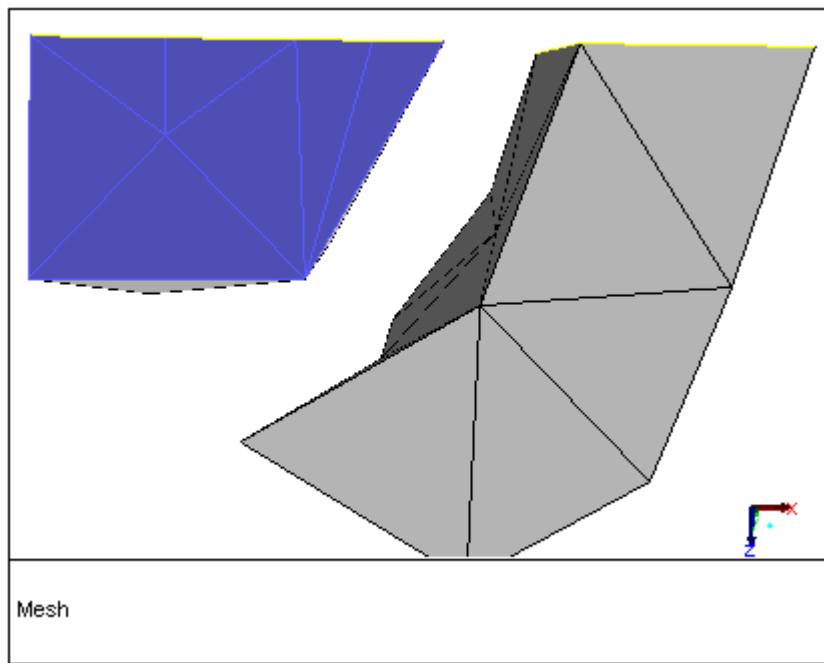
Clear the **Selections** list and select the nodes in the correct order (i.e., first select the corner node, and then select the neighboring node). Merge the two nodes. In this case, the corner node will be retained after merging the nodes, since it was selected first. This procedure is described in the subsequent steps.

---

- b. Click **Clear** in the **Operation** group box.
- c. Select **node** in the **Filter** list.
- d. Select the corner node where the **symmetry** zone meets with the **inlet** zone and the **wall2** zone and the neighboring node (highlighted before the **Selections** list was cleared). See [Figure 2.8: Nodes to be Merged \(p. 19\)](#).

**Figure 2.8: Nodes to be Merged**

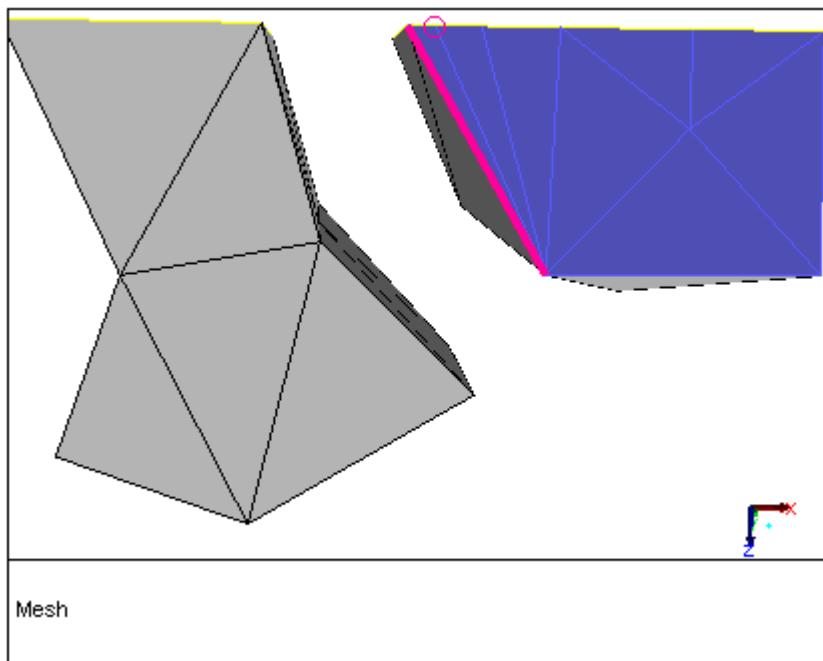
- e. Click **Merge** in the **Operation** group box (Figure 2.9: Surface Mesh After Merging Nodes (p. 19)).

**Figure 2.9: Surface Mesh After Merging Nodes**

2. Repair the next highly skewed face.
  - a. Click **Next** in the **Modify Boundary** dialog box.

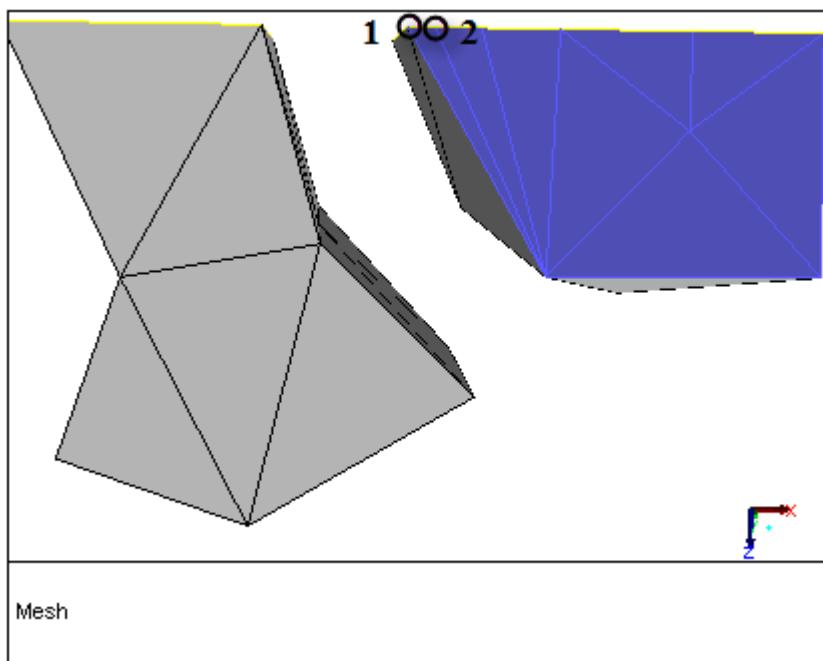
The display shows the face with the next highest skewness ([Figure 2.10: Face with the Next Greatest Skewness \(p. 20\)](#)). The face highlighted is the face on the opposite corner of the **inlet** boundary.

**Figure 2.10: Face with the Next Greatest Skewness**



- b. Clear the **Selections** list.
- c. Select **node** in the **Filter** list.
- d. Select the nodes as shown in [Figure 2.11: Nodes to be Merged \(p. 20\)](#).

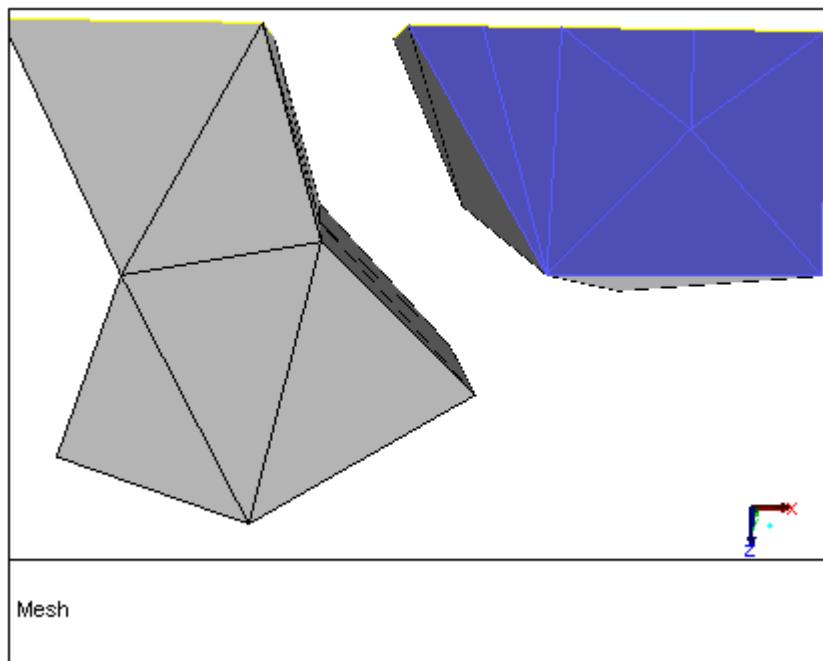
**Figure 2.11: Nodes to be Merged**



- e. Click **Merge**.

The modified mesh after merging the nodes is shown in [Figure 2.12: Surface Mesh After Merging Nodes \(p. 21\)](#).

**Figure 2.12: Surface Mesh After Merging Nodes**



#### Note

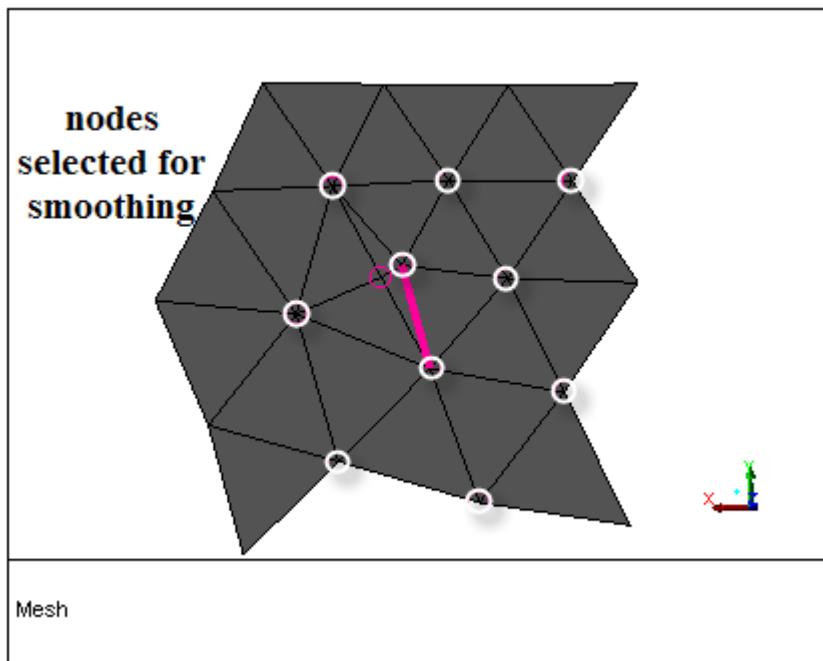
The next two faces that are selected on clicking **Next** can also be modified using the node merging operation. Complete these operations as described earlier.

3. Modify the mesh by smoothing nodes.

- a. Click **Next**.

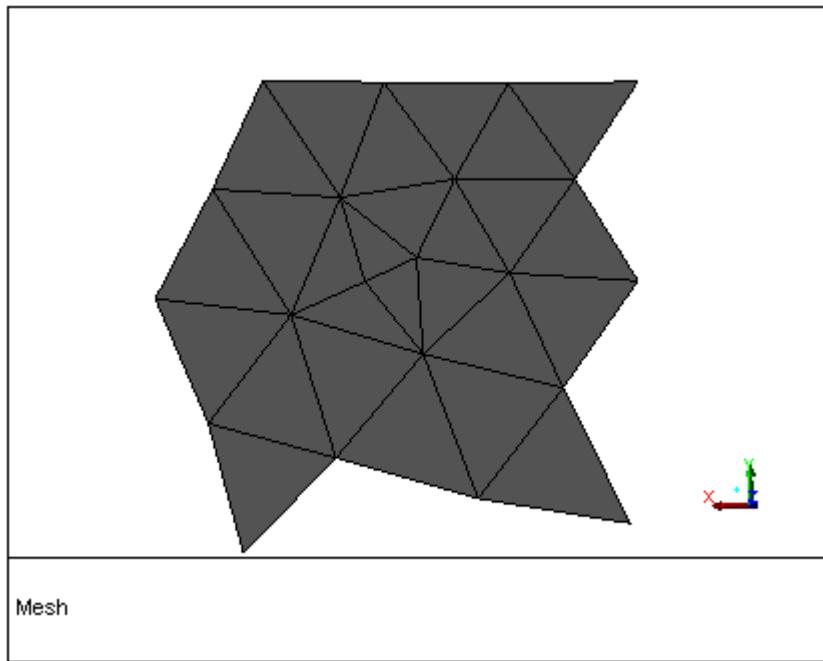
A face located in the middle of one of the internal walls is highlighted ([Figure 2.13: Face to be Modified with Node Smoothing \(p. 22\)](#)).

- b. Select **node** in the **Filter** list.
- c. Select several nodes surrounding the highlighted face (as shown in [Figure 2.13: Face to be Modified with Node Smoothing \(p. 22\)](#)).

**Figure 2.13: Face to be Modified with Node Smoothing**

- d. Click **Smooth** in the **Operation** group box.

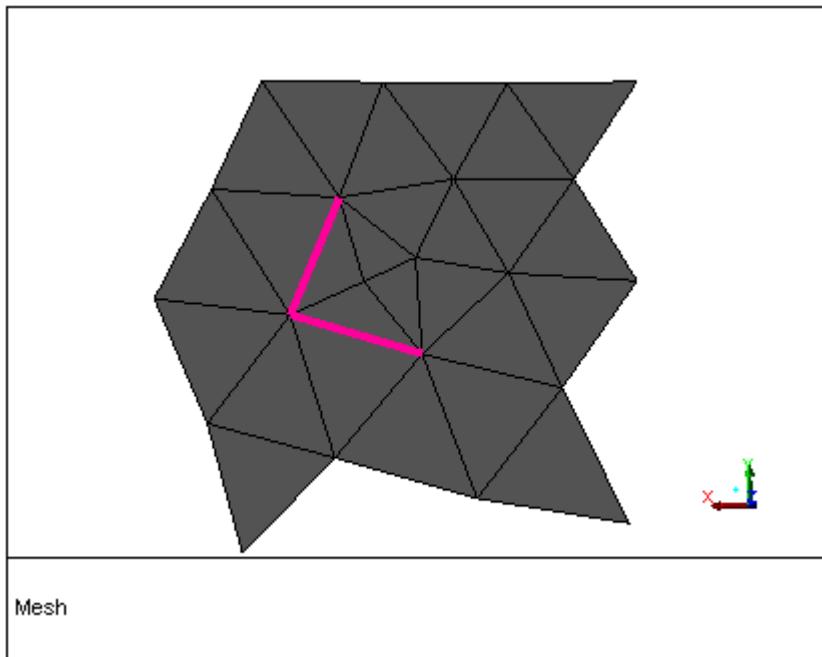
The nodes will be smoothed to make the surrounding faces as uniform in size as possible (see [Figure 2.14: Surface Mesh After Node Smoothing \(p. 22\)](#)).

**Figure 2.14: Surface Mesh After Node Smoothing**

From this point onward, the tutorial attempts to demonstrate some of the additional face modification tools that are available using the cluster of cells shown in [Figure 2.14: Surface Mesh After Node Smoothing \(p. 22\)](#).

4. Modify the mesh by edge swapping.
  - a. Select **edge** in the **Filter** list.
  - b. Select the edges to be swapped ([Figure 2.15: Edges Selected for Swapping \(p. 23\)](#)).

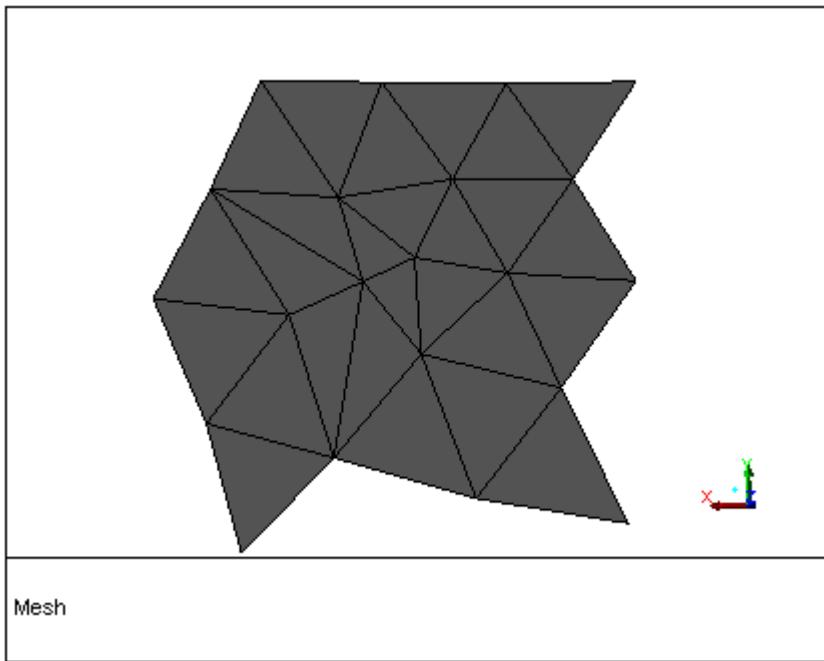
**Figure 2.15: Edges Selected for Swapping**



- c. Click **Swap** in the **Operation** group box.

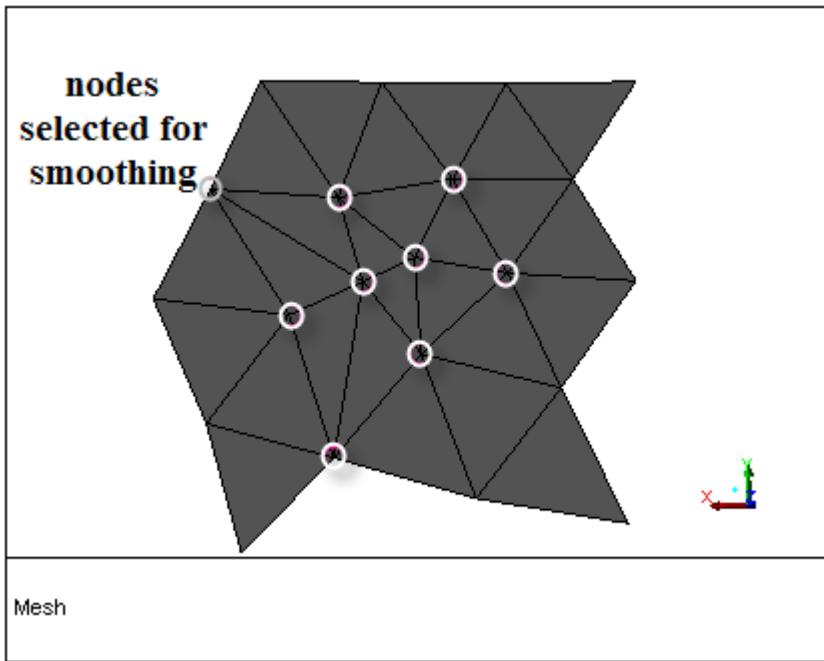
The selected edges will be swapped and the mesh retriangulated ([Figure 2.16: Surface Mesh After Edge Swapping \(p. 24\)](#)). This operation did little to produce a better quality mesh. You can use node smoothing to fix this problem.

**Figure 2.16: Surface Mesh After Edge Swapping**

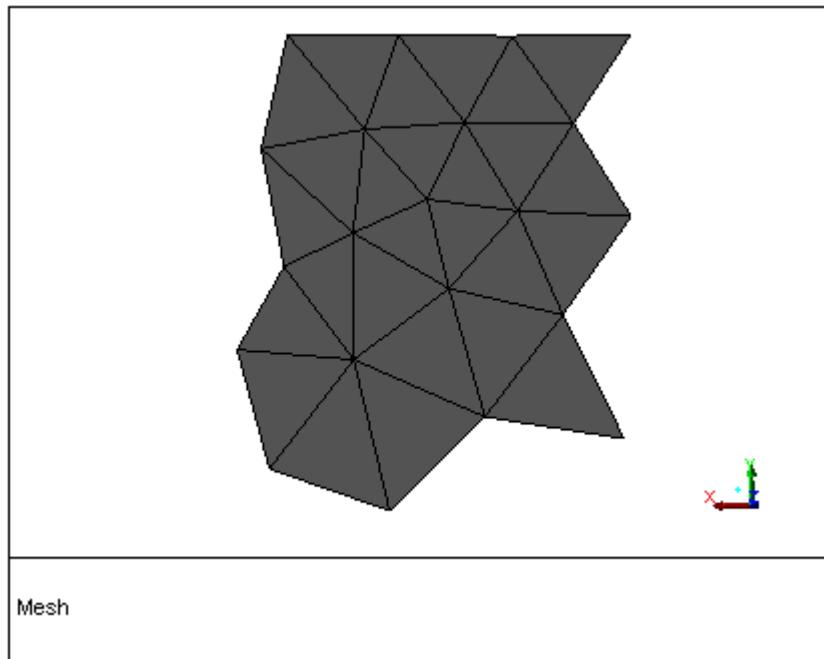


- d. Select **node** in the **Filter** list.
- e. Select the nodes in the vicinity of the swapped edge (Figure 2.17: Nodes Selected for Smoothing (p. 24)).

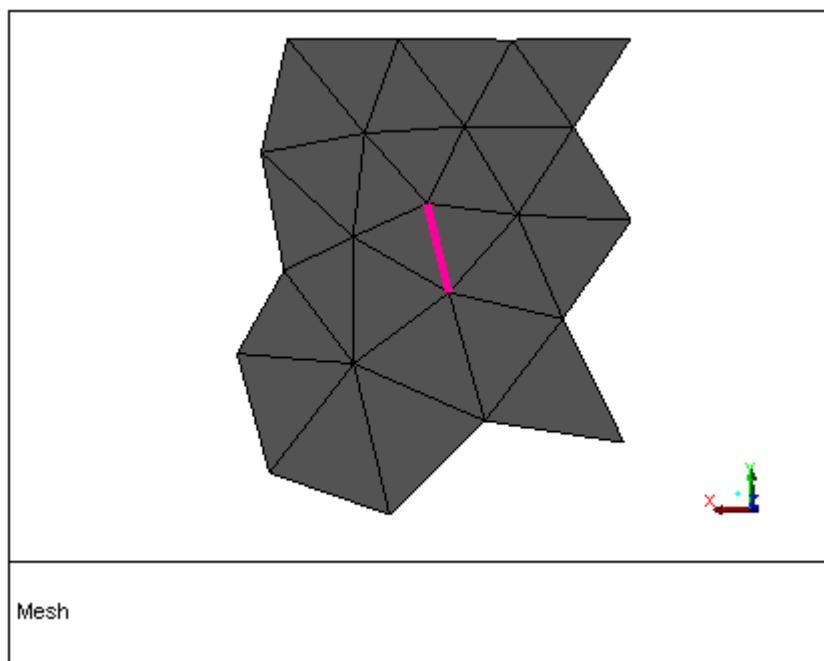
**Figure 2.17: Nodes Selected for Smoothing**



- f. Click **Smooth** in the **Operation** group box (Figure 2.18: Surface Mesh After Node Smoothing (p. 25)).

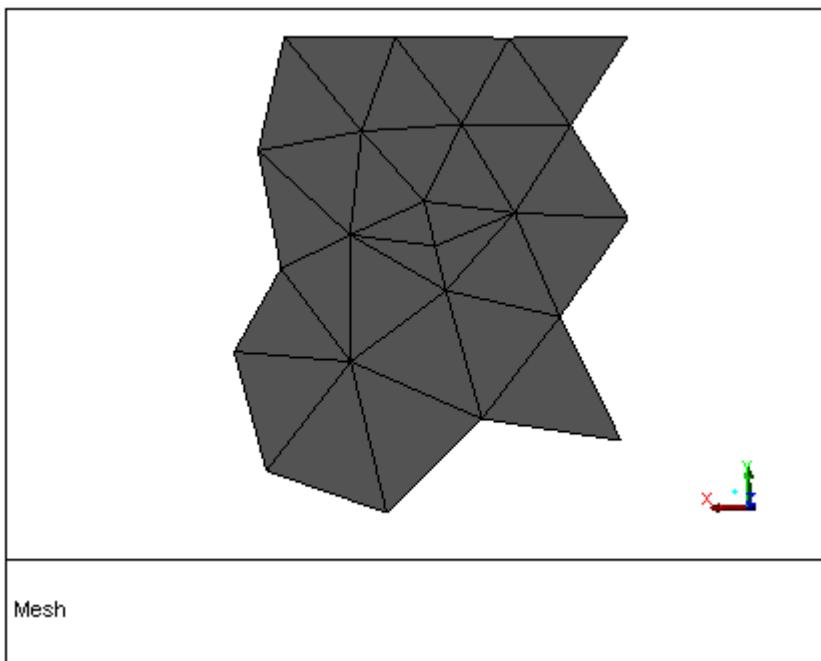
**Figure 2.18: Surface Mesh After Node Smoothing**

5. Modify the mesh by splitting edges.
  - a. Select **edge** in the **Filter** list.
  - b. Select the edge to be split ([Figure 2.19: Edge Selected for Splitting \(p. 25\)](#)).

**Figure 2.19: Edge Selected for Splitting**

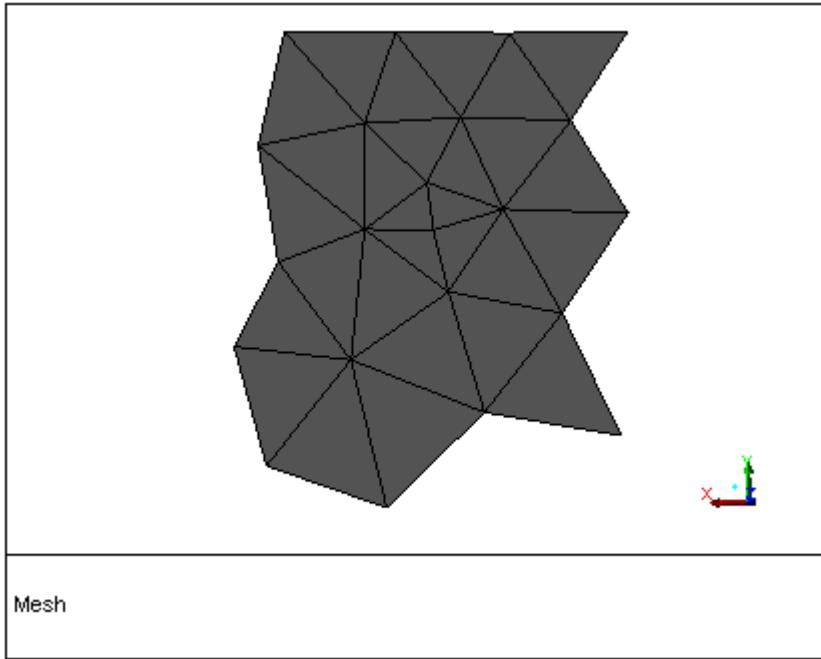
- c. Click **Split** in the **Operation** group box ([Figure 2.20: Surface Mesh After Edge Splitting \(p. 26\)](#)).

**Figure 2.20: Surface Mesh After Edge Splitting**



- d. Perform node smoothing by selecting several nodes around the split edge and clicking **Smooth** (Figure 2.21: Surface Mesh After Node Smoothing (p. 26)).

**Figure 2.21: Surface Mesh After Node Smoothing**

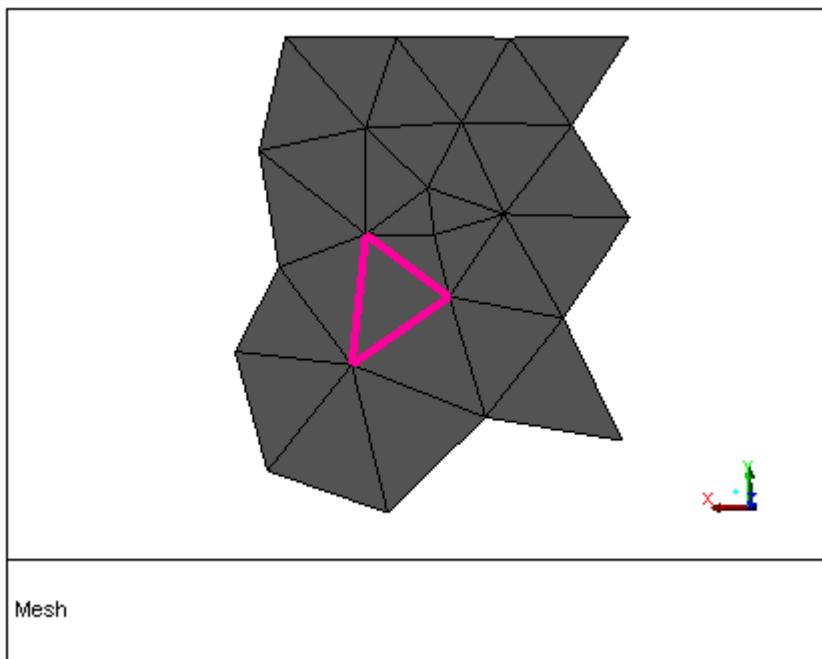


6. Modify the mesh by splitting faces.

- a. Select **face** in the **Filter** list.

- b. Select the face to be split (Figure 2.22: Face Selected for Splitting (p. 27)).

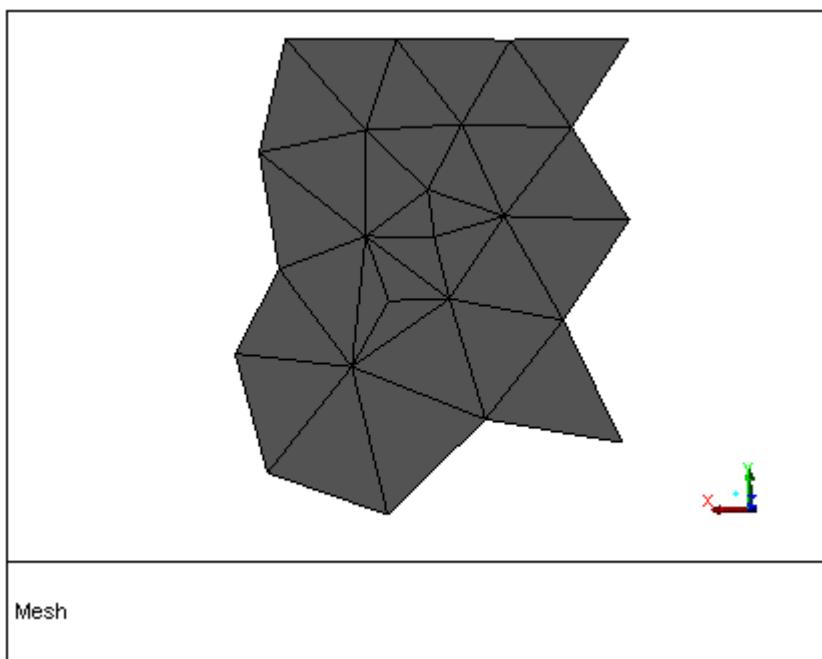
**Figure 2.22: Face Selected for Splitting**



Mesh

- c. Click **Split** in the **Operation** group box to split the face (Figure 2.23: Surface Mesh After Splitting the Face (p. 27)).

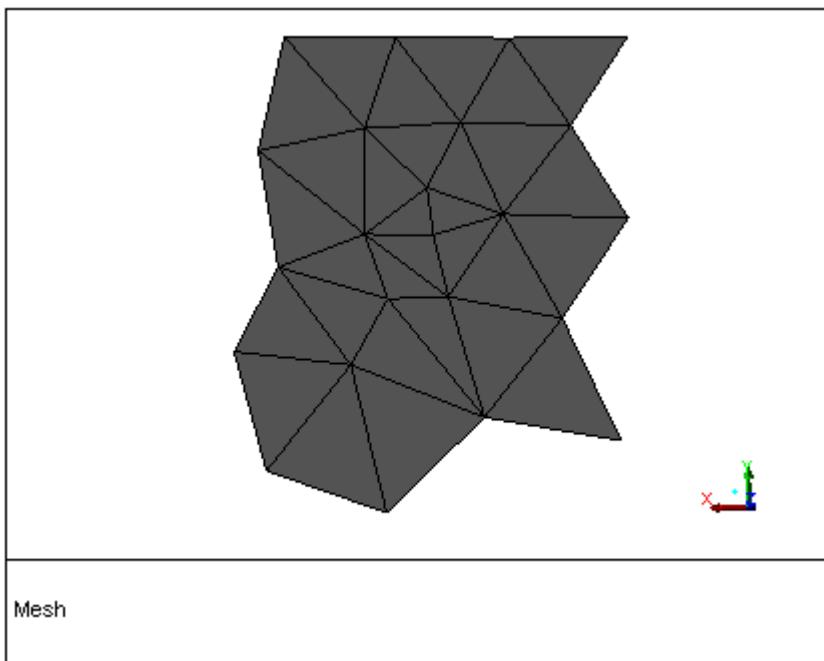
**Figure 2.23: Surface Mesh After Splitting the Face**



Mesh

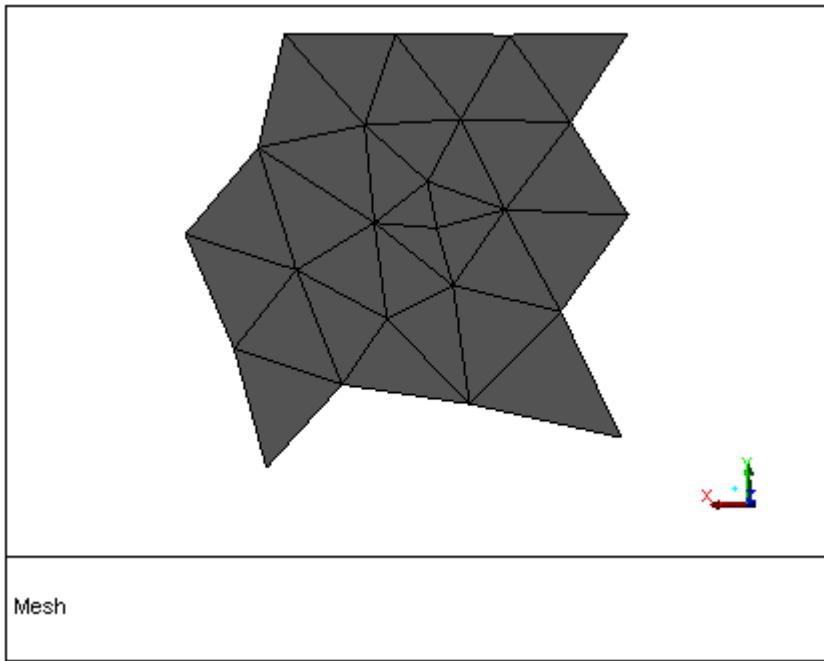
- d. Swap the edges of the split face (Figure 2.24: Surface Mesh After Edge Swapping (p. 28)).

**Figure 2.24: Surface Mesh After Edge Swapping**



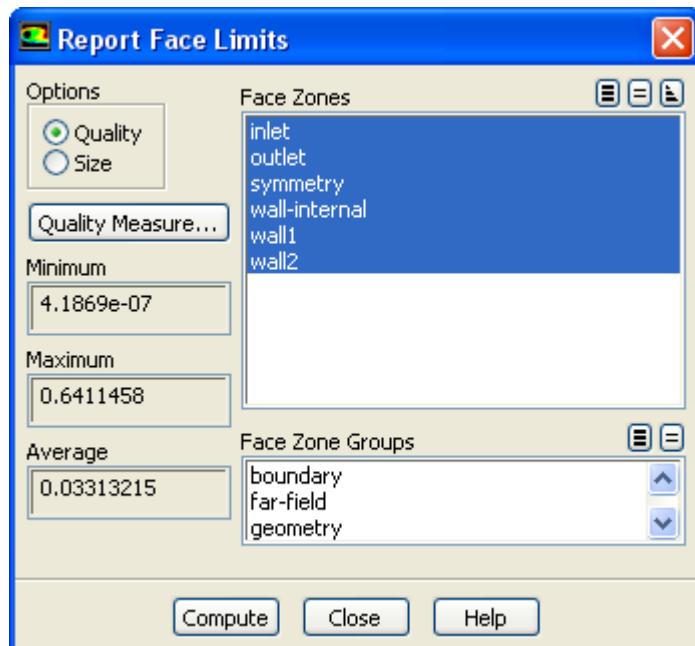
- e. Smooth the nodes in the vicinity of the split face ([Figure 2.25: Surface Mesh After Node Smoothing \(p. 28\)](#)).

**Figure 2.25: Surface Mesh After Node Smoothing**



7. Check the maximum face skewness.

**Report → Face Limits...**



- Select **Quality** in the **Options** list.
- Select all the zones in the **Face Zones** selection list.
- Click **Compute**.

The **Minimum**, **Maximum**, and **Average** face skewness will be reported.

- Close the **Report Face Limits** dialog box.

The maximum face skewness at this point in the tutorial is less than 0.65. There are nine faces with a skewness greater than 0.6 (this information was obtained from the **Face Distribution** dialog box). You can try and reduce the maximum face skewness to a value less than 0.6 using the face modification tools described previously.

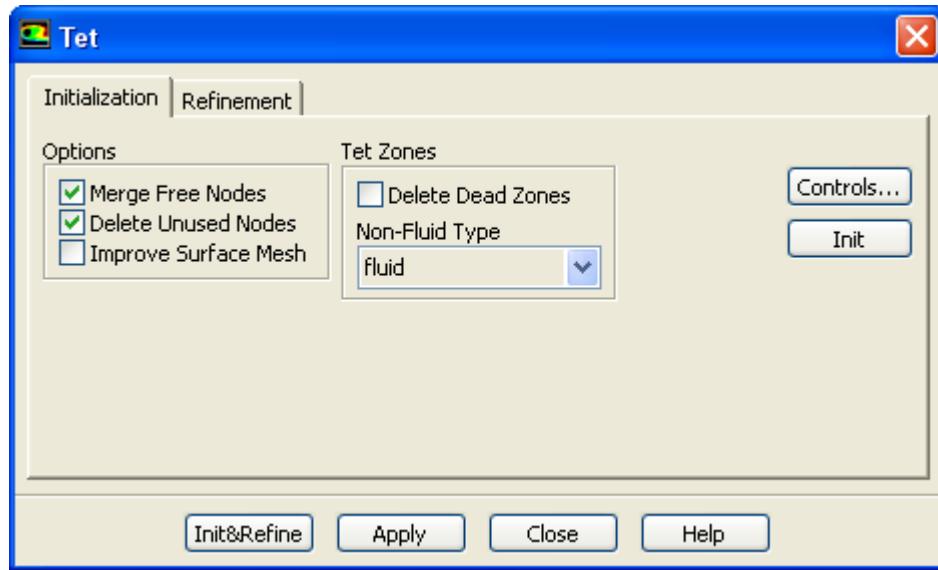
- Close the **Modify Boundary** dialog box.

## 2.11. Generate a Multiple Region Volume Mesh

There are multiple regions in this mesh (four to be exact). To mesh the whole domain, you need to change the non-fluid type declaration to **fluid** in the **Initialization** tab of the **Tet** dialog box and then generate the volume mesh.

- Change the **Non-Fluid Type** from **dead** to **fluid**.

**Mesh → Tet...**



- a. Select **fluid** in the **Non-Fluid Type** drop-down list in the **Tet Zones** group box.
- b. Click **Apply** and close the **Tet** dialog box.

### Note

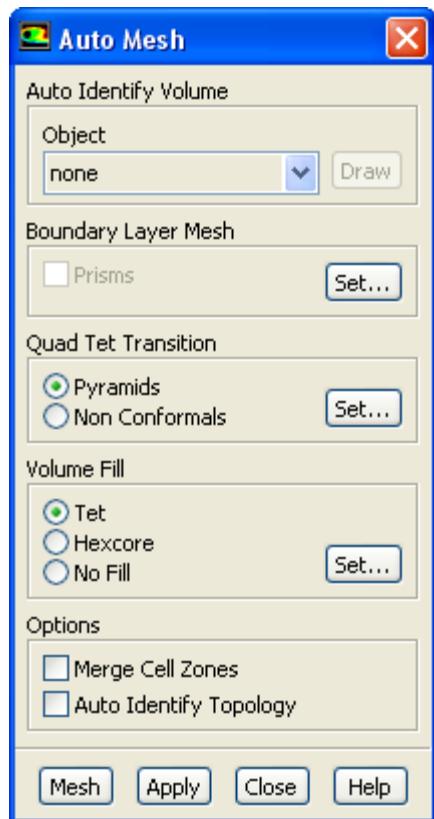
By default, the cell zone with the largest volume is automatically made the active fluid zone. The remaining cell zones (non-fluid zones) are treated as dead zones and not refined. Hence, if you want to mesh multiple zones, change the **Non-Fluid Type** to **solid** or **fluid** depending on the problem.

When **Non-Fluid Type** is set to a type other than **dead**, all the zones are treated as active zones and automatically refined.

If the mesh has only one zone, this operation is not necessary.

2. Generate the volume mesh.

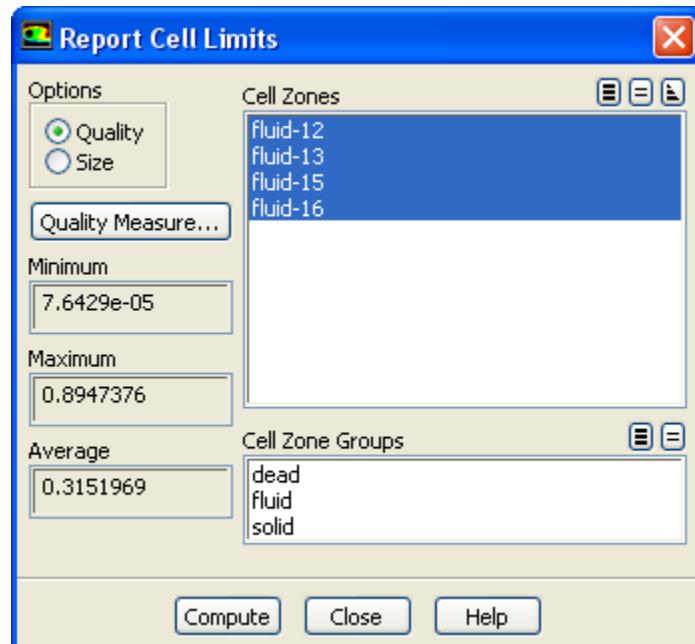
**Mesh → Auto Mesh...**



- Retain the default settings and click **Mesh**.
- Close the **Auto Mesh** dialog box.

## 2.12. Check the Volume Mesh Quality

**Report** → **Cell Limits...**



1. Select all the zones in the **Cell Zones** selection list.
2. Click **Compute** to report the **Maximum**, **Minimum**, and **Average** cell skewness values.
3. Close the **Report Cell Limits** dialog box.

## 2.13. Check and Save the Volume Mesh

1. Check the mesh.

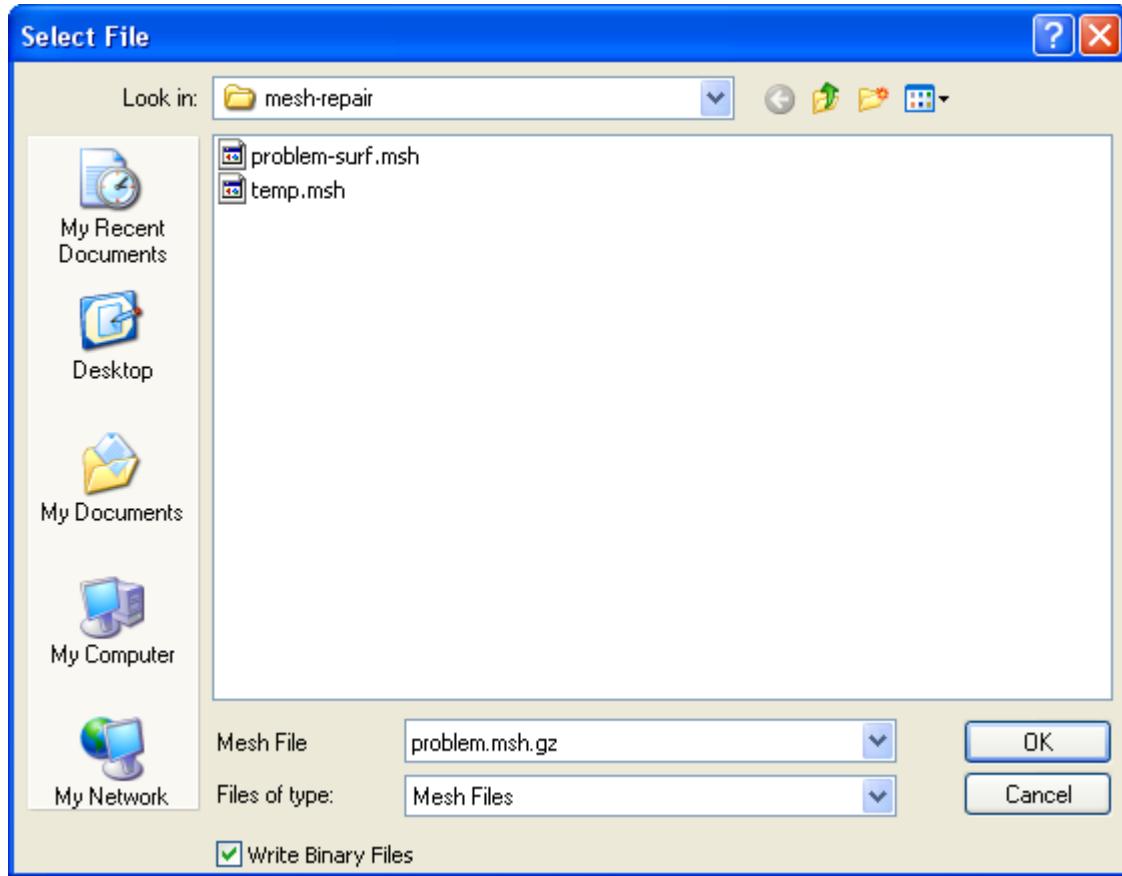
Check the mesh to ensure it has no negative cell volumes or left-handed faces before saving the mesh file.

### Mesh → Check

The printed results of the check show no problems, hence the mesh is valid for use in the solver.

2. Save the mesh.

### File → Write → Mesh...



- Enter **problem.msh.gz** for **Mesh File** and click **OK**.
3. Exit ANSYS FLUENT.

### File → Exit

## 2.14. Summary

This tutorial demonstrated the use of some mesh repair tools available to fix known deficiencies in an existing boundary mesh.



---

## Chapter 3: Tetrahedral Mesh Generation

---

The mesh generation process is highly automated in ANSYS FLUENT. In most cases, you can use the **Auto Mesh** feature to create the volume mesh from the surface mesh. However, in some cases, the boundary mesh may contain irregularities or highly skewed boundary faces that can lead to an unacceptable volume mesh or cause a failure while generating the initial mesh. As a rule of thumb, you need to check the boundary mesh before attempting to generate the volume mesh. This tutorial demonstrates how to do the following:

- Create a user-defined group for easier selection of boundary surfaces.
- Generate the tetrahedral volume mesh using the various refinement options available.
- Compare the mesh generated using the skewness-based and advancing front refinement methods.
- Examine the effect of the size function.
- Examine the effect of the growth factor.
- Create a local refinement region.

### 3.1. Prerequisites

This tutorial assumes that you have little experience with the meshing mode in ANSYS FLUENT, but that you are familiar with the graphical user interface.

### 3.2. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

To prepare for running this tutorial:

1. Download the tutorial input file (`tet-mesh.zip`) for the tutorial.
2. Unzip `tet-mesh.zip`.

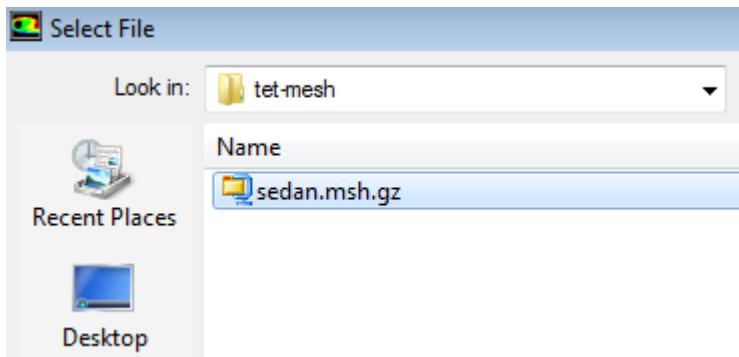
The file `sedan.msh.gz` can be found in the `tet-mesh` folder created on unzipping the file.

3. Start ANSYS FLUENT in meshing mode. For detailed steps, refer to [Starting ANSYS FLUENT in Meshing Mode \(p. 4\)](#).

### 3.3. Read and Display the Boundary Mesh

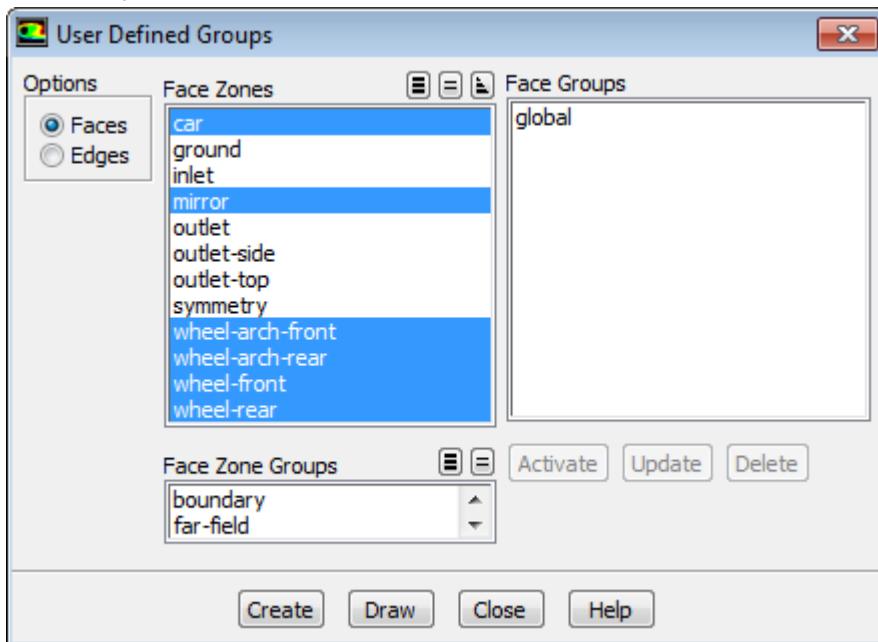
1. Read the boundary mesh.

**File → Read → Boundary Mesh...**



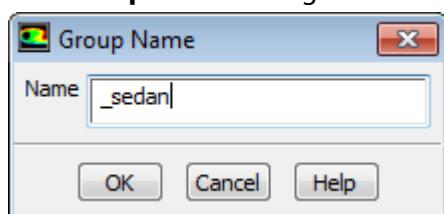
- a. Select **sedan.msh.gz**.
- b. Click **OK**.
2. Create a user-defined group for easier selection of the surfaces defining the sedan.

**Boundary → Zone → Group...**



- a. Select **car**, **mirror**, **wheel-arch-front**, **wheel-arch-rear**, **wheel-front**, and **wheel-rear** from the **Face Zones** selection list.
- b. Click **Create**.

The **Group Name** dialog box will open, prompting you to specify the group name.



- c. Enter **\_sedan** for **Name** and click **OK** to close the **Group Name** dialog box.

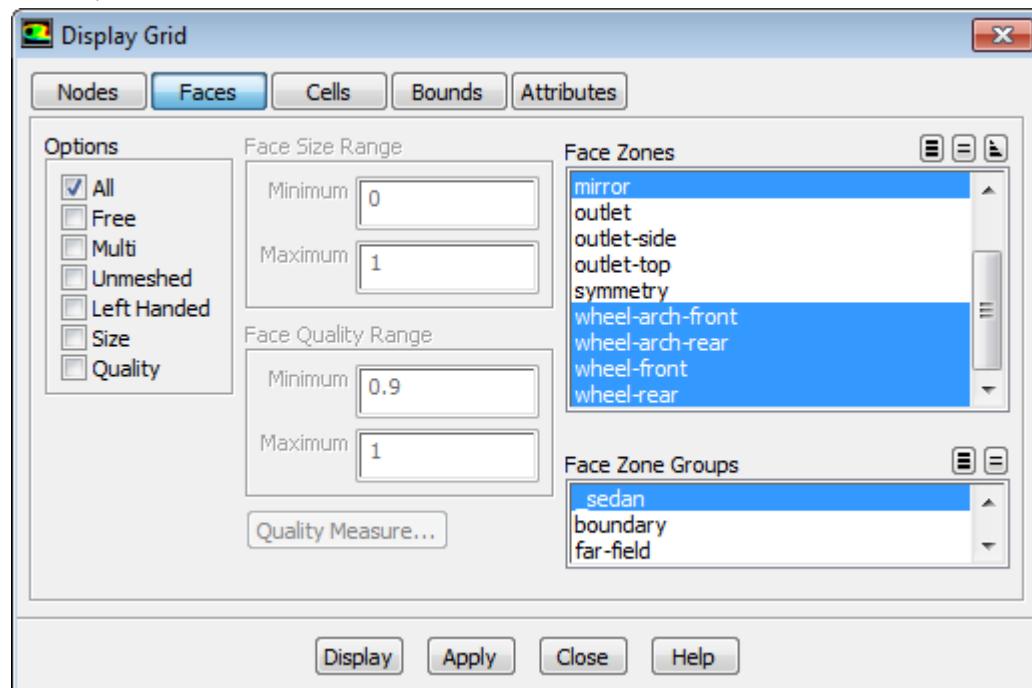
The **\_sedan** group will now be available in the **Face Groups** list in the **User Defined Groups** dialog box.

### Tip

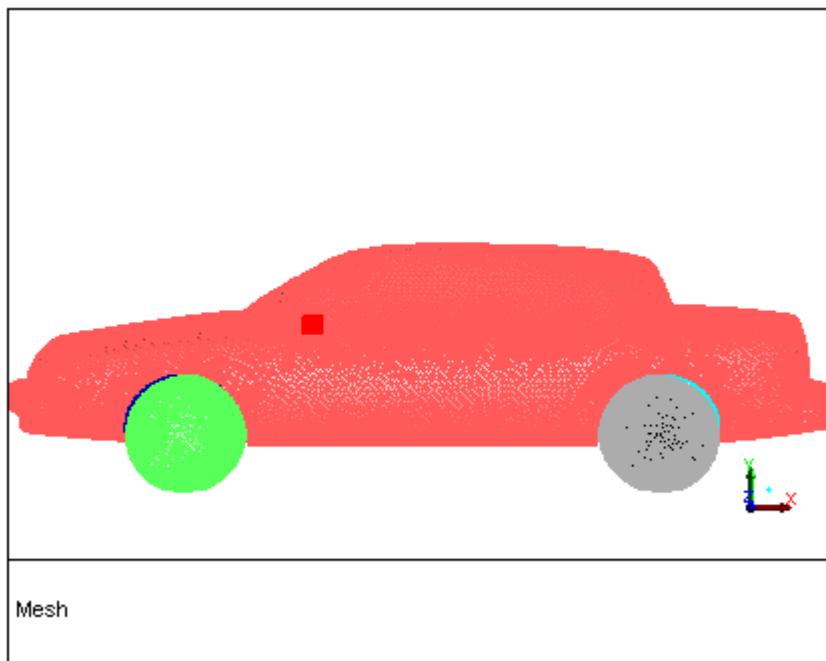
The use of the underscore (\_) in the group name allows the group to be listed at the top of the **Face Zone Groups** list in the respective dialog boxes.

- d. Close the **User Defined Groups** dialog box.
3. Display the boundary mesh ([Figure 3.1: Grid Display \(p. 38\)](#)).

### Display → Grid...



- a. Select **\_sedan** in the **Face Zone Groups** selection list to select all the boundary zones defining the car in the **Face Zones** selection list.
- b. Click the **Attributes** tab and enable **Filled** and **Lights**.
- c. Click the **Colors...** button to open the **Grid Colors** dialog box.
  - i. Select **Color by ID** in the **Options** list.
  - ii. Close the **Grid Colors** dialog box.
- d. Click **Display**.

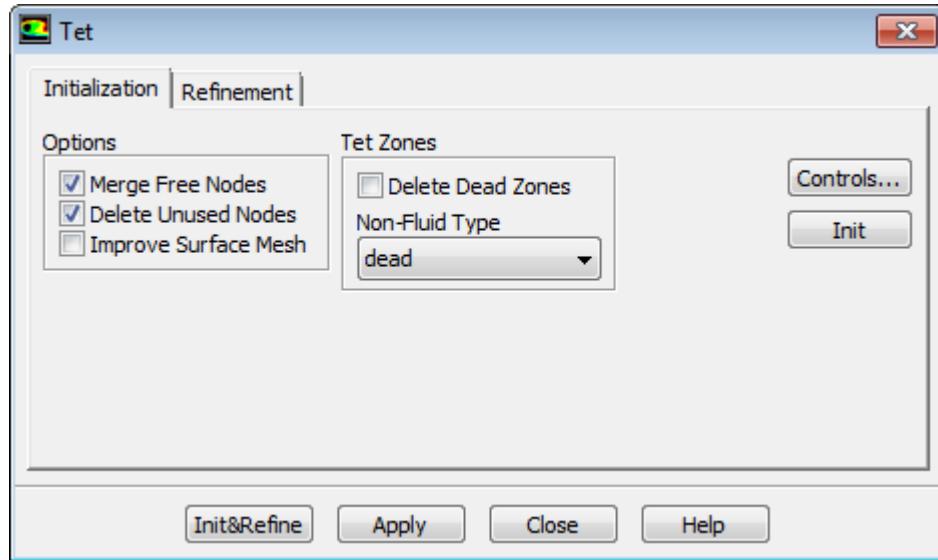
**Figure 3.1: Grid Display**

- e. Close the **Display Grid** dialog box.

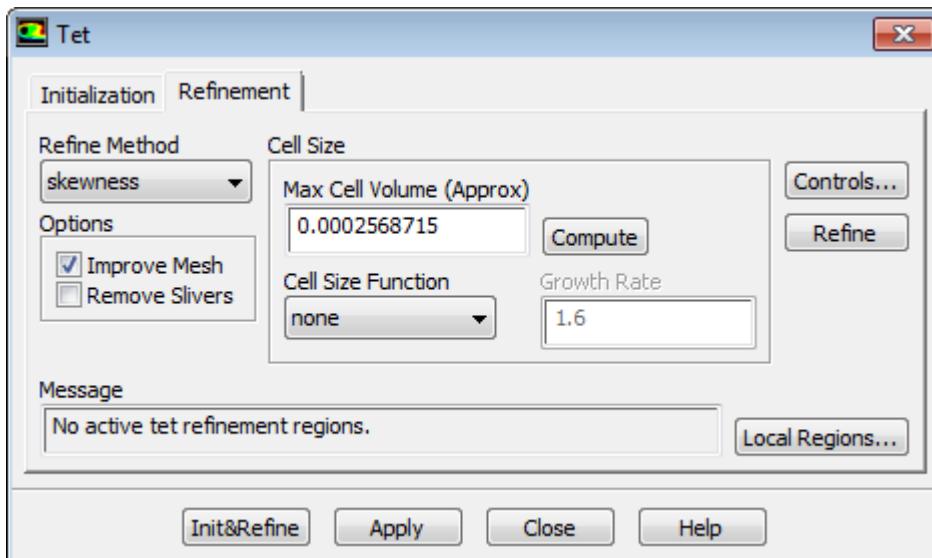
### 3.4. Generate the Mesh using the Skewness-Based Refinement Method

1. Specify the meshing parameters.

Mesh → Tet...



- a. Retain the default settings in the **Initialization** tab.
- b. Click the **Refinement** tab.



- i. Select **skewness** in the **Refine Method** drop-down list.
- ii. Select **none** in the **Cell Size Function** drop-down list.
- iii. Retain the default value ( $2.57 \times 10^{-4}$ ) for **Max Cell Volume**.

The default value for maximum cell volume is calculated as the volume of an ideal equilateral tetrahedron with edge length equal to the length of the longest edge in the domain.

You can use the following commands to verify the value:

- A. `/report/edge-size-limits` to obtain the minimum, maximum, and average edge length.
- B. `/mesh/tet/local-regions/ideal-vol` to calculate the volume of an ideal equilateral tetrahedron ( $\text{side} \times \text{side} \times \text{side} \times \frac{\sqrt{2}}{12}$ ) with side equal to the maximum edge length.

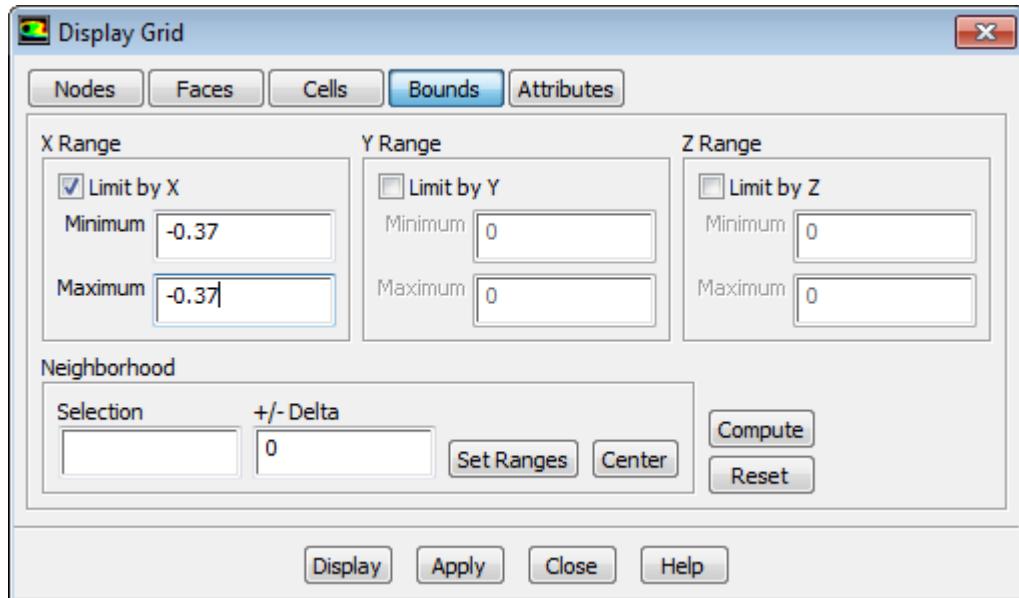
The longest edge may be connected to shorter edges and not characteristic of the facet or maximum volume value. Hence, you may want to measure the length of some of the edges on the outer boundary and then calculate the ideal volume. Select two nodes on the edge and use the hot-key **Ctrl + D** to obtain the edge length.

In this case, an edge length of 0.1 would seem appropriate (giving a volume of  $1.18 \times 10^{-4}$ ), but the mesh would be larger and slower to generate. Hence, you will use the default value for the maximum cell volume.

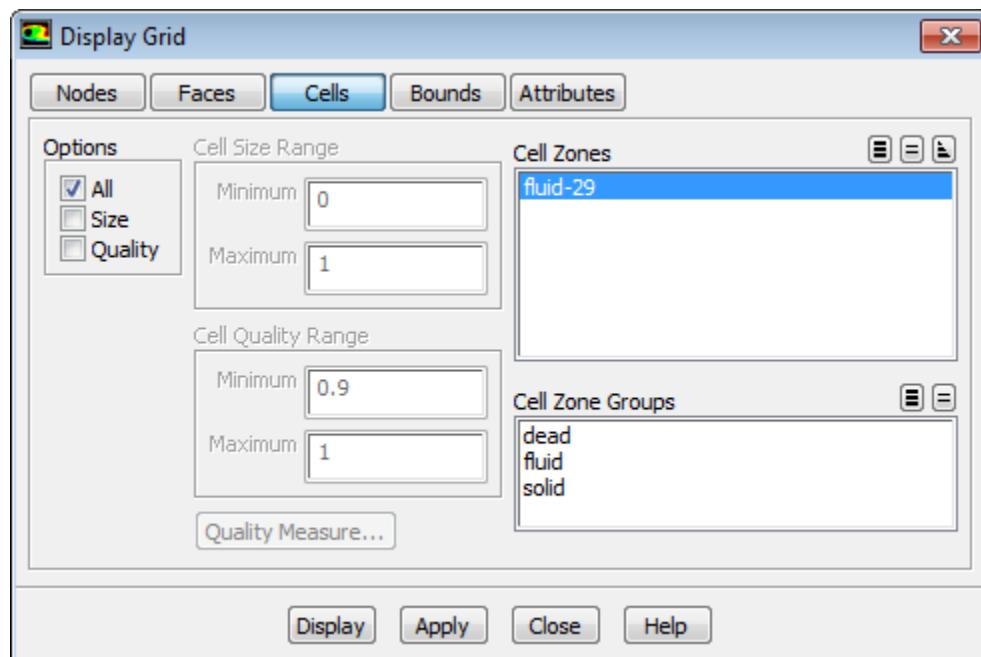
- c. Click **Apply** and **Init & Refine**.
  - d. Close the **Tet** dialog box.
2. Examine the mesh.

**Display → Grid...**

- a. Display the mesh on a slide through the mirror and the car (Figure 3.2: Slide of Cells at X = -0.37 (p. 41)).

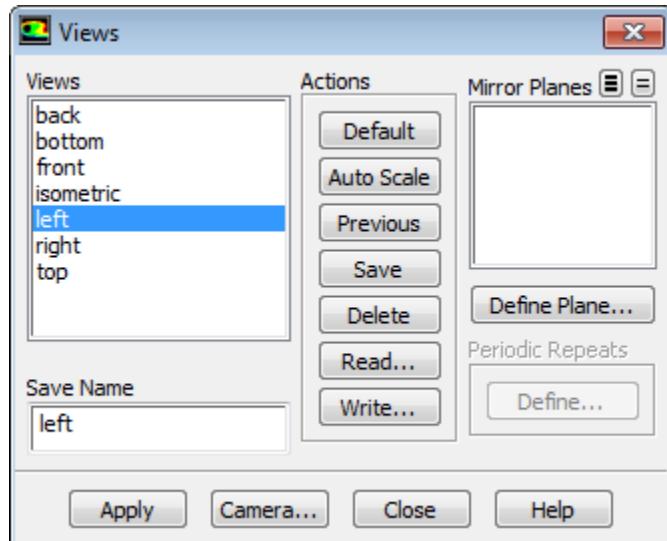


- Click the **Bounds** tab and enable **Limit by X**.
- Enter  $-0.37$  for **Minimum** and **Maximum** in the **X Range** group box.
- Click the **Cells** tab and select the fluid zone in the **Cell Zones** selection list.



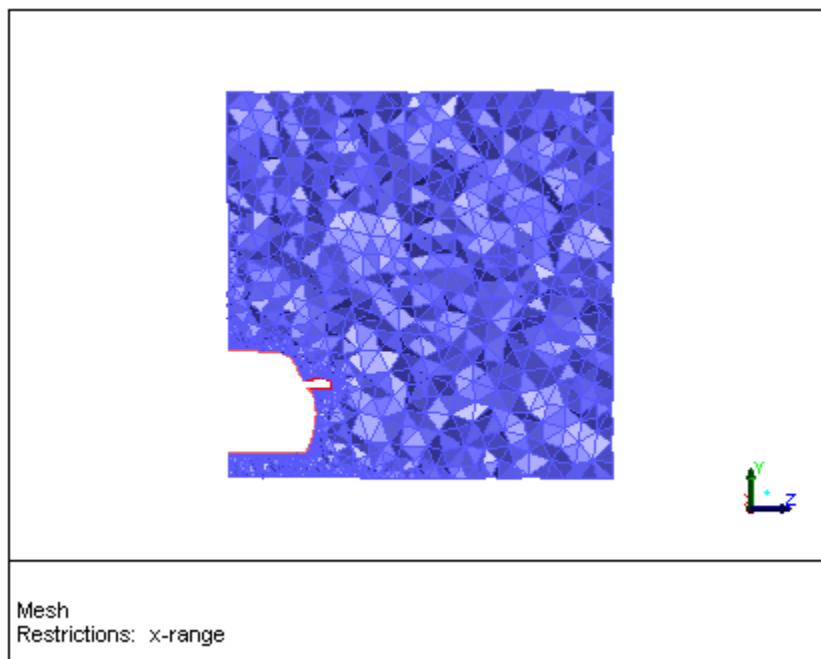
- Enable **All** in the **Options** group box and click **Display**.
- Display the **left** view.

**Display → Views...**



- A. Select **left** in the **Views** list and click **Apply**.
- B. Click **Auto Scale**.
- C. Close the **Views** dialog box.

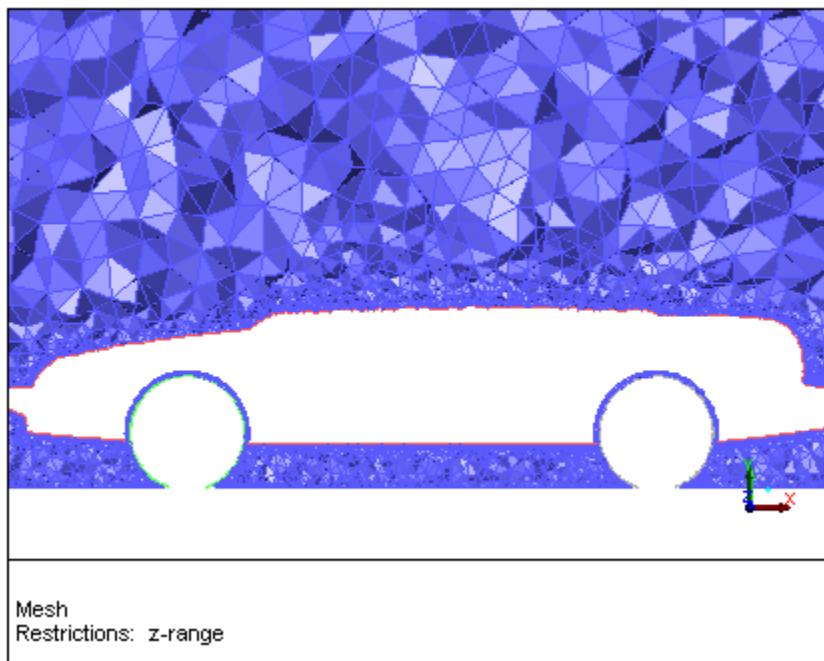
**Figure 3.2: Slide of Cells at X = -0.37**



- b. Display the mesh on a slide through the wheels (Figure 3.3: Slide of Cells at Z = 0.38 (p. 42)).
- i. Click **Reset** in the **Bounds** tab of the **Display Grid** dialog box.
  - ii. Enable **Limit by Z** and enter 0.38 for **Minimum** and **Maximum** in the **Z Range** group box.
  - iii. Click **Display** and display the **front** view.

- iv. Zoom in to the sedan to examine the cell growth.

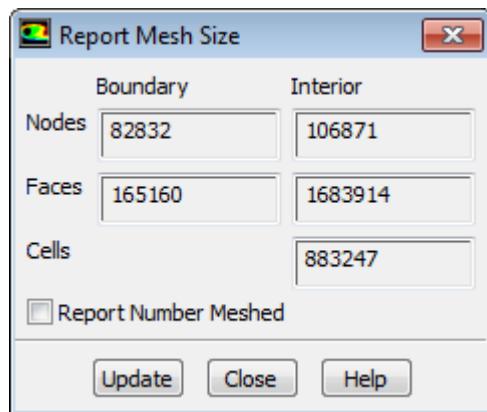
**Figure 3.3: Slide of Cells at Z = 0.38**



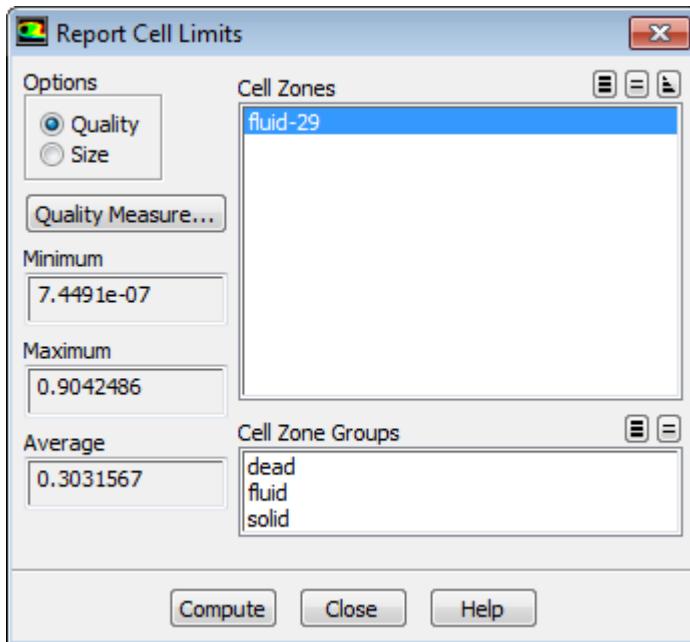
You can see that the cells inside the domain are not larger than those on the outer boundary.

- v. Close the **Display Grid** dialog box.
- c. Check the number of cells.

**Report → Mesh Size...**



- i. Click **Update**.
- The number of cells is 883247. The exact number may differ on different platforms.
- ii. Close the **Report Mesh Size** dialog box.
- d. Check the maximum skewness.

**Report → Cell Limits...**

- i. Select the fluid zone in the **Cell Zones** selection list.
- ii. Click **Compute**.

The maximum skewness is 0.904, which is acceptable. The average skewness is 0.303.

- iii. Close the **Report Cell Limits** dialog box.

### 3.5. Generate the Mesh using the Skewness-Based Refinement Method and a Size Function

1. Delete the previous volume mesh.

**Mesh → Clear**

2. Specify the meshing parameters.

**Mesh → Tet...**

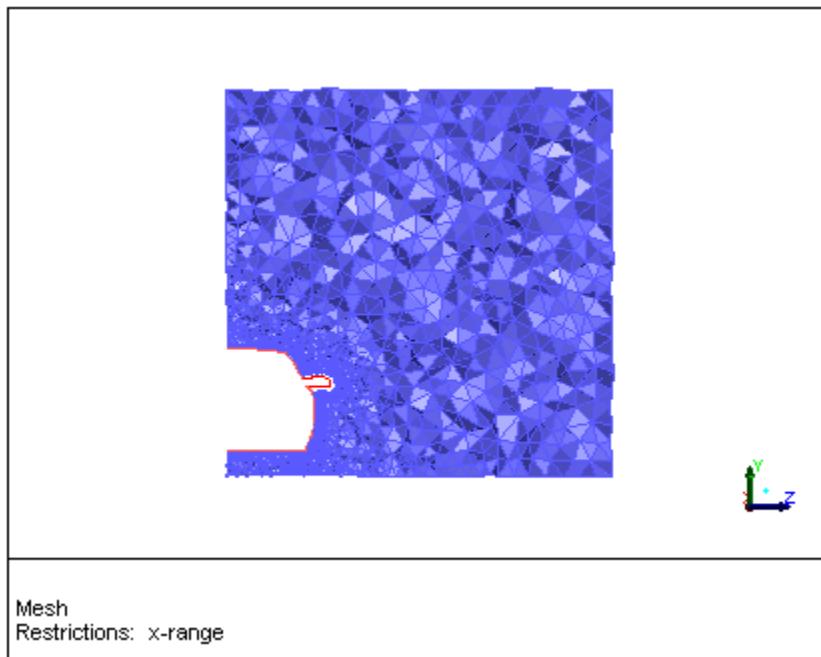
- a. Retain the settings in the **Initialization** tab.
- b. Click the **Refinement** tab.
  - i. Retain the selection of **skewness** in the **Refine Method** drop-down list.
  - ii. Select **geometric** in the **Cell Size Function** drop-down list and enter **1 . 3** for **Growth Rate**.
  - iii. Retain the default value (**2 . 57e-4**) for **Max Cell Volume**.
  - iv. Click **Apply** and **Init & Refine**.
  - v. Close the **Tet** dialog box.

- c. Examine the mesh.

**Display → Grid...**

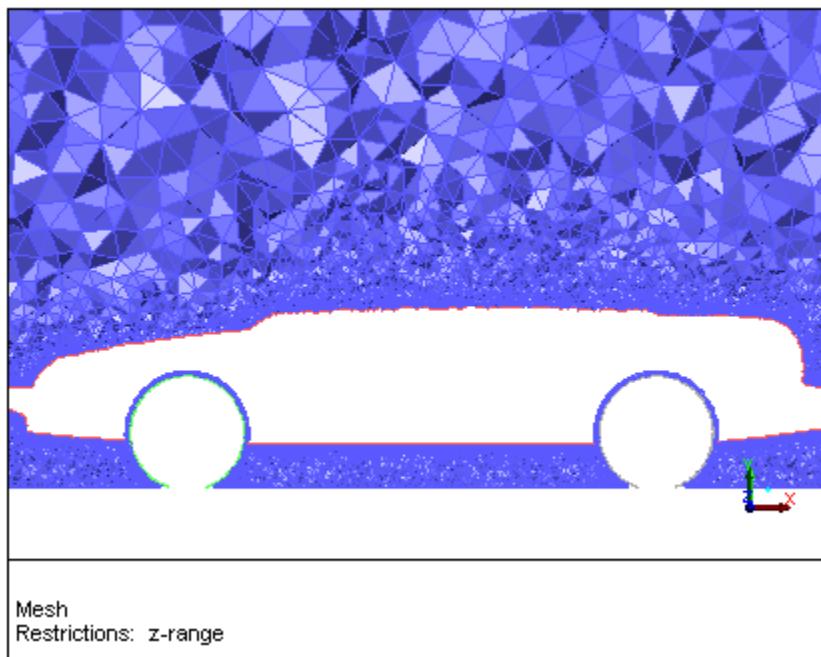
- i. Display the mesh on a slide through the mirror and the car ( $x = -0.37$ ). See [Figure 3.4: Slide of Cells at X = -0.37 \(p. 44\)](#).

**Figure 3.4: Slide of Cells at X = -0.37**



- ii. Display the mesh on a slide through the wheels ( $z = 0.38$ ). See [Figure 3.5: Slide of Cells at Z = 0.38 \(p. 44\)](#).

**Figure 3.5: Slide of Cells at Z = 0.38**



- d. Check the number of cells.

**Report → Mesh Size...**

The number of cells is 1657410. The exact number may differ on different platforms.

- e. Check the maximum skewness.

**Report → Cell Limits...**

The maximum skewness is 0.904, which is acceptable. The average skewness is 0.248.

You can see that the transition between small and large cells is smoother than that for the previous mesh. The transition is smoother when the specified growth rate is closer to 1.

## 3.6. Generate the Mesh using the Advancing Front Refinement Method and a Size Function

1. Delete the previous volume mesh.

**Mesh → Clear**

2. Specify the meshing parameters.

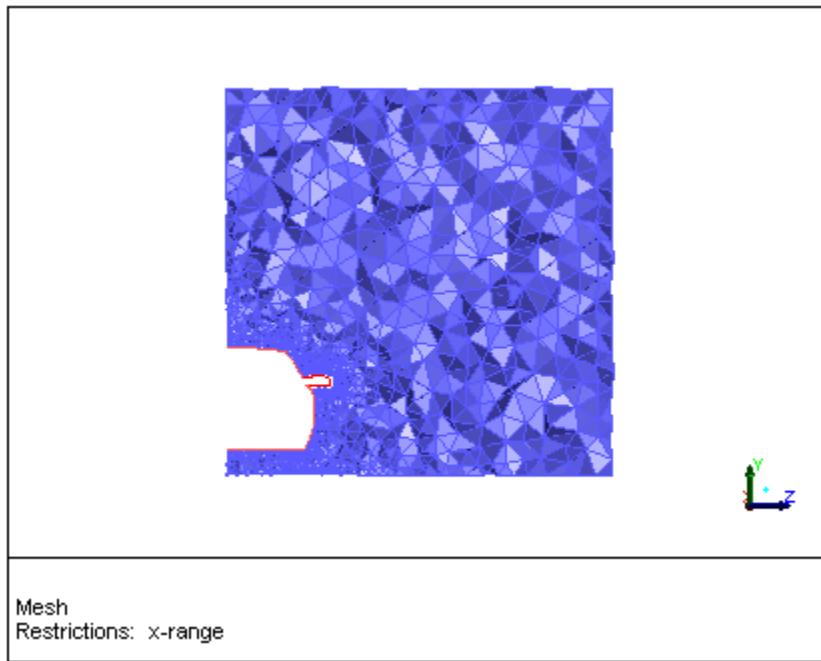
**Mesh → Tet...**

- a. Retain the settings in the **Initialization** tab.
- b. Click the **Refinement** tab.
  - i. Select **adv-front** in the **Refine Method** drop-down list.
  - ii. Retain the selection of **geometric** in the **Cell Size Function** drop-down list and retain 1.3 for **Growth Rate**.
  - iii. Retain the default value (2.57e-4) for **Max Cell Volume**.
  - iv. Click **Apply** and **Init & Refine**.
  - v. Close the **Tet** dialog box.
- c. Examine the mesh.

**Display → Grid...**

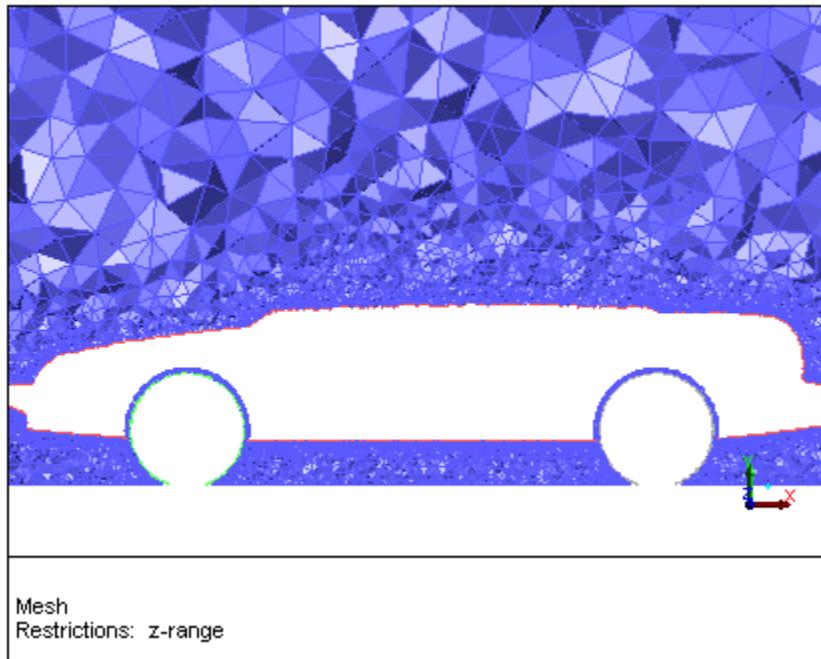
- i. Display the mesh on a slide through the mirror and the car ( $x = -0.37$ ). See [Figure 3.6: Slide of Cells at X = -0.37 \(p. 46\)](#).

**Figure 3.6: Slide of Cells at X = -0.37**



- ii. Display the mesh on a slide through the wheels ( $z = 0.38$ ). See [Figure 3.7: Slide of Cells at Z = 0.38 \(p. 46\)](#).

**Figure 3.7: Slide of Cells at Z = 0.38**



- d. Check the number of cells.

**Report → Mesh Size...**

The number of cells is 1125494. The exact number may differ on different platforms.

- e. Check the maximum skewness.

#### **Report → Cell Limits...**

The maximum skewness is 0.904, which is acceptable. The average skewness is 0.264.

#### **Note**

- The quality is very similar to that obtained with the skewness-based refinement algorithm.
- As far as the number of cells is concerned, for a strict volume criterion, the advancing front method will generate more cells, but for a relaxed maximum volume criterion, the skewness method will generate more cells.
- For a mesh of size similar to that considered in this tutorial, tet refinement for the advancing front method is approximately 1.8 times faster when compared with the skewness method. The speedup will increase for bigger size meshes.

## **3.7. Examine the Effect of the Growth Factor**

1. Clear the mesh.
2. Specify the meshing parameters.

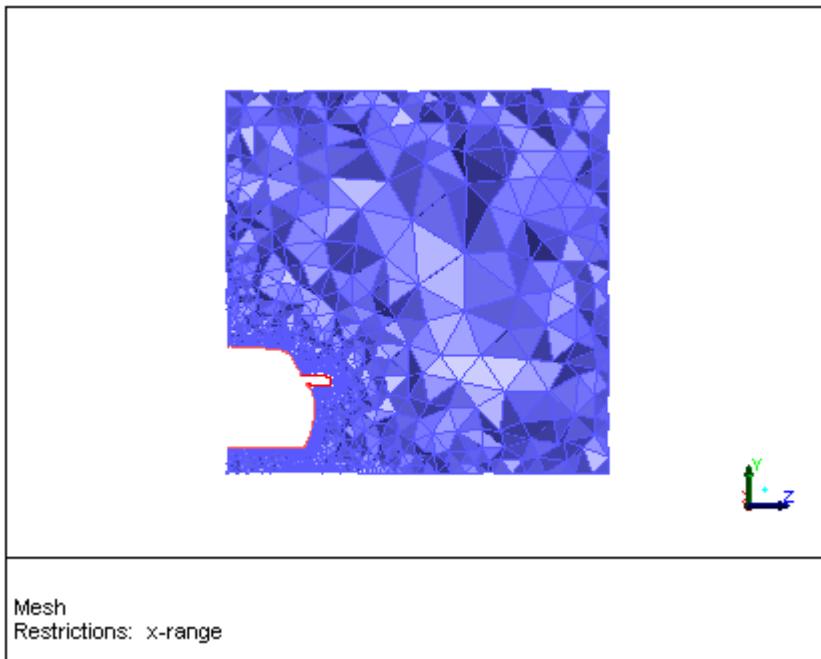
#### **Mesh → Tet...**

- a. Retain the selection of **adv-front** in the **Refine Method** drop-down list and the **Growth Rate** of 1 . 3, respectively.
- b. Enter 2e-2 for **Max Cell Volume** in the **Refinement** tab of the **Tet** dialog box.
- c. Click **Apply** and **Init & Refine**.
- d. Close the **Tet** dialog box.

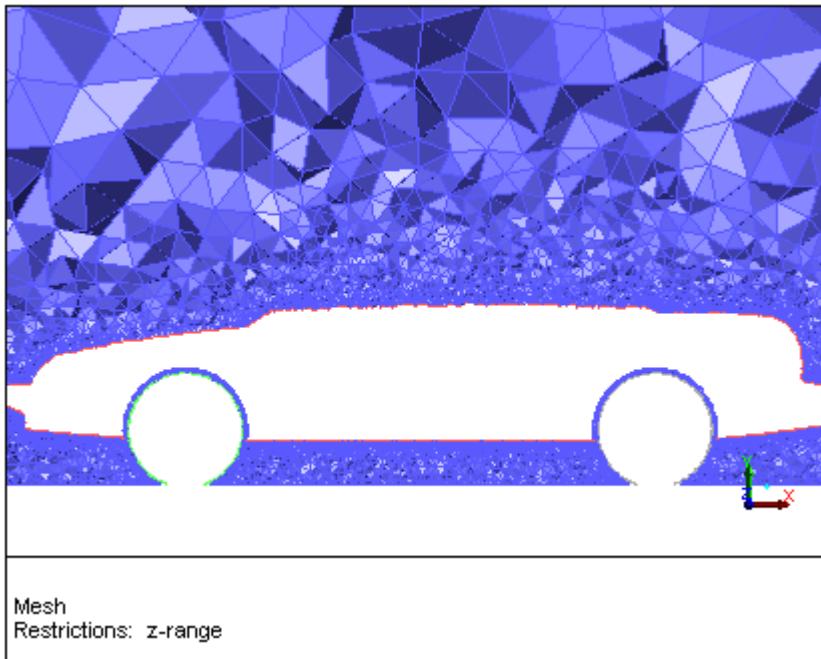
3. Examine the mesh.

#### **Display → Grid...**

- a. Display the mesh on a slide through the mirror and the car ( $x = -0.37$ ). See [Figure 3.8: Slide of Cells at X = -0.37 \(p. 48\)](#).

**Figure 3.8: Slide of Cells at X = -0.37**

- b. Display the mesh on a slide through the wheels ( $z = 0.38$ ). See [Figure 3.9: Slide of Cells at Z = 0.38 \(p. 48\)](#).

**Figure 3.9: Slide of Cells at Z = 0.38**

- c. Check the number of cells.

**Report → Mesh Size...**

The number of cells is 1093103. The exact number may differ on different platforms.

- d. Check the maximum skewness.

**Report → Cell Limits....**

The maximum skewness is 0.904, while the average skewness is 0.266.

4. Clear the mesh.
5. Modify the meshing parameters.

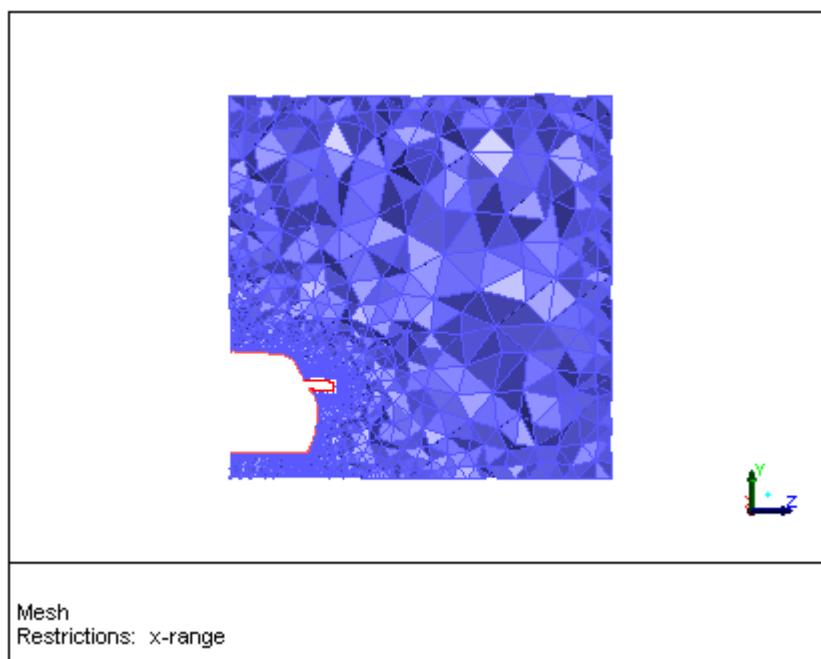
**Mesh → Tet...**

- a. Retain the selection of **adv-front** in the **Refine Method** drop-down list and the **Max Cell Volume** of  $2e-2$ , respectively.
  - b. Enter 1.25 for **Growth Rate** in the **Refinement** tab of the **Tet** dialog box.
  - c. Click **Apply** and **Init & Refine**.
  - d. Close the **Tet** dialog box.
6. Examine the mesh.

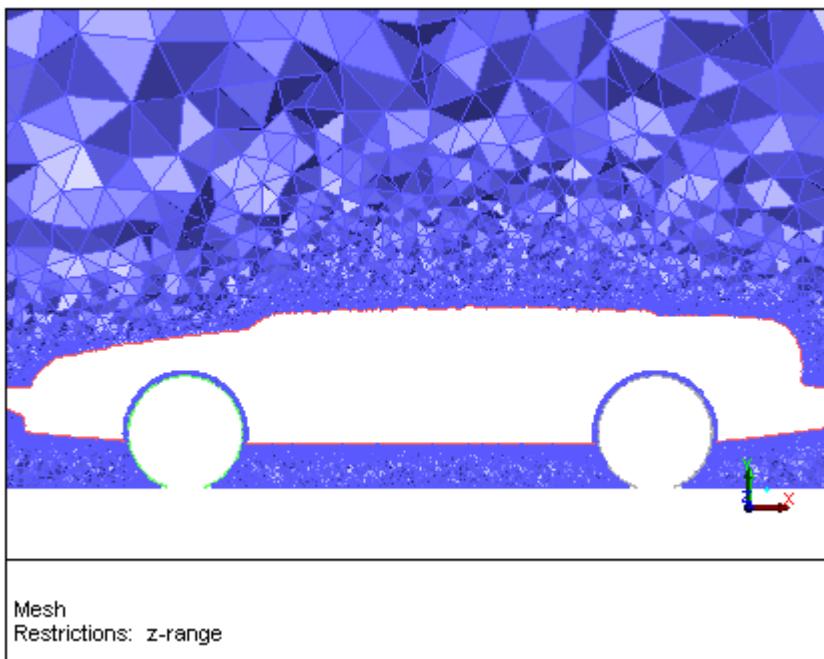
**Display → Grid...**

- a. Display the mesh on a slide through the mirror and the car ( $x = -0.37$ ).

**Figure 3.10: Slide of Cells at  $X = -0.37$**



- b. Display the mesh on a slide through the wheels ( $z = 0.38$ ). See [Figure 3.11: Slide of Cells at  \$Z = 0.38\$  \(p. 50\)](#).

**Figure 3.11: Slide of Cells at Z = 0.38**

- c. Check the number of cells.

**Report → Mesh Size...**

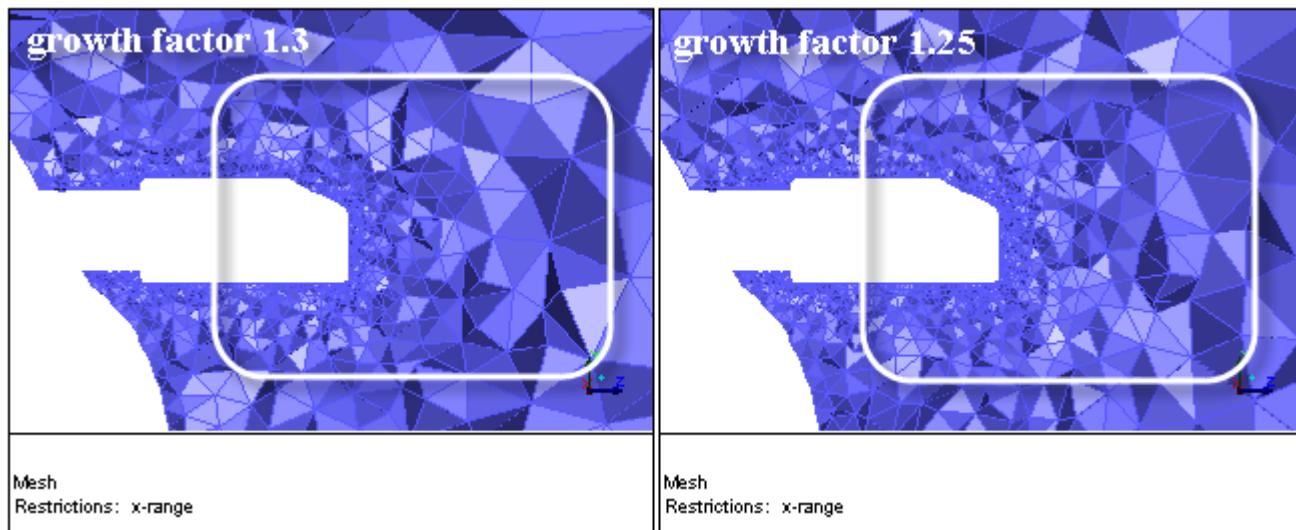
The number of cells is 1285028. The exact number may differ on different platforms.

- d. Check the maximum skewness.

**Report → Cell Limits...**

The maximum skewness is 0.904, while the average skewness is 0.251.

For the mesh generated, the mesh transition is smoother (see [Figure 3.12: Comparison of Meshes Based on Growth Factor \(p. 51\)](#)), however the number of cells generated is significantly more.

**Figure 3.12: Comparison of Meshes Based on Growth Factor**

## 3.8. Generate a Local Refinement in the Wake of the Car

You can define the local size regions to be meshed at the same time as the global mesh initialization and refinement. Multiple regions, each with different maximum cell volume can be defined and activated during the automatic mesh generation process. This section demonstrates the use of a local refinement region in the wake of the car.

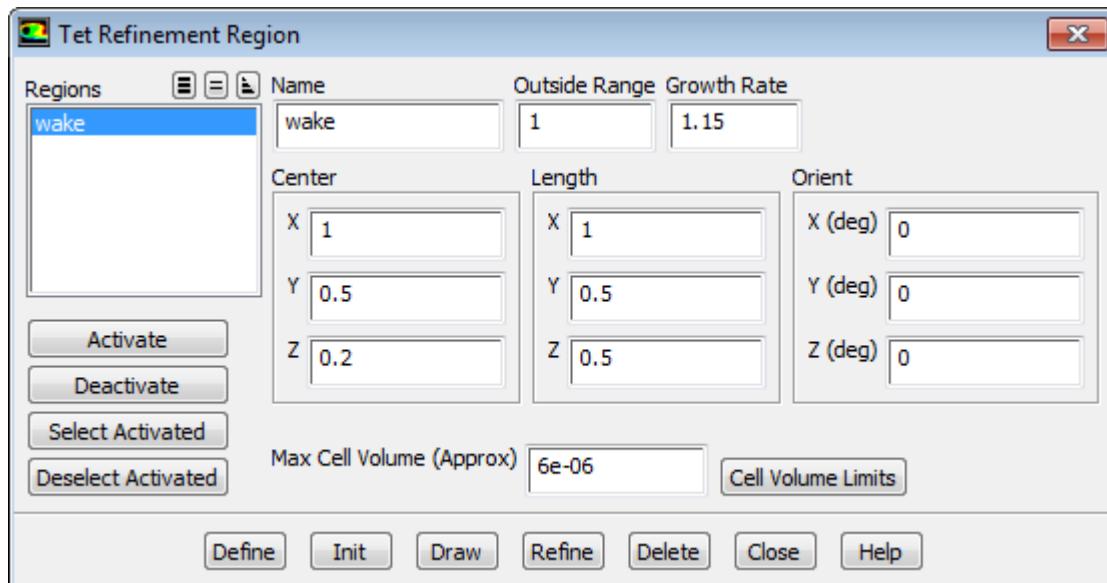
1. Clear the mesh.
2. Display the car.

**Display → Grid...**

- a. Click **Reset** in the **Bounds** tab.
  - b. Retain the selection of **\_sedan** in the **Face Zone Groups** selection list in the **Faces** tab and click **Display**.
  - c. Close the **Display Grid** dialog box.
3. Specify the meshing parameters.
- a. Retain the previous settings in the **Initialization** tab of the **Tet** dialog box.

**Mesh → Tet...**

- b. Click the **Local Regions...** button in the **Refinement** tab to open the **Tet Refinement Region** dialog box.



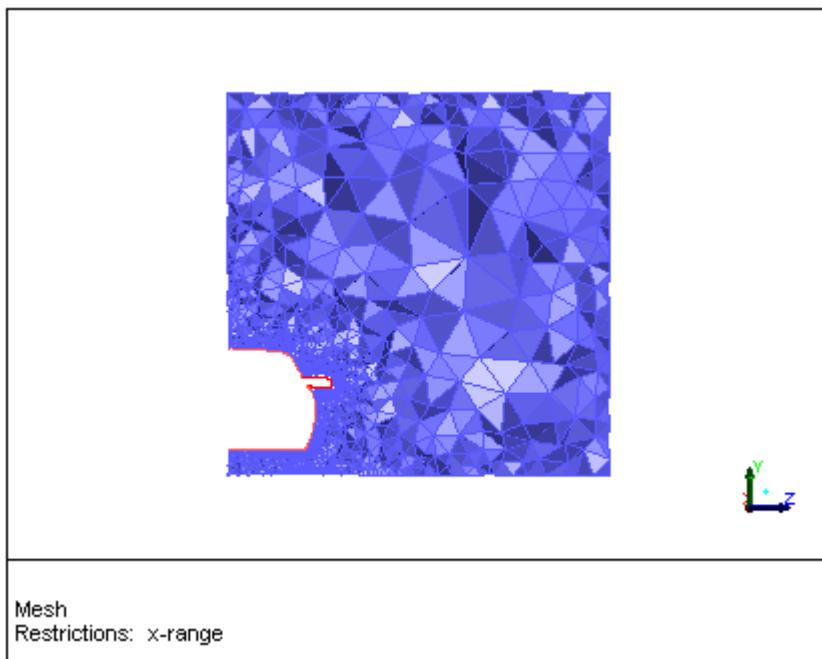
- i. Enter **wake** for **Name**.
- ii. Enter  $(1, 0.5, 0.2)$  for **Center** and  $(1, 0.5, 0.5)$  for **Length**.
- iii. Retain the default orientation of  $(0, 0, 0)$ .
- iv. Enter  $6e-6$  for **Max Cell Volume**.
- v. Retain the value of  $1$  for **Outside Range** and enter  $1.15$  for **Growth Rate**.
- vi. Click **Draw** to see the extents of the region and the maximum cell volume specified.
- vii. Click **Define** to define the wake region.
- viii. Click **Activate** for the region to be taken into account during refinement.
- ix. Close the **Tet Refinement Region** dialog box.

The **Message** field will report that there is one active tet refinement region.

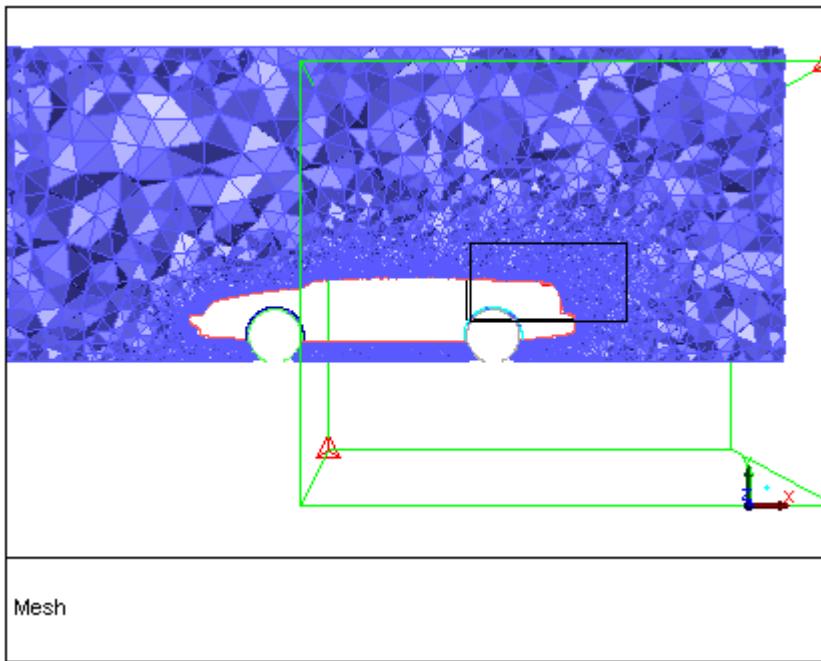
- c. Retain the selection of **adv-front** in the **Refine Method** drop-down list and the **Max Cell Volume** of  $2e-2$ , respectively.
  - d. Enter  $1.3$  for **Growth Rate**.
  - e. Click **Apply** and **Init & Refine**.
  - f. Close the **Tet** dialog box.
4. Examine the mesh.

#### **Display → Grid...**

- a. Display the mesh on a slide through the mirror and the car ( $x = -0.37$ ). See [Figure 3.13: Slide of Cells at  \$X = -0.37\$  \(p. 53\)](#).

**Figure 3.13: Slide of Cells at X = -0.37**

- b. Display the mesh on a slide through the wheels ( $z = 0.38$ ).
- c. Display the refinement region along with the cells (Figure 3.14: Slide of Cells at  $Z = 0.38$  with the Refinement Region (p. 54)).
  - i. Click the **Local Regions...** button in the **Tet** dialog box to open the **Tet Refinement Region** dialog box.  
**Mesh → Tet...**
  - ii. Make sure **wake** is selected in the **Regions** selection list and click **Draw**.
  - iii. Close the **Tet Refinement Region** dialog box.

**Figure 3.14: Slide of Cells at Z = 0.38 with the Refinement Region**

- d. Check the number of cells.

**Report → Mesh Size...**

The number of cells is 1127988. The exact number may differ on different platforms.

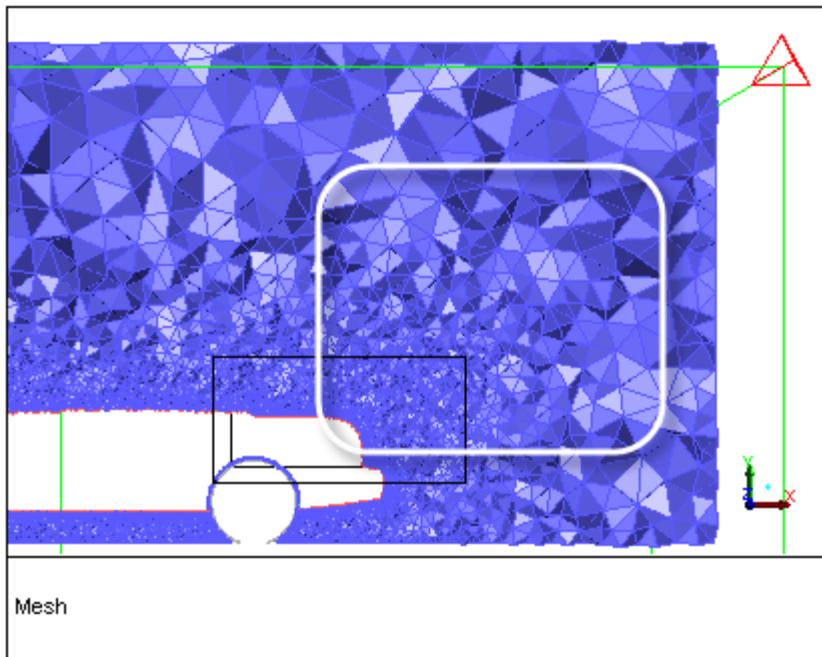
- e. Check the maximum skewness.

**Report → Cell Limits...**

The maximum skewness is 0.904, while the average skewness is 0.264.

For the mesh generated, the local growth rate defined results in a smooth transition between the small cells in the wake region and the larger cells in the rest of the domain (see [Figure 3.15: Transition Between Cells in Locally Refined Region and the Rest of the Domain \(p. 55\)](#)). Further manual operations to obtain better quality are not required in this case.

**Figure 3.15: Transition Between Cells in Locally Refined Region and the Rest of the Domain**



## 3.9. Check and Save the Volume Mesh

1. Check the mesh.

**Mesh → Check**

Various checks will be performed on the mesh and their progress will be reported in the console. Make sure the minimum volume reported is a positive number.

2. Save the mesh.

**File → Write → Mesh...**

- a. Enter `sedan-vol.msh.gz` for **Mesh File**.
- b. Click **OK** to save the volume mesh.
3. Exit ANSYS FLUENT.

**File → Exit**

## 3.10. Summary

This tutorial demonstrated the tetrahedral mesh generation process using both the refinement methods available. It also examined the effect of the size function and the growth factor on the generated mesh. The quality of the mesh generated is similar for both the refinement methods available. However, for most cases, the advancing front method will be faster due to a greater number of cells generated per second. The use of local refinement regions was also demonstrated.



---

## Chapter 4: Zonal Hybrid Mesh Generation

---

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 the meshing mode in ANSYS FLUENT 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:

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

### 4.1. Prerequisites

This tutorial assumes that you have little experience with the meshing mode in ANSYS FLUENT, but that you are familiar with the graphical user interface.

### 4.2. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

To prepare for running this tutorial:

1. Download the tutorial input file (`zonal-hybrid.zip`) for the tutorial.
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.

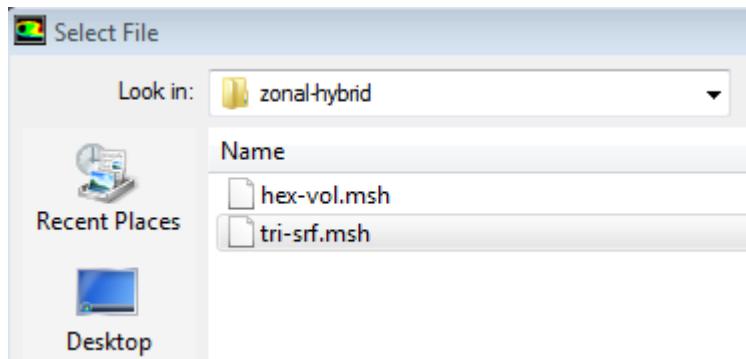
3. Start ANSYS FLUENT in meshing mode. For detailed steps, refer to [Starting ANSYS FLUENT in Meshing Mode \(p. 4\)](#).

## 4.3. Generate the Tetrahedral Mesh Using Pyramids to Transition Between the Hexahedral and Tetrahedral Mesh

### Read and Display the Mesh

1. Read the two mesh files.

**File → Read → Mesh...**



- a. Select hex-vol.msh.

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

- b. Select tri-srf.msh.

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

- c. Click **OK**.

---

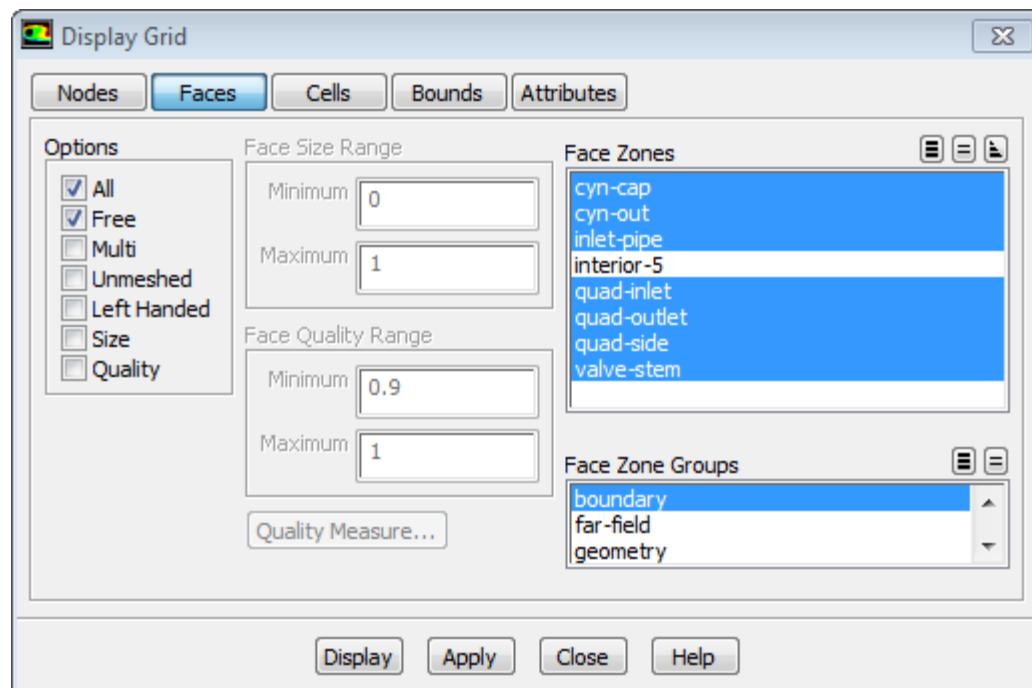
#### Important

ANSYS FLUENT 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.

---

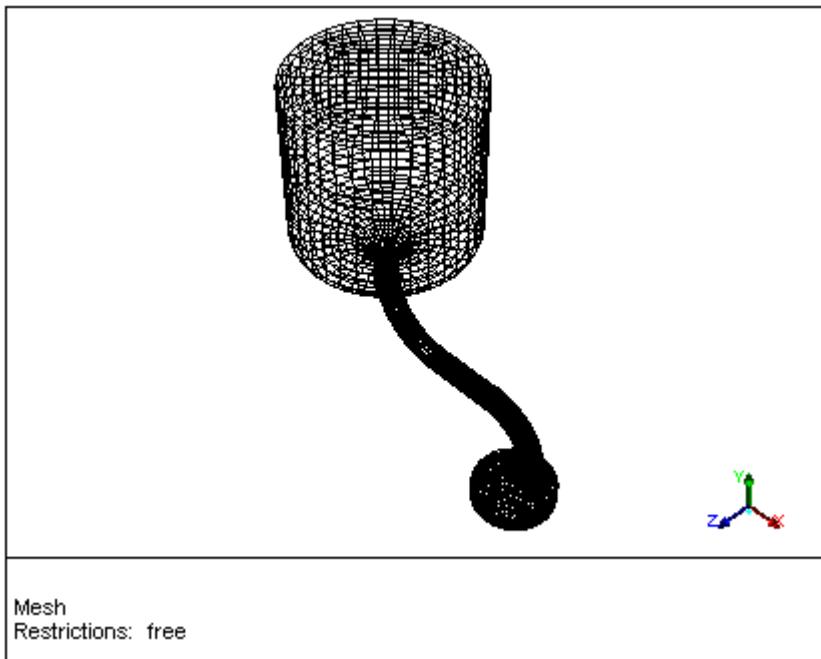
2. Display the boundary mesh ([Figure 4.1: Boundary Mesh for the Valve Port \(p. 60\)](#)).

**Display → Grid...**

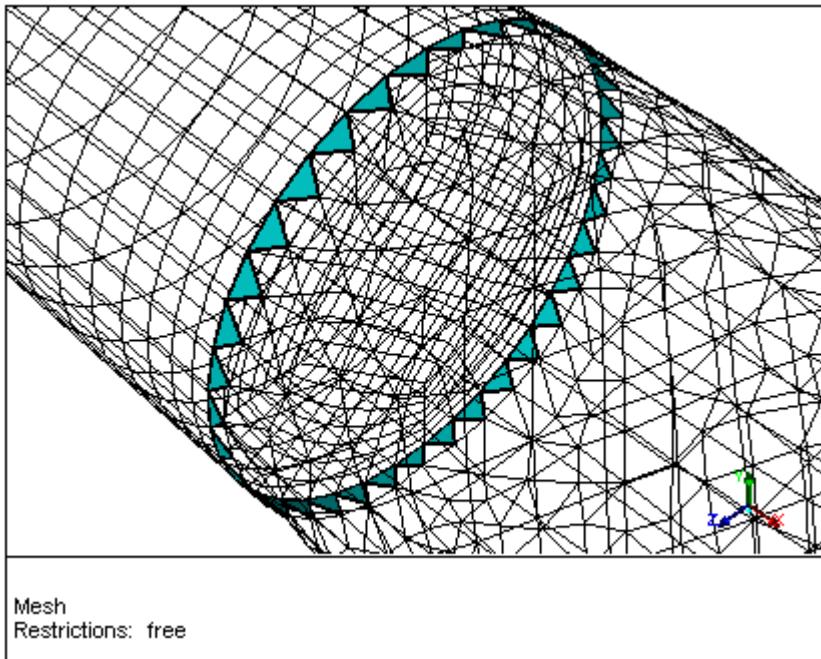


- a. Select **boundary** in the **Face Zone Groups** selection list to select all the boundary zones in the **Face Zones** selection list.
- b. Make sure **Free** is enabled (in addition to the default, **All**) in the **Options** group box.

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**.

**Figure 4.1: Boundary Mesh for the Valve Port**

- e. Zoom in to focus on the free nodes (Figure 4.2: Free Nodes at the Intersection of the Tri and Quad Boundary Meshes (p. 60)).

**Figure 4.2: Free Nodes at the Intersection of the Tri and Quad Boundary Meshes**

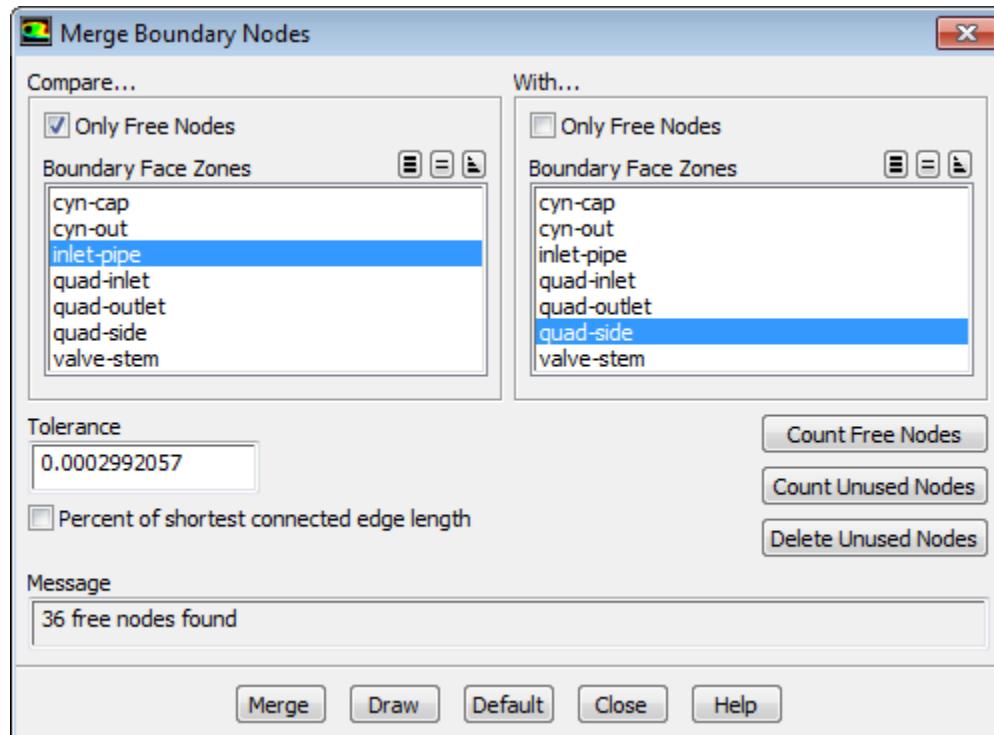
In Figure 4.2: Free Nodes at the Intersection of the Tri and Quad Boundary Meshes (p. 60), the triangular faces that use the free nodes on the boundary between the tri and quad-face zones are colored.

- f. Close the **Display Grid** dialog box.

## Merge the Free Nodes on the Tri/Quad Border

This section demonstrates the merging of 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**.

The number of free nodes will be reported in the **Message** box.

4. Click **Merge** to merge the free nodes.**Note**

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. Make sure **inlet-pipe** and **quad-side** are still selected in the **Compare...** and **With...** group boxes, respectively, and 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** dialog box.

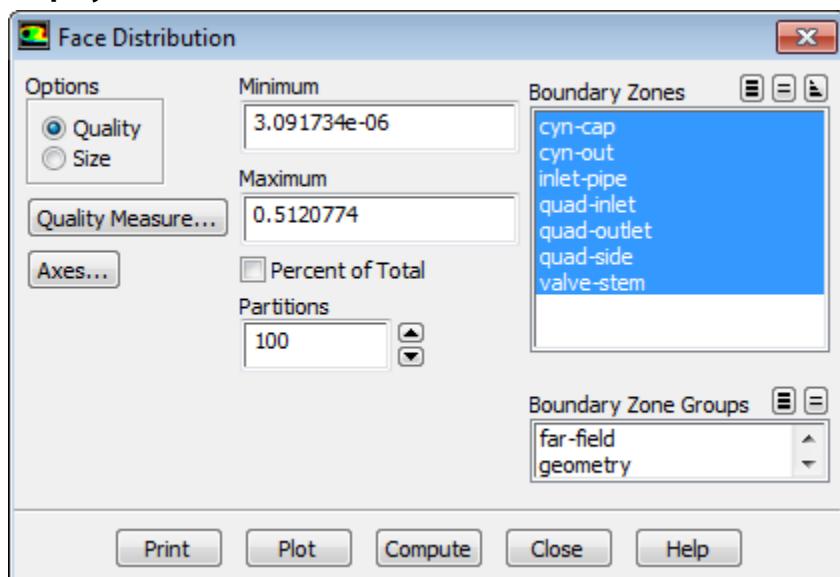
## 9. Save the mesh file.

**File → Write → Mesh...**

a. Enter hex-tri-merged.msh for **Mesh File**.b. Click **OK** to save the mesh.

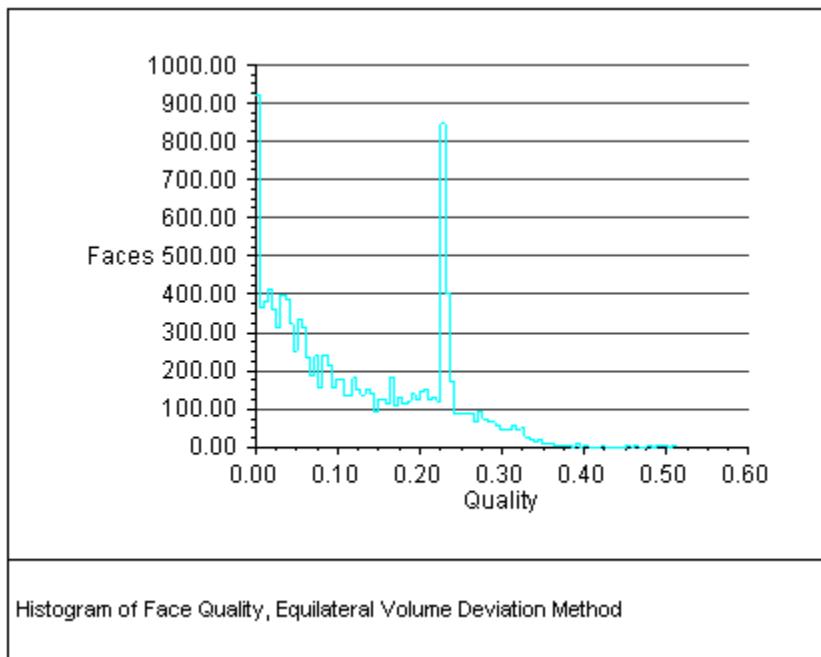
## Check the Skewness Distribution of the Boundary Mesh

**Display → Plot → Face Distribution...**



1. Select all the zones in the **Boundary Zones** selection list.
2. Click **Compute**.
3. Click **Plot** ([Figure 4.3: Boundary Mesh Skewness Distribution \(p. 63\)](#)).

**Figure 4.3: Boundary Mesh Skewness Distribution**



### Tip

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 [Tetrahedral Mesh Generation \(p. 35\)](#).

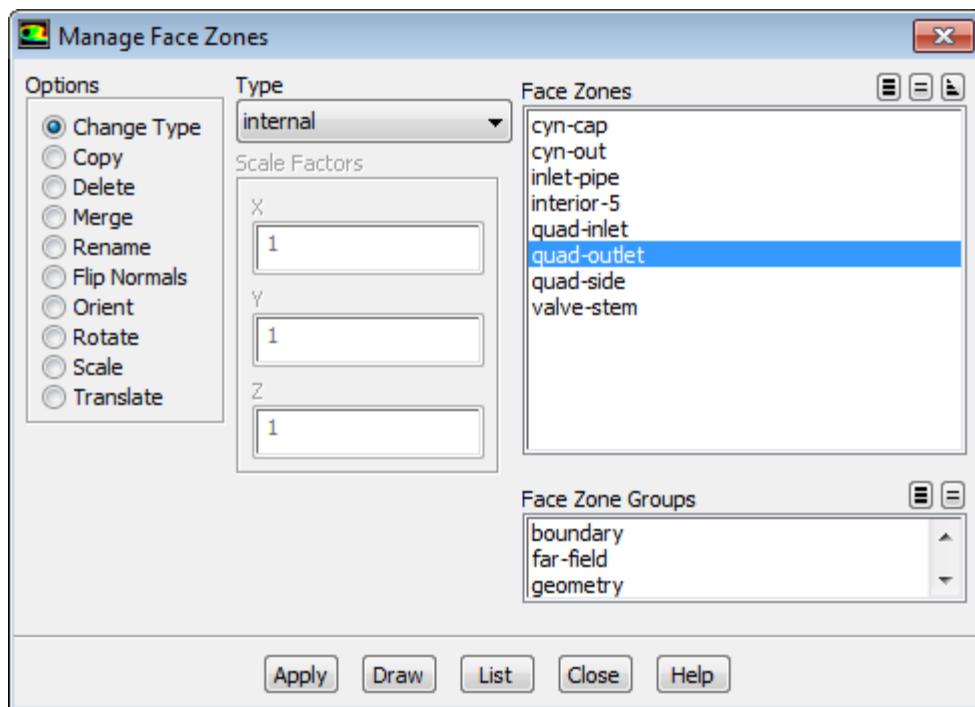
## Generate the Mesh

This section demonstrates the Auto Mesh procedure and the use of 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.

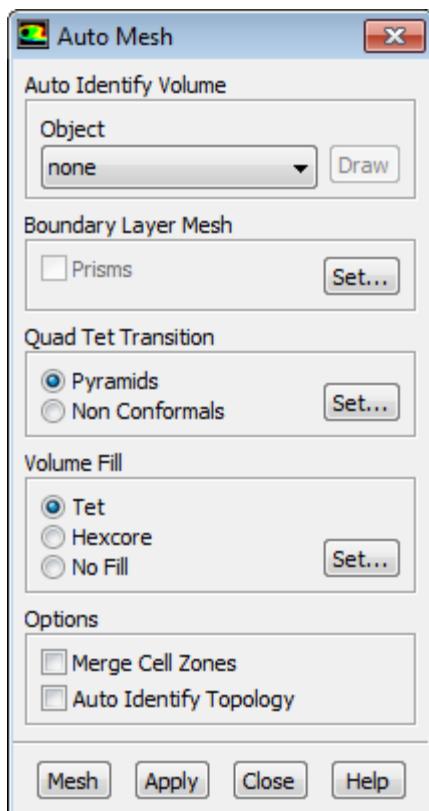
### Warning

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

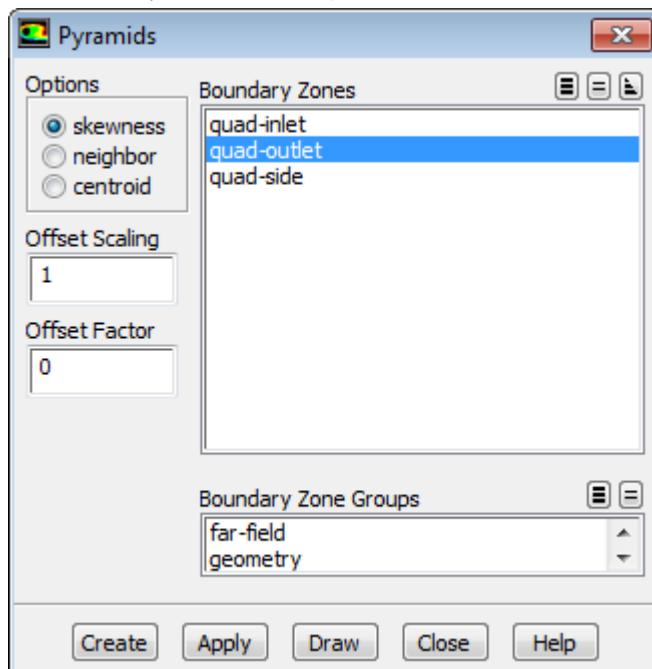
- c. Click **Apply** and close the **Manage Face Zones** dialog box.

2. Set the meshing parameters.

**Mesh → Auto Mesh...**

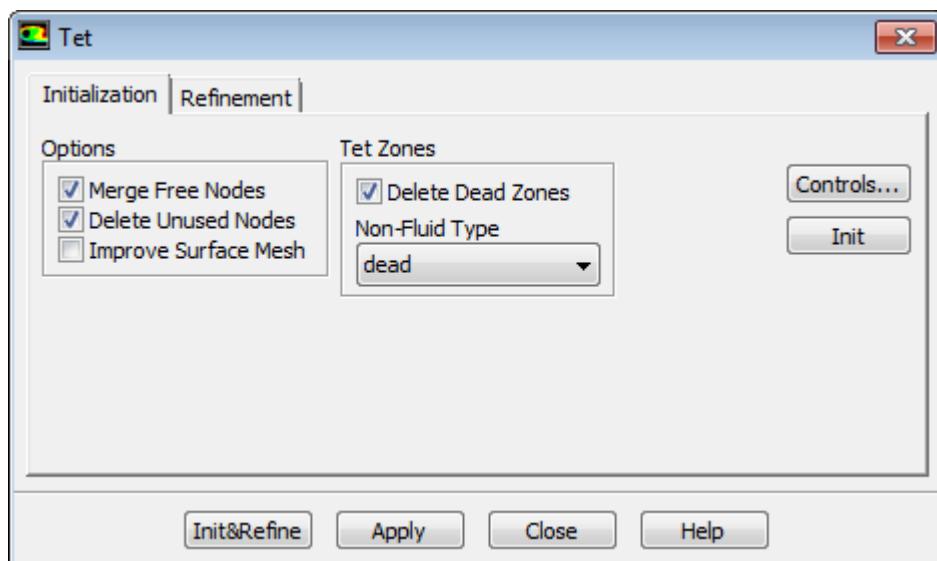


- Retain the selection of **Pyramids** in the **Quad Tet Transition** list and click the **Set...** button to open the **Pyramids** dialog box.

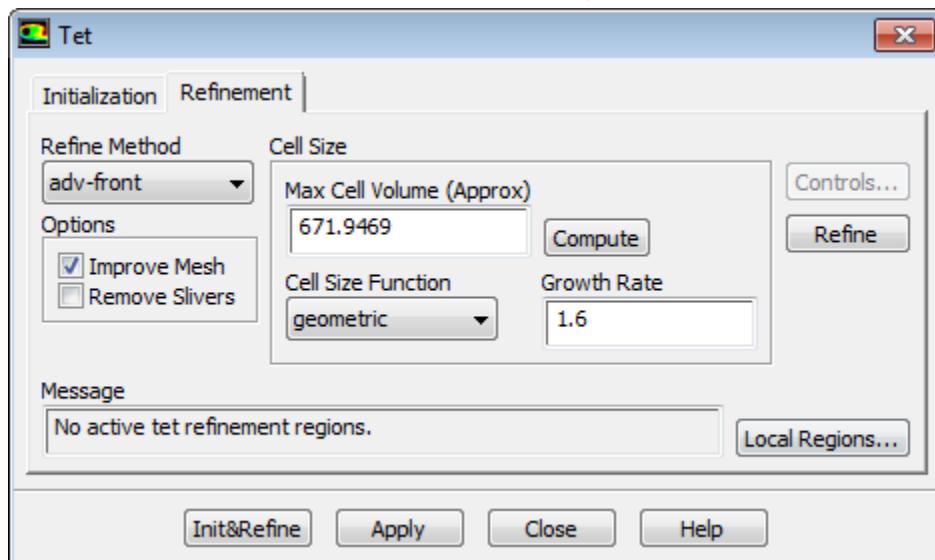


- Select **quad-outlet** in the **Boundary Zones** selection list.
- Retain the selection of **skewness** in the **Options** list.
- Click **Apply**.

- iv. Close the **Pyramids** dialog box.
- b. Retain the selection of **Tet** in the **Volume Fill** list and click the **Set...** button to open the **Tet** dialog box.



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



- ii. Retain the default settings in the **Refinement** tab.
- iii. Click **Apply** and close the **Tet** dialog box.
- c. Click **Apply** in the **Auto Mesh** dialog box.
- d. Preserve the existing hexahedral mesh.

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

- e. Click **Mesh** in the **Auto Mesh** dialog box.

The maximum and average skewness values reported at the end of the meshing are approximately 0.847 and 0.367, respectively.

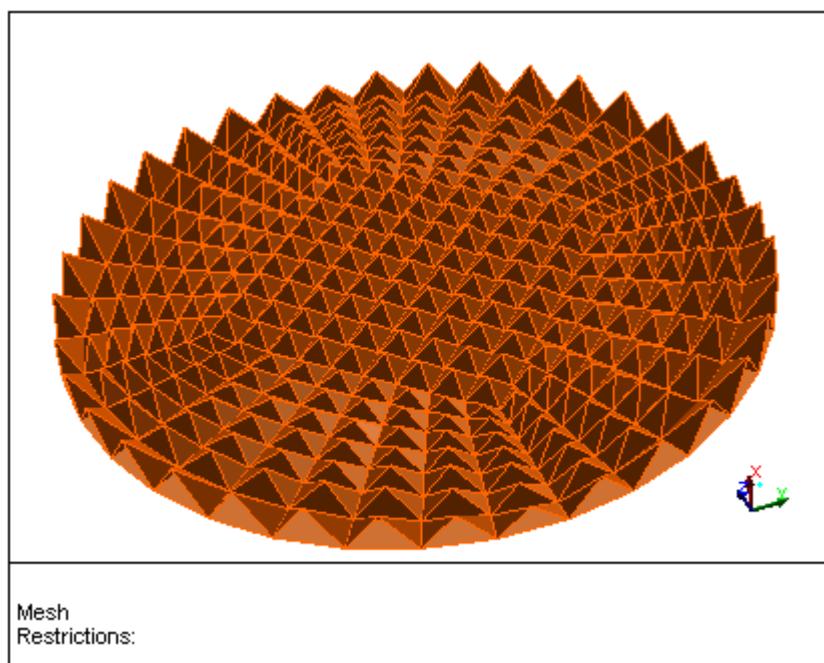
- f. Close the **Auto Mesh** dialog box.

3. Display the pyramid cap.

**Display → Grid...**

- a. Deselect all the previous selections in the **Face Zones** selection list in the **Faces** tab and then select **quad-outlet** and **quad-outlet-pyramid-cap-#**.
- 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 4.4: Pyramid Cap \(p. 67\)](#).

**Figure 4.4: Pyramid Cap**



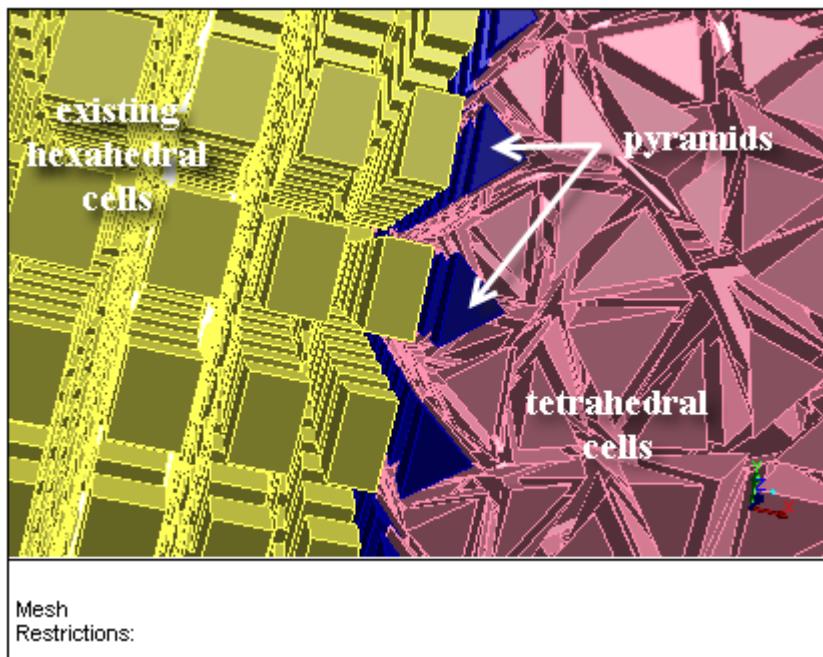
4. Examine the transition between the hexahedral and tetrahedral mesh.

**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**.

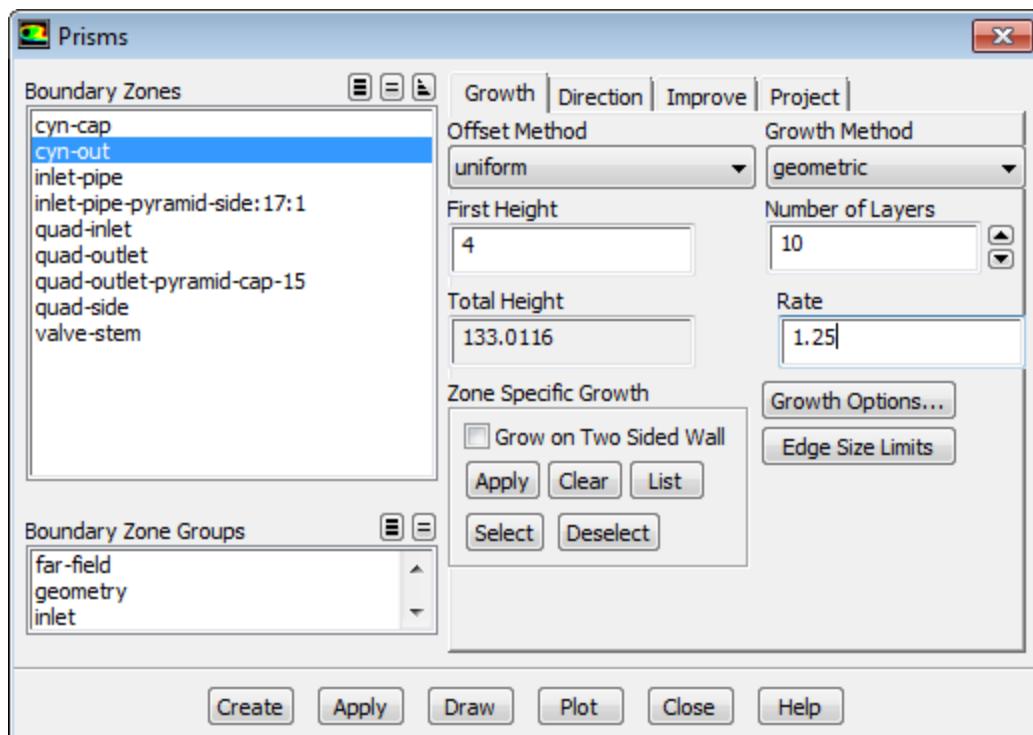
- e. Click the **Colors...** button to open the **Grid Colors** dialog box.
  - i. Select **Color by ID** in the **Options** list.
  - ii. Close the **Grid Colors** dialog box.
- f. Click **Display** and zoom in close to the boundary between the hexahedral and tetrahedral mesh (Figure 4.5: Pyramid Transition Between the Hexahedral and Tetrahedral Mesh (p. 68)).

**Figure 4.5: Pyramid Transition Between the Hexahedral and Tetrahedral Mesh**



## 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** dialog box 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 ( $4 \times 1.25$ ), and so on.

- c. Enter 10 for **Number of Layers**.

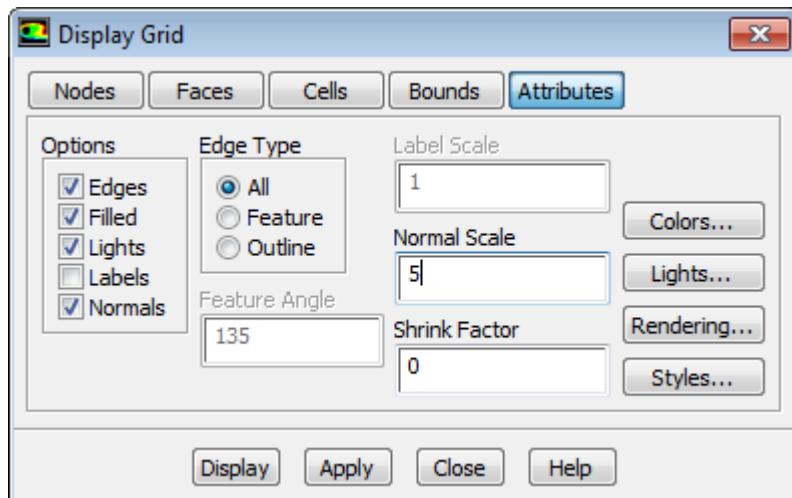
The **Total Height** added by the prisms is slightly more than 133.

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**.

---

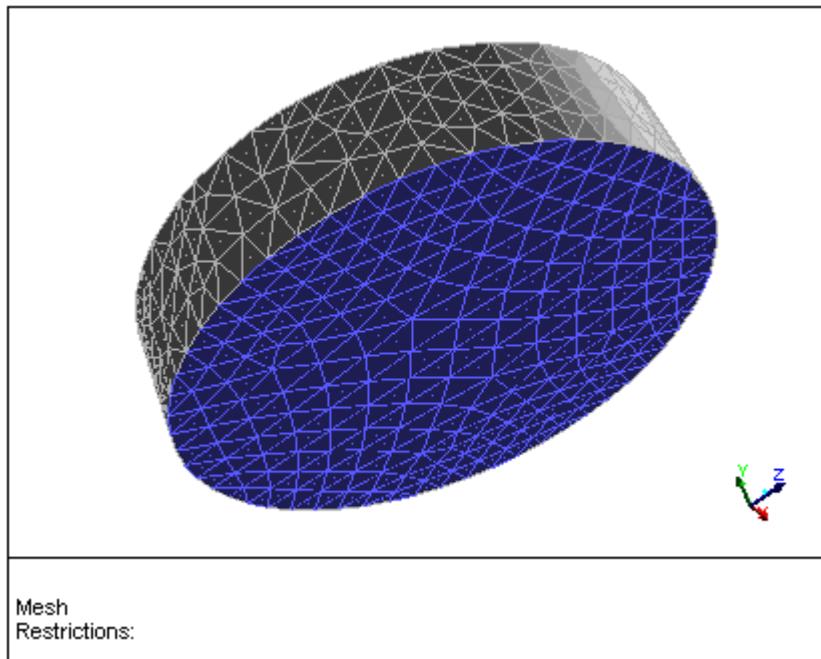
**Tip**

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 4.6: Cylinder Normals in Wrong Direction (p. 70)).

**Figure 4.6: Cylinder Normals in Wrong Direction**



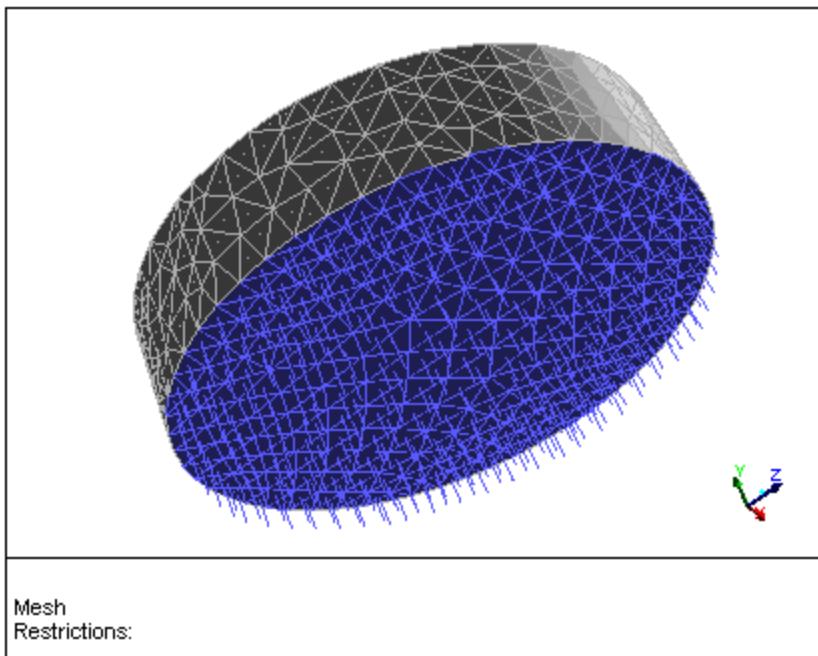
In [Figure 4.6: Cylinder Normals in Wrong Direction \(p. 70\)](#), the normals are not pointing out from the bottom of the cylinder. Since they need to point out (i.e., down), you need to flip them.

- e. Flip the normals on the **cyn-out** zone.

**Boundary → Manage...**

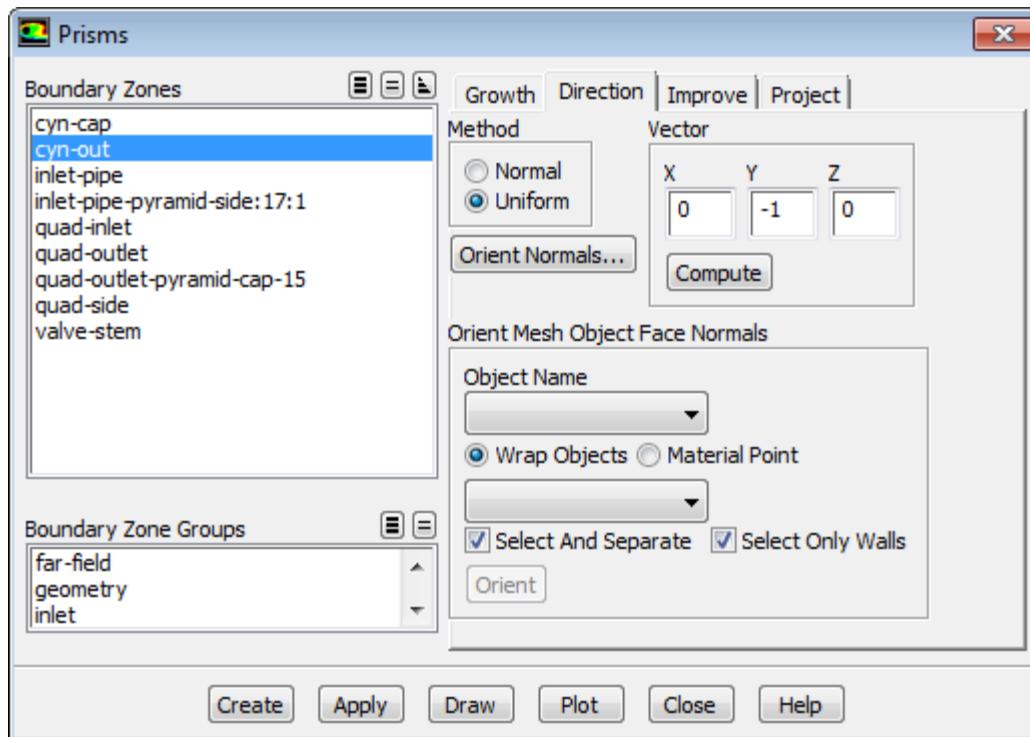
- i. Select **cyn-out** in the **Face Zones** list.
- ii. Select **Flip Normals** in the **Options** list.
- iii. Click **Apply** and close the **Manage Face Zones** dialog box.
- iv. Click **Display** and close the **Display Grid** dialog box.

**Figure 4.7: Cylinder with Normals in Correct Direction**



In [Figure 4.7: Cylinder with Normals in Correct Direction \(p. 71\)](#), the normals are pointing in the correct direction. The prisms built will extend the cylinder below its current bottom.

4. Specify the growth direction for the prisms.



- a. Click the **Direction** tab and 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.

- b. Click **Compute** in the **Vector** group box to update the normal direction vector for the **cyn-out** zone.
5. Click **Apply** to save the prism parameters.
6. Save an intermediate mesh file (`temp.msh`).

**File → Write → Mesh...**

### Tip

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.

7. Click **Create**.

The layers of prisms will be created and a summary of the new zones that have been created will be printed in the console:

```
Prism Layer Summary:
3920 wedge cells created in new zone prism-cells-#.
9128 mixed 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-#**, containing the wedge prism cells.
- prism-cap-#**, the new bottom of the cylinder (with triangular faces).
- prism-side-#**, containing the quadrilateral boundary faces on the outside of the cylinder.

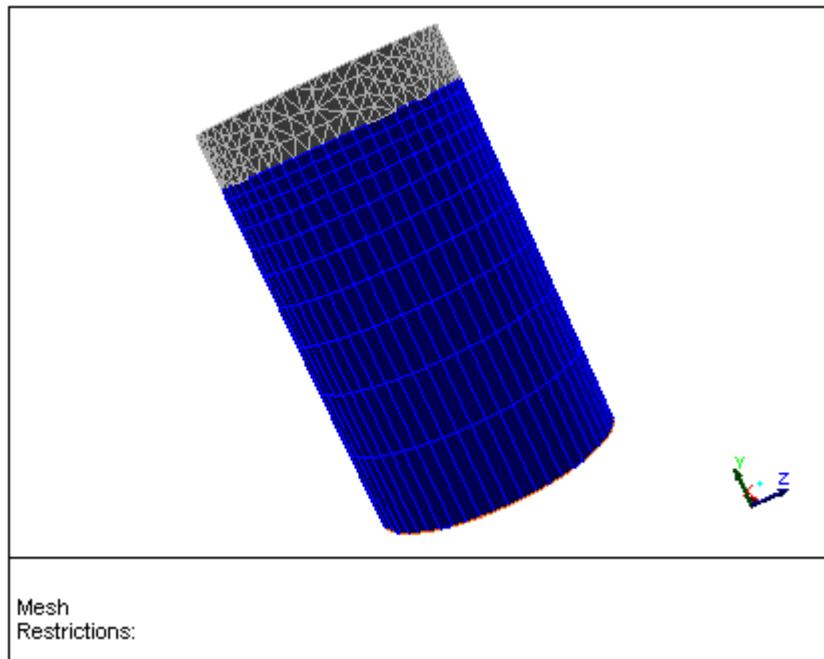
8. Close the **Prisms** dialog box.
9. Display the new boundaries of the cylinder (Figure 4.8: Cylinder Extended Using Prisms (p. 73)).

**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** dialog box.

**Figure 4.8: 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.

**Warning**

It is recommended that you select **internal** instead of **interior** for the boundary type. If you clear the mesh, all interior zones will be removed, but the internal zones will be retained. When you transfer 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**.

---

**Extra**

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

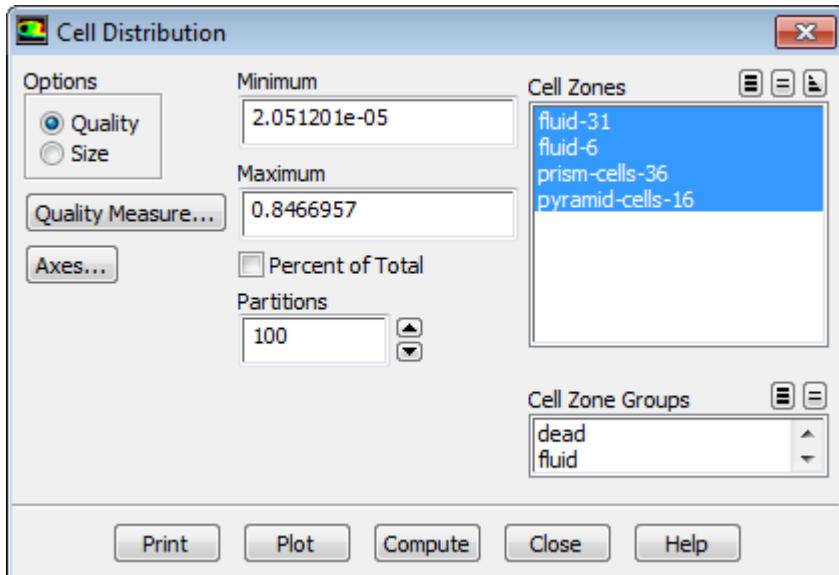
---

- c. Close the **Manage Face Zones** dialog box.

## Check and Save the Volume Mesh

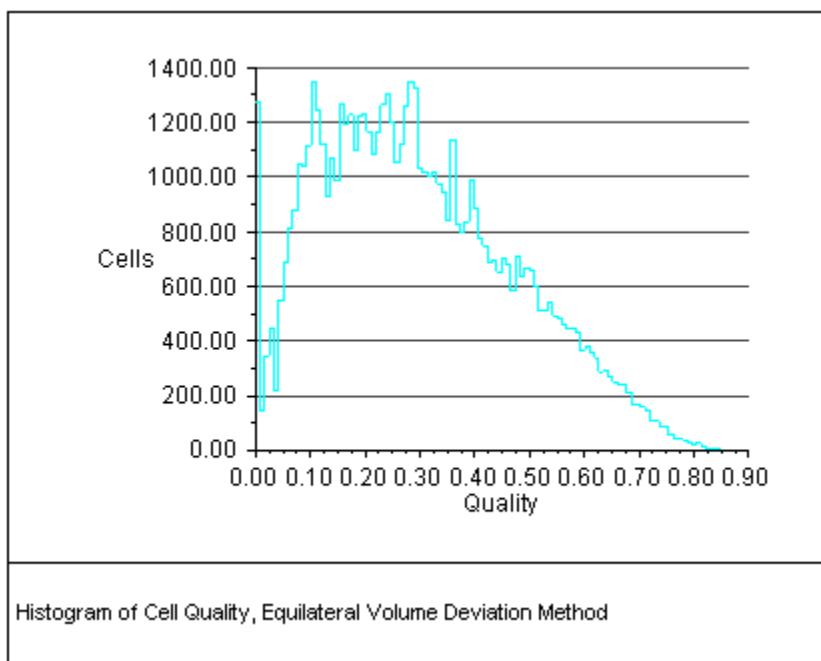
1. Check the skewness of the entire volume mesh.
  - a. Plot the cell skewness distribution ([Figure 4.9: Cell Skewness Distribution \(p. 75\)](#)).

**Display** → **Plot** → **Cell Distribution...**



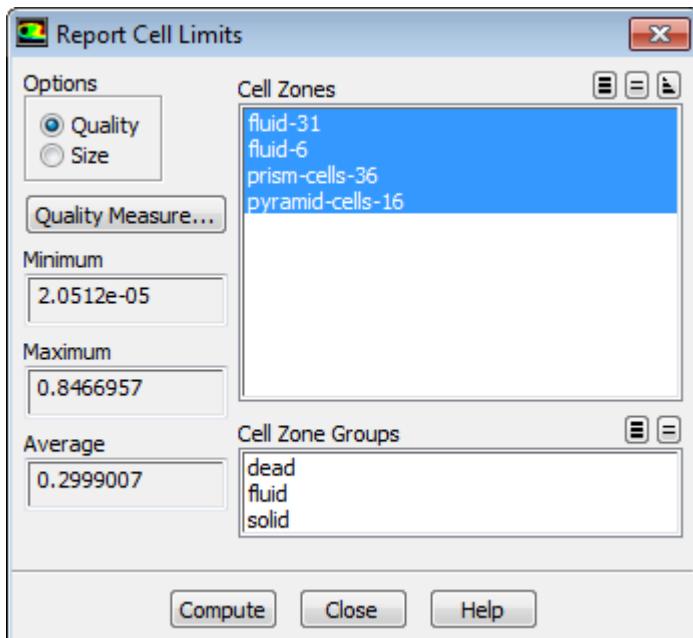
- i. Select all the zones in the **Cell Zones** selection list.
- ii. Click **Compute**.
- iii. Click **Plot**.

**Figure 4.9: Cell Skewness Distribution**



- iv. Close the **Cell Distribution** dialog box.
- b. Report the worst cell skewness.

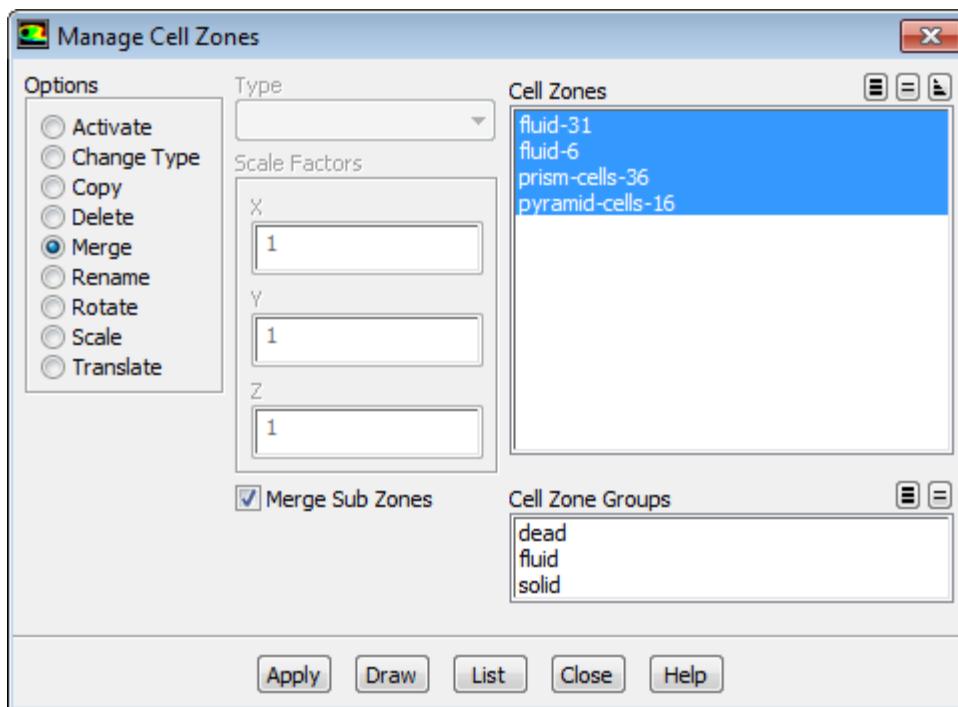
**Report → Cell Limits...**



- i. Select all the zones in the **Cell Zones** selection list.
  - ii. Click **Compute**.
  - iii. Close the **Report Cell Limits** dialog box.
2. Merge the four cell zones.

The hexahedral, pyramid, prism, and tetrahedral cells are all part of the same fluid region in this example. Hence, there is no need to retain four separate cell zones. You will now 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, the face zones associated with the cell zones will be merged, where appropriate.

- Click **Apply**.

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

- Close the **Manage Cell Zones** dialog box.

- Check the volume mesh.

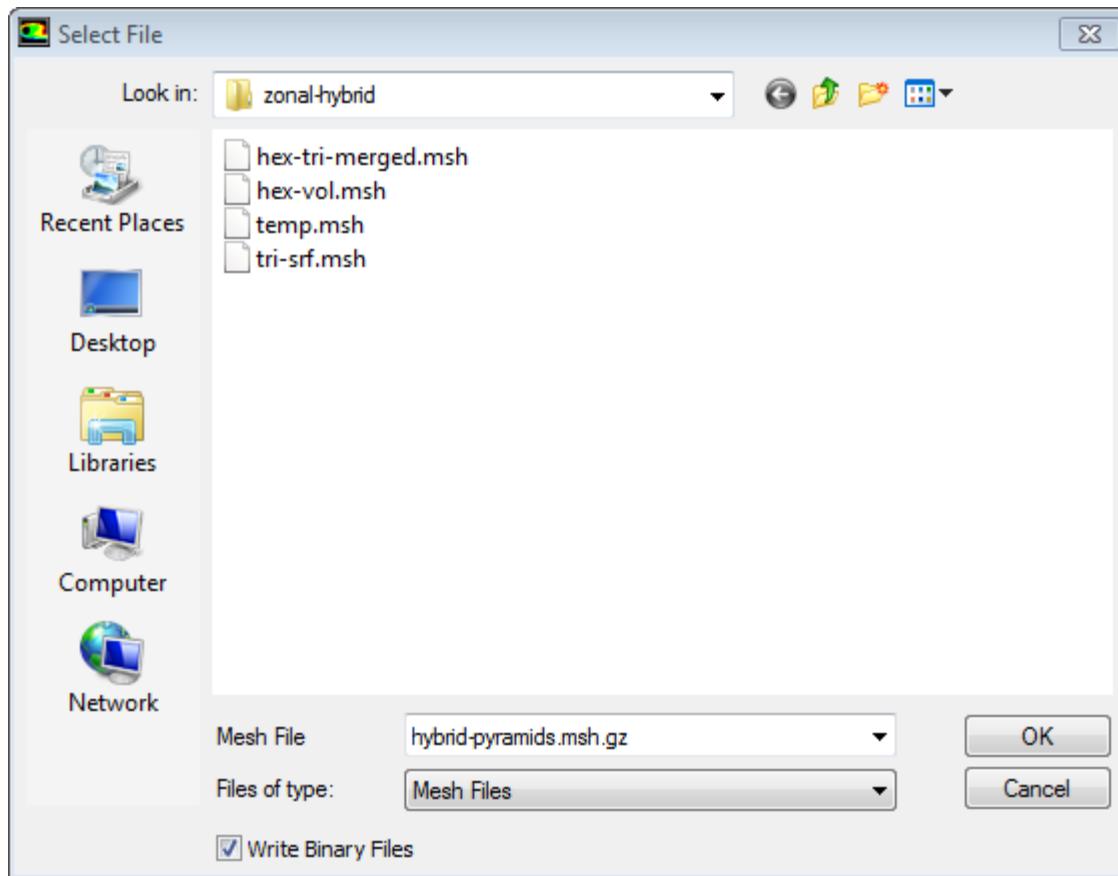
Before saving the mesh file, check it to ensure that it has no negative cell volumes or left-handed faces.

#### **Mesh → Check**

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

- Save the mesh.

**File → Write → Mesh...**



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

## 4.4. Generate the Tetrahedral Mesh Using a Non-Conformal Transition Between the Hexahedral and Tetrahedral Mesh

This section demonstrates the Auto Mesh procedure and the use of 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 [Viscous Hybrid Mesh Generation \(p. 83\)](#).

---

### Note

The procedure outlined in this section is similar to that described in the previous section, and hence is less explicit.

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

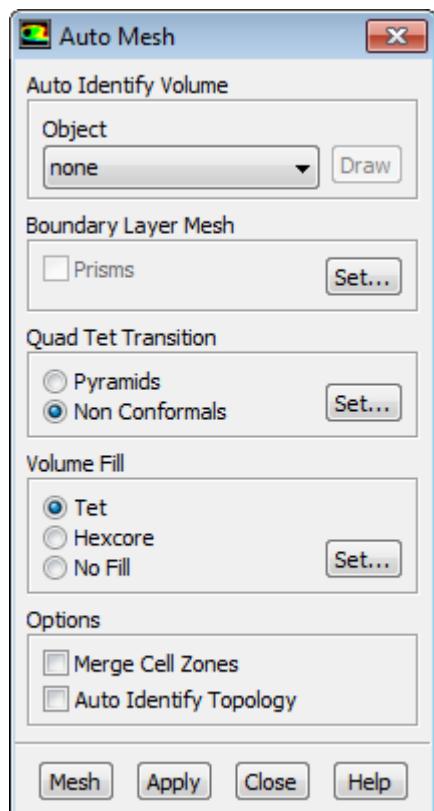
**File → Read → Mesh...**

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

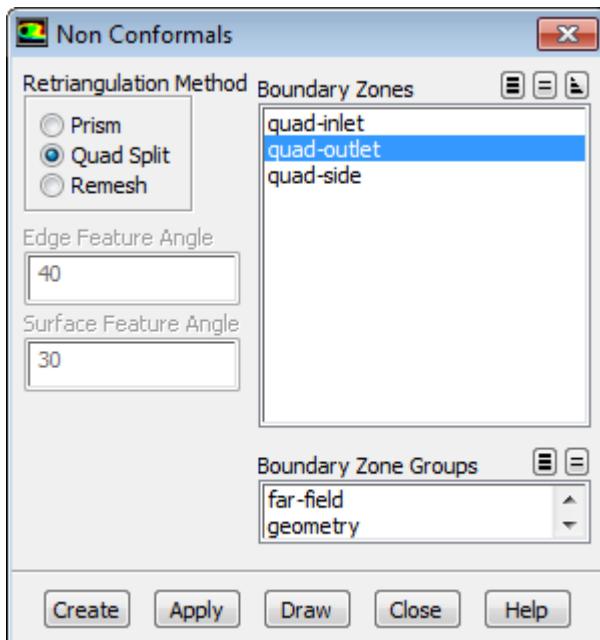
**Boundary → Manage...**

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** dialog box.



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

```
> /mesh/tet/preserve-cell-zone
()
Cell Zones(1) [()] fluid*
Cell Zones(2) [()]
```
- e. Click **Mesh** in the **Auto Mesh** dialog box.

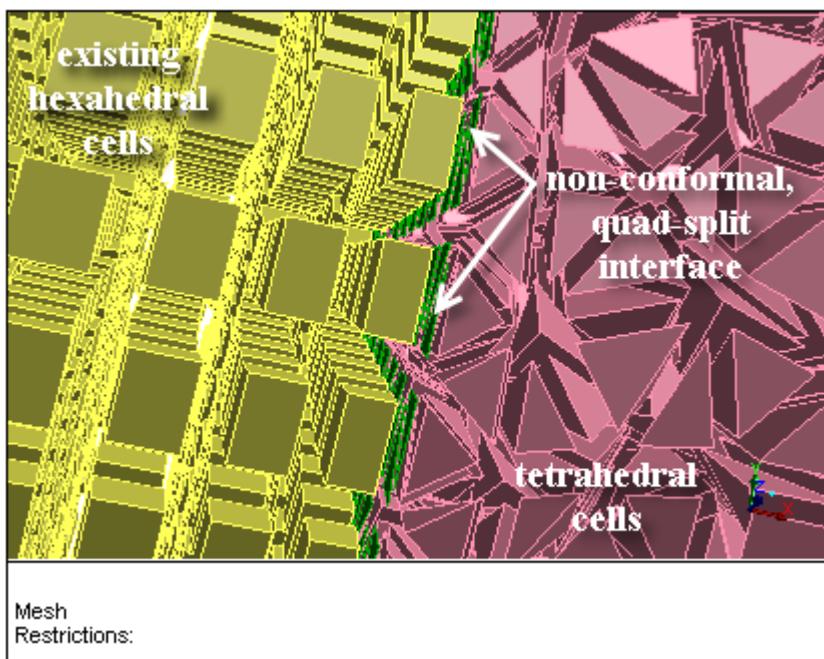
The maximum and average skewness values reported at the end of the meshing are approximately 0.849 and 0.368, respectively.
- f. Close the **Auto Mesh** dialog box.

4. Examine the transition between the hexahedral 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** dialog box.
  - i. Select **Color by ID** in the **Options** list.
  - ii. Close the **Grid Colors** dialog box.
- h. Click **Display** and zoom in close to the boundary between the hexahedral and tetrahedral mesh.

**Figure 4.10: Non Conformal Transition Between Hexahedral and Tetrahedral Mesh**




---

### Extra

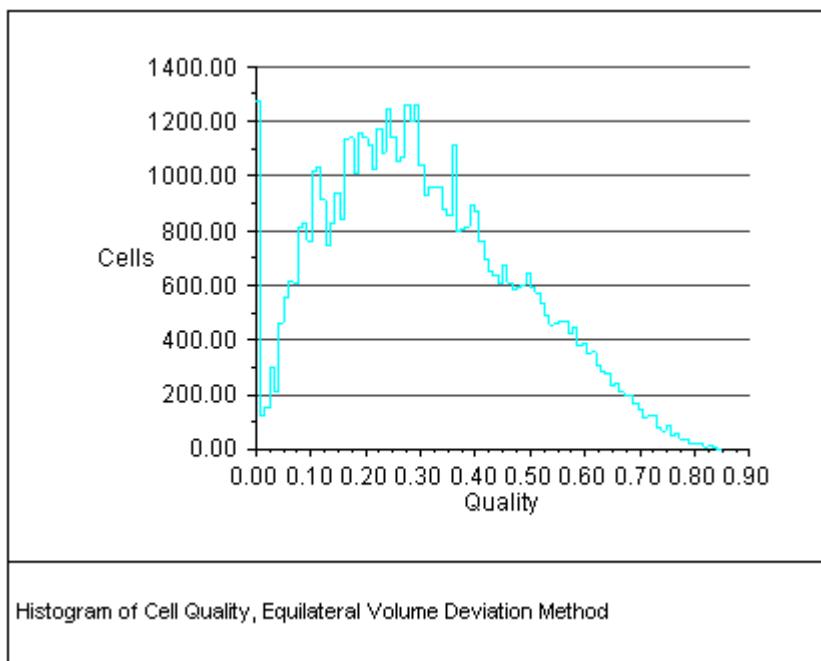
If required, you may extend the mesh using prisms as described in [Extend the Mesh Using Prisms \(p. 68\)](#). Change the type for the appropriate boundaries, as required.

---

5. Check the skewness of the entire volume mesh.
  - a. Plot the cell skewness distribution ([Figure 4.11: Cell Skewness Distribution \(p. 82\)](#)).

**Display → Plot → Cell Distribution...**

**Figure 4.11: Cell Skewness Distribution**



- b. Report the worst cell skewness.

**Report → Cell Limits...**

- 6. Merge the cell zones.

**Mesh → Manage...**

- 7. Check the volume mesh.

**Mesh → Check**

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

**File → Write → Mesh...**

- 9. Exit ANSYS FLUENT.

**File → Exit**

## 4.5. 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.

---

## Chapter 5: Viscous Hybrid Mesh Generation

---

In cases where you want to resolve the boundary layer, it is often more efficient to use prismatic cells in the boundary layer rather than tetrahedral cells. The prismatic cells allow you to resolve the normal gradients associated with boundary layers with fewer cells. The resulting mesh is referred to as a "viscous" hybrid mesh.

You can create a viscous hybrid mesh by growing prisms from the faces on the surface mesh. High quality prism elements are created near the boundary and tetrahedral elements in the rest of the domain. Automatic proximity detection and height adjustment while growing prisms in a narrow gap are also supported.

This tutorial demonstrates the mesh generation procedure for a viscous hybrid mesh, starting from a triangular boundary mesh for a sedan car body. This tutorial demonstrates how to do the following:

- Read the mesh file and display the boundary mesh.
- Check for free and unused nodes.
- Check the skewness of the boundary faces.
- Set parameters for growing prism cells allowing shrinkage and manual tetrahedral meshing.
- Set parameters for growing prism cells ignoring areas of proximity and automatic meshing.
- Examine the prisms in areas of proximity and sharp angles.
- Check the skewness of the entire volume mesh.
- Check and save the volume mesh.

### 5.1. Prerequisites

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

### 5.2. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

To prepare for running this tutorial:

1. Download the tutorial input file (`prisms.zip`) for the tutorial.
2. Unzip `prisms.zip`.

The file `sedan.msh.gz` can be found in the `prisms` folder created on unzipping the file.

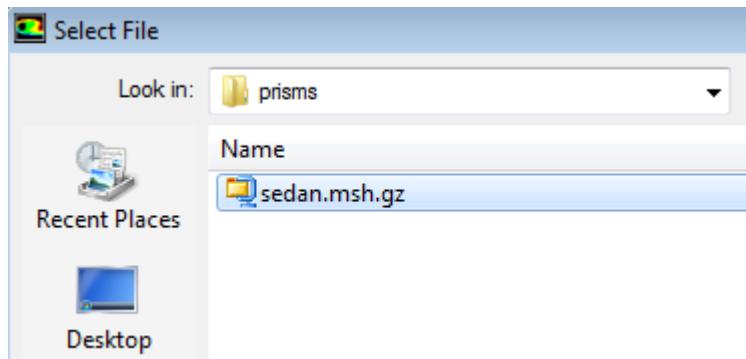
- Start ANSYS FLUENT in meshing mode. For detailed steps, refer to [Starting ANSYS FLUENT in Meshing Mode \(p. 4\)](#).

## 5.3. Generate the Mesh Using the Allow Shrinkage Option and Manual Tetrahedral Meshing

### Read and Display the Boundary Mesh

- Read the mesh file.

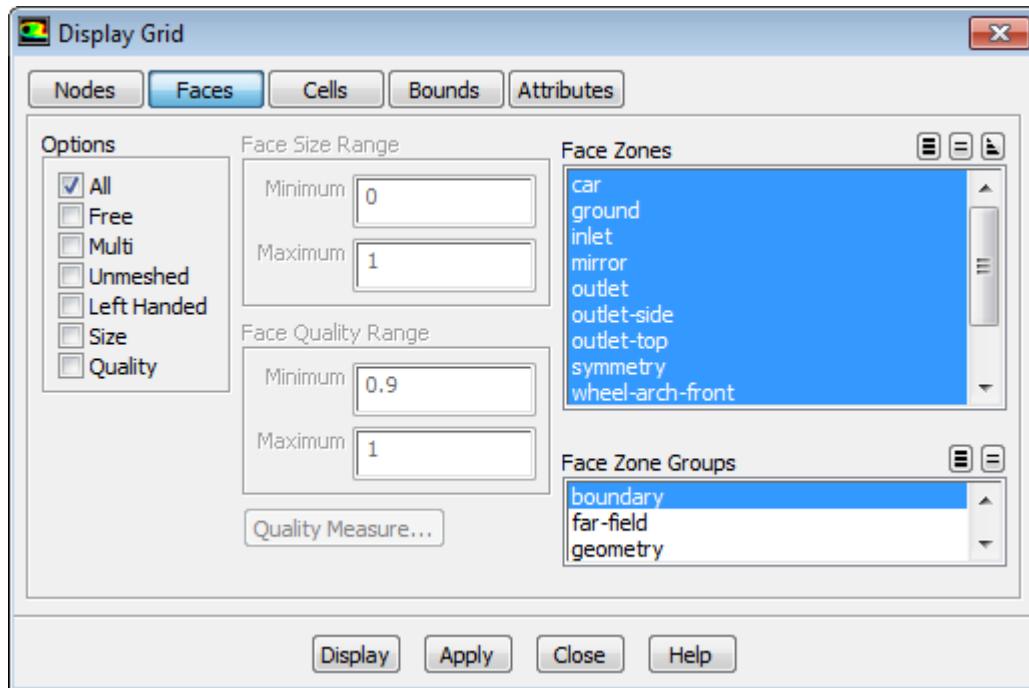
**File → Read → Boundary Mesh...**



- Select `sedan.msh.gz`.
- Click **OK**.

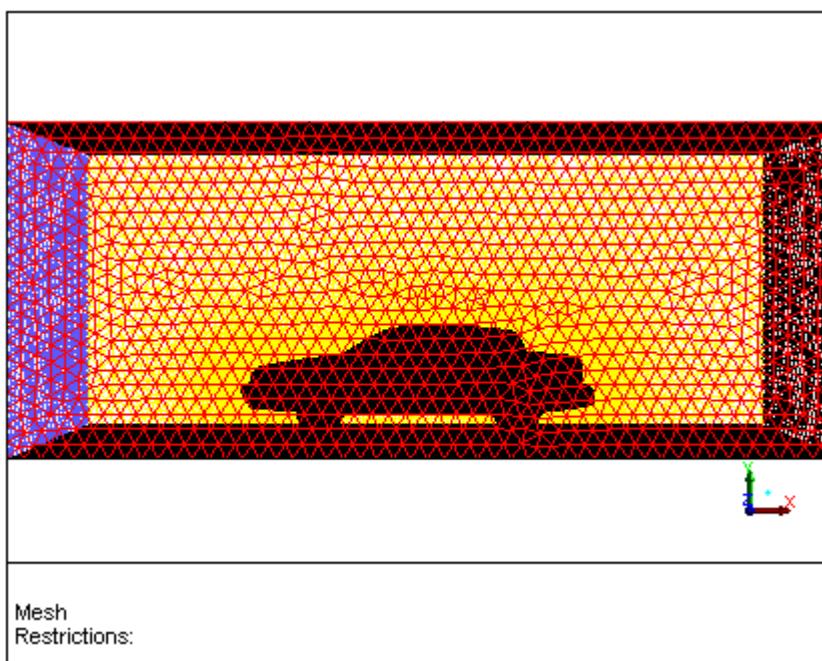
- Display the boundary mesh ([Figure 5.1: Boundary Mesh for the Sedan \(p. 85\)](#)).

**Display → Grid...**



- a. Select **boundary** in the **Face Zone Groups** selection list to select all boundary zones in the **Face Zones** selection list.
- b. Click **Display**.

**Figure 5.1: Boundary Mesh for the Sedan**

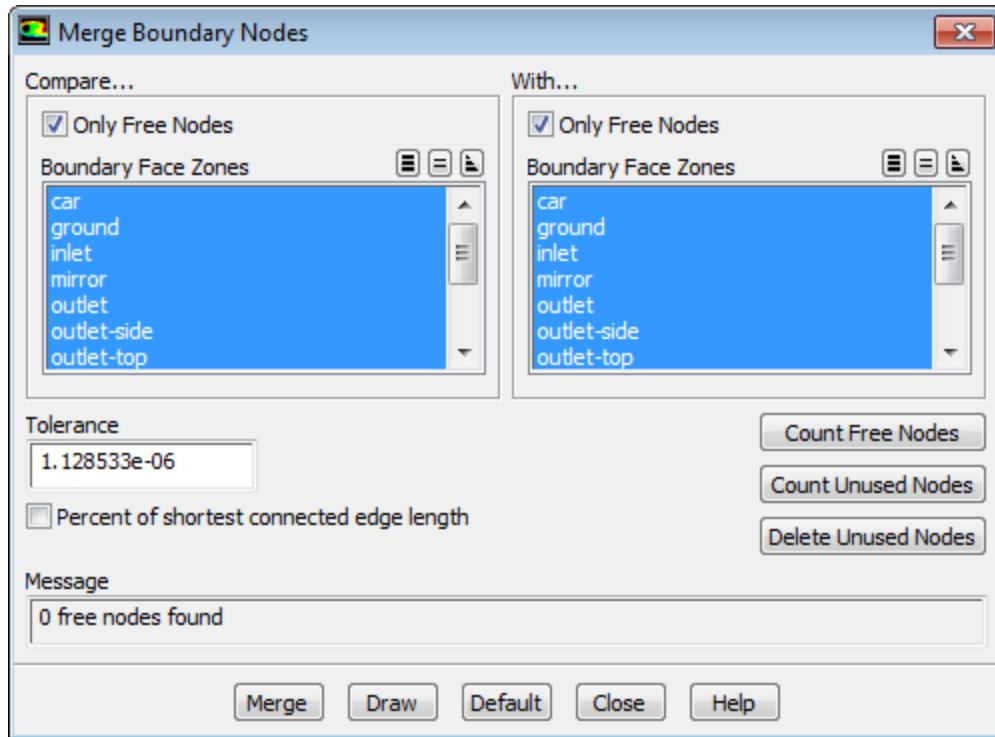


The mesh contains the boundary mesh of the sedan and the tunnel. Prisms will be generated on the body, the mirror, the wheels, and the ground. Critical areas are the wheel/ground intersection, the region of proximity of the wheels and wheel arches, and the mirror.

- c. Close the **Display Grid** dialog box.

## Check for Free and Unused Nodes

**Boundary → Merge Nodes...**



1. Click **Count Free Nodes**.

The number of free nodes is reported in the **Message** box. Click **Merge** to remove free nodes, if any.

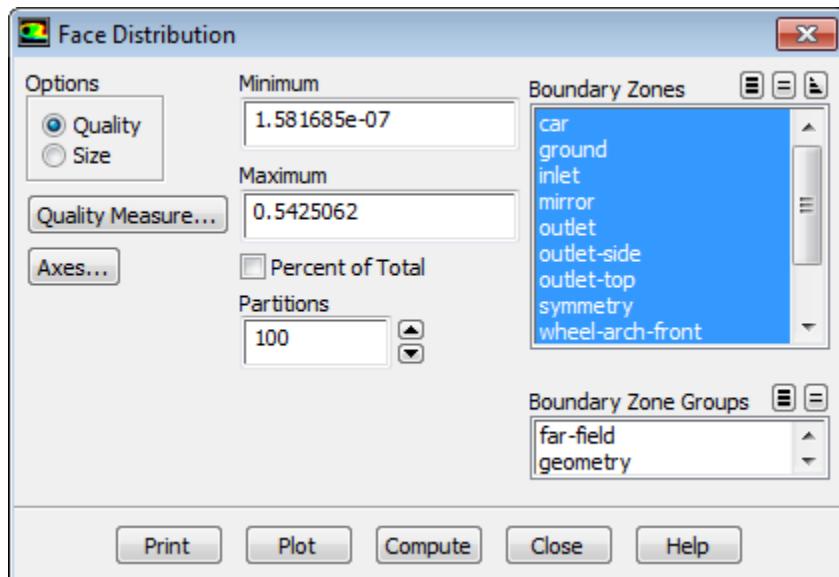
2. Click **Count Unused Nodes**.

The number of unused nodes is reported in the **Message** box. Click **Delete Unused Nodes** to remove unused nodes, if any. Free nodes are nodes associated with free edges. There should not be any free nodes unless there are thin walls in the geometry. If free nodes are located between a zone you are building prisms from and an adjacent zone, the mesher will be unable to project to and retriangulate the adjacent zone.

3. Close the **Merge Boundary Nodes** dialog box.

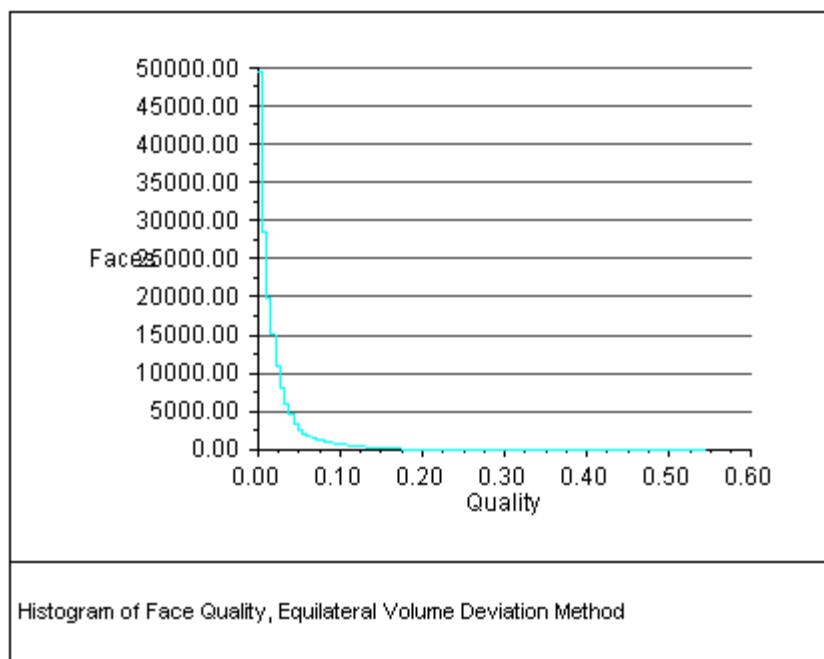
## Check the Quality of the Surface Mesh

**Display → Plot → Face Distribution...**



1. Select all the surfaces in the **Boundary Zones** selection list.
2. Click **Compute**.  
The maximum skewness value reported is 0.543 which is good enough to generate a hybrid mesh. When generating prisms on a surface mesh, the quality must not be higher than 0.7 or even 0.6 if many layers are to be extruded.
3. Click **Plot** (Figure 5.2: Surface Mesh Quality (p. 87)).

**Figure 5.2: Surface Mesh Quality**

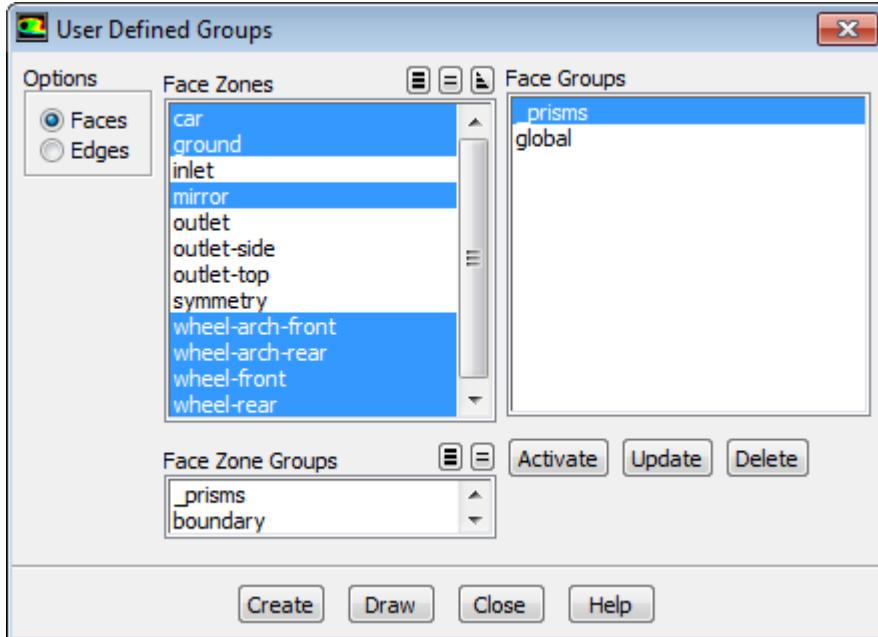


4. Close the **Face Distribution** dialog box.

## Generate the Mesh

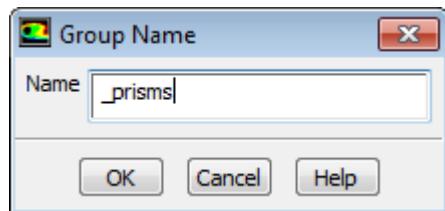
- Create a user-defined group for easier selection of the zones on which prisms are to be generated.

**Boundary → Zone → Group...**



- Select **car**, **ground**, **mirror**, **wheel-arch-front**, **wheel-arch-rear**, **wheel-front**, and **wheel-rear** in the **Face Zones** selection list.
- Click **Draw** and verify that the zones selected are appropriate.
- Click **Create**.

The **Group Name** dialog will open, prompting you to specify the group name.



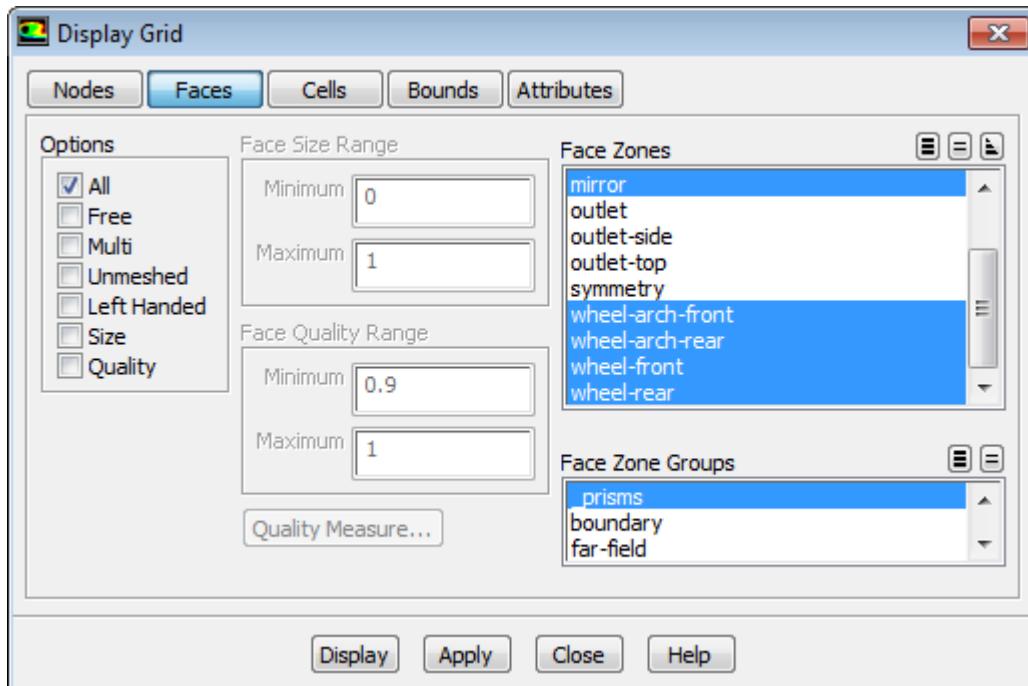
- Enter **\_prisms** for **Name** and click **OK**.

### Tip

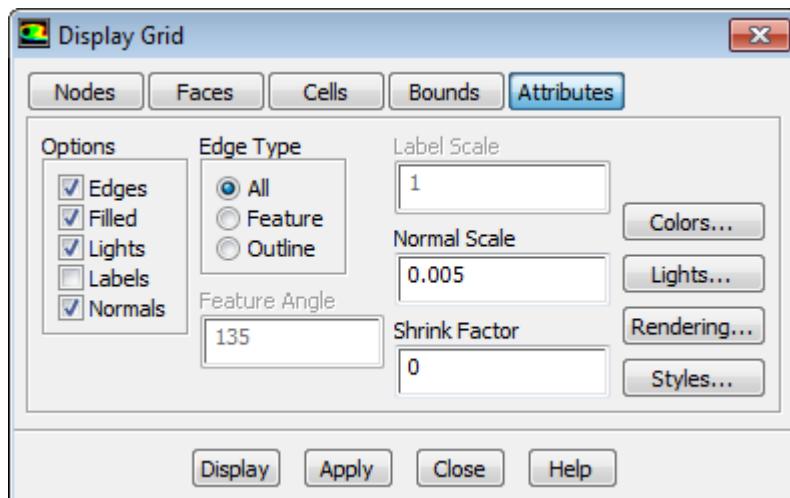
The use of the underscore (\_) in the group name allows the group to be listed at the top of the **Face Zone Groups** list.

- Close the **User Defined Groups** dialog box.
- Verify that the normals are correctly oriented.

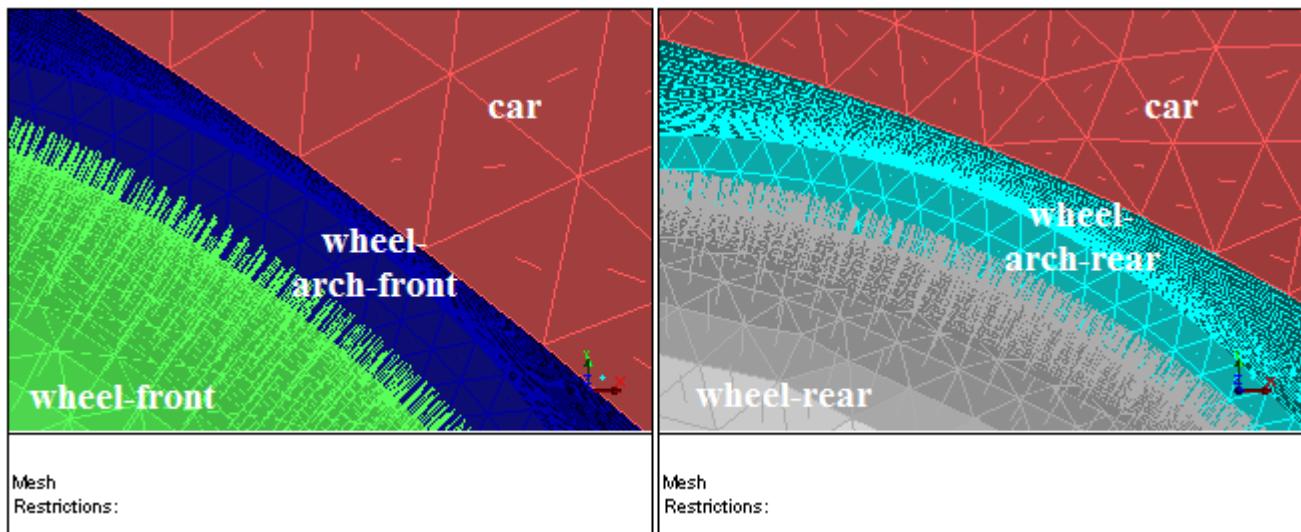
**Display → Grid...**



- Deselect the previous selection of **boundary** and select **\_prisms** in the **Face Zone Groups** selection list in the **Faces** tab.
- Click the **Attributes** tab and enable **Normals**.



- Enter 0.005 for **Normal Scale**.
- Enable **Filled** and **Lights** in the **Options** group box in the **Attributes** tab of the **Display Grid** dialog box.
- Click the **Colors...** button to open the **Grid Colors** dialog box.
- Select **Color by ID** in the **Options** list and close the **Grid Colors** dialog box.
- Click **Display** (Figure 5.3: Normals on the Wheel and Wheel Arch Zones (p. 90)).

**Figure 5.3: Normals on the Wheel and Wheel Arch Zones**

The normals on the selected zones in the **\_prisms** group point inside the flow region, except for those on the **wheel-arch-front** and **wheel-arch-rear** zones.

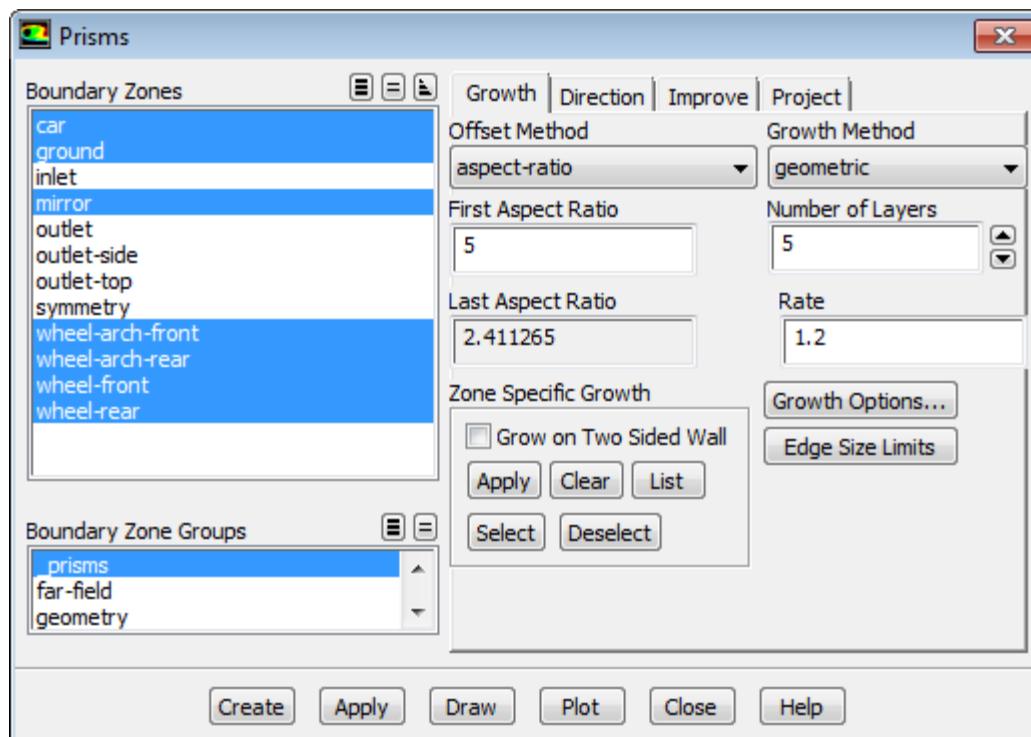
[Figure 5.3: Normals on the Wheel and Wheel Arch Zones \(p. 90\)](#) shows the normals on the wheels and wheel arches. You will need to reorient the normals on the **wheel-arch-front** and **wheel-arch-rear** zones for the prisms to be grown in the correct direction.

### Note

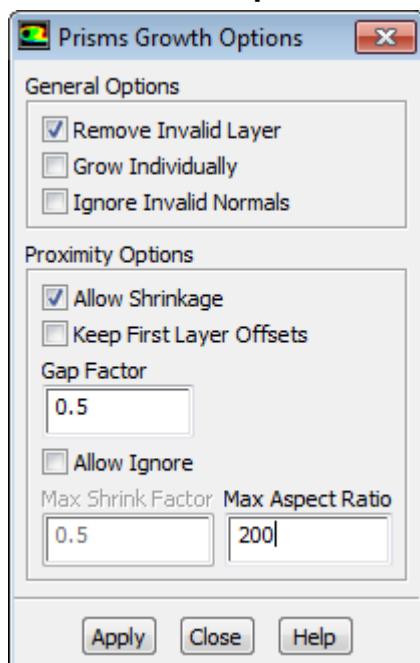
During prism meshing, the normals will always be oriented in the direction of most of the facets. Hence, if a small region is wrongly oriented, there will not be a problem with prisms grown.

- h. Close the **Display Grid** dialog box.
3. Set the prism meshing parameters.

**Mesh → Prisms...**



- Select **\_prisms** in the **Boundary Zone Groups** selection list.
- Select **aspect-ratio** in the **Offset Method** drop-down list and enter 5 for **First Aspect Ratio**.
- Select **geometric** in the **Growth Method** drop-down list and enter **1 . 2** for **Rate**.
- Set the **Number of Layers** to 5.
- Click **Apply** (next to **Create**).
- Click the **Growth Options...** button to open the **Prisms Growth Options** dialog box.



- i. Make sure **Allow Shrinkage** is enabled.

The **Allow Shrinkage** option allows you to enable the prism proximity algorithms, which prevent the prism cells colliding with each other in areas of proximity and sharp angles.

### Tip

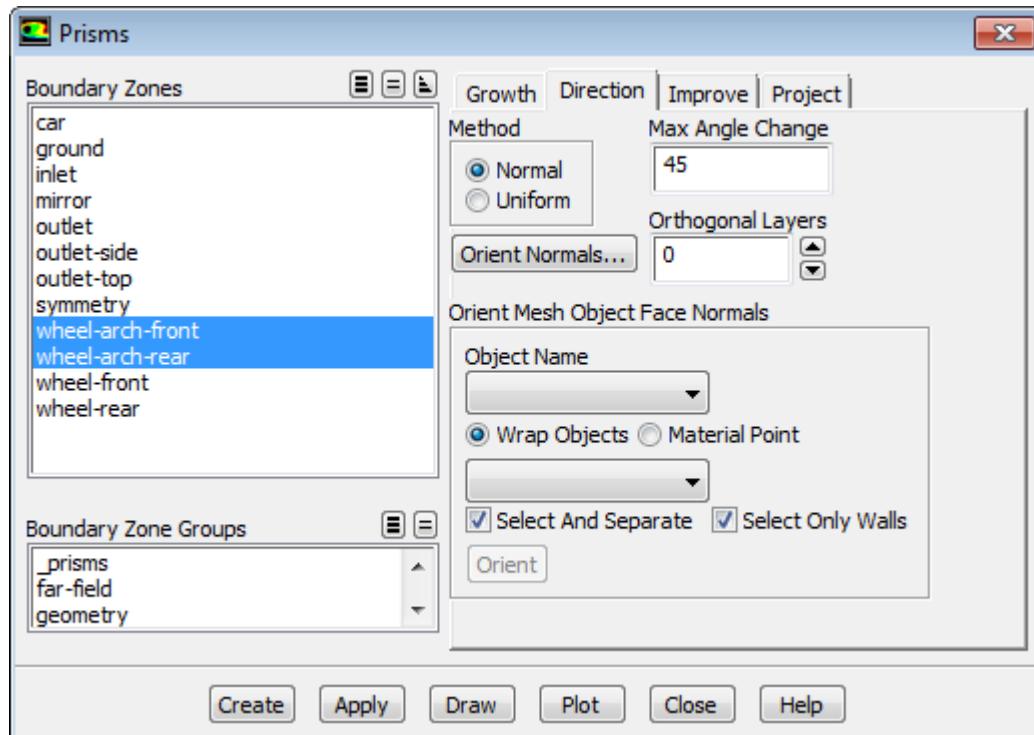
You can use the command /mesh/prism/controls/proximity/smoothing-rate to obtain a better transition between the shrunk and unshrunk layers. Reducing the value specified can improve the mesh transition.

- ii. Enter 200 for **Max Aspect Ratio** and click **Apply**.

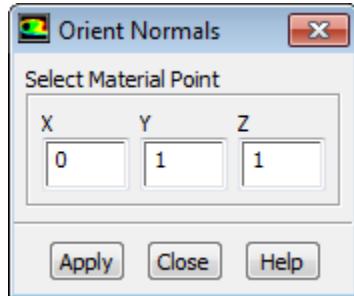
The **Max Aspect Ratio** limits the shrinkage of the prism layers based on the local edge length, while the **Gap Factor** value controls the gap between the opposing prism layers.

- iii. Close the **Prisms Growth Options** dialog box.

- g. Click the **Direction** tab in the **Prisms** dialog box.

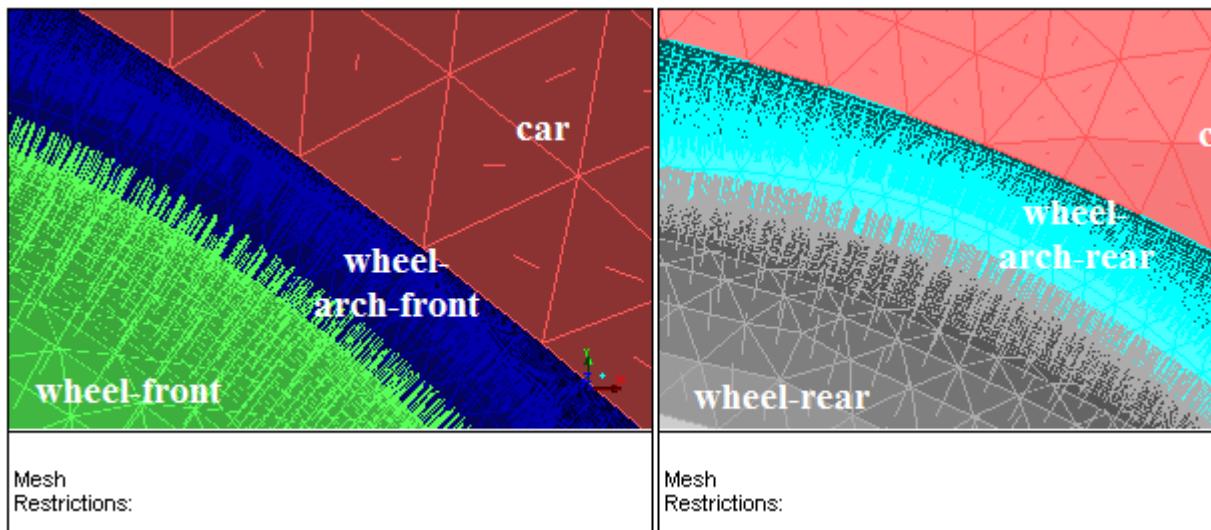


- i. Deselect all the zones selected in the **Boundary Zones** selection list and select only **wheel-arch-front** and **wheel-arch-rear**.
- ii. Click the **Orient Normals...** button to open the **Orient Normals** dialog box.

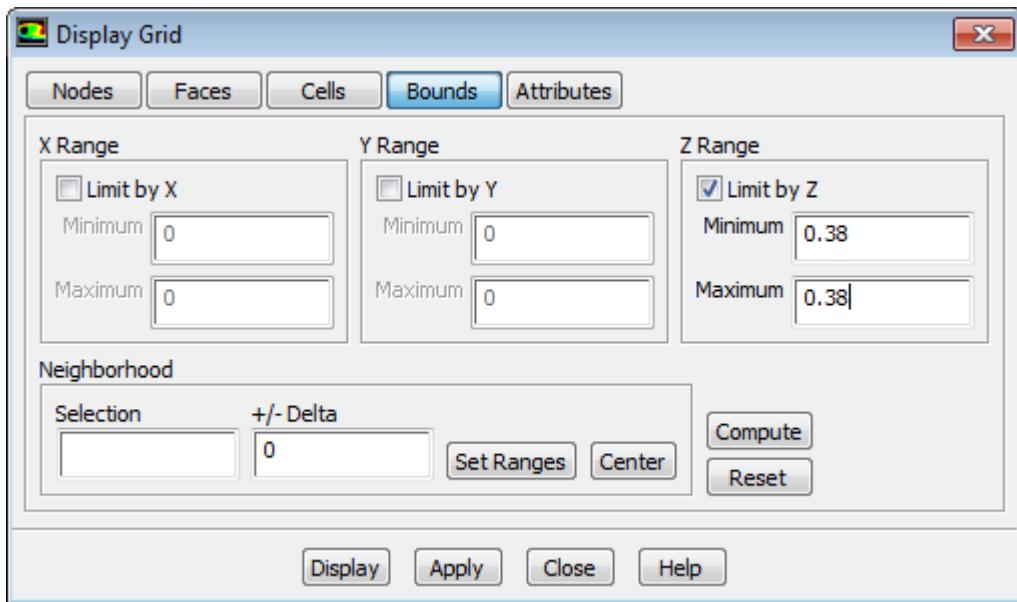


- A. Enter a location within the flow region (e.g., 0, 1, 1 for **X**, **Y**, and **Z**, respectively).
- B. Click **Apply** to orient the normals correctly (Figure 5.4: Normals on Wheels and Wheel Arches After Reorienting (p. 93)).

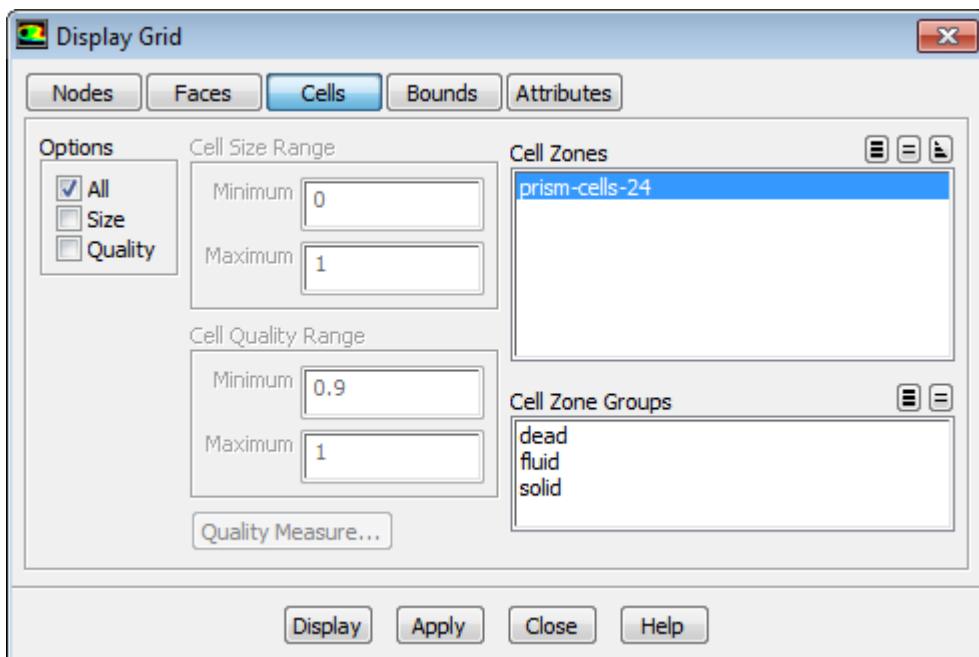
**Figure 5.4: Normals on Wheels and Wheel Arches After Reorienting**



- C. Close the **Orient Normals** dialog box.
  - h. Deselect the zones in the **Boundary Zones** selection list and select **\_prisms** in the **Boundary Zone Groups** selection list.
  - i. Click **Create**.
  - j. Close the **Prisms** dialog box.
4. Examine the prisms generated by displaying a slide of cells at  $z = 0.38$ .
- Display → Grid...**
- a. Retain the selection of the previously selected zones in the **Face Zones** selection list.
  - b. Click the **Bounds** tab and enable **Limit by Z**.

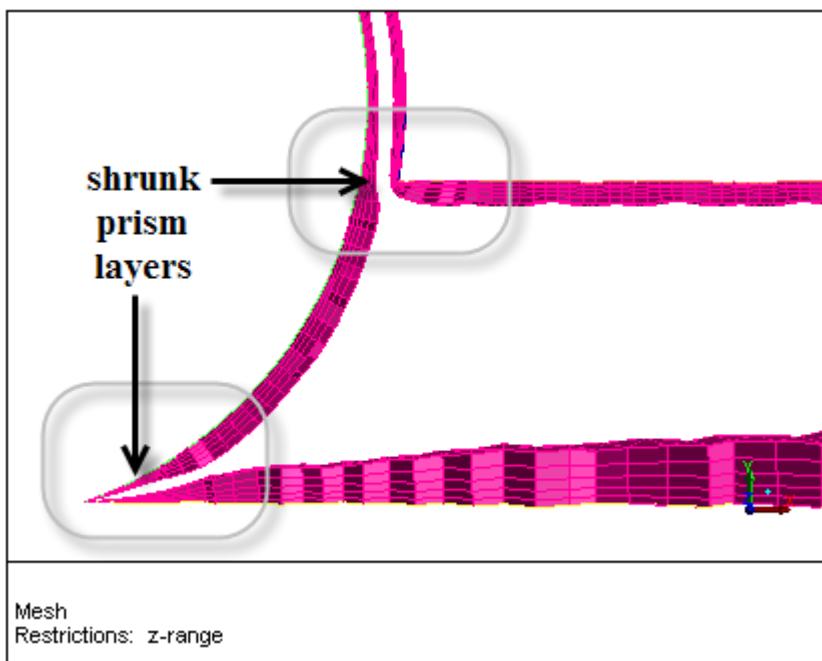


- c. Enter 0.38 for both **Minimum** and **Maximum** in the **Z Range** group box.
- d. Click the **Cells** tab and select the prism cells zone (**prism-cells-#**, where # is the zone ID) in the **Cell Zones** selection list.



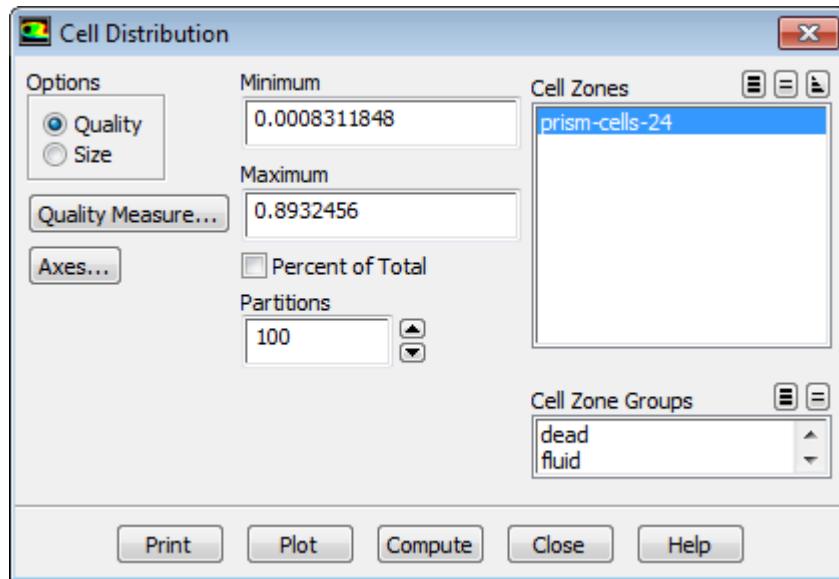
- e. Enable **All** in the **Options** group box and click **Display**.  
Make sure the display of normals is disabled in the Attributes tab.
- f. Zoom in to the region of the wheel/ground intersection and the gap between the car body and the wheels ([Figure 5.5: Prisms Shrunk in Areas of Proximity and Sharp Corners \(p. 95\)](#)).

**Figure 5.5: Prisms Shrunk in Areas of Proximity and Sharp Corners**



- g. Close the **Display Grid** dialog box.
5. Check the quality of the prism cells generated.

**Display → Plot → Cell Distribution...**



- a. Select the prism cell zone in the **Cell Zones** selection list and click **Compute**.

The quality reported is around 0.893.

### Tip

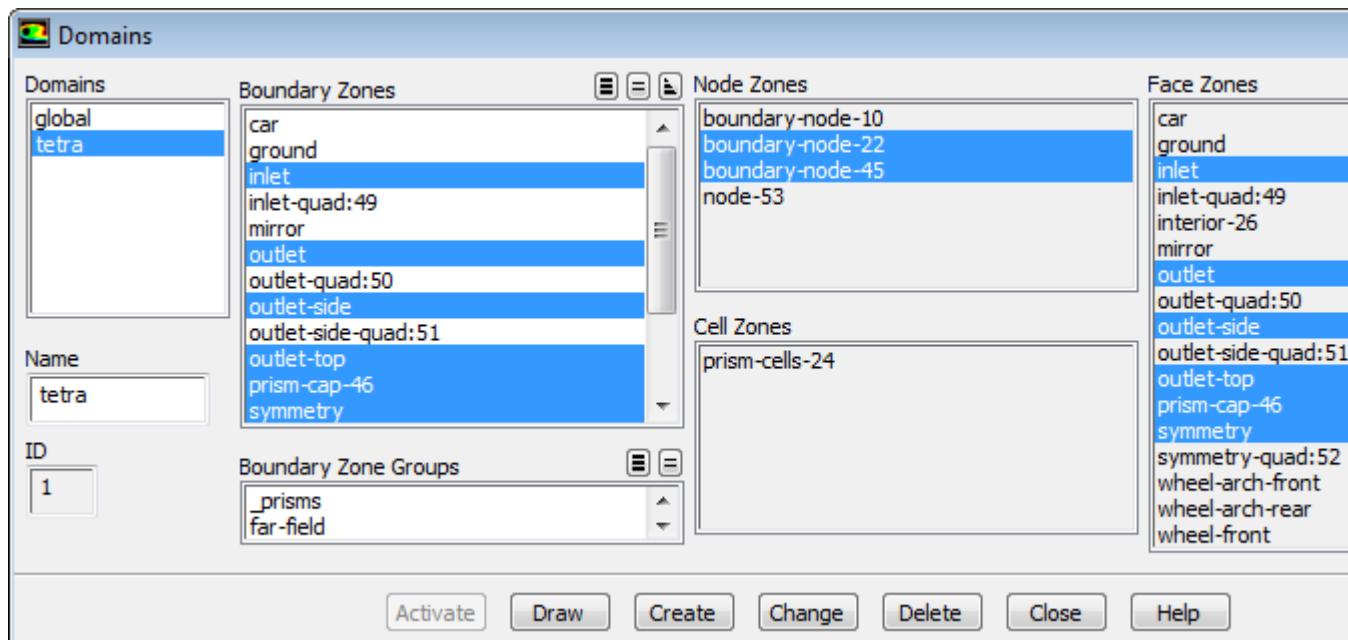
If the prism cell quality is greater than 0.9, you can use the options in the **Prism Improve** dialog box to improve the quality.

**Mesh → Tools → Prism → Improve...**

- b. Close the **Cell Distribution** dialog box.

6. Define the tetrahedral domain.

**Mesh → Domains...**



- a. Select **inlet**, **outlet**, **outlet-side**, **outlet-top**, **prism-cap-#**, and **symmetry** in the **Boundary Zones** selection list.
- b. Enter **tetra** for **Name** and click **Create**.

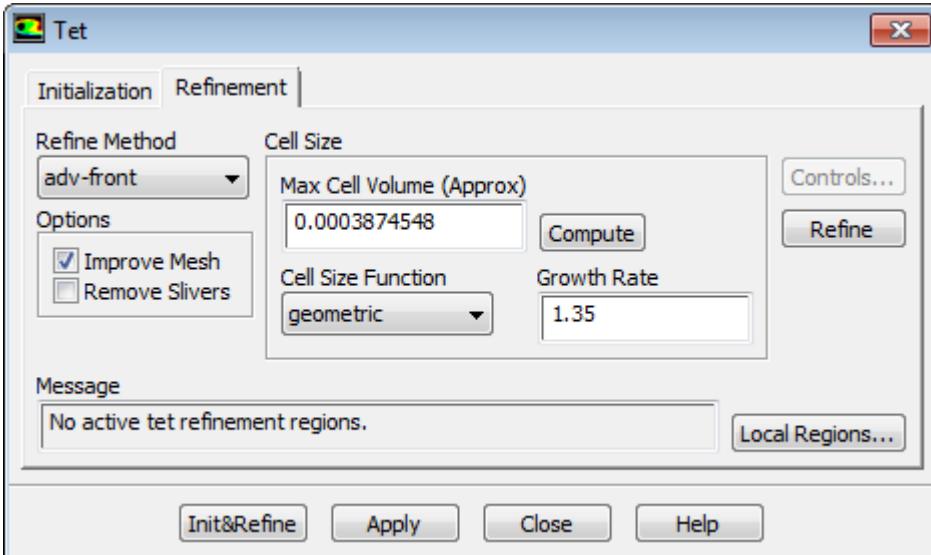
The domain **tetra** will be created and activated.

- c. Verify that all the zones required are included in the defined domain.

**Display → Grid...**

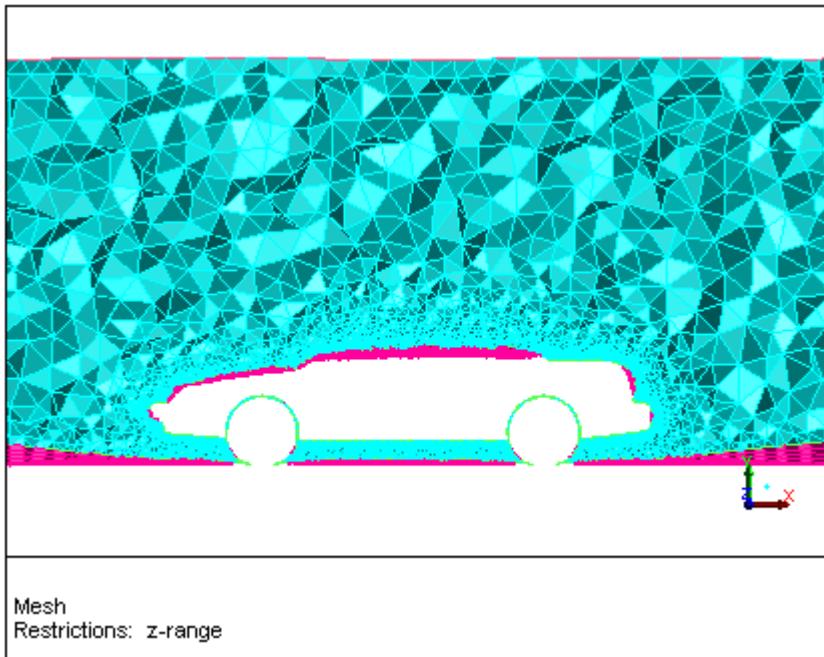
- i. Click the **Bounds** tab and click **Reset**.
- ii. Select all the zones in the **Face Zones** selection list in the **Faces** tab.
- iii. Enable **Free** in the **Options** group box.

Enabling the display of free nodes allows you to verify whether the domain is correctly defined. If any zone is not included in the domain, its adjacent zones will have free nodes.

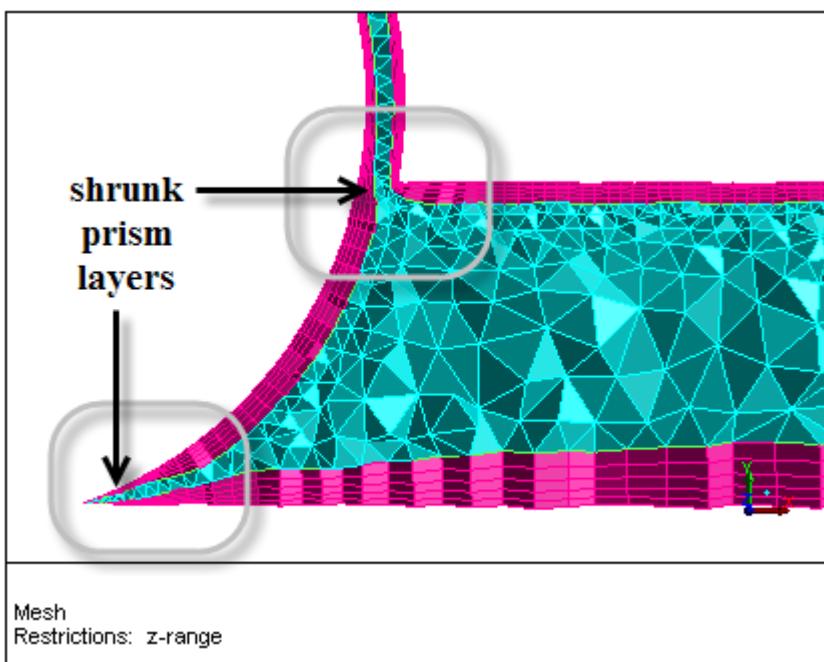
- iv. Click **Display**.
  - v. Close the **Display Grid** dialog box.
  - d. Close the **Domains** dialog box.
7. Set the parameters for tetrahedral meshing.
- Mesh → Tet...**
- a. Retain the settings in the **Initialization** tab.
  - b. Click the **Refinement** tab and retain the selection of **adv-front** in the **Refine Method** drop-down list.
  - c. Select **geometric** in the **Cell Size Function** drop-down list and enter **1.35** for **Growth Rate**.
- 
- The screenshot shows the 'Tet' dialog box with the 'Refinement' tab selected. Under 'Refine Method', 'adv-front' is chosen. In the 'Cell Size' section, 'Max Cell Volume (Approx)' is set to 0.0003874548, 'Cell Size Function' is set to 'geometric', and 'Growth Rate' is set to 1.35. The 'Options' section contains checked boxes for 'Improve Mesh' and 'Remove Slivers'. A message box at the bottom states 'No active tet refinement regions.' and includes a 'Local Regions...' button. At the bottom are 'Init&Refine', 'Apply', 'Close', and 'Help' buttons.
- d. Click **Apply** and **Init & Refine**.
  - e. Close the **Tet** dialog box.
8. Check the quality of the mesh.
- Display → Plot → Cell Distribution...**
- The maximum skewness reported is around 0.953.
9. Activate the global domain.
- Mesh → Domains...**
- a. Select **global** in the **Domains** list and click **Activate**.
  - b. Close the **Domains** dialog box.
10. Examine the mesh by displaying a slide of cells at  $z = 0.38$ .

**Display → Grid...**

- a. Enable **Limit by Z** in the **Bounds** tab and enter 0 . 38 for both **Minimum** and **Maximum**.
- b. Select the fluid and prism cell zones in the **Cell Zones** selection list in the **Cells** tab.
- c. Enable **All** in the **Options** group box and click **Display** (Figure 5.6: Slide of Cells at z = 0.38 (p. 98)).

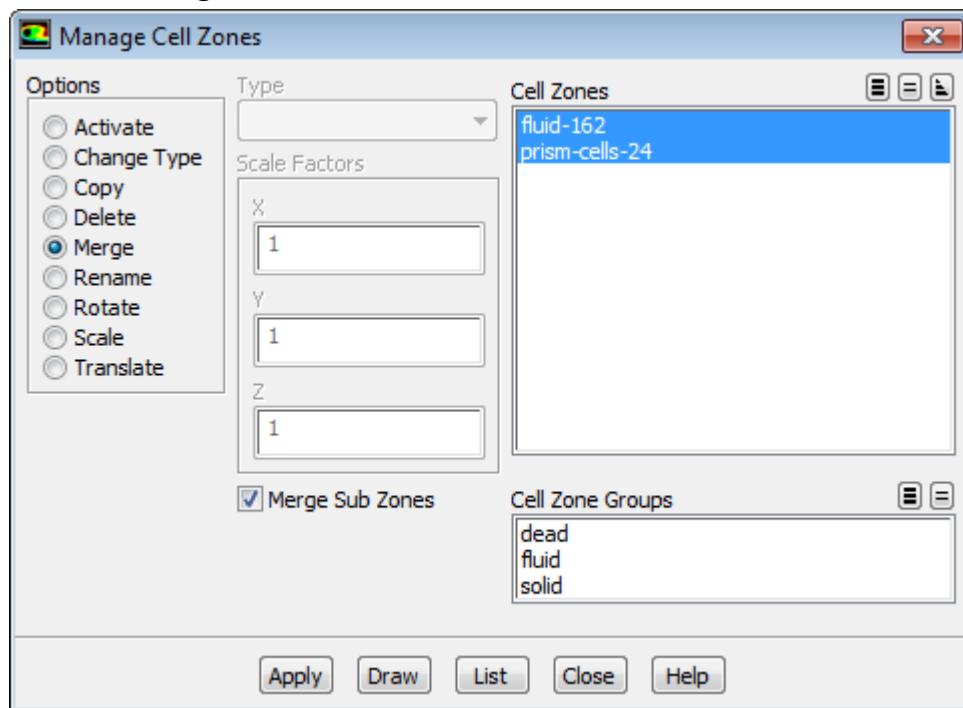
**Figure 5.6: Slide of Cells at z = 0.38**

- d. Examine the areas of proximity (Figure 5.7: Prisms Shrunk in Areas of Proximity (p. 98)).

**Figure 5.7: Prisms Shrunk in Areas of Proximity**

11. Merge the cell zones generated.

**Mesh → Manage...**



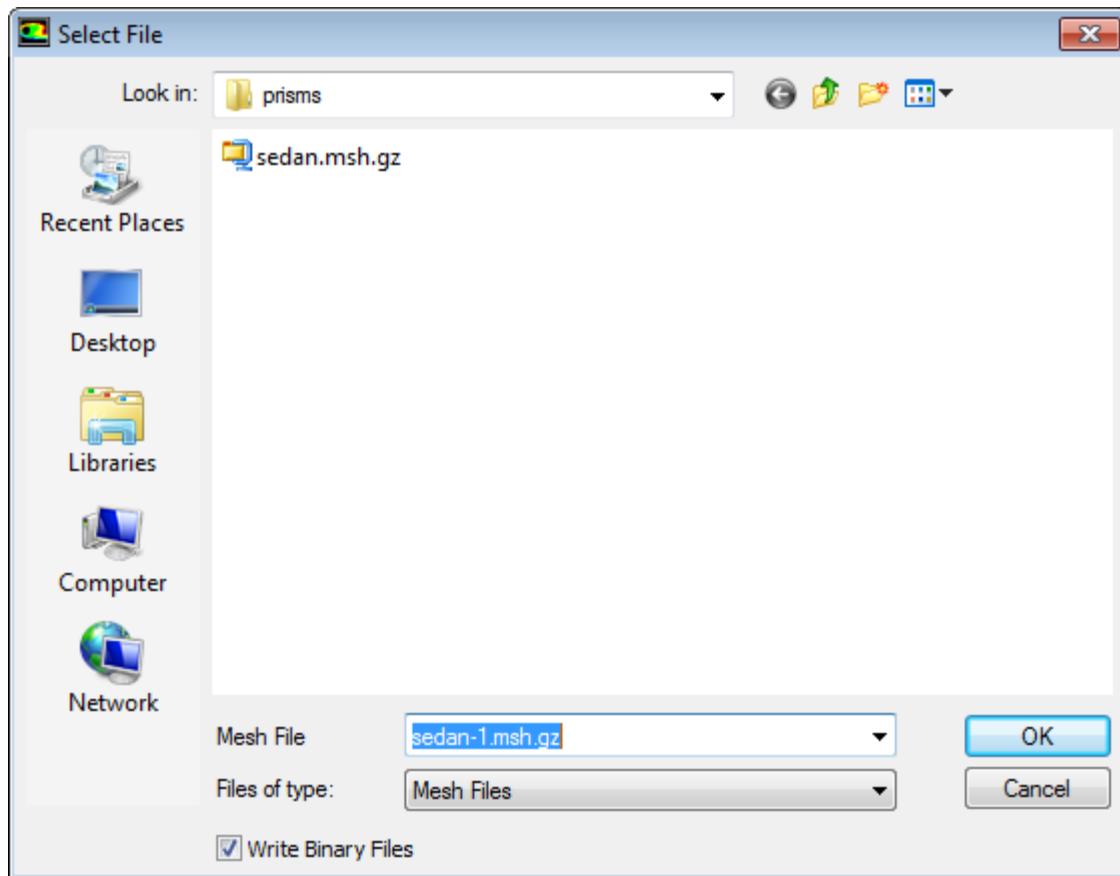
- a. Select the fluid and prism cell zones in the **Cell Zones** selection list.
- b. Select **Merge** in the **Options** list.
- c. Enable **Merge Sub Zones** and click **Apply**.
- d. Close the **Manage Cell Zones** dialog box.

12. Check the mesh.

**Mesh → Check**

13. Save the mesh (sedan-1.msh.gz).

**File → Write → Mesh...**



- a. Enter `sedan-1.msh.gz` for **Mesh File**.
- b. Click **OK**.

## 5.4. Generate the Mesh Using the Allow Ignore Option and Automatic Meshing

The previous section described the set up and manual meshing procedure for a viscous mesh with prisms and tetrahedra. This demonstrated how to work with domains. This section demonstrates the use of the **Auto Mesh** option to set all the meshing parameters and automatically generate the viscous mesh in a single step, thereby removing the need for creating the domain excluding the prism region as an intermediate step.

1. Read the mesh file (`sedan.msh.gz`).

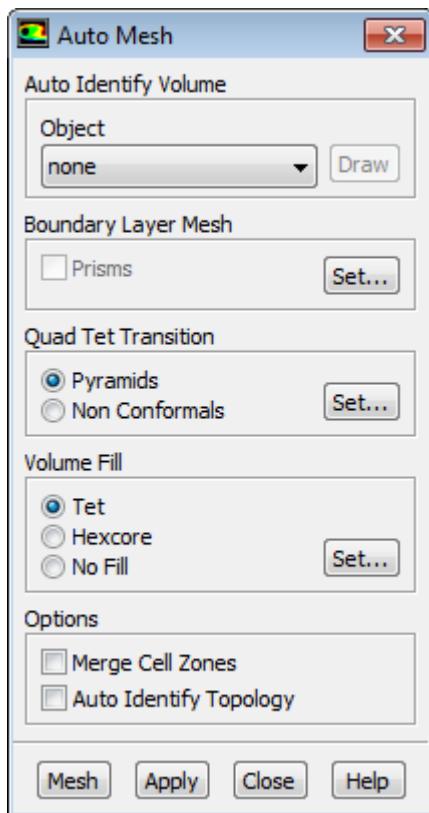
**File → Read → Mesh...**

2. Display the grid and verify that the normals are correctly oriented.

**Display → Grid...**

The normals on the **car**, **mirror**, **wheel-arch-front**, **wheel-arch-rear**, **wheel-front**, and **wheel-rear** point outward, while those on the **ground** point upward. You will need to reorient the normals on the **wheel-arch-front** and **wheel-arch-rear** zones.

3. Set the meshing parameters.

**Mesh → Auto Mesh...**

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

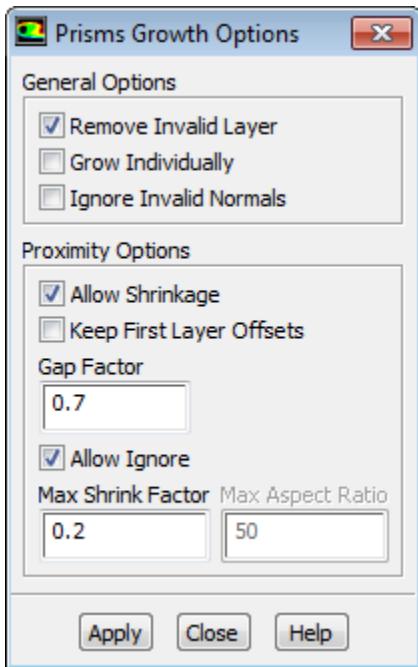
- Click the **Set...** button in the **Boundary Layer Mesh** group box to open the **Prisms** dialog box.
  - Click the **Direction** tab in the **Prisms** dialog box.
    - Select **wheel-arch-front** and **wheel-arch-rear** and click **Orient Normals...** to open the **Orient Normals** dialog box.
    - Enter 0, 1, 1 for **X**, **Y**, **Z**, respectively.
    - Click **Apply** and close the **Orient Normals** dialog box.
  - Click the **Growth** tab and set the prism growth parameters.
    - Select **car**, **ground**, **mirror**, **wheel-arch-front**, **wheel-arch-rear**, **wheel-front**, and **wheel-rear** in the **Boundary Zones** selection list.
    - Select **aspect-ratio** in the **Offset Method** drop-down list and enter 5 for **First Aspect Ratio**.
    - Select **geometric** in the **Growth Method** drop-down list and enter 1 . 2 for **Rate**.
    - Set **Number of Layers** to 5.

- E. Click **Apply** in the **Zone Specific Growth** group box.

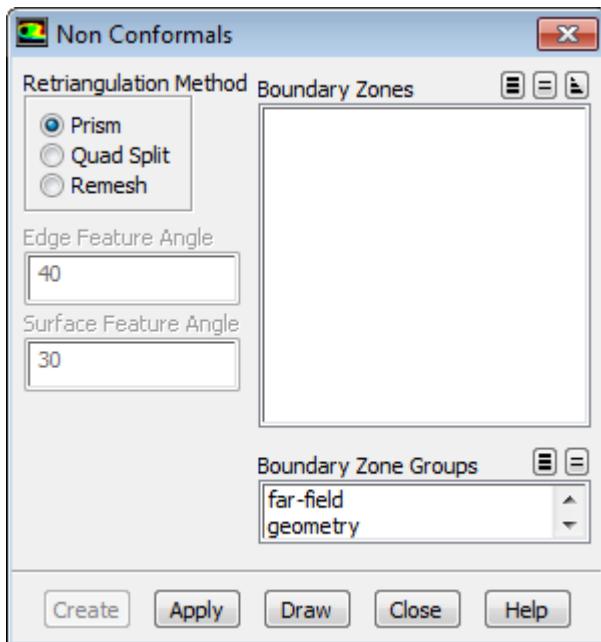
**Warning**

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

- iii. Click the **Growth Options...** button to open the **Prisms Growth Options** dialog box.



- A. Retain the **Allow Shrinkage** option and enable **Allow Ignore**.
- B. Enter 0.7 for **Gap Factor** and 0.2 for **Max Shrink Factor**, respectively.
- C. Click **Apply** and close the **Prisms Growth Options** dialog box.
- iv. Click **Apply** and close the **Prisms** dialog box.
- b. Enable **Prisms** in the **Auto Mesh** dialog box.
- c. Select **Non Conformals** in the **Quad Tet Transition** group box and click the **Set...** button to open the **Non Conformals** dialog box.

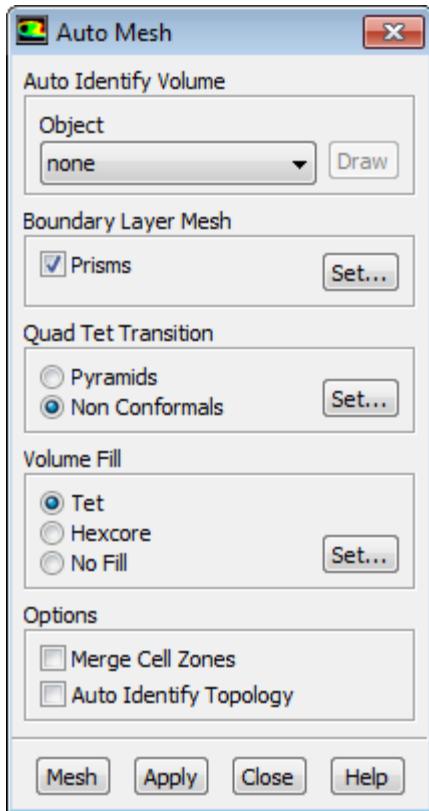


- i. Retain the selection of **Prism** in the **Retriangulation Method** list and click **Apply**.

When the **Quad Split** method of retriangulation is selected, each quadrilateral face zone will be copied and the quadrilaterals on the copied zone will then be split to form triangles. This method is recommended when the quadrilaterals are close to perfect squares (low aspect ratio).

In this case, however, the quadrilaterals are of high aspect ratio, and the use of the **Quad Split** method would create highly skewed triangles, and consequently highly skewed tetrahedra. Of the remaining retriangulation methods available, the **Remesh** option will only take into account the nodes on the edge loop of the quadrilateral zone during retriangulation. The **Prism** option will, however, consider the 'ribs' (lines joining the base nodes and the cap nodes along the prism-side) during the retriangulation, thus, giving a better quality triangular mesh for the curved prism sides.

- ii. Close the **Non Conformals** dialog box.
- d. Retain the selection of **Tet** in the **Volume Fill** group box and click the **Set...** button to open the **Tet** dialog box.
  - i. Retain the settings in the **Initialization** tab.
  - ii. Click the **Refinement** tab and retain the selection of **adv-front** in the **Refine Method** drop-down list.
  - iii. Select **geometric** in the **Cell Size Function** drop-down list and enter **1 . 35** for **Growth Rate**.
  - iv. Click **Apply** and close the **Tet** dialog box.



- e. Click **Mesh**.
  - f. Close the **Auto Mesh** dialog box.
4. Examine the mesh by displaying a slide of cells at  $z = 0.38$ .

**Display → Grid...**

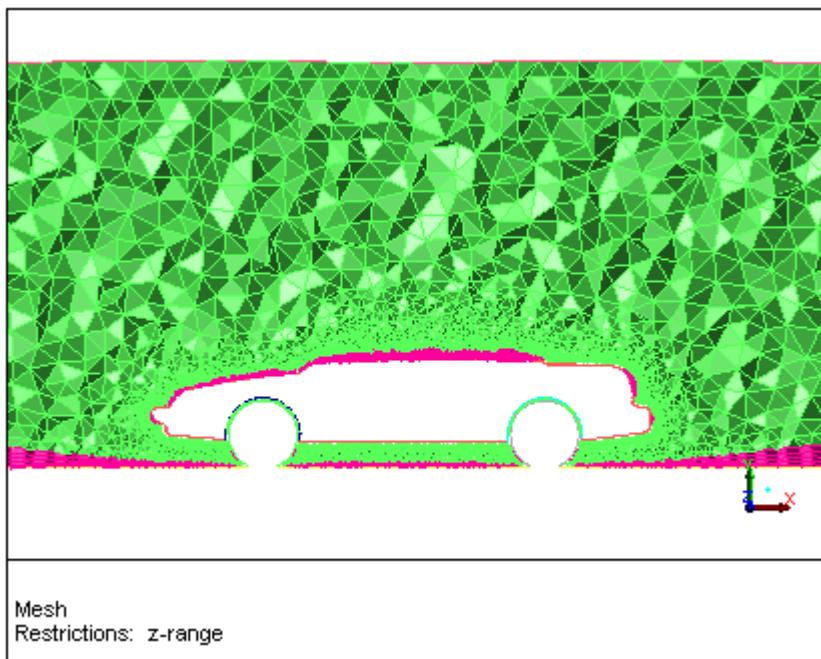
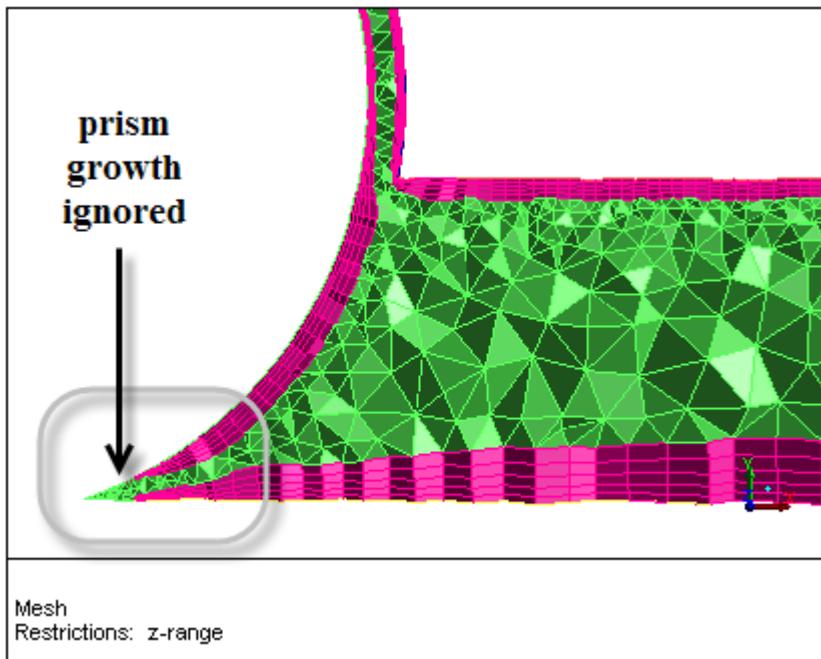
**Figure 5.8: Slide of Cells at z = 0.38****Figure 5.9: Prisms Ignored in Areas of Proximity**

Figure 5.9: Prisms Ignored in Areas of Proximity (p. 105) shows that prism layer growth was ignored in areas of proximity such as sharp corners.

5. Check the quality of the mesh.

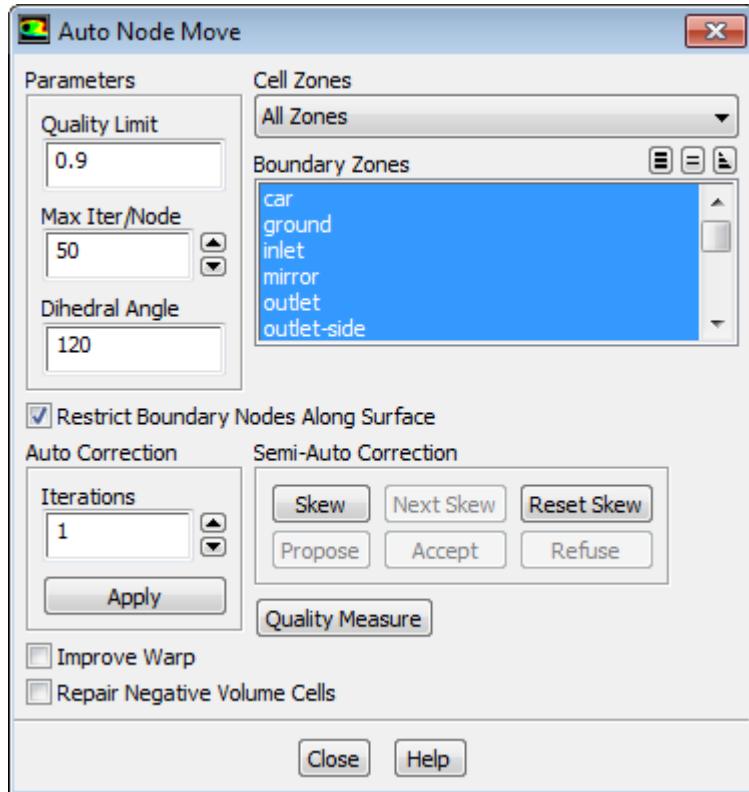
**Display → Plot → Cell Distribution...**

The maximum skewness of the prism cells is around 0.82, while that of the tetrahedral cells is 0.973.

6. Merge the cell zones.

**Mesh → Manage...**

7. Improve the volume mesh using the **Auto Node Move** utility.

**Mesh → Tools → Auto Node Move...**

- Retain the value of **0 . 9** for **Quality Limit**.
- Retain the selection of **All Zones** in the **Cell Zones** drop-down list.
- Select all the zones in the **Boundary Zones** selection list.
- Retain the other default settings and click **Apply** in the **Auto Correction** group box.

The maximum skewness is now reported to be around 0.91, which is acceptable.

8. Check the mesh.

**Mesh → Check**

9. Save the mesh (`sedan-2.msh.gz`).

## 5.5. Summary

This tutorial demonstrated the creation of a viscous hybrid mesh starting from a triangular mesh. The controls available for creating prisms from multiple zones and the additional growth options for areas of proximity and sharp corners were demonstrated. The tutorial also demonstrated the use of the **Auto Mesh** tool for creating the viscous hybrid mesh.

---

## Chapter 6: Hexcore Mesh Generation

---

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. 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:

- Read and display the mesh.
- Check the skewness of the surface mesh.
- Improve the boundary mesh quality.
- Generate the hexcore mesh.
- Examine the effect of buffer layers on the hexcore mesh.
- Generate the hexcore mesh in conjunction with boundary layer (prism) meshing and local refinement.

### 6.1. Prerequisites

This tutorial assumes that you have some experience with the meshing mode in ANSYS FLUENT, and that you are familiar with the graphical user interface.

### 6.2. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

To prepare for running this tutorial:

1. Download the tutorial input file (`hexcore.zip`) for the tutorial.
2. Unzip `hexcore.zip`.

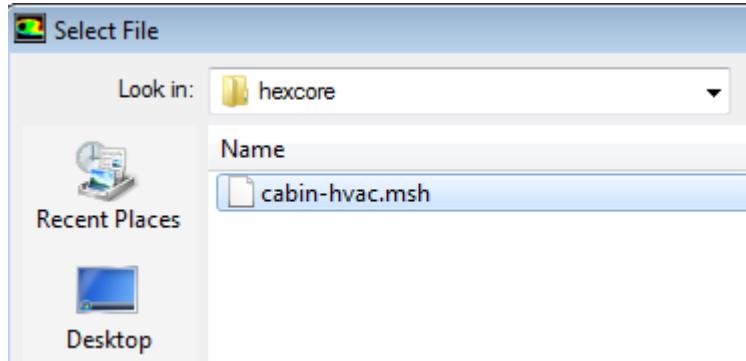
The file, `cabin-hvac.msh` can be found in the `hexcore` folder created on unzipping the file.

3. Start ANSYS FLUENT in meshing mode. For detailed steps, refer to [Starting ANSYS FLUENT in Meshing Mode \(p. 4\)](#).

## 6.3. Read and Display the Mesh

1. Read the mesh.

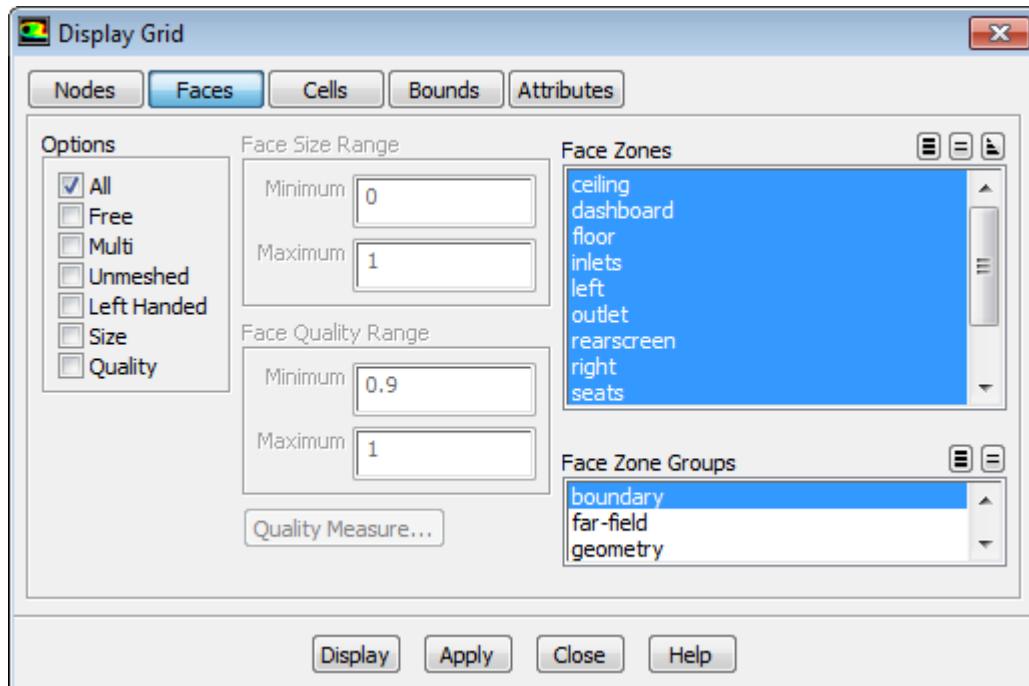
**File → Read → Mesh...**



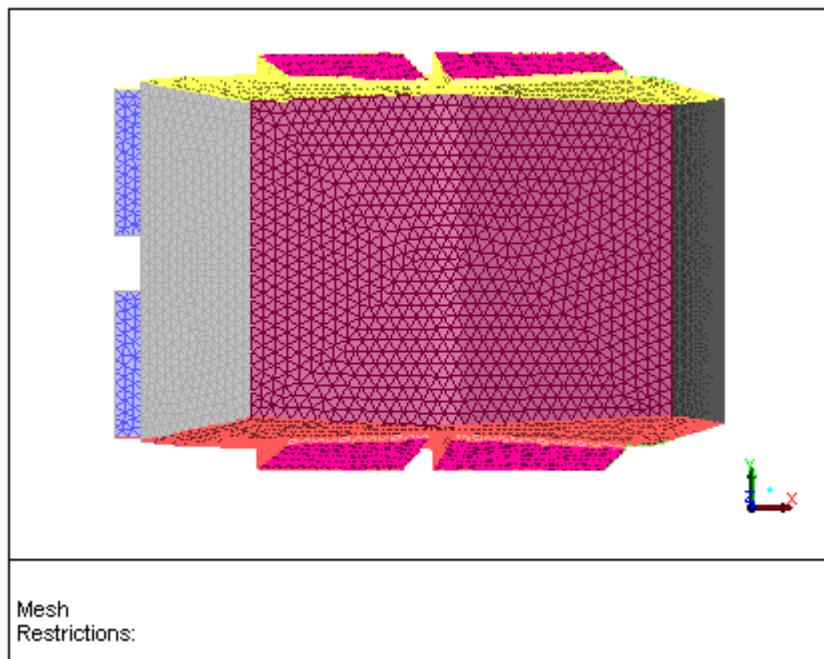
- a. Select `cabin-hvac.msh`.
- b. Click **OK**.

2. Display the mesh (Figure 6.1: Grid Display (p. 109)).

**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**.

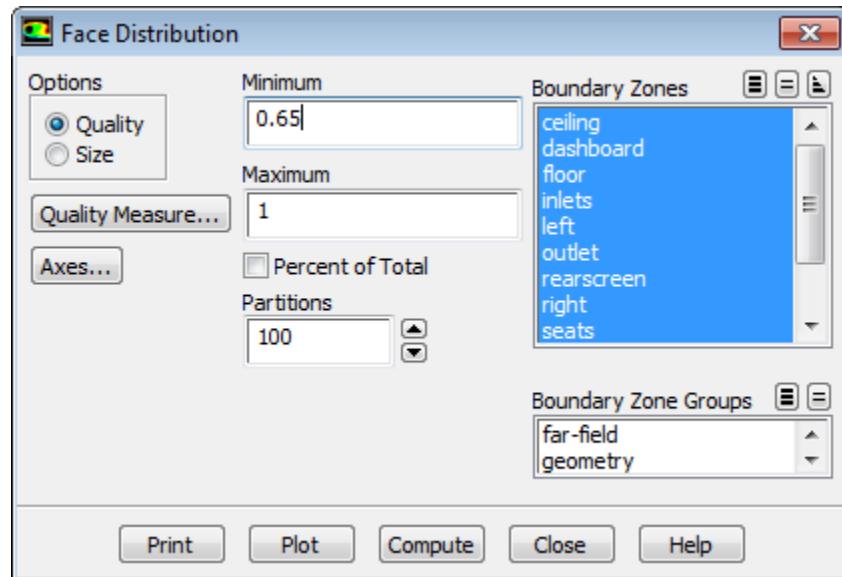
**Figure 6.1: Grid Display**

- c. Close the **Display Grid** dialog box.

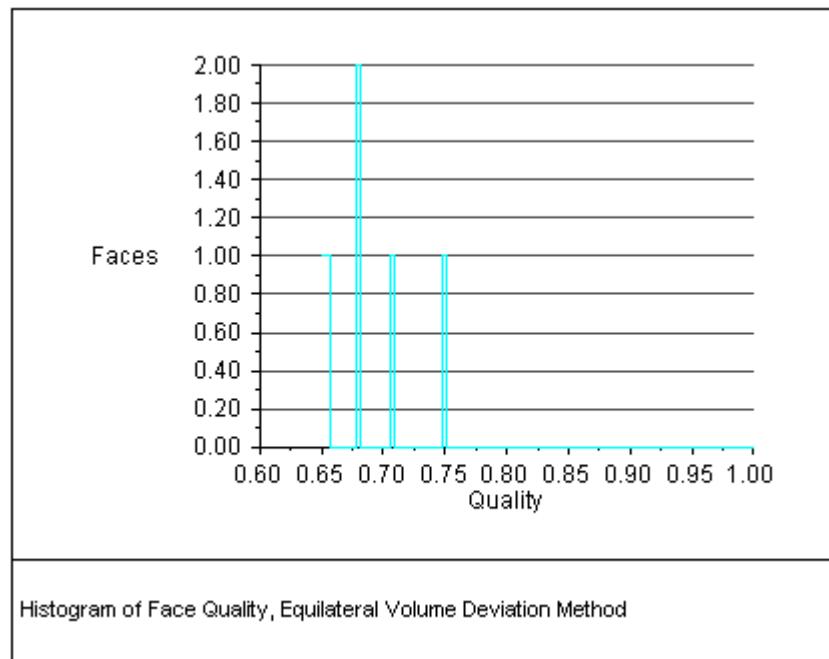
## 6.4. Check the Skewness of the Surface Mesh

Find the number of faces with a skewness greater than 0.65.

**Display → Plot → Face Distribution...**



1. Select all the zones in the **Boundary Zones** selection list.
2. Enter 0 . 65 for **Minimum**.
3. Click **Plot** (Figure 6.2: Skewness Distribution Above 0.65 (p. 110)).

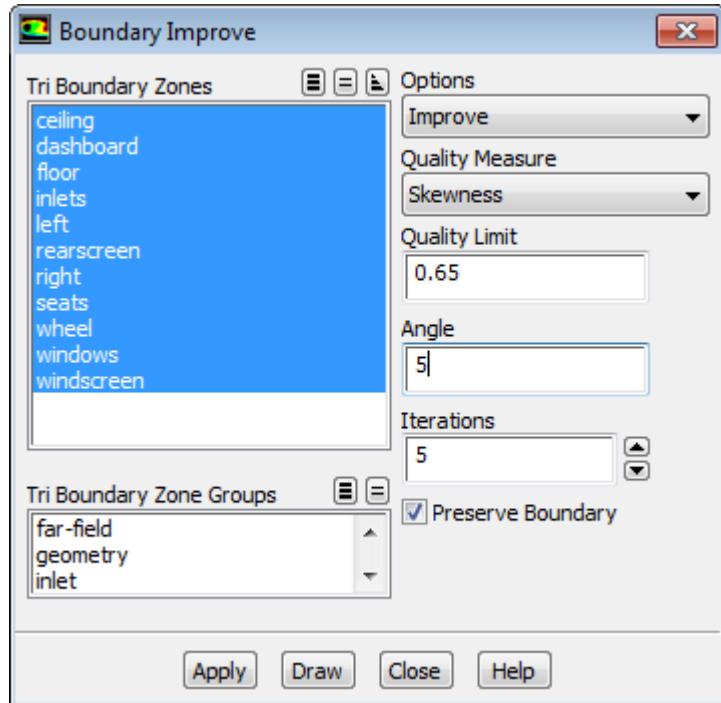
**Figure 6.2: Skewness Distribution Above 0.65**

4. Close the **Face Distribution** dialog box.

## 6.5. Improve the Boundary Mesh

1. Improve the faces having skewness greater than 0.65.

**Boundary** → **Mesh** → **Improve...**



- 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**.

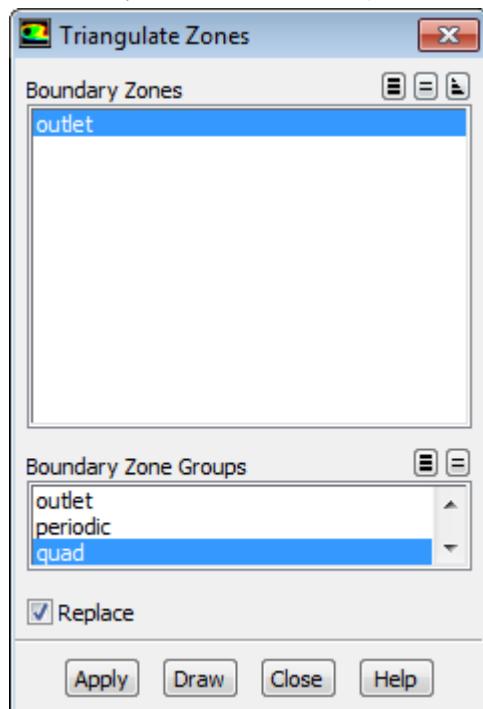
Specifying a lower value for **Angle** will reduce the modifications made to the boundary and better maintain the geometry.

- d. Retain the value of 5 for **Iterations** and click **Apply**.

The 6 faces above the specified maximum quality (0.65) will be fixed and it will be reported that the current maximum quality is approximately 0.649.

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

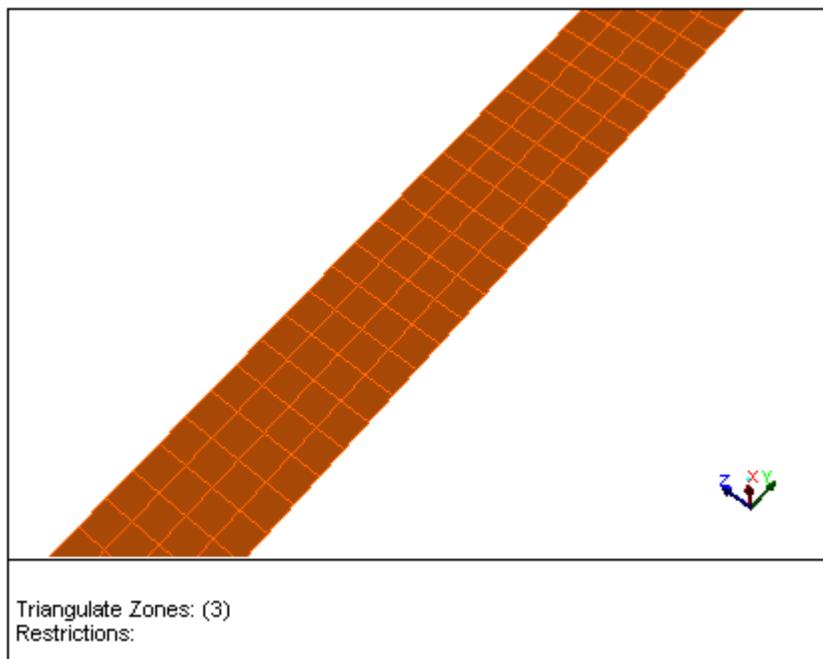
**Boundary** → **Mesh** → **Triangulate...**



- 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 6.3: Quad Faces in the outlet Zone (p. 112)).

**Figure 6.3: Quad Faces in the outlet Zone**

- c. Retain the selection of **Replace** and click **Apply**.

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

- d. Close the **Triangulate Zones** dialog box.

### **Extra**

You can also create a non-conformal interface to allow quad faces during the hexcore mesh initialization. In this case, the triangular mesh on all quad faces available in the domain will be created automatically. 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** dialog box.
- 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** dialog box.

4. Save the mesh file.

**File → Write → Mesh...**

## 6.6. 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

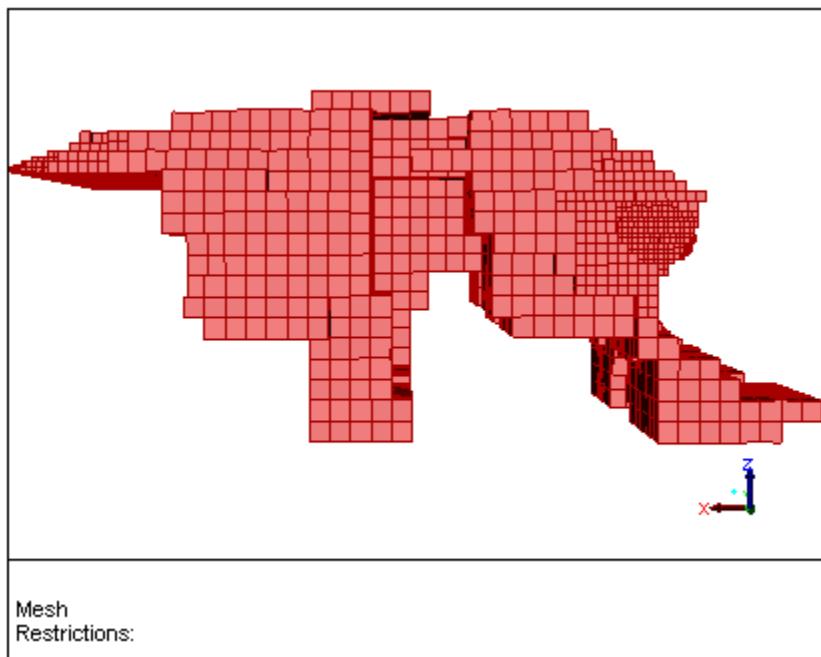
The maximum skewness reported at the end of the refinement process is not the final value. Some slivers can be removed.

- c. Display the parent face.

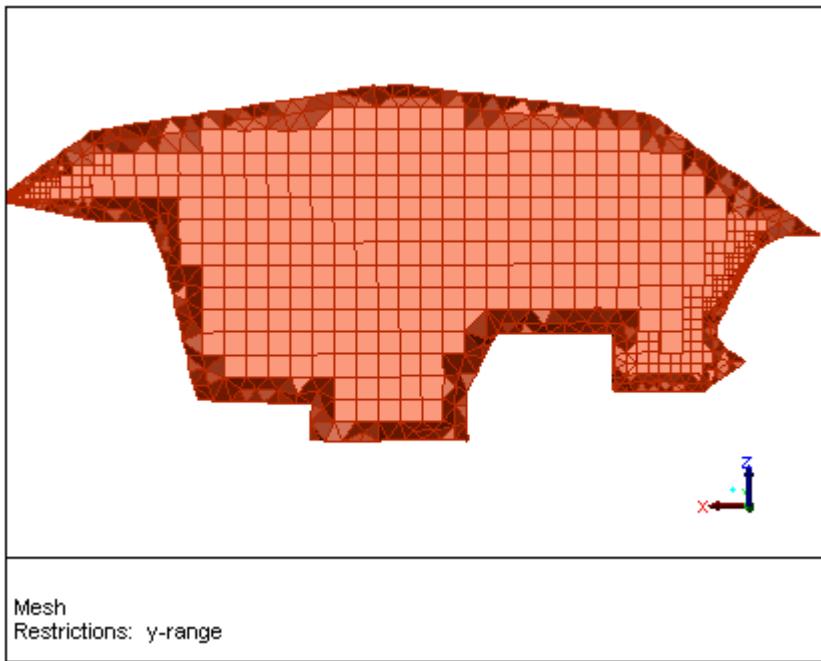
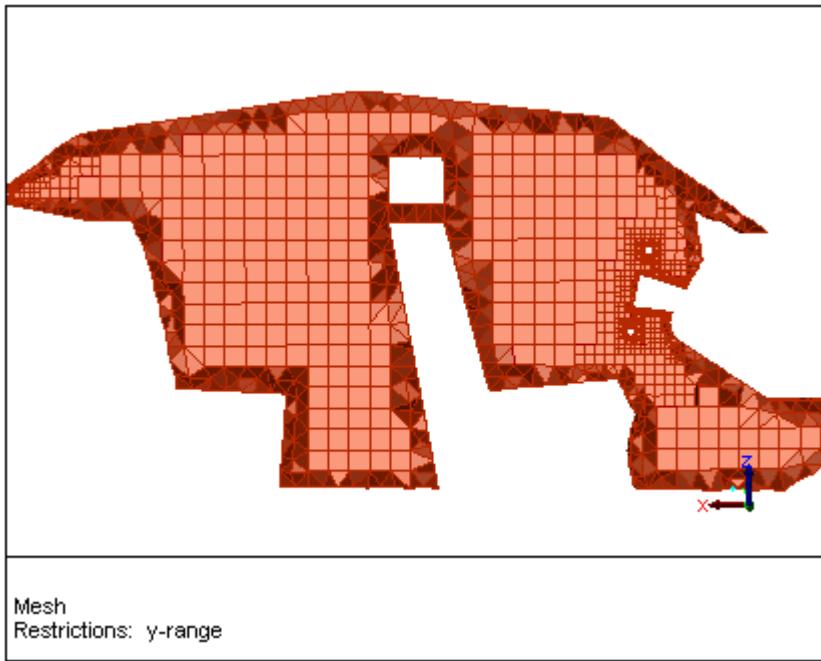
**Display → Grid...**

- i. Click the **Faces** tab. Deselect any previous selections and select **parent-face-#** in the **Face Zones** selection list.
- ii. Click **Display** ([Figure 6.4: Parent Face for the Hexcore Mesh \(p. 113\)](#)).

**Figure 6.4: Parent Face for the Hexcore Mesh**



- d. Display a slide of cells at  $y = 50$  and  $y = -525$  ([Figure 6.5: Slide of Cells at  \$y = 50\$  for the Hexcore Mesh With Buffer Layers = 1 \(p. 114\)](#) and [Figure 6.6: Slide of Cells at  \$y = -525\$  for the Hexcore Mesh With Buffer Layers = 1 \(p. 114\)](#)).

**Figure 6.5: Slide of Cells at  $y = 50$  for the Hexcore Mesh With Buffer Layers = 1****Figure 6.6: 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 around 179041. The exact number may vary slightly on different platforms.

- b. Close the **Report Mesh Size** dialog box.
  
- 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 0.850.

- b. Close the **Report Cell Limits** dialog box.

## 6.7. Examine the Effect of the Buffer Layers on the Hexcore Mesh

The number of **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...**

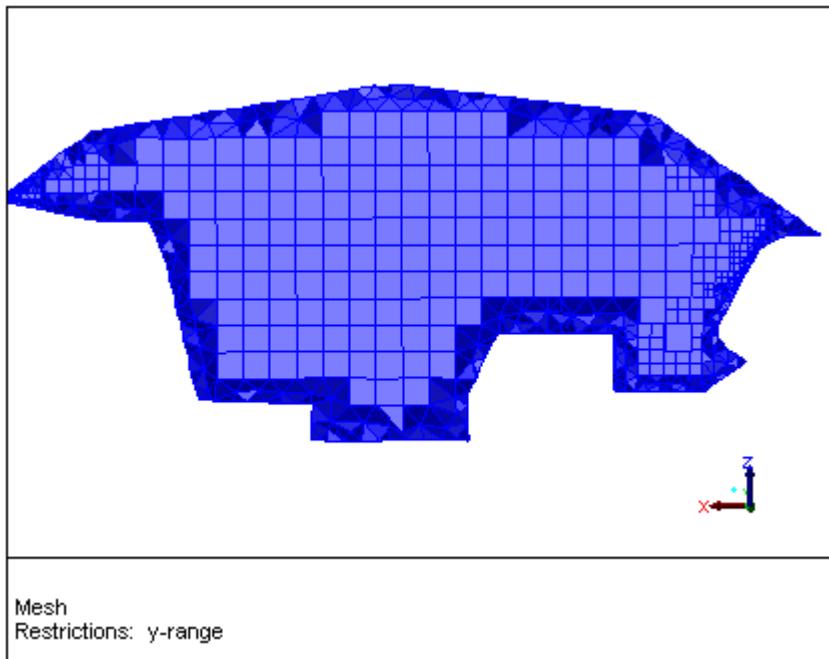
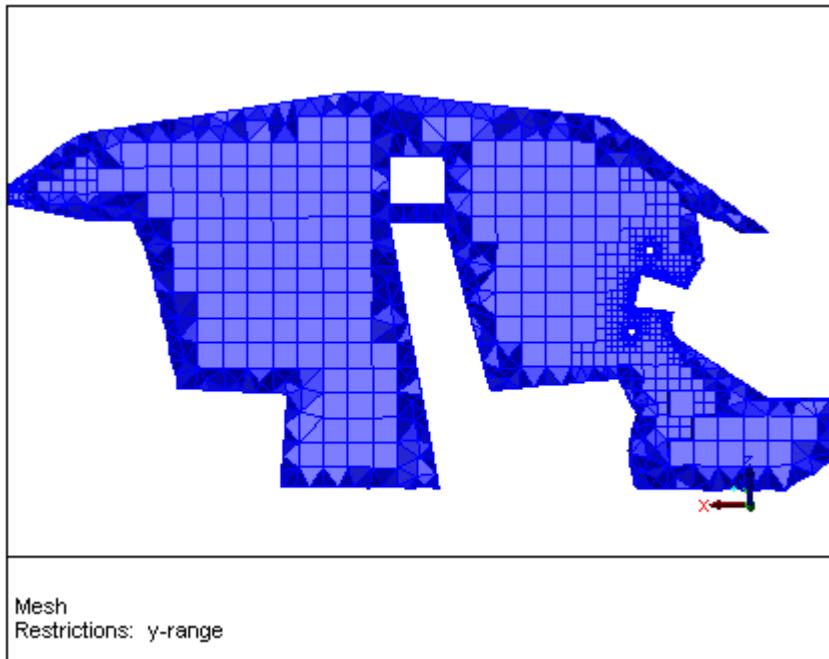
1. Click **Compute** to determine the value for **Max Cell Length** based on the current mesh.
  
2. Set **Buffer Layers** to 0 and click **Apply**.

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

3. Click **OK** in the **Warning** dialog box.
  
4. Click **Create**.

The **Warning** dialog box will appear again.

5. Click **OK** in the **Warning** dialog box.
  
3. Display the slide of cells at  $y = 50$  and  $y = -525$  ([Figure 6.7: Slide of Cells at  \$y = 50\$  for the Hexcore Mesh With Buffer Layers = 0 \(p. 116\)](#) and [Figure 6.8: Slide of Cells at  \$y = -525\$  for the Hexcore Mesh With Buffer Layers = 0 \(p. 116\)](#)).

**Figure 6.7: Slide of Cells at  $y = 50$  for the Hexcore Mesh With Buffer Layers = 0****Figure 6.8: Slide of Cells at  $y = -525$  for the Hexcore Mesh With Buffer Layers = 0**

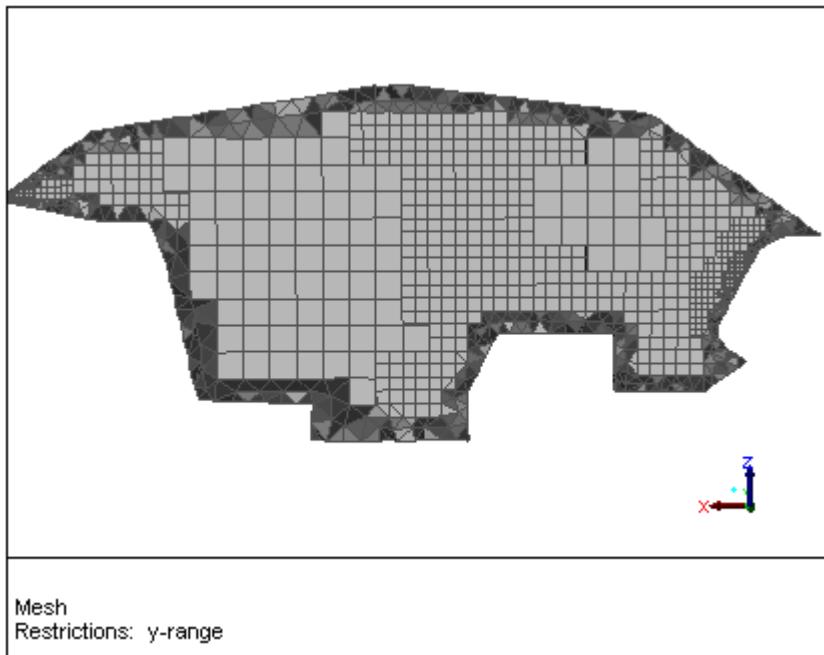
4. Check the maximum skewness and the number of cells.

For buffer layers = 0, the number of cells is around 150297 and the maximum skewness is around 0.865. The exact values may vary on different platforms.

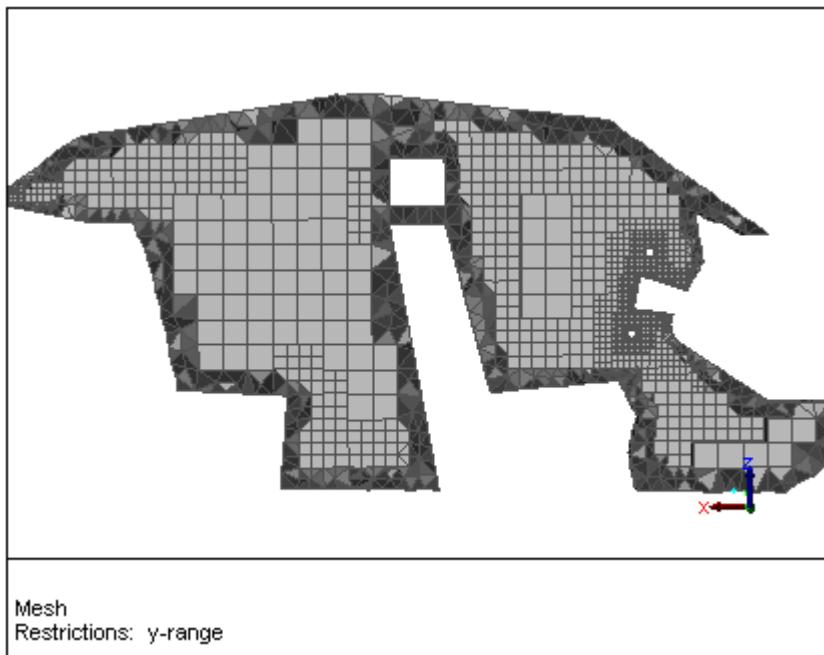
5. Clear the mesh.
6. Set **Buffer Layers** to 2 and generate the hexcore mesh.

7. Display the slide of cells at  $y = 50$  and  $y = -525$  ([Figure 6.9: Slide of Cells at  \$y = 50\$  for the Hexcore Mesh With Buffer Layers = 2 \(p. 117\)](#) and [Figure 6.10: Slide of Cells at  \$y = -525\$  for the Hexcore Mesh With Buffer Layers = 2 \(p. 117\)](#)).

**Figure 6.9: Slide of Cells at  $y = 50$  for the Hexcore Mesh With Buffer Layers = 2**



**Figure 6.10: 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 around 223967 and the maximum skewness is around 0.849. The exact values may vary on different platforms. Hence, the number of buffer layers has a strong impact on the number of cells.

## 6.8. 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** dialog box.
- 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**.

---

### Tip

Larger normals are easier to see in the grid display.

---

- e. Click **Display**.

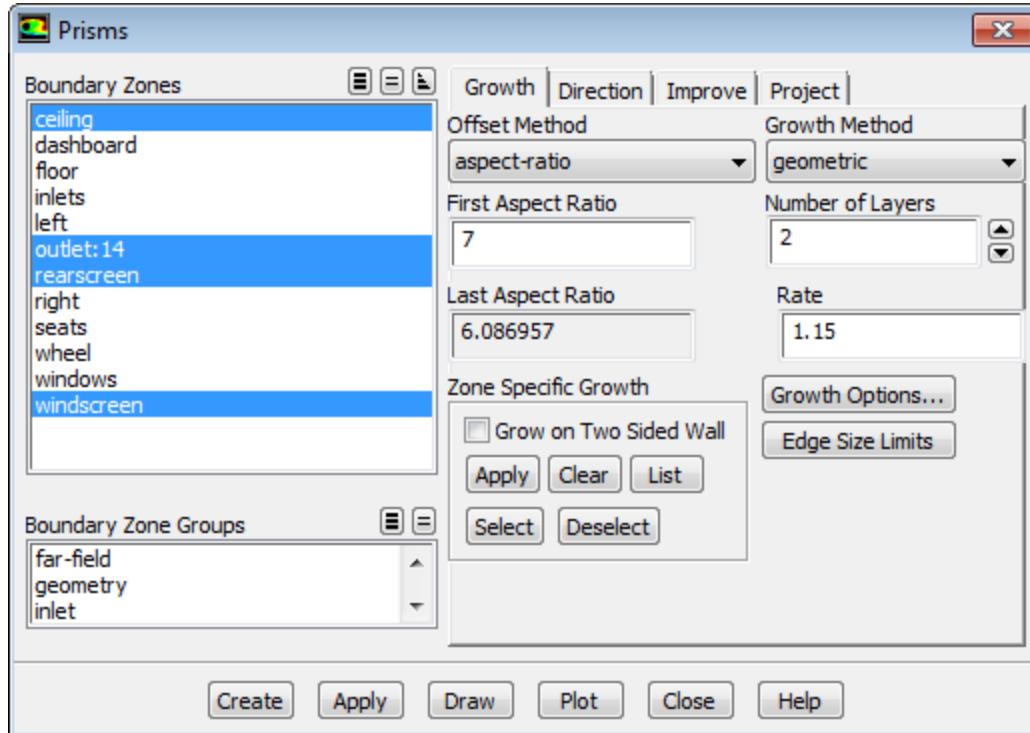
If you zoom in, the normals on the selected zones point inward.

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** dialog box.



- ii. Select **ceiling**, **outlet:#**, **reascreen**, 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.

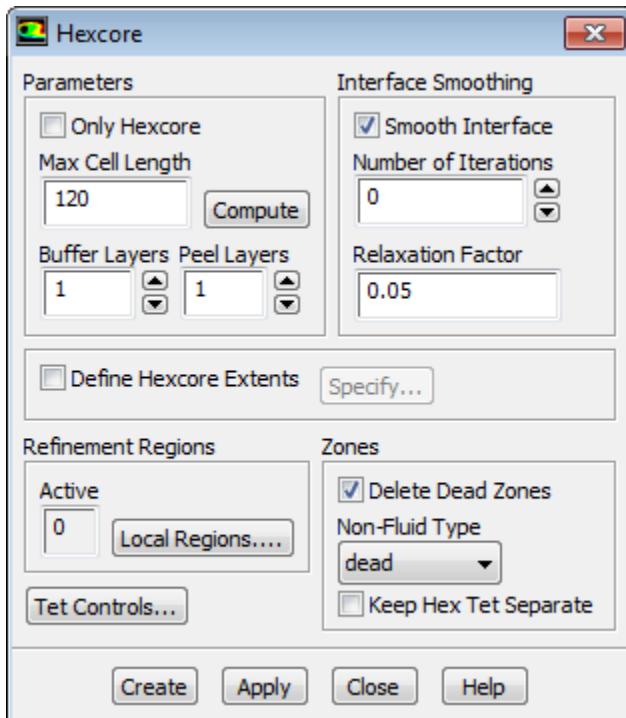
### Warning

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

- vii. Close the **Prisms** dialog box.
- viii. Enable **Prisms** in the **Boundary Layer Mesh** group box in the **Auto Mesh** dialog box.
- b. Display all the surfaces except **right** using the **Display Grid** dialog box.  
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** dialog box and click the **Set...** button to open the **Hexcore** dialog box.
  - 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 60.



- iii. Enter 120 for **Max Cell Length** and click **Apply** in the **Hexcore** dialog box.

This is the maximum length for the hexahedral cells in the volume mesh.

- iv. Set **Buffer Layers** to 1.
- v. Enable **Delete Dead Zones** and **Keep Hex Tet Separate** in the **Zones** group box.

---

#### Note

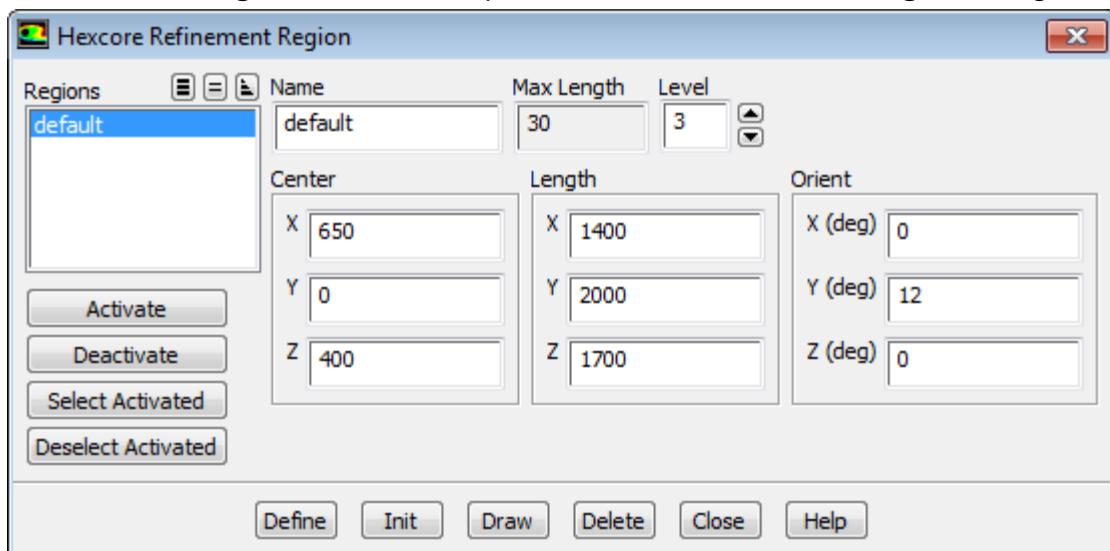
The **Keep Hex Tet Separate** option is used here primarily for better visualization of the mesh.

---

- vi. Click the **Tet Controls...** button to open the **Tet** dialog box.
  - A. Retain the default settings in the **Tet** dialog box and click **Apply**.
  - B. Close the **Tet** dialog box.
- vii. Define and visualize the refinement region.

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

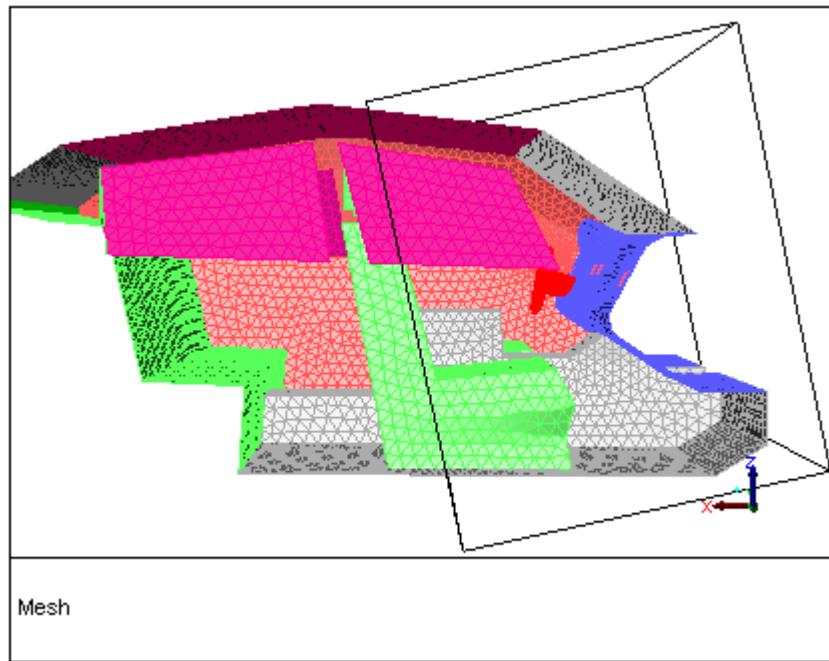
- A. Click the **Local Regions...** button to open the **Hexcore Refinement Region** dialog box.



- B. Retain default for **Name**.  
C. Set **Level** to 3.  
D. Enter the following parameters:

	<b>Cen- ter</b>	<b>Length</b>	<b>Ori- ent</b>
X	650	1400	0
Y	0	2000	12
Z	400	1700	0

- E. Click **Define**.  
F. Click **Draw** to visualize the refinement region (Figure 6.11: Refinement Region (p. 122)).

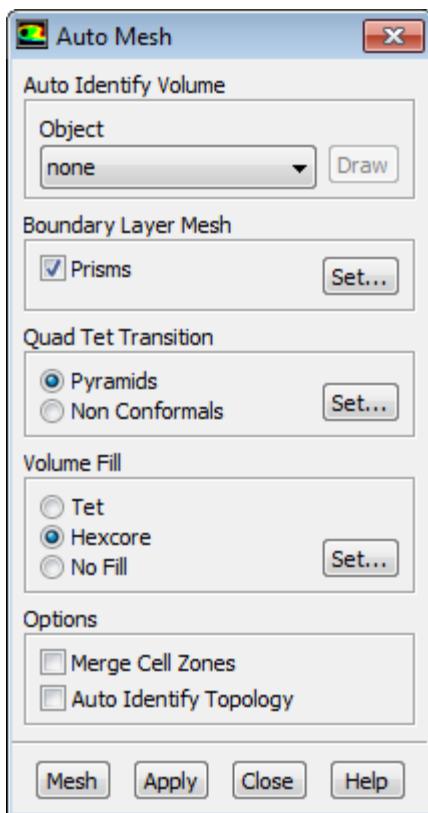
**Figure 6.11: Refinement Region**

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

H. Close the **Hexcore Refinement Region** dialog box.

The **Active** field in the **Hexcore** dialog box will show that there is one active hexcore refinement region.

d. Click **Apply** and close the **Hexcore** dialog box.



4. Click **Mesh** in the **Auto Mesh** dialog box to generate the mesh.
5. Close the **Auto Mesh** dialog box.
6. Examine the mesh by zone type.

**Mesh → Manage...**

7. Select all the zones in the **Cell Zones** selection list and click **List**.

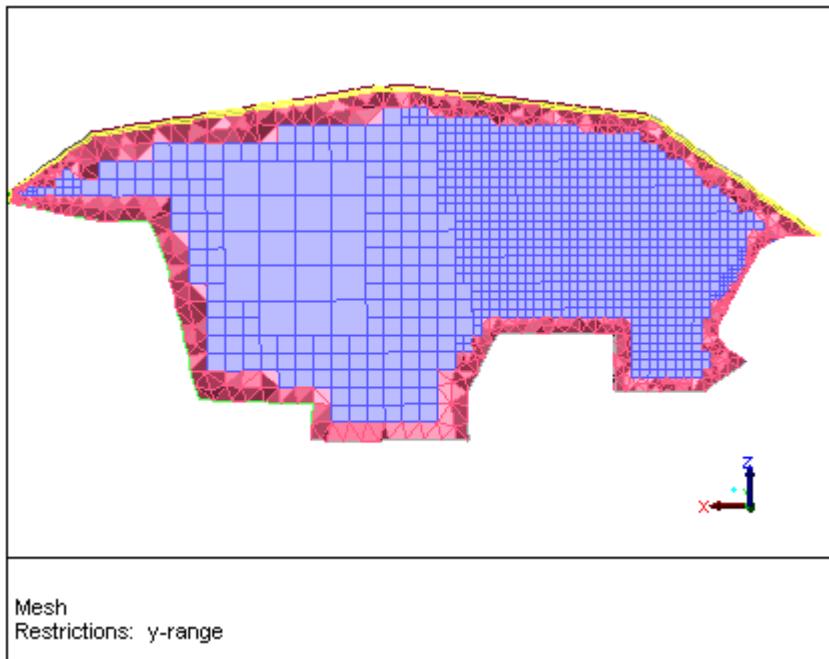
You can also draw the cell zones to identify the respective zones. There are around 11728 prism cells (**prism-cells-#**), 190486 tetrahedral cells (**fluid-#**), and 60035 hexahedral cells (**fluid-#:#**).

8. Check the cell count and the maximum skewness.

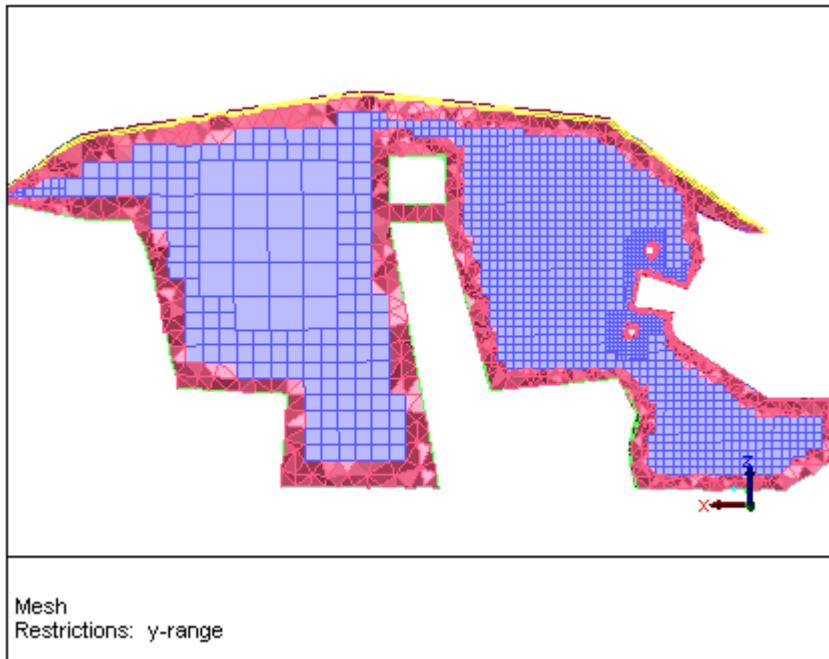
The number of cells is around 262249 and the maximum skewness is around 0.907. The exact number may vary on different platforms.

9. Display the slide of cells at  $y = 50$  and  $y = -525$  ([Figure 6.12: Slide of Cells at  \$y = 50\$  \(p. 124\)](#) and [Figure 6.13: Slide of Cells at  \$y = -525\$  \(p. 124\)](#)).

**Figure 6.12: Slide of Cells at  $y = 50$**



**Figure 6.13: Slide of Cells at  $y = -525$**



10. Check the edge length of the largest hex cell.

You should get a value close to 120.

11. Save the mesh file.

**File → Write → Mesh...**

12. Exit ANSYS FLUENT.

**File → Exit**

## 6.9. 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.



---

## Chapter 7: Generating the Hexcore Mesh to Domain Boundaries

---

When generating the hexcore mesh for external flow domains, it may not be necessary to have a tetrahedral mesh at the domain boundaries. For such cases, you can generate the hexcore mesh to the domain boundaries, thereby also reducing the cell count. This tutorial demonstrates the generation of the hexcore mesh to the domain boundaries for a sedan car.

This tutorial demonstrates how to do the following:

- Read the mesh file and display the boundary mesh.
- Set parameters for generating the hexcore mesh manually.
- Set parameters for creating prism layers and the hexcore mesh using automatic meshing.
- Set parameters for creating prism layers and the hexcore mesh using TUI commands.
- Check and save the volume mesh.

### 7.1. Prerequisites

This tutorial assumes that you have little experience with the meshing mode in ANSYS FLUENT, but that you are familiar with the graphical user interface.

### 7.2. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

To prepare for running this tutorial:

1. Download the tutorial input file (`hexcore-boundaries.zip`) for the tutorial.
2. Unzip `hexcore-boundaries.zip`.

The files `sedan.msh.gz`, `sedan-nonaligned.msh.gz`, and `sedan_hexcore_to_boundaries.jou` can be found in the `hexcore-boundaries` folder created on unzipping the file.

3. Start ANSYS FLUENT in meshing mode. For detailed steps, refer to [Starting ANSYS FLUENT in Meshing Mode \(p. 4\)](#).

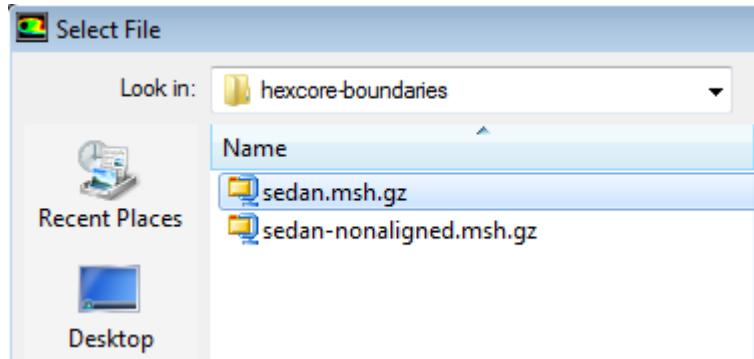
### 7.3. Manually Generate the Hexcore Mesh to the Boundaries

This section demonstrates the setting of parameters for generating the hexcore mesh. The outer boundaries will be deleted prior to the hexcore mesh generation. You will use the manual mesh generation procedure.

## Read and Display the Boundary Mesh

1. Read the mesh file.

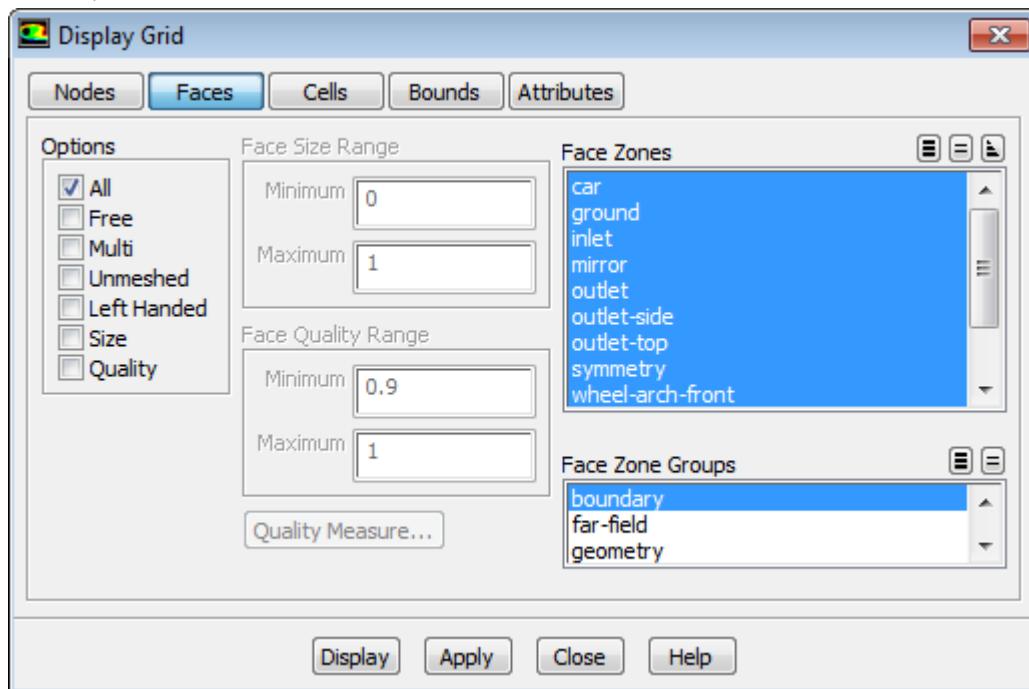
**File → Read → Boundary Mesh...**



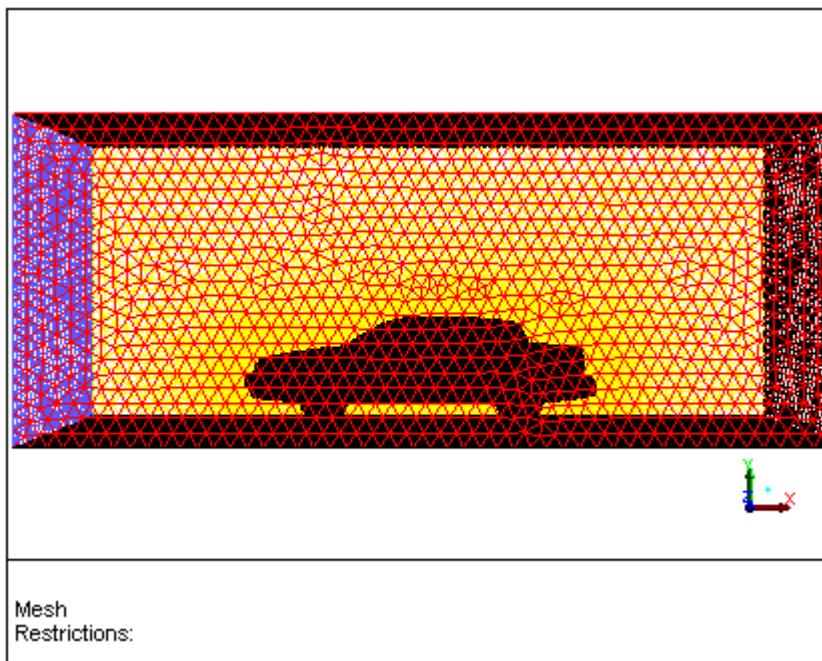
- a. Select **sedan.msh.gz** in the **Files** list.
- b. Click **OK**.

2. Display the boundary mesh (Figure 7.1: Boundary Mesh for the Sedan (p. 129)).

**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**.

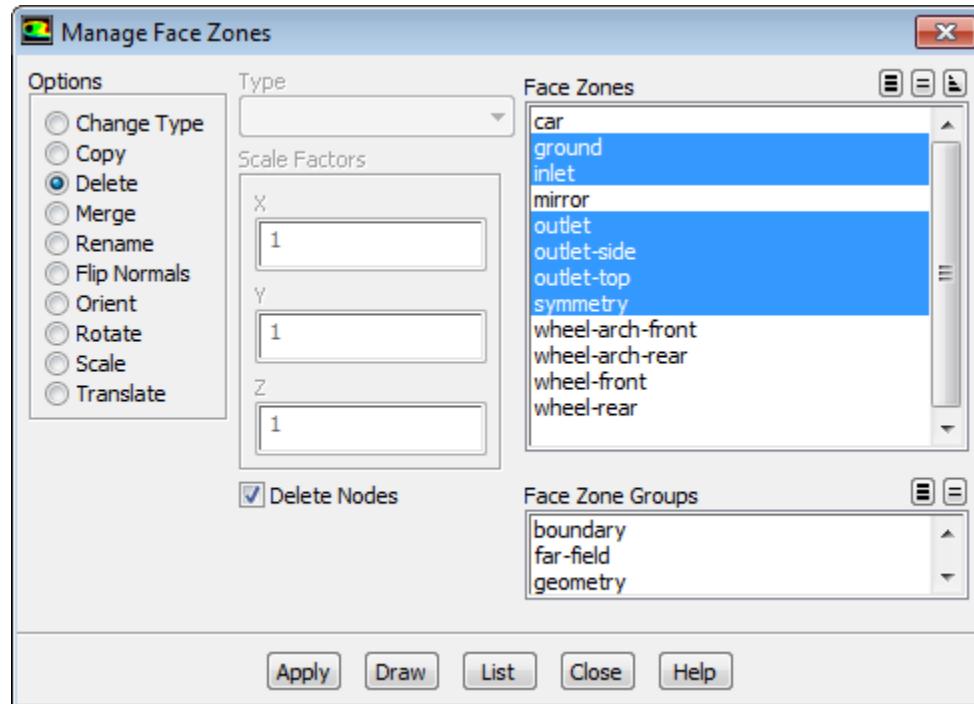
**Figure 7.1: Boundary Mesh for the Sedan**

The mesh contains the boundary mesh of the sedan and the outer box. You will initially generate the hexcore mesh without using the outer box boundaries to define the domain extents.

- Close the **Display Grid** dialog box.

## Delete the Outer Box Boundaries

**Boundary → Manage...**



- Select **ground**, **inlet**, **outlet**, **outlet-side**, **outlet-top**, and **symmetry** in the **Face Zones** selection list.

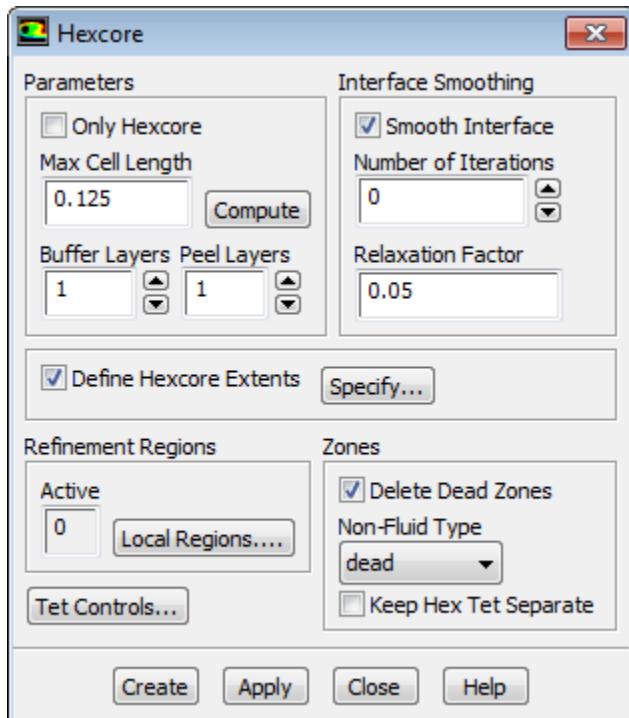
2. Select **Delete** in the **Options** list.
3. Retain the **Delete Nodes** option.
4. Click **Apply**.

A **Question** dialog box will appear asking you to confirm if you want to delete the selected zones.

5. Click **Yes** in the **Question** dialog box.
6. Select all the zones in the **Face Zones** selection list
7. Click **Draw**.
8. Close the **Manage Face Zones** dialog box.

## Generate the Hexcore Mesh

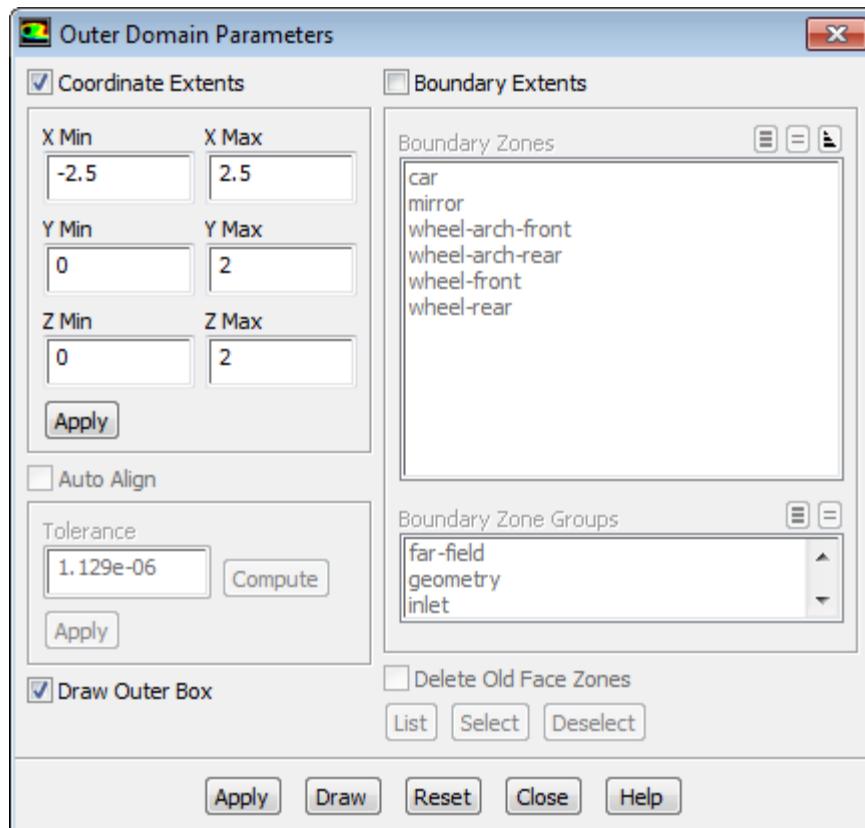
**Mesh → Hexcore...**



1. Enter 0.125 for **Max Cell Length**.
2. Retain the setting of both **Buffer Layers** and **Peel Layers** to 1.
3. Enable **Define Hexcore Extents**.

The **Specify...** button will be enabled.

4. Click the **Specify...** button to open the **Outer Domain Parameters** dialog box.



- a. Enable **Coordinate Extents**.
  - b. Enter the domain extents as follows:
 

**X Min = -2 . 5, Y Min = 0, Z Min = 0**

**X Max = 2 . 5, Y Max = 2, Z Max = 2**
  - c. Click **Apply** in the **Coordinate Extents** group box.
  - d. Enable **Draw Outer Box** and click **Draw**.
  - e. Click **Apply**.
  - f. Close the **Outer Domain Parameters** dialog box.
5. Enable **Delete Dead Zones** in the **Zones** group box.

#### Note

Enabling **Delete Dead Zones** ensures that the volume mesh is not generated inside the sedan body and the wheels, thereby making the meshing process faster.

6. Click **Create**.
7. Close the **Hexcore** dialog box.

## Examine the Mesh

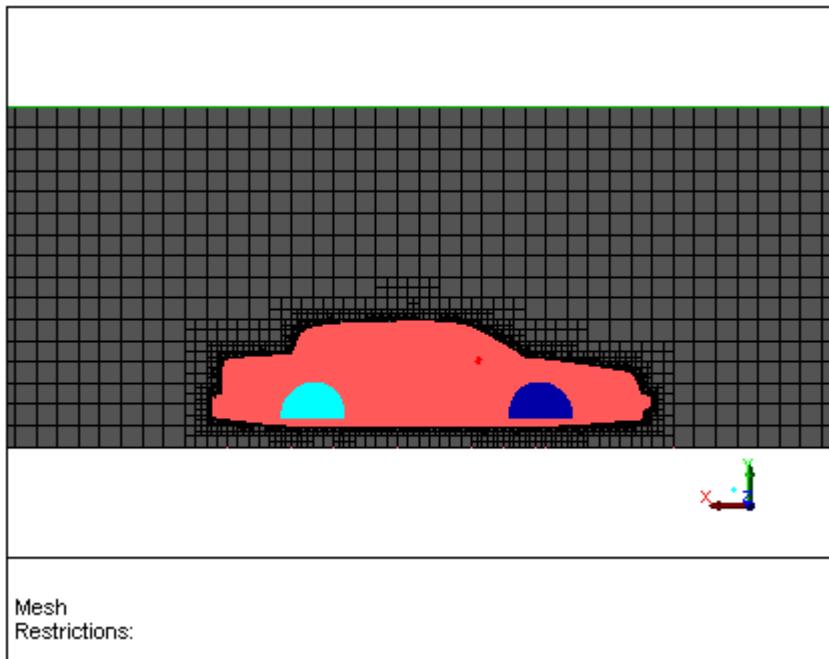
**Display → Grid...**

1. Deselect the previous selections and select **boundary** in the **Face Zone Groups** selection list.
2. Click the **Attributes** tab.
  - a. Enable **Filled** and **Lights** in the **Options** group box.
  - b. Click the **Colors...** button to open the **Grid Colors** dialog box.
    - i. Select **Color by ID** in the **Options** list.
    - ii. Close the **Grid Colors** dialog box.
3. Display the **back** view.

**Display → Views...**

- a. Select **back** from the **Views** list.
  - b. Click **Apply**.
  - c. Close the **Views** dialog box.
4. Click **Display** in the **Display Grid** dialog box.

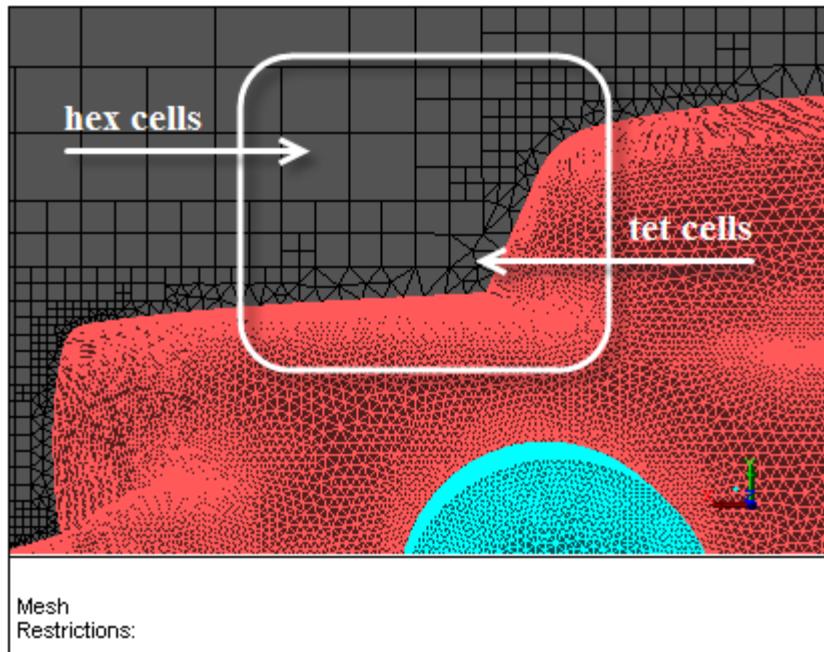
**Figure 7.2: Hexcore Mesh Up to Domain Boundaries**



In [Figure 7.2: Hexcore Mesh Up to Domain Boundaries \(p. 132\)](#), you can see that the hex cells are generated up to the extents of the domain defined in the **Hexcore** dialog box. For easy recognition, the newly created zones are named **wall-x-max**, **wall-x-min**, **wall-y-max**, etc.

5. Close the **Display Grid** dialog box.
6. Zoom in to the region shown in [Figure 7.3: Hexcore Mesh Up to Domain Boundaries—Zoomed View \(p. 133\)](#).

**Figure 7.3: Hexcore Mesh Up to Domain Boundaries—Zoomed View**



In [Figure 7.3: Hexcore Mesh Up to Domain Boundaries—Zoomed View \(p. 133\)](#), you can see that the mesh on the boundaries is made up of hex cells, except near the sedan.

7. Check the mesh.

**Mesh → Check**

8. Save the mesh file.

**File → Write → Mesh...**

## 7.4. Automatically Generate the Hexcore Mesh to the Boundaries with Prism Layers

This section demonstrates the setting of parameters for creating prism layers on the sedan body and ground, and for generating the hexcore mesh. The outer boundaries must be retained for the prisms to attach onto them. You will use the automatic mesh generation procedure.

### Read and Display the Boundary Mesh

1. Read the mesh file (`sedan-nonaligned.msh`).

**File → Read → Boundary Mesh...**

The mesh file contains the boundary mesh of the sedan as well as the outer box.

---

### Note

In order to create the hexcore mesh up to selected boundaries, each boundary should be separated by an angle of 90 degrees. For example, the auto-align tool will not work if the tunnel wall and roof are in a single boundary zone. The outer box in this tutorial constitutes individual zones separated by 90 degrees.

If you have a boundary mesh with the wind tunnel and roof in a single boundary zone, you will need to separate the zones by an angle of 90 degrees or less using the **Separate Face Zones** dialog box.

**Boundary → Zone → Separate...**

---

2. Display the grid.

**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** and close the **Display Grid** dialog box.

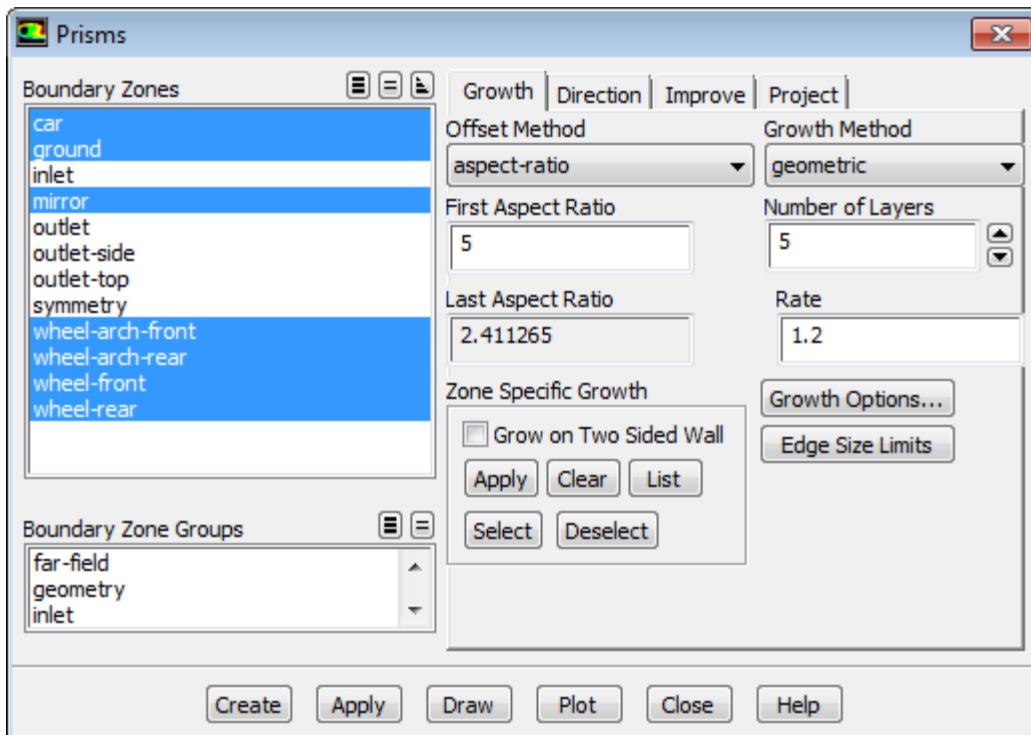
## Set the Meshing Parameters

**Mesh → Auto Mesh...**

1. Set the prism meshing parameters.

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

- a. Click the **Set...** button in the **Boundary Layer Mesh** group box to open the **Prisms** dialog box.

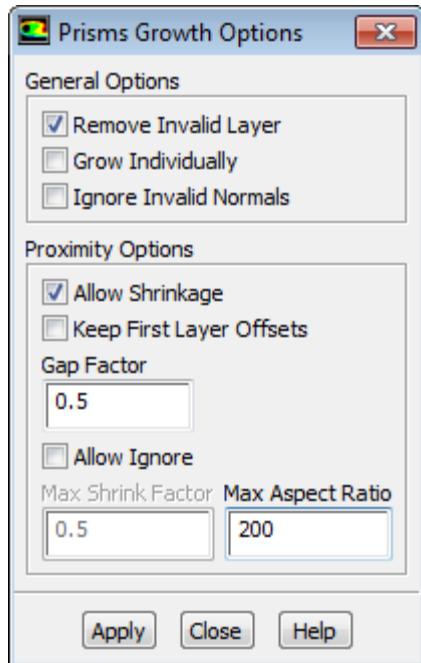


- b. Select **car**, **ground**, **mirror**, **wheel-arch-front**, **wheel-arch-rear**, **wheel-front**, and **wheel-rear** in the **Boundary Zones** selection list.
- c. Select **aspect-ratio** in the **Offset Method** drop-down list and enter 5 for **First Aspect Ratio**.
- d. Select **geometric** in the **Growth Method** drop-down list and enter **1 . 2** for **Rate**.
- e. Set **Number of Layers** to 5.
- f. In the **Zone Specific Growth** group box, click **Apply**.

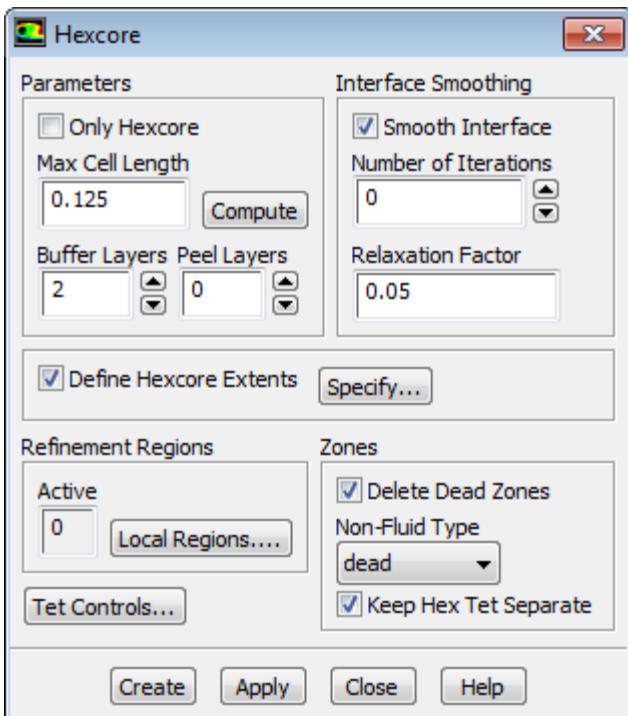
### Warning

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

- g. Click the **Growth Options...** button to open the **Prisms Growth Options** dialog box.



- i. Retain the **Allow Shrinkage** option in the **Proximity Options** group box.
  - ii. Enter **200** for **Max Aspect Ratio**.
  - iii. Click **Apply**
  - iv. Close the **Prisms Growth Options** dialog box.
  - h. Click **Apply** in the **Prisms** dialog box,
  - i. Close the **Prisms** dialog box.
  - j. Enable **Prisms** in the **Boundary Layer Mesh** group box in the **Auto Mesh** dialog box.
2. Set the hexcore meshing parameters.
- a. Select **Hexcore** in the **Volume Fill** list in the **Auto Mesh** dialog box.
  - b. Click the **Set...** button to open the **Hexcore** dialog box.



c. Enter 0.125 for **Max Cell Length**.

d. Set **Buffer Layers** to 2.

Setting the number of buffer layers to 2 ensures smoother size variation.

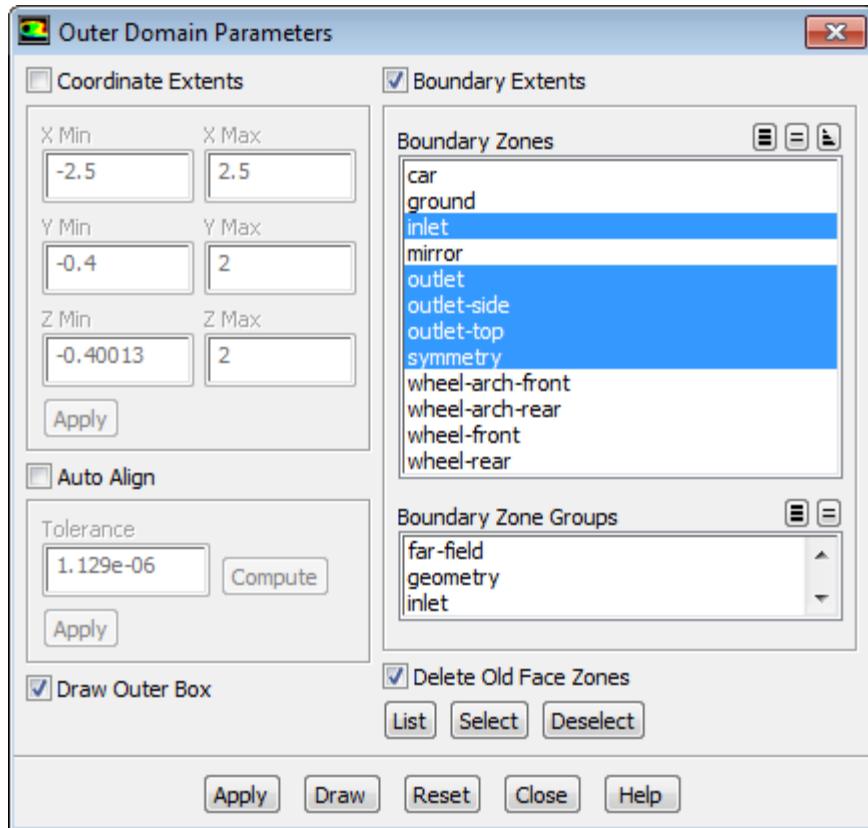
e. Set **Peel Layers** to 0 and click **Apply**.

The peel layer controls the gap between the hexahedra core and the geometry. The lower the peel layer specified, the fewer the tetrahedral cells generated.

f. Enable **Define Hexcore Extents**.

The **Specify...** button will be enabled.

g. Click the **Specify...** button to open the **Outer Domain Parameters** dialog box.



i. Enable **Boundary Extents**.

The **Boundary Extents** option is recommended over the **Coordinate Extents** option when the boundaries are explicitly defined.

ii. Select **inlet**, **outlet**, **outlet-side**, **outlet-top**, and **symmetry** in the **Boundary Zones** selection list.

---

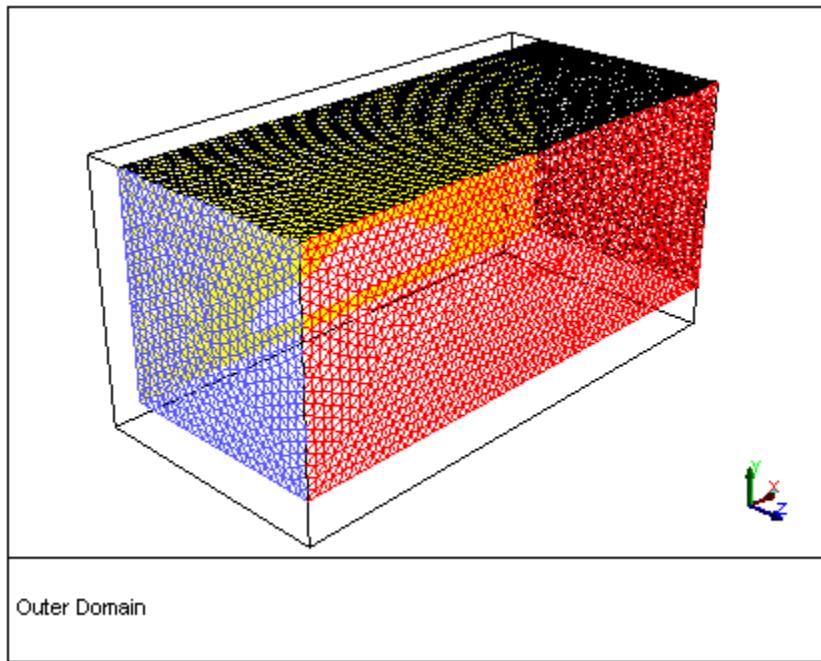
**Note**

The **ground** is not selected as prism layers will be grown from the ground during the automatic mesh generation process.

---

iii. Click **Apply**.

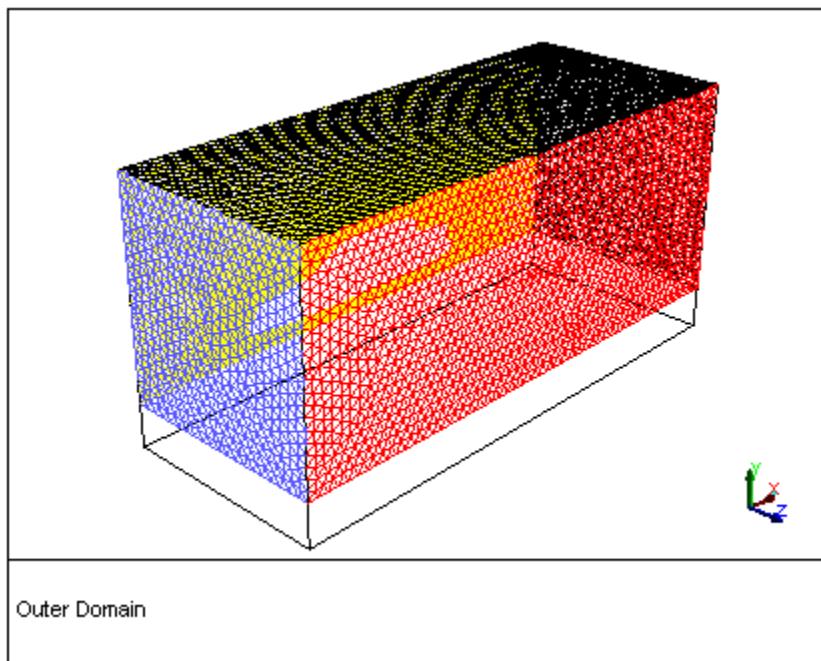
iv. Enable **Draw Outer Box** and click **Draw** (see [Figure 7.4: Outer Box for the Hexcore Mesh \(p. 139\)](#)).

**Figure 7.4: Outer Box for the Hexcore Mesh**

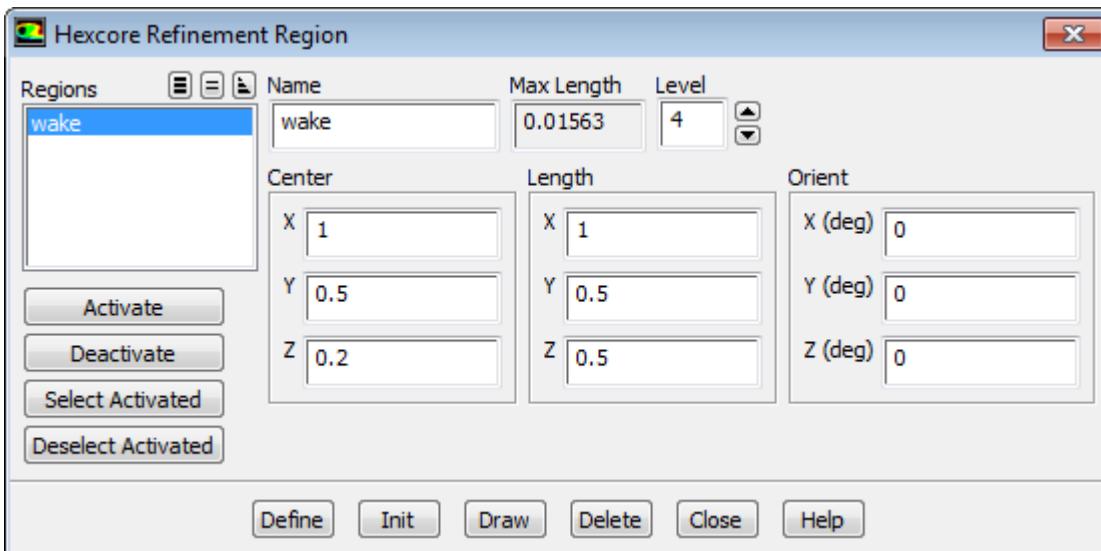
In [Figure 7.4: Outer Box for the Hexcore Mesh \(p. 139\)](#), the outer box snaps to all the selected boundaries, except the symmetry. This indicates that the symmetry zone is misaligned with the axis, and the hexcore mesh cannot be grown to this boundary. You need to align the symmetry zone to be able to grow the hexcore mesh to the symmetry.

- v. Select only **symmetry** in the **Boundary Zones** selection list and enable **Auto Align**.
  - vi. Click **Compute** in the **Auto Align** group box.
  - vii. Retain the computed tolerance and click **Apply** in the **Auto Align** group box.
- An **Information** dialog box will appear, informing you that the auto align operation may deform the geometry permanently.
- Click **OK** in the **Information** dialog box.
- viii. Deselect previous selections in the **Boundary Zones** list and click **Select** to reselect the zones to which the hexcore mesh is to be generated.
  - ix. Click **Apply** in the **Outer Domain Parameters** dialog box.
  - x. Make sure **Draw Outer Box** is enabled, and then click **Draw** (see [Figure 7.5: Outer Box for the Hexcore Mesh After Using Auto Align \(p. 140\)](#)).

In [Figure 7.5: Outer Box for the Hexcore Mesh After Using Auto Align \(p. 140\)](#), the outer box snaps to the selected boundaries.

**Figure 7.5: Outer Box for the Hexcore Mesh After Using Auto Align**

- xi. Enable **Delete Old Face Zones**, and then click **Apply**.  
The original outer box zones will be replaced by those created during the hexcore meshing process.
- xii. Close the **Outer Domain Parameters** dialog box.
- h. Click the **Tet Controls...** button to open the **Tet** dialog box.
  - i. Retain the default settings in the **Initialization** tab.
  - ii. Click the **Refinement** tab and retain the selection of **geometric** in the **Cell Size Function** drop-down list.
  - iii. Enter **1 . 3** for **Growth Rate** and click **Apply**.
  - iv. Close the **Tet** dialog box.
- i. Click the **Local Regions...** button to open the **Hexcore Refinement Region** dialog box which will be used to define a local refinement region.



- i. Enter **wake** for **Name**.

- ii. Set **Level** to 4.

The hexcore mesh is based on a Cartesian grid, hence, the maximum length inside the local region is defined as a factor of the **Max Length** defined. The maximum length in the region is equal to  $\frac{\text{MaxLength}}{2^{(\text{level}-1)}}$ . In this case, the **Max Length** is equal to

$$\frac{0.125}{2^{(4-1)}} = 0.01563.$$

- iii. Enter (1, 0.5, 0.2) for **Center** and (1, 0.5, 0.5) for **Length**, respectively.
- iv. Retain the default orientation.
- v. Click **Draw** and check the region extents and maximum length within the region.
- vi. Click **Define** to create the region.
- vii. Retain the selection of **wake** in the **Regions** selection list and click **Activate**.
- viii. Close the **Hexcore Refinement Region** dialog box.

The **Active** field in the **Hexcore** dialog box indicates that there is one active hexcore refinement region.

- j. Enable **Delete Dead Zones** in the **Zones** group box.
- k. Enable **Keep Hex Tet Separate**.

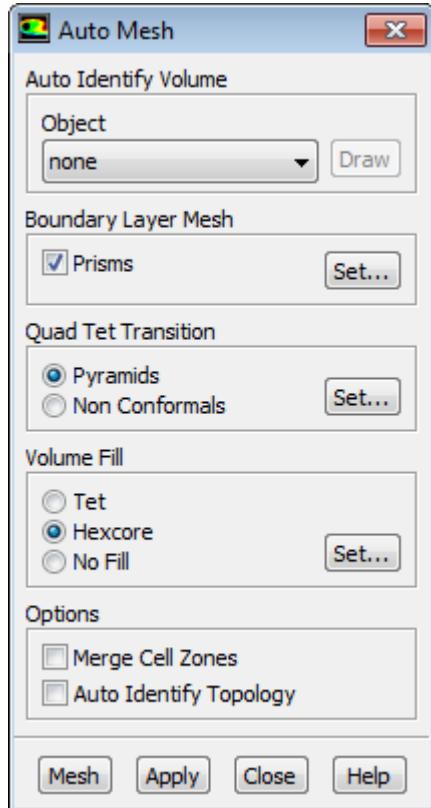
---

#### Note

The **Keep Hex Tet Separate** option is used here primarily for better visualization of the mesh.

- I. Click **Apply**.

- m. Close the **Hexcore** dialog box.



3. Click **Mesh**.
4. Close the **Auto Mesh** dialog box.

## Examine the Mesh

**Display → Grid...**

1. Deselect the previous selections and select **car**, **ground**, **inlet**, **inlet-#**, **mirror**, **outlet**, **outlet-#**, **outlet-side**, **outlet-side-#**, **outlet-top**, **symmetry**, **symmetry-#**, **wheel-arch-front**, **wheel-arch-rear**, **wheel-front**, and **wheel-rear** in the **Face Zones** selection list.
2. Click the **Attributes** tab.
  - a. Enable **Filled** and **Lights** in the **Options** group box.
  - b. Click the **Colors...** button to open the **Grid Colors** dialog box.
    - i. Select **Color by ID** in the **Options** list.
    - ii. Close the **Grid Colors** dialog box.
3. Display the **back** view.

**Display → Views...**

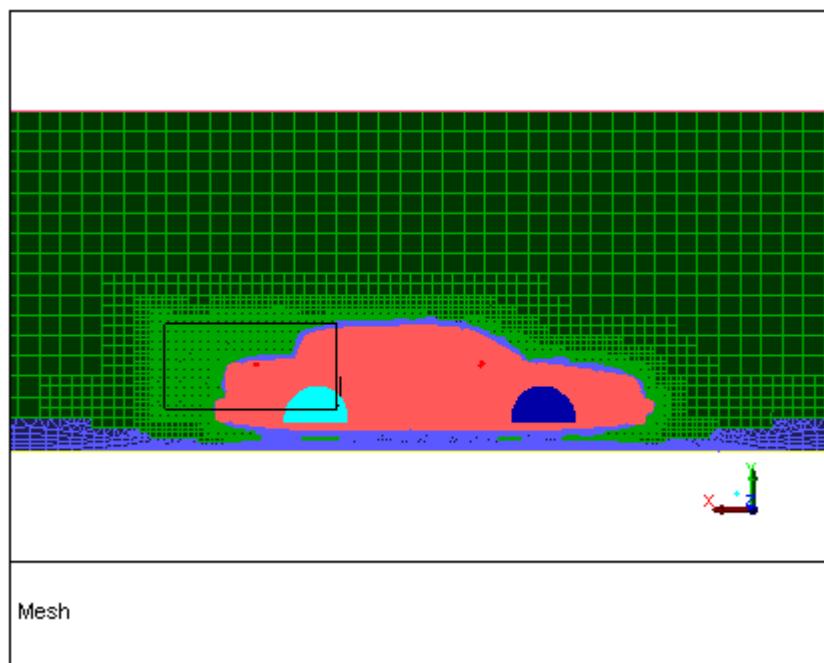
4. Click **Display** in the **Display Grid** dialog box.
5. Display the refinement region along with the mesh.

**Mesh → Hexcore...**

- a. Click the **Local Regions...** button in the **Hexcore** dialog box to open the **Hexcore Refinement Region** dialog box.
- b. Make sure **wake** is selected in the **Regions** selection list.
- c. Click **Draw**.
- d. Close the **Hexcore Refinement Region** dialog box.

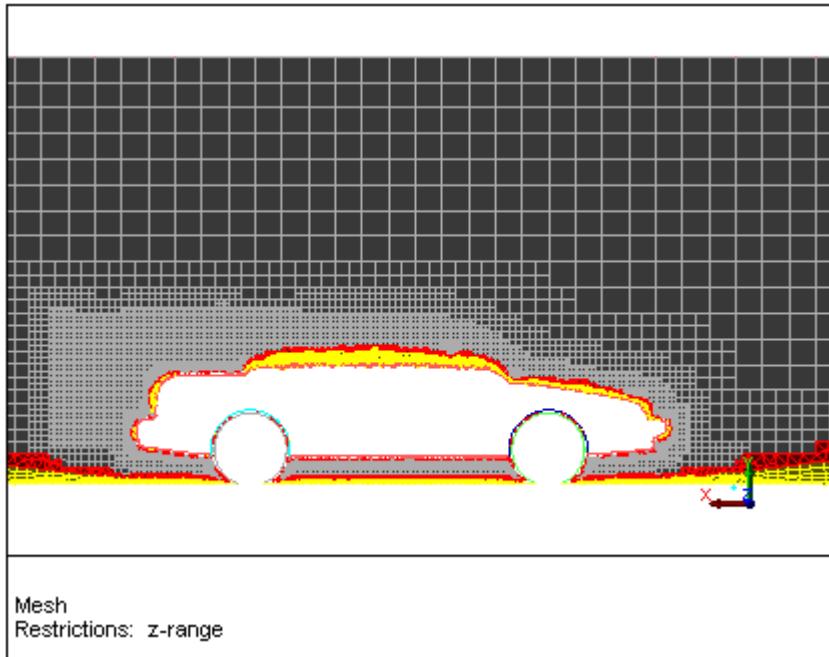
In [Figure 7.6: Hexcore Mesh to Selected Boundaries \(p. 143\)](#), you can see that the hex cells are generated up to the selected boundaries. You can also see the refinement region.

**Figure 7.6: Hexcore Mesh to Selected Boundaries**



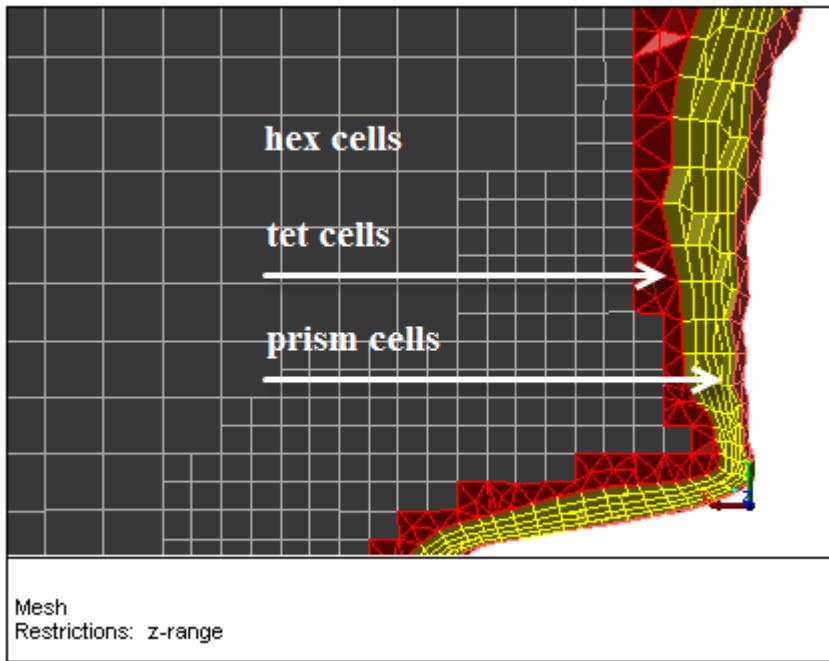
6. Click the **Bounds** tab in the **Display Grid** dialog box.
  - a. Enable **Limit by Z**.
  - b. Enter 0 . 37 for **Minimum** and **Maximum** in the **Z Range** group box.
7. Click the **Cells** tab in the **Display Grid** dialog box
  - a. Select all the zones in the **Cell Zones** selection list.
  - b. Enable **All** in the **Options** group box.
8. Click **Display**.

**Figure 7.7: Slide of Cells at z = 0.37**

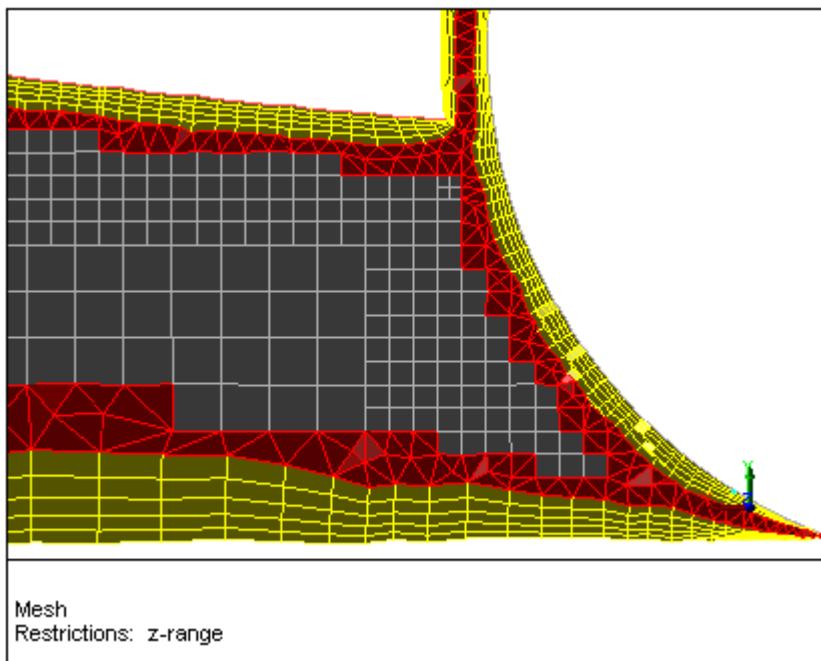


9. Zoom in to the rear of the sedan.

**Figure 7.8: Hexcore Mesh to Selected Boundaries—Zoomed View**



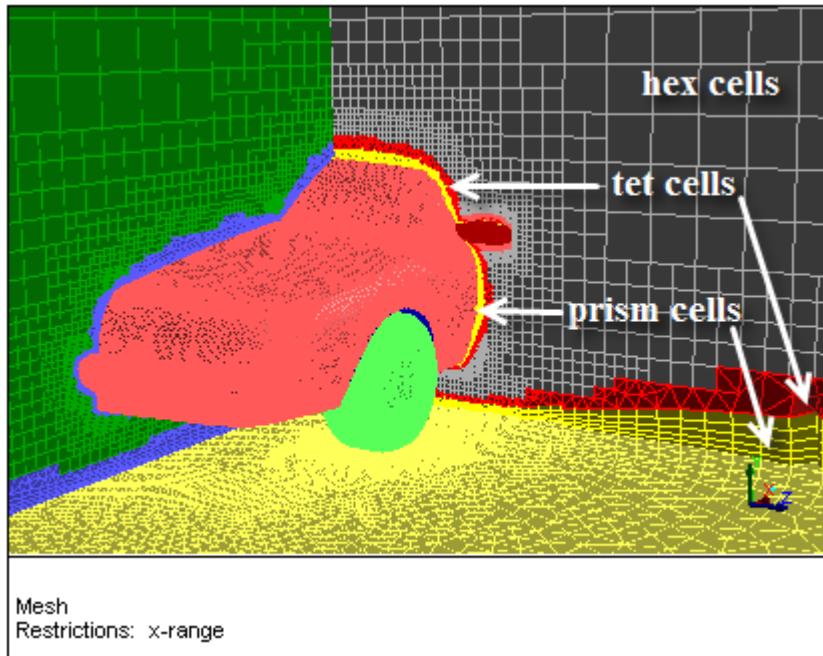
10. Zoom in to the region of the wheel/ground intersection.

**Figure 7.9: Hexcore Mesh to Selected Boundaries—Wheel/Ground Intersection**

11. Display the mesh overlaid with a section of cells at  $x = -0.37$  (Figure 7.10: Hexcore Mesh to Selected Boundaries—Cells at  $x = -0.37$  (p. 146)).
  - a. Click the **Bounds** tab, and then click **Reset**.
  - b. In the **Cells** tab of the **Display Grid** dialog box, deselect the fluid zones in the **Cell Zones** selection list.
  - c. Select **car**, **ground**, **mirror**, **symmetry**, **symmetry-#**, **wheel-arch-front**, **wheel-arch-rear**, **wheel-front**, and **wheel-rear** in the **Face Zones** selection list.
  - d. Click **Display**.
  - e. Enable the overlaying of graphics.  
**Display → Scene...**
    - i. Select all the zones in the **Names** selection list.
    - ii. Enable **Overlays** in the **Scene Composition** group box.
    - iii. Click **Apply**.
    - iv. Close the **Scene Description** dialog box.
  - f. Click the **Cells** tab in the **Display Grid** dialog box and make sure **All** is enabled in the **Options** group box.
  - g. Select the fluid zones in the **Cell Zones** selection list.

- h. On the **Bounds** tab, enable **Limit by X**, and then enter  $-0.37$  for both **Minimum** and **Maximum** in the **X Range** group box.
- i. Click **Display** and zoom in as shown in Figure 7.10: Hexcore Mesh to Selected Boundaries—Cells at  $x = -0.37$  (p. 146).

**Figure 7.10: Hexcore Mesh to Selected Boundaries—Cells at  $x = -0.37$**



12. Check the mesh.

**Mesh → Check**

13. Save the mesh file.

**File → Write → Mesh...**

14. Exit ANSYS FLUENT.

**File → Exit**

## 7.5. Generate the Hexcore Mesh to the Boundaries with Prism Layers Using TUI Commands

This section demonstrates the use of a journal file to generate the hexcore mesh to selected boundaries. You will set parameters for creating prism layers on the sedan body and ground, and for generating the hexcore mesh. The outer boundaries must be retained for the prisms to attach onto them.

1. View the contents of the journal file (`sedan_hexcore_to_boundaries.jou`) in a viewer.

All lines starting with a semicolon (;) indicate comments.

```
;Read the mesh file
/file/read-mesh sedan-nonaligned.msh.gz
```

```

;Display the grid
/display/boundary-grid * ,

;Specify the prism growth parameters
/mesh/prism/controls/zone-specific-growth/apply-growth car ground mirror wheel-arch-front wheel-arch-rear
/mesh/prism/controls/proximity/max-aspect-ratio 200

;Specify the hexcore meshing parameters
/mesh/hexcore/controls/maximum-cell-length 0.125
/mesh/hexcore/controls/buffer-layers 2
/mesh/hexcore/controls/peel-layers 0
;Hexcore to selected boundaries
/mesh/hexcore/controls/define-hexcore-extents? yes
/mesh/hexcore/controls/outer-domain-params/specify-boundaries? yes
;;Select the planar boundaries to which hexcore mesh is to be generated
/mesh/hexcore/controls/outer-domain-params/boundaries inlet outlet outlet-side outlet-top symmetry ,
;;Use auto-align to axis-align the boundaries to which hexcore mesh is to be generated
/mesh/hexcore/controls/outer-domain-params/auto-align? yes
/mesh/hexcore/controls/outer-domain-params/auto-align-tolerance ,
/mesh/hexcore/controls/outer-domain-params/auto-align-boundaries
;Make sure delete-old-face-zones is enabled
/mesh/hexcore/controls/outer-domain-params/delete-old-face-zones? yes
;Tet controls
/mesh/tet/controls/cell-size-function geometric 1.3
;Hexcore refinement region
/mesh/hexcore/local-regions/define "wake" 4 1 0.5 0.2 1 0.5 0.5 0 0 0
/mesh/hexcore/local-regions/activate "wake"
;Hexcore zones parameters
/mesh/hexcore/controls/delete-dead-zones? yes

;Generate the mesh
/mesh/auto-mesh "" yes pyramids hexcore yes no

;Examine the mesh
/display/boundary-grid car ground inlet* mirror outlet* outlet-side* outlet-top symmetry* wheel-arch-front
/display/set/filled-grid? yes
/display/set/lights/lights-on? yes
/display/set/colors/color-by-type? no
/display/set-grid/all-nodes? no
/display/set-grid/z-range 0.37 0.37
/display/set-grid/all-cells? yes
/display/all-grid fluid* ()
/display/view/restore-view back
/display/view/auto-scale

/mesh/check-mesh

```

2. Start a new ANSYS FLUENT session in meshing mode.
3. Read the journal file `sedan_hexcore_to_boundaries.jou`.

**File → Read → Journal...**

## 7.6. Summary

This tutorial demonstrated the generation of a hexcore mesh up to the domain boundaries for a sedan car. It also demonstrated the creation of prism layers and the use of local refinement regions in conjunction with the hexcore mesh generation.



---

## Chapter 8: Using the Boundary Wrapper

---

Geometries imported from various CAD packages often contain gaps and/or overlaps between surfaces. Repairing such geometries manually is a tedious and time-consuming process. The boundary wrapper can be used to repair such geometries automatically, thereby reducing the time required for preprocessing.

The wrapping procedure is based on the Cartesian grid (or overlay grid) approach. Initially, a coarse Cartesian grid is overlaid on the input geometry and the intersection between the Cartesian grid and the geometry is calculated. The intersecting cells are identified and a watertight faceted representation is created along the boundary of these cells. The nodes on the faceted representation are projected onto the input geometry resulting in a wrapper surface that closely resembles the input geometry.

This tutorial demonstrates how to do the following:

- Read and display the mesh.
- Perform pre-wrapping operations to close holes in the geometry.
- Initialize the wrapper.
- Check the region to be wrapped.
- Refine the Cartesian grid using the local size function.
- Wrap the surface and imprint necessary features.
- Check the deviation of the wrapper surface from the original geometry.
- Perform post-wrapping operations to improve wrapper surface quality.
- Create a tunnel encompassing the geometry and generate the volume mesh.
- Improve the volume mesh quality using automatic node movement.

The V-8 engine geometry used in this tutorial is courtesy of Platinum Pictures Multimedia Inc., [www.3dcafe.com](http://www.3dcafe.com), and Michael Barthels.

### 8.1. Prerequisites

This tutorial assumes that you have some experience with the meshing mode in ANSYS FLUENT, and that you are familiar with the graphical user interface.

### 8.2. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

To prepare for running this tutorial:

1. Download the tutorial input file (`wrapper.zip`) for the tutorial.

2. Unzip `wrapper.zip`.

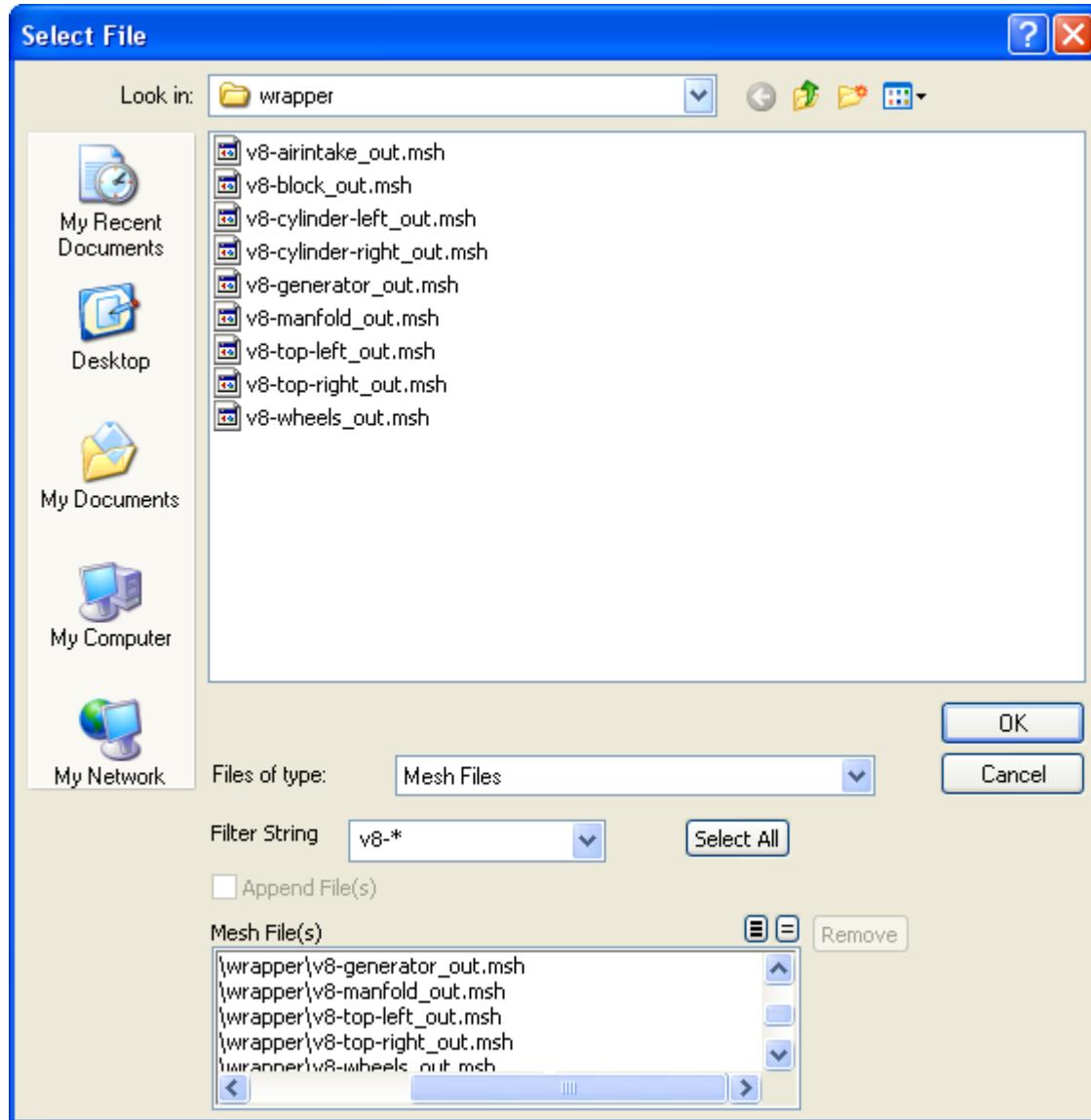
The files `v8-airintake_out.msh`, `v8-block_out.msh`, `v8-cylinder-left_out.msh`, `v8-cylinder-right_out.msh`, `v8-generator_out.msh`, `v8-manifold_out.msh`, `v8-top-left_out.msh`, `v8-top-right_out.msh`, and `v8-wheels_out.msh` can be found in the `v8` folder created on unzipping the file.

3. Start ANSYS FLUENT in meshing mode. For detailed steps, refer to [Starting ANSYS FLUENT in Meshing Mode \(p. 4\)](#).

### 8.3. Read and Display the Mesh

1. Read the mesh files using a filter to select multiple files.

**File → Read → Mesh...**



a. Enter the filter string `v8-* .msh` in the **Filter String** text box.

- b. Click **Select All** to select the mesh files.

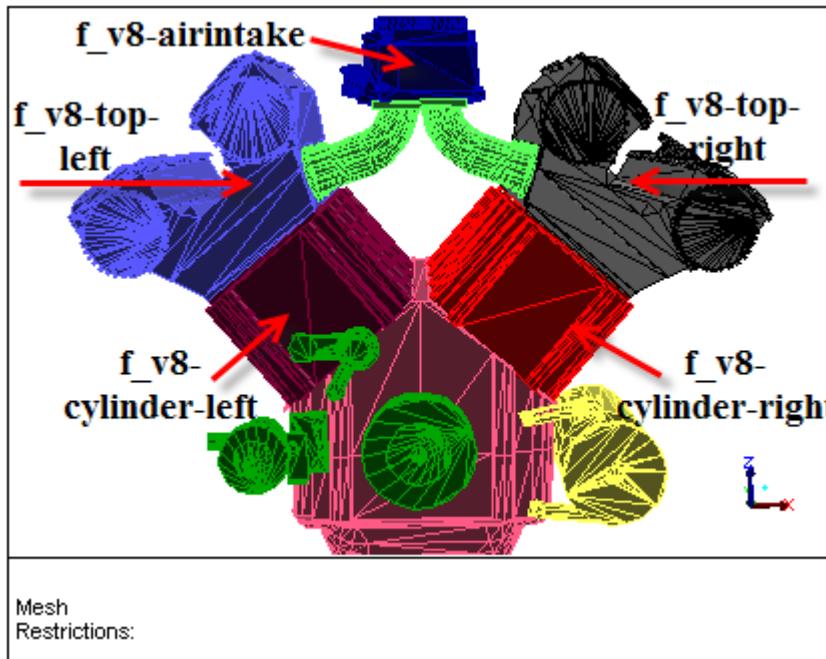
All the respective mesh files will now be added to the **Mesh File(s)** list.

- c. Click **OK**.

2. Display the mesh ([Figure 8.1: Grid Display \(p. 152\)](#)).

#### **Display → Grid...**

- a. Select all the zones in the **Face Zones** selection list.
- b. Click the **Attributes** tab and enable **Filled** and **Lights** in addition to the default, **Edges**.
- c. Click the **Colors...** button to open the **Grid Colors** dialog box.
  - i. Select **Color by ID** in the **Options** list.
  - ii. Close the **Grid Colors** dialog box.
- d. Click the **Rendering...** button to open the **Display Options** dialog box.
  - i. Select **All** in the **Animation Option** drop-down list.
  - ii. Enable **Double Buffering**.
  - iii. Select **Software Z-buffer** in the **Hidden Surface Removal Method** drop-down list.
  - iv. Click **Apply** and close the **Display Options** dialog box.
- e. Click **Display** in the **Display Grid** dialog box and rotate the geometry about the x-axis to obtain the view shown in [Figure 8.1: Grid Display \(p. 152\)](#).

**Figure 8.1: Grid Display****Tip**

You can save the view using the **Views** dialog box and restore the saved view whenever necessary.

## 8.4. Perform Pre-Wrapping Operations to Close Holes in the Geometry

The geometry contains some large holes (holes significantly larger than the mesh size that will be used) which preferably should be manually closed before proceeding with the wrapping operations. The meshing mode in ANSYS FLUENT provides a variety of options for closing arbitrary openings between zones, arbitrary openings in the same zone, coplanar openings, etc.

This section demonstrates the closing of the holes in the following zones:

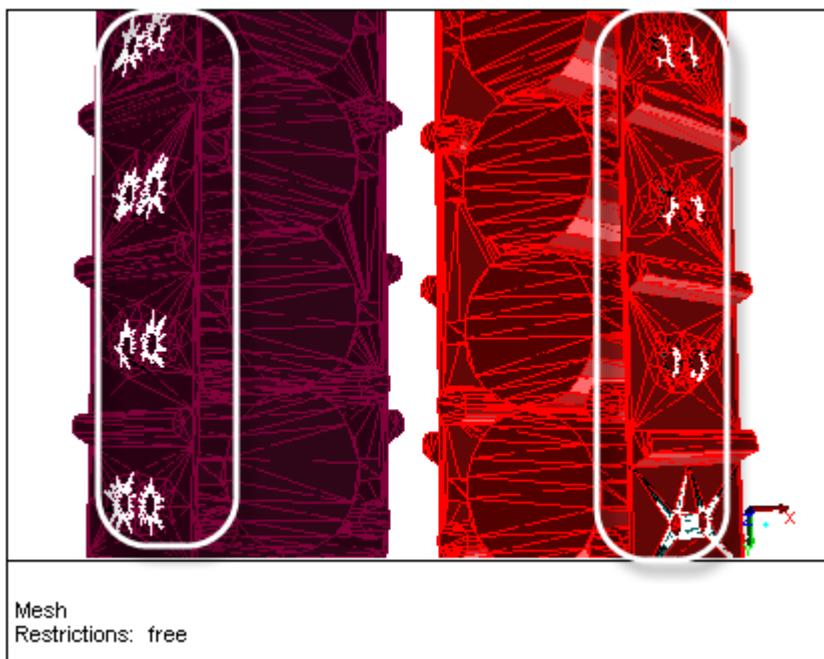
- **f\_v8-cylinder-left** and **f\_v8-cylinder-right** (see [Figure 8.2: Holes to be Closed in the f\\_v8-cylinder-left and f\\_v8-cylinder-right Zones \(p. 153\)](#))
- **f\_v8-airintake** (see [Figure 8.4: Holes to be Closed in the f\\_v8-airintake Zone \(p. 154\)](#))
- **f\_v8-top-right** (see [Figure 8.7: Coplanar Holes in the f\\_v8-top-right Zone \(p. 157\)](#))
- **f\_v8-top-left** (see [Figure 8.10: Coplanar Holes in the f\\_v8-top-left Zone \(p. 159\)](#))

1. Close the holes in the **f\_v8-cylinder-left** and **f\_v8-cylinder-right** zones.

- a. Select the **f\_v8-cylinder-left** and **f\_v8-cylinder-right** zones in the **Display Grid** dialog box.
- b. Enable **Free** in the **Options** group box.

- c. Click **Display** and rotate the geometry to obtain the view shown in [Figure 8.2: Holes to be Closed in the f\\_v8-cylinder-left and f\\_v8-cylinder-right Zones \(p. 153\)](#).

**Figure 8.2: Holes to be Closed in the f\_v8-cylinder-left and f\_v8-cylinder-right Zones**



- d. Select the **f\_v8-cylinder-left** and **f\_v8-cylinder-right** zones in the display window using the right-mouse button.

Here, **zone** is the default selection for **Filter**.

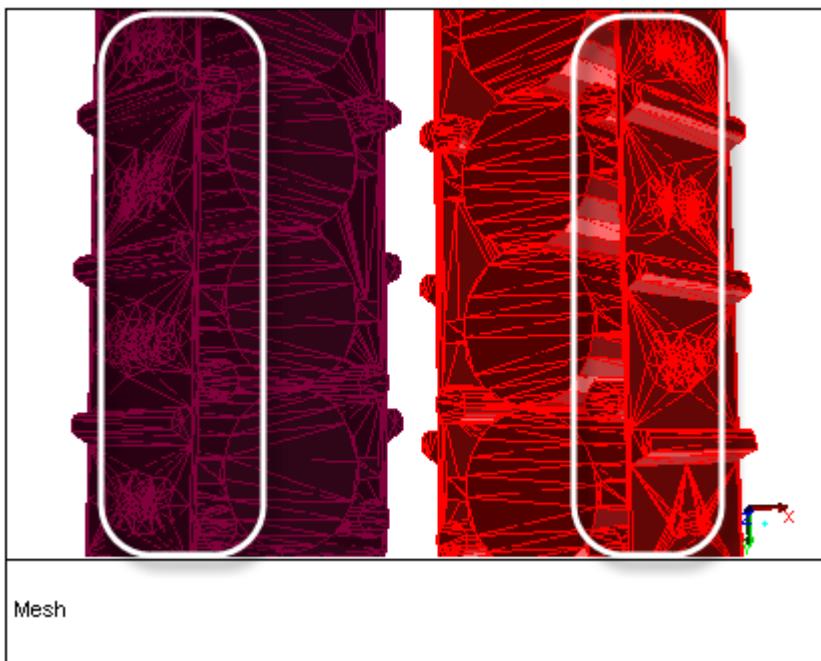
- e. Use the command /boundary/modify/repair to close the holes ([Figure 8.3: Holes Closed in the f\\_v8-cylinder-left and f\\_v8-cylinder-right Zones \(p. 154\)](#)).

OR

Use the hot-key **Ctrl-R** to close the holes.

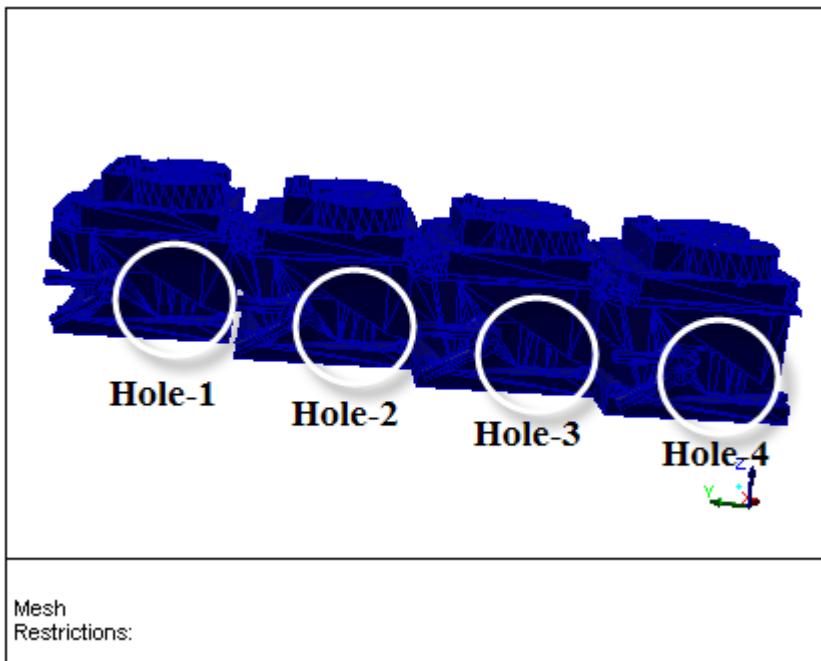
[Figure 8.3: Holes Closed in the f\\_v8-cylinder-left and f\\_v8-cylinder-right Zones \(p. 154\)](#) shows the holes closed in the **f\_v8-cylinder-left** and **f\_v8-cylinder-right** zones.

**Figure 8.3: Holes Closed in the f\_v8-cylinder-left and f\_v8-cylinder-right Zones**



2. Close the holes in the **f\_v8-airintake** zone.
  - a. Deselect the previously selected zones and select **f\_v8-airintake** in the **Display Grid** dialog box.
  - b. Disable **Free** and click **Display**. Rotate the geometry to obtain the view shown in [Figure 8.4: Holes to be Closed in the f\\_v8-airintake Zone \(p. 154\)](#).

**Figure 8.4: Holes to be Closed in the f\_v8-airintake Zone**



Four holes are visible in the **f\_v8-airintake** zone. You will close only three holes in this step, the remaining hole will be closed later.

### Warning

The remaining hole in the **f\_v8-airintake** zone will be closed in [Examine the Region to be Wrapped \(p. 163\)](#) after demonstrating the use of the **Pan Regions** and **Trace Path** features to detect and trace a leak in the geometry.

- c. Zoom in to the region shown in [Figure 8.5: Nodes Selected to Close Hole-1 in the f\\_v8-airintake Zone \(p. 155\)](#).
- d. Select **node** in the **Filter** list in the **Modify Boundary** dialog box.

### Boundary → Modify...

OR

Use the hot-key, **Ctrl-N** to select **node** as the **Filter**.

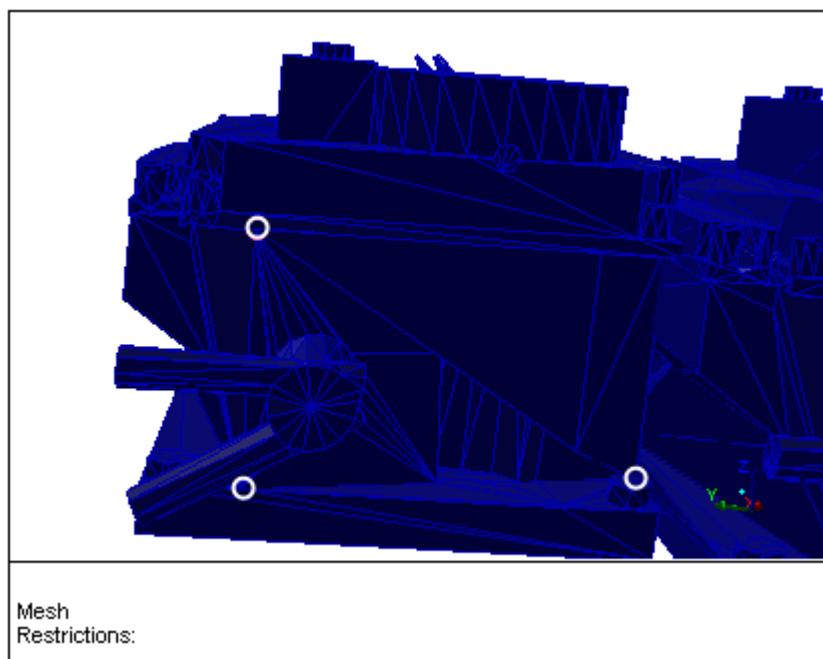
- e. Select the three nodes surrounding the hole using the right mouse button ([Figure 8.5: Nodes Selected to Close Hole-1 in the f\\_v8-airintake Zone \(p. 155\)](#)).

---

### Tip

If you select the incorrect node(s), use **Esc** to deselect the last node selected or use **F2** to deselect all the previously selected nodes.

**Figure 8.5: Nodes Selected to Close Hole-1 in the f\_v8-airintake Zone**



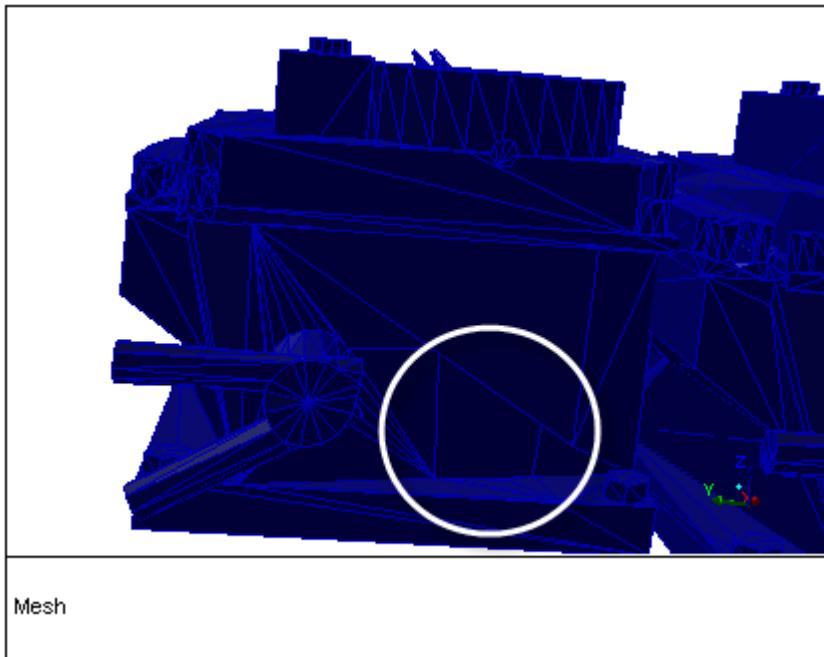
- f. Click **Create** in the **Operation** group box.

OR

Use **F5**.

A face partially covering the hole is created, making the hole smaller than the mesh size used in this tutorial (see [Figure 8.6: Closed Hole \(Hole-1\) in the f\\_v8-airintake Zone \(p. 156\)](#)).

**Figure 8.6: Closed Hole (Hole-1) in the f\_v8-airintake Zone**

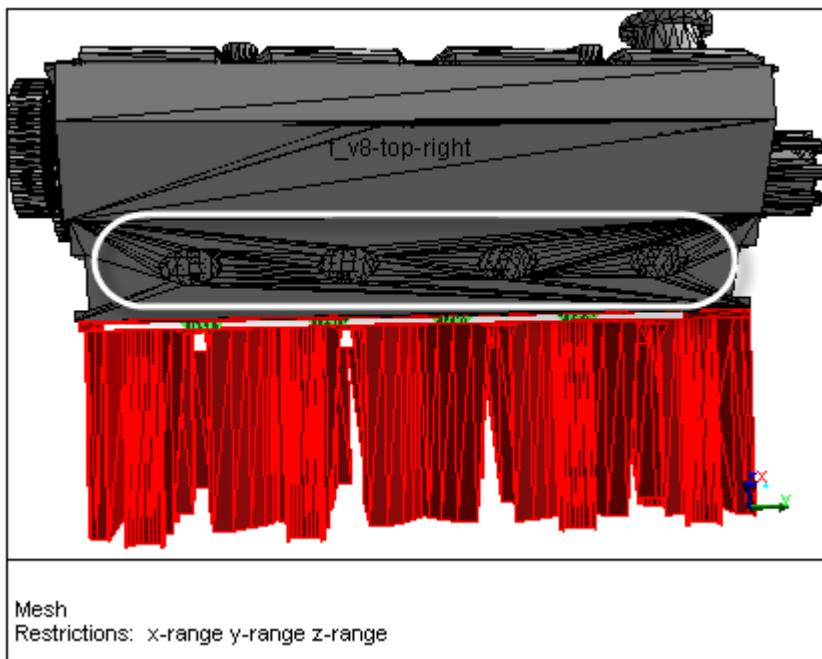


- g. Similarly, close the second and third holes (Hole-2 and Hole-3) in the **f\_v8-airintake** zone.

**Warning**

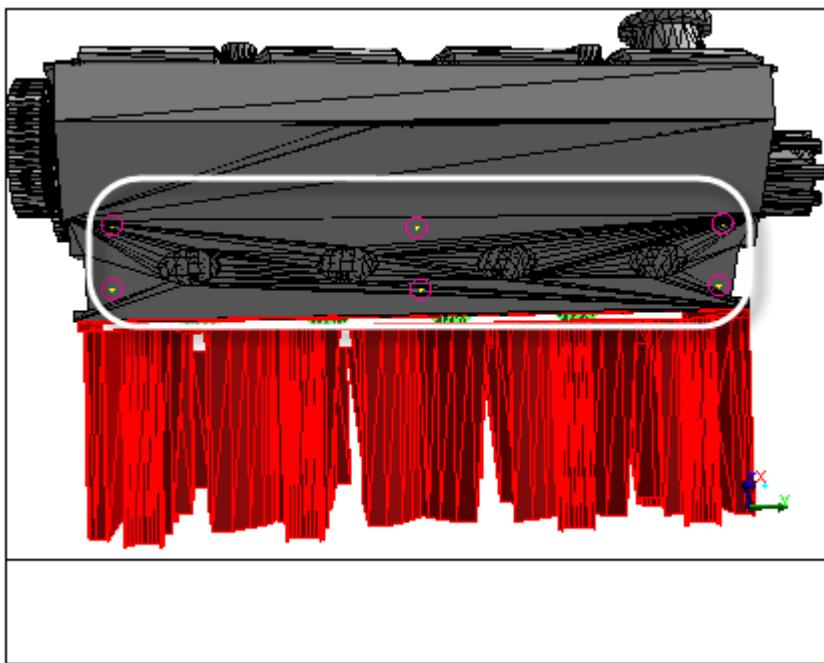
The closing of the remaining hole will be demonstrated in [Examine the Region to be Wrapped \(p. 163\)](#).

3. Close the holes in the **f\_v8-top-right** zone.
- Select all the zones in the **Display Grid** dialog box and click **Display**.
  - Select **zone** in the **Filter** list in the **Modify Boundary** dialog box (or use the hot-key, **Ctrl-Z**).
  - Select the **f\_v8-top-right** zone and click the **Set Ranges** button in the **Bounds** tab of the **Display Grid** dialog box.
  - Click **Display** and rotate the geometry to obtain the view shown in [Figure 8.7: Coplanar Holes in the f\\_v8-top-right Zone \(p. 157\)](#).

**Figure 8.7: Coplanar Holes in the f\_v8-top-right Zone**

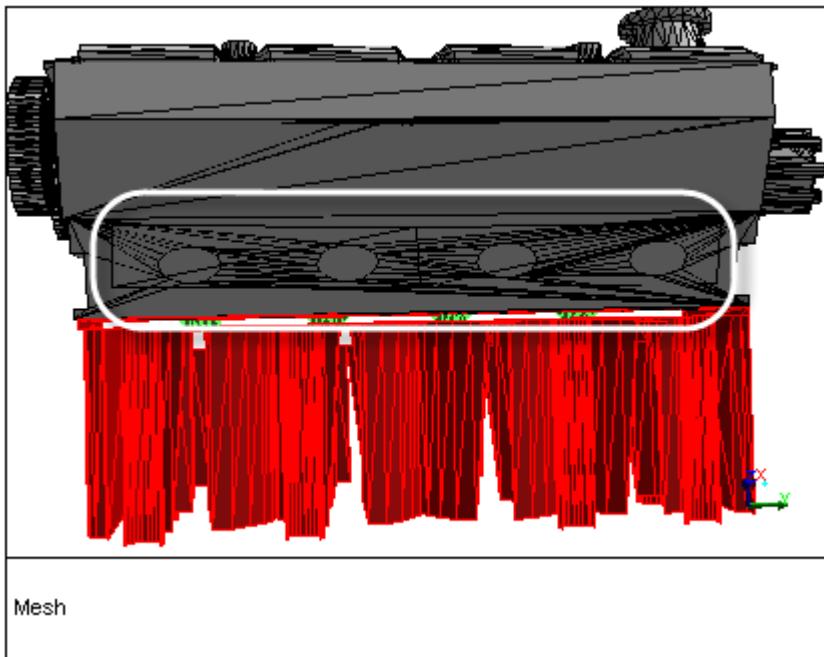
- e. Clear the **Selections** list in the **Modify Boundary** dialog box and select **position** in the **Filter** list.  
OR  
Use **F2** to clear the **Selections** list and the hot-key, **Ctrl-X** to select **position** as the **Filter**.
- f. Select the six positions shown in [Figure 8.8: Nodes Selected to Close Holes in the f\\_v8-top-right Zone \(p. 158\)](#) using the right mouse button.  
The positions should be selected in either clockwise or anti-clockwise order.
- g. Click **Create** in the **Operation** group box (or use **F5**).  
Six boundary nodes are created at the selected positions. These nodes are automatically selected in the **Selections** list.

**Figure 8.8: Nodes Selected to Close Holes in the f\_v8-top-right Zone**



- h. Retain the selection of the newly created nodes in the **Selections** list.
- i. Select **zone** in the **Filter** list and select the **f\_v8-top-right** zone (or use the hot-key, **Ctrl-Z**).
- j. Select **node** in the **Filter** list (hot-key, **Ctrl-N**) and click **Create** (or use **F5**).

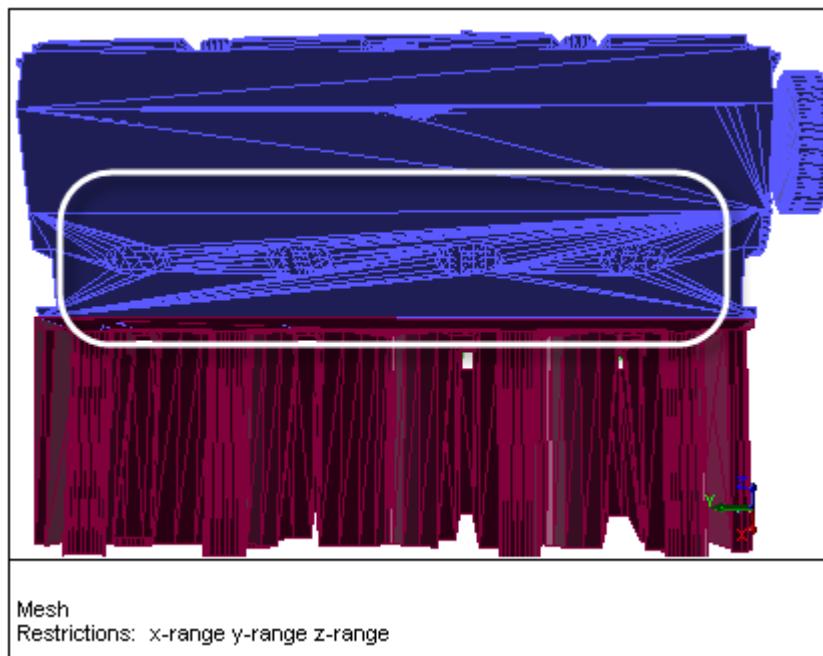
**Figure 8.9: Holes Closed in the f\_v8-top-right Zone**



4. Close the holes in the **f\_v8-top-left** zone by creating a plane surface.

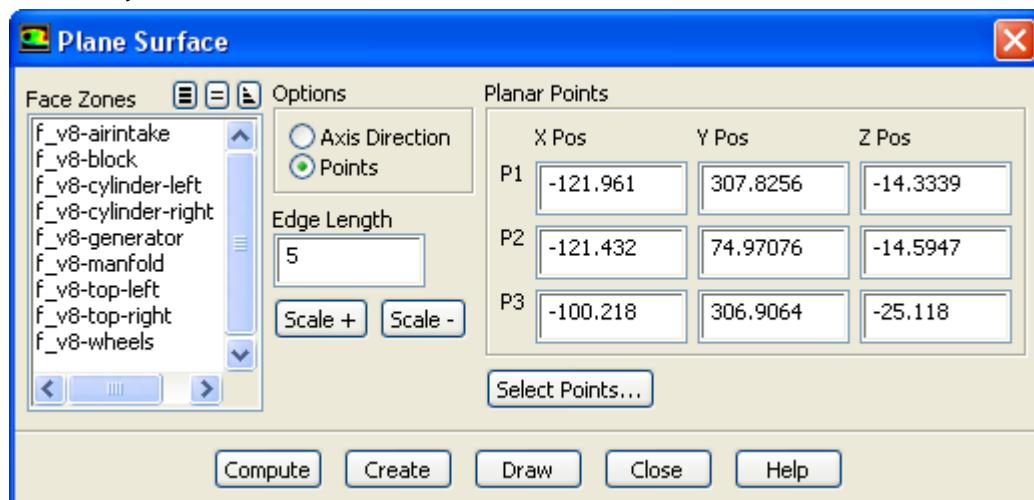
- Click **Reset** in the **Bounds** tab in the **Display Grid** dialog box.
- Display the grid.
- Select **zone** in the **Filter** list in the **Modify Boundary** dialog box (or use the hot-key, **Ctrl-Z**).
- Select the **f\_v8-top-left** zone and click the **Set Ranges** button in the **Bounds** tab of the **Display Grid** dialog box.
- Click **Display** and rotate the geometry to obtain the view shown in [Figure 8.10: Coplanar Holes in the f\\_v8-top-left Zone \(p. 159\)](#).

**Figure 8.10: Coplanar Holes in the f\_v8-top-left Zone**



- Create a plane surface to cover the coplanar holes.

**Boundary → Create → Plane Surface...**

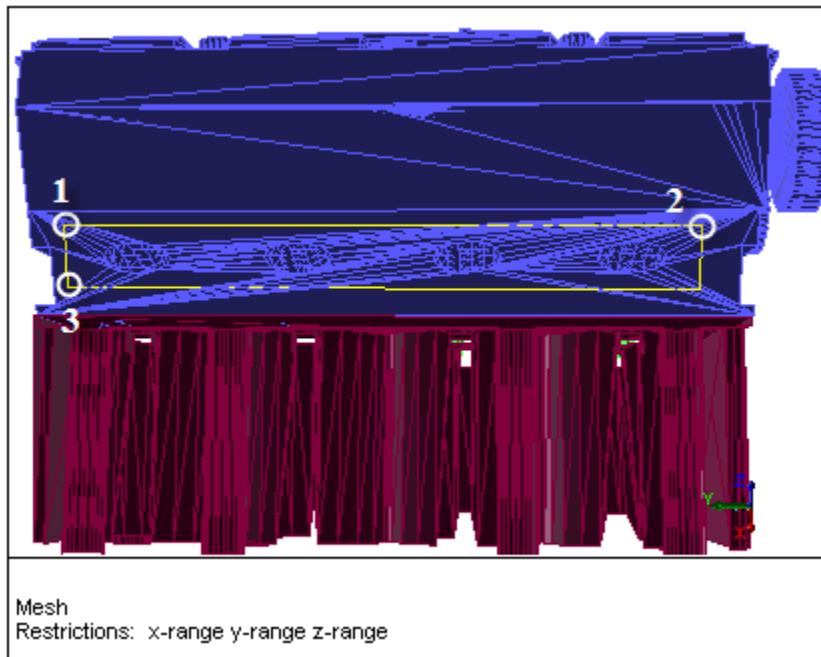


- Select **Points** in the **Options** list.

- ii. Click **Select Points...** to select the points defining the plane surface.
- iii. Select three points to define a plane using the right mouse button (Figure 8.11: Selection of Points for Creating the Plane Surface (p. 160)).

Select the two points along the longest side first. The coordinates of the three selected points will be updated in the **Planar Points** group box.

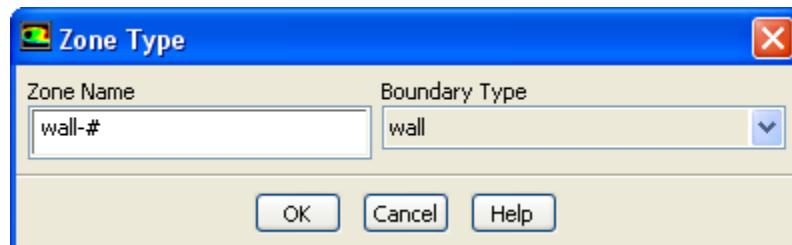
**Figure 8.11: Selection of Points for Creating the Plane Surface**



- iv. Enter 5 for **Edge Length**.

- v. Click **Create**.

The **Zone Type** dialog box will open, displaying the default entry for **Zone Name**.



- vi. Click **OK** in the **Zone Type** dialog box.

The newly created zone (**wall-#**, where # is the zone ID) will be added to the **Face Zones** selection list.

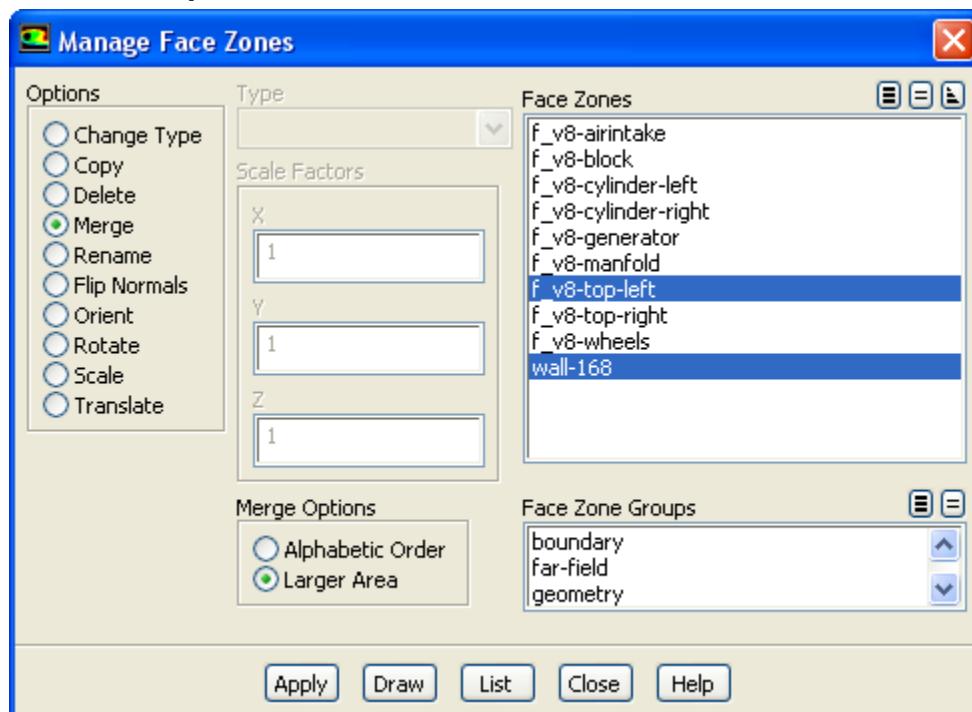
- vii. Close the **Plane Surface** dialog box.

- g. Merge the newly created zone (**wall-#**) with the **f\_v8-top-left** zone using the **Larger Area** option.

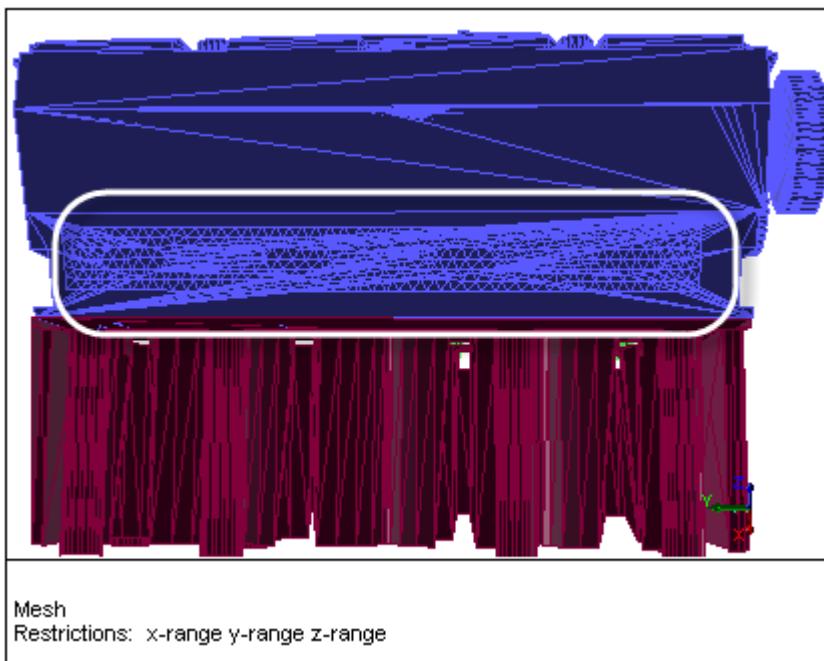
The name of the zone having a larger area will be retained after merging the zones.

**Boundary → Manage...**

- Select **f\_v8-top-left** and **wall-#** in the **Face Zones** selection list.



- Select **Merge** in the **Options** list.
  - Select **Larger Area** in the **Merge Options** list and click **Apply**.  
The **wall-#** zone will be merged with the **f\_v8-top-left** zone.
  - Close the **Manage Face Zones** dialog box.
- 
- Click **Display** in the **Display Grid** dialog box to see the recently closed holes ([Figure 8.12: Holes Closed in the f\\_v8-top-left Zone Using a Plane Surface \(p. 162\)](#)).

**Figure 8.12: Holes Closed in the f\_v8-top-left Zone Using a Plane Surface**

5. Save the mesh (engine.msh.gz).

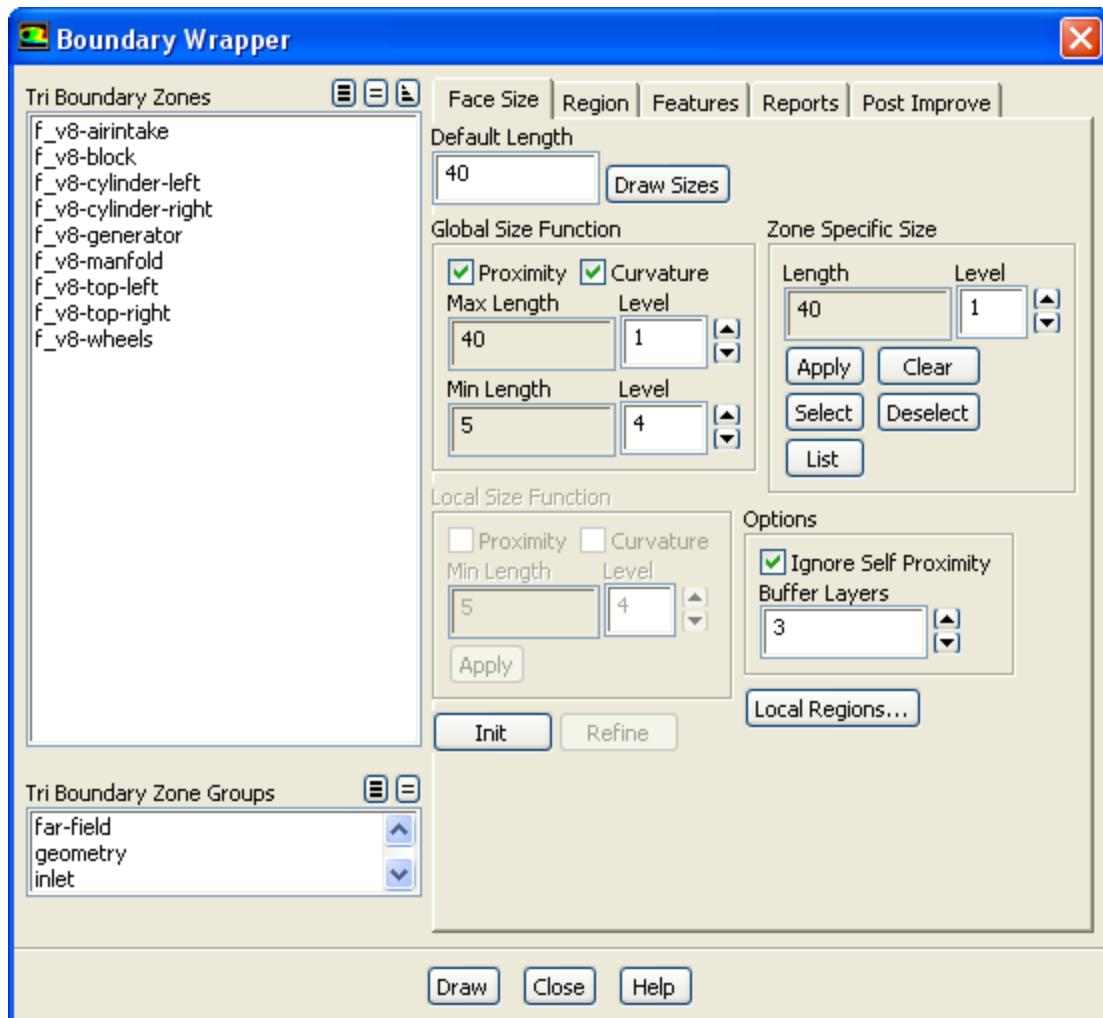
**File → Write → Mesh...**

## 8.5. Initialize the Surface Wrapper

The accuracy of the wrapper depends on the cell size distribution of the Cartesian grid. You can specify the cell size of the Cartesian grid according to your requirement. Finer cells give better results, but also increase the computational time.

The **Min Length** is the minimum allowable cell size in the Cartesian grid. In this case, the targeted **Min Length** is 5 mm. This is achieved by setting the **Default Length** to 40 and the **Min Length Level** to 4.

**Boundary → Wrap...**

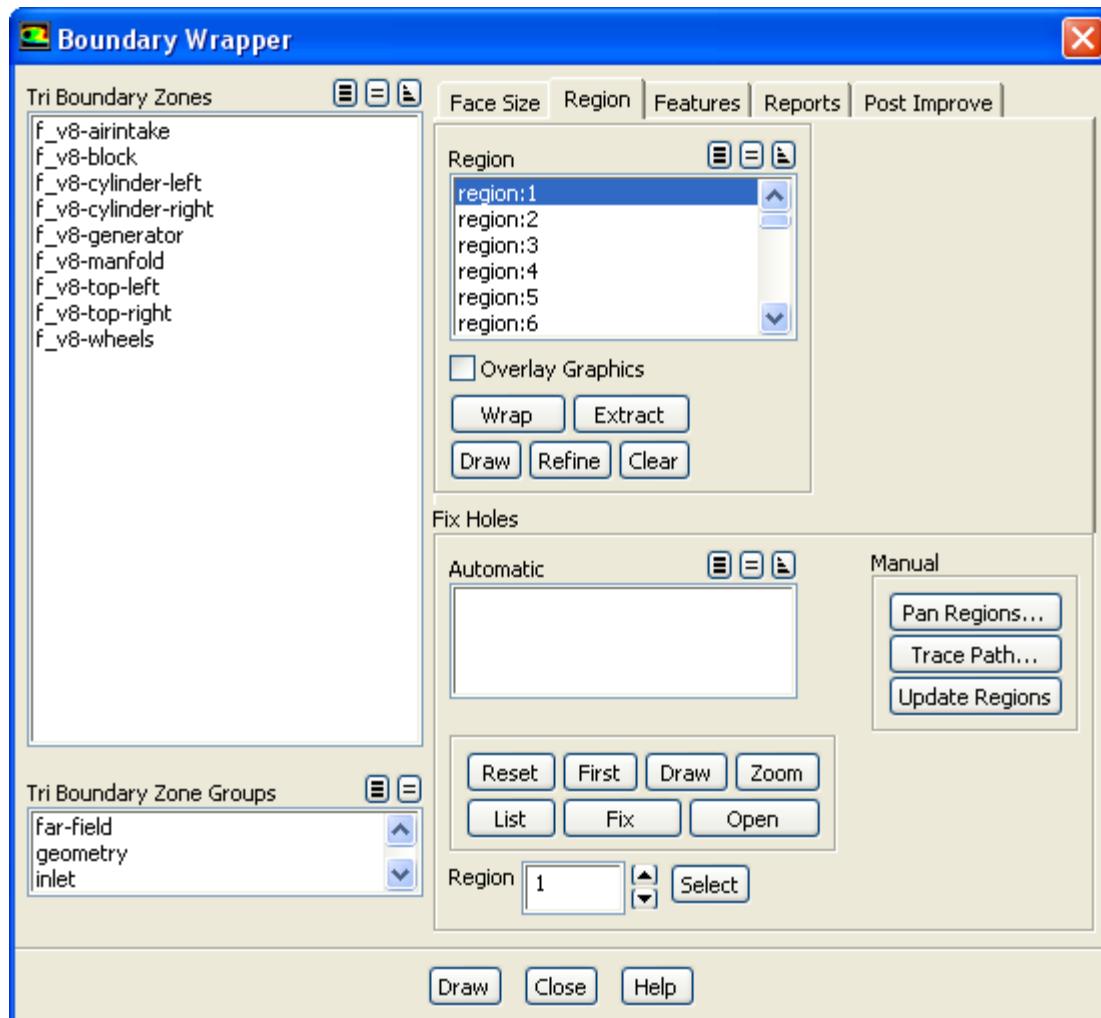


1. Enter 40 for **Default Length** and press **Enter**.
2. Enable **Proximity** and **Curvature** in the **Global Size Function** group box.
3. Enable **Ignore Self Proximity** and increase **Buffer Layers** to 3 in the **Options** group box.
4. Click **Init**.

A Cartesian hanging node grid will be created and refined based on the defined size functions. The grid will then be intersected with the geometry, thereby creating a number of Cartesian closed regions. A report of the regions created will be provided.

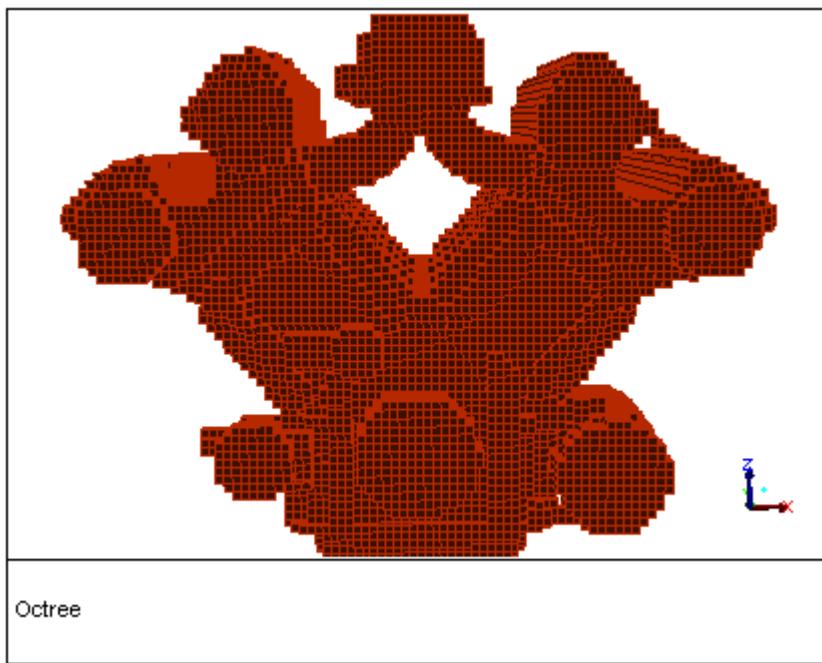
## 8.6. Examine the Region to be Wrapped

1. Click **Reset** in the **Bounds** tab of the **Display Grid** dialog box.
2. Click the **Region** tab in the **Boundary Wrapper** dialog box.



3. Retain the selection of **region:1** and click **Draw** in the **Region** group box to draw the largest region.

Figure 8.13: Region to be Wrapped (p. 165) shows the region to be wrapped.

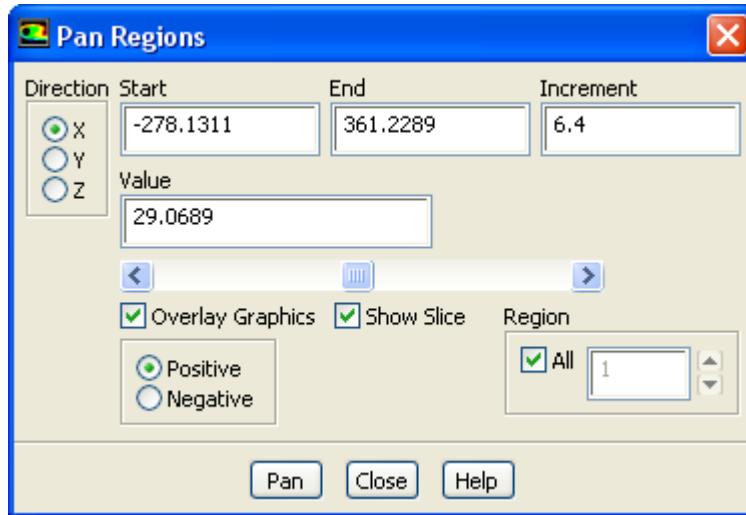
**Figure 8.13: Region to be Wrapped**

4. Pan the region to examine grid distribution and to search for holes.

- a. Display the entire geometry.

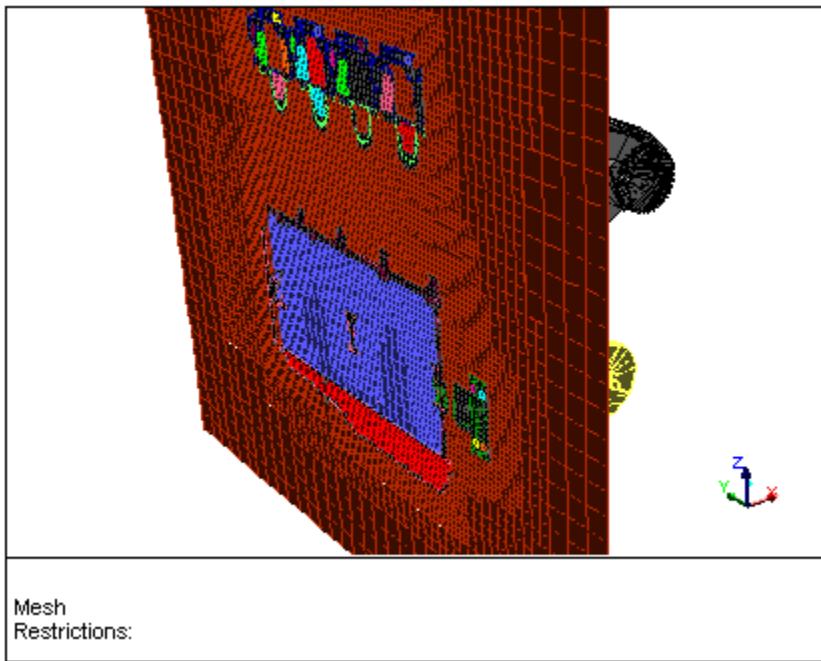
**Display → Grid...**

- b. Click the **Pan Regions...** button in the **Region** tab of the **Boundary Wrapper** dialog box to open the **Pan Regions** dialog box.



- i. Enable **Overlay Graphics**.
- ii. Move the slider bar to position the pan plane as shown in [Figure 8.14: Pan Plane with the Overlaid Geometry \(p. 166\)](#) (**Value** ≈29).

You may need to switch between **Positive** and **Negative** to view the geometry on either side of the pan plane.

**Figure 8.14: Pan Plane with the Overlaid Geometry**

- iii. Zoom in to the region of the leak ([Figure 8.15: Region of the Leak \(p. 166\)](#)).

**Figure 8.15: Region of the Leak**

The presence of the main region color on the inside ([Figure 8.14: Pan Plane with the Overlaid Geometry \(p. 166\)](#)) is an indication of a possible leak. In this case, the remaining hole in the **f\_v8-airintake** zone has caused this leak. You can switch between **Positive** and **Negative** to flip the geometry, if required.

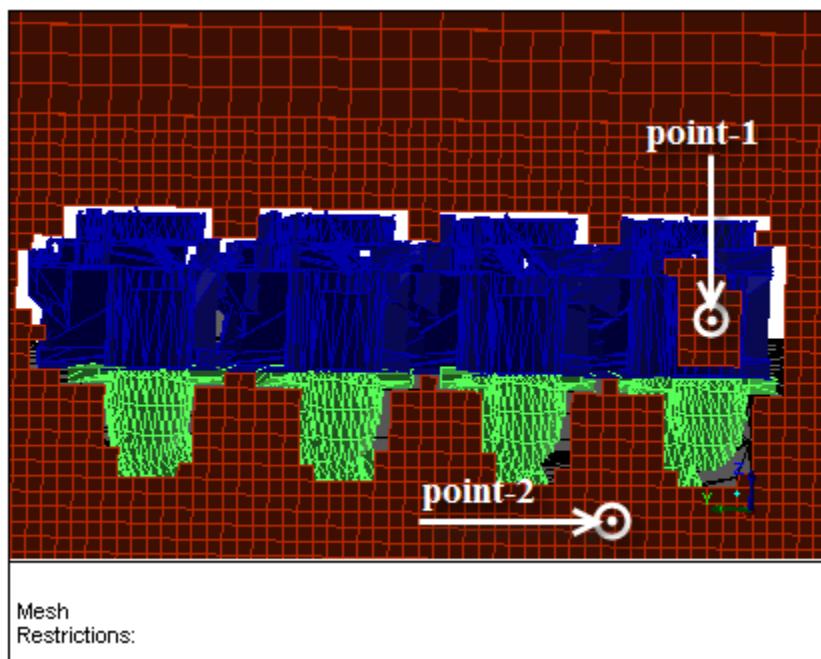
- iv. Disable **All** in the **Region** group box and set **Region** to 1 (see [Figure 8.16: Detecting the Leakage using the Trace Path dialog box \(p. 167\)](#)).

- c. Use the **Trace Path** feature to detect the leakage.
  - i. Click the **Trace Path...** button in the **Region** tab of the **Boundary Wrapper** dialog box to open the **Trace Path** dialog box.



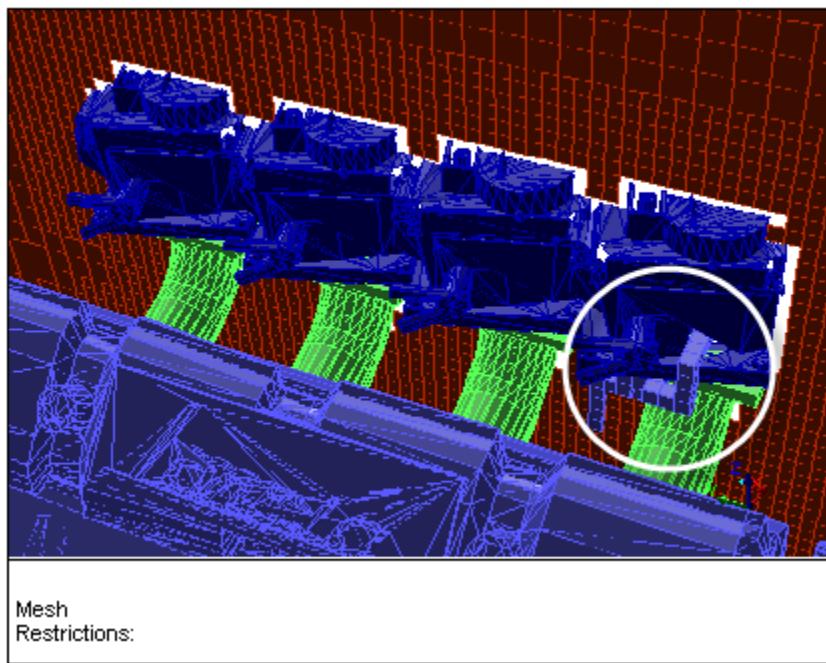
- ii. Click the **Select Points...** button and select two points as shown in Figure 8.16: Detecting the Leakage using the Trace Path dialog box (p. 167).

**Figure 8.16: Detecting the Leakage using the Trace Path dialog box**



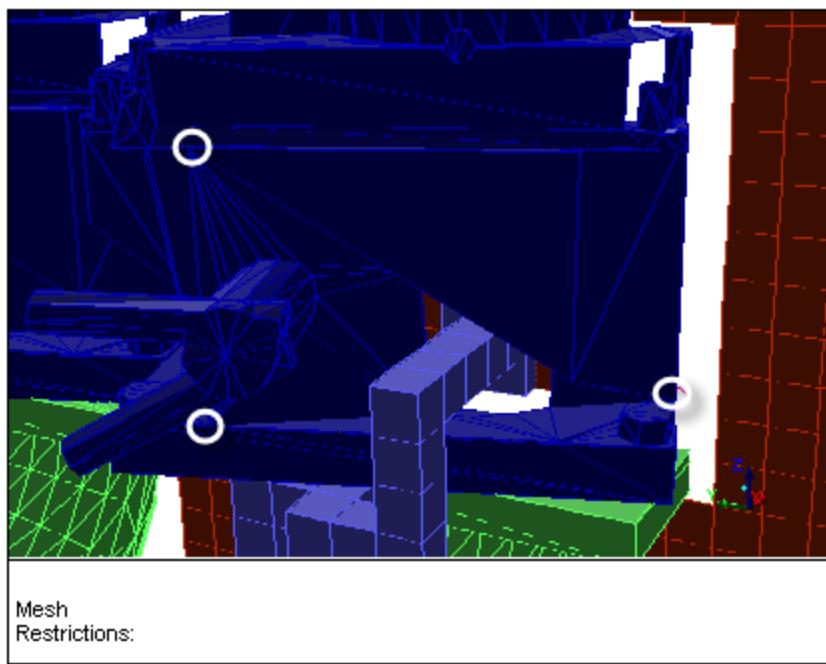
- iii. Click **Trace** (Figure 8.17: Display of Geometry with the Traced Path (p. 168)).

The trace path is displayed by coloring all the faces in the path.

**Figure 8.17: Display of Geometry with the Traced Path****Note**

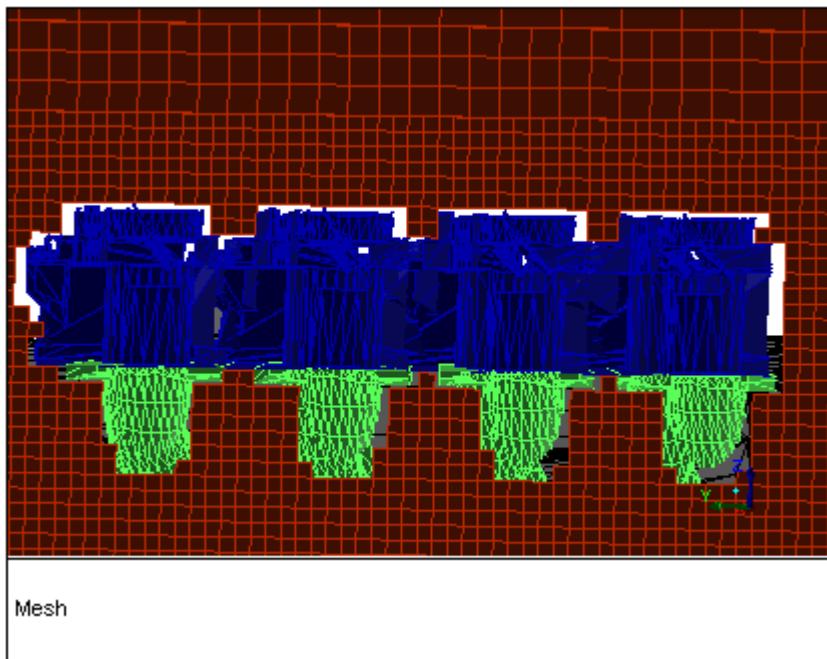
You may need to switch between **Positive** and **Negative** in the **Pan Regions** dialog box to obtain the display in [Figure 8.17: Display of Geometry with the Traced Path \(p. 168\)](#).

- iv. Zoom in to the hole and select three nodes as shown in [Figure 8.18: Nodes Selected to Close the Hole in the f\\_v8-airintake Zone \(p. 168\)](#).

**Figure 8.18: Nodes Selected to Close the Hole in the f\_v8-airintake Zone**

- v. Click **Create** in the **Modify Boundary** dialog box or use the hot-key **F5** to close the hole.
5. Select all the geometry in the **Tri Boundary Zones** selection list in the **Boundary Wrapper** dialog box.
6. Click **Update Regions** in the **Region** tab to update the region based on the newly added face (see [Figure 8.19: Region Updated for the Closed Hole \(p. 169\)](#)).

**Figure 8.19: Region Updated for the Closed Hole**

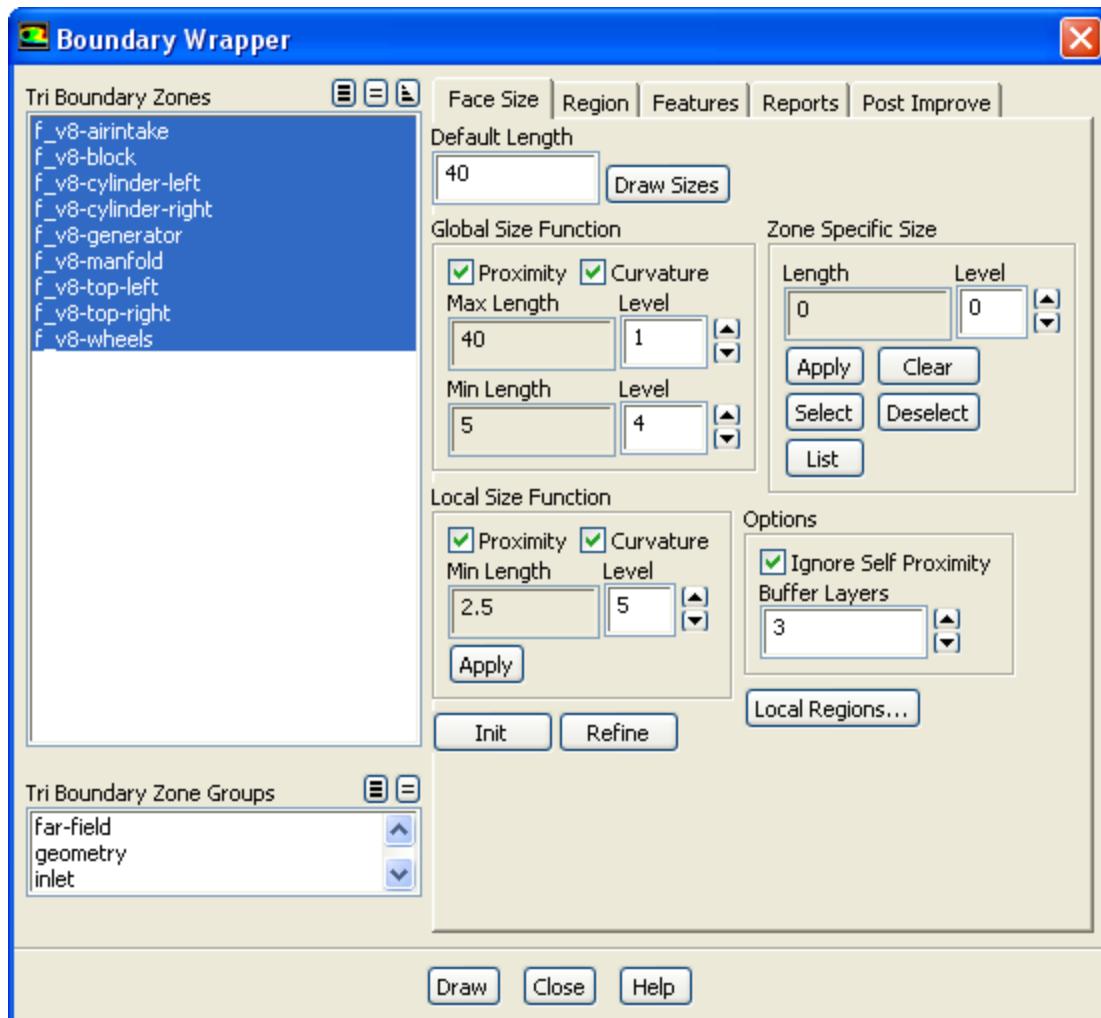


There are no more big leakages in the geometry. You will however see later that there are leakages having size smaller than 5 mm but bigger than 2.5 mm.

7. Close the **Pan Regions** and **Trace Path** dialog boxes.

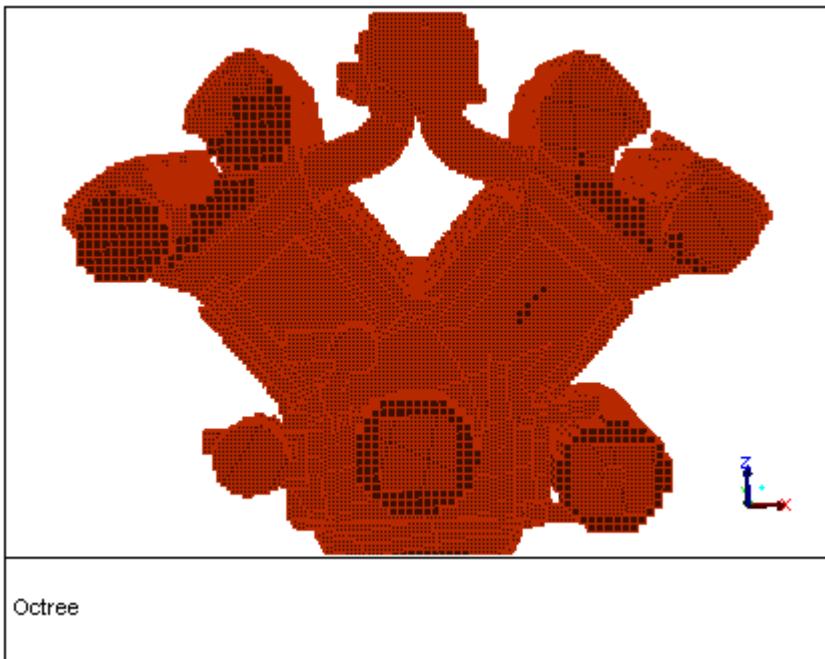
## 8.7. Refine the Main Region

This section demonstrates the refining of the Cartesian grid using the **Local Size Function**.



1. Enable **Proximity** and **Curvature** in the **Local Size Function** group box in the **Face Size** tab of the **Boundary Wrapper** dialog box.
2. Set **Level** to 5 in the **Local Size Function** group box.
3. Select all the surfaces in the **Tri Boundary Zones** selection list.
4. Click **Apply** in the **Local Size Function** group box.
5. Click **Refine**.

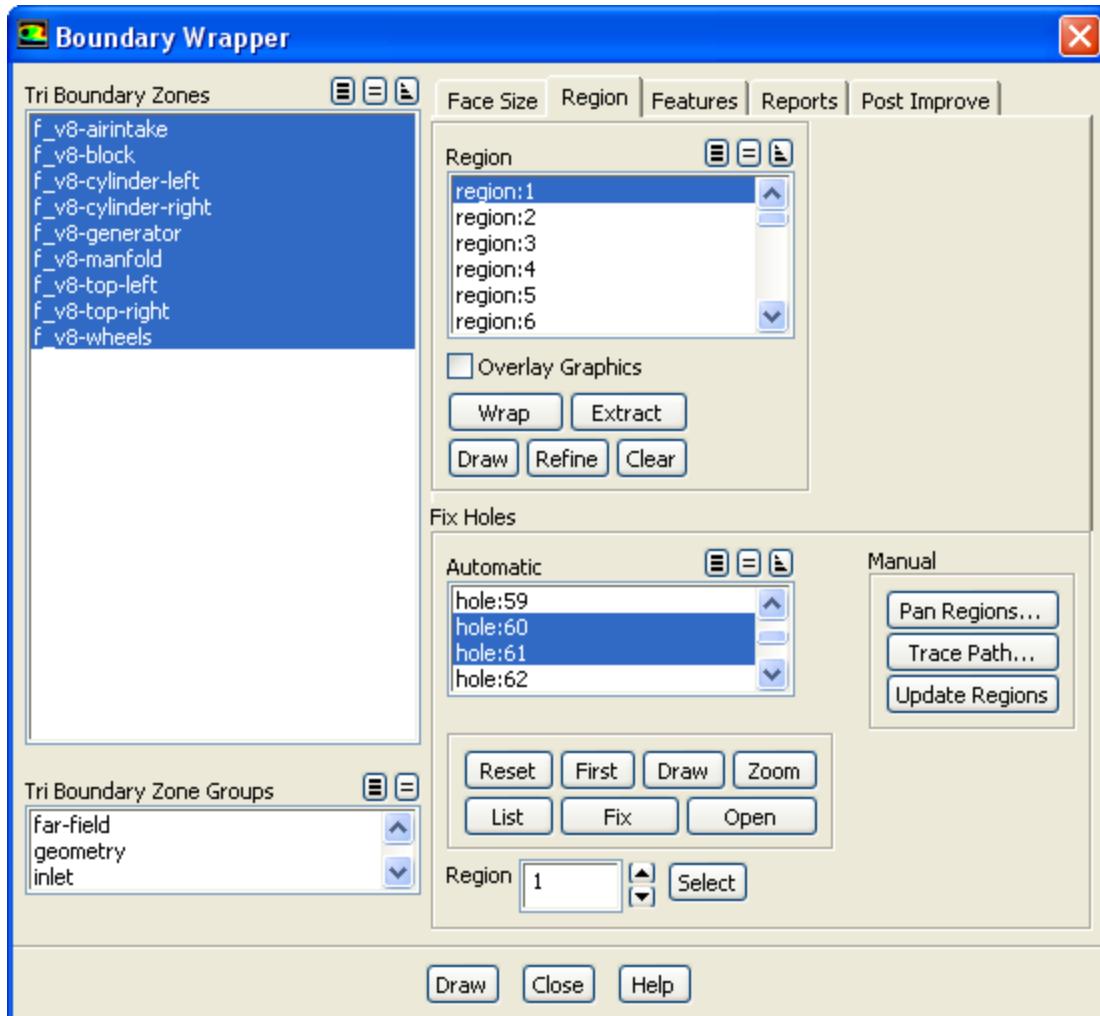
After refining the region, **region:1** is now much finer as seen in [Figure 8.20: Region After Refining Using Local Size Function \(p. 171\)](#).

**Figure 8.20: Region After Refining Using Local Size Function**

The minimum edge length is now reduced to 2.5 mm, smaller than some leakages. Even though the edge length is now smaller than the leakages, no regions will be subdivided. The cells at the potential holes will be marked and labeled as a "hole".

## 8.8. Close Small Holes Automatically

This section demonstrates the automatic fixing of all the potential holes for **region:1**.



- Set **Region** to 1 and click **Select** in the **Fix Holes** group box in the **Region** tab of the **Boundary Wrapper** dialog box.

All the potential holes created for **region:1** will be selected in the **Automatic** selection list.

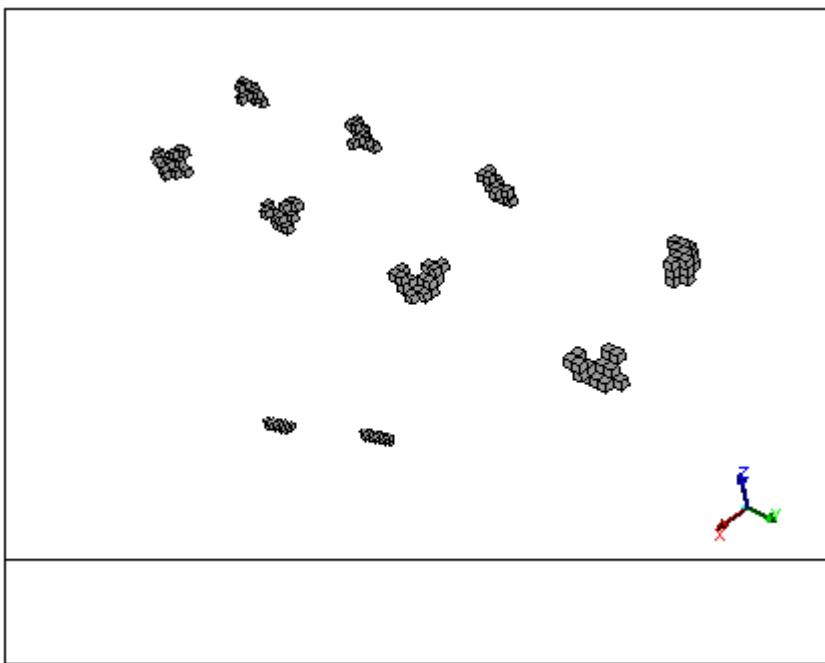
- Deselect the previously selected zones in the **Face Zones** selection list in the **Faces** tab of the **Display Grid** dialog box.

**Display → Grid...**

- Click **Display**.

This will allow you to display only the potential holes.

- Click **Draw** in the **Fix Holes** group box in the **Region** tab of the **Boundary Wrapper** dialog box (Figure 8.21: Potential Holes in region:1 (p. 173)).

**Figure 8.21: Potential Holes in region:1**

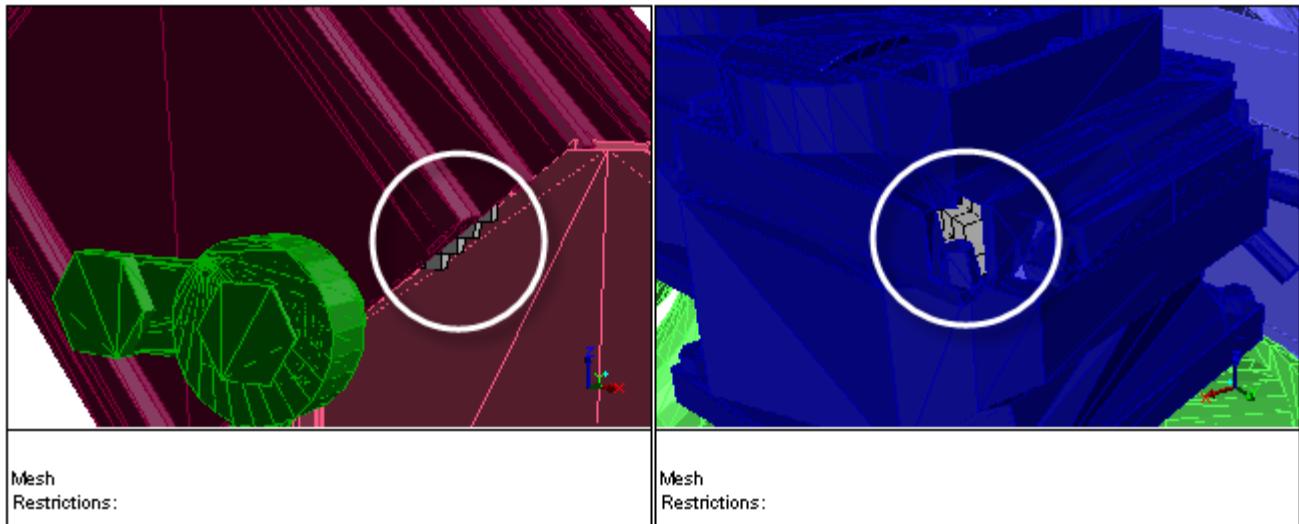
5. Examine the potential holes.

- a. Click **First** in the **Fix Holes** group box.

The display will be limited to the vicinity of the first selected hole in the **Automatic** selection list. The **First** button will be replaced by **Next**.

- b. Click **Next** to view the next selected hole.  
c. Click **Next** repeatedly to successively display the selected holes.

[Figure 8.22: Display of Holes \(p. 173\)](#) shows examples of holes in the wrapper region.

**Figure 8.22: Display of Holes**

You will now fix the holes using the automatic hole fixing option.

### 6. Fix the holes in **region:1**.

- a. Retain the selection of the holes for **region:1** in the **Automatic** selection list in the **Region** tab of the **Boundary Wrapper** dialog box.
- b. Click **Fix** in the **Fix Holes** group box in the **Region** tab in the **Boundary Wrapper** dialog box.
- c. Select all the zones in the **Face Zones** selection list in the **Display Grid** dialog box.
- d. Click **Display** (Figure 8.23: Holes Fixed Using the Automatic Hole Fixing Option (p. 174)).

**Figure 8.23: Holes Fixed Using the Automatic Hole Fixing Option**

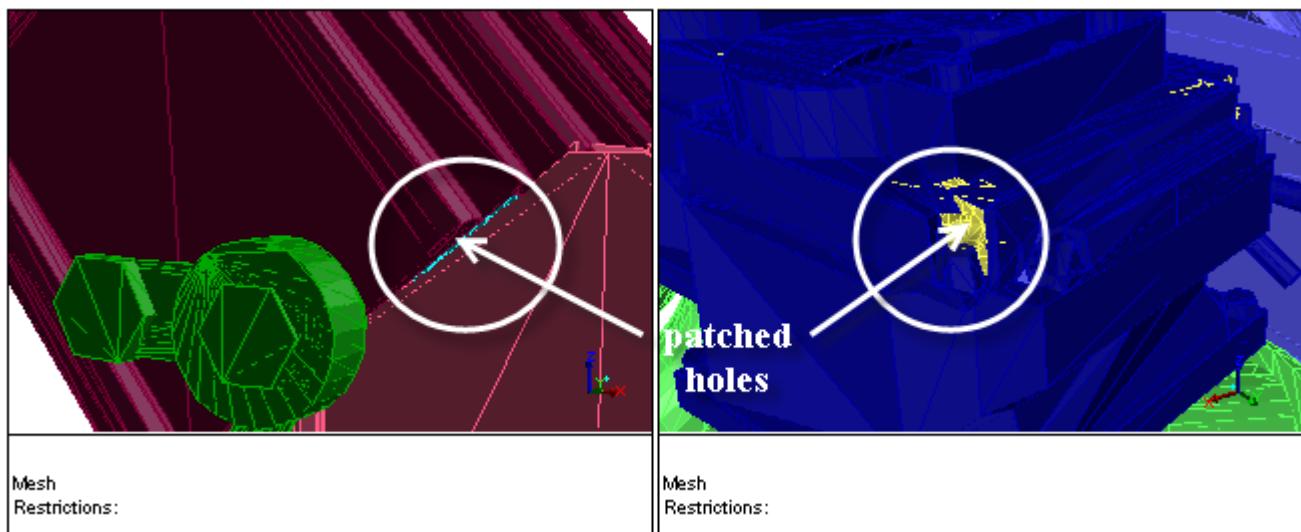


Figure 8.23: Holes Fixed Using the Automatic Hole Fixing Option (p. 174) shows the holes closed by the automatic hole fixing operation.

---

#### Note

For bigger models having a large number of holes, this step may be omitted as the time taken may be considerable, as well as some of the "holes" might not be real holes. In this case, though the initial wrapper surface may have more cells having higher skewness, the final mesh would have similar quality as the corresponding mesh with all the holes fixed.

---

## 8.9. Wrap the Main Region

1. Select **region:1** in the **Region** list in the **Region** tab of the **Boundary Wrapper** dialog box.
2. Click **Wrap**.

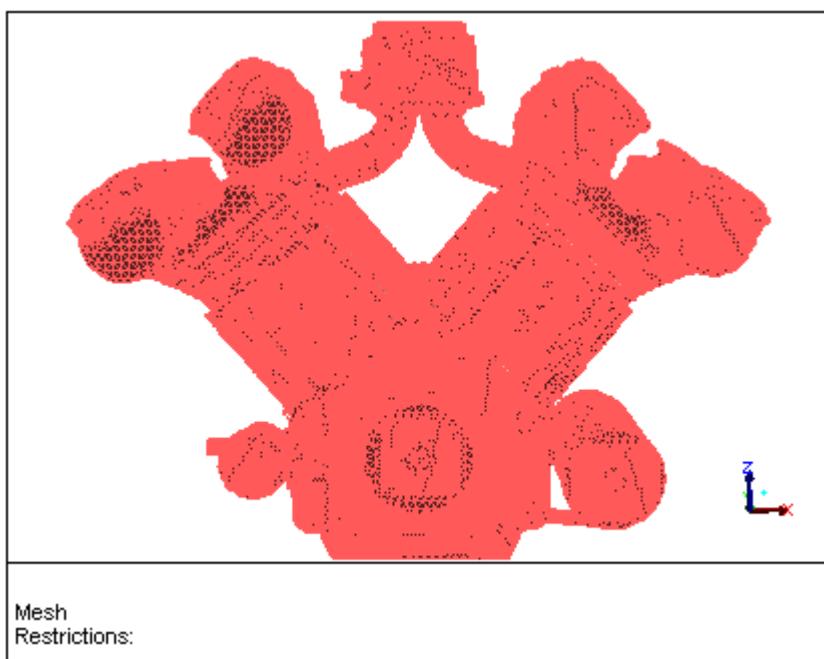
A **Question** dialog box will appear, asking if you want to delete all the regions. Deleting the regions will reduce the peak memory.

3. Click **Yes** in the **Question** dialog box.

The wrapper surface for **region:1, wrapper-surf-#**, will be created and will be available in the **Tri Boundary Zones** selection list.

- Display the wrapper surface (Figure 8.24: Wrapper Surface (p. 175)).

**Figure 8.24: Wrapper Surface**



You may need to manipulate the display in the graphics window to obtain the view shown in Figure 8.24: Wrapper Surface (p. 175).

- Save the mesh file (engine00.msh.gz).

## 8.10. Capture Features

- Click the **Features** tab in the **Boundary Wrapper** dialog box.
- Select all the geometry whose features are to be captured in the **Tri Boundary Zones** selection list.

---

### Note

Make sure that the patched holes (zones with the **vs\_** prefix) and the wrapper surface are not selected in the **Tri Boundary Zones** list.

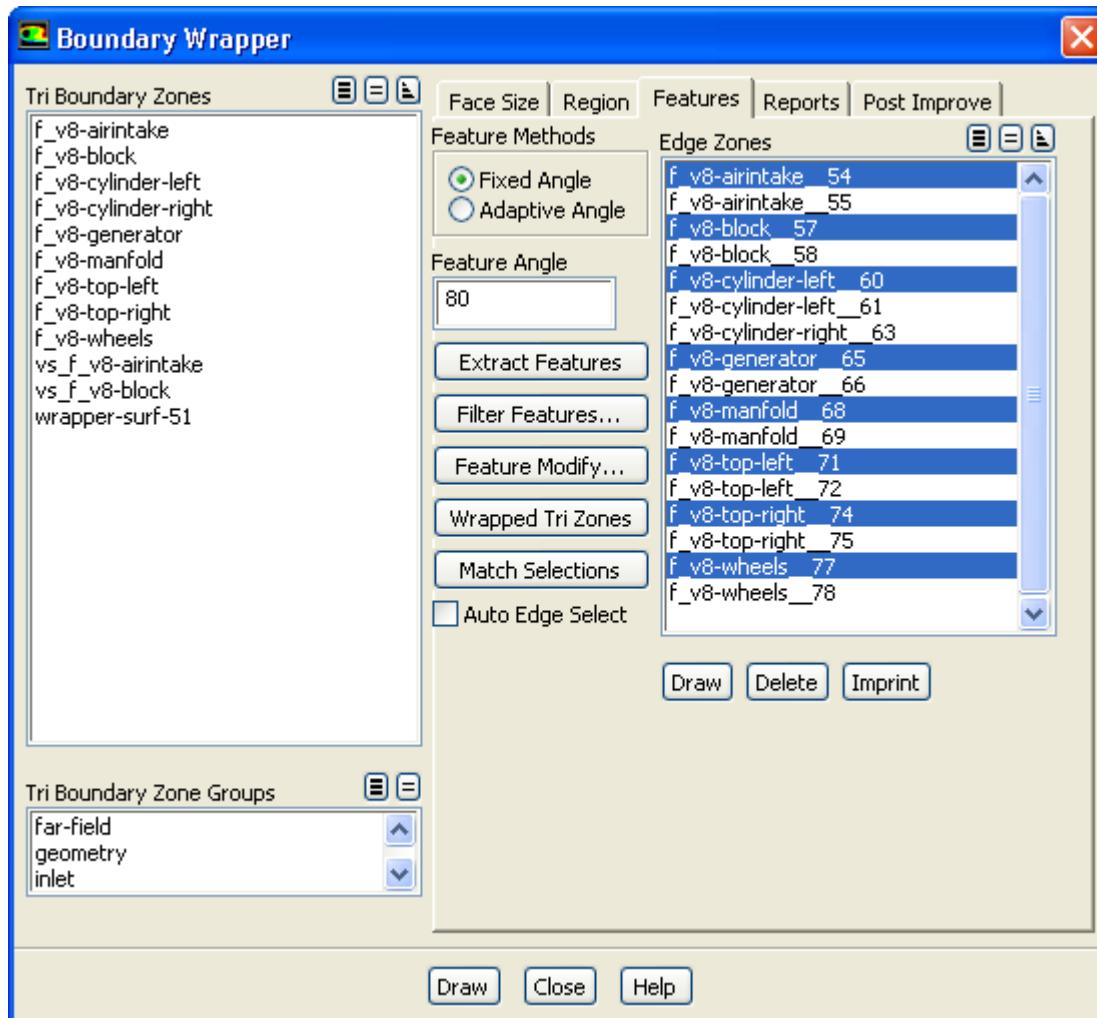
- Click **Extract Features**.

The extracted features will now be available in the **Edge Zones** selection list.

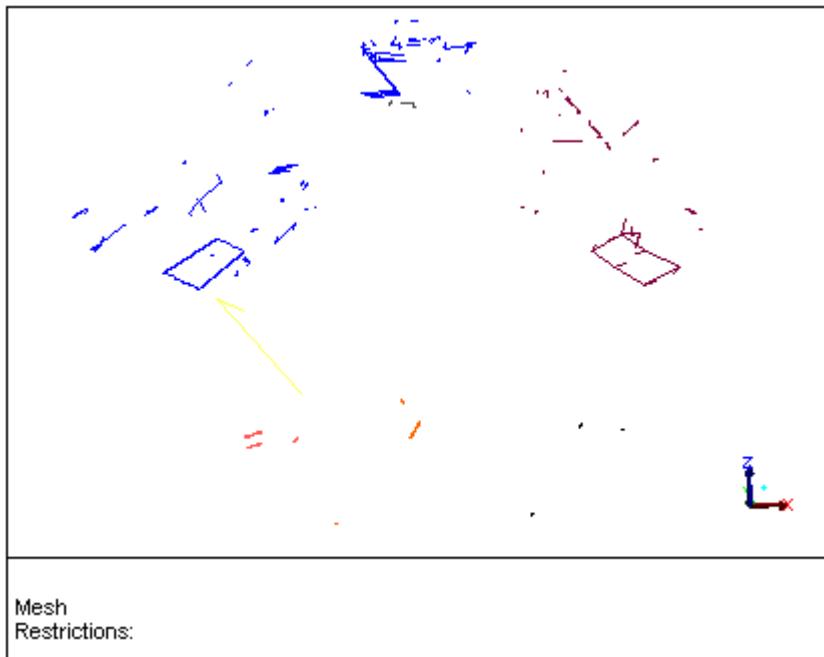
- Select all the zones in the **Edge Zones** selection list and click **Draw** (below the **Edge Zones** selection list).

You can reproject the wrapper surface onto the important features extracted. If the extracted features include details which are not required (e.g., embossed company logos, etc.), they may be deleted before proceeding with the imprinting.

5. Draw the edge zones individually to determine the important features required for imprinting.



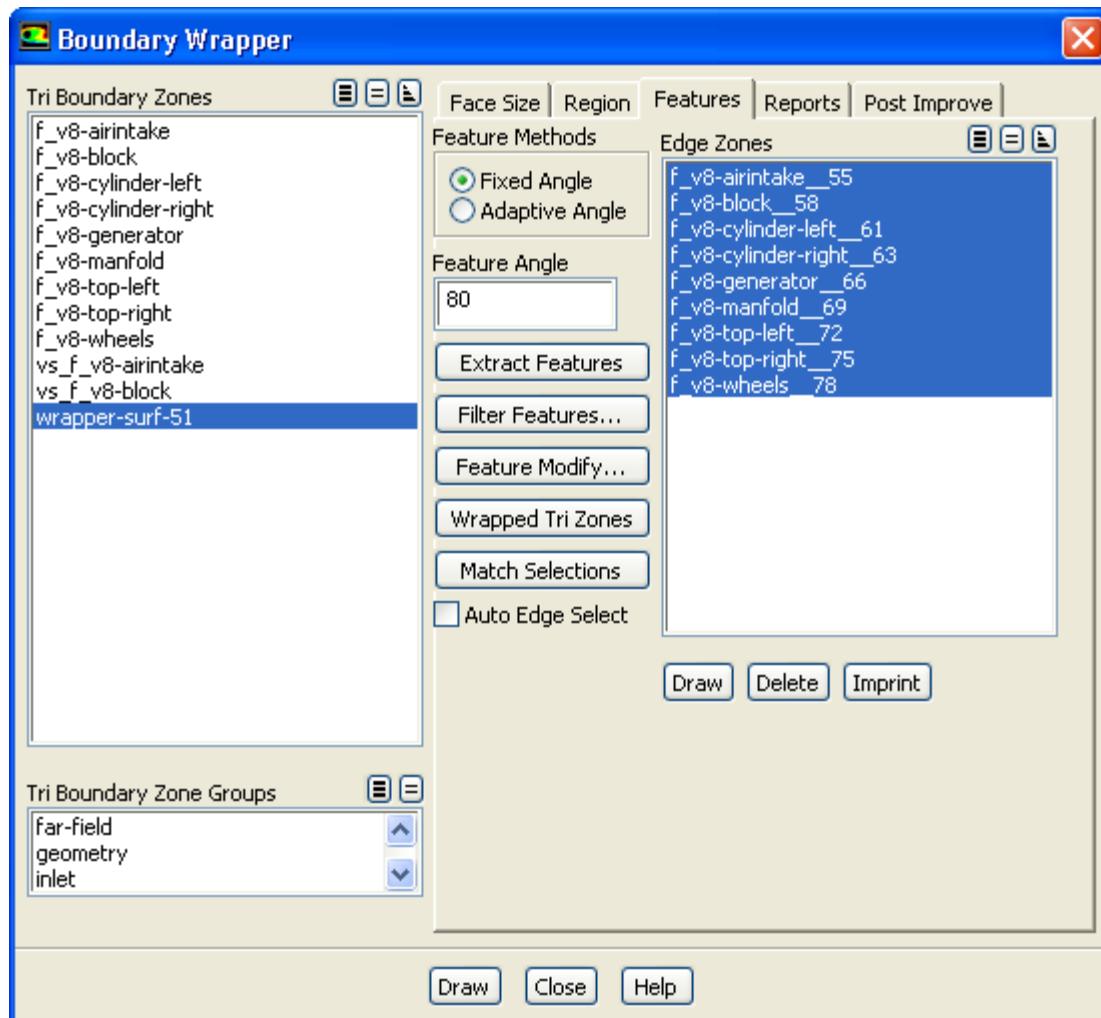
6. Select the unnecessary edge zones in the **Edge Zones** selection list in the **Boundary Wrapper** dialog box.
7. Click the **Draw** button below the **Edge Zones** selection list (Figure 8.25: Insignificant Features to be Deleted (p. 177)).

**Figure 8.25: Insignificant Features to be Deleted**

8. Click **Delete** to delete the insignificant features.
  9. Select only the wrapper surface (**wrapper-surf-#**) in the **Tri Boundary Zones** selection list and click **Draw**.
- Zoom in to the region shown in [Figure 8.26: Wrapper Surface and Contours of Distance During Imprinting \(p. 179\)](#).
10. Select all the features to be imprinted in the **Edge Zones** selection list and click **Draw**.
  11. Click the **Reports** tab in the **Boundary Wrapper** dialog box and click **Draw Contours**.

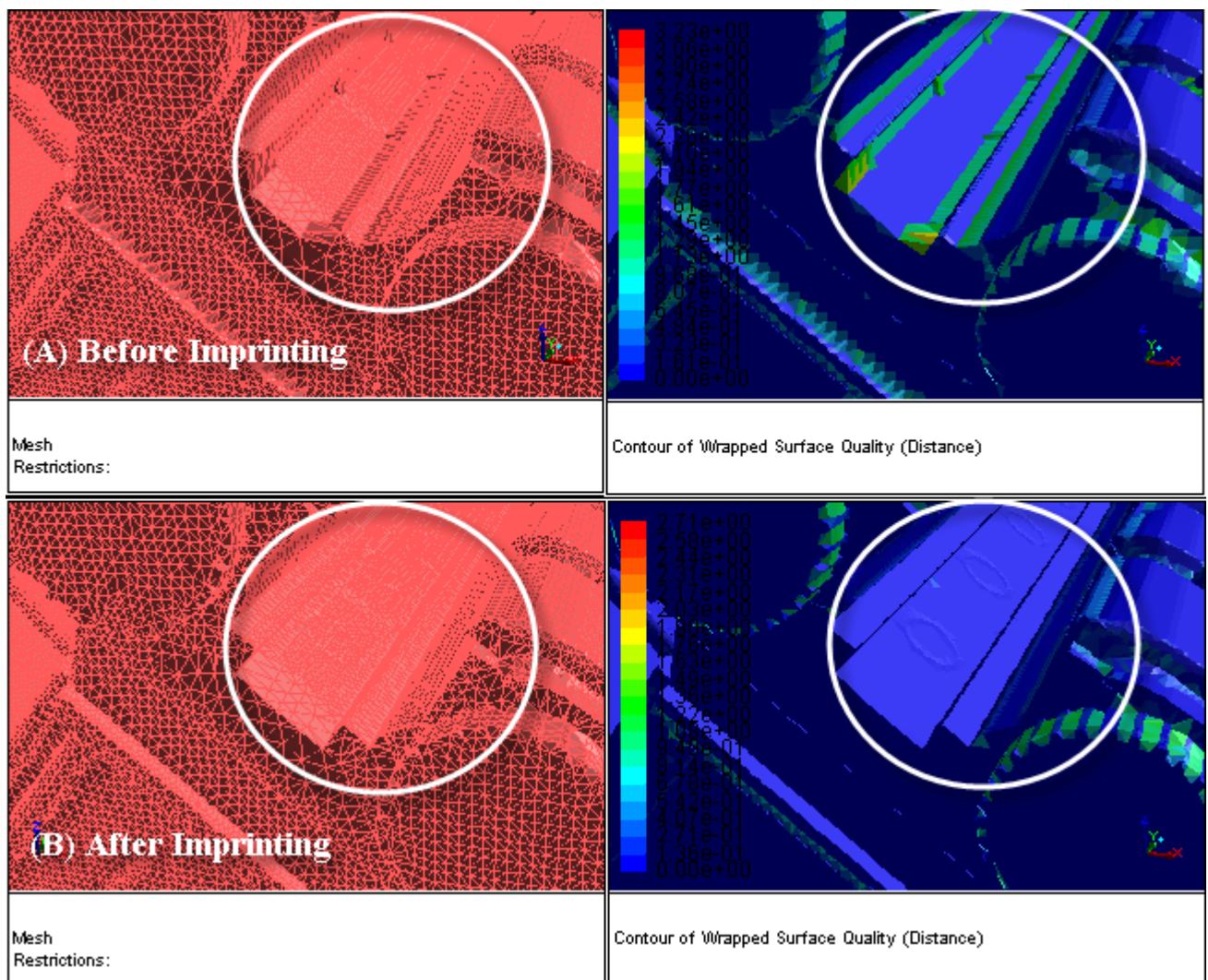
You can use the contours of wrapped surface quality to verify the imprinting of features ([Figure 8.26: Wrapper Surface and Contours of Distance During Imprinting \(p. 179\)](#)).

12. Retain the selection of the wrapper surface and all the features to be imprinted in the **Tri Boundary Zones** and **Edge Zones** selection lists, respectively.



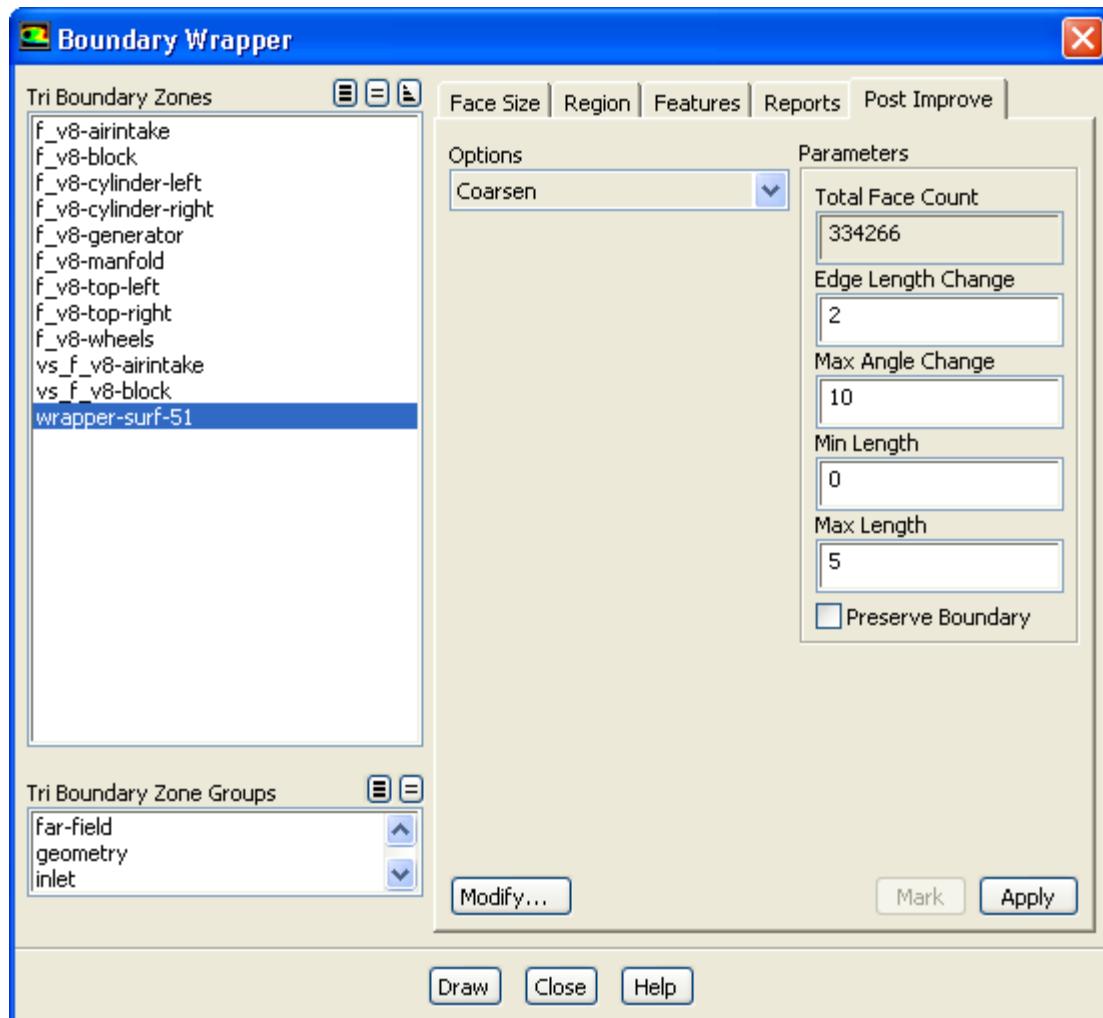
13. Click **Imprint**.
14. Display the wrapper surface to see the effect of imprinting.
15. Click **Draw Contours** in the **Reports** tab of the **Boundary Wrapper** dialog box.
16. Retain the selection of the wrapper surface in the **Tri Boundary Zones** selection list and the features to be imprinted in the **Edge Zones** selection list in the **Features** tab.
17. Click **Imprint** again.

[Figure 8.26: Wrapper Surface and Contours of Distance During Imprinting \(p. 179\)](#) shows the wrapper surface and contours of distance before and after imprinting.

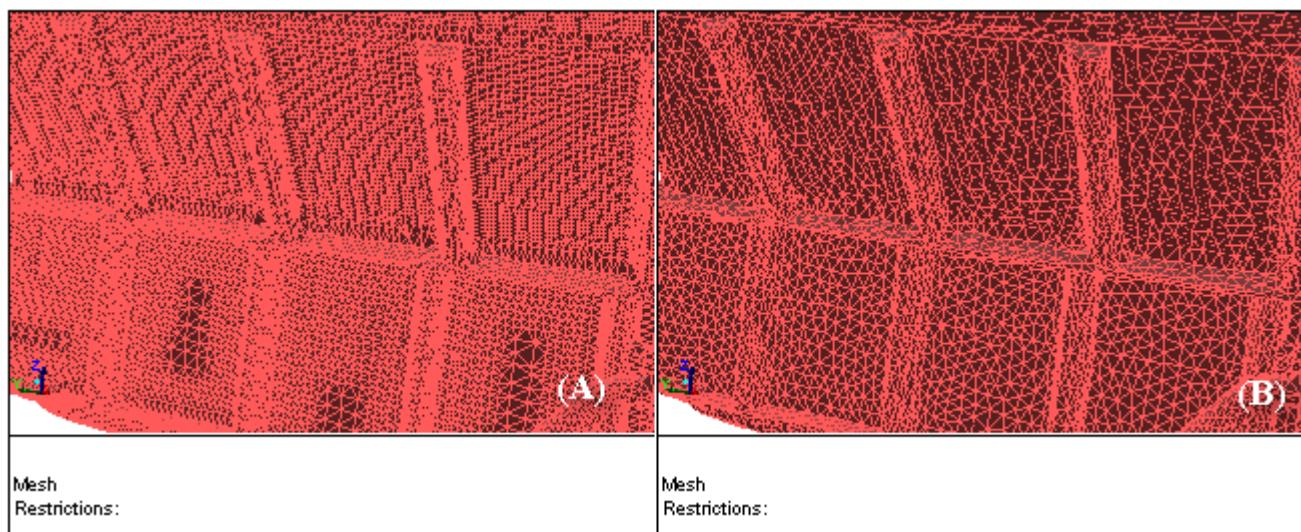
**Figure 8.26: Wrapper Surface and Contours of Distance During Imprinting**

## 8.11. Post Wrapping Operations

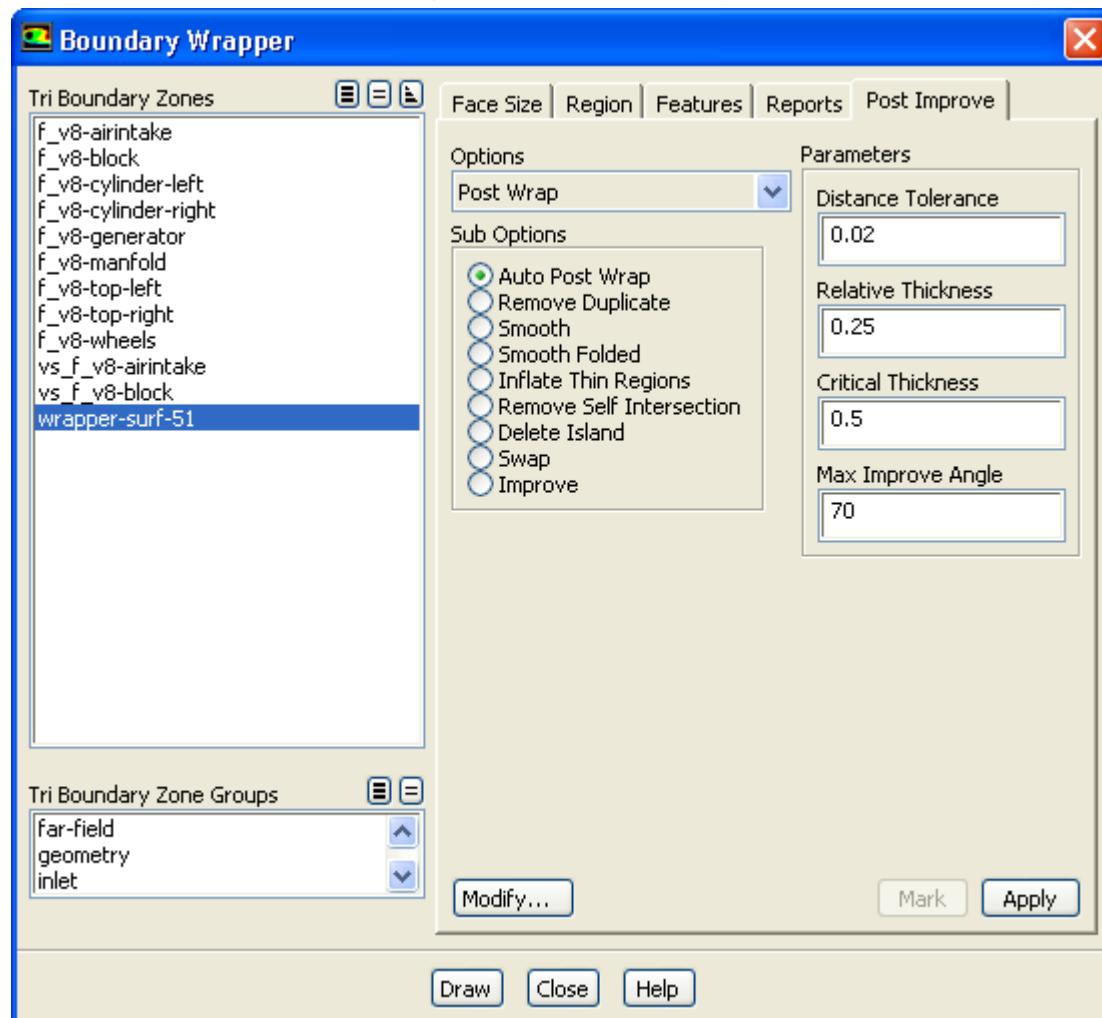
1. Click the **Post Improve** tab of the **Boundary Wrapper** dialog box.
2. Coarsen the wrapper surface.



- a. Retain the selection of **Coarsen** in the **Options** drop-down list.
- b. Retain the value of 2 for **Edge Length Change** and enter 10 for **Max Angle Change**, respectively.
- c. Retain the value of 0 for **Min Length** and enter 5 for **Max Length**.
- d. Click **Apply**.

**Figure 8.27: Wrapper Surface (A) Before and (B) After Coarsening**

- e. Save the mesh (engine-01.msh.gz).
  
- 3. Use the automated post wrapping option to improve the wrapper surface.



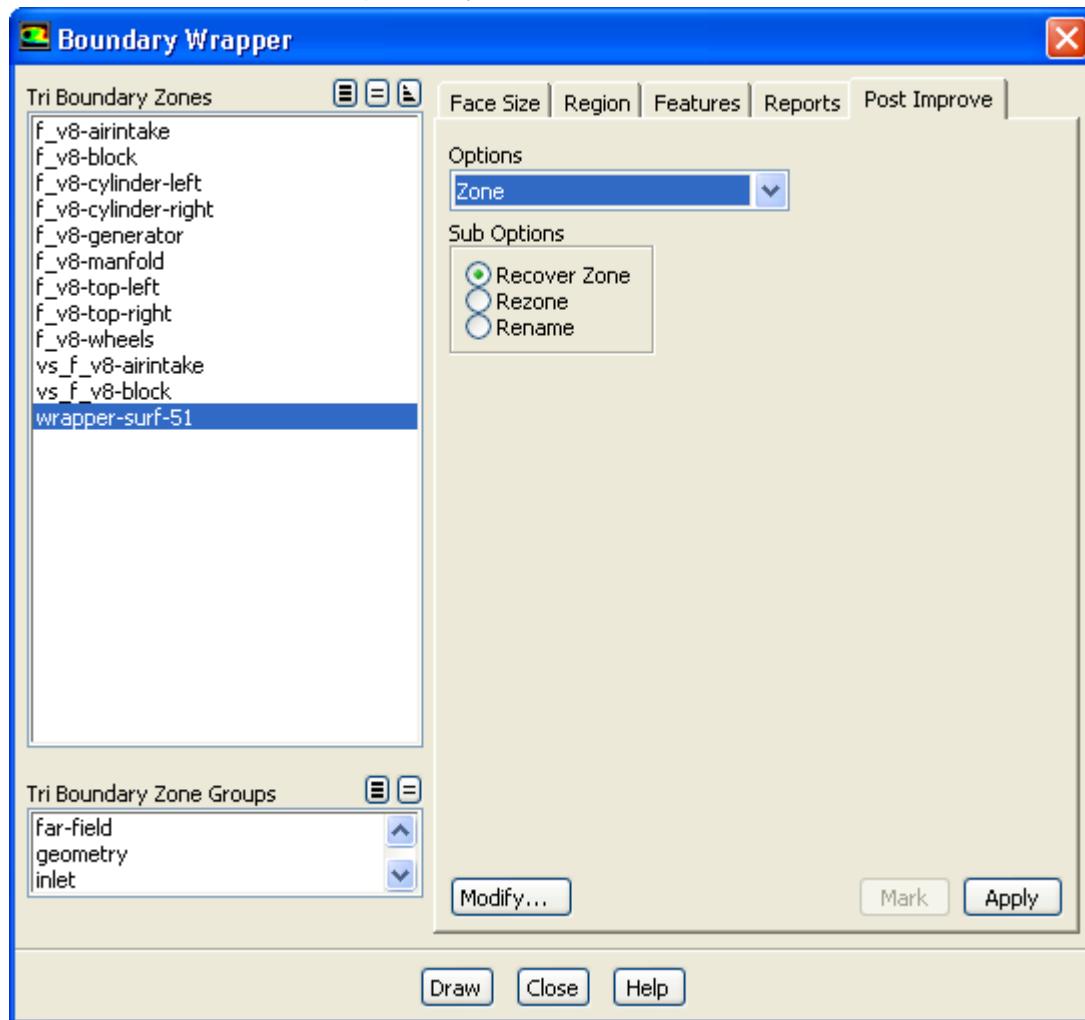
- a. Retain the selection of the wrapper surface in the **Tri Boundary Zones** selection list.
- b. Select **Post Wrap** in the **Options** drop-down list and **Auto Post Wrap** in the **Sub Options** list.
- c. Enter 0.5 for **Critical Thickness**.

The value specified is 20% of the minimum size (2.5 mm).

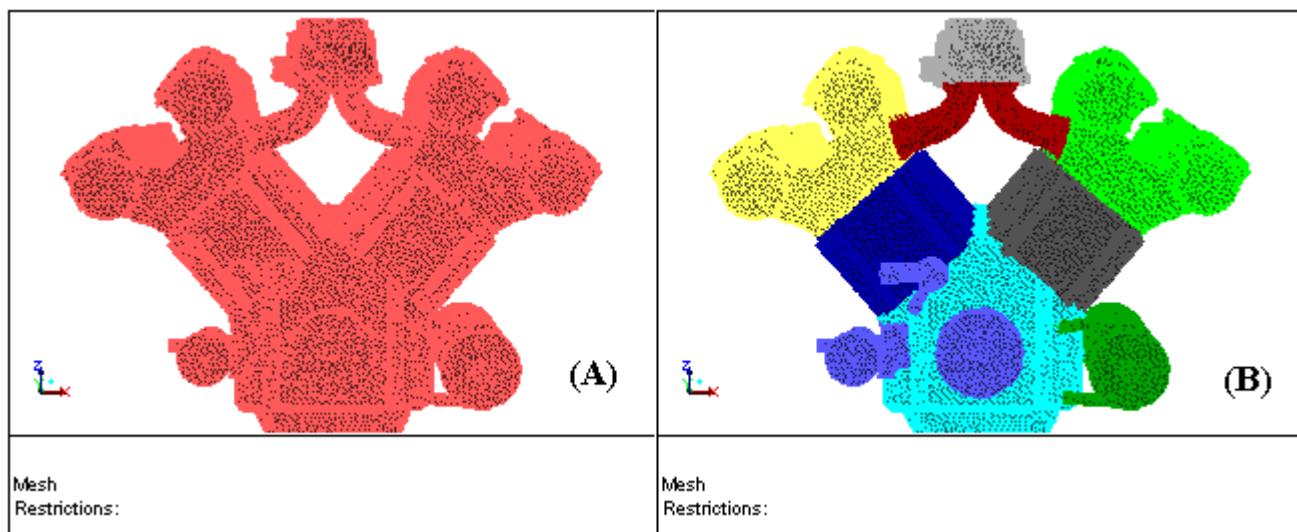
- d. Retain the default settings for the remaining parameters and click **Apply**.

The **Auto Post Wrap** option will perform all the remaining post wrapping operations in an optimal order to provide a valid surface mesh of as good quality as possible, without destroying any features.

4. Save the mesh (engine-02.msh.gz).
5. Extract zones based on the geometry.

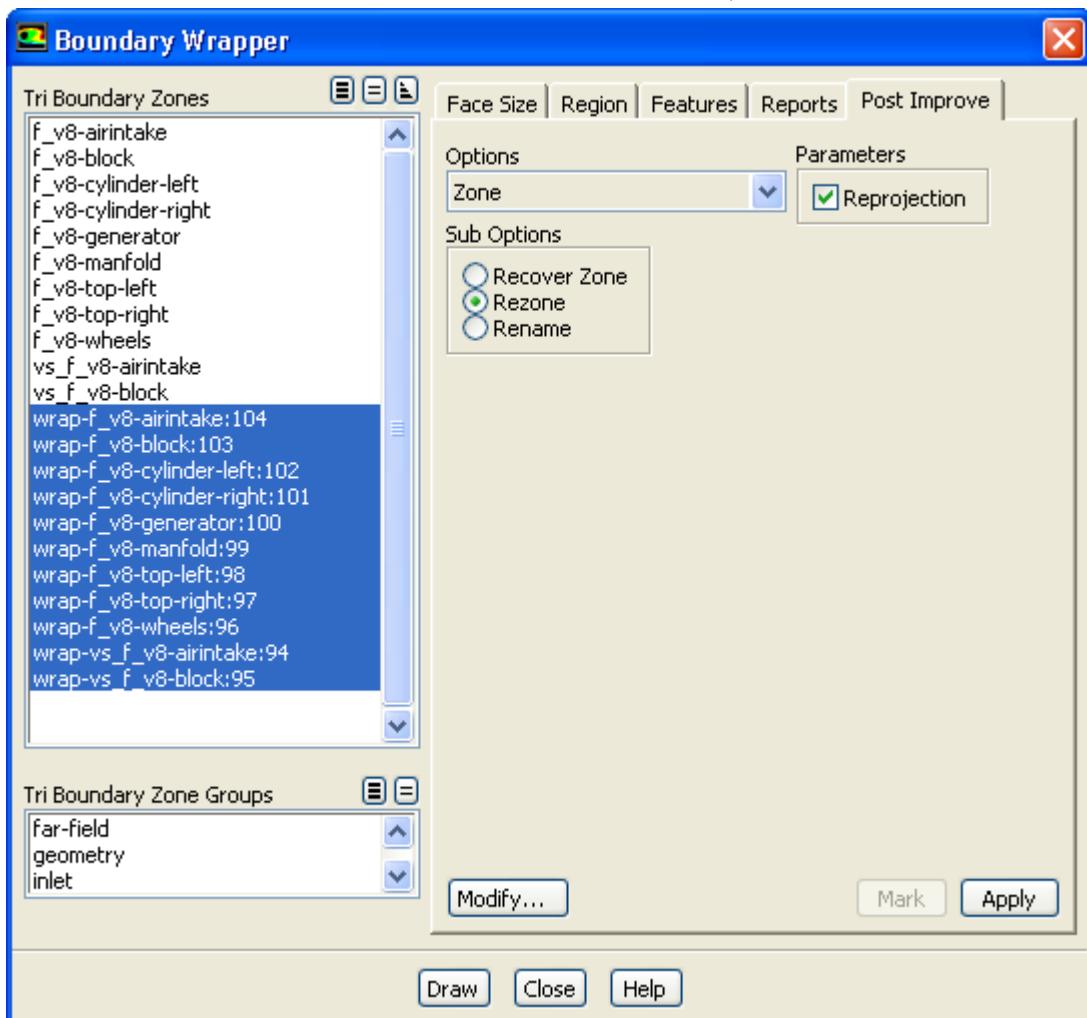


- a. Retain the selection of the wrapper surface in the **Tri Boundary Zones** selection list.
- b. Select **Zone** in the **Options** drop-down list and **Recover Zone** in the **Sub Options** list.
- c. Click **Apply** (Figure 8.28: Wrapper Surface (A) Before and (B) After Recovering Zones (p. 183)).

**Figure 8.28: Wrapper Surface (A) Before and (B) After Recovering Zones**

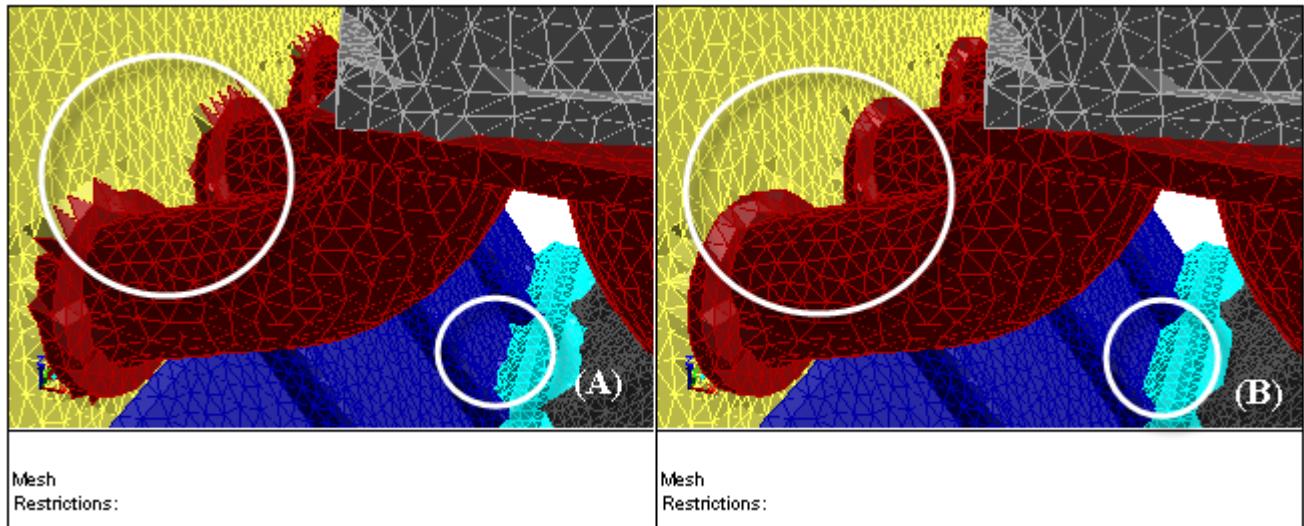
The wrapper surface will be separated into zones based on the zones in the original geometry.  
The extracted wrapper zones will be prefixed by **wrap-**.

- Select the extracted wrapper surfaces in the **Tri Boundary Zones** selection list.



- e. Retain the selection of **Zone** in the **Options** drop-down list and select **Rezone** in the **Sub Options** list.
- f. Retain the **Reprojection** option in the **Parameters** group box.
- g. Click **Apply** (Figure 8.29: Wrapper Surface (A) Before and (B) After Rezoning (p. 184)).

**Figure 8.29: Wrapper Surface (A) Before and (B) After Rezoning**



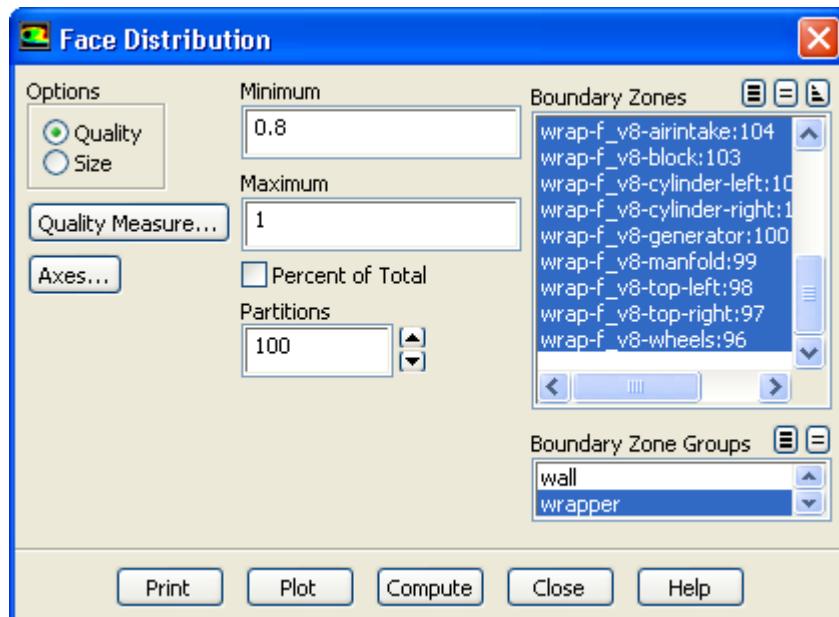
- h. Save the mesh (engine-03.msh.gz).
6. Merge the small area face zones with the neighboring zones using TUI commands:

```
>/boundary/merge-small-face-zones
Minimum area [0.01] 500
```

```
Merged wrap-vs_f_v8-block:# with wrap-f_v8-block:# 
Merged wrap-vs_f_v8-airintake:# with wrap-f_v8-airintake:# 
Merged 2 zones
```

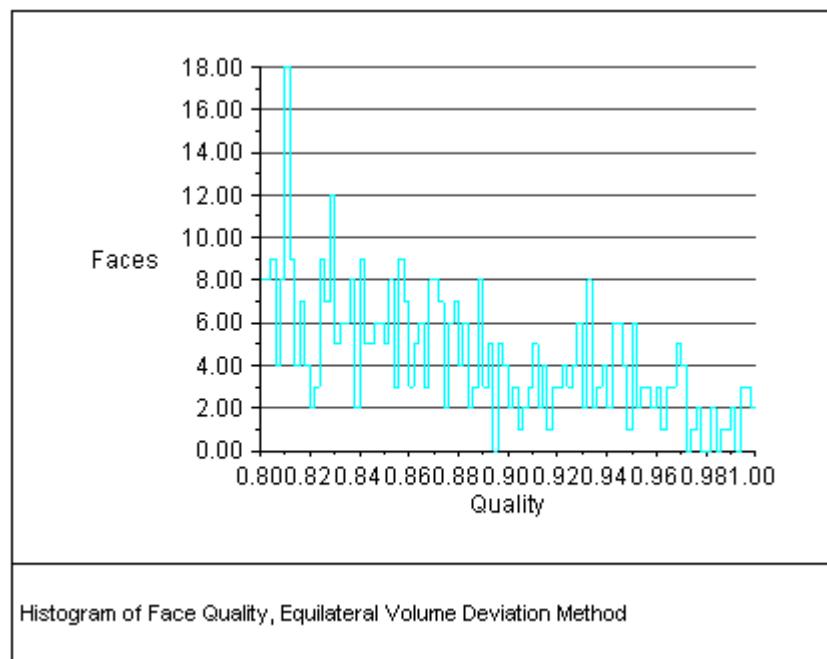
7. Plot the face skewness distribution in the range 0.8 to 1.0.

**Display → Plot → Face Distribution...**

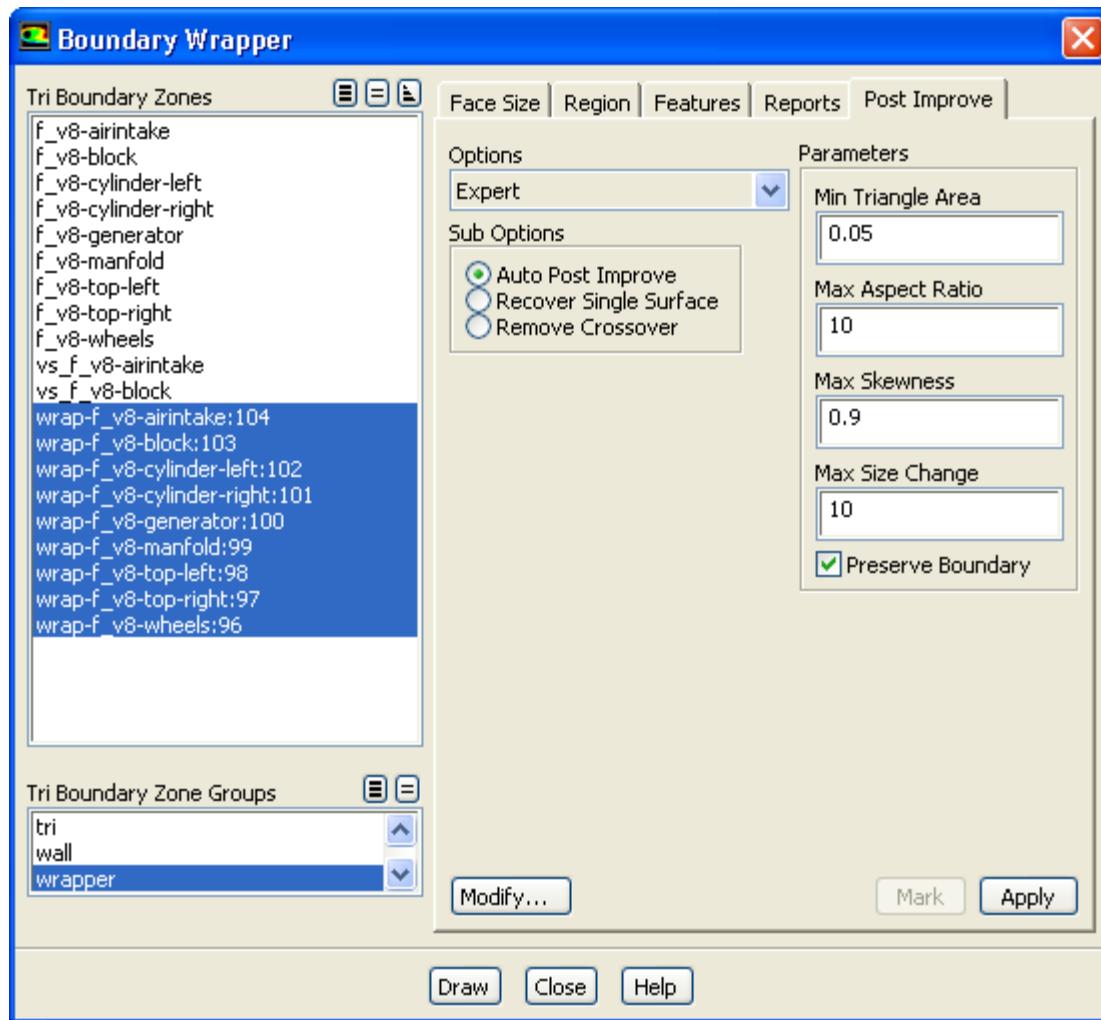


- Select **wrapper** in the **Boundary Zone Groups** selection list to select all the wrapper zones in the **Boundary Zones** selection list.
- Enter 0.8 for **Minimum**.
- Click **Plot** (Figure 8.30: Face Skewness Distribution Between 0.8 and 1.0 (Before Auto Post Improve) (p. 185)).

**Figure 8.30: Face Skewness Distribution Between 0.8 and 1.0 (Before Auto Post Improve)**



- Close the **Face Distribution** dialog box.
- Improve the wrapper surfaces using the **Auto Post Improve** option.

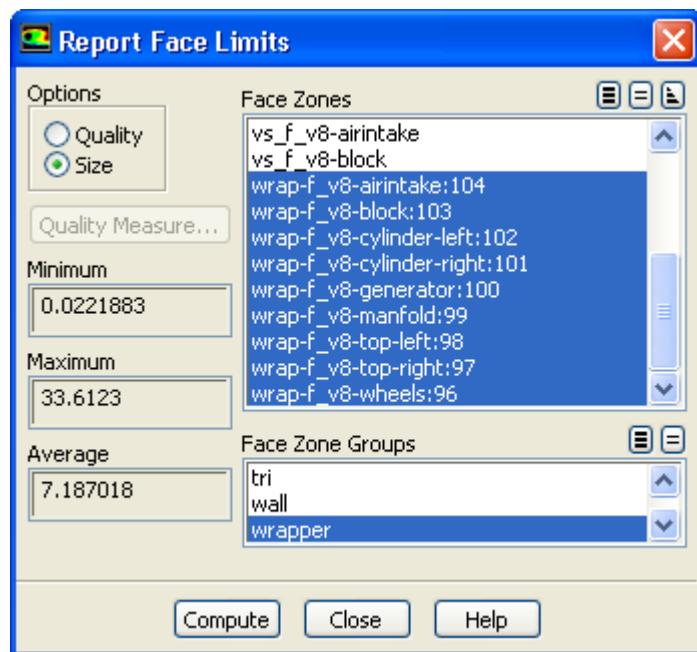


- Select **Expert** in the **Options** drop-down list and **Auto Post Improve** in the **Sub Options** list.
- Enter **0 . 05** for **Min Triangle Area**.

#### Note

All faces having area smaller than the specified value will be removed. Hence, it is suggested to first find the smallest triangle area before specifying the value.

**Report → Face Limits...**



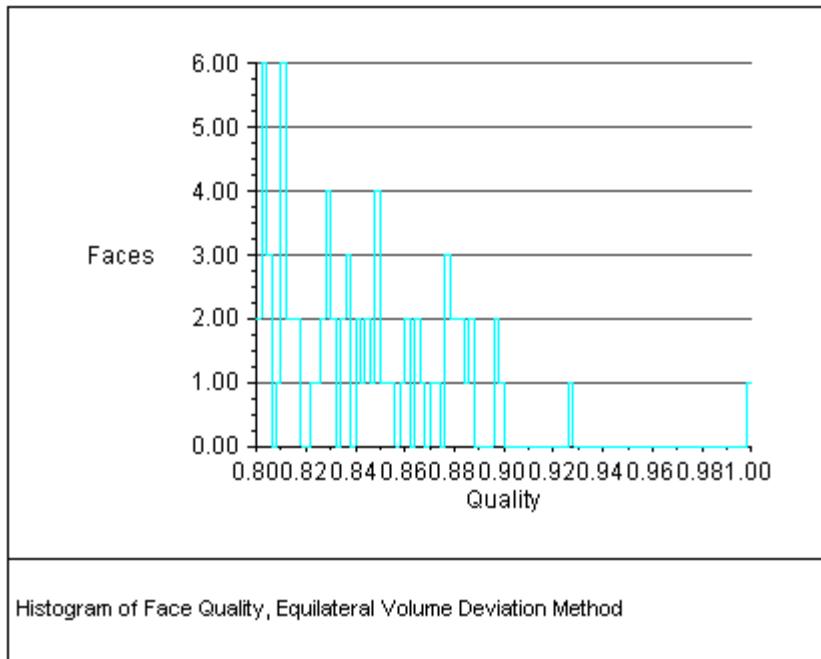
1. Select **wrapper** in the **Face Zone Groups** list to select all the wrapper zones.
2. Select **Size** in the **Options** list.
3. Click **Compute**.
4. Close the **Report Face Limits** dialog box.

The value specified for **Min Triangle Area** can be decided accordingly.

- 
- c. Enter 0.9 for **Max Skewness** and 10 for **Max Size Change**, respectively.
  - d. Click **Apply**.
  - e. Plot the face skewness distribution in the range 0.8 to 1.0 ([Figure 8.31: Face Skewness Distribution Between 0.8 and 1.0 \(After First Auto Post Improve\) \(p. 188\)](#)).

**Display → Plot → Face Distribution...**

**Figure 8.31: Face Skewness Distribution Between 0.8 and 1.0 (After First Auto Post Improve)**



In Figure 8.31: Face Skewness Distribution Between 0.8 and 1.0 (After First Auto Post Improve) (p. 188), you can see that there are two faces above 0.9.

- f. Enter 0.1 for **Min Triangle Area** and 0.8 for **Max Skewness**, respectively.
- g. Click **Apply**.

The worst quality is reported to be 0.8.

9. Delete island faces on the wrapper zones using TUI commands:

```
>/boundary/delete-island-faces
()
Boundary Face Zones(1) [()] wrap-*
Boundary Face Zones(2) [()]
Critical Absolute Face Count [15]
Critical Relative Face Count [1]
... collecting linked face zones
9 face zones were collected
... extracting separated face groups
3 groups were extracted.
... processing small face groups
face count of the largest group: 117685
relative island count: 1.000000
critical counts for groups to delete: 15
0 faces were deleted
```

10. Save the mesh (engine-04.msh.gz).

11. Delete the original geometry.

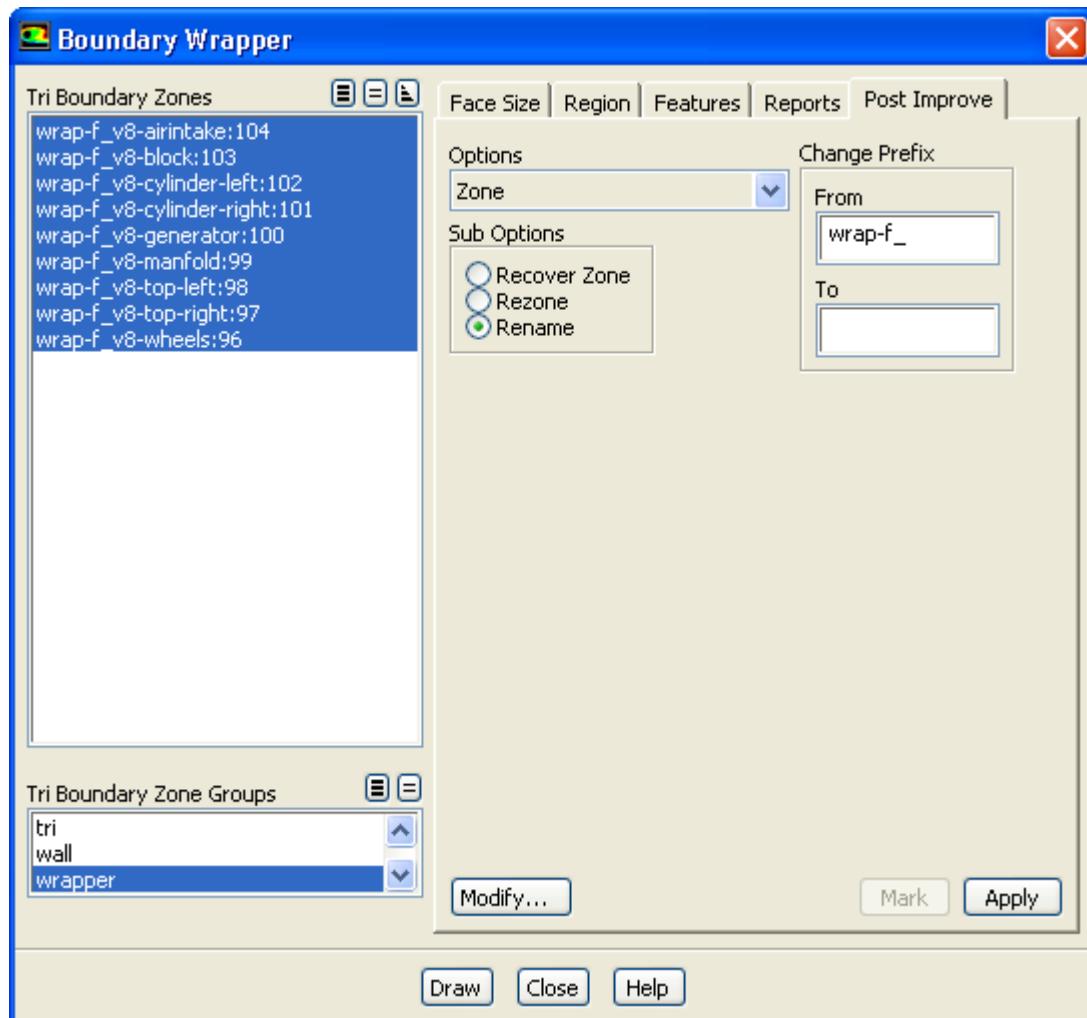
**Boundary → Manage...**

- a. Select the original geometry and patched holes in the **Face Zones** selection list.

- b. Select **Delete** in the **Options** list and click **Apply**.

A **Question** dialog box will appear, asking you to confirm if you want to delete the selected zones.

- c. Click **Yes** in the **Question** dialog box.  
d. Close the **Manage Face Zones** dialog box.
12. Rename the wrapper surfaces.

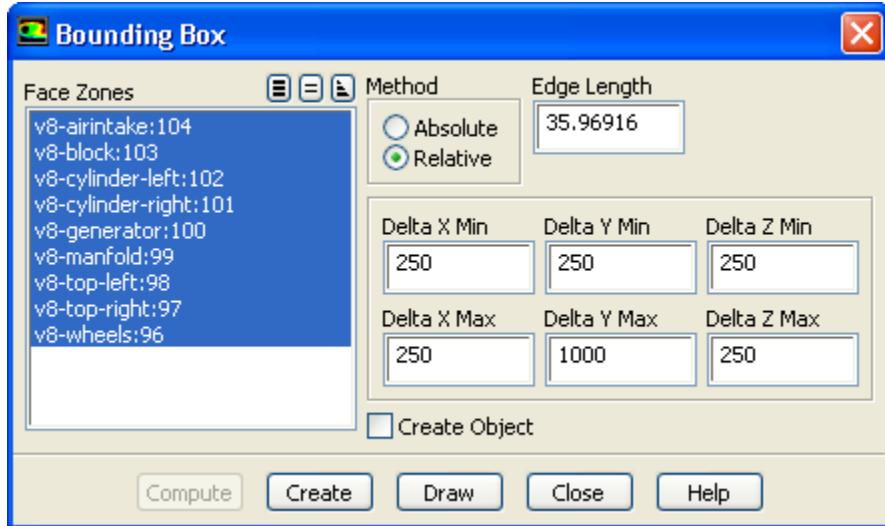


- a. Select the wrapper zones in the **Tri Boundary Zones** selection list in the **Boundary Wrapper** dialog box.
- b. Select **Zone** from the **Options** drop-down list and **Rename** in the **Sub Options** list.
- c. Enter `wrap-f_` in the **From** field.  
Leave the **To** field blank so that the prefix will be removed.
- d. Click **Apply**.

13. Close the **Boundary Wrapper** dialog box.

## 8.12. Create the Tunnel

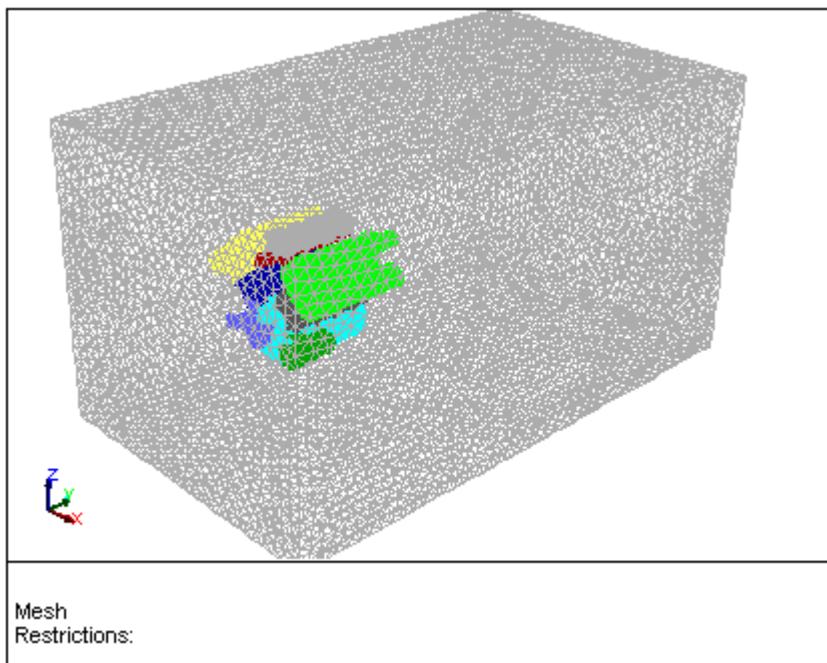
**Boundary → Create → Bounding Box...**



1. Select all the surfaces in the **Face Zones** selection list and click **Compute**.
2. Select **Relative** in the **Method** list.
3. Enter the values for the extents of the bounding box as shown in the **Bounding Box** dialog box.
4. Click **Create**.

The **Zone Name** dialog box will open.

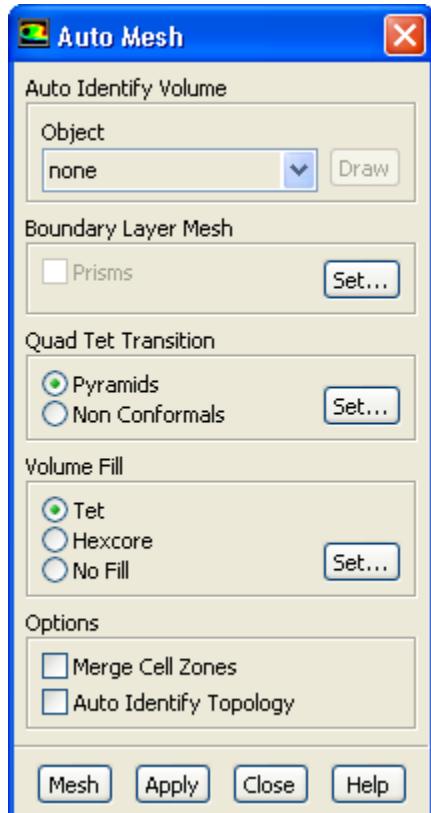
5. Enter tunnel for **Zone Name** and click **OK**.
6. Close the **Bounding Box** dialog box.
7. Display the boundary mesh ([Figure 8.32: Mesh with the Tunnel \(p. 191\)](#)).

**Figure 8.32: Mesh with the Tunnel**

Disable **Filled** in the **Attributes** tab of the **Display Grid** dialog box to obtain the display shown in Figure 8.32: Mesh with the Tunnel (p. 191).

## 8.13. Generate the Volume Mesh

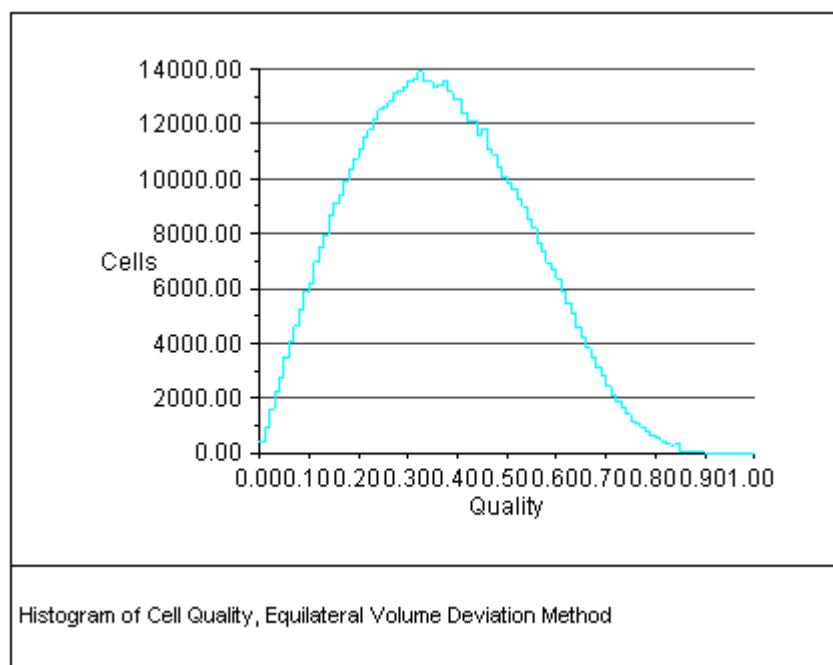
**Mesh → Auto Mesh...**



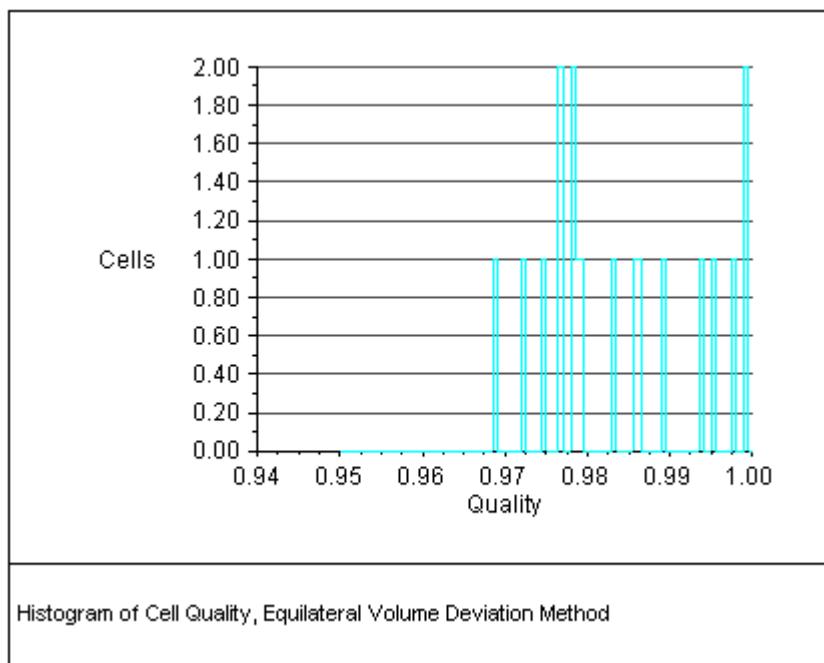
1. Retain the selection of **Tet** in the **Volume Fill** group box and click the **Set...** button to open the **Tet** dialog box.
  - a. Enable **Delete Dead Zones** in the **Tet Zones** group box.
  - b. Click **Apply** and close the **Tet** dialog box.
2. Click **Mesh** in the **Auto Mesh** dialog box.
3. Close the **Auto Mesh** dialog box.
4. Check the quality of the volume mesh ([Figure 8.33: Cell Quality Distribution \(p. 192\)](#)).

**Display → Plot → Cell Distribution...**

**Figure 8.33: Cell Quality Distribution**



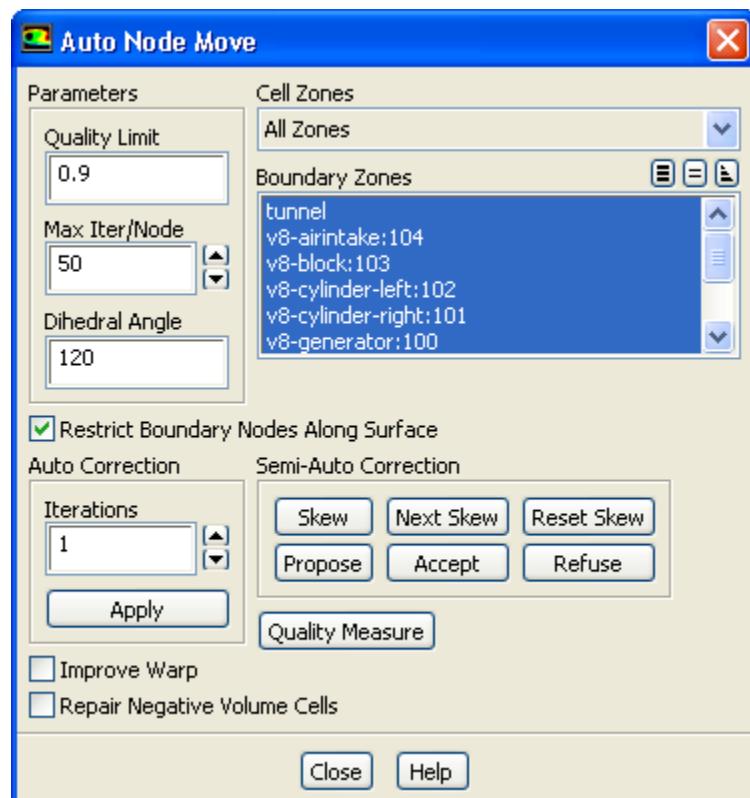
5. Plot the skewness distribution above 0.95 ([Figure 8.34: Cell Quality Distribution Above 0.95 \(p. 193\)](#)).

**Figure 8.34: Cell Quality Distribution Above 0.95**

## 8.14. Improve the Volume Mesh

This section demonstrates the improvement of the volume mesh such that the maximum skewness is around 0.95.

**Mesh → Tools → Auto Node Move...**



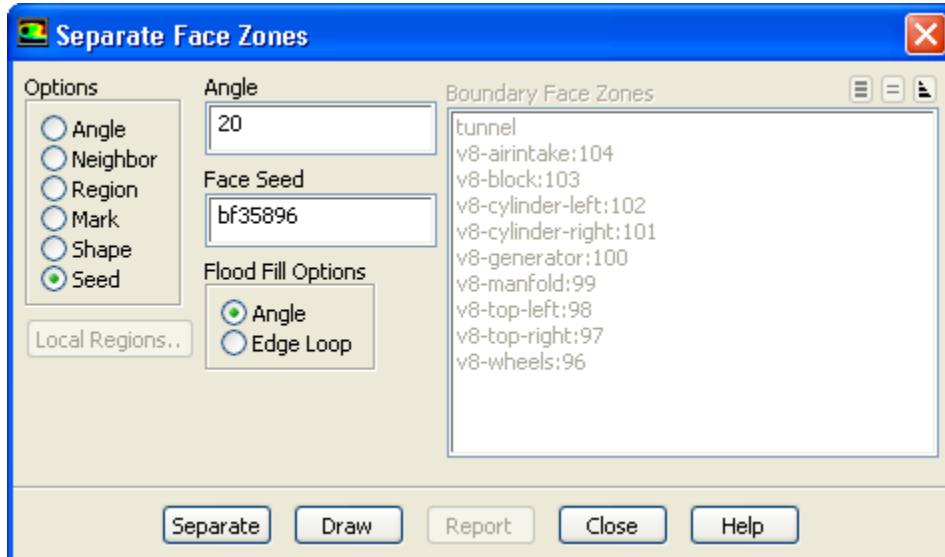
1. Select all the zones in the **Boundary Zones** selection list.
2. Click **Apply** in the **Auto Correction** group box.
3. Check the skewness distribution above 0.95.
4. Enter 0.95 for **Quality Limit** and 0 for **Dihedral Angle**, respectively.
5. Disable **Restrict Boundary Nodes Along Surface**.
6. Click **Apply** in the **Auto Correction** group box.

The maximum skewness reported is around 0.95, which is acceptable.

7. Close the **Auto Node Move** dialog box.

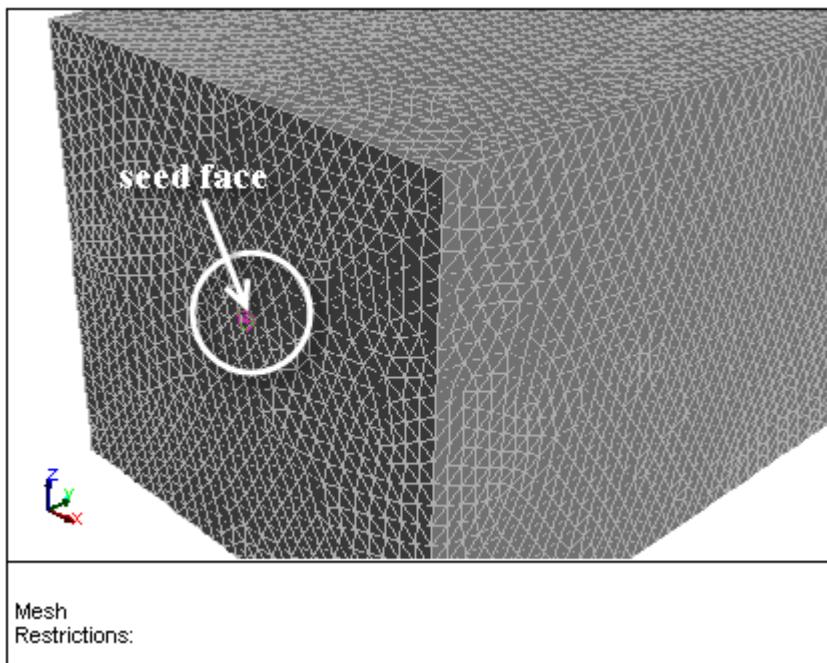
## 8.15. Separate the Tunnel Inlet and Outlet

**Boundary** → **Zone** → **Separate...**



1. Select **Seed** in the **Options** list.
2. Enter 20 for **Angle**.
3. Select the seed face as shown in Figure 8.35: Seed Face for Separating the Tunnel Inlet (p. 195).

**Figure 8.35: Seed Face for Separating the Tunnel Inlet**



4. Click **Separate**.

The zone **tunnel-#** will be created, where # is the zone ID.

5. Rename **tunnel-#** to **inlet**.

**Boundary → Manage...**

- a. Select **tunnel-#** in the **Face Zones** selection list.

- b. Select **Rename** in the **Options** list.

- c. Enter **inlet** for **Name** and click **Apply**.

6. Similarly, separate the tunnel outlet and rename it to **outlet**.

7. Save the mesh (`engine-final.msh.gz`).

8. Exit ANSYS FLUENT.

## 8.16. Summary

This tutorial demonstrated the wrapping procedure for a V-8 engine mesh. You initially performed pre-wrapping operations to close large holes in the geometry. You then initialized the wrapper, examined the region to be wrapped and updated the region to account for the fixed leakage. The tutorial also demonstrated the use of the automatic hole fixing functionality to close smaller holes detected when refining the Cartesian grid using local size functions. After wrapping the main region, and imprinting necessary features of the geometry, you performed post-wrapping operations to improve the wrapper surface quality. You then created a tunnel encompassing the geometry and generated the volume mesh. The tutorial also described the procedure for using the **Auto Node Move** functionality to improve the quality of the volume mesh.



---

## Chapter 9: CutCell Mesh Generation

---

CutCell meshing is a general purpose meshing technique which is suitable for a large range of applications. Due to the large fraction of hex cells in the generated mesh, the CutCell mesh can often produce better results than regular tetrahedral meshes. This meshing method uses a direct surface and volume approach without needing cleanup or decomposition, thereby reducing the turnaround time required for meshing. This tutorial demonstrates the generation of the CutCell mesh for a manifold.

This tutorial demonstrates how to do the following:

- Import the CAD geometry.
- Examine the objects.
- Create capping surfaces for the inlet and outlets.
- Define size functions.
- Create the CutCell mesh.
- Set parameters and generate prisms for the CutCell mesh.
- Improve the CutCell mesh quality.

### 9.1. Prerequisites

This tutorial assumes that you have little experience with the meshing mode in ANSYS FLUENT, but that you are familiar with the graphical user interface.

### 9.2. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

To prepare for running this tutorial:

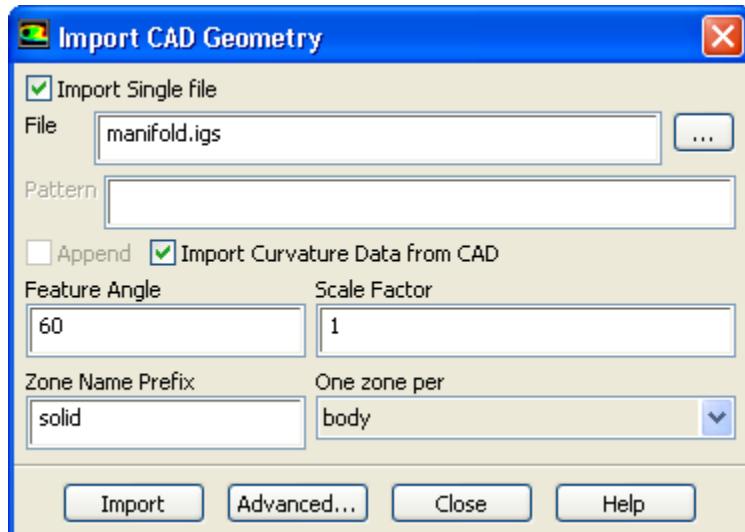
1. Download the tutorial input file `cutcell.zip` for the tutorial.
2. Unzip `cutcell.zip`.

The file `manifold.igs` can be found in the `cutcell` folder created on unzipping the file.

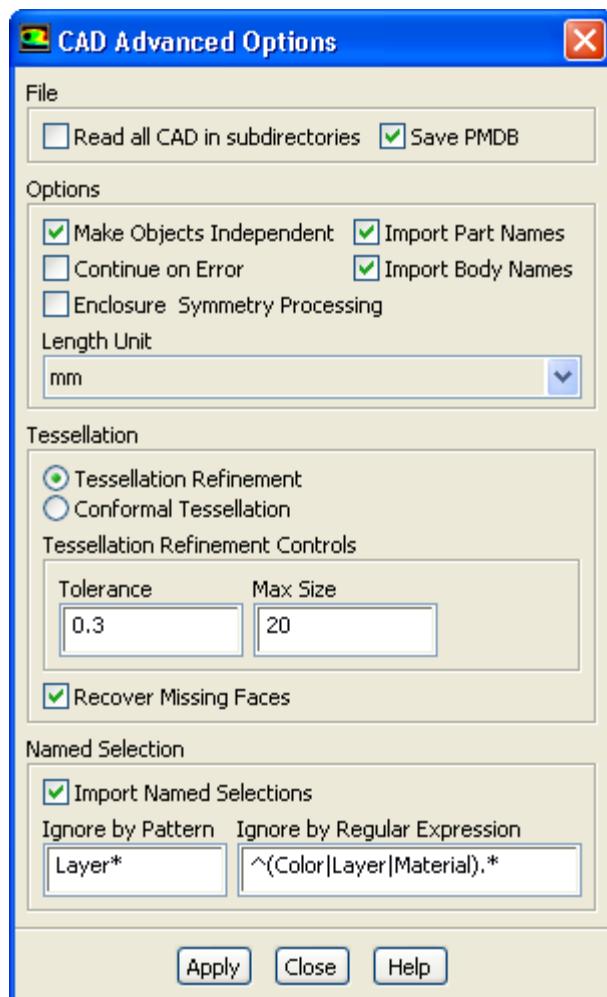
3. Start ANSYS FLUENT in meshing mode. For detailed steps, refer to [Starting ANSYS FLUENT in Meshing Mode \(p. 4\)](#).

### 9.3. Import the CAD Geometry

**File → Import → CAD...**



1. Ensure that **Import Single file** is enabled.
2. Click and select the file `manifold.igs`.
3. Enable **Import Curvature Data from CAD**.
4. Enter a value of 60 for **Feature Angle**.
5. Enter `solid` for **Zone Name Prefix**.
6. Retain the value of 1 for **Scale Factor** and the selection of **body** in the **One zone per** drop-down list, respectively.
7. Click the **Advanced...** button to open the **CAD Advanced Options** dialog box.

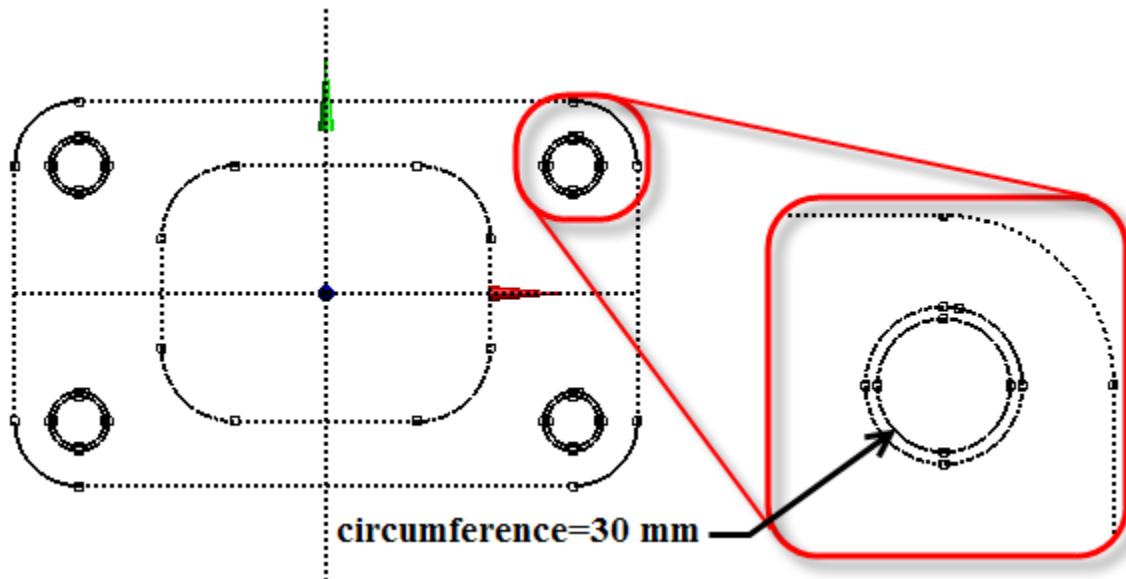


- Enable **Save PMDB** in the **File** group box.
- Select **mm** in the **Length Unit** drop-down list.
- Retain the selection of **Tessellation Refinement** in the **Tessellation** group box.
  - Specify values of 0 . 3 and 20 for **Tolerance** and **Max Size**, respectively.

---

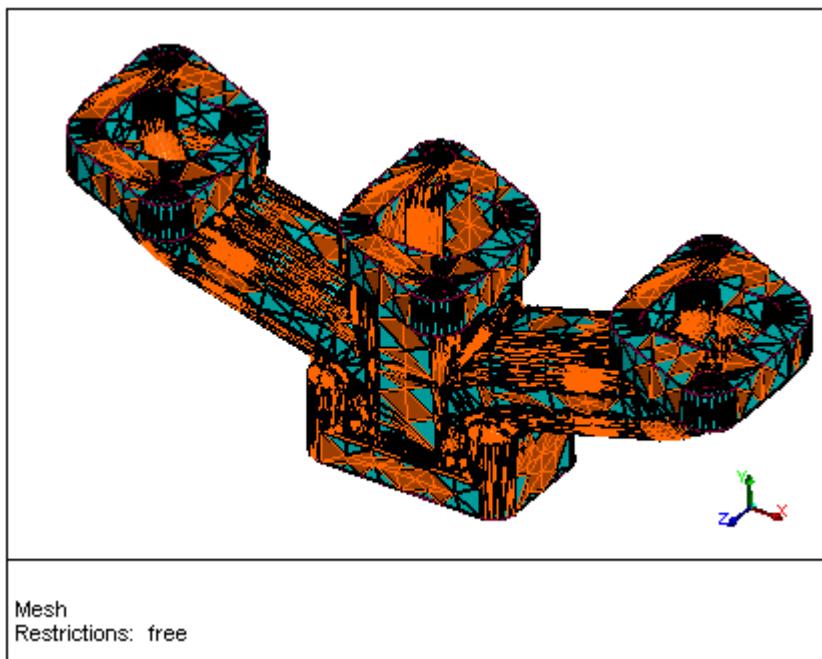
### Note

In this case, the smallest feature in the geometry is 30 mm in circumference (see [Figure 9.1: Determining the Minimum Size and Tessellation Tolerance \(p. 200\)](#)).

**Figure 9.1: Determining the Minimum Size and Tessellation Tolerance**

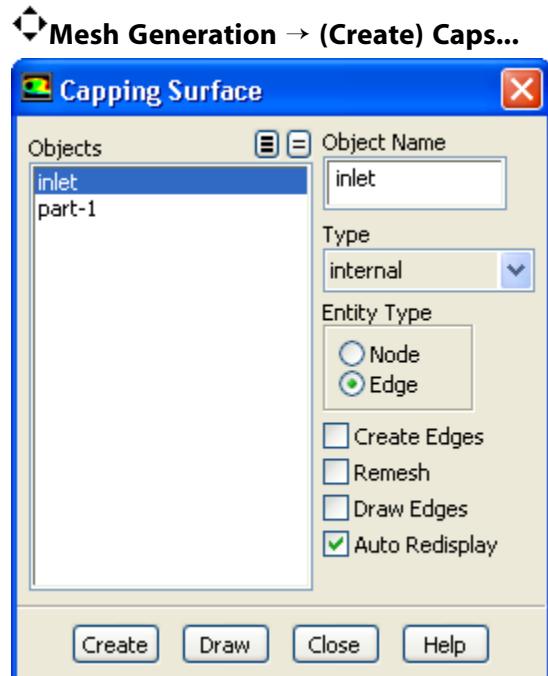
Hence the minimum size to be used would be approximately 3 mm. It is recommended that you use a **Tolerance** value 1/10th the intended minimum size (0.3).

- d. Retain the selection of **Import Named Selections** in the **Named Selection** group box.
  - e. Click **Apply** and close the **CAD Advanced Options** dialog box.
8. Click **Import** in the **Import CAD Geometry** dialog box.
9. Close the **Import CAD Geometry** dialog box.
10. Display the imported geometry object.
-  **Mesh Generation** → **(Objects) Manage...**
- a. Select **part-1** in the **Objects** selection list.
  - b. Enable **Free** in the **Face Options** group box in the **Mesh Generation** task page to display free nodes.
  - c. Click **Draw** in the **Manage Objects** dialog box (Figure 9.2: Geometry Object (p. 201)).

**Figure 9.2: Geometry Object**

- d. Close the **Manage Objects** dialog box.

## 9.4. Create Capping Surfaces for the Inlet and Outlets



1. Create the capping surface for the inlet.

- a. Enter **inlet** for **Object Name**.

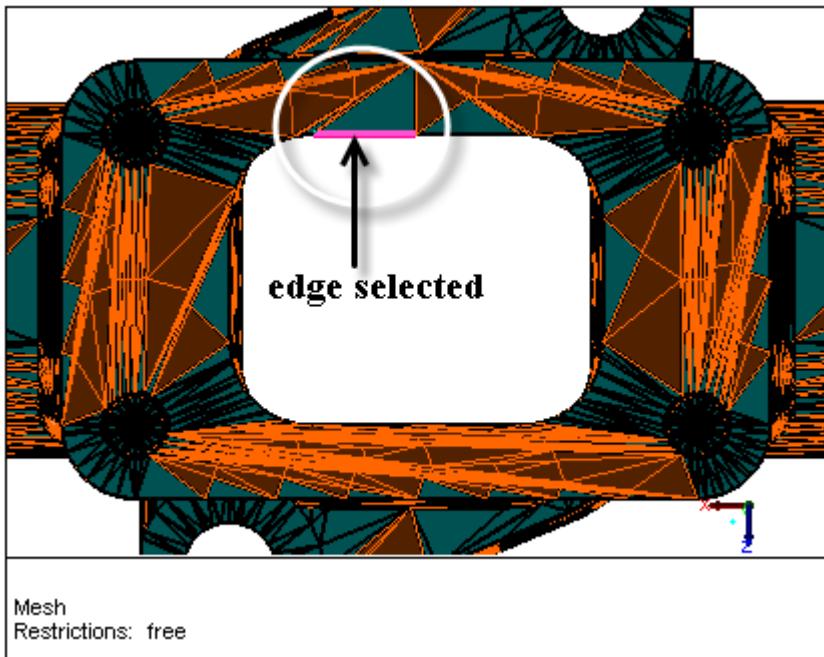
- b. Select **internal** in the **Type** drop-down list.

### Important

Though the capping surfaces to be created represent an inlet/outlet zone, they are essentially zero-thickness baffle surfaces. To allow the recovery of baffle surfaces by the CutCell mesher, the type must be set to **internal**, which will be recovered as a **wall**. You can then change the type of recovered boundary zone to the appropriate type (see [Post CutCell Meshing Cleanup Operations \(p. 215\)](#)).

- c. Select **Edge** in the **Entity Type** list.  
d. Select the edge shown in [Figure 9.3: Edge Selected for Capping the Inlet \(p. 202\)](#).

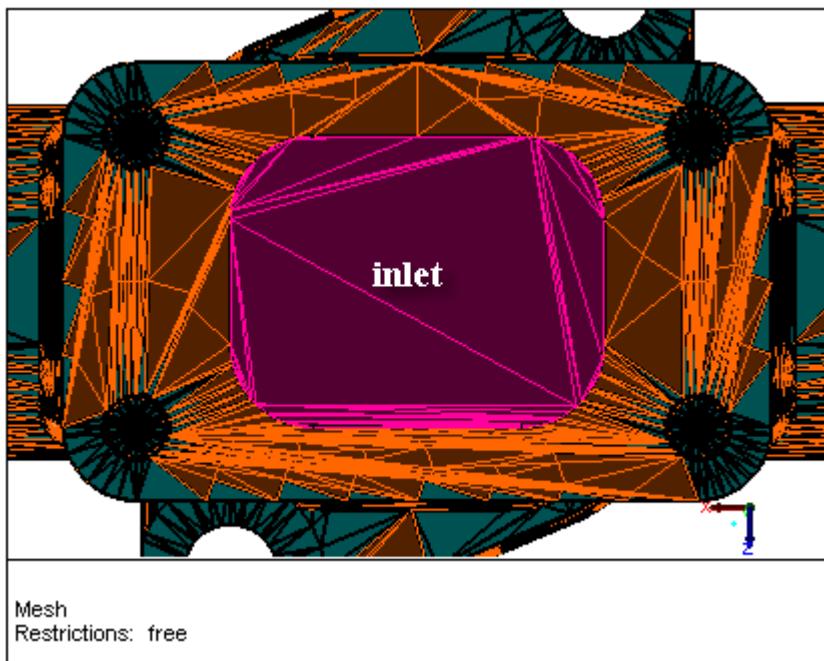
**Figure 9.3: Edge Selected for Capping the Inlet**



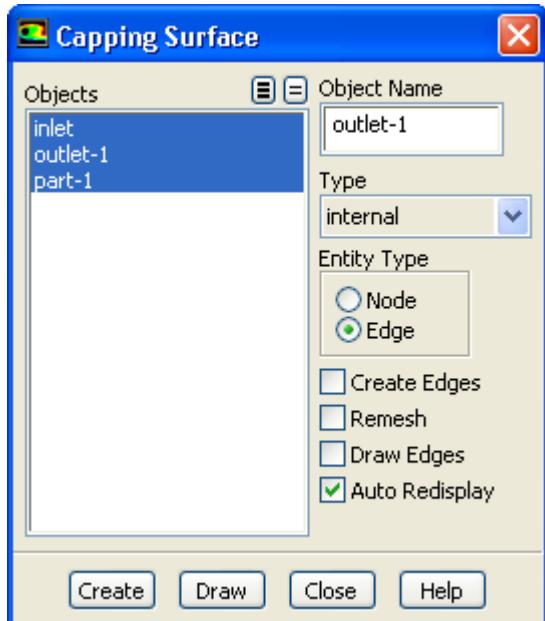
- e. Disable **Remesh** and enable **Auto Redisplay**.  
f. Click **Create**.

A wrap object (**inlet**) is created and added to the **Objects** selection list.

- g. Select **inlet** and **part-1** in the **Objects** list and click **Draw** ([Figure 9.4: Capping Surface Created for the Inlet \(p. 203\)](#)).

**Figure 9.4: Capping Surface Created for the Inlet**

2. Create the capping surface for the outlets.



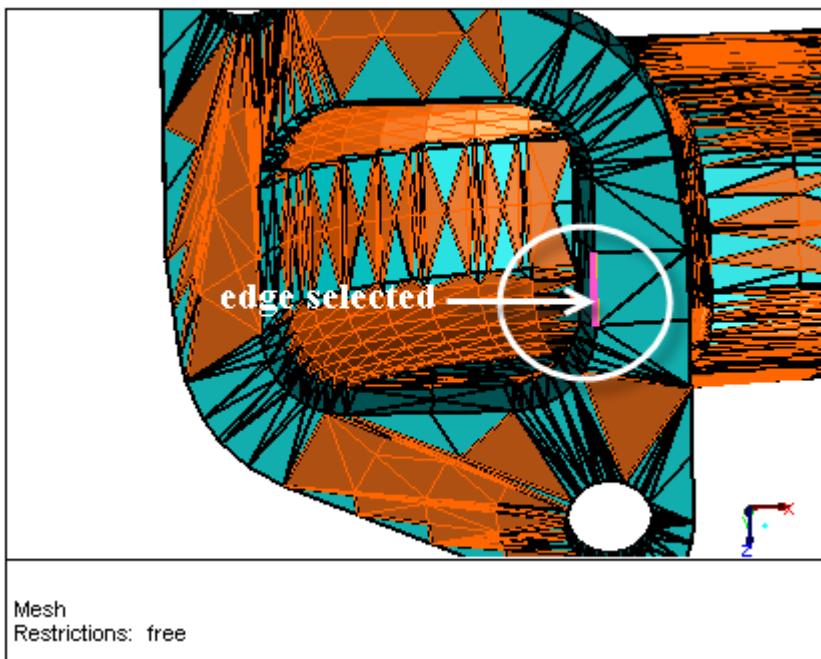
- a. Enter **outlet-1** for **Object Name**.
- b. Select **internal** in the **Type** drop-down list.

As described, to allow the recovery of baffle surfaces by the CutCell mesher, the type must be set to **internal**, which will be recovered as a **wall**. You can then change the type of recovered boundary zone to the appropriate type (see [Post CutCell Meshing Cleanup Operations \(p. 215\)](#)).

- c. Select **Edge** in the **Entity Type** list.

- d. Select the edge shown in [Figure 9.5: Edge Selected for Capping outlet-1 \(p. 204\)](#).

**Figure 9.5: Edge Selected for Capping outlet-1**

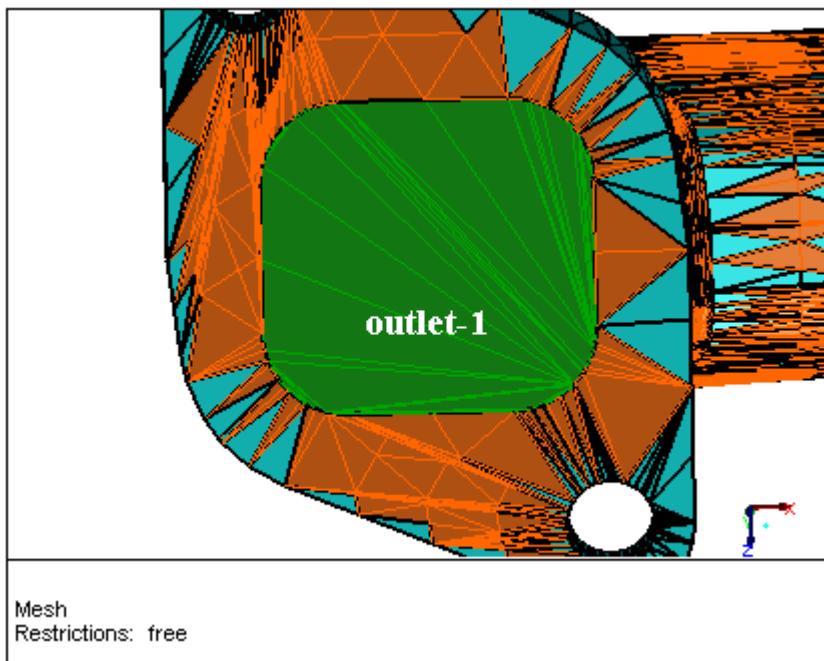


- e. Ensure that **Remesh** is disabled.

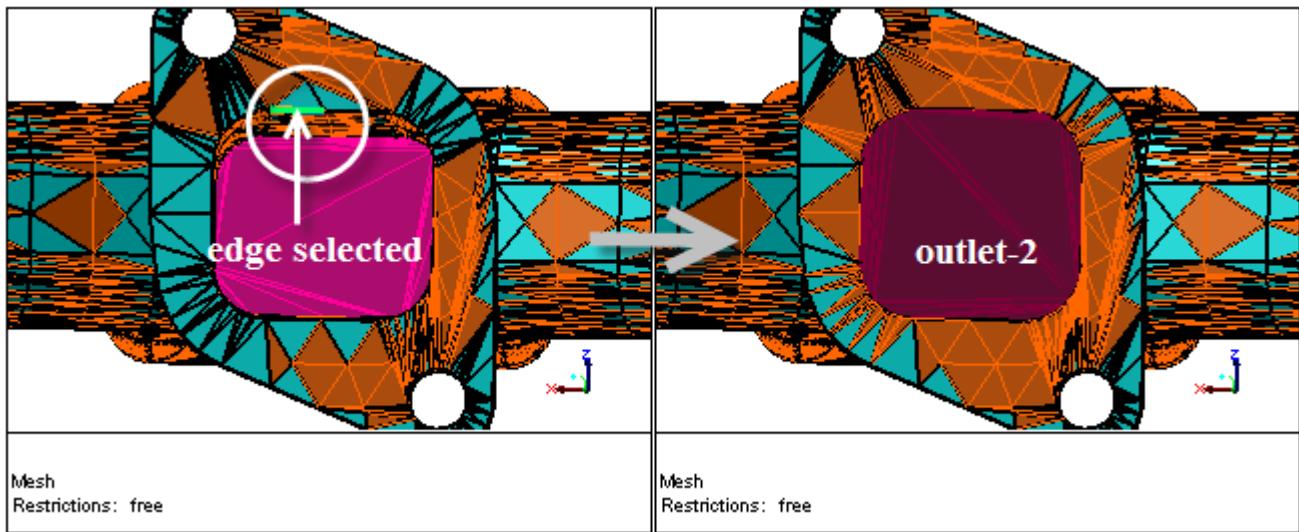
- f. Click **Create**.

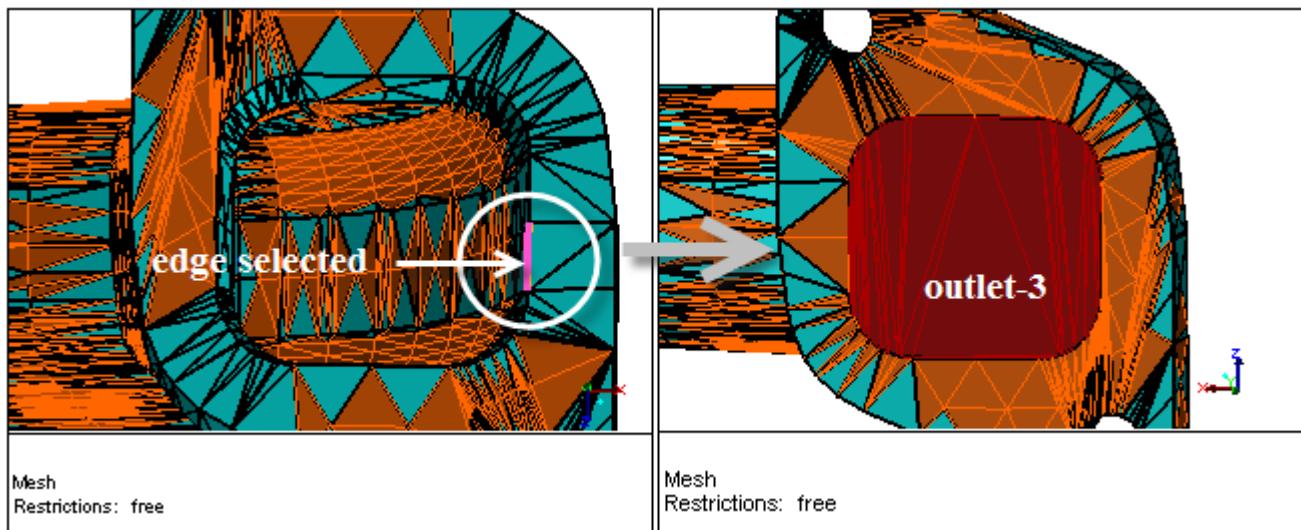
A wrap object (**outlet-1**) is created and added to the **Objects** selection list.

- g. Select **inlet**, **outlet-1** and **part-1** in the **Objects** list and click **Draw** ([Figure 9.6: Capping Surface Created for outlet-1 \(p. 205\)](#)).

**Figure 9.6: Capping Surface Created for outlet-1**

- h. Similarly, create the capping surfaces **outlet-2** (Figure 9.7: Capping Surface Created for outlet-2 (p. 205)) and **outlet-3** (Figure 9.8: Capping Surface Created for outlet-3 (p. 206)).

**Figure 9.7: Capping Surface Created for outlet-2**

**Figure 9.8: Capping Surface Created for outlet-3**

3. Close the **Capping Surface** dialog box.
4. Display the all the objects.
  - a. Select all the objects in the **Objects** selection list and click **Draw**.
  - b. Disable the **Free** option in the **Mesh Generation** task page.

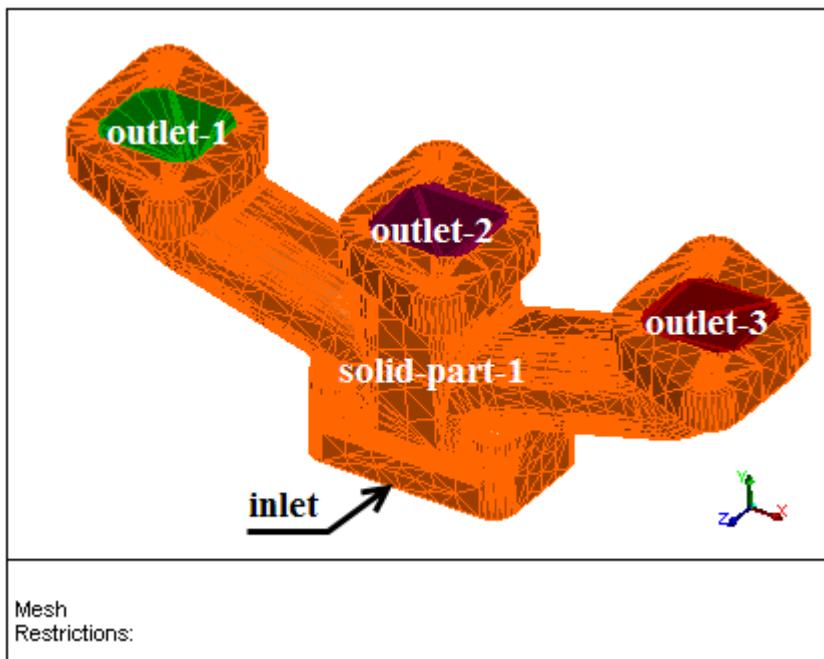
---

#### Note

Few free nodes still remain, however, it is not necessary to have a fully connected model as a prerequisite for generating the CutCell mesh.

---

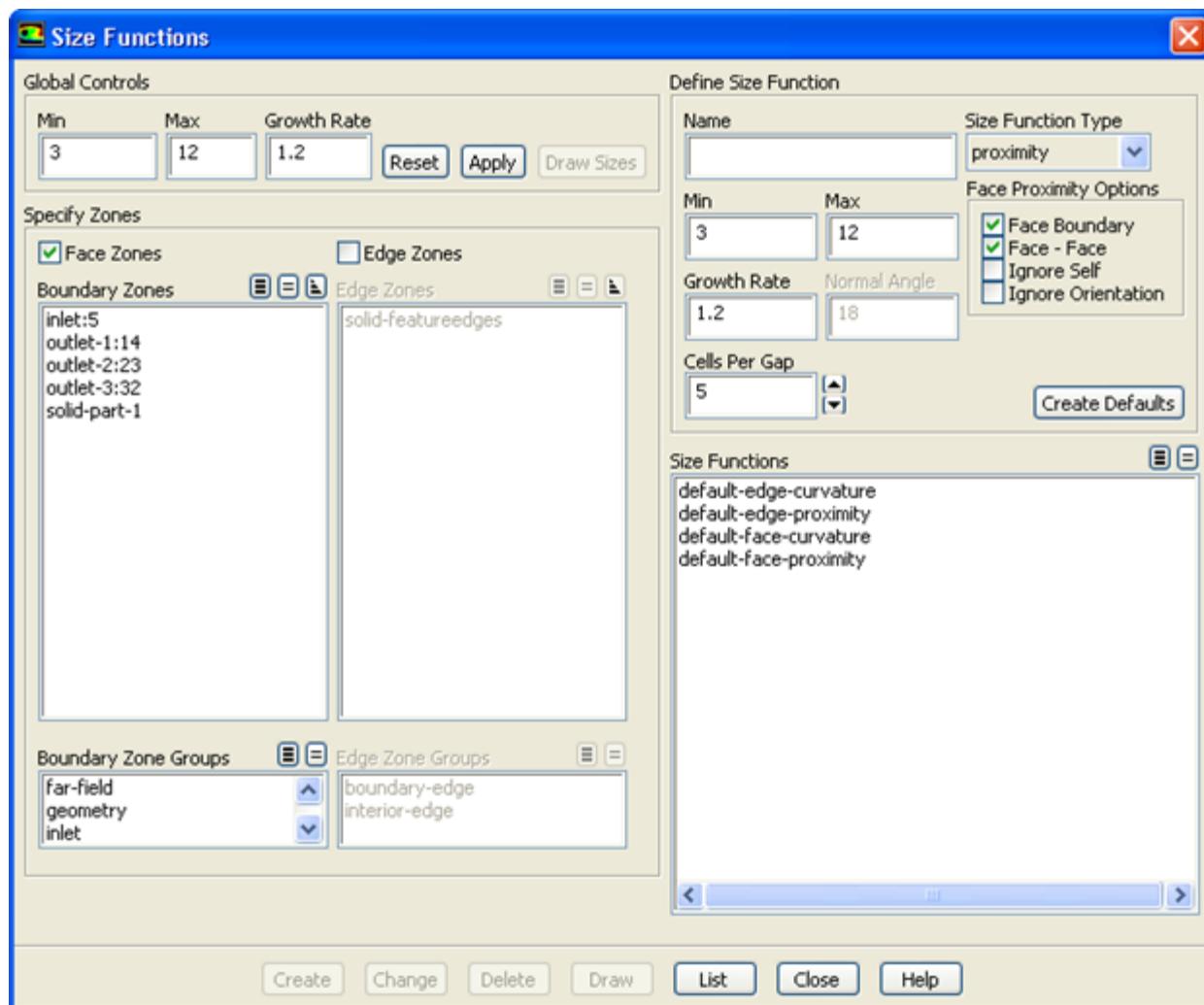
5. Click **Draw** again ([Figure 9.9: Geometry and Wrap Objects \(p. 207\)](#)).

**Figure 9.9: Geometry and Wrap Objects**

## 9.5. Set Up Size Functions

◆ **Mesh Generation → (Create) Size Functions...**

- Set up default size functions based on face and edge curvature and proximity.



- Enter values of 3 and 12 for **Min** and **Max**, respectively in the **Global Controls** group box.

#### Note

The minimum and maximum sizes should be in the ratio  $2^n$ . If not, the maximum size will be automatically set to a value close to but smaller than the specified maximum size, which is in the ratio of  $2^n$  with the minimum size specified.

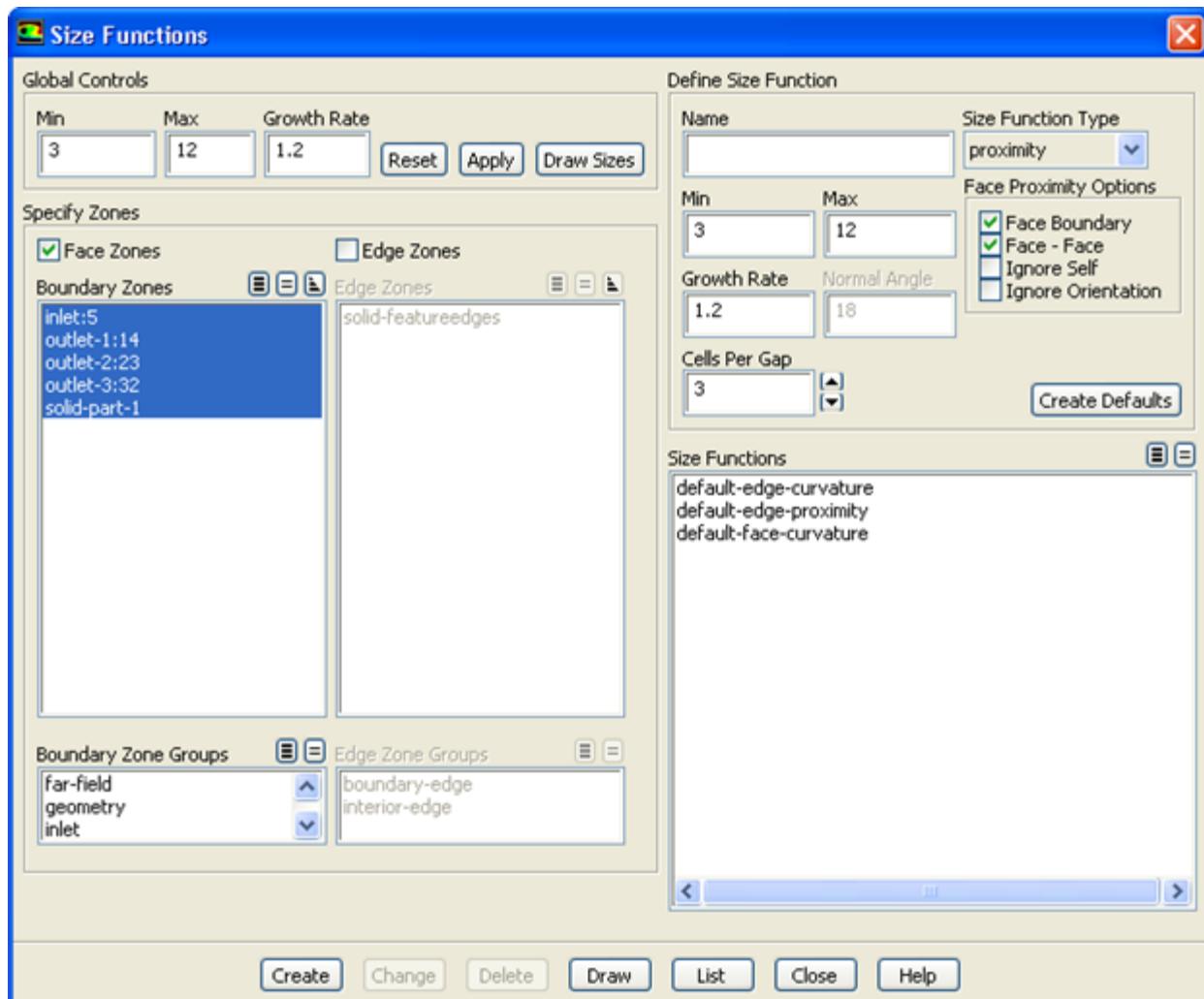
- Retain the value of 1.2 for **Growth Rate**.
- Click **Apply**.

You can use the **Draw Sizes** option to display red boxes of the specified global minimum and maximum sizes over the selected zones.

- Click **Create Defaults** in the **Define Size Function** group box.

The curvature and proximity size functions will be listed in the **Size Functions** selection list.

- Delete the face proximity size function.

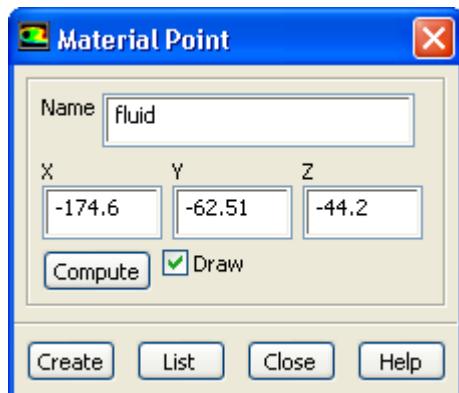


- i. Select **default-face-proximity** in the **Size Functions** selection list.
- ii. Click **Delete**.
- f. Close the **Size Functions** dialog box.

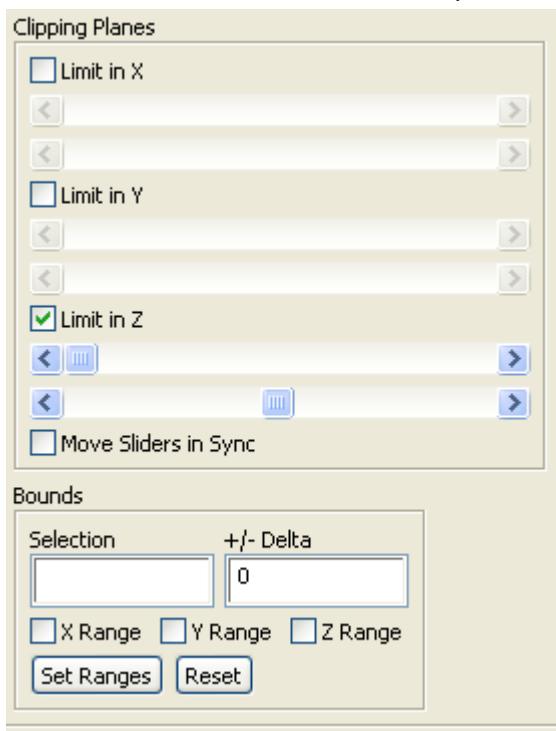
## 9.6. Generate the CutCell Mesh

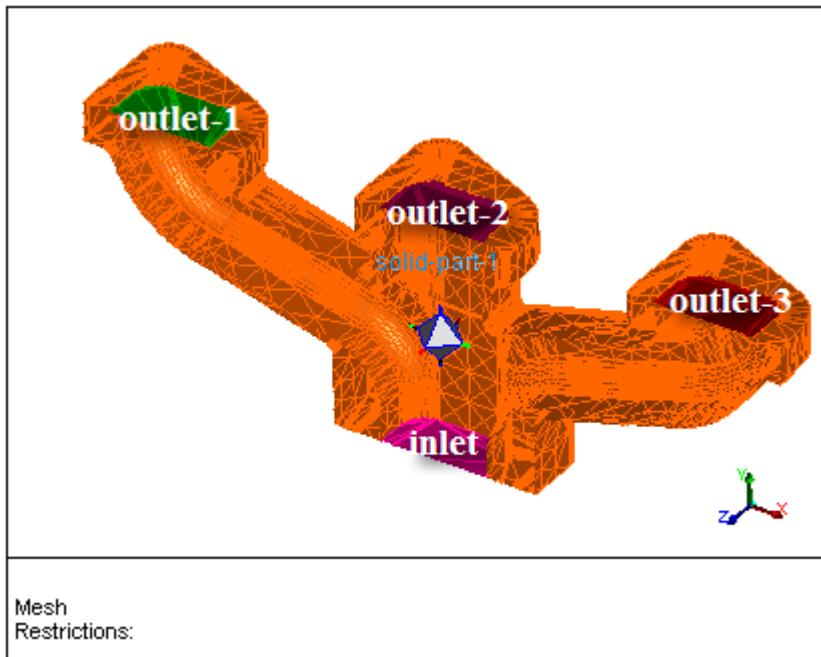
1. Define the material point for internal flow.

**Mesh Generation → (Create) Material Point...**



- Ensure that the selection filter is set to zone (**Ctrl+Z**).
- Select the zone **solid-part-1** in the graphics window (Figure 10.2: Material Point Created (p. 233)).
- Click **Compute** in the **Material Point** dialog box.
- Enable **Draw** to view the material point computed.
- Enable **Limit in Z** in the **Clipping Planes** group box in the **Mesh Generation** task page and adjust the lower slider to see the material point as shown in Figure 9.10: Creating the Material Point (p. 211).



**Figure 9.10: Creating the Material Point**

- f. Enter fluid for **Name** and click **Create**.
- g. Close the **Material Point** dialog box.
2. Set the parameters for mesh quality improvement.

For a relatively simple model like this, the mesh can be improved beyond the quality set as default. You will thus modify the improve parameters, specifying a tighter quality limit for the node movement operation.

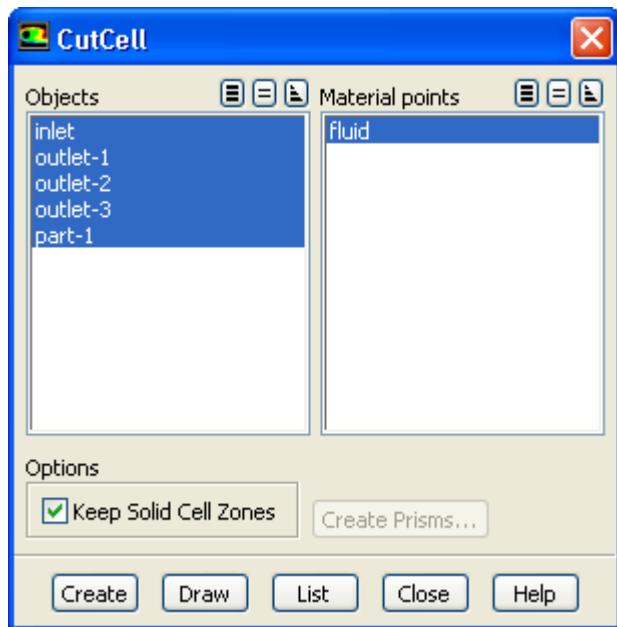
```
>/mesh/cutcell/set/set-post-snap-parameters
Quality limit for Auto Node Move [0.85] 0.75
Iterations for Auto Node Move [1]
Iterations per node in Auto Node Move [50]
Restrict nodes on surfaces [yes]
Dihedral angle for sharp feature angle [120]
Quality limit for cavity remeshing [0.9]
```

### Note

The default quality measure (set using the command `/mesh/cutcell/set/set-cutcell-quality-method`) for the improve operations is ortho skew. To verify quality values using the **Report Cell Limits** dialog box, change the **Quality Measure** to **Ortho Skew** before checking the quality.

3. Generate the CutCell mesh.

**Mesh Generation → (Volume Mesh) CutCell...**



- Select all the objects in the **Objects** selection list.
- Select **fluid** in the **Material points** selection list.
- Retain the selection of **Keep Solid Cell Zones**.
- Click **Create**.

A **Question** dialog box appears, asking if you want to proceed even though some objects do not comprise edges.

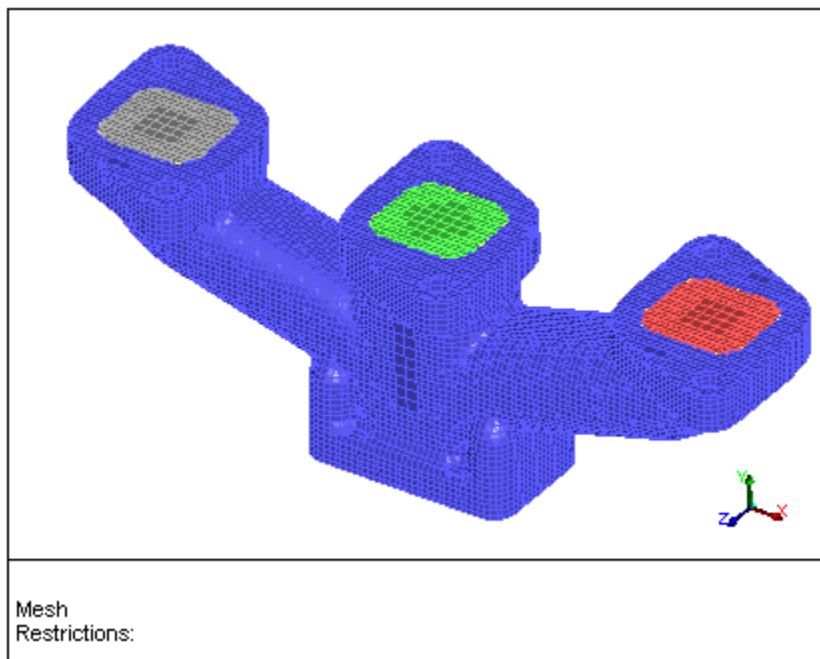
- Click **Yes** in the **Question** dialog box.

The face zones are separated by cell neighbor and normals on face zones connected to the fluid cell zones are oriented into the fluid zone. A face zone group (**\_fluid**) is created for the face zones of each fluid cell zone. Additionally, the defaults for post volume mesh prism generation will be set.

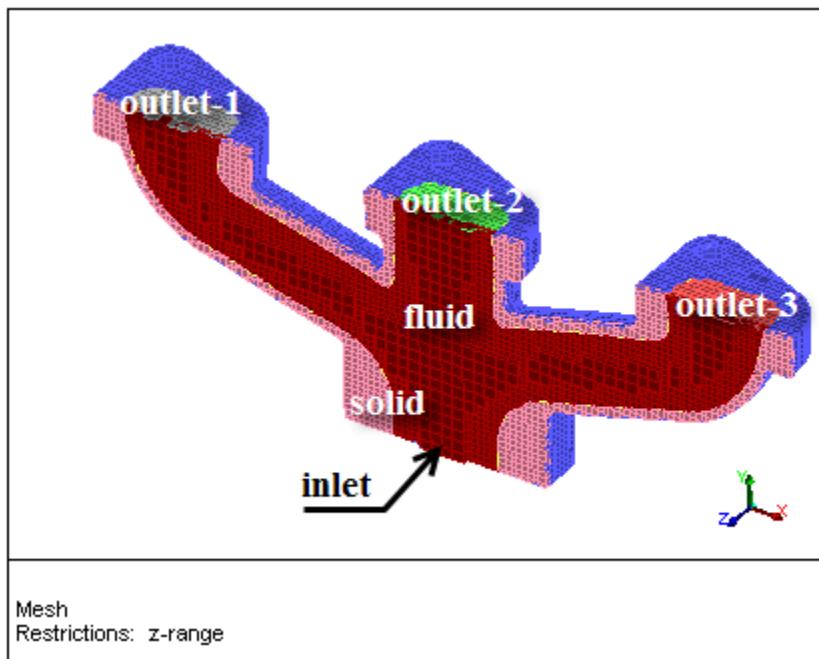
#### 4. Examine the mesh.

##### **Display → Grid...**

- Display the surface mesh.
  - Select all the zones prefixed by **cutcell-boundary** in the **Face Zones** selection list.
  - Click **Display** (Figure 9.11: CutCell Surface Mesh (p. 213)).

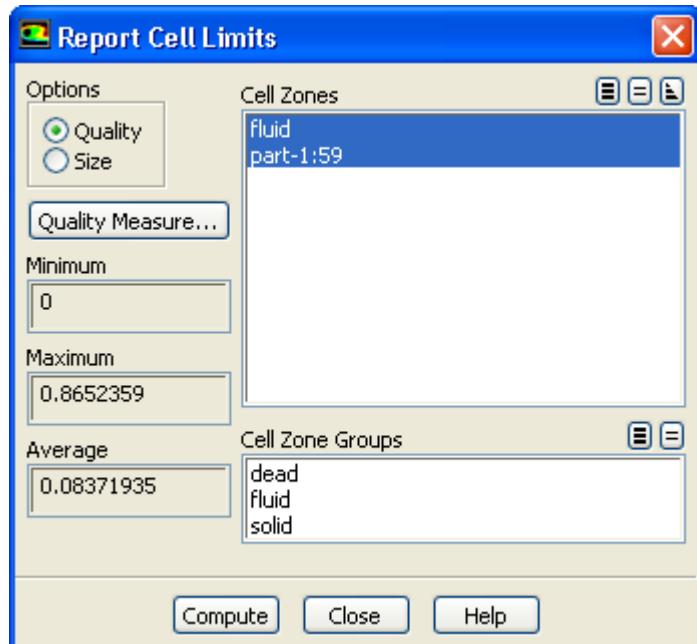
**Figure 9.11: CutCell Surface Mesh**

- b. Display a section through the solid and fluid at  $z = -50$  overlaid with the surface mesh.
  - i. Retain the selected zones and click **Compute** in the **Bounds** tab of the **Display Grid** dialog box.
  - ii. Disable **Limit by X** and **Limit by Y**.
  - iii. Enter  $-50$  for **Maximum** in the **Z Range** group box and click **Display**.
  - iv. Enable the overlaying of graphics.  
**Display → Scene...**
    - A. Select all the zones in the **Names** selection list.
    - B. Enable **Overlays** in the **Scene Composition** group box.
    - C. Click **Apply** and close the **Scene Description** dialog box.
  - v. Click the **Cells** tab in the **Display Grid** dialog box and enable **All** in the **Options** group box.
  - vi. Select all the cell zones in the **Cell Zones** selection list.
  - vii. Enter  $-50$  for **Minimum** and retain the value of  $-50$  for **Maximum** in the **Z Range** group box in the **Bounds** tab.
  - viii. Click **Display** ([Figure 9.12: CutCell Mesh–Section at  \$z = -50\$  \(p. 214\)](#)).

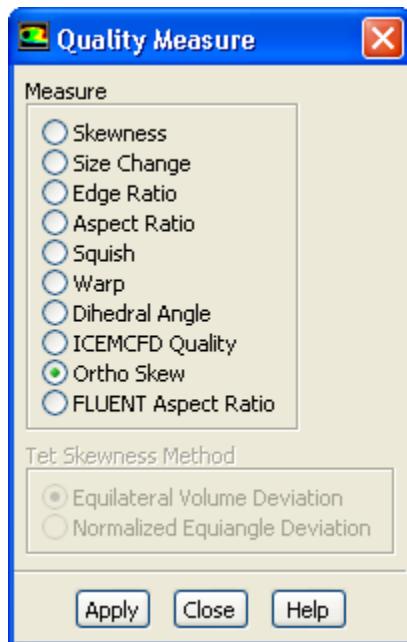
**Figure 9.12: CutCell Mesh-Section at z = -50**

5. Check the quality of the CutCell mesh.

**Report → Cell Limits...**



- a. Select all the zones in the **Cell Zones** selection list.
- b. Set the quality measure to ortho skew.
  - i. Click the **Quality Measure...** button to open the **Quality Measure** dialog box.



- ii. Select **Ortho Skew** in the **Measure** list.
  - iii. Click **Apply** and close the **Quality Measure** dialog box.
- c. Click **Compute**.

The maximum quality reported is around 0.865.

## 9.7. Post CutCell Meshing Cleanup Operations

1. Clean up the CutCell mesh.

Operations such as deleting dead zones, deleting geometry/wrap objects, deleting edge zones, removing face/cell zone name prefixes and/or suffixes, deleting unused faces and nodes are performed during the cleanup operation.

### Mesh Generation → (Volume Mesh) Cleanup

A **Question** dialog box appears, asking you to confirm that you want to proceed with the cleanup operation.

- Click **Yes** in the **Question** dialog box to perform the cleanup operation.

2. Change the zone types for the inlet and outlet zones.

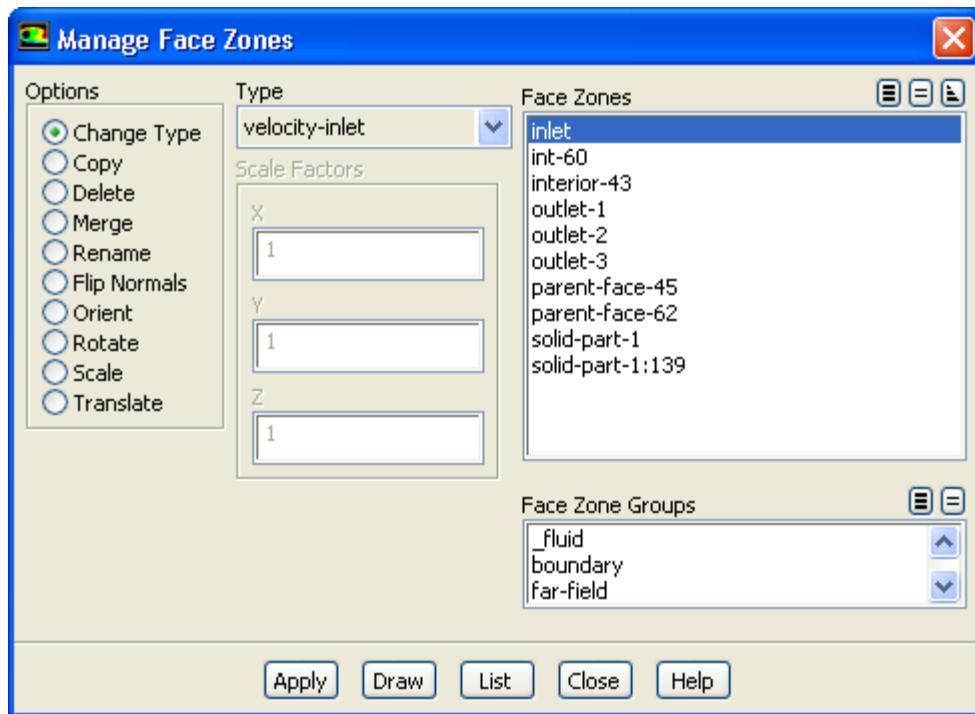
---

#### Important

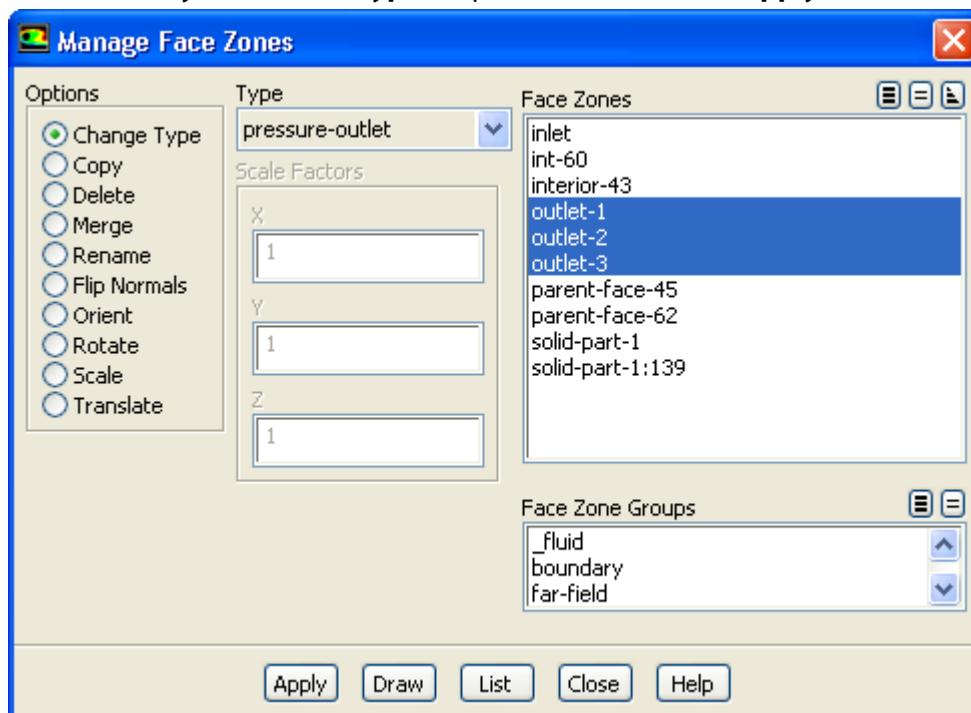
The inlet and outlet zones are recovered as **wall** zones. You will now change the boundary zone type to the appropriate type.

---

#### Boundary → Manage...



- Select the inlet zone, **inlet** and in the **Face Zones** selection list.
- Retain the selection of **Change Type** in the **Options** list.
- Select **velocity-inlet** in the **Type** drop-down list and click **Apply**.



- Select the outlet zones, **outlet-1**, **outlet-2**, and **outlet-3** in the **Face Zones** selection list.
- Retain the selection of **Change Type** in the **Options** list.
- Select **pressure-outlet** in the **Type** drop-down list and click **Apply**.

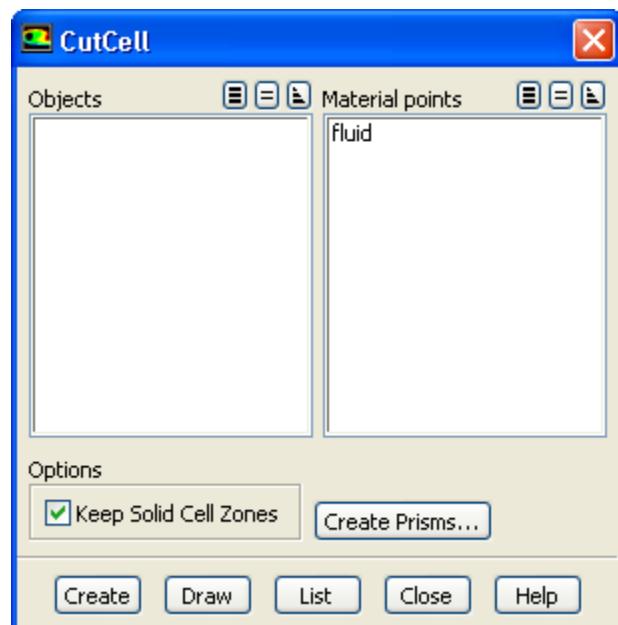
- g. Close the **Manage Face Zones** dialog box.
3. Save the volume mesh.

**File → Write → Mesh...**

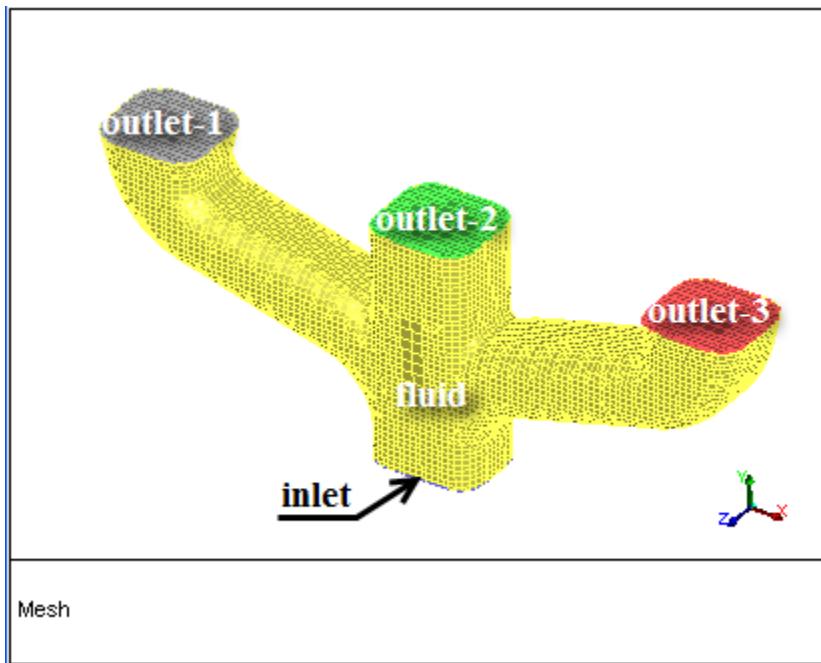
## 9.8. Generating Prisms for the CutCell Mesh

◆ **Mesh Generation → (Volume Mesh) CutCell...**

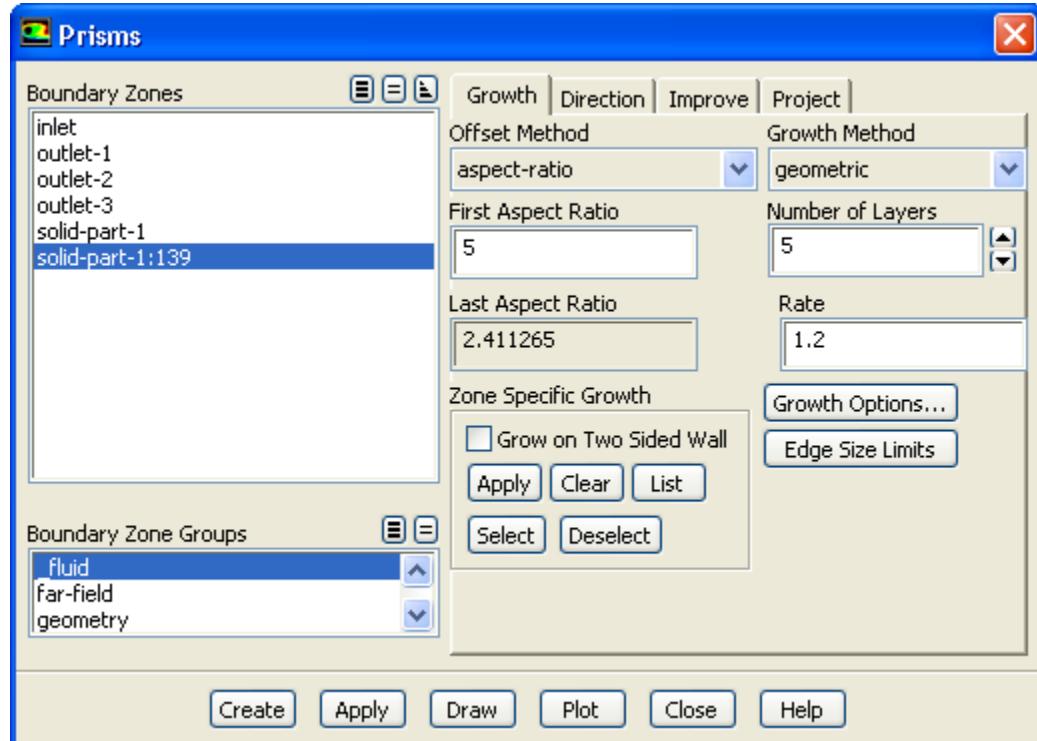
The **Create Prisms...** button is now enabled in the **CutCell** dialog box.



1. Set the parameters for prism growth.
  - a. Click the **Create Prisms...** button to open the **Prisms** dialog box.
  - b. Examine the face zone group created for the face zones of the fluid cell zones and determine the face zones for which prism meshing parameters are to be specified.
    - i. Select **\_fluid** in the **Boundary Zone Groups** selection list and click **Draw** ([Figure 9.13: Fluid Group \(p. 218\)](#)).

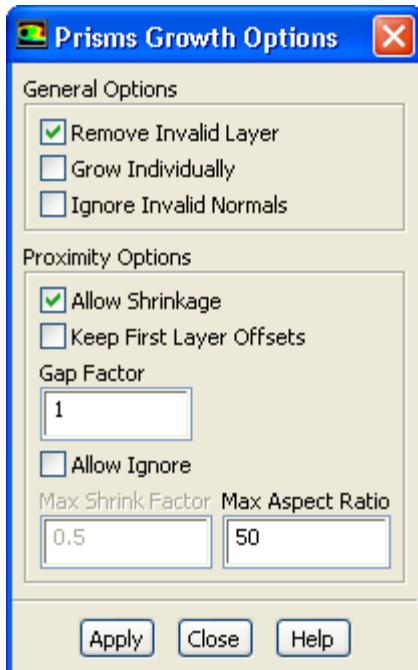
**Figure 9.13: Fluid Group**

- ii. Deselect all the zones corresponding to the inlets and outlets in the **Boundary Zones** selection list.



- c. Select **aspect-ratio** in the **Offset Method** drop-down list.
- d. Select **geometric** in the **Growth Method** drop-down list.
- e. Enter 5 for **First Aspect Ratio**.

- f. Set **Number of Layers** to 5.
- g. Enter 1.2 for **Rate**.
- h. Click **Apply** in the **Zone Specific Growth** group box.
- i. Click the **Growth Options...** button to open the **Prisms Growth Options** dialog box.



- i. Enter 1 for **Gap Factor**.
- ii. Click **Apply** and close the **Prisms Growth Options** dialog box.
- j. Click **Apply** and close the **Prisms** dialog box.

2. Click **Create** in the **Prisms** dialog box.

A **Question** dialog box will appear, asking if you want to morph the existing volume mesh.

- Click **Yes** in the **Question** dialog box to generate the prism layers.

3. Close the **Prisms** and **CutCell** dialog boxes.
4. Check the quality of the CutCell mesh.

#### **Report → Cell Limits...**

- a. Select all the zones in the **Cell Zones** selection list.
- b. Set the quality measure to orthoskew.
  - i. Click the **Quality Measure...** button to open the **Quality Measure** dialog box.
  - ii. Select **Ortho Skew** in the **Measure** list.

- iii. Click **Apply** and close the **Quality Measure** dialog box.
  - c. Click **Compute**.  
The quality reported is around 0.846.
  - d. Close the **Report Cell Limits** dialog box.
5. Improve the mesh quality.

Again, the mesh quality can be improved beyond the smoothing quality limits set as default. You will modify the improve parameters, specifying a tighter quality limit for the node movement operation.

- a. Set the parameters for mesh quality improvement.

```
>/mesh/cutcell/set/set-post-morph-parameters
  Quality limit for Auto Node Move [0.9] 0.75
  Iterations for Auto Node Move [1]
  Iterations per node in Auto Node Move [50]
  Restrict nodes on surfaces [yes] no
  Dihedral angle for sharp feature angle [120]
  Quality limit for cavity remeshing [0.95]
```

---

### Note

The prism cap in this case is an internal boundary zone, and hence, you can allow full freedom of node movement to improve quality.

---

- b. Improve the mesh quality using the following command:

```
>/mesh/cutcell/modify/post-morph-improve
()
Cell Zones(1) [()] fluid*
Cell Zones(2) [()]
()
Face Zones(1) [()] *cap*
Face Zones(2) [()]
```

The worst quality is reported to be around 0.748.

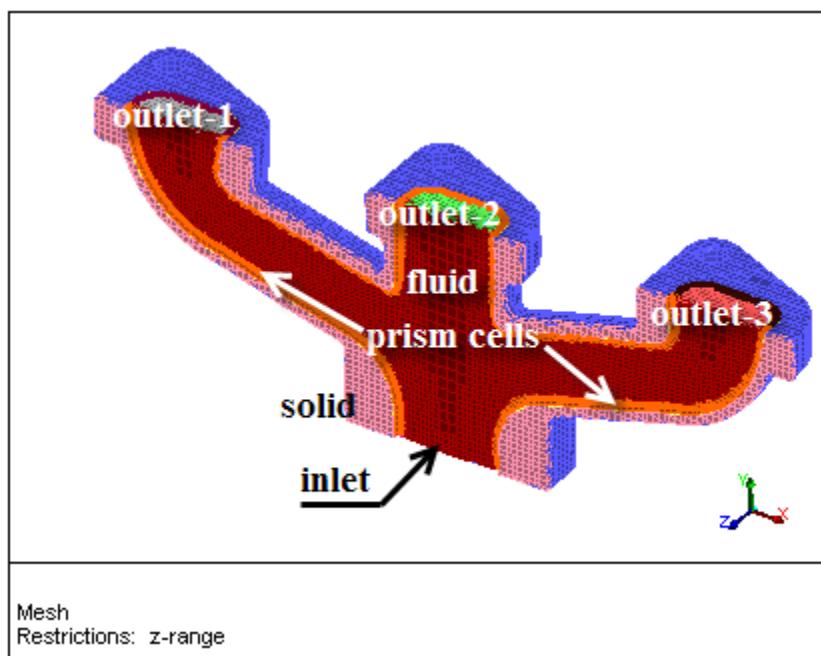
- c. Display a section through the solid and fluid at z = -50 overlaid with the surface mesh.
  - i. Select all the boundary zones in the **Face Zones** selection list.
  - ii. Click **Compute** in the **Bounds** tab of the **Display Grid** dialog box.
  - iii. Disable **Limit by X** and **Limit by Y**.
  - iv. Enter -50 for **Maximum** in the **Z Range** group box and click **Display**.
  - v. Enable the overlaying of graphics.

### Display → Scene...

- A. Select all the zones in the **Names** selection list.

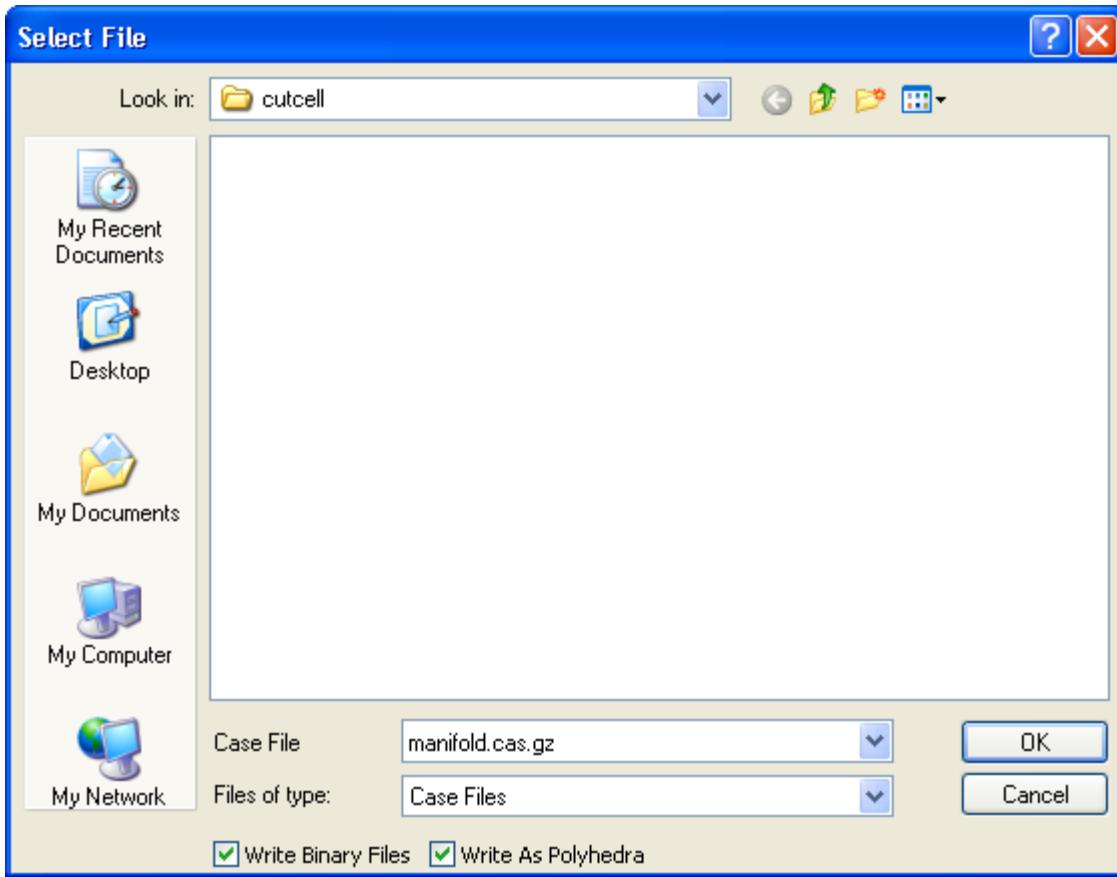
- B. Enable **Overlays** in the **Scene Composition** group box.
- C. Click **Apply** and close the **Scene Description** dialog box.
- vi. Click the **Cells** tab in the **Display Grid** dialog box and enable **All** in the **Options** group box.
- vii. Select all the cell zones in the **Cell Zones** selection list.
- viii. Enter  $-50$  for **Minimum** and retain the value of  $-50$  for **Maximum** in the **Z Range** group box in the **Bounds** tab.
- ix. Click **Display** (Figure 9.12: CutCell Mesh–Section at  $z = -50$  (p. 214)).

**Figure 9.14: CutCell Mesh With Prisms**



- d. Save the case file.

**File → Write → Case...**



- i. Enable **Write As Polyhedra**.

---

#### Note

The case will not produce a valid mesh in the solution mode in ANSYS FLUENT without the **Write as Polyhedra** option due to hanging node configurations on the internal boundary between the solid and the fluid cell zones. This may be eliminated using a fixed size on this boundary.

---

- ii. Enter **manifold.cas.gz** for **Case File** and click **OK**.

A message appears, indicating that the case file being written from the meshing mode will be incompatible with previously saved data files.

- iii. Click **OK** to close the **Information** dialog box.

6. Exit ANSYS FLUENT.

## 9.9. Summary

This tutorial demonstrated the procedure for generating the CutCell mesh for a manifold. You created capping surfaces and defined size functions and a material point for the CutCell mesh. You then created and examined the CutCell mesh, and used the cleanup operation. Further, you set parameters and

generated prisms for the CutCell mesh. The tutorial also demonstrated the improvement of the CutCell mesh after prism generation, and finally saving the file for use in the solution mode in ANSYS FLUENT.



---

## Chapter 10: Object Based Mesh Generation

---

The object based meshing approach allows you to generate a tetrahedral, hexcore, or hybrid volume mesh based on meshing objects from the imported geometry. This approach involves creating a conformally connected surface mesh using the object wrapping and sewing operations before generating the volume mesh. This tutorial demonstrates the generation of the volume mesh for a mixer pipe in the meshing mode in ANSYS FLUENT. It also demonstrates the transfer of the mesh to the solution mode, the set up and solution of the CFD problem, and visualizing the results.

This tutorial demonstrates how to do the following:

- Start the parallel version of ANSYS FLUENT in meshing mode.
- Import the CAD geometry.
- Display and manipulate meshing objects.
- Define size functions.
- Create the volume mesh.
- Transfer the mesh to solution mode.
- Set up the CFD simulation in solution mode.
- Examine the results.

### 10.1. Prerequisites

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

### 10.2. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

To prepare for running this tutorial:

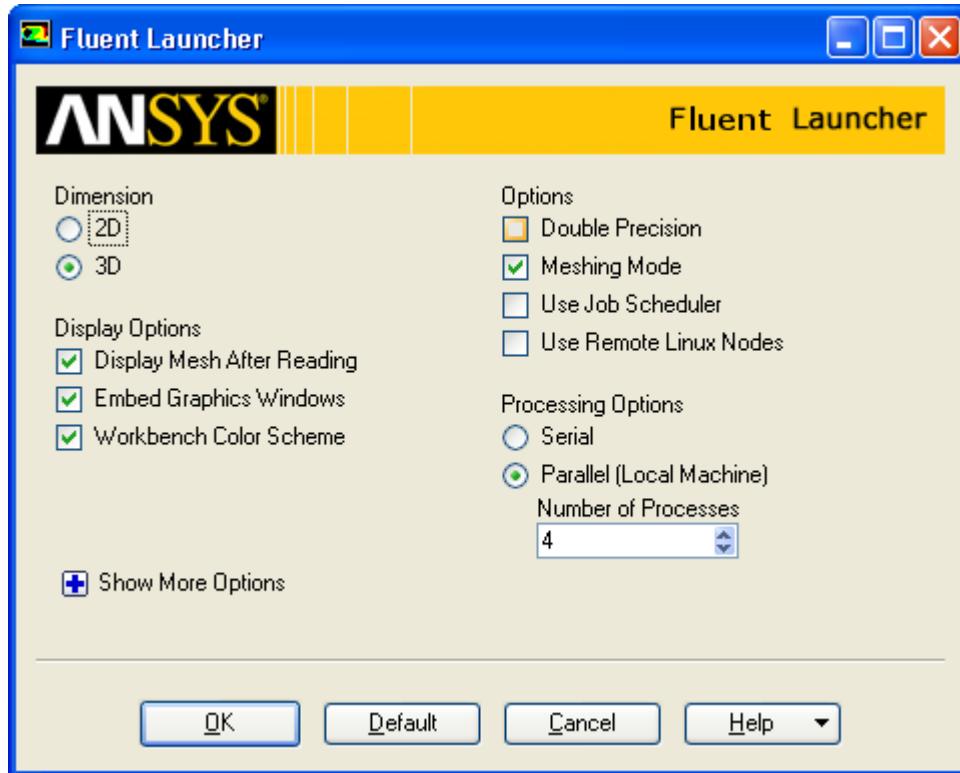
1. Download the tutorial input file `object-mesh.zip` for the tutorial.
2. Unzip `object-mesh.zip`.

The file `mixer-t.stp` can be found in the `object-mesh` folder created on unzipping the file.

## 10.3. Starting ANSYS FLUENT in Meshing Mode

1. Open the FLUENT Launcher by clicking the Windows **Start** menu, then selecting **FLUENT 14.5** in the **Fluid Dynamics** sub-menu of the **ANSYS 14.5** program group.

**Start → All Programs → ANSYS 14.5 → Fluid Dynamics → FLUENT 14.5**



2. Select the appropriate start up options.
  - a. Ensure that **3D** is selected in the **Dimension** list.
  - b. Enable **Meshing Mode** under **Options**.
  - c. Retain the **Display Mesh After Reading**, **Embed Graphics Windows**, and **Workbench Color Scheme** options.

---

### Note

The selected preferences will be retained for future sessions.

---

- d. Select **Parallel (Local Machine)** in the **Processing Options** list and enter 4 for **Number of Processes**.

---

### Important

ANSYS FLUENT in meshing mode can only use a single process (even if the session is started in parallel using more than one process). By default, for all parallel simulations started with more than one process, in meshing mode, the FLUENT session

will be started by using one process and, when switching to solution mode, FLUENT will automatically spawn the remaining parallel node processes.

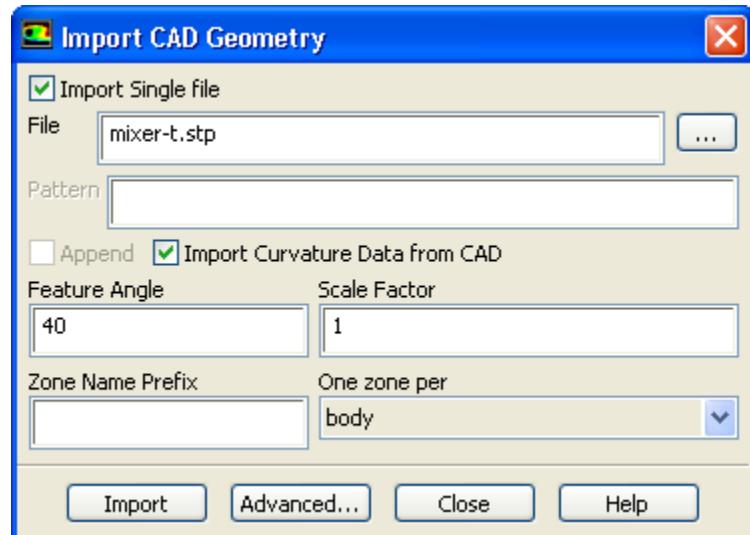
3. Set the path to the working directory.
  - a. Click the **Show More Options** button.
  - b. Enter the path to the working directory by double-clicking the **Working Directory** text box and typing.

Alternatively, you can click the browse button (  ) next to the **Working Directory** text box and browse to the directory, using the **Browse For Folder** dialog box.

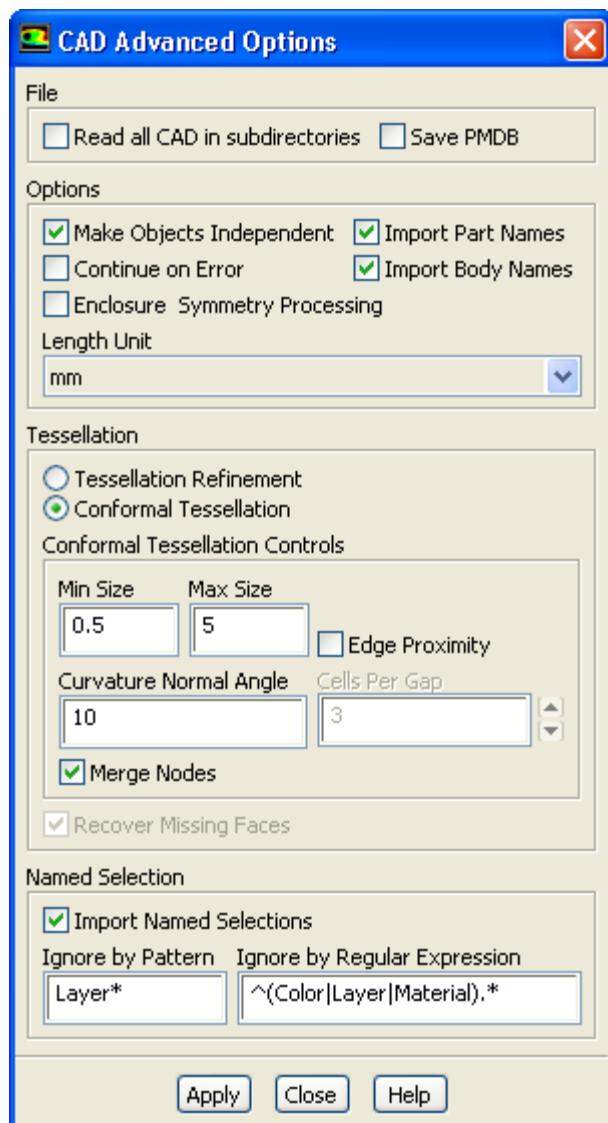
4. Click **OK** to start the parallel version of ANSYS FLUENT in meshing mode.

## 10.4. Import the CAD Geometry

**File → Import → CAD...**



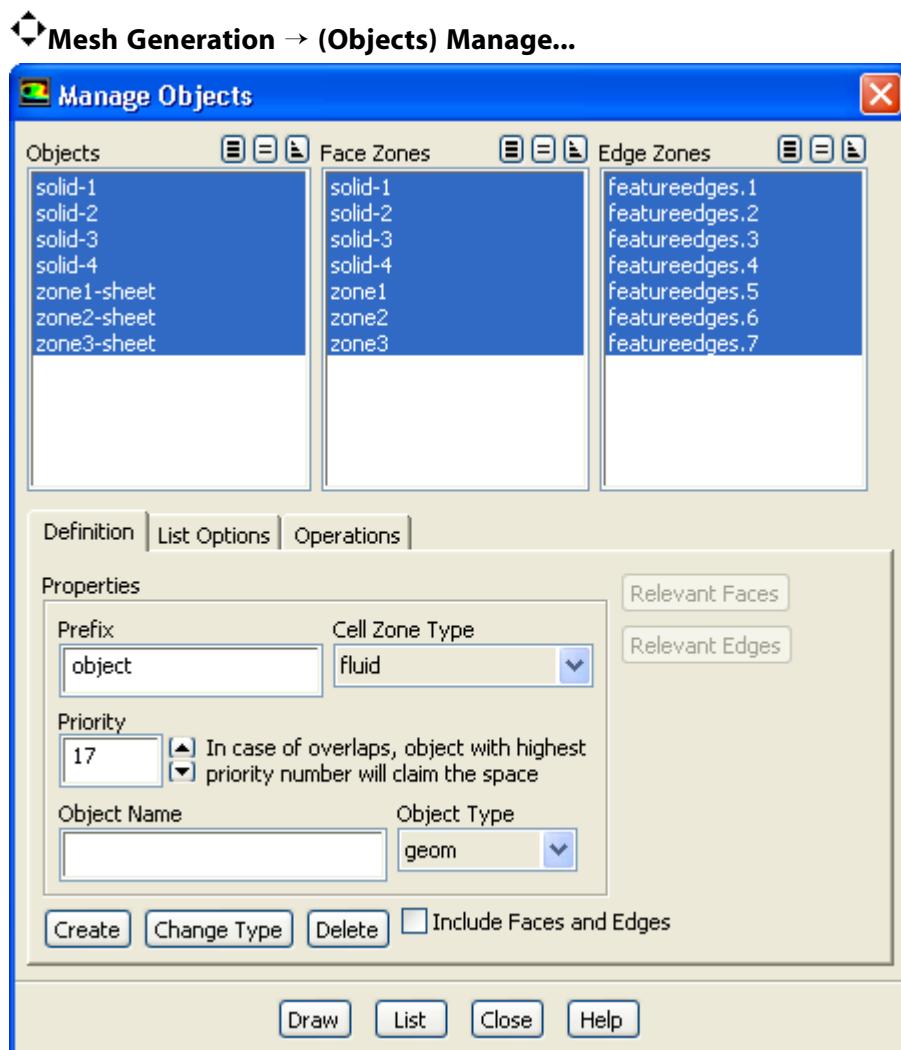
1. Ensure that **Import Single file** is enabled.
2. Click  and select the file `mixer-t.stp`.
3. Enable **Import Curvature Data from CAD**.
4. Retain the values of 40 for **Feature Angle** and 1 for **Scale Factor**, respectively.
5. Retain the selection of **body** in the **One zone per** drop-down list.
6. Click the **Advanced...** button to open the **CAD Advanced Options** dialog box.



- a. Select **mm** in the **Length Unit** drop-down list.
  - b. Select **Conformal Tessellation** in the **Tessellation** group box.
    - i. Specify values of 0.5 and 5 for **Min Size** and **Max Size**, respectively.
    - ii. Retain the default value of 10 for **Curvature Normal Angle**.
  - c. Retain the selection of **Import Named Selections** in the **Named Selection** group box.
  - d. Click **Apply** and close the **CAD Advanced Options** dialog box.
7. Click **Import** in the **Import CAD Geometry** dialog box.
  8. Close the **Import CAD Geometry** dialog box.

## 10.5. Prepare the Geometry

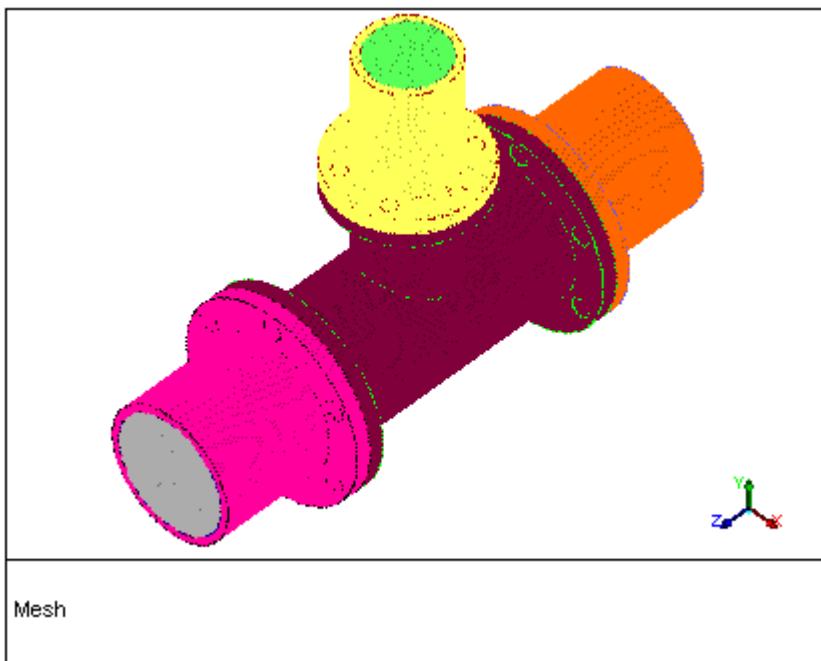
1. Display the imported geometry objects.



- Select all the objects in the **Objects** selection list.
- Click **Draw**.
- View the isometric view.

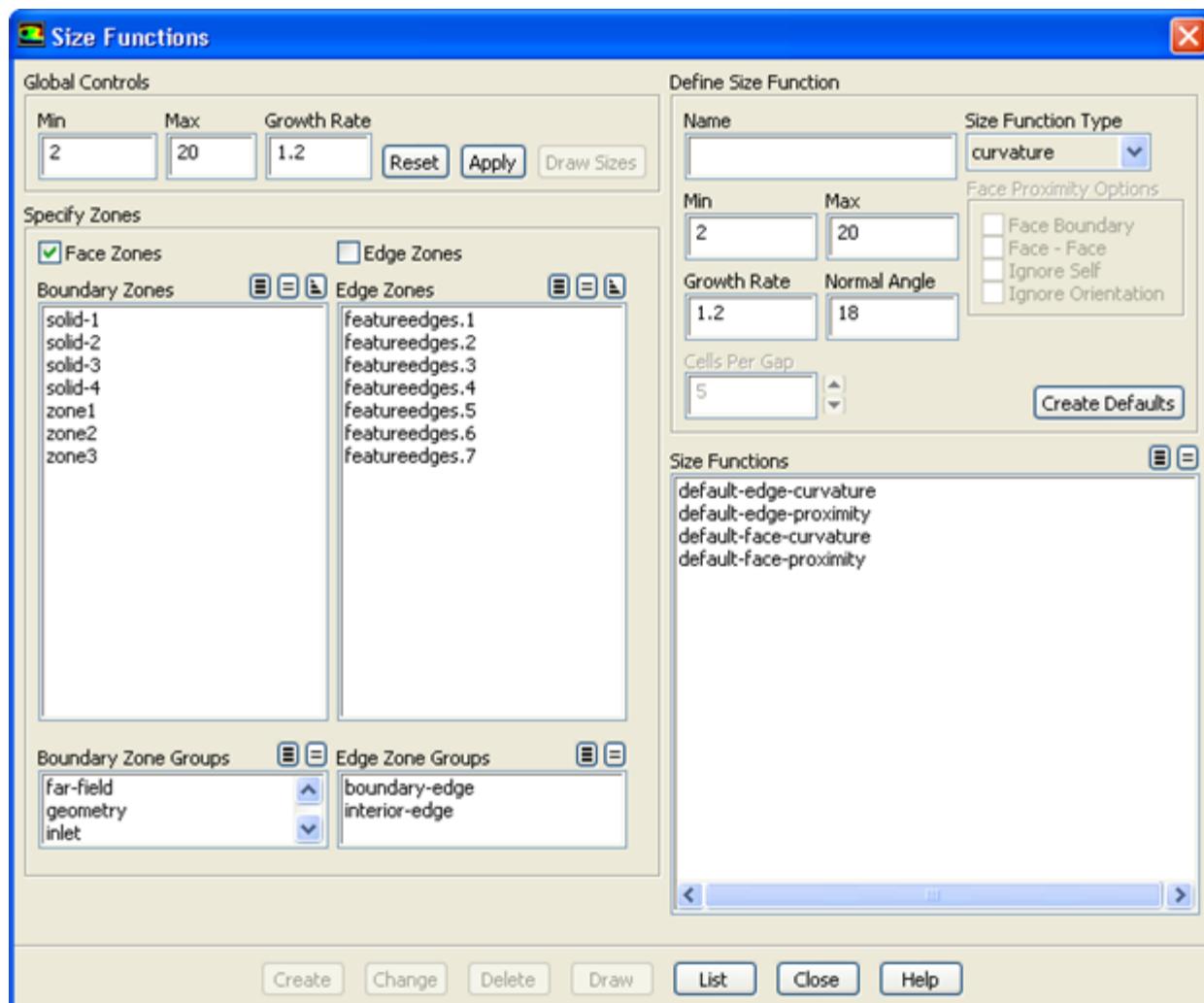
**Display → Views...**

**Figure 10.1: Geometry Objects**



- d. Close the **Manage Objects** dialog box.
2. Set up default size functions based on face and edge curvature and proximity.

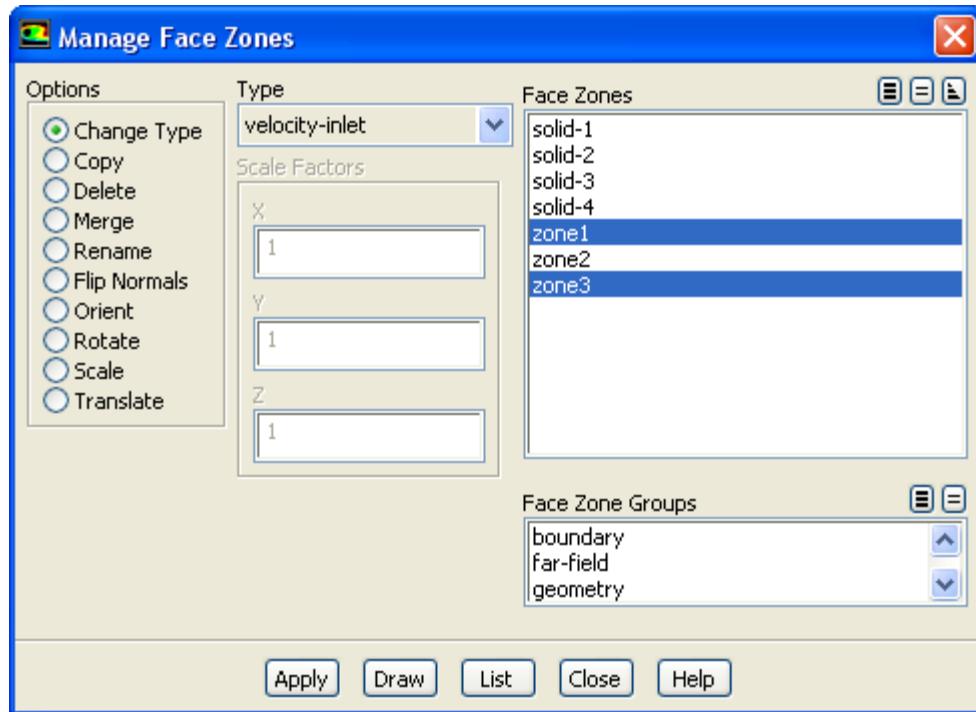
Mesh Generation → Size Functions...



- Enter values of 2 and 20 for **Min** and **Max**, respectively in the **Global Controls** group box.
  - Click **Apply**.
  - Click **Create Defaults** in the **Define Size Function** group box.
- The curvature and proximity size functions will be listed in the **Size Functions** selection list.
- Set the appropriate zone types for the inlet and outlet zones.

Setting the appropriate type avoids growth of prism layers on the inlet and outlet zones when the volume mesh is generated.

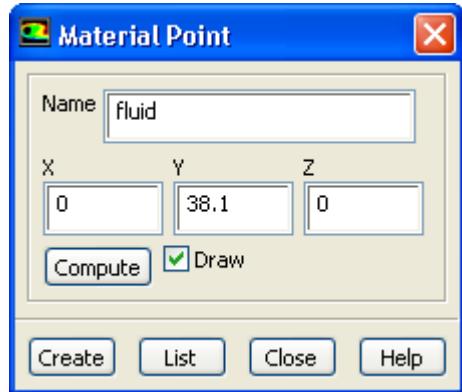
**Boundary → Manage...**



- a. Select the inlet zones, **zone1** and **zone3**, in the **Face Zones** selection list.
- b. Retain the selection of **Change Type** in the **Options** list.
- c. Select **velocity-inlet** in the **Type** drop-down list and click **Apply**.
- d. Select the outlet zone, **zone2**, in the **Face Zones** selection list.
- e. Retain the selection of **Change Type** in the **Options** list.
- f. Select **pressure-outlet** in the **Type** drop-down list and click **Apply**.

4. Define the material point for internal flow.

#### Mesh Generation → Material Point...

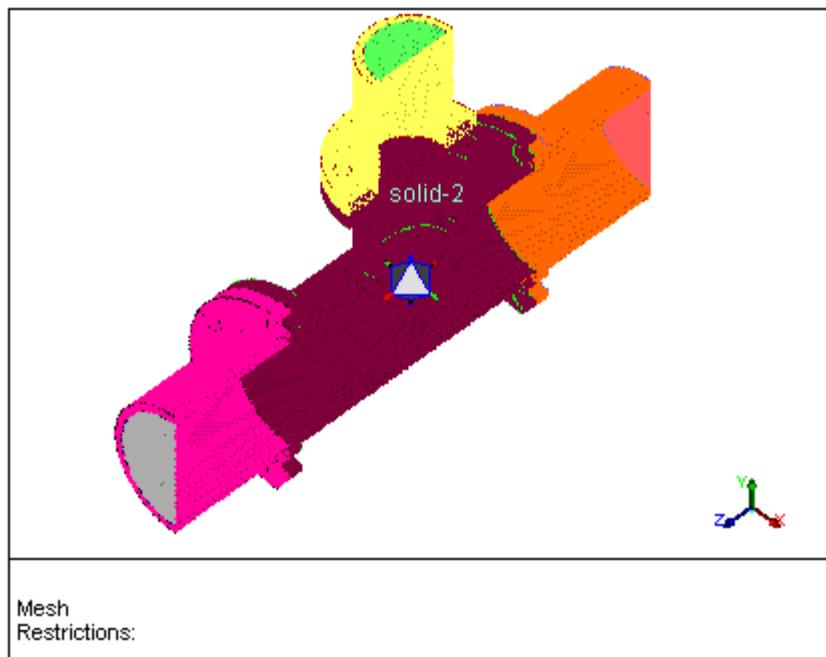


- a. Ensure that the selection filter is set to zone (**Ctrl+Z**).
- b. Select the zone **solid-2** in the graphics window (Figure 10.2: Material Point Created (p. 233)).

- c. Click **Compute** in the **Material Point** dialog box.
- d. Enter a value of 0 for the X coordinate.
- e. Enable **Draw** to view the material point computed.
- f. Enable **Limit in X** in the **Clipping Planes** group box in the **Mesh Generation** task page and adjust the lower slider to see the material point as shown in [Figure 10.2: Material Point Created \(p. 233\)](#).



**Figure 10.2: Material Point Created**



g. Enter fluid for **Name** and click **Create**.

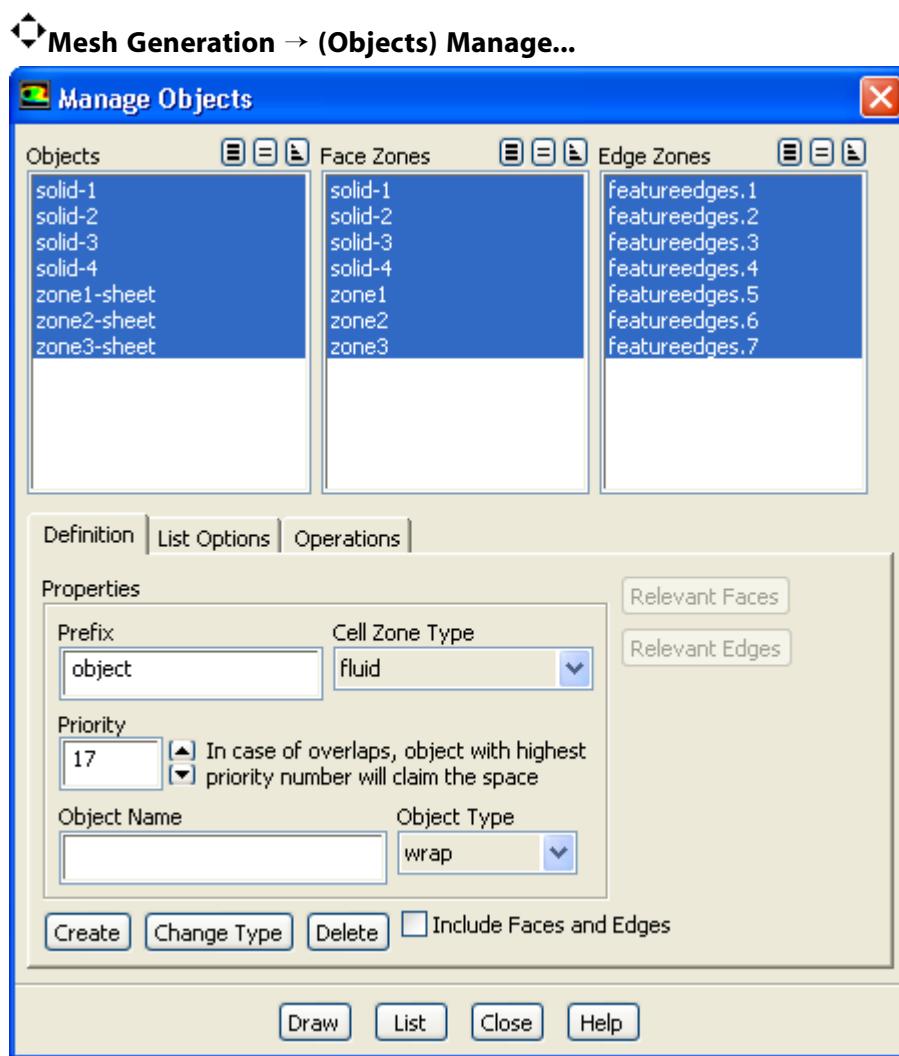
h. Close the **Material Point** dialog box.

## 10.6. Sewing Objects

- Convert the imported geometry objects to wrap objects.

### Important

Since conformal tessellation options were selected when the CAD file was imported, the geometry objects can be converted to wrap objects without the intermediate object wrapping step. If conformal tessellation options are not used during import, an additional object wrapping step is necessary to make the objects conformal.



- Select all the geometry objects in the **Objects** selection list.
- Select **wrap** in the **Object Type** drop-down list in the **Properties** group box.
- Click **Change Type**.

A **Question** dialog box will appear, indicating that it is assumed the selected objects have conformal facetting.

- d. Click **Yes** in the **Question** dialog box to proceed.

All objects will be converted to wrap type.

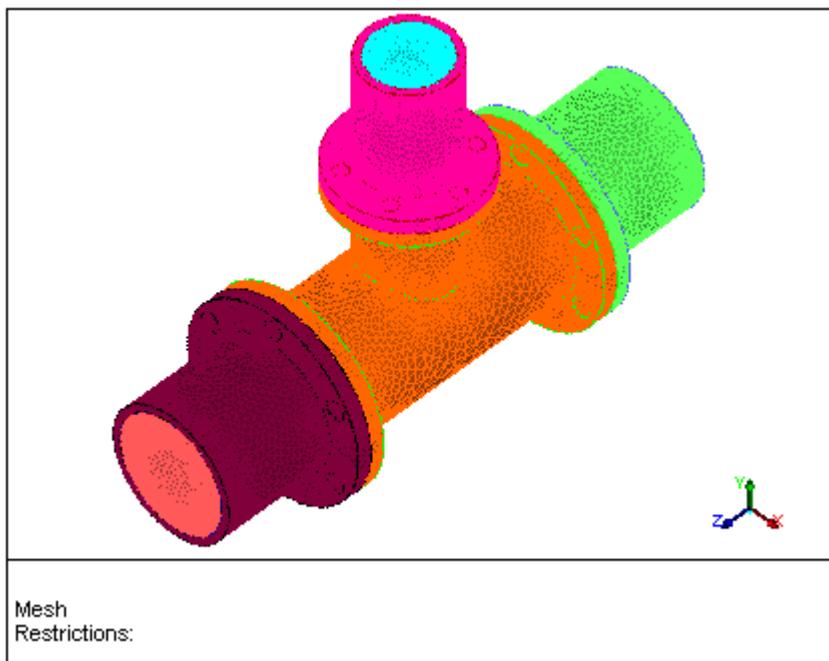
- e. Close the **Manage Objects** dialog box.
2. Sew the wrap objects to obtain the connected surface mesh.



- a. Select all the wrap objects in the **Objects** selection list.
- b. Enter **mesh-object** for **New Object Name**.
- c. Click **Sew**.

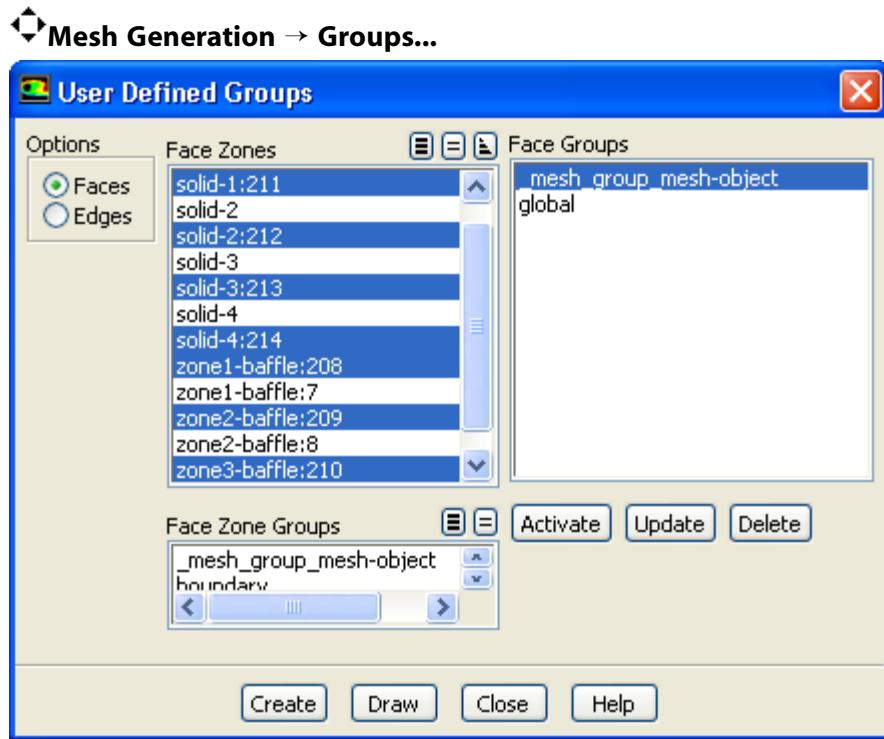
The mesh object (**mesh-object**) is created along with a face zone group (**\_mesh\_group\_mesh-object**).

3. Select the mesh object created (**mesh-object**) in the **Objects** list and click **Draw** (Figure 10.3: Mesh Object Created (p. 236)).

**Figure 10.3: Mesh Object Created**

4. Close the **Sew** dialog box.
5. Activate the face zone group comprising the mesh object face zones.

Activating the face zone group comprising the mesh object face zones allows you to view only the mesh object face zones in the lists in all dialog boxes.

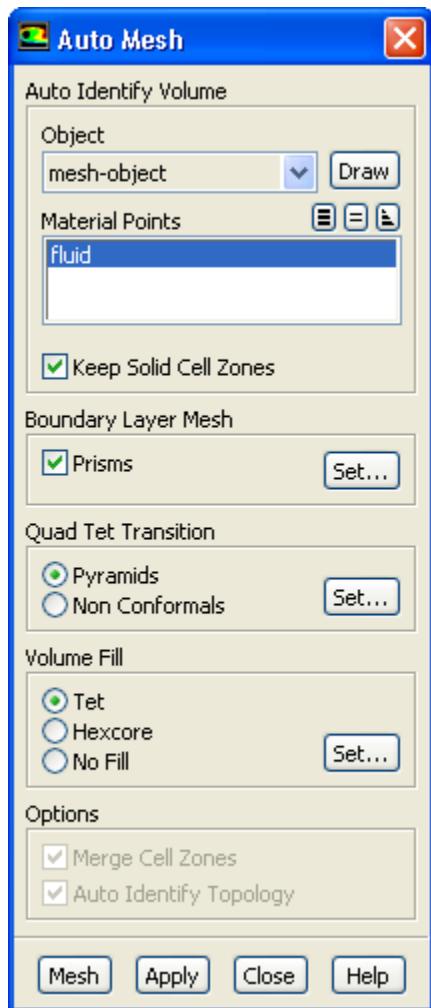


- a. Select **\_mesh\_group\_mesh-object** in the **Face Groups** selection list.

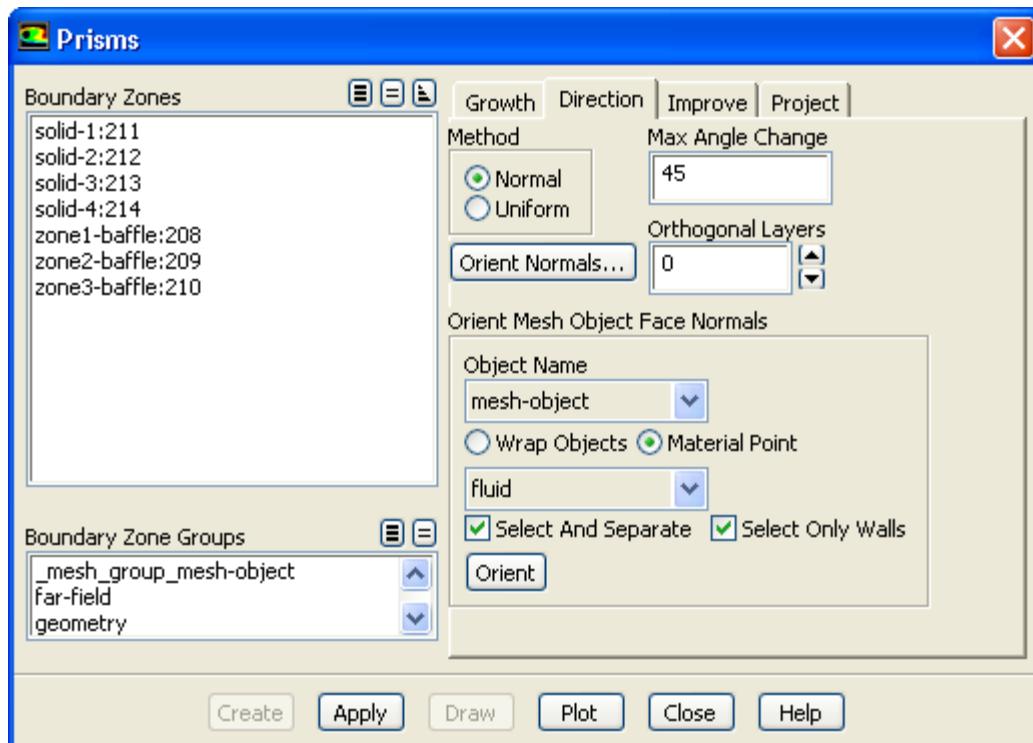
- b. Click **Activate** and close the **User Defined Groups** dialog box.

## 10.7. Generate the Volume Mesh

### Mesh Generation → Auto Mesh...



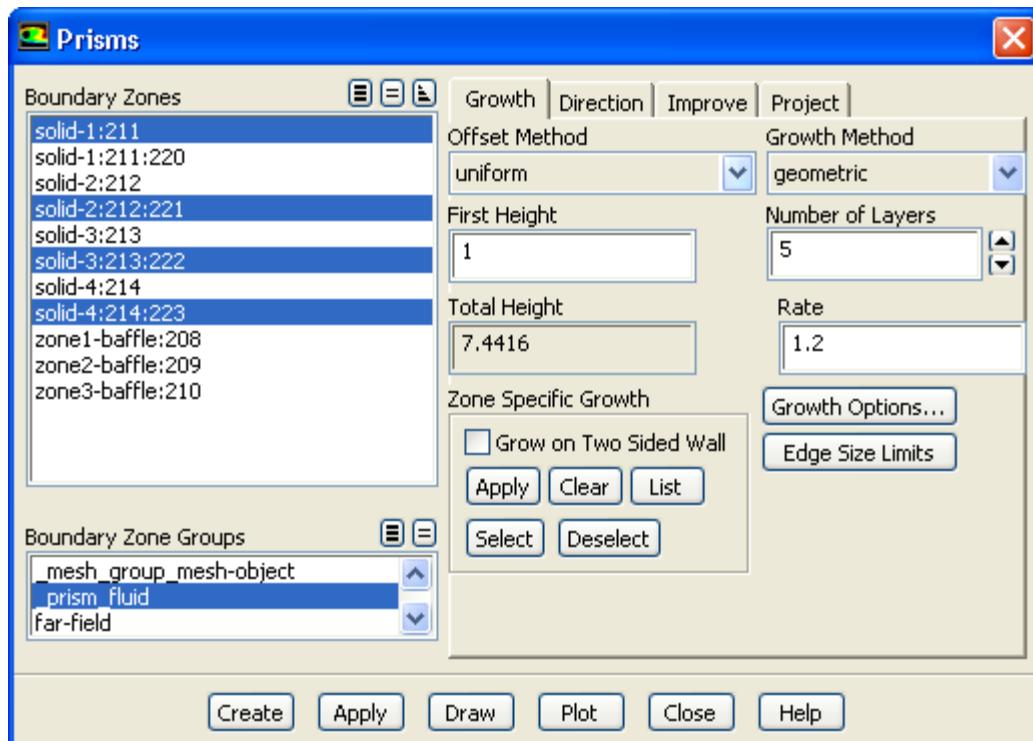
1. Select **mesh-object** in the **Object** drop-down list in the **Auto Identify Volume** group box.
2. Select **fluid** in the **Material Points** selection list.
3. Retain the selection of **Keep Solid Cell Zones**.
4. Set the prism meshing parameters.
  - a. Click the **Set...** button in the **Boundary Layer Mesh** group box to open the **Prisms** dialog box.
  - b. Orient the normals for prism growth.



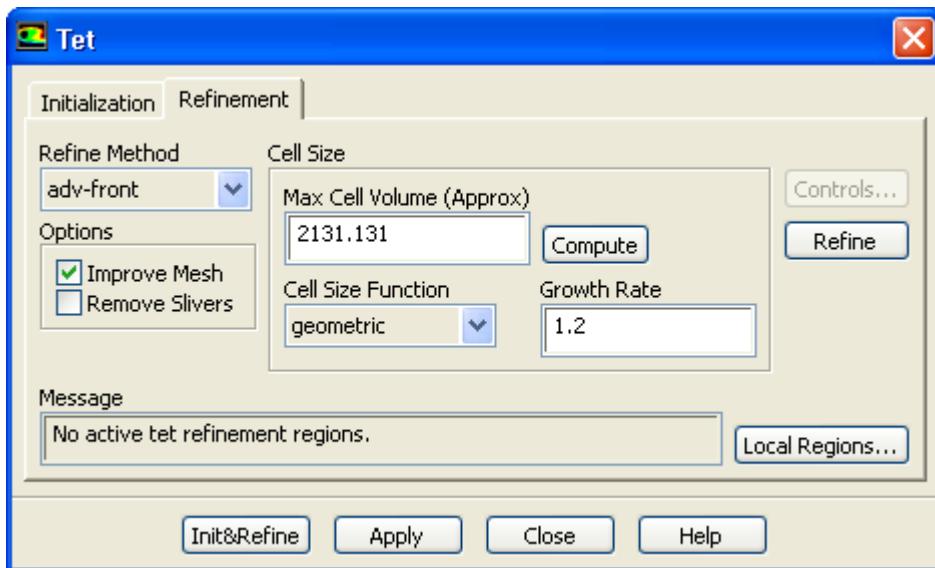
- i. Select **mesh-object** in the **Object Name** drop-down list in the **Orient Mesh Object Face Normals** group box in the **Direction** tab.
- ii. Select **Material Point** and then select **fluid** in the drop-down list.
- iii. Retain the selection of **Select and Separate** and **Select Only Walls**.
- iv. Click **Orient**.

A new face boundary zone group comprising the prism base zones will be created (**\_prism\_fluid**).

- c. Specify the prism growth parameters.



- i. Select **\_prism\_fluid** in the **Boundary Zone Groups** selection list.
- ii. Retain the selection of **uniform** in the **Offset Method** drop-down list and the value **1** for **First Height** in the **Growth** tab.
- iii. Select **geometric** in the **Growth Method** drop-down list and enter **1.2** for **Rate**.
- iv. Set the **Number of Layers** to **5**.
- v. Click **Apply** in the **Zone Specific Growth** group box.
- vi. Click **Apply** and close the **Prisms** dialog box.
- d. Enable **Prisms** in the **Boundary Layer Mesh** group box in the **Auto Mesh** dialog box.
5. Retain the selection of **Pyramids** in the **Quad Tet Transition** list.
6. Retain the selection of **Tet** in the **Volume Fill** list and click the **Set...** button to open the **Tet** dialog box.
  - a. Retain the default settings in the **Initialization** tab.

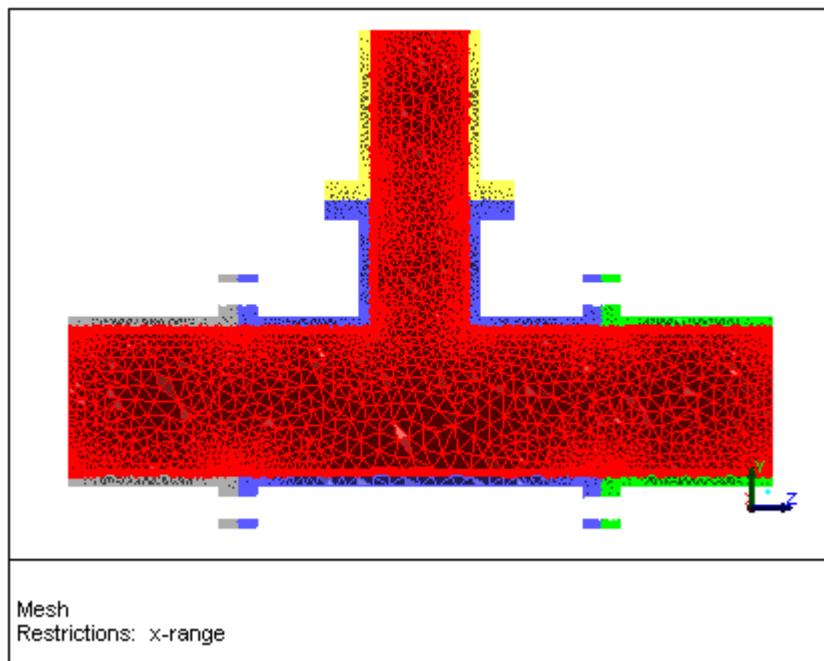


- b. Click **Compute** for **Max Cell Volume (Approx)** in the **Cell Size** group box in the **Refinement** tab. A **Question** dialog box will appear, asking if you want to compute the maximum cell size based on the mesh objects.
- c. Click **Yes** in the **Question** dialog box.
- d. Enter **1 . 2** for **Growth Rate**.
- e. Click **Apply** and close the **Tet** dialog box.
7. Click **Mesh** in the **Auto Mesh** dialog box.
8. Close the **Auto Mesh** dialog box.
9. Examine the volume mesh.

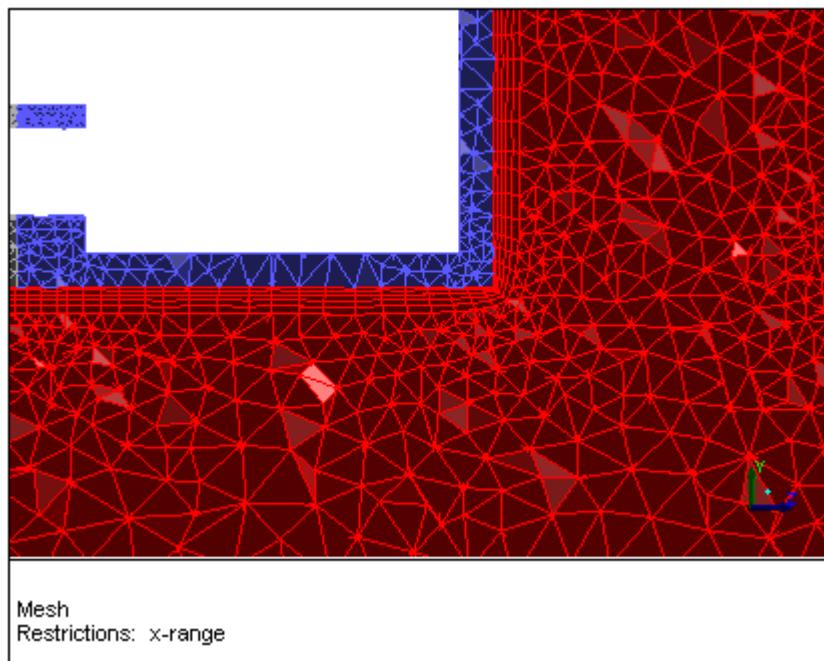
#### **Display → Grid...**

- a. Enable **All** in the **Options** group box and select all the cell zones in the **Cell Zones** selection list in the **Cells** tab.
- b. Enable **Limit by X** and set the **Minimum** and **Maximum** values to 0 in the **X Range** group box in the **Bounds** tab.
- c. Click **Display**.
- d. Display the left view ([Figure 10.4: Volume Mesh Generated \(p. 241\)](#)).

#### **Display → Views...**

**Figure 10.4: Volume Mesh Generated**

- e. Zoom in to the area shown in [Figure 10.5: Volume Mesh—Zoomed View \(p. 241\)](#).

**Figure 10.5: Volume Mesh—Zoomed View**

It is recommended that you have at least 3–5 cells across the solid zones for an accurate conjugate heat transfer simulation.

- f. Close the **Display Grid** dialog box.
10. Clean up the volume mesh.

Operations such as deleting dead zones, deleting geometry/wrap objects, deleting edge zones, removing face/cell zone name prefixes and/or suffixes, deleting unused faces and nodes are performed during the cleanup operation.

#### ◀ Mesh Generation → (Volume Mesh) Cleanup

A **Question** dialog box appears, asking you to confirm that you want to proceed with the cleanup operation.

- Click **Yes** in the **Question** dialog box to perform the cleanup operation.

## 10.8. Transfer the Mesh to Solution Mode

1. Check the mesh.

Check the mesh to ensure it has no negative cell volumes or left-handed faces before saving the mesh file.

#### Mesh → Check

The printed results of the check show no problems, hence the mesh is valid for use in the solver.

2. Check the mesh quality.

Check that the mesh quality is adequate for use in the solver.

#### Mesh → Check Quality

3. Save the mesh file (`mixer.msh.gz`).

#### File → Write → Mesh...

4. Transfer the mesh to solution mode.

- a. Click **Switch to Solution** (  ) in the **Mode** toolbar.

A **Question** dialog box will appear, asking you to confirm that you want to switch to solution mode.

- b. Click **Yes** in the **Question** dialog box to transfer the mesh to solution mode.

---

#### Important

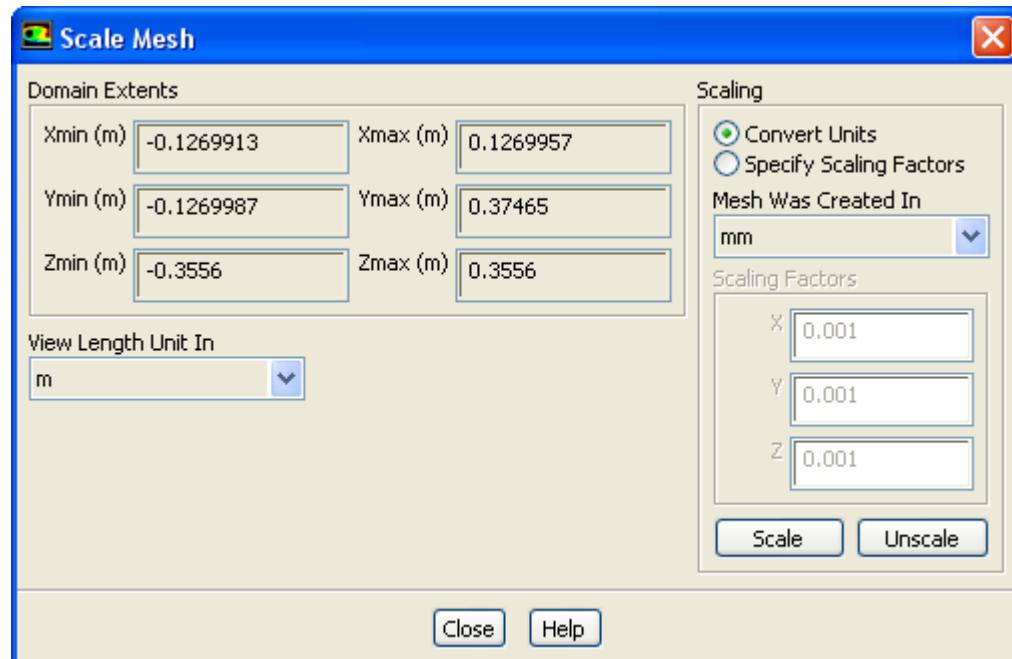
When switching to solution mode, FLUENT will automatically spawn the remaining parallel node processes and the mesh will be automatically partitioned for use in the solver.

---

## 10.9. Solution Setup

1. Scale the mesh to meters.

↳ General → Scale...

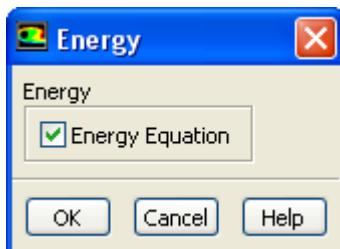


- a. Select **mm** in the **Mesh Was Created In** drop-down list.
  - b. Click **Scale**.
- The domain extents will be updated in the **Scale Mesh** dialog box.
- c. Close the **Scale Mesh** dialog box.
2. Set up the models for the CFD simulation.

↳ Models

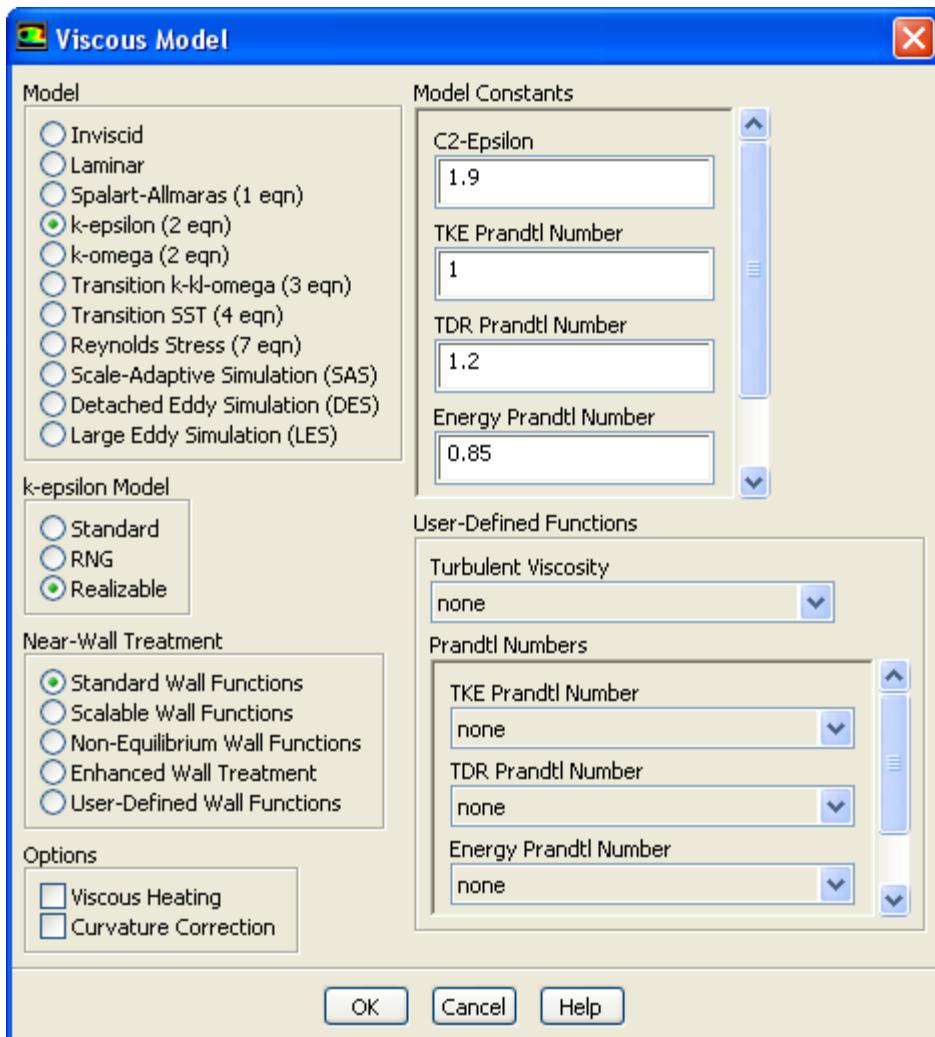
- a. Enable heat transfer by activating the energy equation.

↳ Models → Energy → Edit...



- Enable **Energy Equation** and click **OK** to close the **Energy** dialog box.
- b. Enable the k-epsilon turbulence model.

↳ Models → Viscous → Edit...

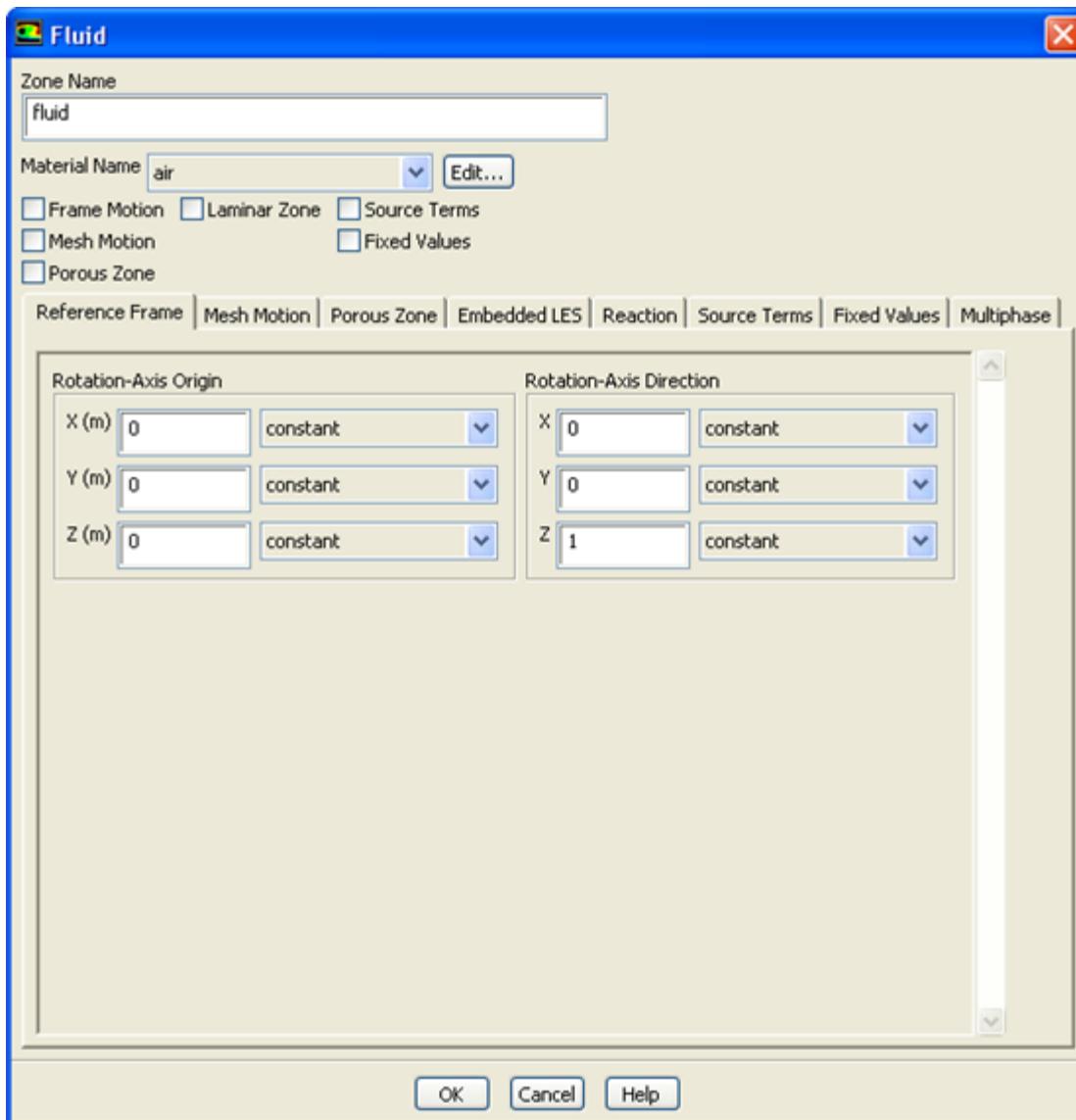


- i. Select **k-epsilon (2 eqn)** in the **Model** list.
  - ii. Select **Realizable** in the **k-epsilon Model** list.
  - iii. Retain the selection of **Standard Wall Functions** in the **Near-Wall Treatment** list.
  - iv. Click **OK** to accept the settings and close the **Viscous Model** dialog box.
3. Set up the cell zone conditions for the CFD analysis.

### Cell Zone Conditions

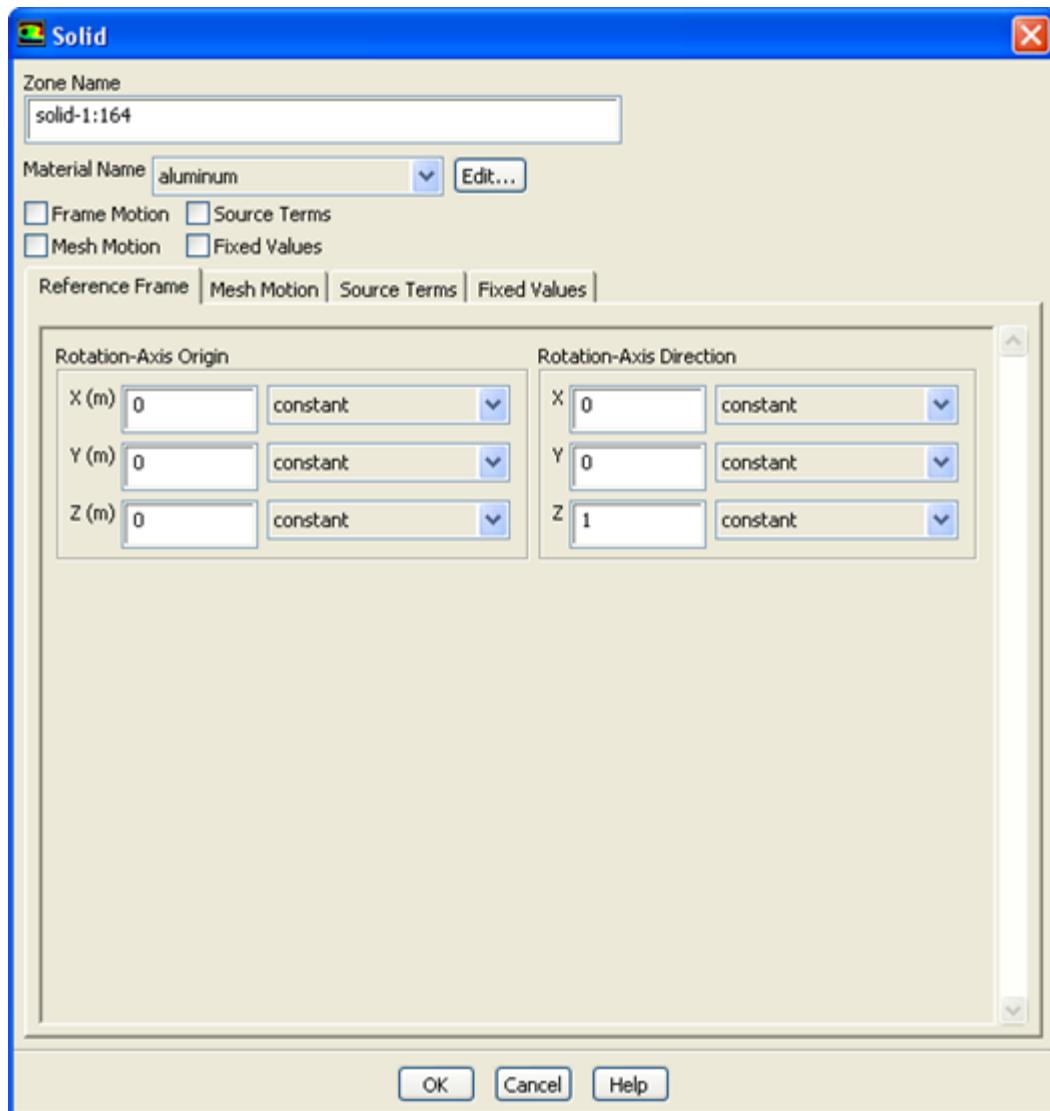
- a. Set the cell zone conditions for the fluid zone.

Cell Zone Conditions → fluid → Edit...



- i. Ensure that **air** is selected in the **Material Name** drop-down list.
  - ii. Retain the defaults for other parameters and click **OK** to close the **Fluid** dialog box.
- b. Set the cell zone conditions for the solid zones.

◆ **Cell Zone Conditions** → █ **solid-1:#** → **Edit...**

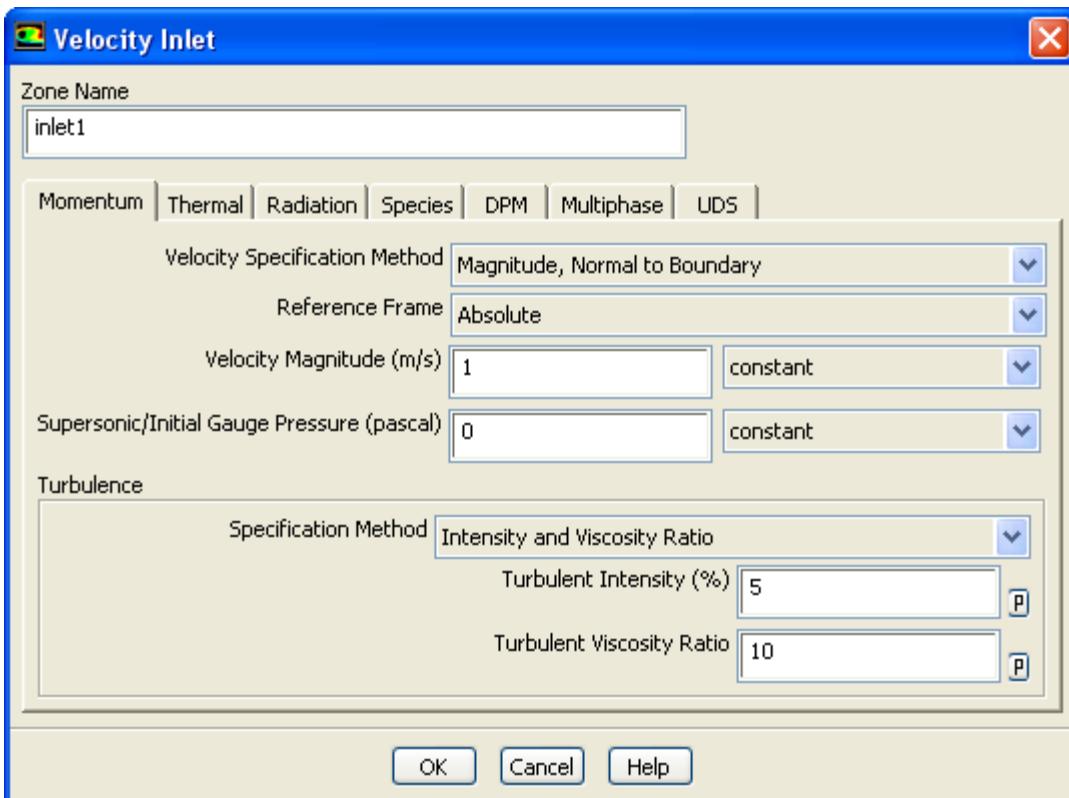


- i. Ensure that **aluminum** is selected in the **Material Name** drop-down list.
  - ii. Retain the defaults for other parameters and click **OK** to close the **Solid** dialog box.
  - iii. Similarly, ensure that the material for all solid zones is set to **aluminum**.
4. Set up the boundary conditions for the CFD analysis.

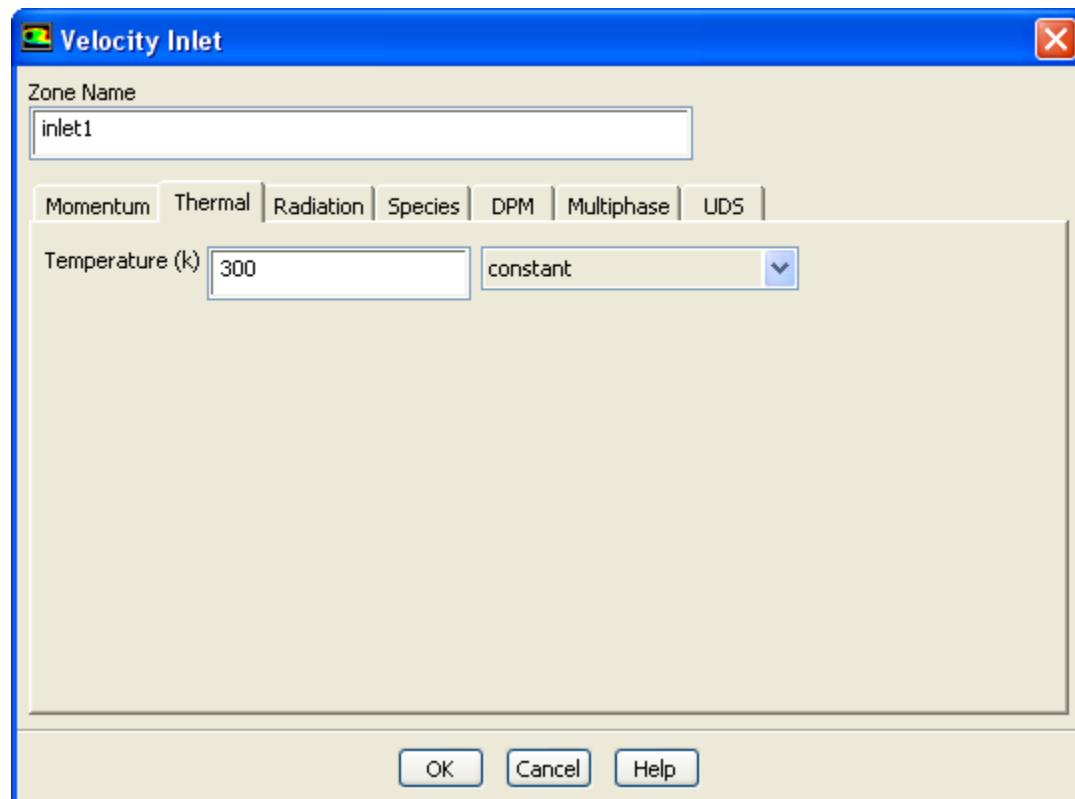
## Boundary Conditions

- a. Set the boundary conditions at the inlet (**inlet1**).

Boundary Conditions → **zone3-baffle** → Edit...



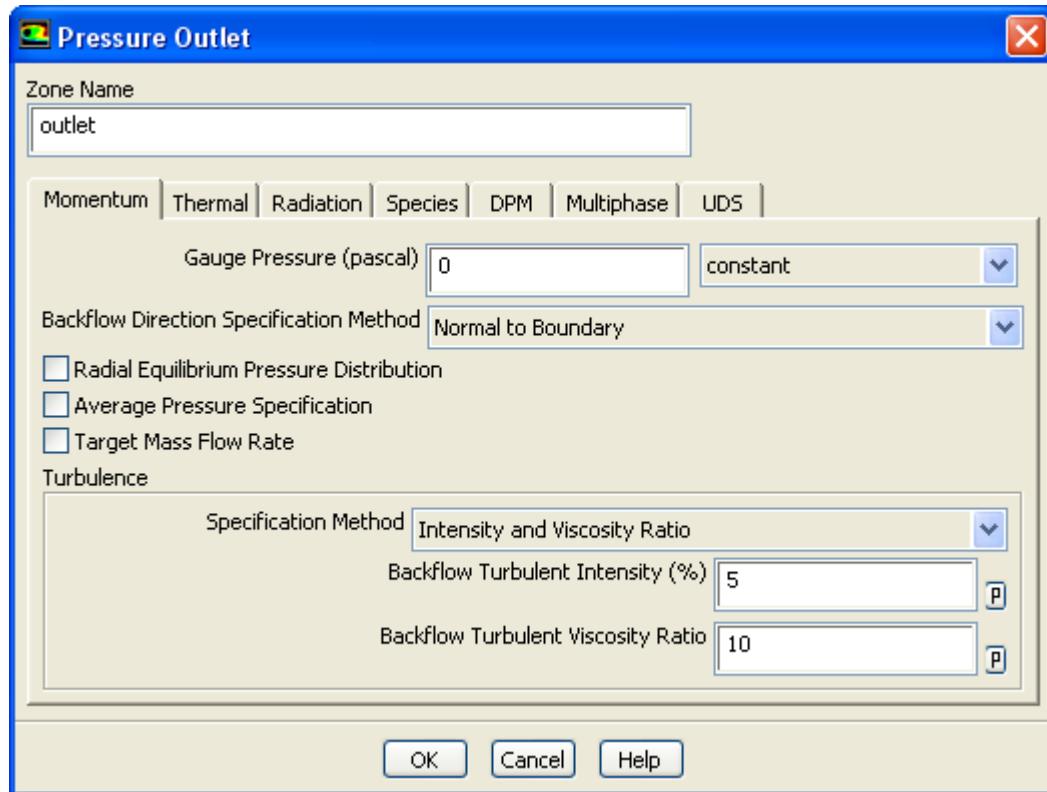
- i. Enter **inlet1** for **Zone Name**.
- ii. Retain the selection of **Magnitude, Normal to Boundary** in the **Velocity Specification Method** drop-down list.
- iii. Enter **1 m/s** for **Velocity Magnitude**.



- iv. Click the **Thermal** tab and enter 300 K for **Temperature**.
  - v. Click **OK** to close the **Velocity Inlet** dialog box.
  - b. Set the boundary conditions at the inlet (**inlet2**).
- Boundary Conditions** → **zone1-baffle** → **Edit...**
- i. Enter **inlet2** for **Zone Name**.
  - ii. Retain the selection of **Magnitude, Normal to Boundary** in the **Velocity Specification Method** drop-down list.
  - iii. Enter 0.5 m/s for **Velocity Magnitude**.
  - iv. Click the **Thermal** tab and enter 350 K for **Temperature**.
  - v. Click **OK** to close the **Velocity Inlet** dialog box.

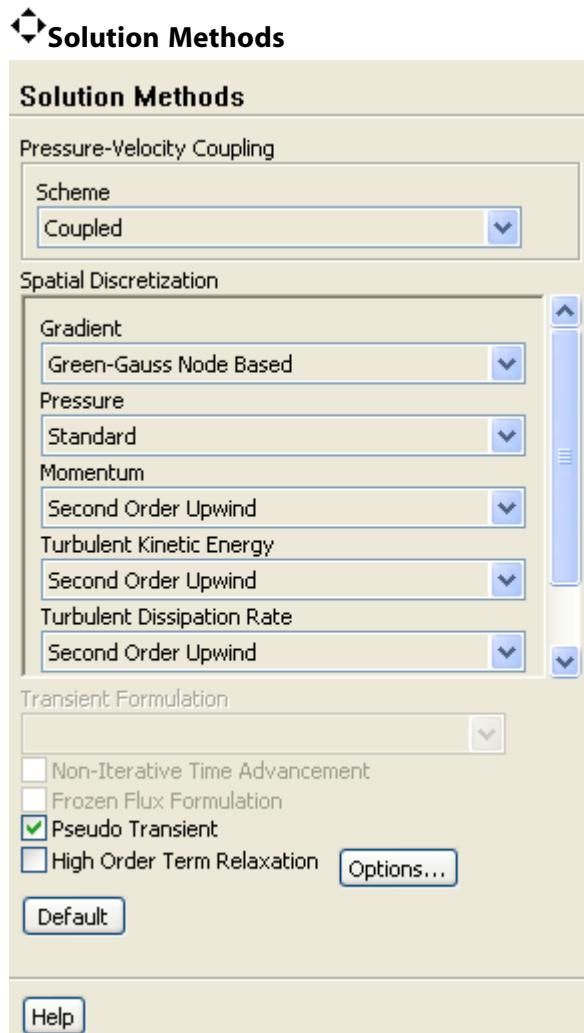
- c. Set the boundary conditions at the outlet (**outlet**).

**Boundary Conditions** → **zone2-baffle** → **Edit...**



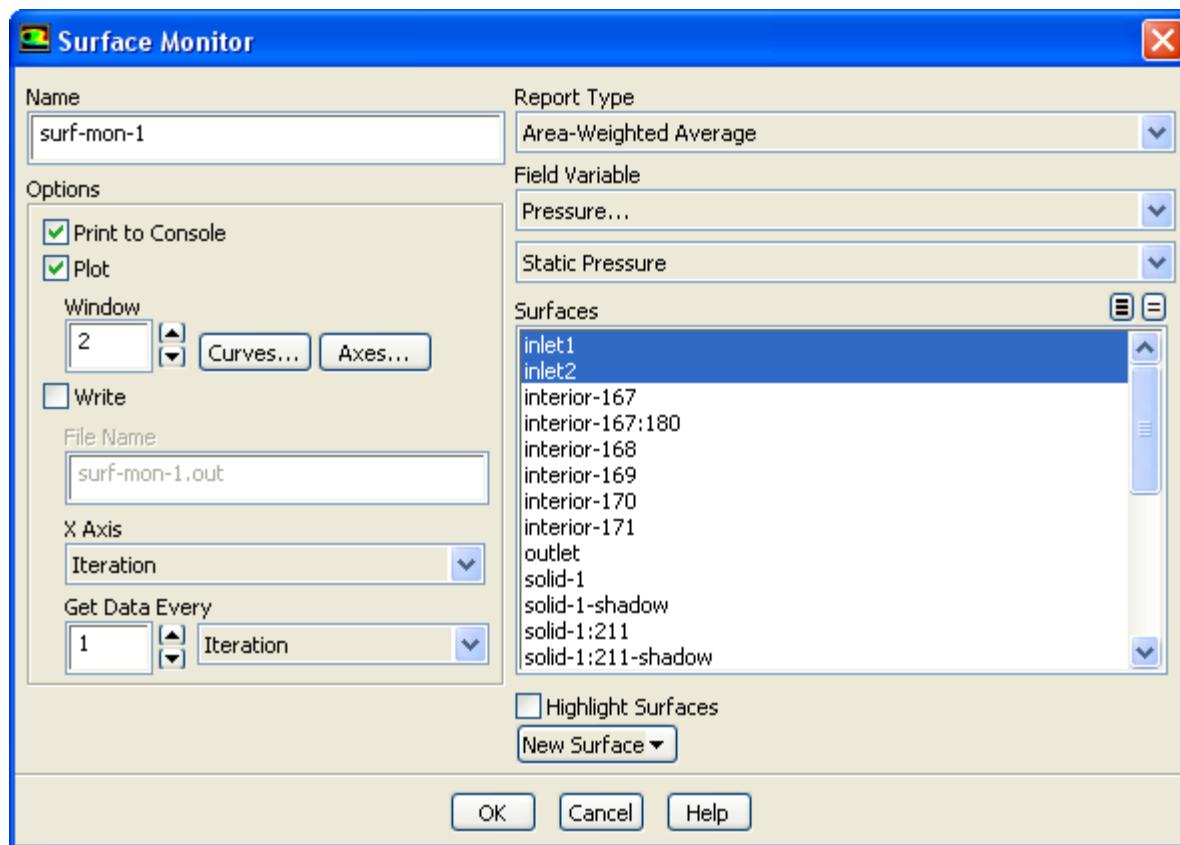
- i. Enter **outlet** for **Zone Name**.
- ii. Retain the default values for all other parameters.
- iii. Click **OK** to close the **Pressure Outlet** dialog box.

5. Set up solution parameters for the CFD solution.



- a. Select **Coupled** in the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box.
  - b. Select **Green-Gauss Node Based** in the **Gradient** drop-down list in the **Spatial Discretization** group box.
  - c. Retain the selection of **Standard** and **Second Order Upwind** in the **Pressure** and **Momentum** drop-down lists, respectively.
  - d. Select **Second Order Upwind** for both **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate**, respectively.
  - e. Enable **Pseudo Transient**.
6. Create a surface monitor at the inlets.

**Monitors (Surface Monitors)** → **Create...**



- a. Enable **Plot** for **surf-mon-1**.
  - b. Select **Area-Weighted Average** in the **Report Type** drop-down list.
  - c. Retain the selection of **Pressure...** and **Static Pressure** in the **Field Variable** drop-down lists.
  - d. Select **inlet1** and **inlet2** in the **Surfaces** selection list.
  - e. Click **OK** to save the surface monitor settings and close the **Surface Monitor** dialog box.
7. Convert the mesh to polyhedra.

- a. Set the feature angle for polyhedra conversion to 60.

```
> (rpsetvar 'polyhedra/conversion-feature-angle 60)
```

#### Note

This angle value will result in less dual cells on the boundary after the conversion to polyhedra. You can also try with the default settings and compare the results.

- b. Convert the mesh domain to polyhedra.

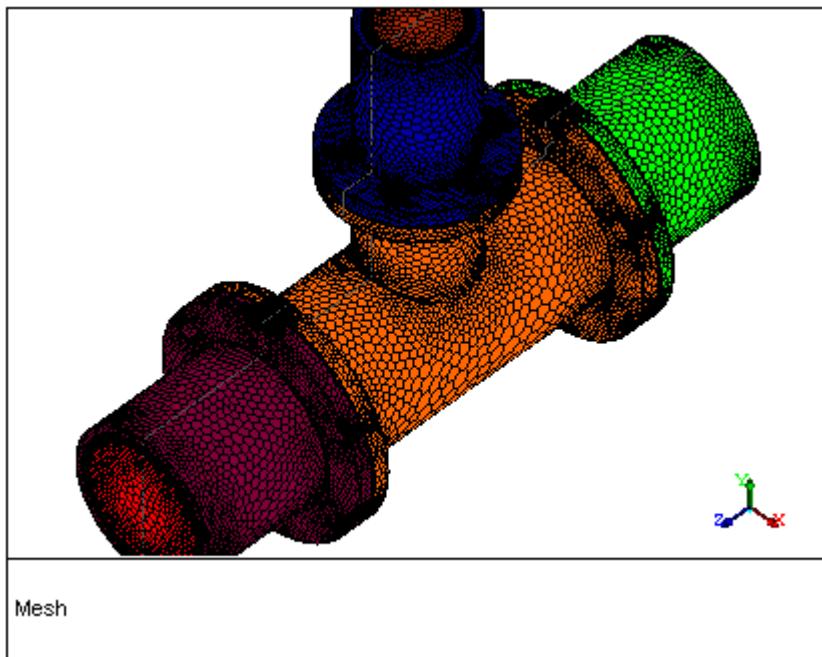
#### **Mesh → Polyhedra → Convert Domain**

- c. Display the polyhedral mesh.

## ◆ General → Display...

The polyhedral mesh is shown in [Figure 10.6: Polyhedral Mesh \(p. 251\)](#).

**Figure 10.6: Polyhedral Mesh**



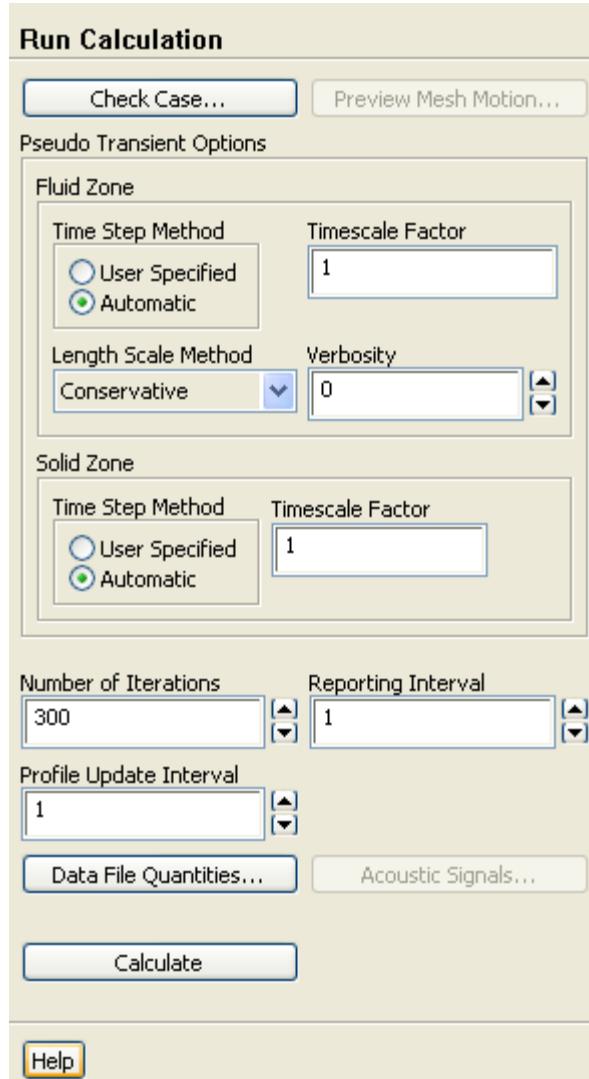
8. Initialize the flow field.

## ◆ Solution Initialization



- a. Retain the selection of **Hybrid Initialization** in the **Initialization Methods** list.
- b. Click **Initialize**.
9. Calculate the solution.

## ◆ Run Calculation

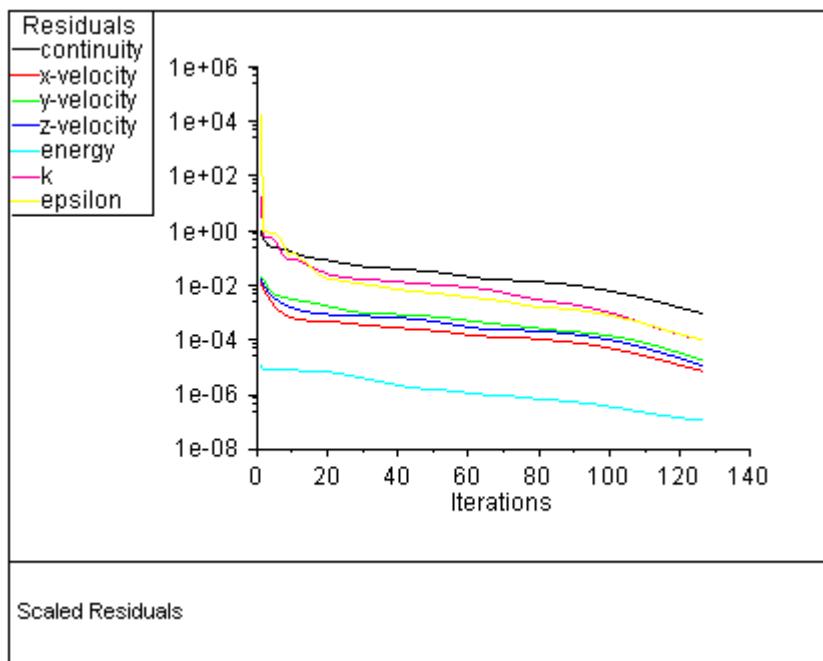
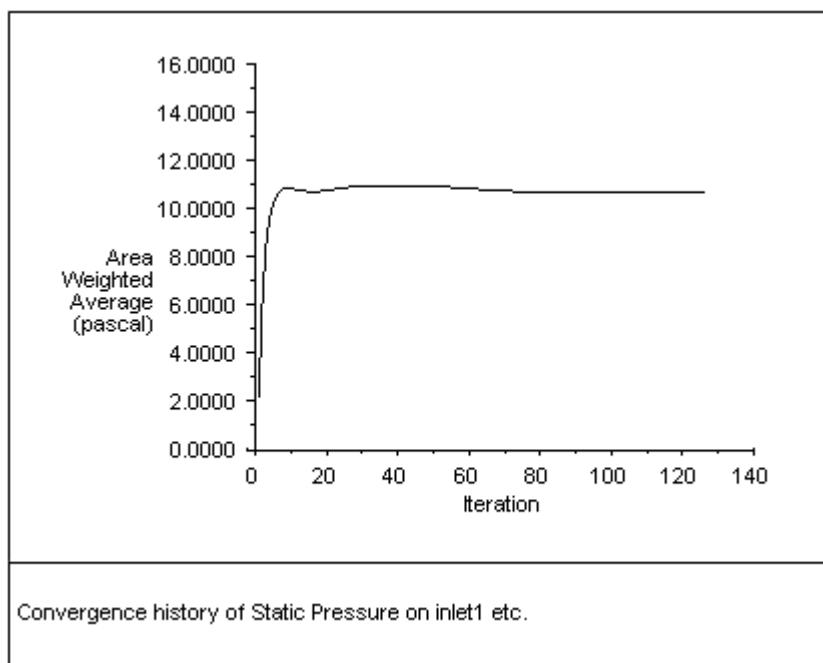


a. Enter 300 for **Number of Iterations**.

b. Click **Calculate**.

The solution converges in approximately 130 iterations.

10. Examine the plots for convergence ([Figure 10.7: Residuals \(p. 253\)](#) and [Figure 10.8: Convergence History of Static Pressure \(p. 253\)](#)).

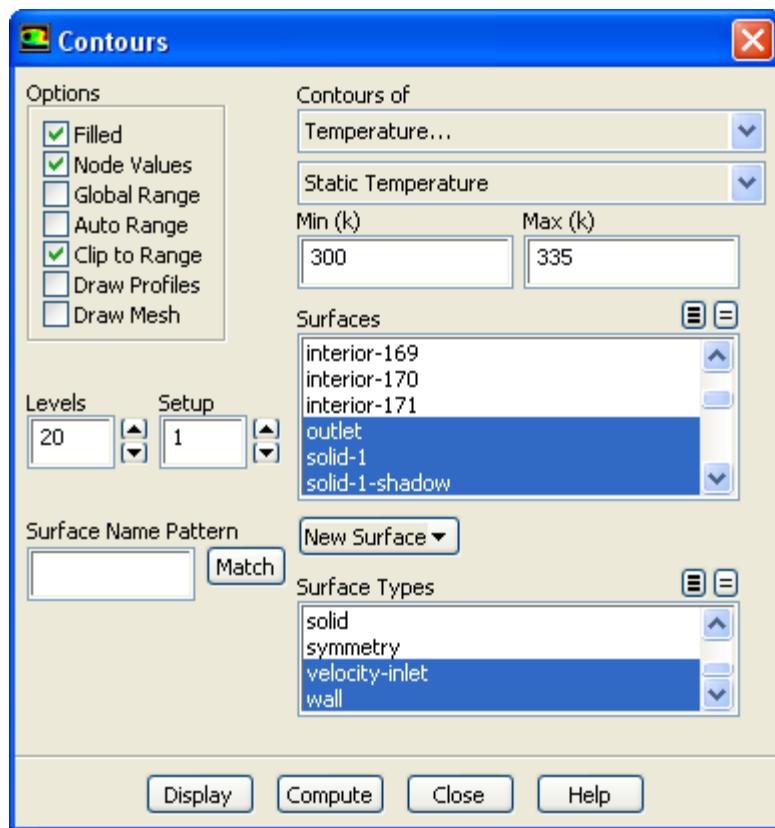
**Figure 10.7: Residuals****Figure 10.8: Convergence History of Static Pressure**

11. Save the case and data files (`mixer.cas.gz` and `mixer.dat.gz`).

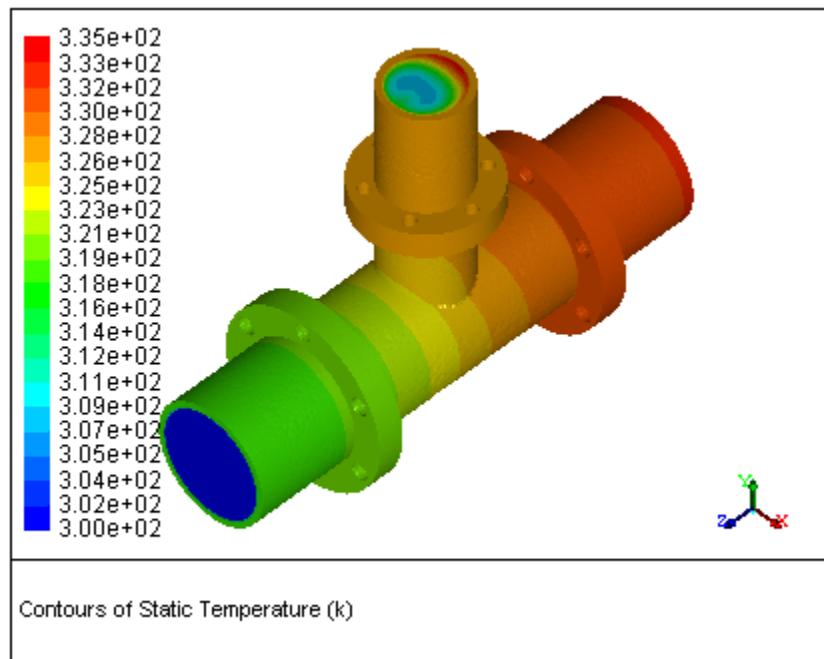
**File → Write → Case & Data...**

12. Examine the results.

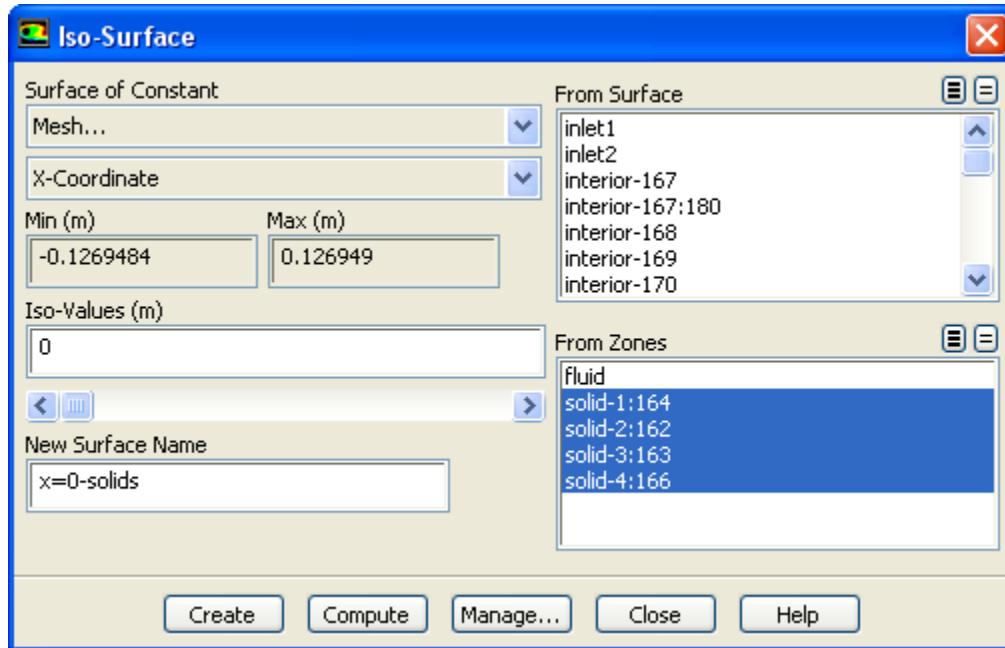
**Graphics and Animations** → **Contours** → **Set Up...**



- a. Display filled contours of temperature on the inlet, outlet, and wall surfaces.
  - i. Select **Temperature...** and **Static Temperature** in the **Contours of** drop-down lists.
  - ii. Select **pressure-outlet**, **velocity-inlet**, and **wall** in the **Surface Types** selection list.
  - iii. Enable **Filled** and **Node Values** in the **Options** group box and disable **Global Range** and **Auto Range**.
  - iv. Enter values of 300 K and 335 K for **Min** and **Max**, respectively.
  - v. Click **Display** (Figure 10.9: Contours of Static Temperature (p. 255)).

**Figure 10.9: Contours of Static Temperature**

- b. Display filled contours of temperature on a cut plane through the solid surfaces.
- i. Select **Iso-Surface...** in the **New Surface** drop-down list to open the **Iso-Surface** dialog box.

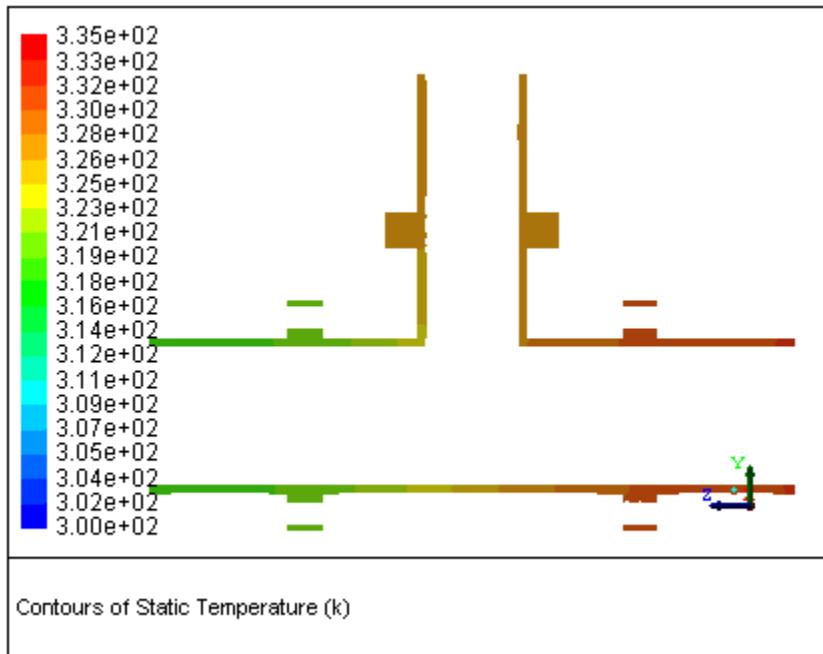


- ii. Select **Mesh...** and **X-Coordinate** in the **Surface of Constant** drop-down lists.
- iii. Select all the solid zones in the **From Zones** selection list.
- iv. Enter 0 for **Iso-Values**.
- v. Enter **x=0-solids** for **New Surface Name**.

- vi. Click **Create** and close the **Iso-Surface** dialog box.
- vii. Deselect the previously selected surfaces in the **Contours** dialog box and select **x=0-solids** in the **Surfaces** selection list.
- viii. Retain the other previous selections and click **Display**.
- ix. Set the right view ([Figure 10.10: Contours of Static Temperature on Cut Plane Through Solids \(p. 256\)](#)).

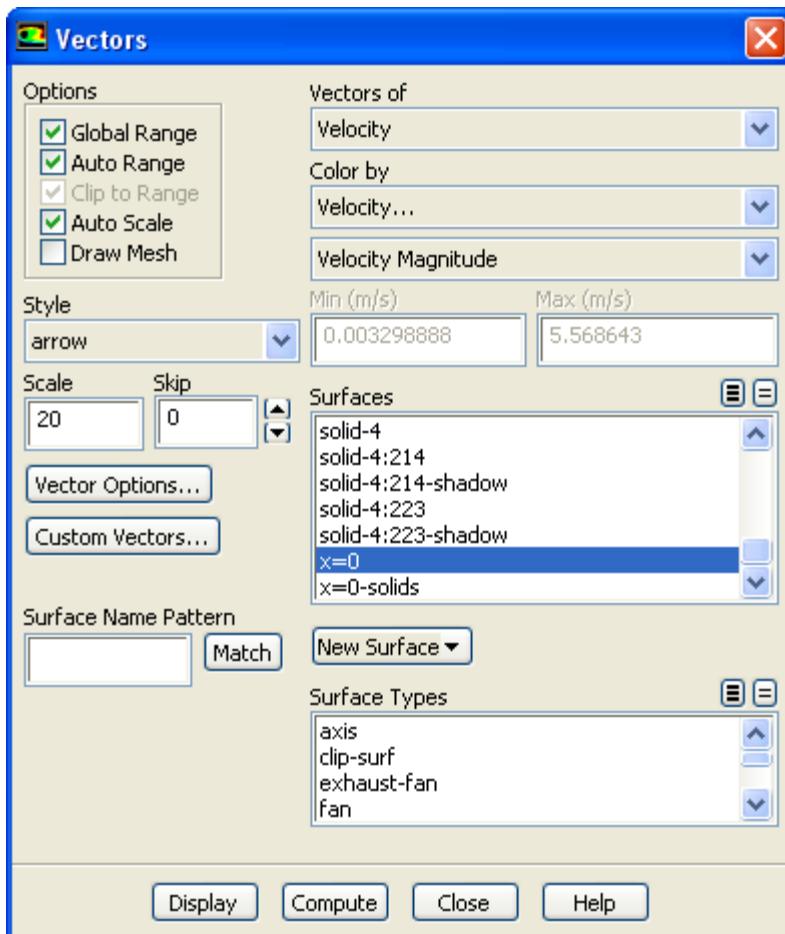
**Display → Views...**

**Figure 10.10: Contours of Static Temperature on Cut Plane Through Solids**



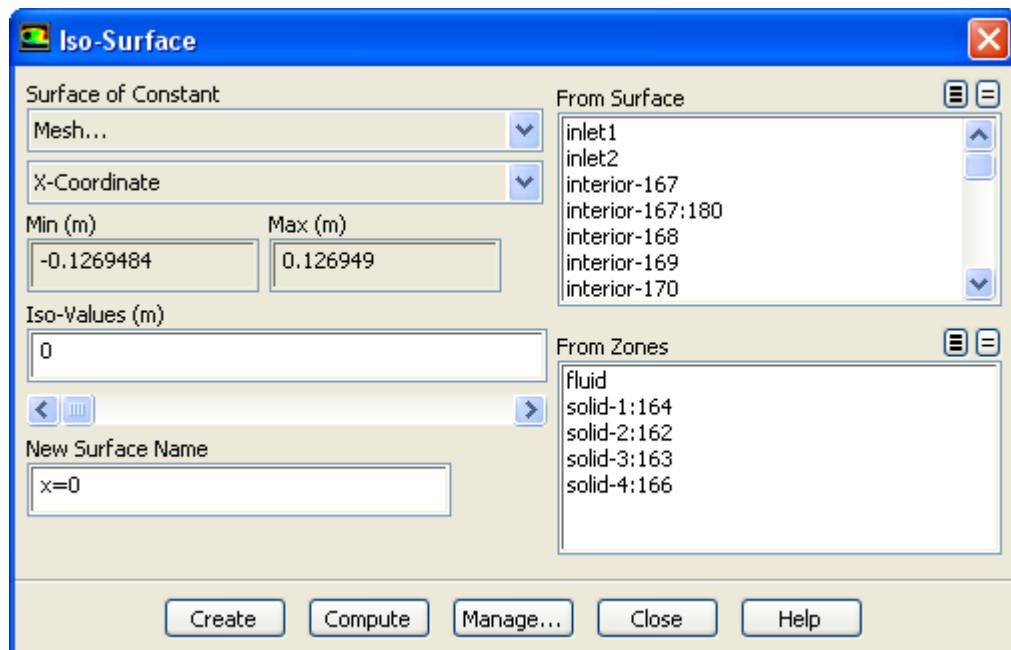
- x. Close the **Contours** dialog box.
- c. Display velocity vectors on the x=0 plane.

**Graphics and Animations** → **Vectors** → **Set Up...**



i. Create an iso-surface for  $x=0$ .

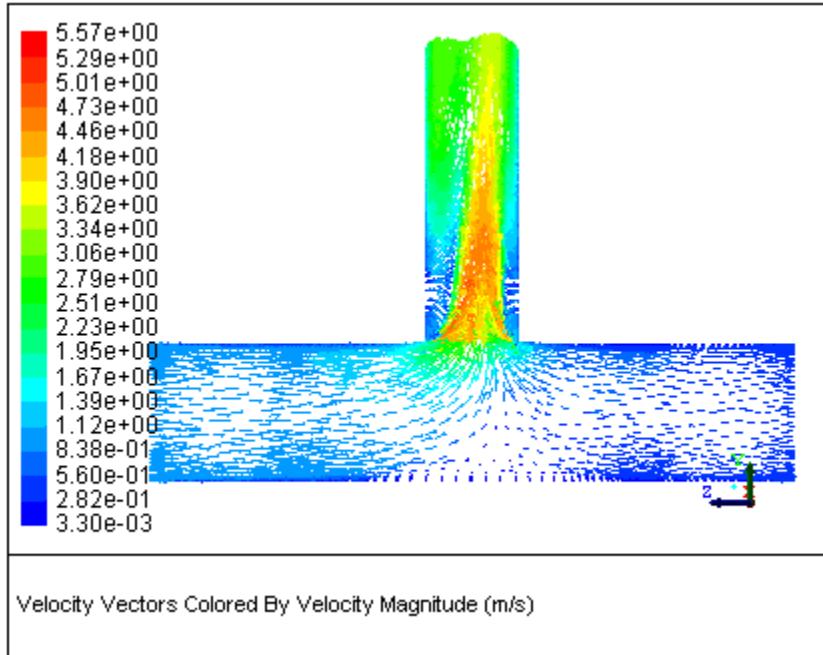
- A. Select **Iso-Surface...** in the **New Surface** drop-down list to open the **Iso-Surface** dialog box.



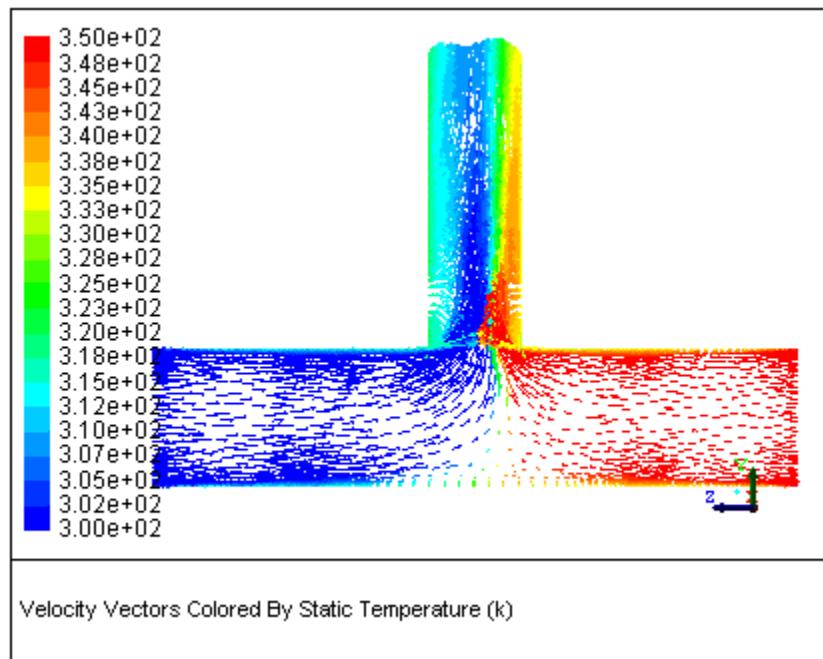
- B. Select **Mesh...** and **X-Coordinate** in the **Surface of Constant** drop-down lists.

- C. Enter 0 for **Iso-Values**.
  - D. Enter  $x=0$  for **New Surface Name**.
  - E. Click **Create** and close the **Iso-Surface** dialog box.
- 
- ii. Select  $x=0$  in the **Surfaces** selection list in the **Vectors** dialog box.
  - iii. Enter 20 for **Scale** and click **Display** ([Figure 10.11: Velocity Vectors on  \$x=0\$  Plane \(p. 258\)](#)).

**Figure 10.11: Velocity Vectors on  $x=0$  Plane**

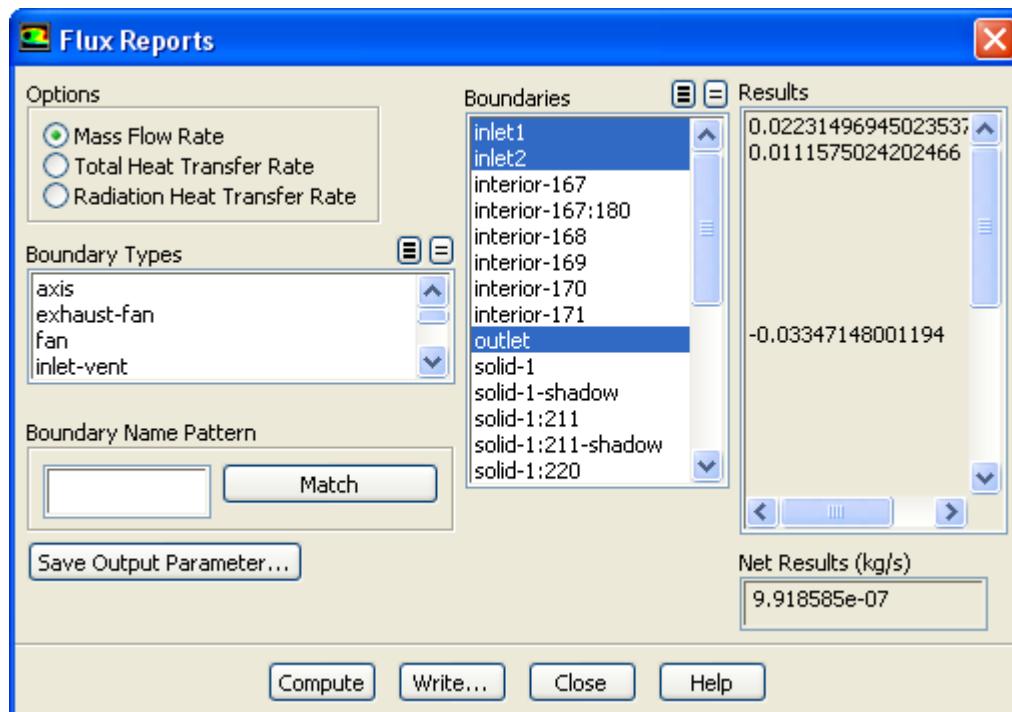


- iv. Select **Temperature...** and **Static Temperature** in the **Color by** drop-down lists.
- v. Retain the other settings and click **Display** ([Figure 10.12: Vectors of Temperature on  \$x=0\$  Plane \(p. 259\)](#)).

**Figure 10.12: Vectors of Temperature on x=0 Plane**

- vi. Close the **Vectors** dialog box.
- d. Check the mass flux balance.

Reports → Fluxes → Set Up...



- i. Retain the selection of **Mass Flow Rate** in the **Options** list.
- ii. Select **inlet1**, **inlet2**, and **outlet** in the **Boundaries** selection list.

iii. Click **Compute**.

The net mass imbalance should be no more than a small fraction (say, 0.5%) of the total flux through the system.

iv. Close the **Flux Reports** dialog box.

## 10.10. Summary

This tutorial demonstrated the procedure for generating the volume mesh for a mixer geometry using the parallel version of ANSYS FLUENT. You defined size functions, manipulated the imported objects, and used the sewing operation to generate a conformal surface mesh. You then created the volume mesh based on the mesh object and material point, and transferred the mesh to the solution mode in ANSYS FLUENT. You set up the CFD solution and examined the results in the solution mode.

---

## Chapter 11:Cavity Remeshing

---

Cavity remeshing is useful in parametric studies as it allows you to add, remove, or replace different parts of an existing mesh. This tutorial demonstrates the procedure for replacing an object in the existing mesh with another by creating a cavity and remeshing it.

This tutorial demonstrates how to do the following:

- Create and remesh a cavity to replace the mirror in a tetrahedral mesh.
- Create and remesh a cavity in a hybrid mesh (tetrahedra and prisms) having a single fluid zone.
- Create and remesh a cavity in a hybrid mesh (tetrahedra and prisms) having multiple fluid zones.
- Create and remesh a cavity in a hexcore mesh.

### 11.1. Prerequisites

This tutorial assumes that you have some experience with the meshing mode in ANSYS FLUENT, and that you are familiar with the graphical user interface.

### 11.2. Preparation

To access tutorials and their input files on the ANSYS Customer Portal, go to <http://support.ansys.com/training>.

To prepare for running this tutorial:

1. Download the tutorial input file (`cavity.zip`) for the tutorial.
2. Unzip `cavity.zip`.

The files, `sedan_tetra.msh.gz`, `sedan_hyb-1zone.msh.gz`, `sedan_hyb-2zones.msh.gz`, `sedan_hexcore.msh.gz`, `mirror.msh.gz`, and `sedan_hyb-2zones-cavity.jou`, can be found in the `cavity` folder created on unzipping the file.

3. Start ANSYS FLUENT in meshing mode. For detailed steps, refer to [Starting ANSYS FLUENT in Meshing Mode \(p. 4\)](#).

### 11.3. Cavity Remeshing For a Tetrahedral Mesh

#### Read and Display Mesh

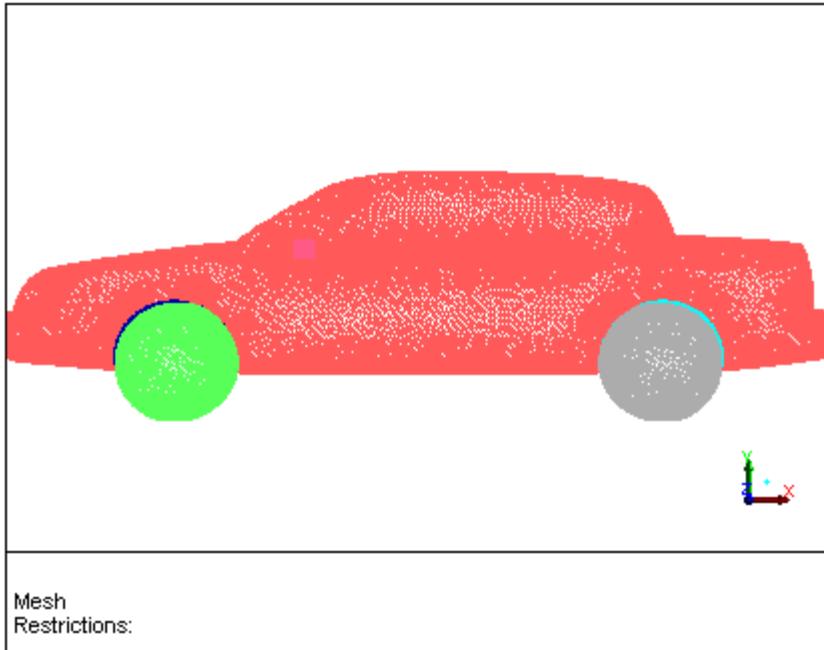
1. Read the mesh file.

**File → Read → Mesh...**

- a. Select **sedan\_tetra.msh.gz**.

- b. Click **OK**.
2. Examine the mesh.
- Display → Grid...**
- a. Select **car, mirror-old, wheel-arch-front, wheel-arch-rear, wheel-front, and wheel-rear**, in the **Face Zones** selection list.
  - b. Click **Display** ([Figure 11.1: Boundary Mesh \(p. 262\)](#)).

**Figure 11.1: Boundary Mesh**



- c. Click the **Cells** tab and enable **All** in the **Options** group box.
- d. Select **fluid** in the **Cell Zone Groups** selection list.
- e. Click the **Bounds** tab and enable **Limit by X** in the **X Range** group box.
- f. Enter  $-0.37$  for both **Minimum** and **Maximum** in the **X Range** group box.
- g. Click the **Attributes** tab  
and enable **Filled** and **Lights**.
- h. Click **Display**.
- i. Display the **left** view.

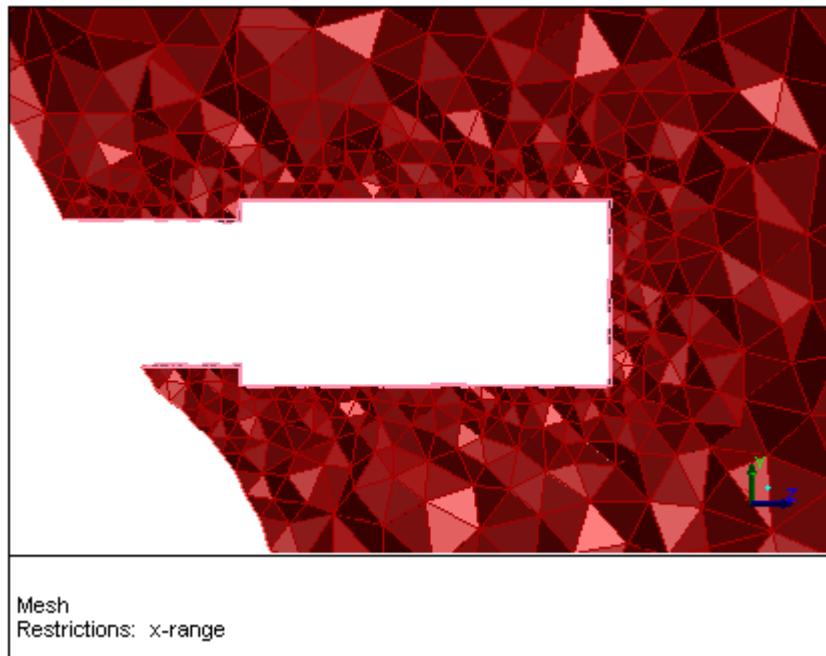
**Display → Views...**

- i. Select **left** in the **Views** list.
- ii. Click **Apply** and close the **Views** dialog box.

- j. Zoom in and examine the mesh around the mirror ([Figure 11.2: Tetrahedral Mesh Near the Mirror \(p. 263\)](#)).

The mesh comprises tetrahedral cells.

**Figure 11.2: Tetrahedral Mesh Near the Mirror**



## Import and Connect the New Mirror

This section demonstrates replacing the mirror (**mirror-old**) by a modified geometry. The node distribution on the boundary on the two mirrors is identical.

1. Read the mesh file (**mirror.msh.gz**).

**File → Read → Mesh...**

- a. Enable **Append File(s)**.
- b. Select **mirror.msh.gz** in the **Files** list and click **OK**.

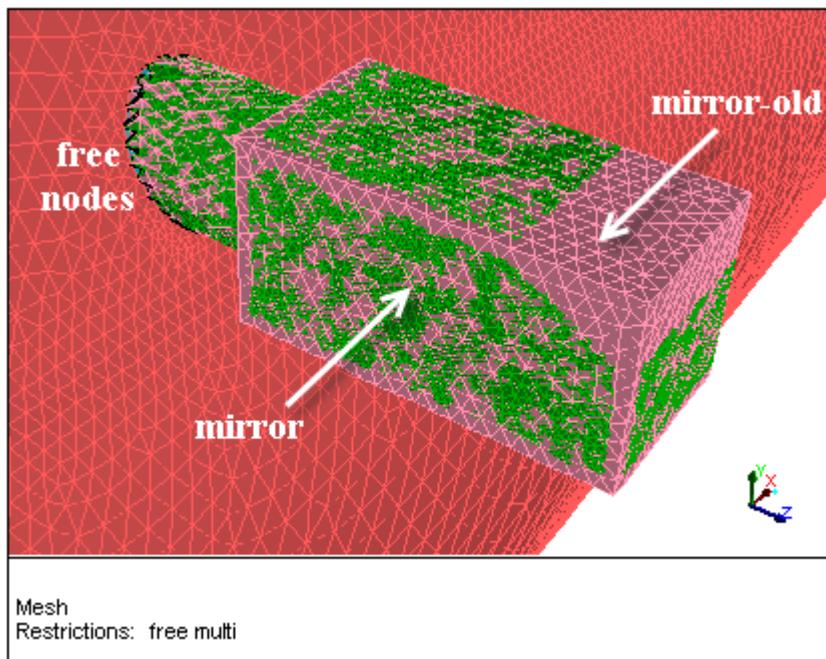
2. Verify that the new mirror is appropriately positioned.

**Display → Grid...**

- a. Select **car**, **mirror**, and **mirror-old** in the **Face Zones** selection list in the **Faces** tab.
- b. Enable **Free** and **Multi** in the **Options** group box.

Make sure that the fluid zone is deselected in the **Cell Zones** selection list in the **Cells** tab. Click **Reset** in the **Bounds** tab.

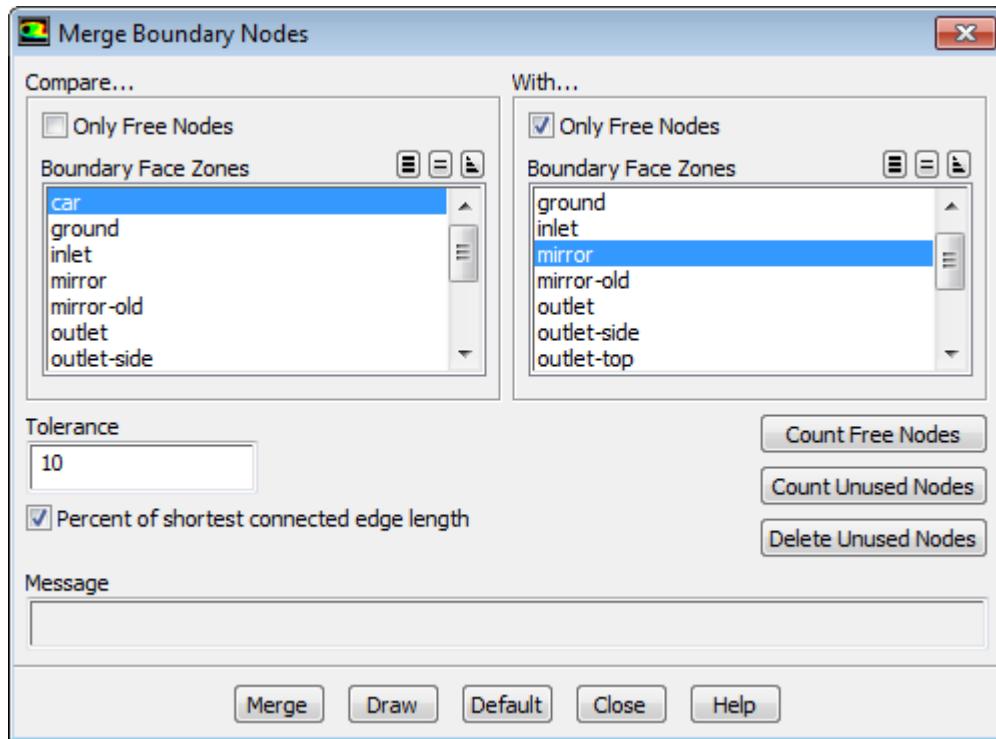
- c. Click **Display** ([Figure 11.3: Free Nodes on the Mirror \(p. 264\)](#)).

**Figure 11.3: Free Nodes on the Mirror**

In Figure 11.3: Free Nodes on the Mirror (p. 264), you can see the free nodes on the boundary of **mirror**. This indicates that the mirror is not connected to the car.

3. Connect the mirror to the car.

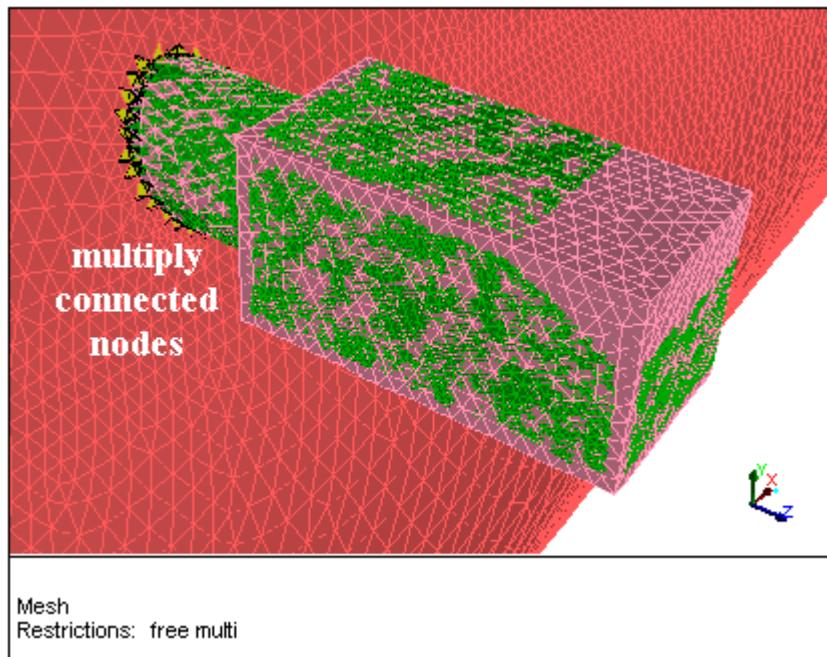
#### **Boundary → Merge Nodes...**



- a. Select only **car** in the **Boundary Face Zones** selection list in the **Compare...** group box and disable **Only Free Nodes**.

- b. Select only **mirror** in the **Boundary Face Zones** selection list in the **With...** group box and retain **Only Free Nodes**.
- c. Enable **Percent of shortest connected edge length** and enter 10 for **Tolerance**.
- d. Click **Merge**.
- e. Click **Display** in the **Display Grid** dialog box.

**Figure 11.4: Multiply Connected Nodes on the Mirror**



In Figure 11.4: Multiply Connected Nodes on the Mirror (p. 265), you can see that there is a multiple connection between the mirrors and the car.

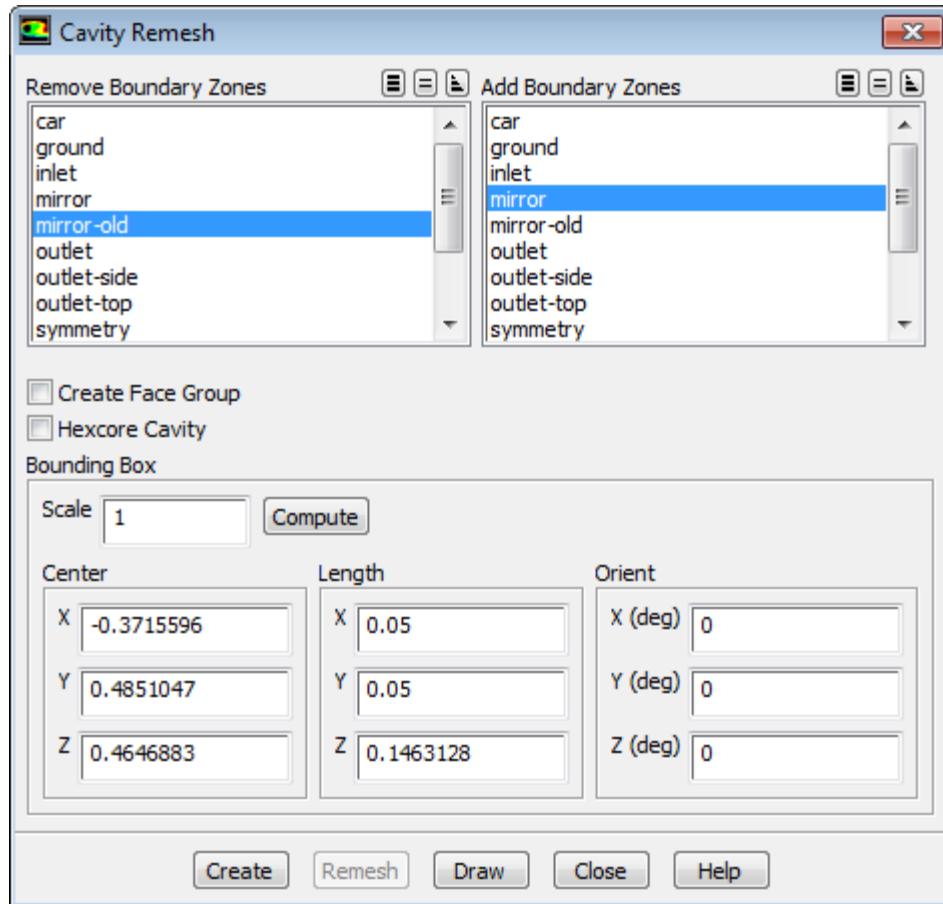
- f. Close the **Merge Boundary Nodes** dialog box.

## Replace the Mirror

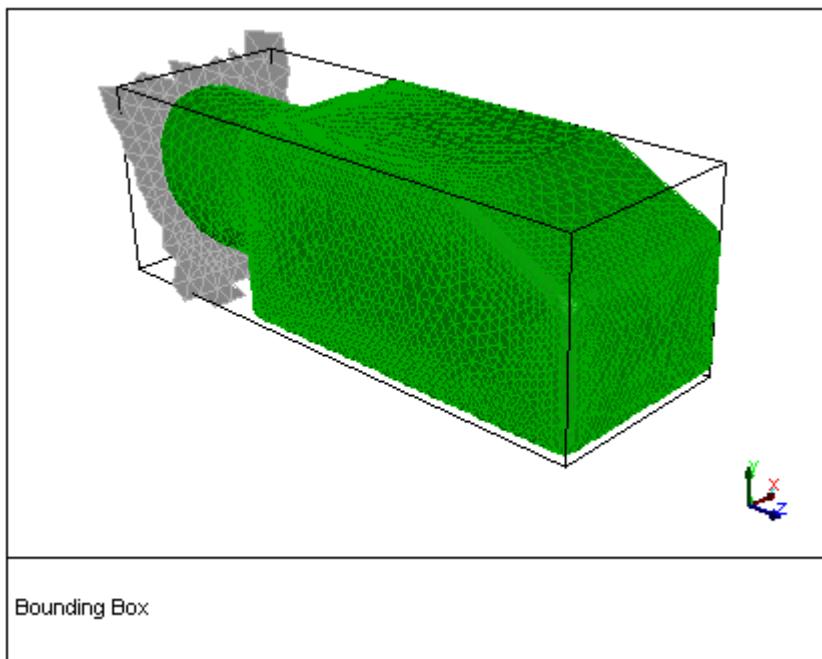
This section demonstrates replacing the existing mirror with a new one, without resetting the volume mesh. To do this, you will remove a portion of the mesh around the mirror.

1. Replace the mirror.

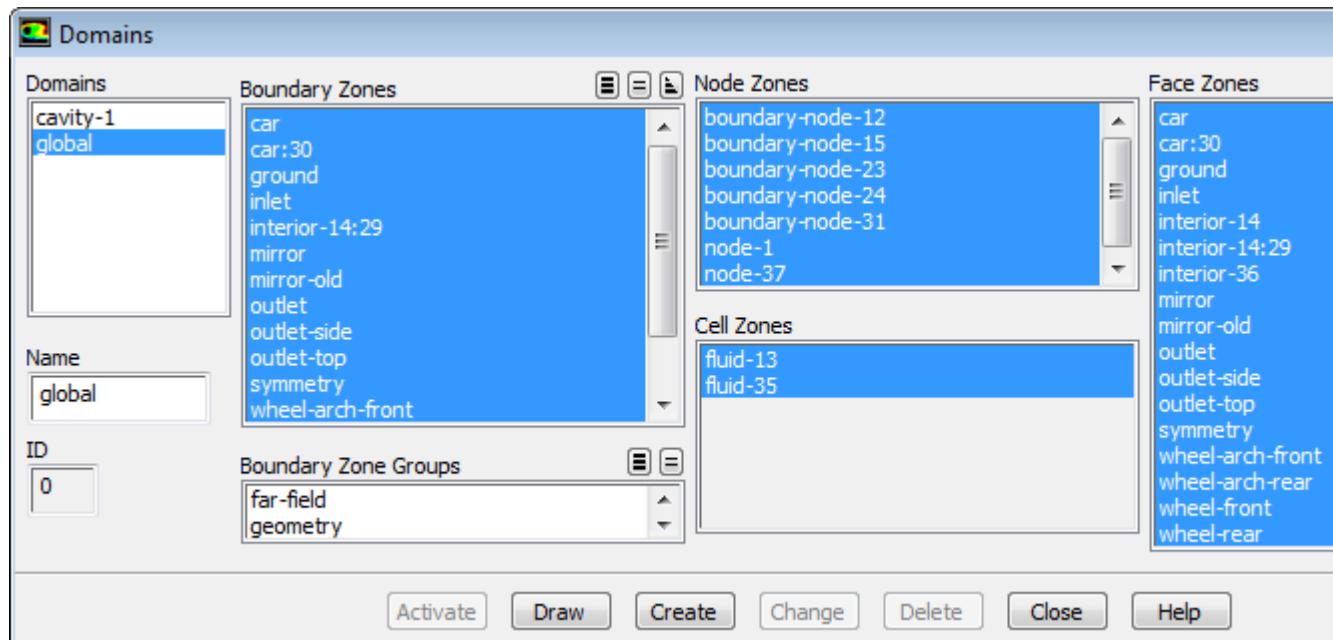
**Mesh → Tools → Cavity Remesh...**



- a. Select **mirror-old** in the **Remove Boundary Zones** selection list.
- b. Select **mirror** in the **Add Boundary Zones** selection list.
- c. Enter 1 for **Scale**.
- d. Click **Compute**.
- e. Click **Draw** to verify that the cavity defined fits the mirror to be replaced.
- f. Click **Create**.
- The car zone is separated into two—one inside (**car:#**) and the other outside the cavity region, an interior surface (**interior-#:#**) is created at the boundary of the cavity, and a domain is defined and activated for the cavity.
- g. Select **car:#** and **mirror** in the **Face Zones** list in the **Display Grid** dialog box.
- h. Click **Display** and close the **Display Grid** dialog box.
- i. Click **Draw** in the **Cavity Remesh** dialog box (Figure 11.5: Cavity Domain Before Meshing (p. 267)).

**Figure 11.5: Cavity Domain Before Meshing**

- j. Close the **Cavity Remesh** dialog box.
2. Mesh the cavity with tetrahedral cells.  
**Mesh → Tet...**
  - a. Retain the default settings and click **Init & Refine**.
  - b. Close the **Tet** dialog box.
3. Activate the global domain.  
**Mesh → Domains...**

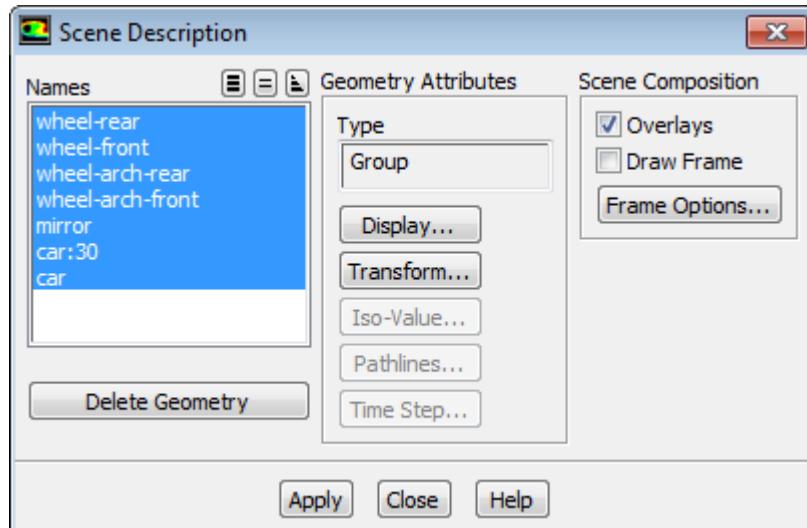


- a. Select **global** in the **Domains** list.
- b. Click **Activate** and close the **Domains** dialog box.
4. Examine the mesh.

#### Display → Grid...

- a. Select **car**, **car:#**, **mirror**, **wheel-arch-front**, **wheel-arch-rear**, **wheel-front**, and **wheel-rear** in the **Face Zones** selection list in the **Faces** tab of the **Display Grid** dialog box.
- b. Disable **Free** and **Multi** in the **Options** group box.
- c. Click **Display**.
- d. Enable the overlaying of graphics.

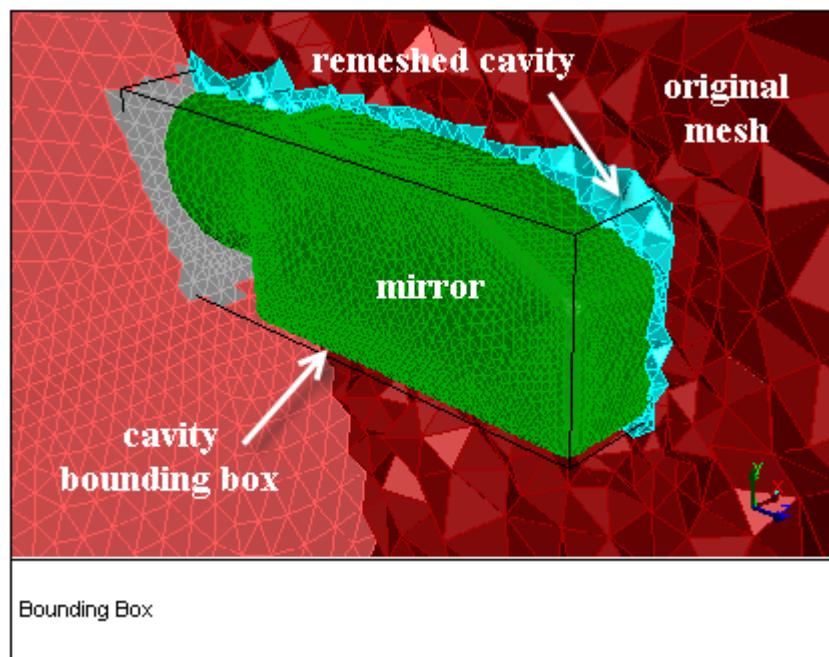
#### Display → Scene...



- i. Select all the zones in the **Names** selection list.
- ii. Enable **Overlays** in the **Scene Composition** group box.
- iii. Click **Apply** and close the **Scene Description** dialog box.
- e. Click the **Cells** tab in the **Display Grid** dialog box and enable **All** in the **Options** group box.
- f. Select the fluid zones in the **Cell Zones** selection list.
- g. Click the **Bounds** tab and enable **Limit by X** in the **X Range** group box.
- h. Enter  $-0.37$  for both **Minimum** and **Maximum** in the **X Range** group box.
- i. Click **Display**.
- j. Click **Draw** in the **Cavity Remesh** dialog box (Figure 11.6: Cavity Remeshed With Tetrahedral Cells (p. 269)).

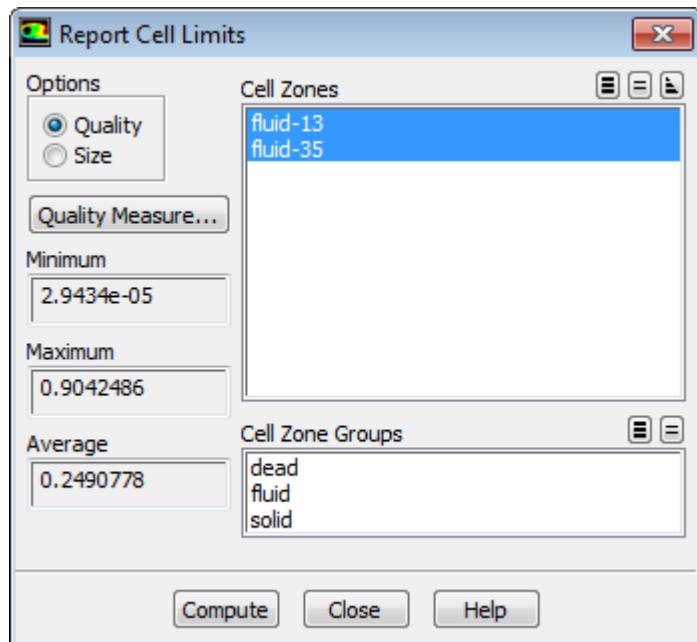
**Mesh → Tools → Cavity Remesh...**

**Figure 11.6: Cavity Remeshed With Tetrahedral Cells**



5. Check the mesh quality.

**Report → Cell Limits...**



- Select all zones in the **Cell Zones** selection list.
- Click **Compute**.

The maximum skewness reported is around 0.904. The exact value may vary slightly on different platforms.

- Merge the cavity domain with the original mesh.

```
>/mesh/cavity/merge-cavity
Insert domain name/id [cavity-1] cavity-1
Into domain name/id [global] global
Merging cell zone fluid-# (id #) with fluid-#
Merging face zone interior-#:# (id #) with interior-#
Merging face zone car:# (id #) with car
```

where, # denotes the respective zone IDs.

- Delete the old mirror.

**Boundary** → **Manage...**

- Select **mirror-old** in the **Face Zones** selection list.
- Select **Delete** in the **Options** list and retain the **Delete Nodes** option.
- Click **Apply**.

A **Question** dialog box will appear, asking you to confirm if you want to delete the selected zone(s).

- Click **Yes** in the **Question** dialog box.
  - Close the **Manage Face Zones** dialog box.
- Check the mesh.

**Mesh → Check**

9. Save the mesh (`sedan-tet-cavity.msh.gz`).

**File → Write → Mesh...**

## 11.4. Cavity Remeshing For a Hybrid Mesh (Tetrahedra and Prisms) Having a Single Fluid Zone

1. Read the mesh file (`sedan_hyb-1zone.msh.gz`).

**File → Read → Mesh...**

- a. Select `sedan_hyb-1zone.msh.gz` in the **Files** list.
- b. Click **OK**.

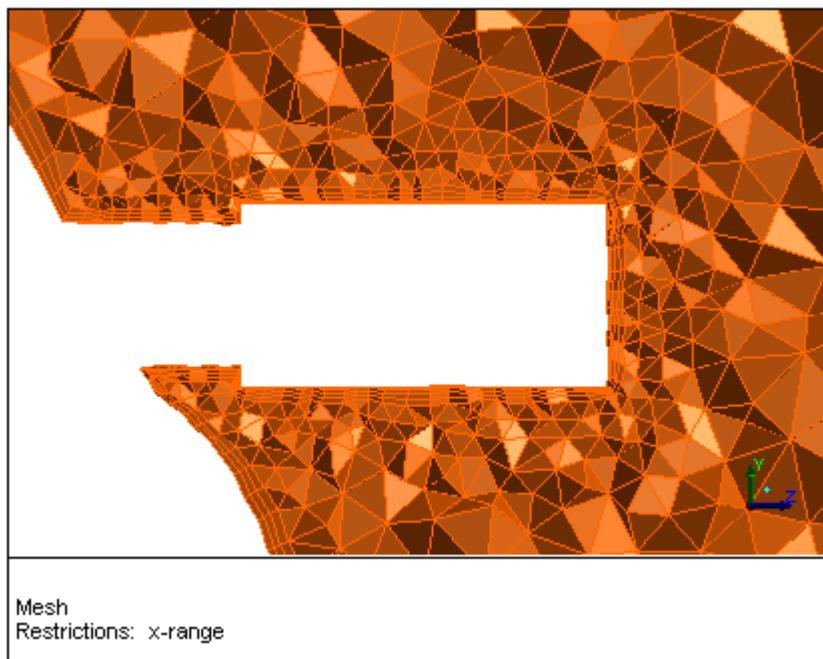
2. Examine the mesh.

**Display → Grid...**

- a. Click the **Cells** tab and enable **All** in the **Options** group box.
- b. Select **fluid** in the **Cell Zone Groups** selection list.
- c. Click the **Bounds** tab and enable **Limit by X** in the **X Range** group box.
- d. Enter  $-0.37$  for both **Minimum** and **Maximum** in the **X Range** group box.
- e. Click the **Attributes** tab and enable **Filled** and **Lights**.
- f. Click **Display**.
- g. Display the **left** view.

**Display → Views...**

- h. Zoom in and examine the mesh around the mirror (Figure 11.7: Hybrid Mesh Near the Mirror (p. 272)).

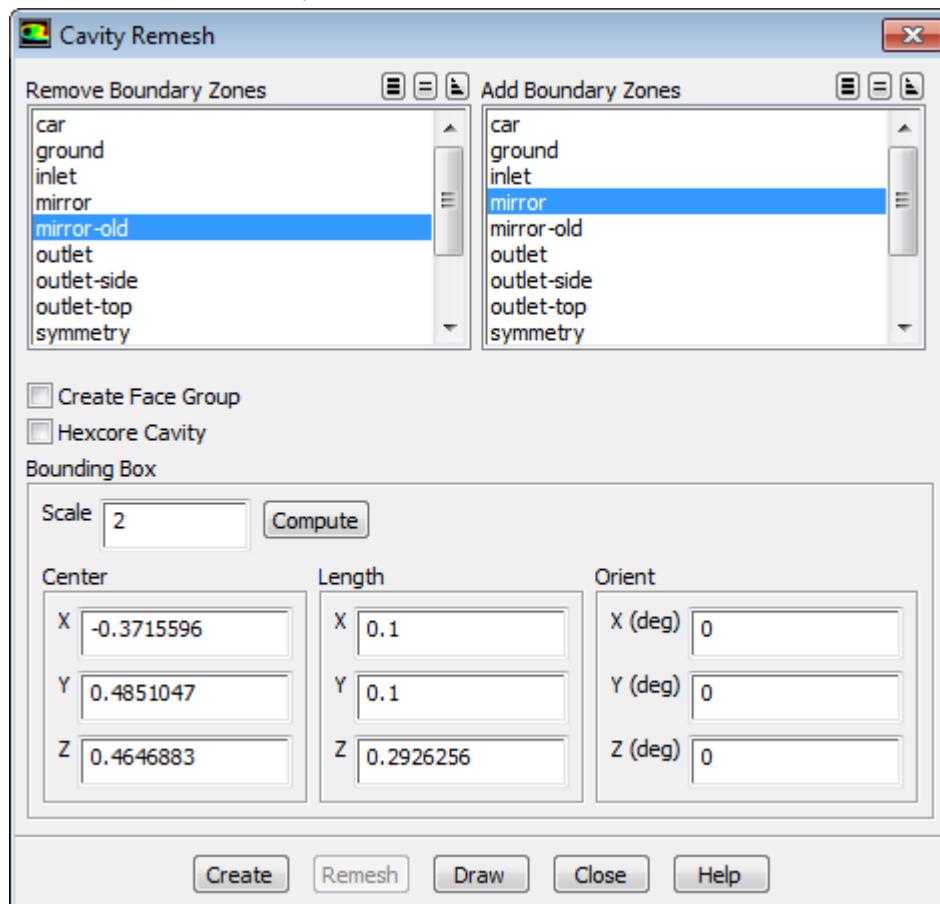
**Figure 11.7: Hybrid Mesh Near the Mirror**

3. Import and connect the mirror.
  - a. Read the mesh file **mirror.msh.gz** with the **Append File(s)** option enabled.  
**File → Read → Mesh...**
    - i. Enable **Append File(s)**.
    - ii. Select **mirror.msh.gz** in the **Files** list and click **OK**.
  - b. Verify that the mirror is appropriately positioned.
    - i. Select **car**, **mirror**, and **mirror-old** in the **Face Zones** selection list in the **Faces** tab of the **Display Grid** dialog box.  
**Display → Grid...**
      - ii. Enable **Free** and **Multi** in the **Options** group box in the **Faces** tab.  
 Make sure that the fluid zone is deselected in the **Cell Zones** selection list in the **Cells** tab. Click **Reset** in the **Bounds** tab.
    - iii. Click **Display**.  
 The presence of free nodes indicates that the mirror is not connected to the car.
  - c. Connect the mirror to the car.  
**Boundary → Merge Nodes...**

- i. Select only **car** in the **Boundary Face Zones** selection list in the **Compare...** group box and disable **Only Free Nodes**.
  - ii. Select only **mirror** in the **Boundary Face Zones** selection list in the **With...** group box and enable **Only Free Nodes**.
  - iii. Enable **Percent of shortest connected edge length** and enter 10 for **Tolerance**.
  - iv. Click **Merge**.
  - v. Click **Display** in the **Display Grid** dialog box.
- Both mirrors are now connected to the car.
- vi. Close the **Merge Boundary Nodes** dialog box.

4. Replace the mirror.

**Mesh → Tools → Cavity Remesh...**



- a. Select **mirror-old** in the **Remove Boundary Zones** selection list.
- b. Select **mirror** in the **Add Boundary Zones** selection list.
- c. Enter 1 for **Scale** and click **Compute**.

- d. Click **Draw**.

---

**Note**

The bounding box for the cavity fits the mirror. However, you need to generate prisms as well as tetrahedral cells in the cavity. Hence, you need to increase the scale.

---

- e. Enter 2 for **Scale** and click **Compute**.

- f. Click **Draw**.

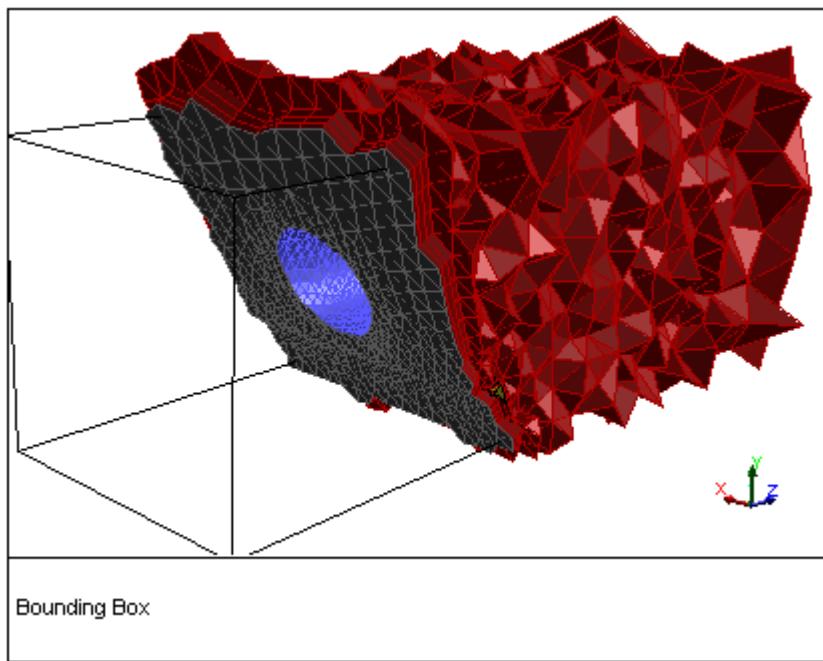
- g. Click **Create**.

The car zone is separated into two—one inside (**car:#**) and the other outside the cavity region, an interior surface (**interior:#:#**) is created at the boundary of the cavity, and a domain is defined and activated for the cavity.

- h. Select all the zones in the **Cavity Remesh** dialog box.

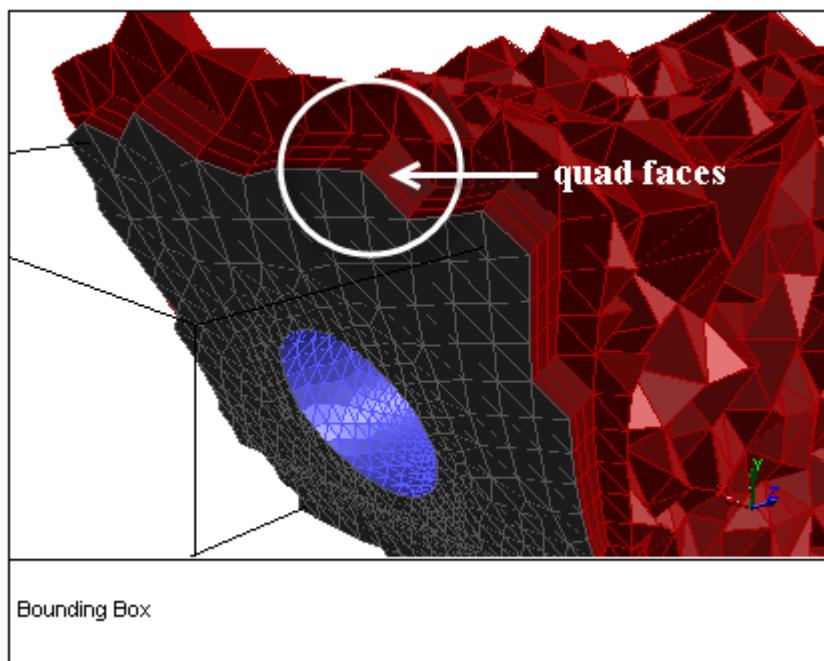
- i. Click **Draw** to view the domain (Figure 11.8: Cavity Domain Before Meshing (p. 274)).

**Figure 11.8: Cavity Domain Before Meshing**



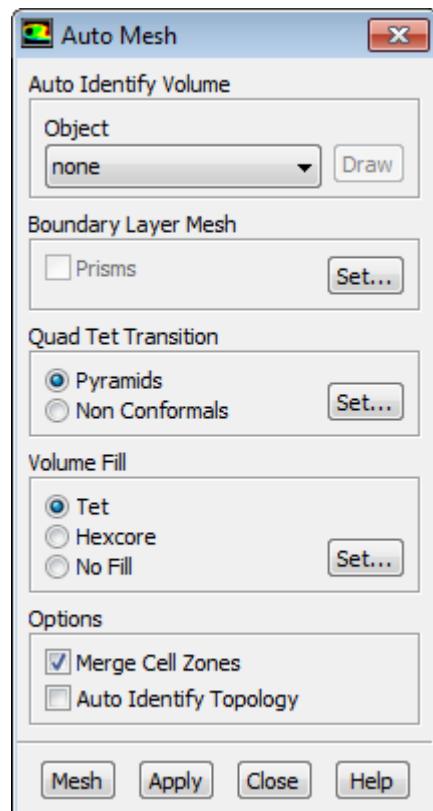
Zoom in to the interior zone (as shown in Figure 11.9: Quad Faces in the Interior (p. 275)), to see the quad faces. When you remesh the cavity, the prisms created during the remeshing will connect to these quad faces.

**Figure 11.9: Quad Faces in the Interior**



- j. Close the **Cavity Remesh** dialog box.
5. Specify the meshing parameters for meshing the cavity.

**Mesh → Auto Mesh...**



- a. Specify the prism meshing parameters.
  - i. Verify that the normals on the mirror are correctly oriented.

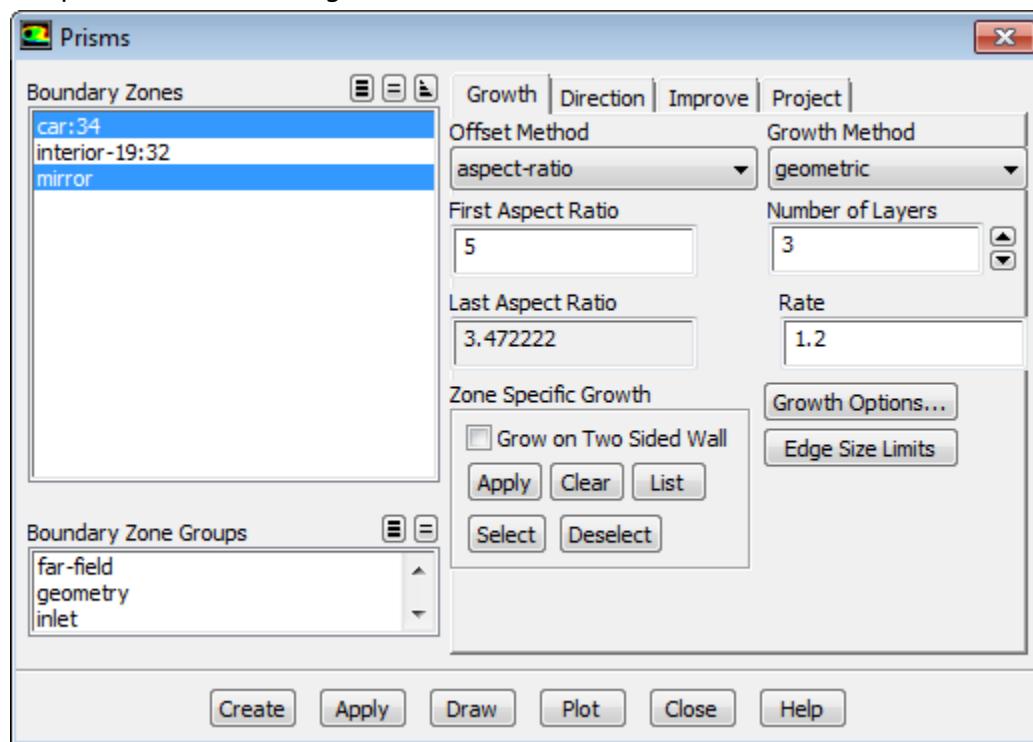
**Display → Grid...**

- A. Select **mirror** in the **Face Zones** selection list in the **Faces** tab.
- B. Enable **Normals** in the **Options** group box in the **Attributes** tab and enter **0.001** for **Normal Scale**.
- C. Click **Display**.

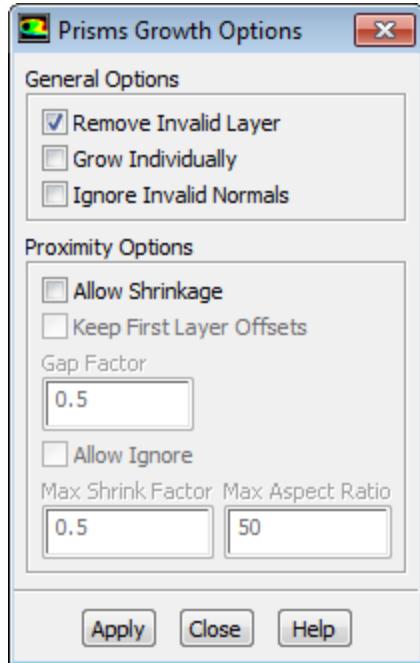
The normals point outward and hence, are correctly oriented.

- D. Disable **Normals** and close the **Display Grid** dialog box.

- ii. Click the **Set...** button in the **Boundary Layer Mesh** group box in the **Auto Mesh** dialog box to open the **Prisms** dialog box.



- iii. Select **car:#** and **mirror** in the **Boundary Zones** selection list.
- iv. Select **aspect-ratio** in the **Offset Method** drop-down list and enter **5** for **First Aspect Ratio**.
- v. Select **geometric** in the **Growth Method** drop-down list and enter **1.2** for **Rate**.
- vi. Set the **Number of Layers** to **3**.
- vii. Click the **Growth Options...** button to open the **Prisms Growth Options** dialog box.

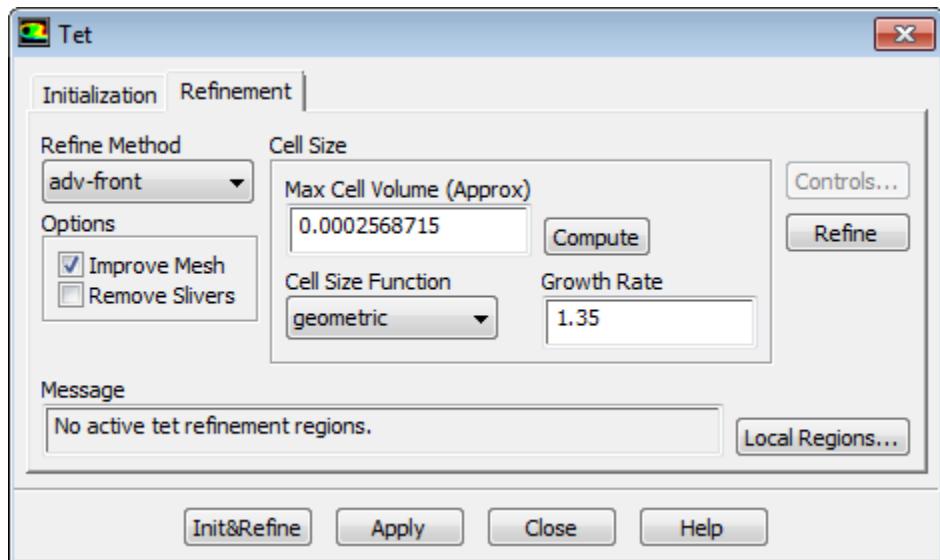


- A. Make sure that **Allow Shrinkage** is disabled in the **Proximity Options** group box.
- B. Click **Apply** and close the **Prisms Growth Options** dialog box.
- viii. Click **Apply** in the **Zone Specific Growth** group box.

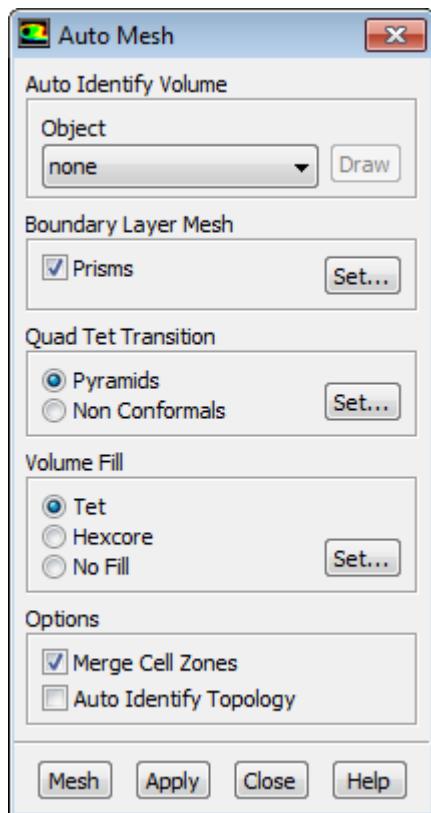
**Warning**

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

- ix. Close the **Prisms** dialog box.
- x. Enable **Prisms** in the **Auto Mesh** dialog box.
- b. Specify the tetrahedral meshing parameters.
  - Retain the selection of **Tet** and click the **Set....** button in the **Volume Fill** group box to open the **Tet** dialog box.



- Click the **Refinement** tab and retain the selection of **adv-front** in the **Refine Method** drop-down list.
- Retain the default value for **Max Cell Volume (Approx)**.
- Select **geometric** in the **Cell Size Function** drop-down list and enter **1 . 35** for **Growth Rate**.
- Click **Apply** and close the **Tet** dialog box.



- Enable **Merge Cell Zones**.

- d. Click **Mesh**.

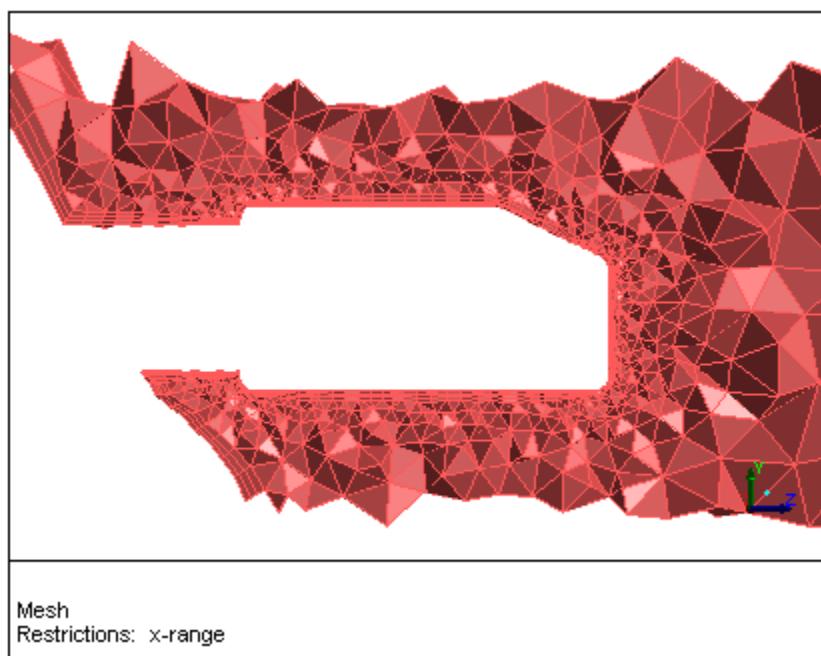
Prism layers will be generated around the mirror and the car according to the defined parameters. A domain will be automatically created and filled with tetrahedra, and finally the zones will be merged together.

- e. Close the **Auto Mesh** dialog box.
- f. Examine the mesh.

**Display → Grid...**

- i. Click the **Cells** tab and enable **All** in the **Options** group box.
- ii. Select **fluid** in the **Cell Zone Groups** selection list.
- iii. Click the **Bounds** tab and enable **Limit by X** in the **X Range** group box.
- iv. Enter  $-0.37$  for both **Minimum** and **Maximum** in the **X Range** group box.
- v. Click **Display** ([Figure 11.10: Remeshed Cavity \(p. 279\)](#)).

**Figure 11.10: Remeshed Cavity**



In [Figure 11.10: Remeshed Cavity \(p. 279\)](#), you can see the three prism layers generated around the mirror and car. The rest of the domain is filled with tetrahedra.

- vi. Activate the global domain.

**Mesh → Domains...**

- A. Select **global** in the **Domains** list and click **Activate**.
- B. Close the **Domains** dialog box.

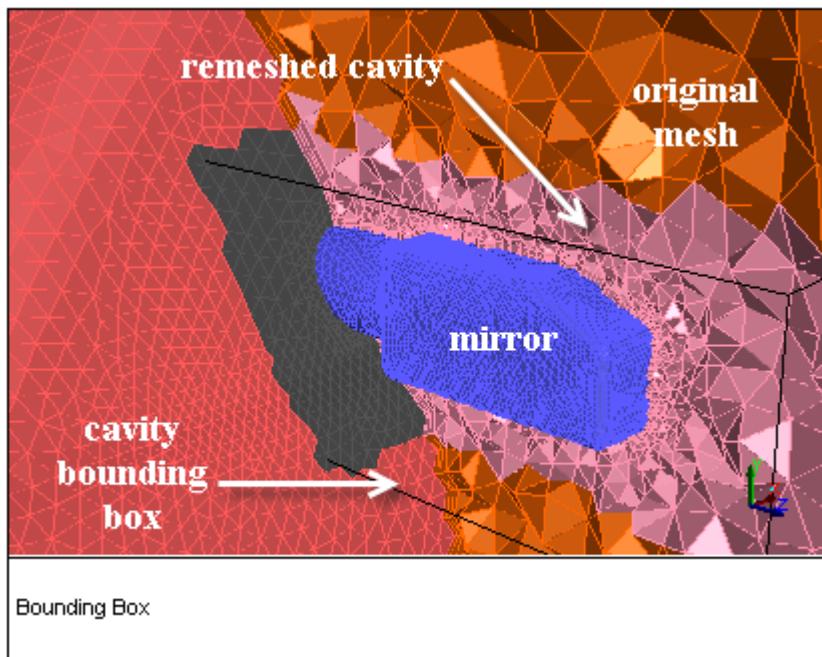
- vii. Display the original mesh along with the remeshed cavity.
  - A. Select **car**, **car:#**, **mirror**, **wheel-arch-front**, **wheel-arch-rear**, **wheel-front**, and **wheel-rear** in the **Face Zones** selection list in the **Faces** tab of the **Display Grid** dialog box.
  - B. Make sure the **fluid** zone is deselected in the **Cells** tab and the display bounds are reset in the **Bounds** tab.
  - C. Click **Display**.
  - D. Enable the overlaying of graphics.

**Display → Scene...**

- E. Select all the zones in the **Names** selection list and enable **Overlays** in the **Scene Composition** group box.
- F. Click **Apply** and close the **Scene Description** dialog box.
- G. Select the fluid zones in the **Cell Zones** selection list in the **Cells** tab of the **Display Grid** dialog box.
- H. Enable **All** in the **Options** group box.
- I. Click the **Bounds** tab and enable **Limit by X** in the **X Range** group box.
- J. Enter **-0.37** for both **Minimum** and **Maximum** in the **X Range** group box.
- K. Click **Display**.
- L. Click **Draw** in the **Cavity Remesh** dialog box ([Figure 11.11: Cavity Remeshed With Hybrid Mesh \(p. 281\)](#)).

**Mesh → Tools → Cavity Remesh...**

**Figure 11.11: Cavity Remeshed With Hybrid Mesh**



In Figure 11.11: Cavity Remeshed With Hybrid Mesh (p. 281), you can see that the mesh outside the cavity has not been modified. The mesh inside the cavity is connected in a conformal manner with the original mesh.

6. Merge the cavity domain with the original mesh.

```
>/mesh/cavity/merge-cavity
Insert domain name/id [tet] cavity-2
Into domain name/id [cavity-2] global
Merging cell zone fluid-# (id #) with fluid-#
Merging face zone interior-#:# (id #) with interior-#
Merging face zone car:# (id #) with car
```

where # denotes the respective zone IDs.

7. Delete the old mirror.

#### **Boundary → Manage...**

- a. Select **mirror-old** in the **Face Zones** selection list.
- b. Select **Delete** in the **Options** list and retain the **Delete Nodes** option.
- c. Click **Apply**.

A **Question** dialog box will open asking you to confirm if you want to delete the selected zone(s).

- d. Click **Yes** to close the **Question** dialog box.
- e. Close the **Manage Face Zones** dialog box.

8. Check the mesh quality.

**Report → Cell Limits...**

The maximum skewness reported is around 0.935. The exact value may vary slightly on different platforms.

9. Check the mesh.

**Mesh → Check**

10. Save the mesh (sedan-hyb-1-cavity.msh.gz).

**File → Write → Mesh...**

## 11.5. Cavity Remeshing For a Hybrid Mesh (Tetrahedra and Prisms) Having Multiple Fluid Zones

This section demonstrates the use of a journal file to remesh a cavity in a mesh having two different fluid zones (prism and tetrahedral cells).

1. View the contents of the journal file (sedan\_hyb-2zones-cavity.jou) in a viewer.

All lines starting with a semicolon (;) indicate comments.

```
;Read the hybrid mesh and the new mirror
/file/read-multiple-mesh "sedan_hyb-2zones.msh.gz" "mirror.msh.gz"

;Display the cells in the plane x=-0.37
/display/set-grid/x-range -0.37 -0.37
/display/set/filled-grid? yes
/display/set/lights/lights-on? yes
//display/set-grid/all-cells? yes
/display/all-grid * ,

;Set the view
/display/view/camera/target -0.37 0.486 0.48
/display/view/camera/position -1 0.486 0.48
/display/view/camera/field 0.35 0.35
/display/view/camera/up-vector 0 1 0

;Merge the free nodes of the imported mirror with 10% edge tolerance
/boundary/merge-nodes car , mirror , no yes yes 10

;Replace the mirror by creating a cavity with a scale of 2
/mesh/cavity/replace-zones mirror-old , mirror , 2 , , , no
/display/all-grid * ,

;Specify prism growth parameters
/mesh/prism/controls/zone-specific-growth/apply-growth mirror car:* , aspect-ratio geometric 3 5 1.2 no
;Disable shrinkage of prism layers
/mesh/prism/controls/proximity/allow-shrinkage? no

;Mesh the cavity with prisms and tetrahedra
/mesh/tet/controls/refine-method adv-front
/mesh/tet/controls/cell-size-function geometric 1.35
/mesh/auto-mesh , yes pyramids tet no no
/display/all-grid * ,

;Activate the global domain
/mesh/domain/activate global

;Merge the fluid zones together
```

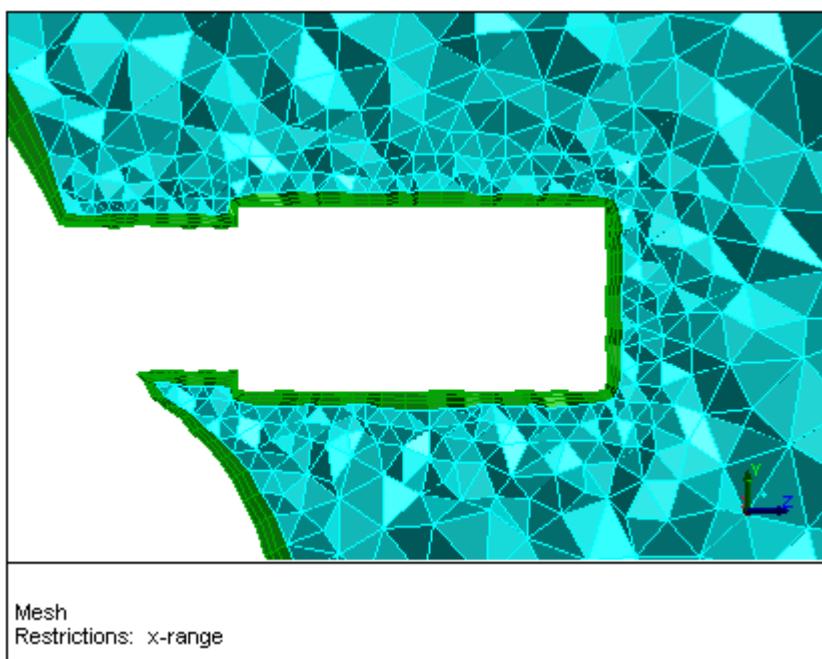
```
/mesh/manage/merge * , yes  
;Display the cells in the plane x = -0.37  
/display/all-grid fluid*,  
  
;Delete the old mirror  
/boundary/manage/delete mirror-old , yes
```

2. Start a new session of ANSYS FLUENT in meshing mode.
3. Read the journal file sedan\_hyb-2zones-cavity.jou.

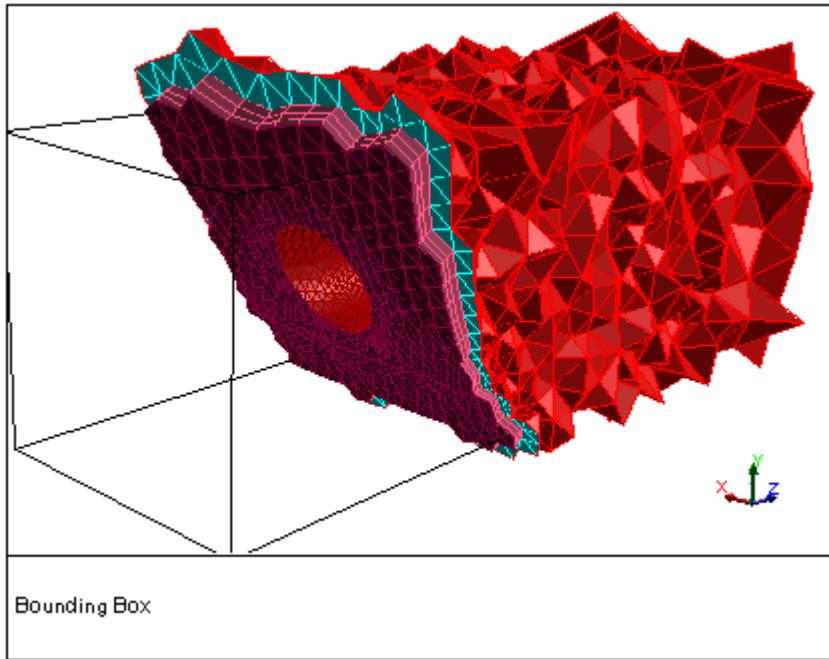
**File → Read → Journal...**

*Figure 11.12: Hybrid Mesh Near the Mirror (p. 283)—Figure 11.15: Cavity Remeshed With Hybrid Mesh (p. 285)* show the mesh at intermediate stages.

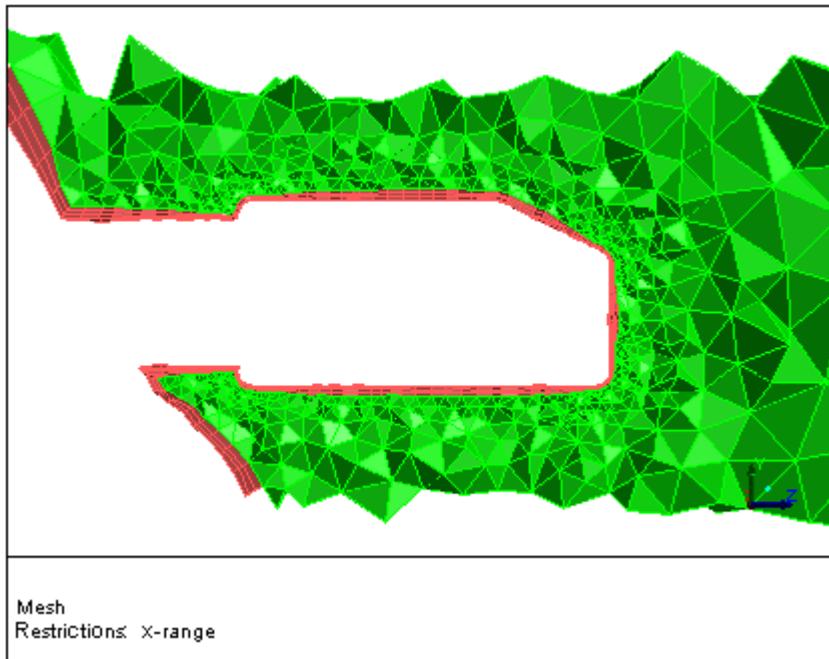
**Figure 11.12: Hybrid Mesh Near the Mirror**

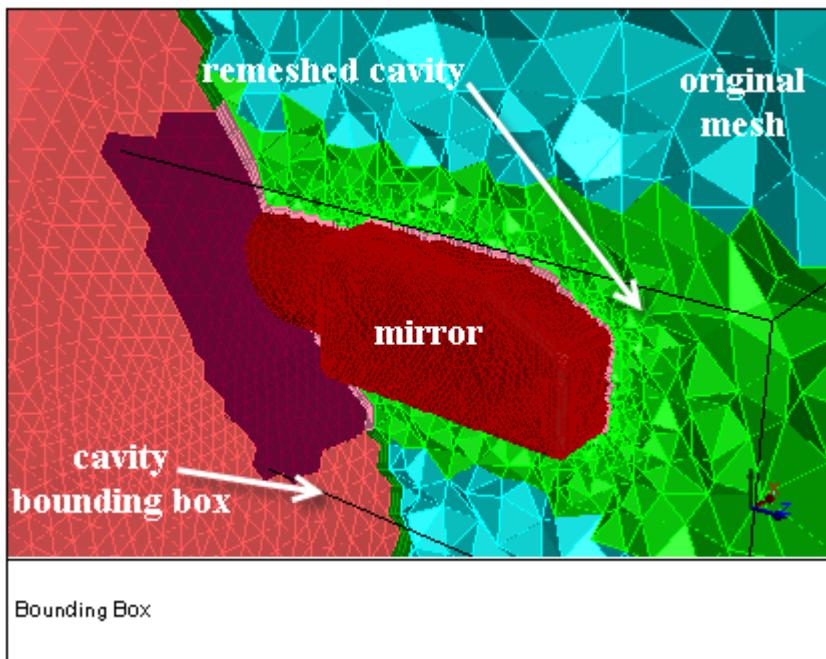


**Figure 11.13: Cavity Domain Before Meshing**



**Figure 11.14: Remeshed Cavity**



**Figure 11.15: Cavity Remeshed With Hybrid Mesh**

## 11.6. Cavity Remeshing For a Hexcore Mesh

The procedure outlined in this section is similar to that described in previous sections, and hence is less explicit.

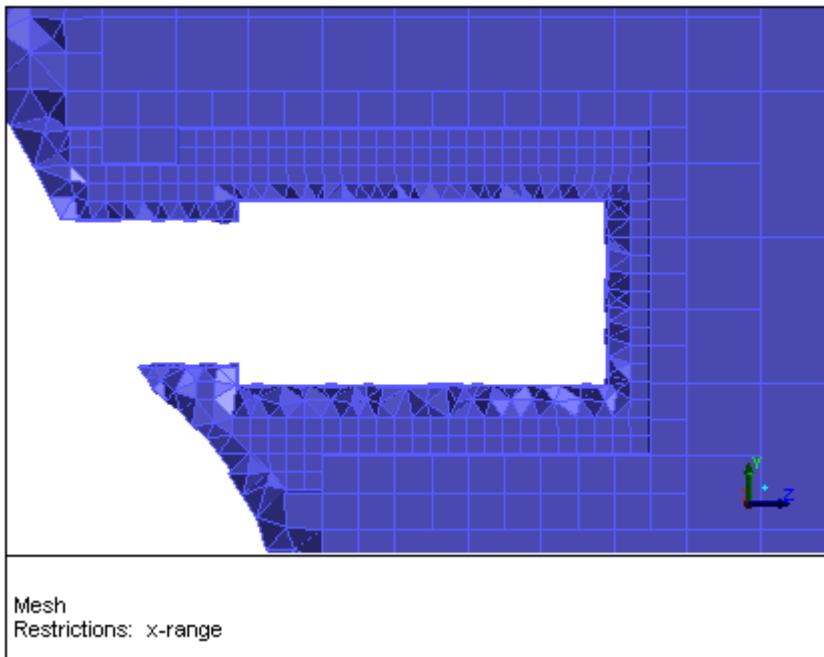
1. Read the mesh files `sedan_hexcore.msh.gz` and `mirror.msh.gz`.

**File → Read → Mesh...**

2. Examine the mesh.

**Display → Grid...**

- a. Enable **All** in the **Options** group box and select the fluid zone in the **Cell Zones** selection list in the **Cells** tab.
  - b. Enable **Limit by X** and enter  $-0.37$  for both **Minimum** and **Maximum** in the **X Range** group box in the **Bounds** tab.
  - c. Click the **Attributes** tab and enable **Filled** and **Lights**.
  - d. Click **Display**.
  - e. Display the **left** view.
- Display → Views...**
- f. Zoom in and examine the mesh around the mirror (Figure 11.16: Hexcore Mesh Near the Mirror (p. 286)).

**Figure 11.16: Hexcore Mesh Near the Mirror**

3. Verify that the mirror is appropriately positioned.

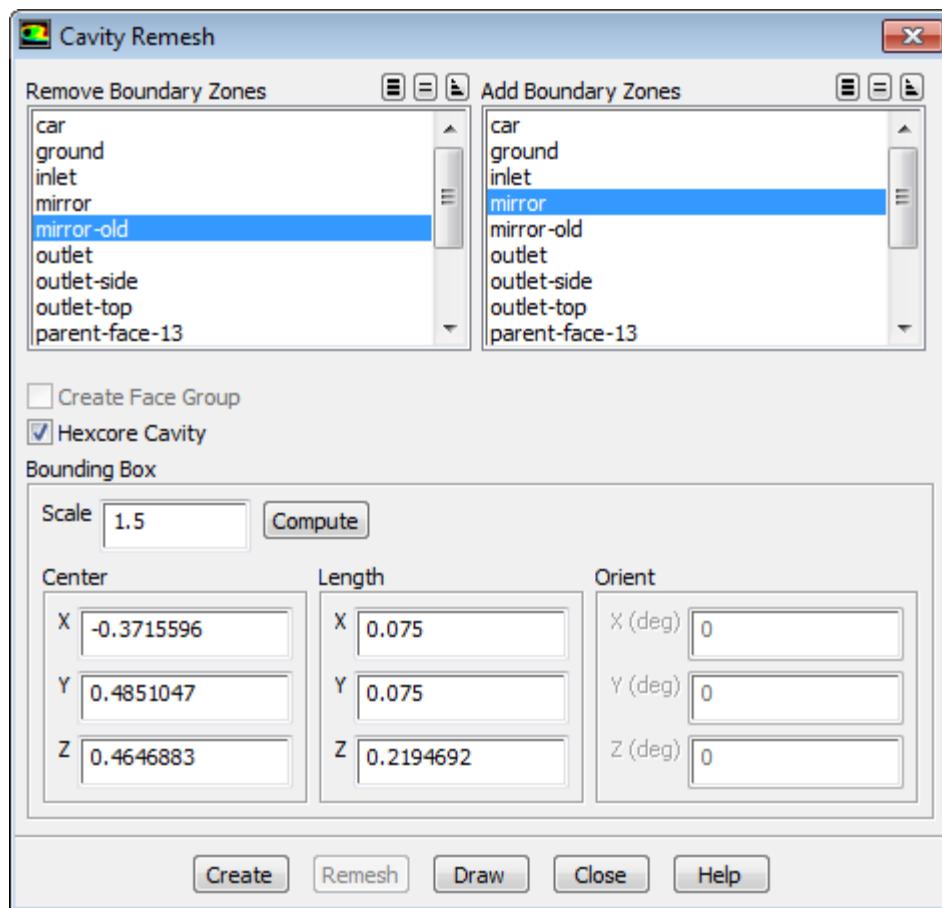
Enable the display of free nodes to check whether the mirror is connected to the car. The presence of free nodes indicates that the mirror is not connected to the car.

4. Connect the new mirror.

**Boundary → Merge Nodes...**

- a. Select only **car** in the **Boundary Face Zones** selection list in the **Compare...** group box and disable **Only Free Nodes**.
  - b. Select only **mirror** in the **Boundary Face Zones** selection list in the **With...** group box and retain the **Only Free Nodes** option.
  - c. Enable **Percent of shortest connected edge length** and enter 10 for **Tolerance**.
  - d. Click **Merge**.
  - e. Close the **Merge Boundary Nodes** dialog box.
5. Replace the old mirror with the new one.

**Mesh → Tools → Cavity Remesh...**



- a. Select **mirror-old** in the **Remove Boundary Zones** selection list and **mirror** in the **Add Boundary Zones** selection list, respectively.
- b. Enable **Hexcore Cavity**.
- c. Enter **1.5** for **Scale** and click **Compute**.
- d. Click **Create**.
6. Click **Remesh** in the **Cavity Remeshing** dialog box.
7. Examine the remeshed cavity by displaying the cells in the plane  $x = -0.37$  ([Figure 11.17: Remeshed Cavity \(p. 288\)](#)).

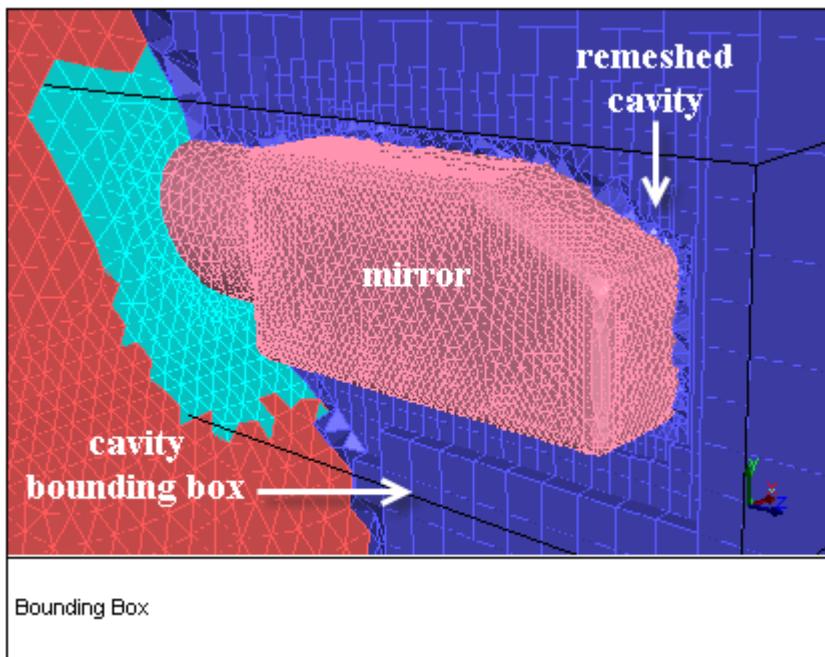
**Display → Grid...**

**Figure 11.17: Remeshed Cavity**

8. Display the original mesh along with the remeshed cavity (Figure 11.18: Remeshed Hexcore Cavity (p. 289)).

- Select **car**, **car:#**, **mirror**, **wheel-arch-front**, **wheel-arch-rear**, **wheel-front**, and **wheel-rear** in the **Face Zones** selection list in the **Display Grid** dialog box.
- Make sure the fluid zone is deselected in the **Cells** tab and the display bounds are reset in the **Bounds** tab.
- Click **Display**.
- Enable the overlaying of graphics in the **Scene Description** dialog box.  
**Display → Scene...**
- Select the fluid zone in the **Cell Zones** selection list and enable **All** in the **Options** group box in the **Cells** tab of the **Display Grid** dialog box.
- Enable **Limit by X** and enter  $-0.37$  for both **Minimum** and **Maximum** in the **X Range** group box in the **Bounds** tab.
- Click **Display**.
- Click **Draw** in the **Cavity Remesh** dialog box.  
**Mesh → Tools → Cavity Remesh...**

**Figure 11.18: Remeshed Hexcore Cavity**



9. Check the mesh.

**Mesh → Check**

10. Save the mesh (sedan-hexcore-cavity.msh.gz).

**File → Write → Mesh...**

11. Exit ANSYS FLUENT.

**File → Exit**

## 11.7. Summary

This tutorial demonstrated the creation of a cavity in an existing mesh to replace an object, and then remeshing the cavity as appropriate. The procedure for creating and remeshing the cavity was demonstrated for a tetrahedral mesh, a hybrid mesh with a single fluid zone, a hybrid mesh with multiple fluid zones, and a hexcore mesh.

