TGrid 5.0 Tutorial Guide

April 2008

ANSYS, Inc.
Centerra Resource Park
10 Cavendish Court
Lebanon, NH 03766

Using This Manual

What's In This Manual

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

- Tutorial 1 is a detailed tutorial designed to introduce the beginner to TGrid. This tutorial provides explicit instructions for all steps in the tutorial.

  The remaining tutorials assume that you have read or solved Tutorial 1, and that you are already familiar with TGrid and its interface. In these tutorials, some steps will not be shown explicitly.

- Tutorial 2 demonstrates the mesh generation procedure for a problem that has multiple regions. It also describes the procedure to generate a volume mesh using the automatic refinement feature of TGrid.
- Tutorial 3 demonstrates the mesh generation procedure for a hybrid mesh, starting from a hexahedral volume mesh and a triangular boundary mesh.
- Tutorial 4 demonstrates the mesh generation procedure for a viscous hybrid mesh, starting from a triangular boundary mesh for a sedan car body.
- Tutorial 5 explains an application from the automotive industry, thus demonstrating how the hexcore mesh can significantly reduce the cell count compared with a fully tetrahedral mesh.
- Tutorial 6 demonstrates the creation of a hexcore mesh upto the domain boundaries for a sedan car.
- Tutorial 7 demonstrates the use of the boundary wrapper to repair an existing geometry. It also describes the procedure to improve the wrapper surface quality.
- Tutorial 8 demonstrates the procedure for replacing an object in the existing mesh with another by creating a cavity and remeshing it.

Where to Find the Files Used in the Tutorials

Each of the tutorials uses an existing mesh file. You will find the appropriate mesh file (and any otherrelevant files used in the tutorial) on the TGrid documentation CD. The **Preparation** step of each tutorial will tell you where to find the necessary files.

Typographical Conventions Used In This Manual

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

- An informational icon ( ) marks an important note.
- A warning icon ( ) marks an important note or warning.
- Different type styles are used to indicate graphical user interface menu items and text interface menu items (e.g., Display Grid panel, `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 panel.
- Instructions for performing each step in a tutorial will appear in standard type. Additional information about a step in a tutorial appears in italicized type.
- A mini flow chart is used to indicate the menu selections that lead you to a specific panel. For example,

  Display ⟶Grid...

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

  The words surrounded by boxes invoke menus (or submenus) and the arrows point from a specific menu toward the item you should select from that menu.

Contents

Tutorial 1. Repairing a Boundary Mesh

Introduction

TGrid offers several tools for mesh repair. 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 in TGrid to find and fix known deficiencies in an existing boundary mesh (a simple 3D geometry).

This tutorial demonstrates how to do the following:

1. Read the mesh file and display the boundary mesh.
2. Check for free and unused nodes.
3. Repair the boundary mesh by recreating missing faces.
4. Use the rezoning feature.
5. Improve the boundary mesh.
6. Check the skewness of the boundary faces.
7. Further repair the boundary mesh.
8. Generate a multiple region volume mesh.
9. Check the quality of the entire volume mesh.
10. Check and save the volume mesh.

Prerequisites

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

Preparation

1. Download mesh-repair.zip from the FLUENT User Services Center to your working directory. This file can be found from the Documentation link on the TGrid product page.

   OR

   Copy mesh-repair.zip from the TGrid documentation CD to your working directory.

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

     cdrom/tgrid5.0/help/tutfiles/

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

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

     cdrom:\tgrid5.0\help\tutfiles\

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

2. Unzip mesh-repair.zip.

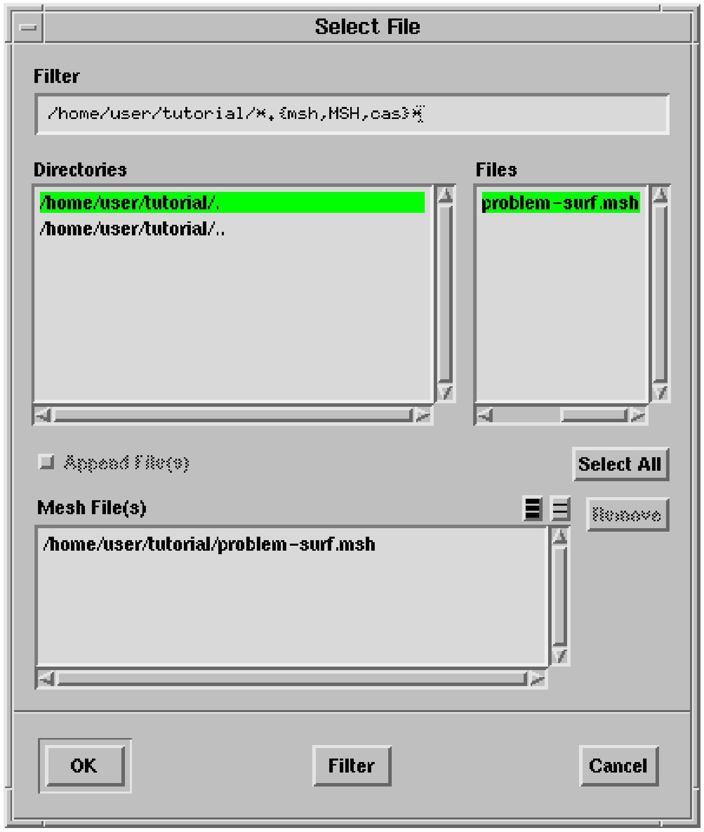
   The file, problem-surf.msh *can be found in the* mesh-repair *folder created on unzipping the file.*

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

Step 1: Reading and Displaying the Boundary Mesh

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

   File ⟶ Read ⟶Boundary Mesh...

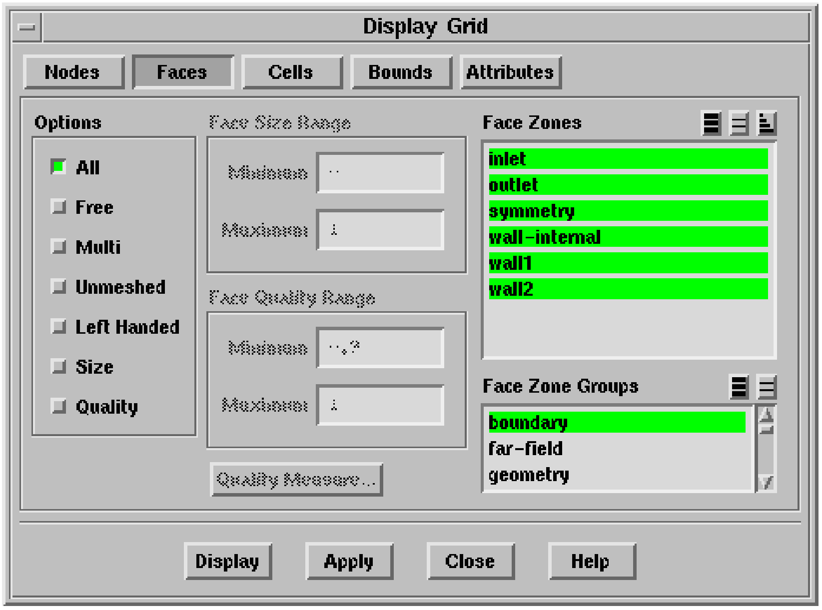


   (a) Select problem-surf.msh in the Files list.

   (b) Click OK.

2. Display the boundary mesh.

   Display →Grid...



   (a) Select boundary in the Face Zone Groups selection list.

   (b) Click Display (Figure 1.1).

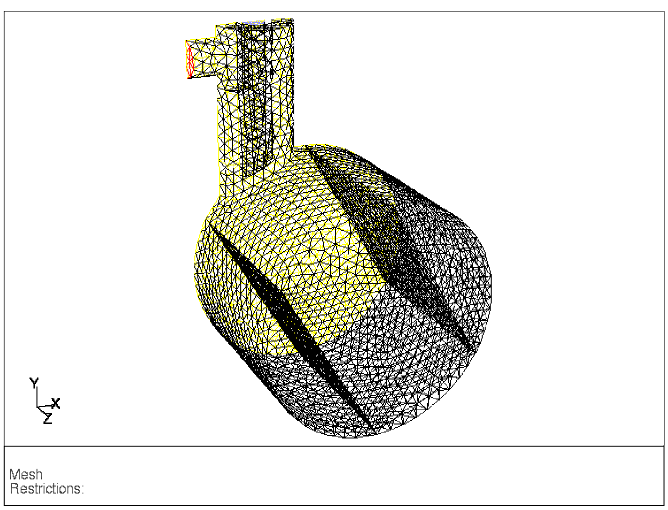


Figure 1.1: Boundary Mesh

   (c) Close the Display Grid panel.

3. Display the boundary mesh with the hidden lines removed.

   Display ⟶Options...

   (a) Enable Hidden Line Removal.

   (b) Click Apply and close the Display Options panel.

   The display will be updated as shown in Figure 1.2.

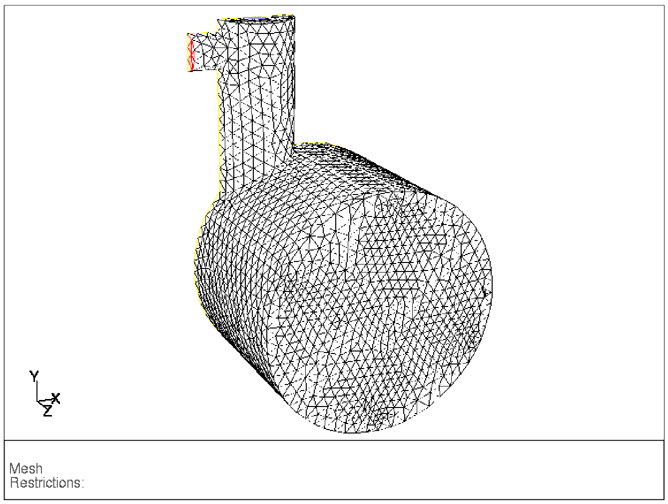


Figure 1.2: Boundary Mesh With Hidden Lines Removed

Step 2: Check for Free and Unused Nodes

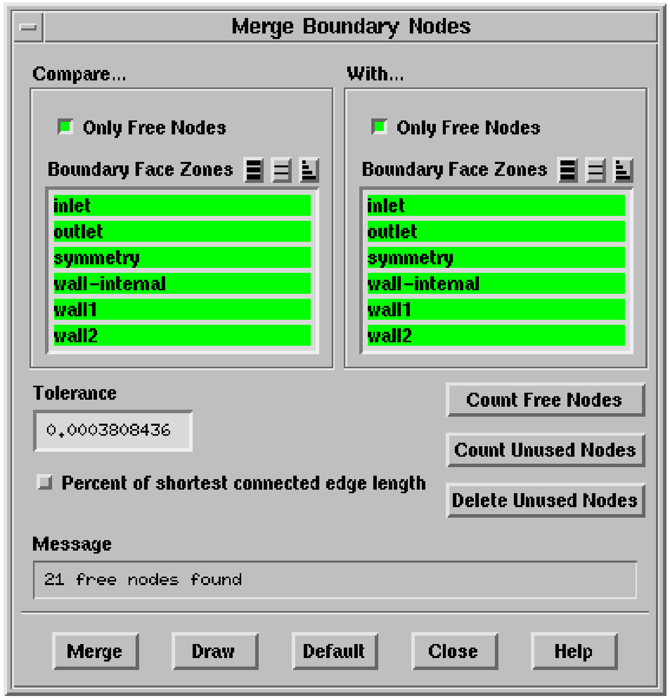
After reading the boundary mesh, check it for topological problems such as free and multiply-connected nodes and faces.

Boundary ⟶Merge Nodes...

1. Click Count Free Nodes.

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

   Here, the free nodes are due to seven missing faces in the surface mesh. In **Step 3**, *you will use* TGrid *mesh repair tools to recreate the missing faces.*



2. Click Count Unused Nodes.

   TGrid *will report the number of unused nodes in the* Message *box. If there are unused nodes, click* Delete Unused Nodes *to remove them.*

3. Close the Merge Boundary Nodes panel.

Step 3: Repair the Boundary Mesh

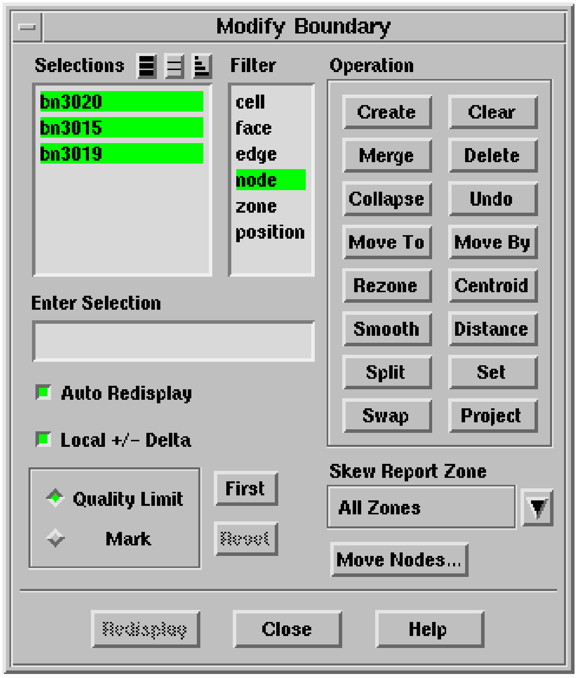
In this step, you will recreate the missing faces to repair the boundary mesh.

1. Zoom in to one of the missing faces (Figure 1.3).

   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 *panel to highlight the faces surrounding the missing face.*

2. Recreate the missing face.

   Boundary ⟶Modify...



   (a) Select node in the Filter list.

   (b) Select the three nodes surrounding the missing face using the right mouse button (see Figure 1.3).

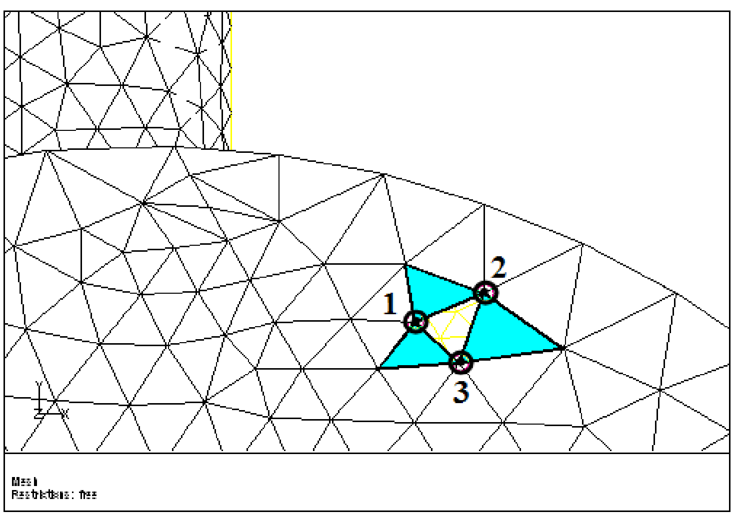


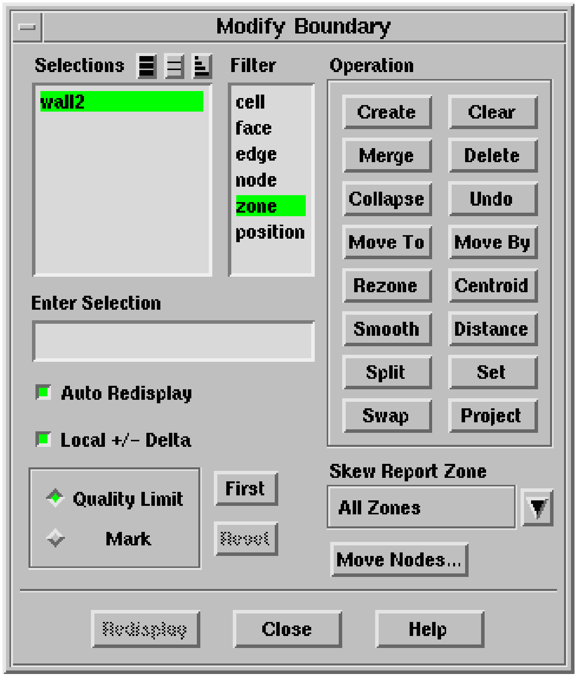
Figure 1.3: Recreating the Missing Face

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.

TGrid *will recreate the missing face.*

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.

TGrid *will display the zone name in the graphics window (Figure 1.4).*

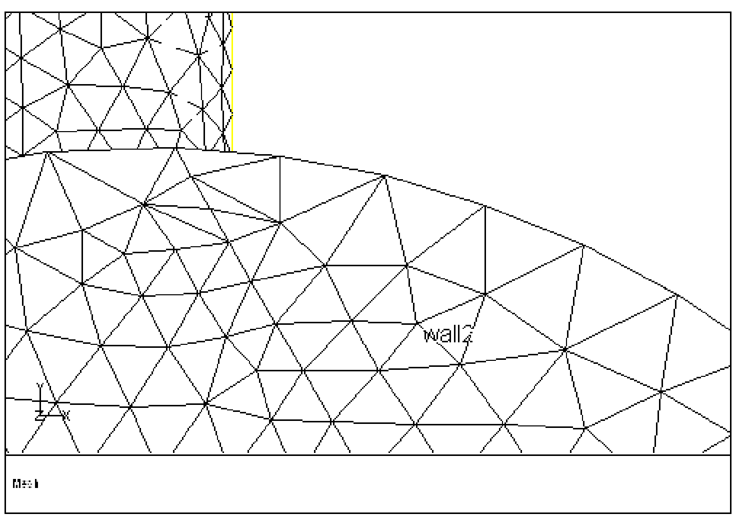


Figure 1.4: Verifying the Zone of the New Face

TGrid *places the face 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 **Step 4**).

4. Similarly, recreate the other missing faces.

5. Save an intermediate mesh file (`temp.msh`).

   File ⟶ Write ⟶ Mesh...

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

Step 4: Use the Rezoning Feature

This step 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 step is complete, 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 1.5).

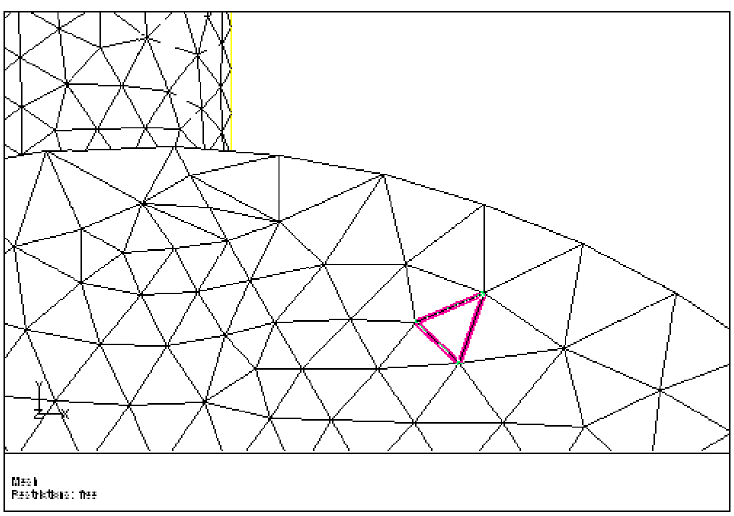
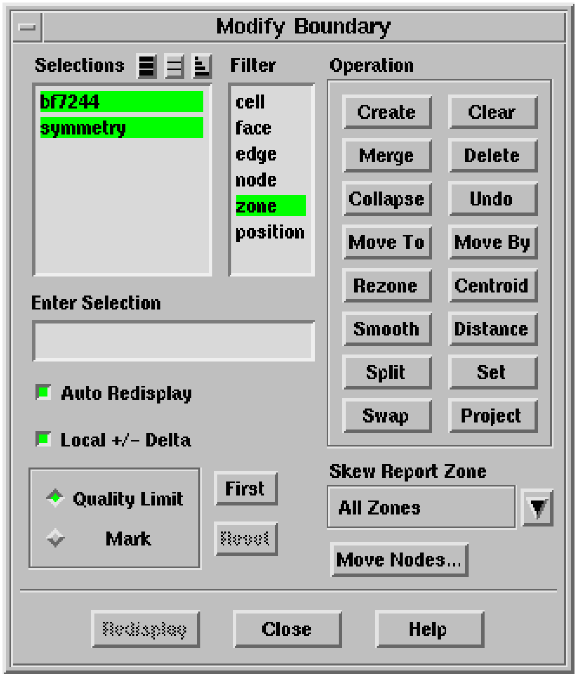


Figure 1.5: Face Selected to be Rezoned

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 *panel will show the face identification number and the zone to which you want to move it.*



5. Click Rezone in the Operation group box.

   TGrid *will move the selected face to the* symmetry *zone (Figure 1.6).*

   ⚠ This step was included only to demonstrate the use of the Rezone option. Move the selected face back to the wall2 zone using Rezone.

6. Close the Modify Boundary panel.

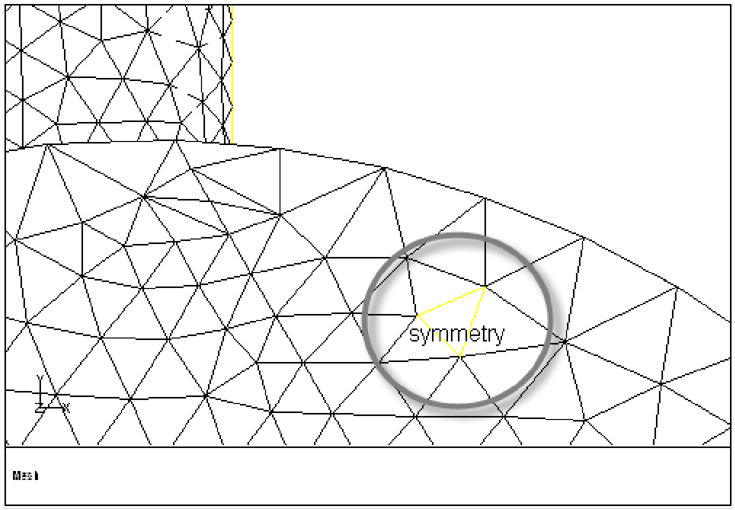
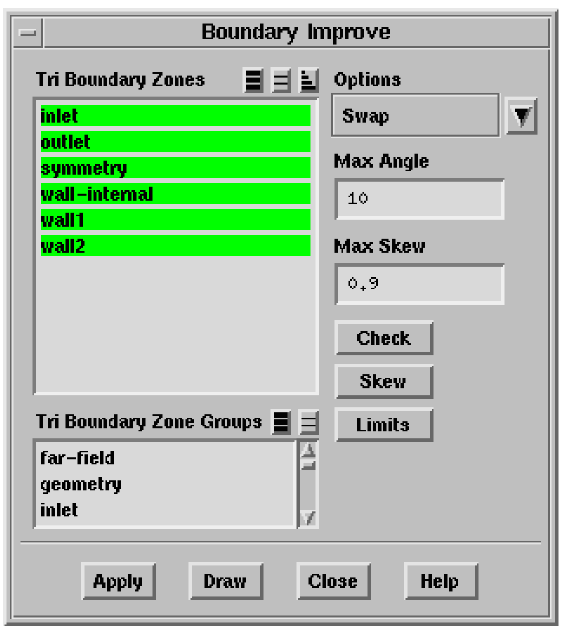


Figure 1.6: Face Rezoned to Symmetry Boundary

Step 5: Improve the Boundary Mesh

Boundary ⟶ Mesh ⟶ Improve...



1. Select all the zones in the Tri Boundary Zones selection list.
2. Select Swap in the Options drop-down list.
3. Click Skew to check if the maximum face skewness is below 0.9.

TGrid *will report that the maximum face skewness is approximately 0.992.*

4. Click Check to check for Delaunay violations in the boundary mesh.

   TGrid *will report the violations in the console.*

5. Retain the default values of `10` and `0.9` for Max Angle and Max Skew, respectively.

6. Click Apply until TGrid reports zero modifications made.

7. Click Skew to verify that the maximum face skewness is below 0.9.

8. Close the Boundary Improve panel.

Step 6: Check the Skewness Distribution of the Boundary Mesh

Display ⟶ Plot ⟶ Face Distribution...

1. Select all the zones in the Boundary Zones selection list.

2. Enter `10` for Partitions.

3. Click Plot (Figure 1.7).

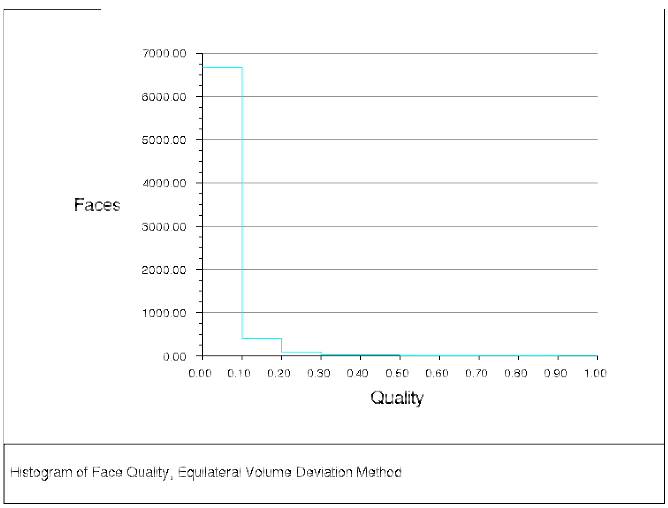


Figure 1.7: Histogram Plot of Face Skewness

4. Click Print.

   TGrid *will print the histogram information 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 panel.

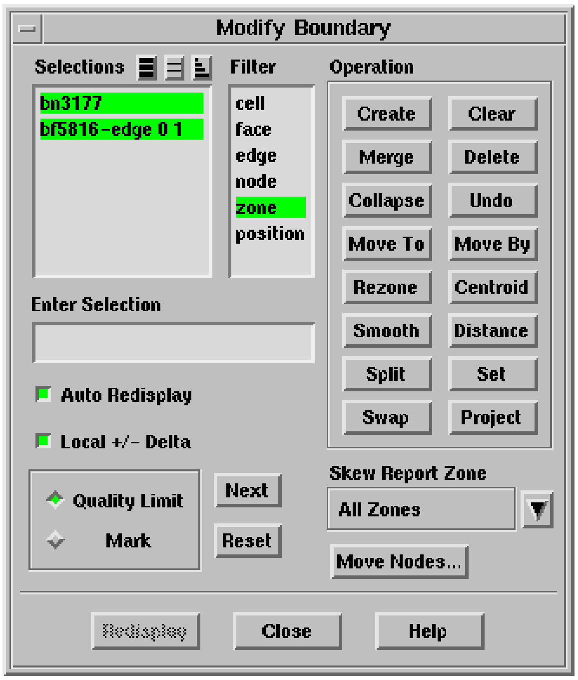
Extra: *This tutorial also aims at reducing the maximum face skewness below 0.6. This tutorial exposes you to some of the mesh repair tools. Then, it is up to you to try and get the maximum face skewness below 0.6.*

Step 7: Repairing the Boundary Mesh Further

In this step, you will repair the mesh 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.

   TGrid *will zoom in on the face having the greatest skewness (Figure 1.8). You will merge the highlighted node with the corner node to repair the skewed face.*

   When merging nodes, the first node selected is the one that remains after merging. 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. The corner node will be retained after merging the nodes, since it was selected first. The procedure is described in the following steps.*

   (b) Click Clear in the Operation group box.

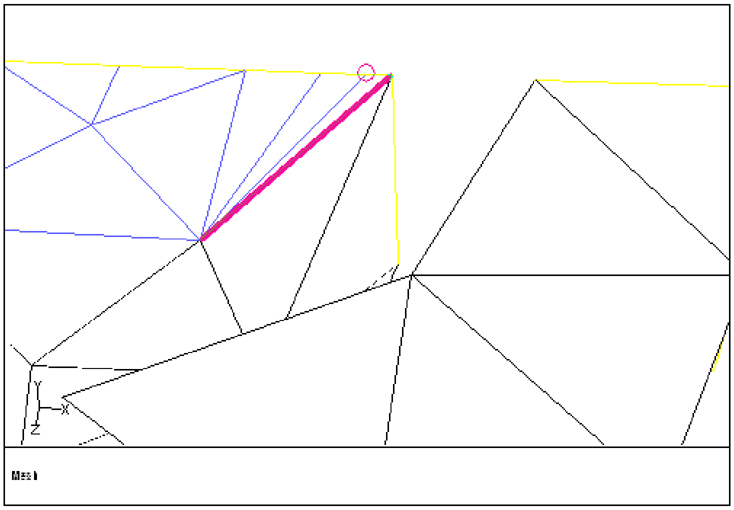


Figure 1.8: Face with the Greatest Skewness

(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 1.9.

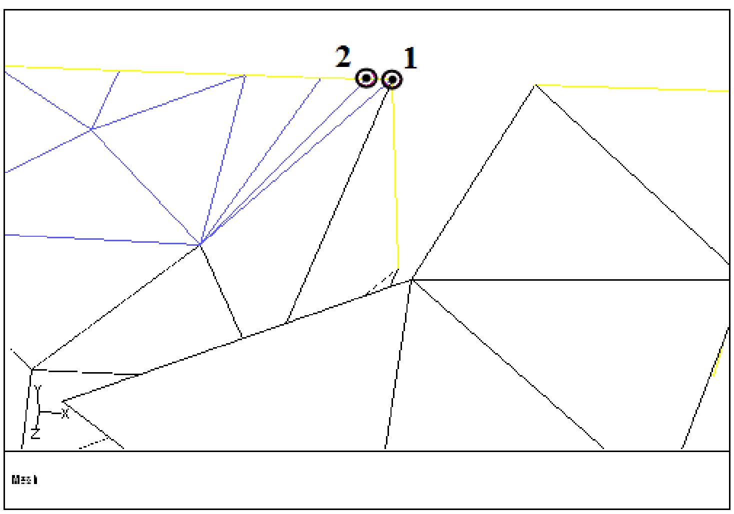


Figure 1.9: Nodes to be Merged

(e) Click Merge in the Operation group box (Figure 1.10).

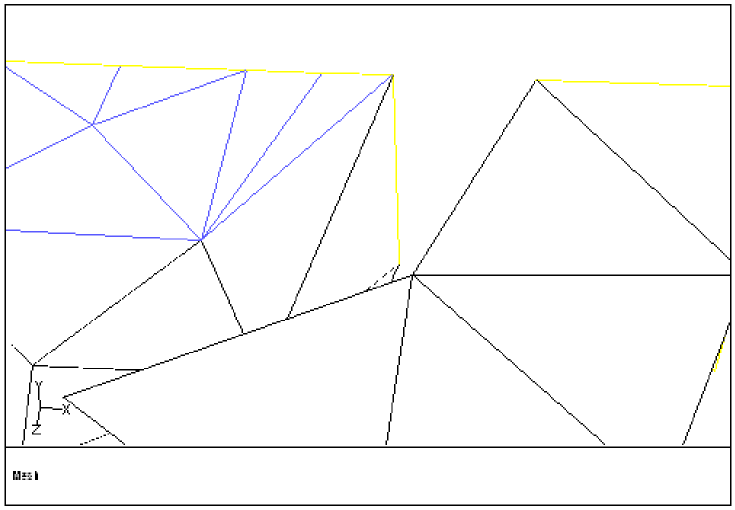


Figure 1.10: Surface Mesh After Merging Nodes

2. Repair the next highly skewed face.

   (a) Click Next in the Modify Boundary panel.

   TGrid *will zoom in on the face with the next highest skewness (Figure 1.11). The face highlighted is the face on the opposite corner of the* inlet *boundary.*

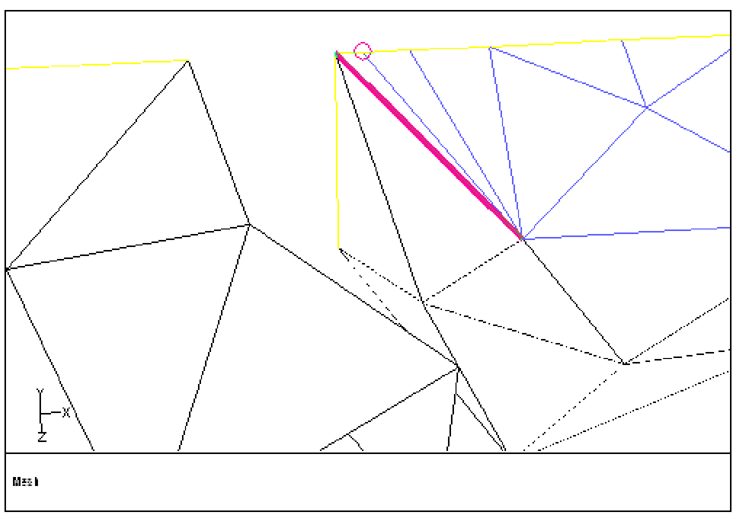


Figure 1.11: Face with the Next Greatest Skewness

   (b) Clear the Selections list.

   (c) Select node in the Filter list.

   (d) Select the nodes as shown in Figure 1.12.

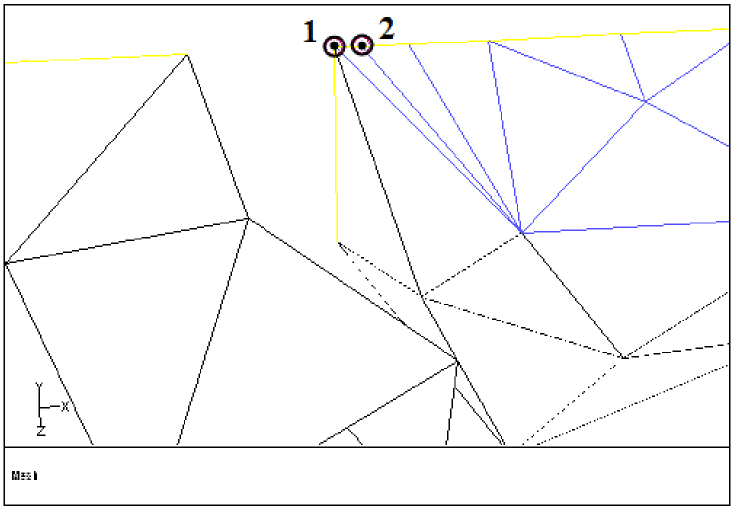


Figure 1.12: Nodes to be Merged

(e) Click Merge.

The modified mesh after merging the nodes is shown in Figure 1.13.

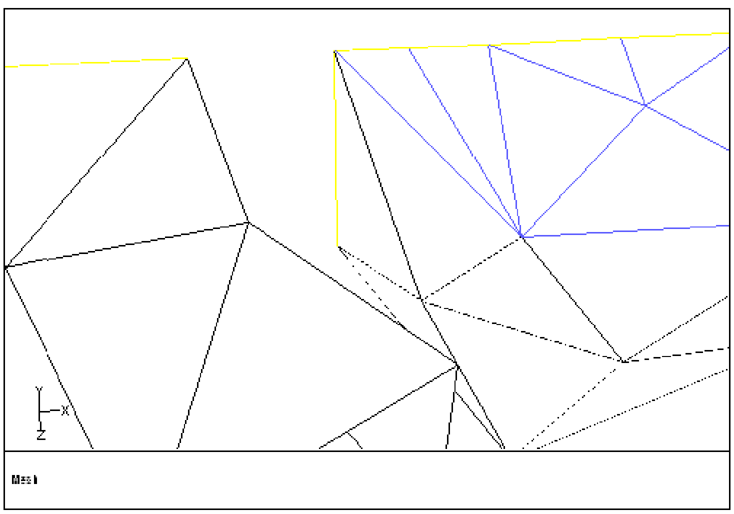


Figure 1.13: Surface Mesh After Merging Nodes

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 in Steps 7.1 and 2.*

3. Modify the mesh by smoothing nodes.

    (a) Click Next.

    TGrid *highlights a face located in the middle of one of the internal walls (Figure 1.14).*

(b) Select node in the Filter list.

(c) Select several nodes surrounding the face highlighted by TGrid (as shown in Figure 1.14).

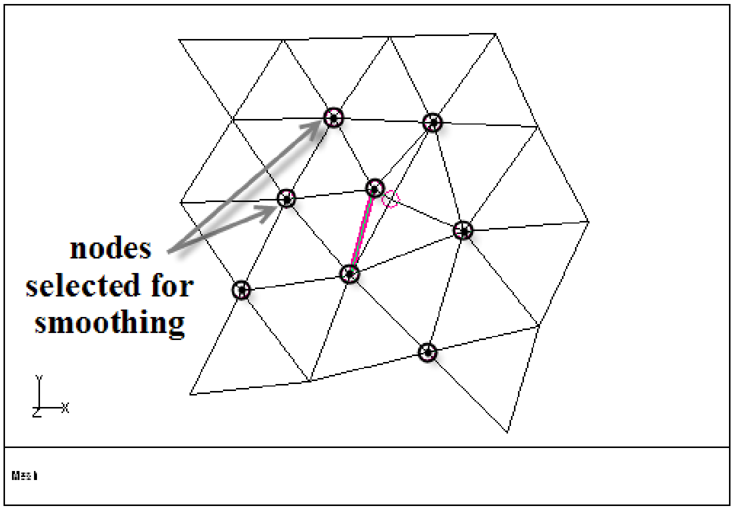


Figure 1.14: Face to be Modified with Node Smoothing

(d) Click Smooth in the Operation group box.

TGrid *performs node smoothing to make the surrounding cells as uniform in size as possible (see Figure 1.15).*

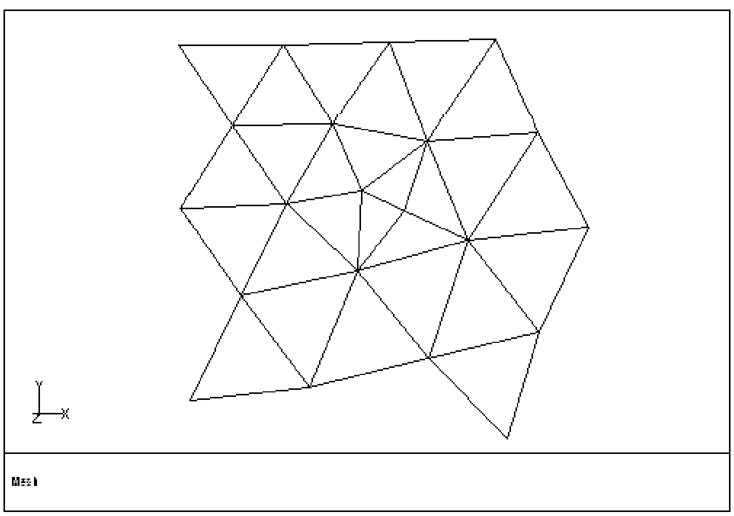


Figure 1.15: Surface Mesh After Node Smoothing

From this point onward, the tutorial attempts to demonstrate some of the additional face modification tools that are available in TGrid *using the cluster of cells shown in Figure 1.15.*

4. Modify the mesh by edge swapping.

   (a) Select edge in the Filter list.

   (b) Select the edges to be swapped (Figure 1.16).

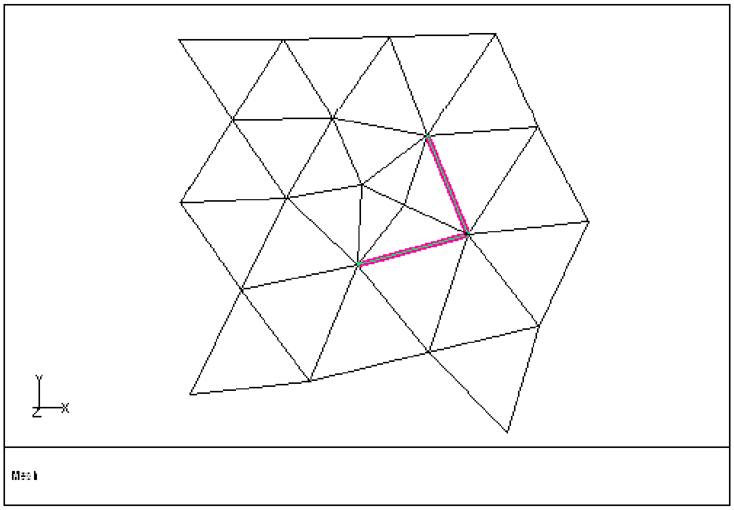


Figure 1.16: Edges Selected for Swapping

   (c) Click Swap in the Operation group box.

   TGrid *will swap the selected edges and retriangulate the mesh (Figure 1.17). This operation did little to produce a better quality mesh. You can use node smoothing to fix this problem.*

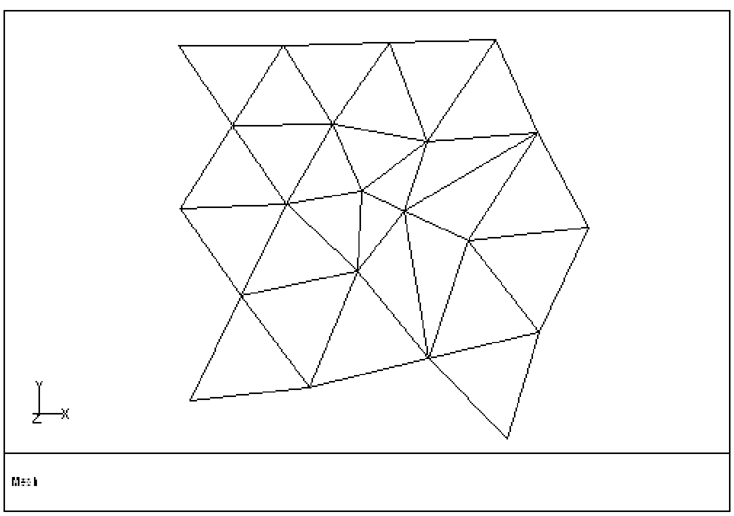


Figure 1.17: Surface Mesh After Edge Swapping

   (d) Select node in the Filter list.

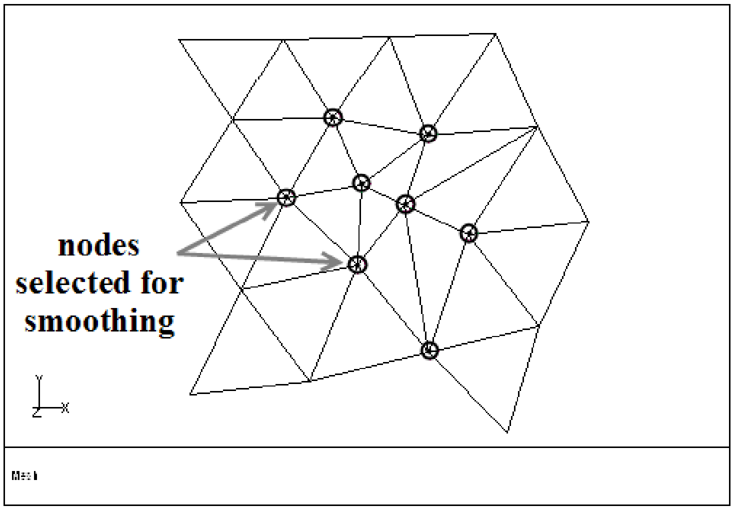


Figure 1.18: Nodes Selected for Smoothing

(e) Select the nodes in the vicinity of the swapped edge (Figure 1.18).

(f) Click Smooth in the Operation group box (Figure 1.19).

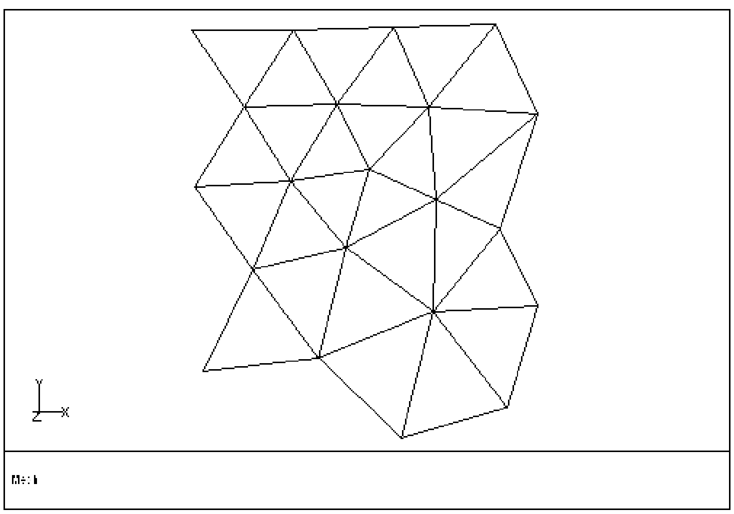


Figure 1.19: Surface Mesh After Node Smoothing

5. Modify the mesh by splitting edges.

   (a) Select edge in the Filter list.

   (b) Select the edge to be split (Figure 1.20).

   (c) Click Split in the Operation group box (Figure 1.21).

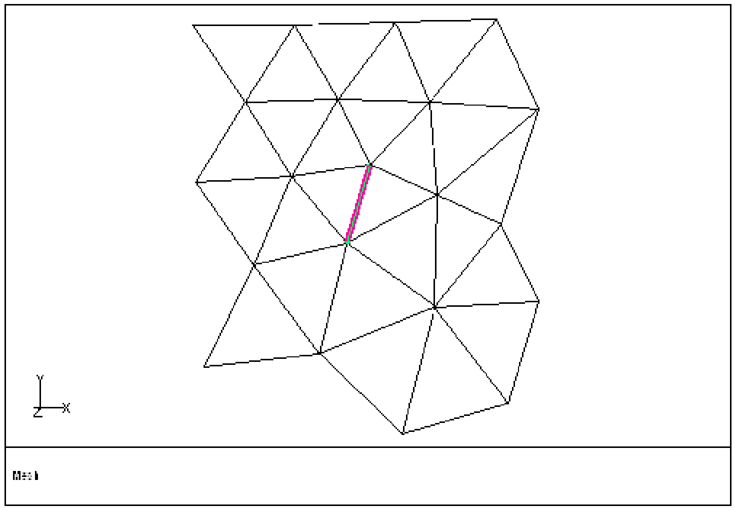


Figure 1.20: Edge Selected for Splitting

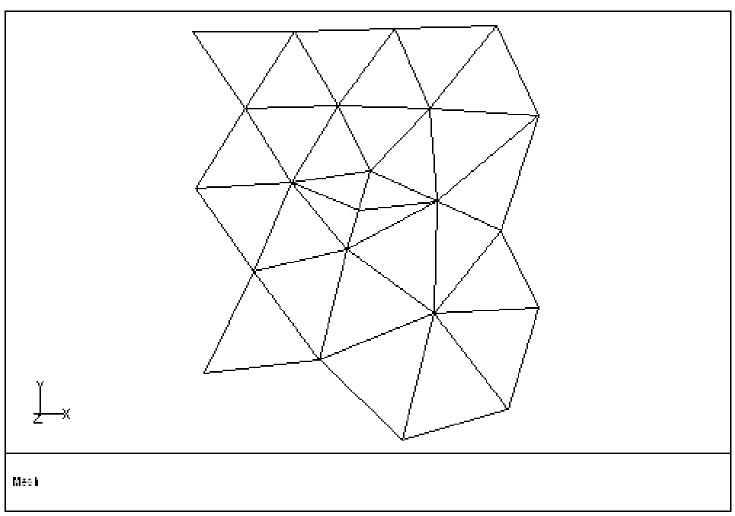


Figure 1.21: Surface Mesh After Edge Splitting

(d) Perform node smoothing by selecting several nodes around the split edge and clicking Smooth (Figure 1.22).

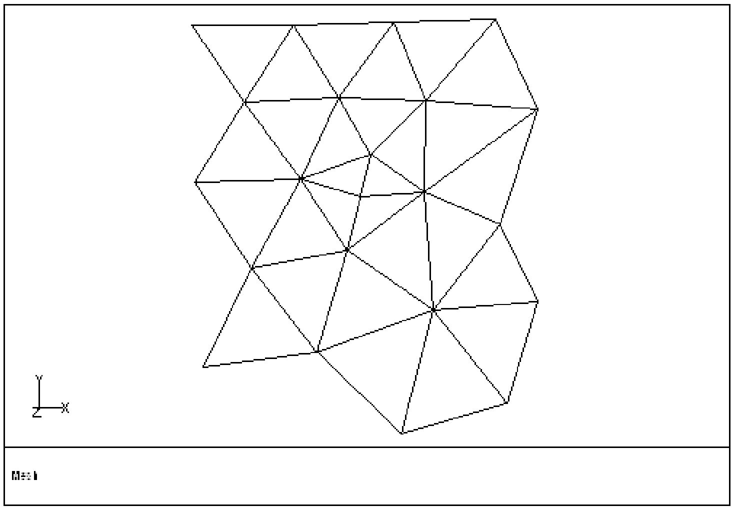


Figure 1.22: Surface Mesh After Node Smoothing

6. Modify the mesh by splitting faces.

   (a) Select face in the Filter list.

   (b) Select the face to be split (Figure 1.23).

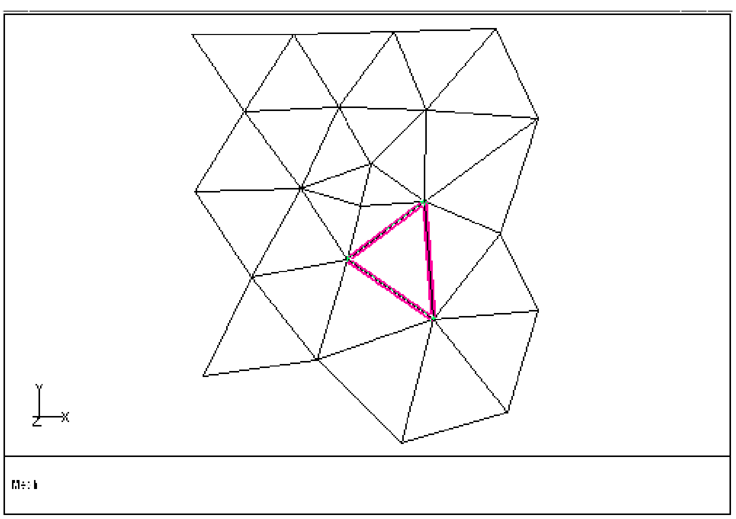


Figure 1.23: Face Selected for Splitting

   (c) Click Split in the Operation group box to split the face (Figure 1.24).

   (d) Swap the edges of the split face (Figure 1.25).

   (e) Smooth the nodes in the vicinity of the split face (Figure 1.26).

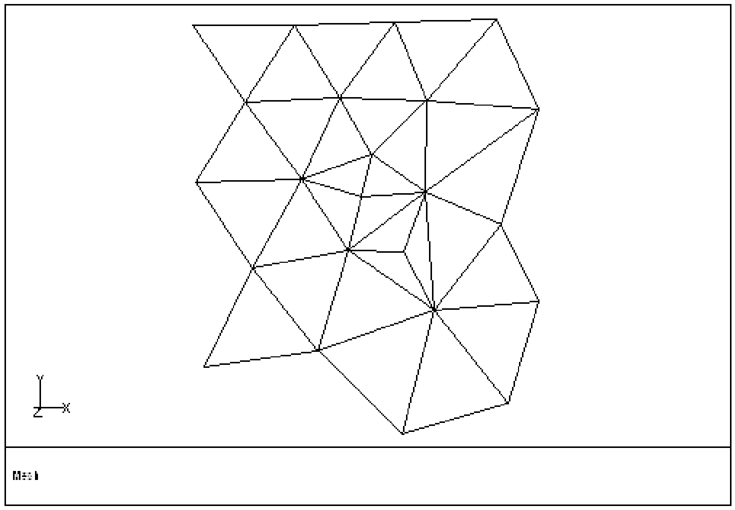


Figure 1.24: Surface Mesh After Splitting the Face

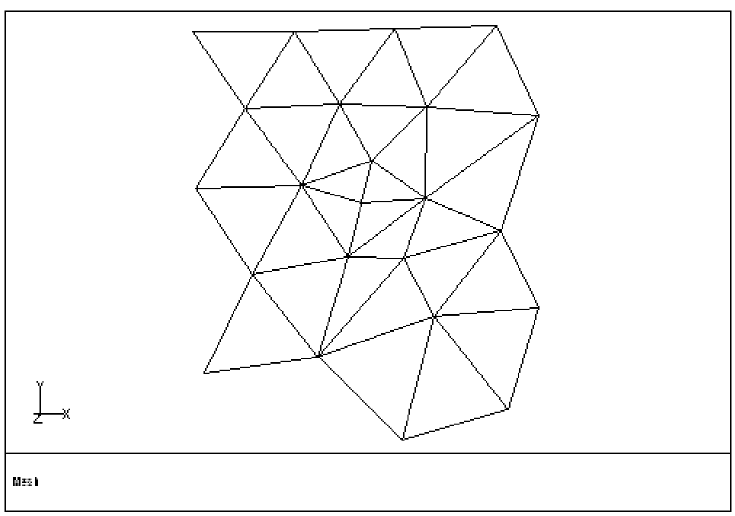


Figure 1.25: Surface Mesh After Edge Swapping

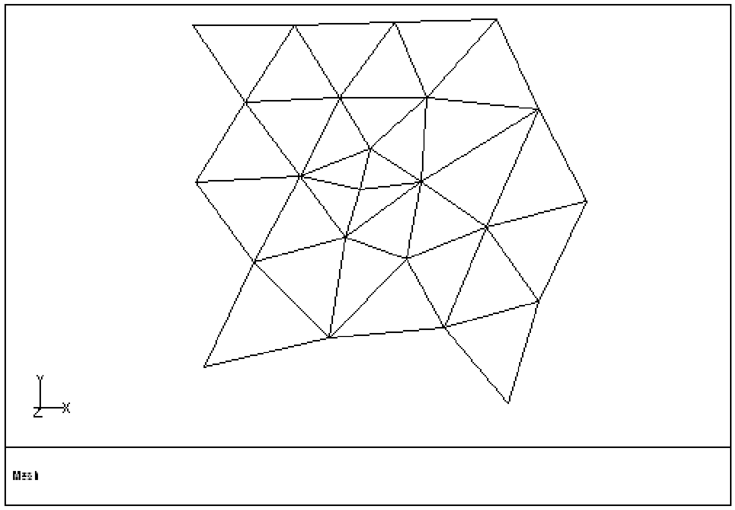
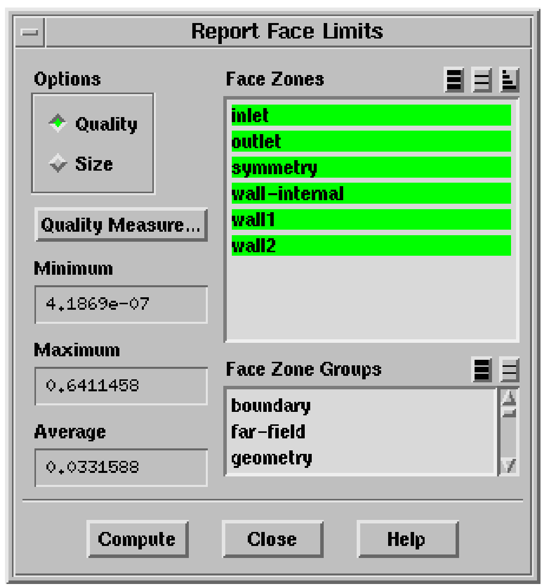


Figure 1.26: Surface Mesh After Node Smoothing

7. Check the maximum face skewness.

   Report ⟶Face Limits...



   (a) Select Quality in the Options list.

   (b) Select all the zones in the Face Zones selection list.

   (c) Click Compute.

   TGrid *will report the* Minimum, Maximum, *and* Average *face skewness.*

   (d) Close the Report Face Limits panel.

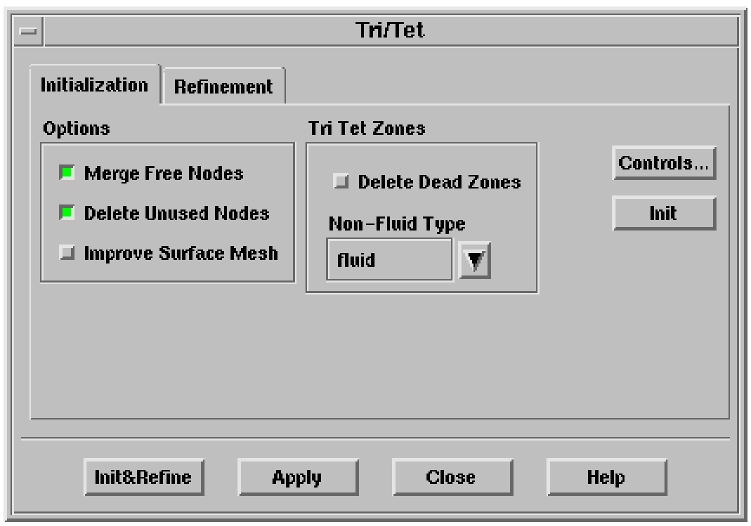
The maximum face skewness at this point in the tutorial is less than 0.65. There are nine faces with a skewness greater than 0.6 (this information was obtained from the Face Distribution *panel). You can try and reduce the maximum face skewness to a value less than 0.6 using the face modification tools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

Step 8: 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* Tri/Tet *panel and then generate the volume mesh.*

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

   Mesh ⟶Tri/Tet...



   (a) Select fluid in the Non-Fluid Type drop-down list in the Tri Tet Zones group box.

   (b) Click Apply and close the Tri/Tet panel.

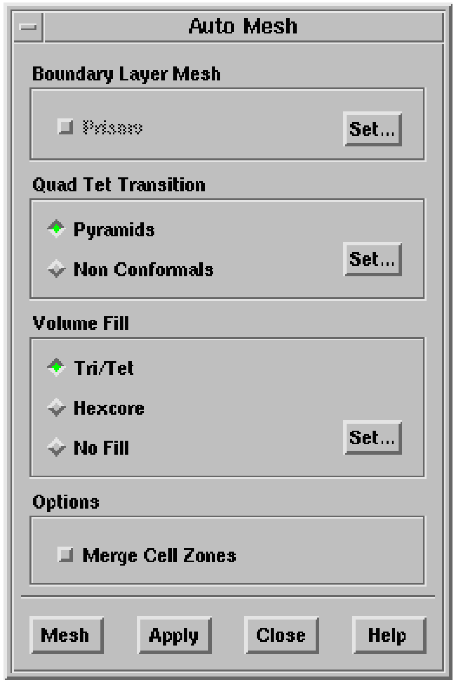
   By default, TGrid *automatically makes the cell zone with the largest volume the active fluid zone.* TGrid *treats the remaining cell zones (non-fluid zones) as dead zones and does not refine them. Hence, if you want to mesh multiple zones, change the* Non-Fluid Type *to* solid *or* fluid *depending on the problem.*

   When Non-Fluid Type *is set to a type other than* dead, TGrid *treats all the zones as active zones and automatically refines these zones.*

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

2. Generate the volume mesh.

   Mesh ⟶Auto Mesh...

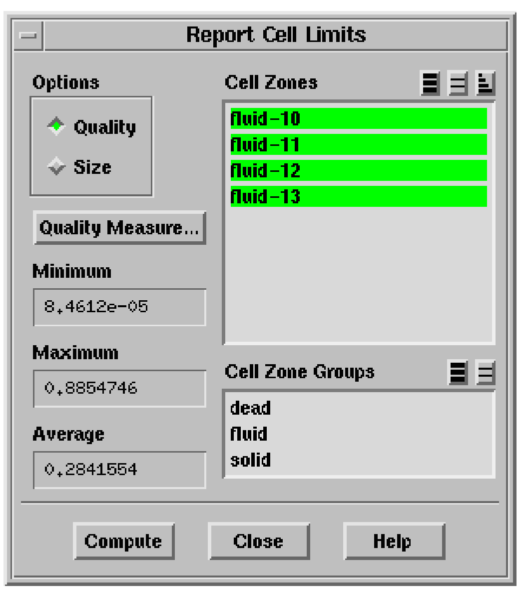


   (a) Retain the default settings and click Mesh.

   (b) Close the Auto Mesh panel.

Step 9: Check the Volume Mesh

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 panel.

Step 10: Check and Save the Volume Mesh

1. Check the mesh.

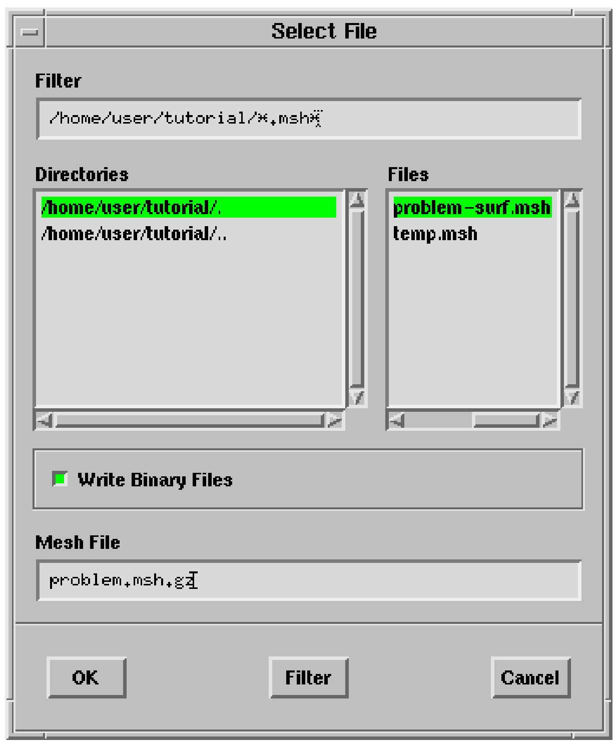
   Check the mesh to ensure it has no negative cell volumes or left-handed faces before saving the mesh file.

   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...



3. Exit TGrid.

   File ⟶Exit

Summary

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

Tutorial 2. Tetrahedral Mesh Generation

Introduction

The mesh generation process is highly automated in TGrid. 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 TGrid to fail 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 sufaces.
- Generate the tetrahedral volume mesh using the various refinement options available in TGrid.
- 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 maximum cell volume.
- Examine the effect of the growth factor.
- Create a local refinement region.

Prerequisites

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

Preparation

1. Download `tet-mesh.zip` from the FLUENT User Services Center. This file can be found from the Documentation link on the TGrid product page.

   OR

   Copy `tet-mesh.zip` from the TGrid documentation CD to your working directory.

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

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

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

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

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

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

2. Unzip `tet-mesh.zip`.

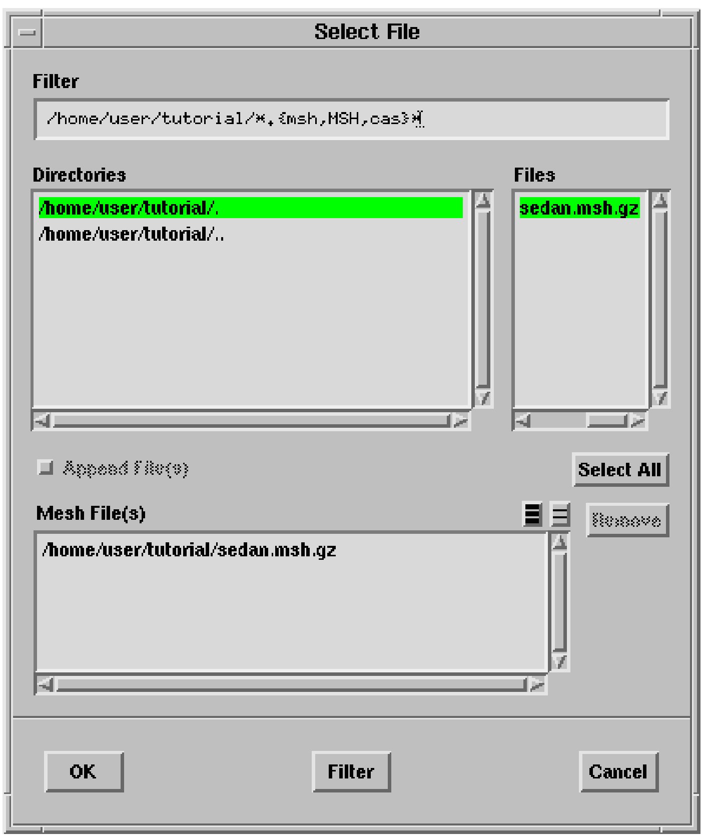
   The file, `sedan.msh.gz` *can be found in the* `tet-mesh` *folder created on unzipping the file.*

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

Step 1: Read and Display the Boundary Mesh

1. Read the boundary mesh.

   File ⟶ Read ⟶ Boundary Mesh...

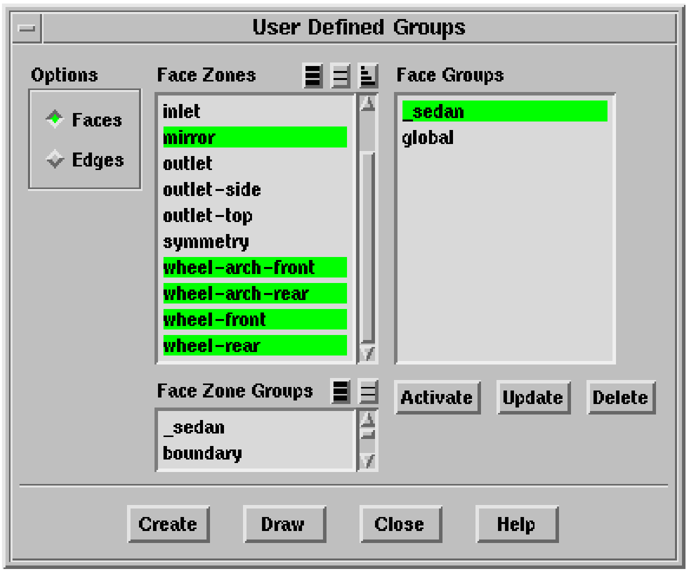


(a) Select sedan.msh.gz in the Files list.

(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.

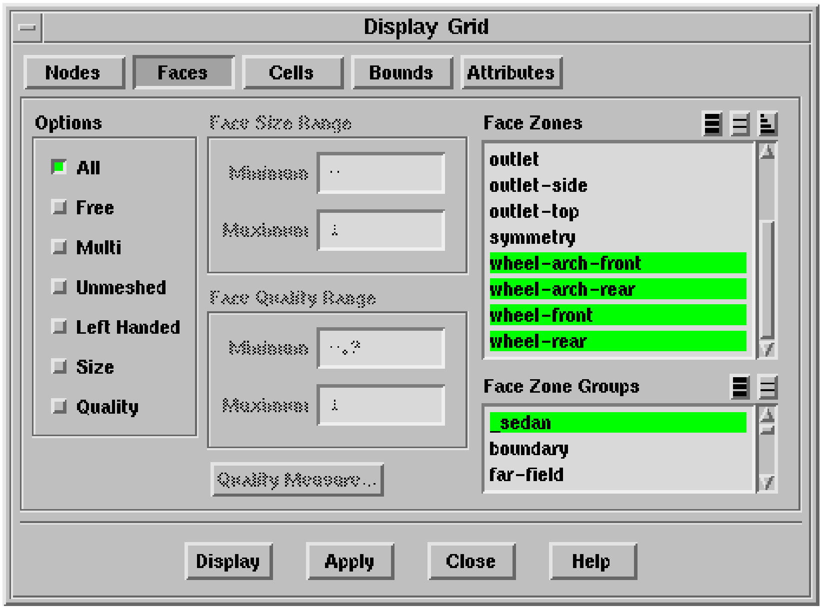
The _sedan *group will now be available in the* Face Groups *list in the* User Defined Groups *panel.*

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 panels.*

(d) Close the User Defined Groups panel.

3. Display the boundary mesh (Figure 2.1).

   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.

   (c) Click the Colors... button to open the Grid Colors panel.

      i. Select Color by ID in the Options list.

      ii. Close the Grid Colors panel.

   (d) Enable Hidden Line Removal in the Display Options panel.

      Display ⟶Options...

   (e) Click Display.

   (f) Close the Display Grid panel.

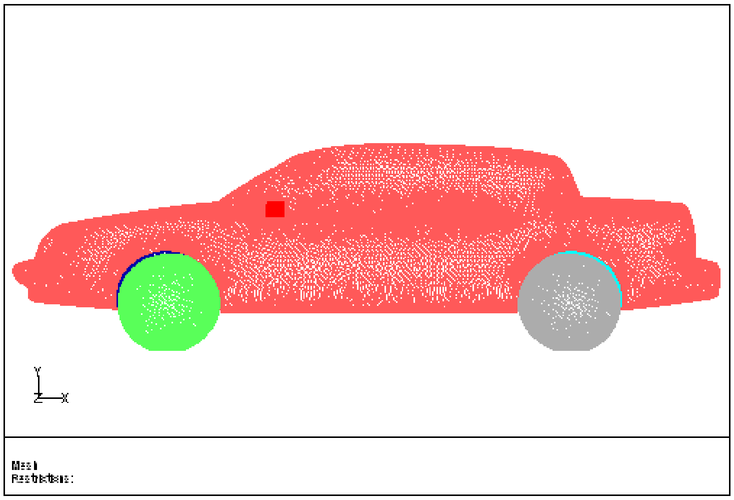
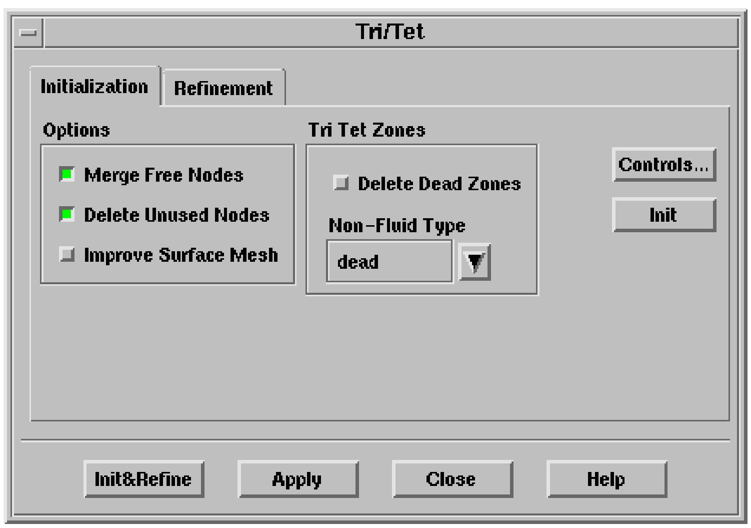


Figure 2.1: Grid Display

Step 2: Generate the Mesh using the Skewness-Based Refinement Method

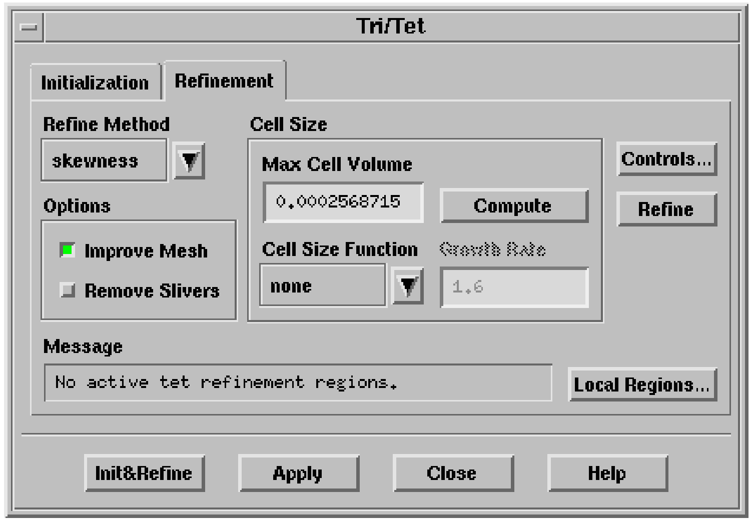
1. Specify the meshing parameters.

   Mesh ⟶Tri/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 (`2.57e-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/tritet/local-regions/ideal-vol` *to calculate the volume of an ideal equilateral tetrahedron (side $\times$ side $\times$ side $\times \dfrac{\sqrt{2}}{12}$) 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 1.18e-4), but the mesh would be larger and slower to generate. Hence, you will use the default value for the maximum cell volume.

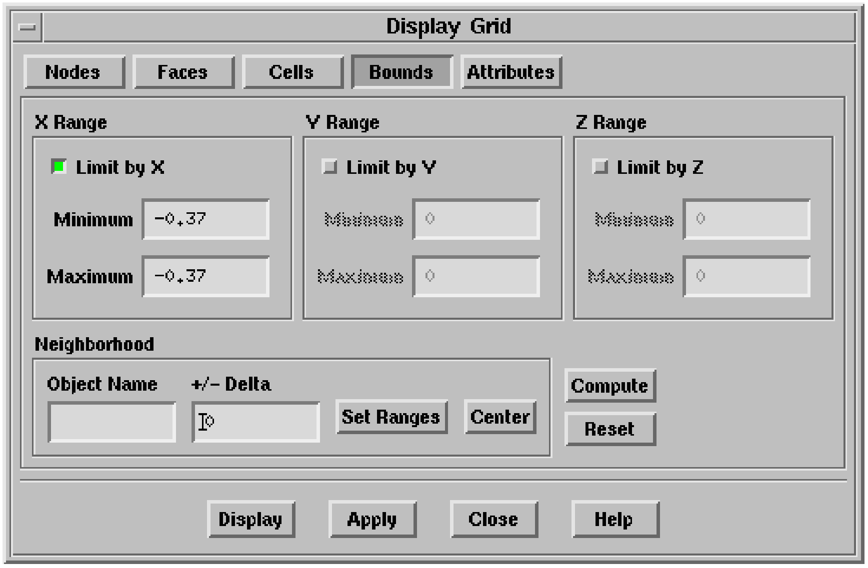
(c) Click Apply and Init&Refine.

(d) Close the Tri/Tet panel.

2. Examine the mesh.

   Display →Grid...

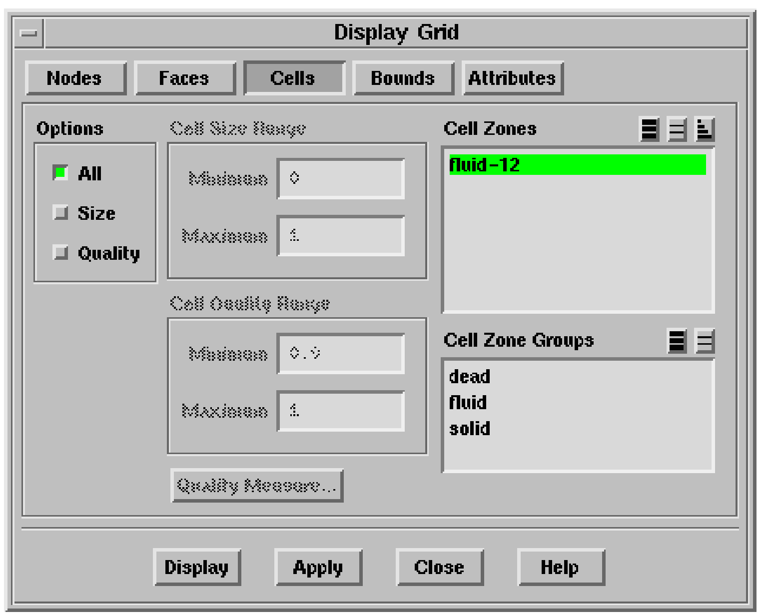
   (a) Display the mesh on a slide through the mirror and the car (Figure 2.2).



   i. Click the Bounds tab and enable Limit by X.

   ii. Enter -0.37 for Minimum and Maximum in the X Range group box.

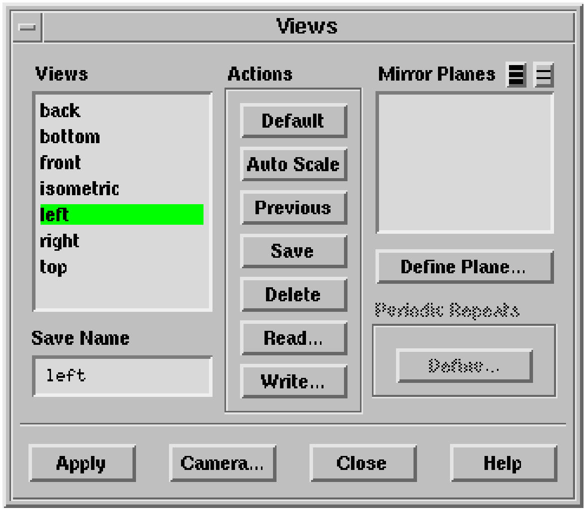
   iii. Click the Cells tab and select the fluid zone in the Cell Zones selection list.



   iv. Enable All in the Options group box and click Display.

v. Display the left view.

Display ⟶Views...



A. Select left in the Views list and click Apply.

B. Click Auto Scale.

C. Close the Views panel.

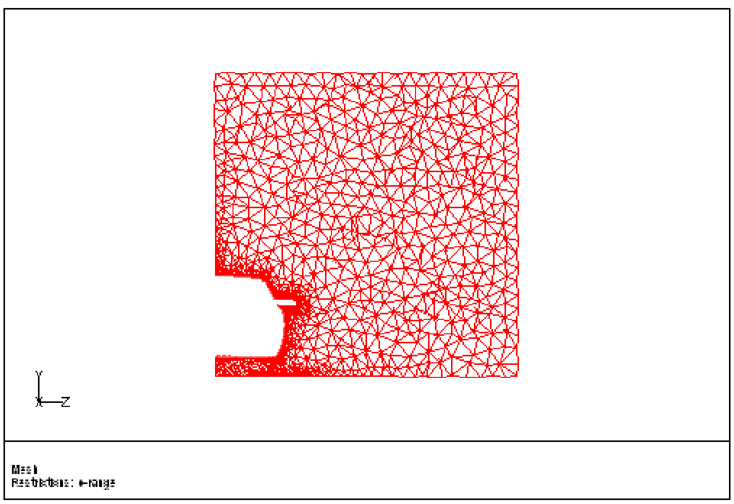


Figure 2.2: Slide of Cells at X = -0.37

(b) Display the mesh on a slide through the wheels (Figure 2.3).

i. Click Reset in the Bounds tab of the Display Grid panel.

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 sedan to examine the cell growth.

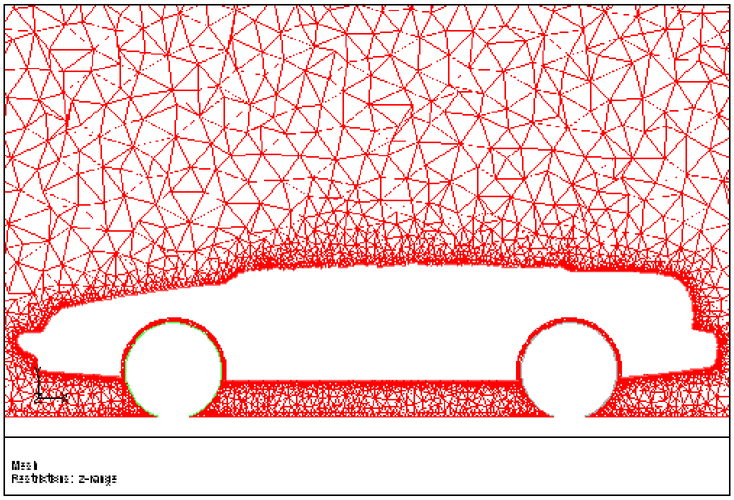


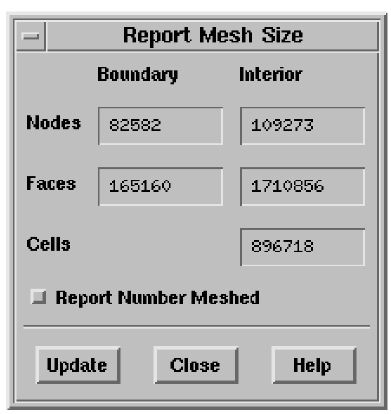
Figure 2.3: Slide of Cells at Z = 0.38

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

v. Close the Display Grid panel.

(c) Check the number of cells.

Report ⟶Mesh Size...



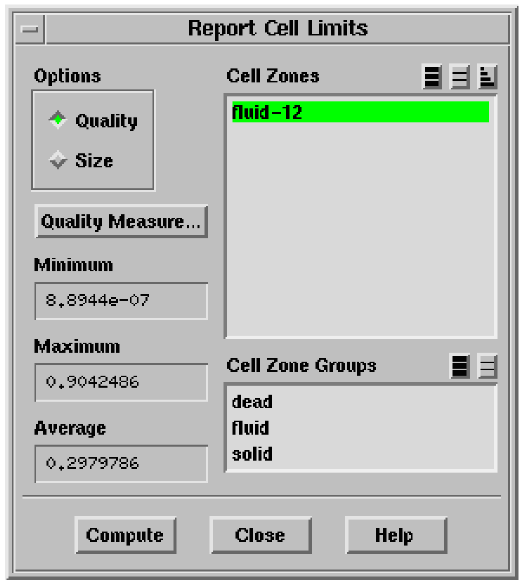
i. Click Update.

The number of cells is 896718. The exact number may differ on different platforms.

ii. Close the Report Mesh Size panel.

(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 0.298.

iii. Close the Report Cell Limits panel.

Step 3: 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 ⟶Tri/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 Tri/Tet panel.

(c) Examine the mesh.

Display ⟶Grid...

i. Display the mesh on a slide through the mirror and the car (x = -0.37). See Figure 2.4.

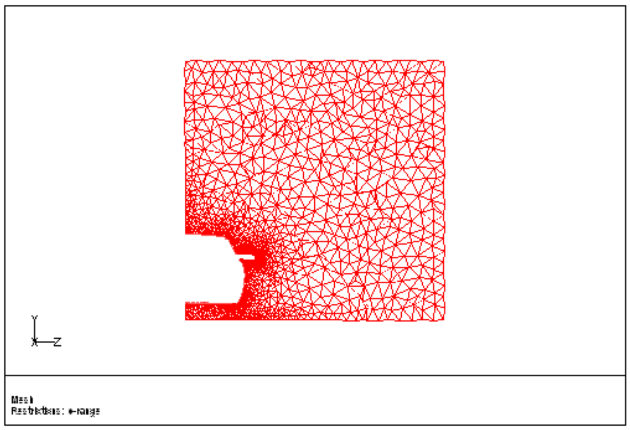


Figure 2.4: Slide of Cells at X = -0.37

ii. Display the mesh on a slide through the wheels (z = 0.38). See Figure 2.5.

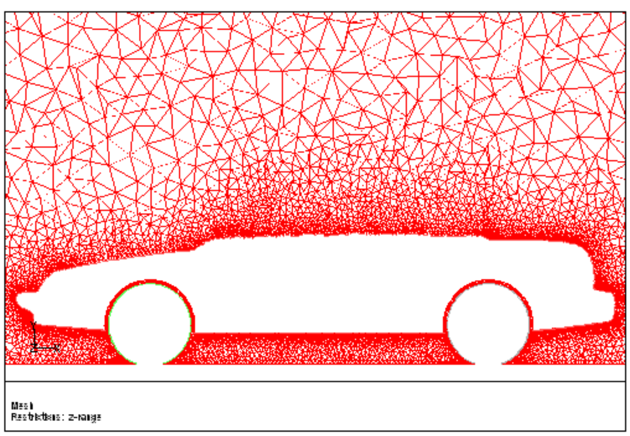


Figure 2.5: Slide of Cells at Z = 0.38

(d) Check the number of cells.

Report →Mesh Size...

The number of cells is 1658326. 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 0.249.

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.

Step 4: 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 →Tri/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 Tri/Tet panel.

   (c) Examine the mesh.

   Display →Grid...

      i. Display the mesh on a slide through the mirror and the car (x = -0.37). See Figure 2.6.

      ii. Display the mesh on a slide through the wheels (z = 0.38). See Figure 2.7.

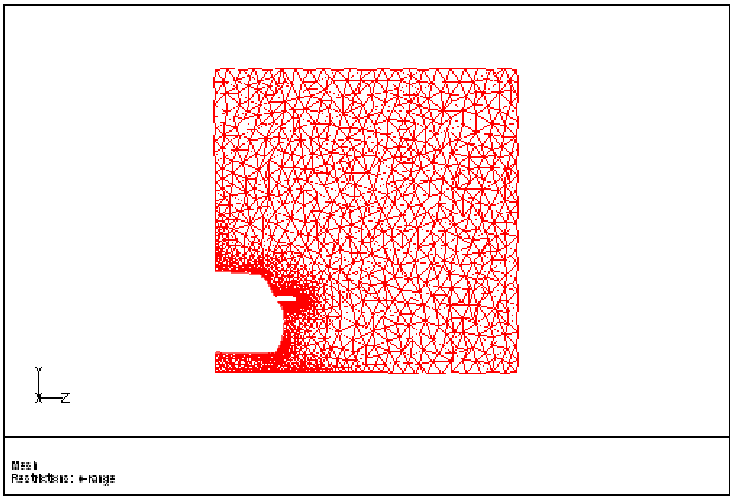


Figure 2.6: Slide of Cells at X = -0.37

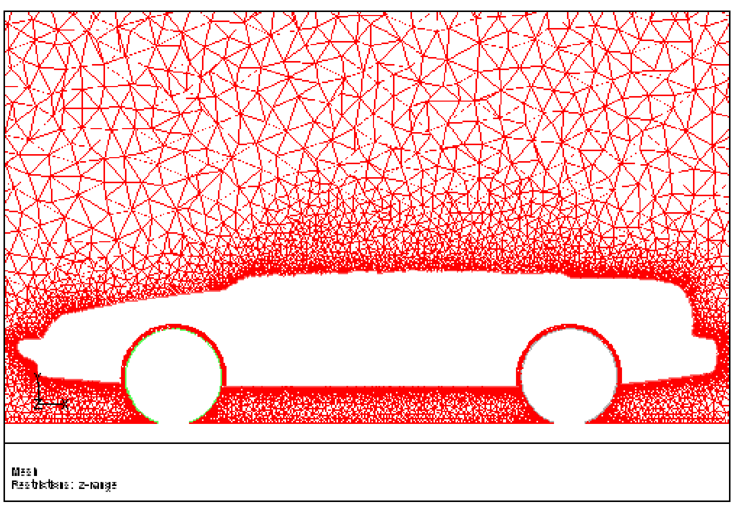


Figure 2.7: Slide of Cells at Z = 0.38

(d) Check the number of cells.

Report ⟶Mesh Size...

The number of cells is 1493701. 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 0.245.

(f) Examine the cell size distribution.

Display ⟶ Plot ⟶Cell Distribution...

i. Select the fluid zone in the Cell Zones selection list.

ii. Select Size from the Options list and click Compute.

The maximum cell volume is 2.23e-4.

Note: *The maximum cell volume (2.23e-4) is lower than the specified value (2.57e-4).*

iii. Close the Cell Distribution panel.

- *The quality is very similar to that obtained with the skewness-based refinement algorithm. Generally, the maximum skewness will be similar for both refinement methods, but the average skewness will be better with the advancing front method.*
- *The maximum cell volume criterion is well respected. In the middle of the domain, the cell volume is defined by that parameter. However, a few cells may still be above the specified size since quality is predominant on the size.*
- *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.*

Step 5: Examine the Effect of the Maximum Cell Volume

1. Clear the mesh.

2. Specify the meshing parameters.

   Mesh ⟶Tri/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 Tri/Tet panel.

   (c) Click Apply and Init&Refine.

   (d) Close the Tri/Tet panel.

3. Examine the mesh.

   Display ⟶Grid...

   (a) Display the mesh on a slide through the mirror and the car (x = -0.37). See Figure 2.8.

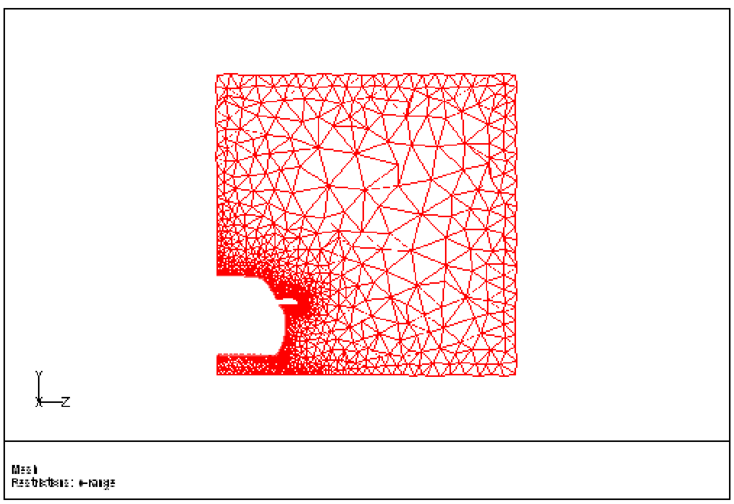


Figure 2.8: Slide of Cells at X = -0.37

   (b) Display the mesh on a slide through the wheels (z = 0.38). See Figure 2.9.

   (c) Check the number of cells.

   Report ⟶Mesh Size...

   The number of cells is 1339210. 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 0.240.

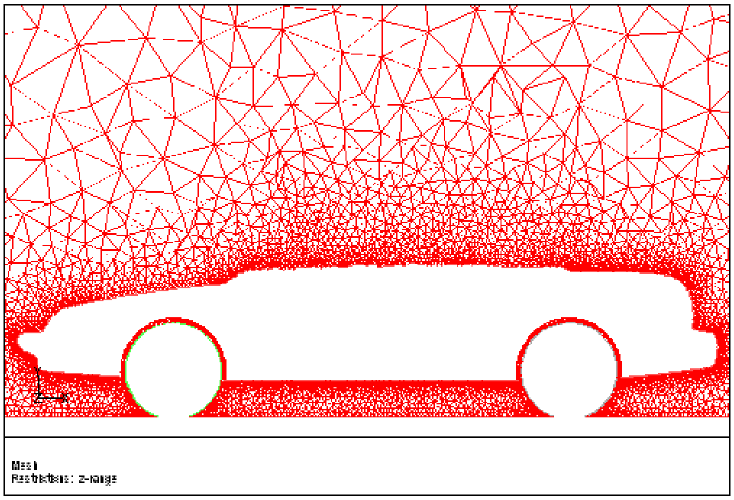


Figure 2.9: Slide of Cells at Z = 0.38

(e) Check the maximum cell volume.

Report ⟶Cell Limits...

i. Select the fluid zone in the Cell Zones selection list.

ii. Select Size from the Options list and click Compute.

The maximum cell volume is 4.69e-3 which is lower than the specified value (2e-2). For the mesh generated, the maximum cell volume is lower than the value specified, indicating that the size distribution was based on the surface mesh, the growth rate, and the quality, and not restricted by the maximum cell volume specified. Hence, the edge length of the cells in the centre of the domain is more than the edge length of the cells on the boundary.

Step 6: Examine the Effect of the Growth Factor

1. Clear the mesh.

2. Specify the meshing parameters.

Mesh ⟶Tri/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 Tri/Tet panel.

(c) Click Apply and Init&Refine.

(d) Close the Tri/Tet panel.

3. Examine the mesh.

   Display →Grid...

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

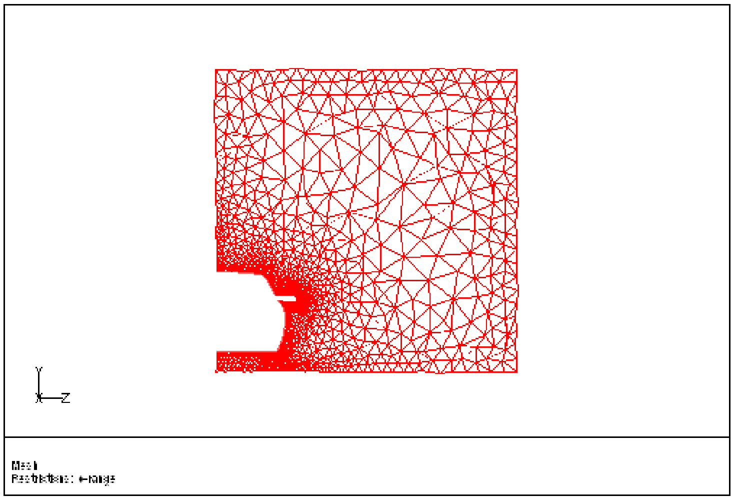


Figure 2.10: Slide of Cells at X = -0.37

   (b) Display the mesh on a slide through the wheels (z = 0.38). See Figure 2.11.

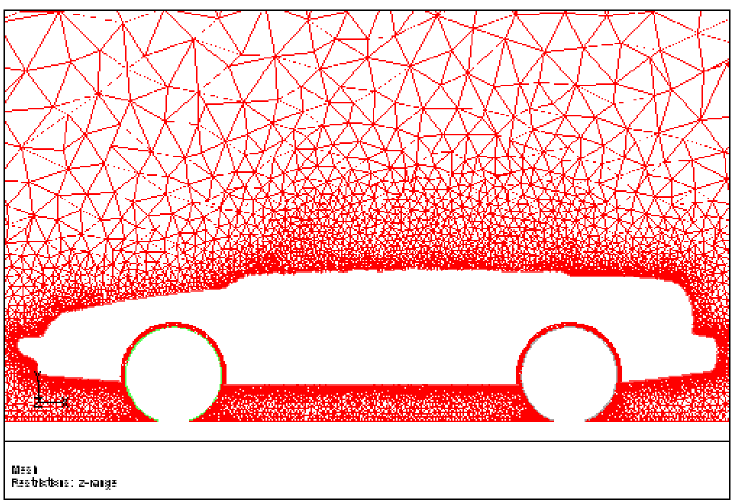


Figure 2.11: Slide of Cells at Z = 0.38

   (c) Check the number of cells.

   Report →Mesh Size...

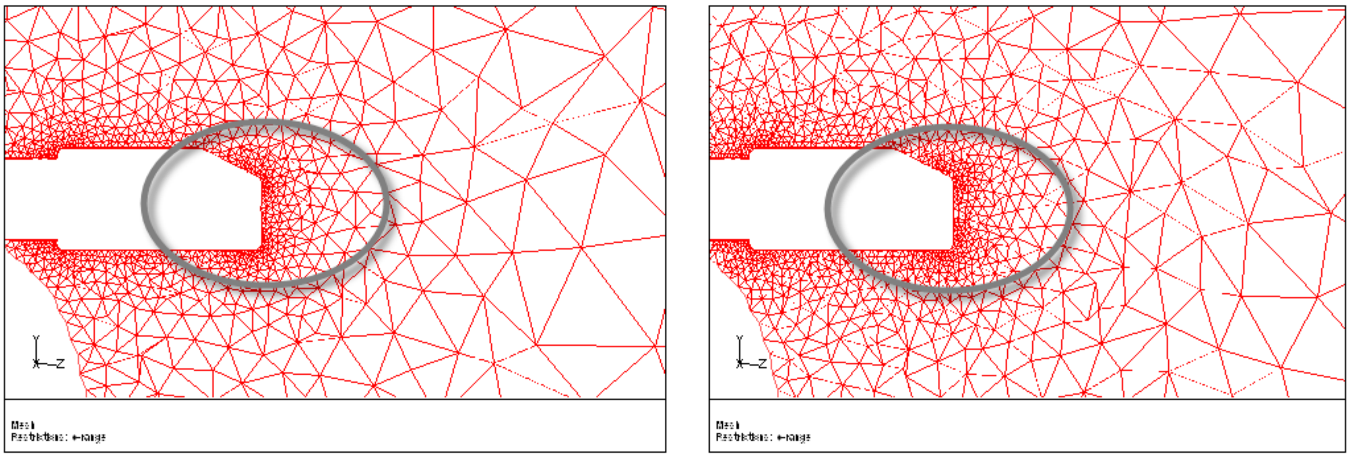
   The number of cells is 1621399. The exact number may differ on different platforms.

(d) Check the maximum skewness.

Report ⟶Cell Limits...

The maximum skewness is 0.904, while the average skewness is 0.230.

For the mesh generated, the mesh transition is smoother (see Figure 2.12), however the number of cells generated is significantly more.



(A) Growth Factor = 1.3 **(B) Growth Factor = 1.25**

Figure 2.12: Comparison of Meshes Based on Growth Factor

Step 7: Generate a Local Refinement in the Wake of the Car

In TGrid *5.0, 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. In this step, you will generate 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.

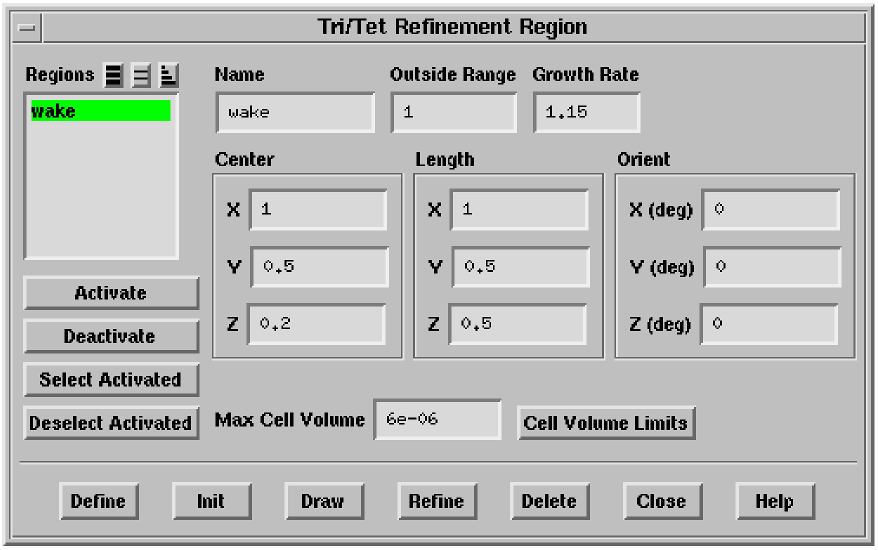
   (c) Close the Display Grid panel.

3. Specify the meshing parameters.

(a) Retain the previous settings in the Initialization tab of the Tri/Tet panel.

Mesh —→Tri/Tet...

(b) Click the Local Regions... button in the Refinement tab to open the Tri/Tet Refinement Region panel.



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 Tri/Tet Refinement Region panel.

TGrid *will report (in the* Message *field) 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 Tri/Tet panel.

4. Examine the mesh.

   Display ⟶Grid...

   (a) Display the mesh on a slide through the mirror and the car (x = -0.37). See Figure 2.13.

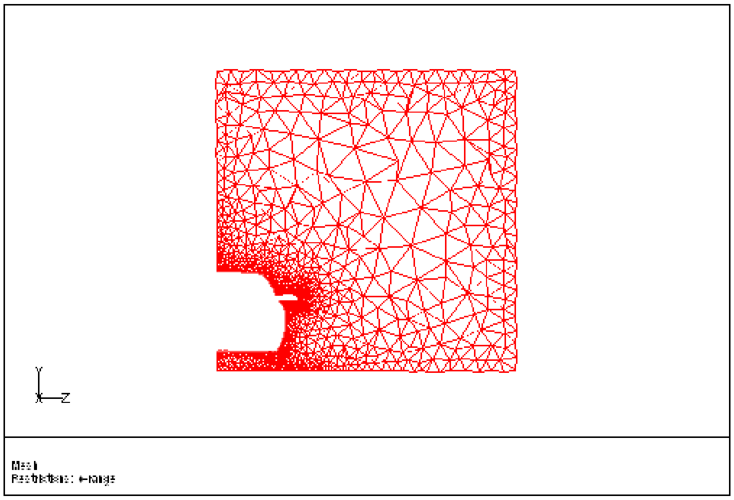
   

   Figure 2.13: Slide of Cells at X = -0.37

   (b) Display the mesh on a slide through the wheels (z = 0.38).

   (c) Display the refinement region along with the cells (Figure 2.14).

      i. Click the Local Regions... button in the Tri/Tet panel to open the Tri/Tet Refinement Region panel.

         Mesh ⟶Tri/Tet...

      ii. Make sure wake is selected in the Regions selection list and click Draw.

      iii. Close the Tri/Tet Refinement Region panel.

   (d) Check the number of cells.

      Report ⟶Mesh Size...

      The number of cells is 1389502. 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 0.238.

      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 2.15). Further manual operations to obtain better quality are not required in this case.

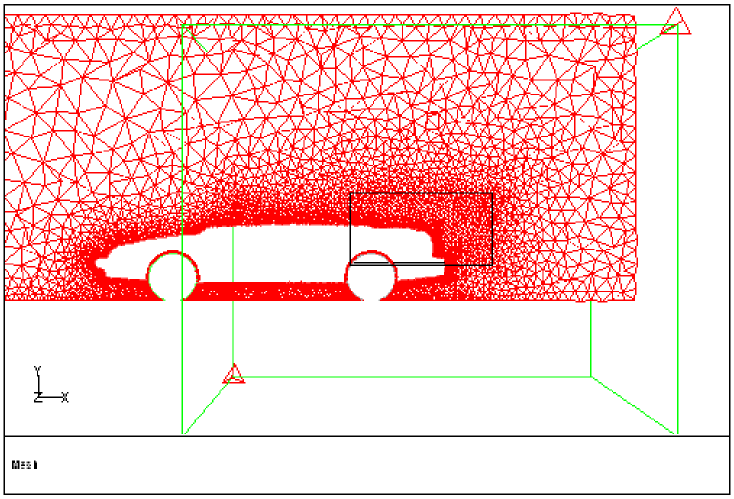


Figure 2.14: Slide of Cells at Z = 0.38 with the Refinement Region

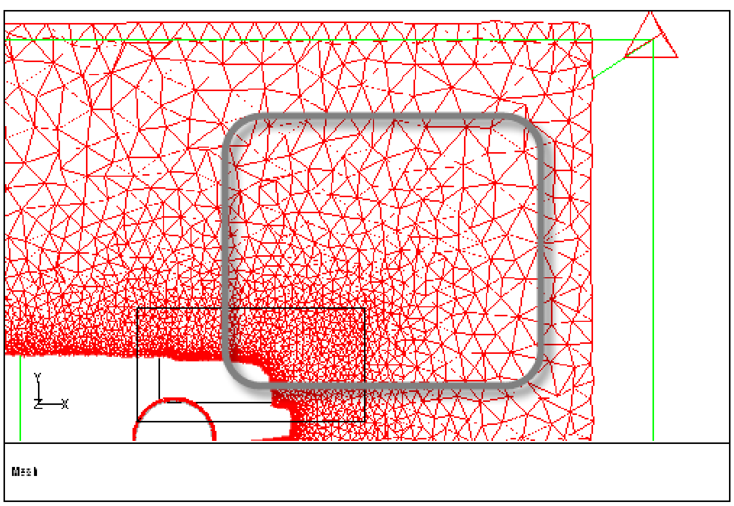


Figure 2.15: Transition Between Cells in Locally Refined Region and the Rest of the Domain

Step 8: Check and Save the Volume Mesh

1. Check the mesh.

   Mesh ⟶Check

   TGrid *will perform various checks on the mesh and report the progress 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 TGrid.

   File ⟶Exit

Summary

This tutorial demonstrated the tetrahedral mesh generation process using both the refinement methods available in TGrid. It also examined the effect of the size function, maximum cell volume, 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.

Tutorial 3. Zonal Hybrid Mesh Generation

Introduction

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

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

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

Prerequisites

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

Preparation

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

OR

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

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

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

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

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

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

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

2. Unzip `zonal-hybrid.zip`.

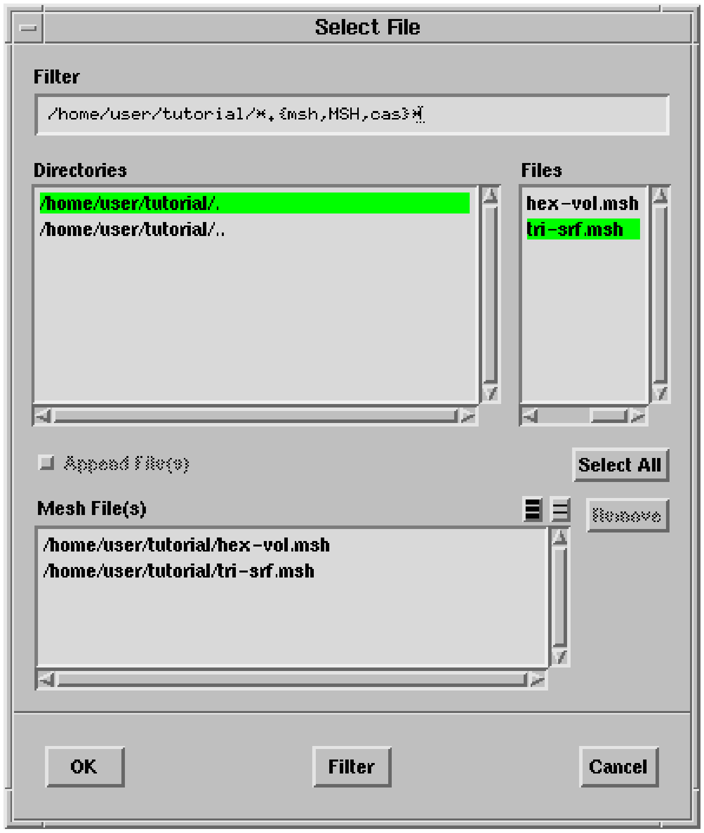
   The files, `hex-vol.msh` *and* `tri-srf.msh` *can be found in the* `zonal-hybrid` *folder created on unzipping the file.*

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

Step 1: Read and Display the Mesh

1. Read the two mesh files.

   File ⟶ Read ⟶ Mesh...



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

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

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

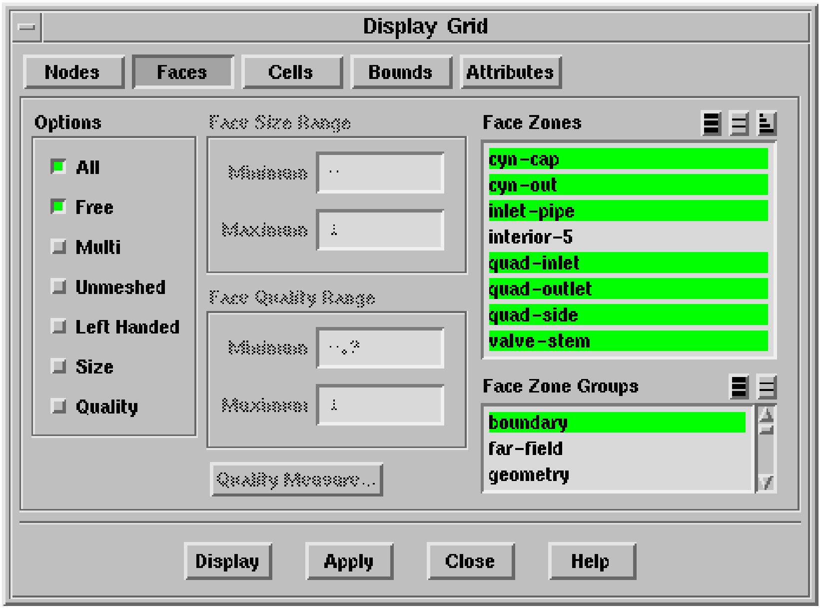
   This file will also be added to the Mesh File(s) *list.*

   (c) Click OK.

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

2. Display the boundary mesh (Figure 3.1).

   Display →Grid...



   (a) Select boundary in the Face Zone Groups selection list to select all the boundary zones in the Face Zones selection list.

   (b) Make sure Free is enabled (in addition to the default, All) in the Options group box.

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

   (c) Click the Attributes tab and disable Filled in the Options group box.

   (d) Click Display.

   (e) Zoom in to focus on the free nodes (Figure 3.2).

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

   (f) Close the Display Grid panel.

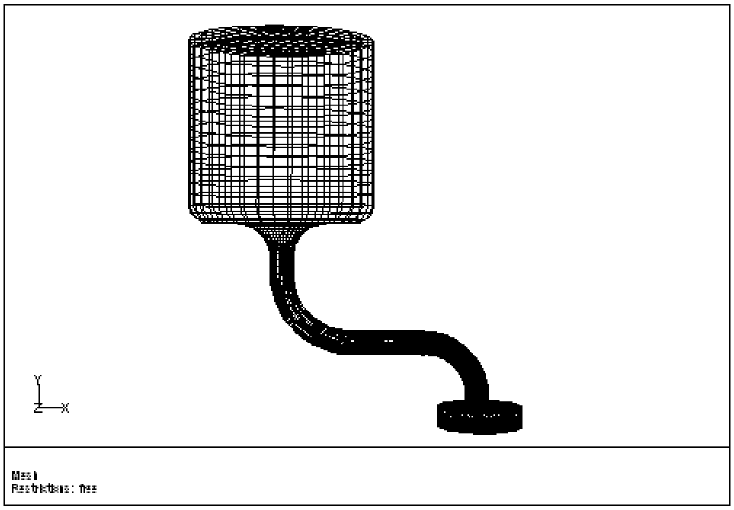


Figure 3.1: Boundary Mesh for the Valve Port

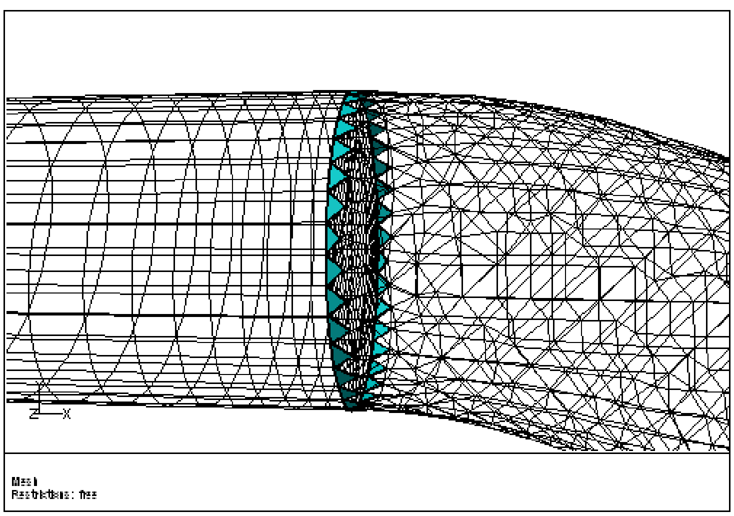


Figure 3.2: Free Nodes at the Intersection of the Tri and Quad Boundary Meshes

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

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

Boundary ⟶Merge Nodes...

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

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

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

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

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

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

3. Click Count Free Nodes.

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

4. Click Merge to merge the free nodes.

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

5. Enter 0.002992057 for Tolerance.

6. Click Merge.

   The remaining nodes should now be merged.

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

8. Close the Merge Boundary Nodes panel.

9. Save the mesh file.

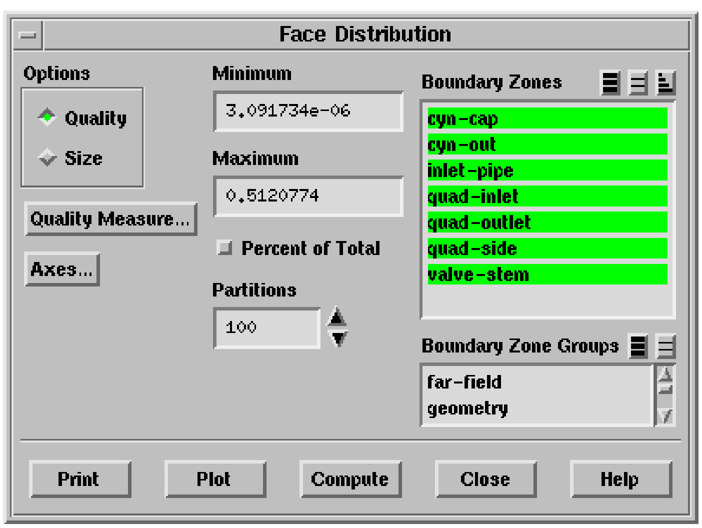
   File ⟶ Write ⟶Mesh...

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

   (b) Click OK to save the mesh.

Step 3: Check the Skewness Distribution of the Boundary Mesh

Display ⟶ Plot ⟶Face Distribution...



1. Select all the zones in the Boundary Zones selection list.

2. Click Compute.

3. Click Plot (Figure 3.3).

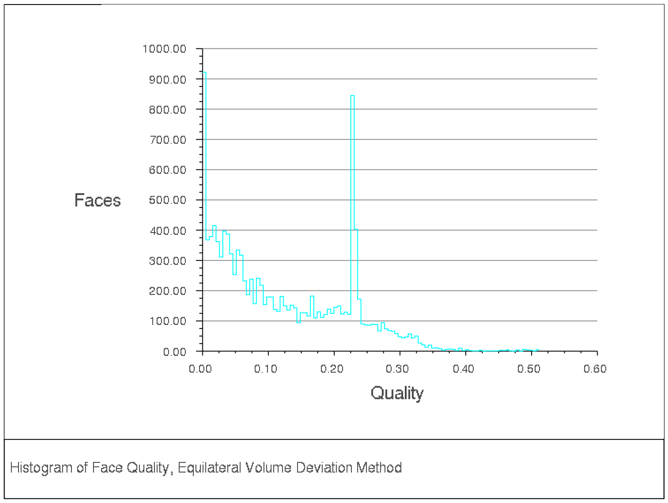


Figure 3.3: Boundary Mesh Skewness Distribution

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

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

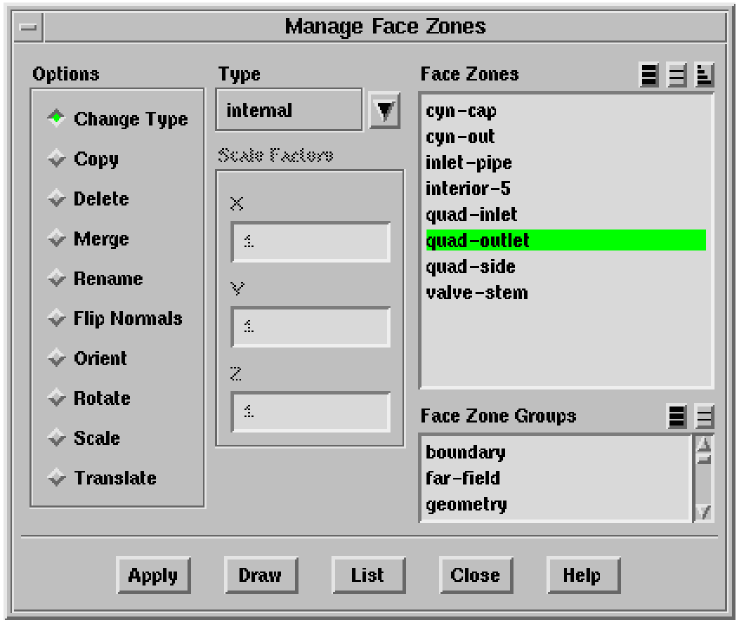
Step 4: Generate the Tetrahedral Mesh Using Pyramids to Transition Between the Hexahedral and Tetrahedral Mesh

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

1. Change the boundary type of quad-outlet.

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

Boundary ⟶Manage...



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

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

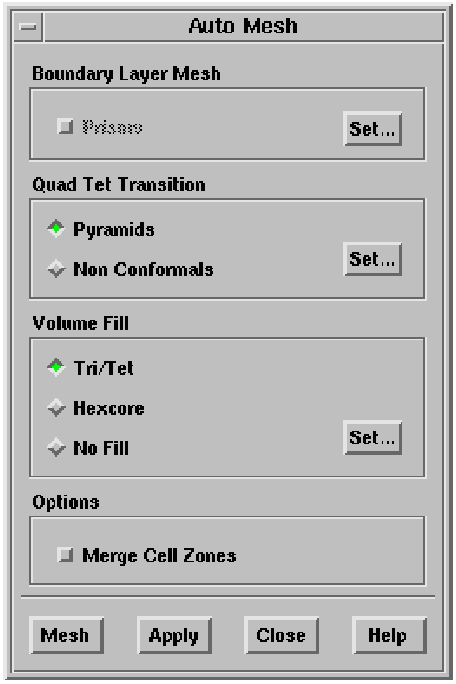
(b) Retain the selection of Change Type in the Options list and select internal in the Type drop-down list.

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

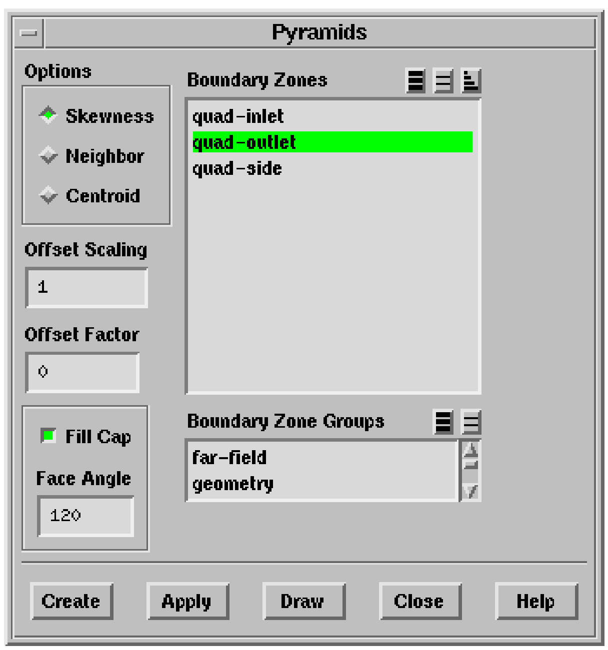
(c) Click Apply and close the Manage Face Zones panel.

2. Set the meshing parameters.

   Mesh ⟶ Auto Mesh...



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



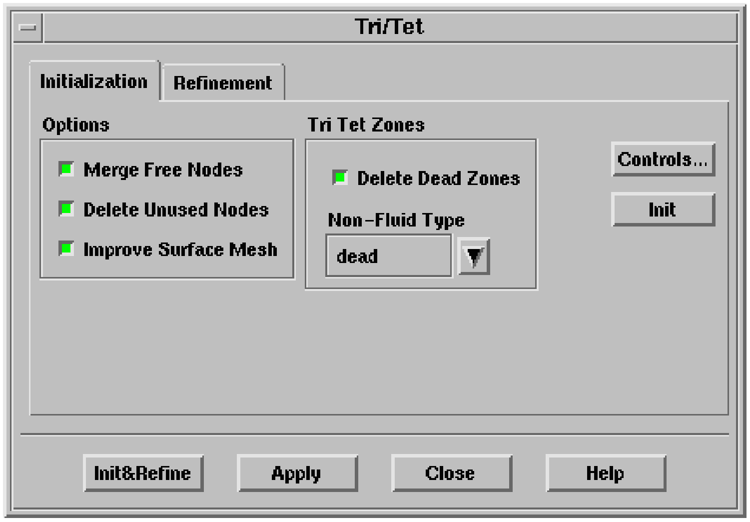
      i. Select quad-outlet in the Boundary Zones selection list.

ii. Retain the selection of Skewness in the Options list.

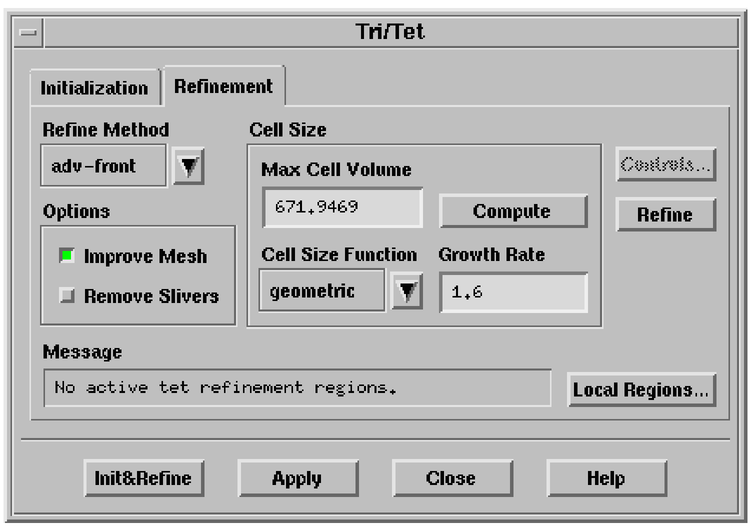
iii. Enable Fill Cap and click Apply.

iv. Close the Pyramids panel.

(b) Retain the selection of Tri/Tet in the Volume Fill list and click the Set... button to open the Tri/Tet panel.



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



ii. Retain the default settings in the Refinement tab.

iii. Click Apply and close the Tri/Tet panel.

(c) Click Apply in the Auto Mesh panel.

(d) Preserve the existing hexahedral mesh.

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

(e) Click Mesh in the Auto Mesh panel.

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

(f) Close the Auto Mesh panel.

3. Display the pyramid cap.

Display ⟶Grid...

(a) Select quad-outlet and quad-outlet-pyramid-cap-# in the Face Zones selection list in the Faces tab.

(b) Disable Free in the Options group box.

(c) Click the Attributes tab and enable Filled and Lights in the Options group box.

(d) Click Display and manipulate the display to obtain the view shown in Figure 3.4.

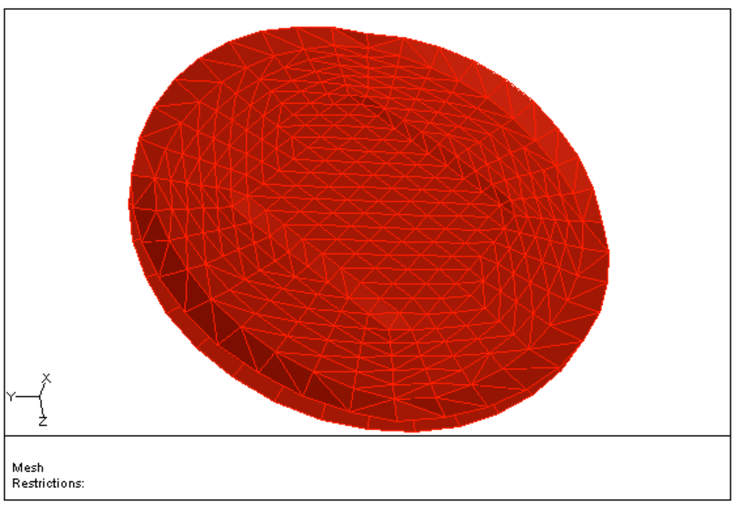


Figure 3.4: Filled Pyramid Cap

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

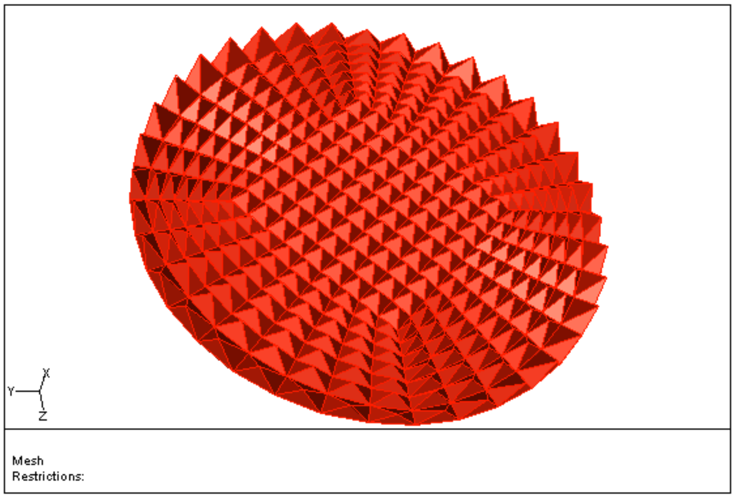


Figure 3.5: Pyramid Cap Without the Fill Cap Option

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

   Display ⟶ Grid...

   (a) Deselect all the previous selections in the Face Zones selection list in the Faces tab.

   (b) Click the Cells tab and select all the zones in the Cell Zones selection list.

   (c) Enable All in the Options group box.

   (d) Click the Attributes tab and enter `0.4` for Shrink Factor.

   (e) Click the Colors... button to open the Grid Colors panel.

      i. Select Color by ID in the Options list.

      ii. Close the Grid Colors panel.

   (f) Click Display and zoom in close to the boundary between the hexahedral and tetrahedral mesh (Figure 3.6).

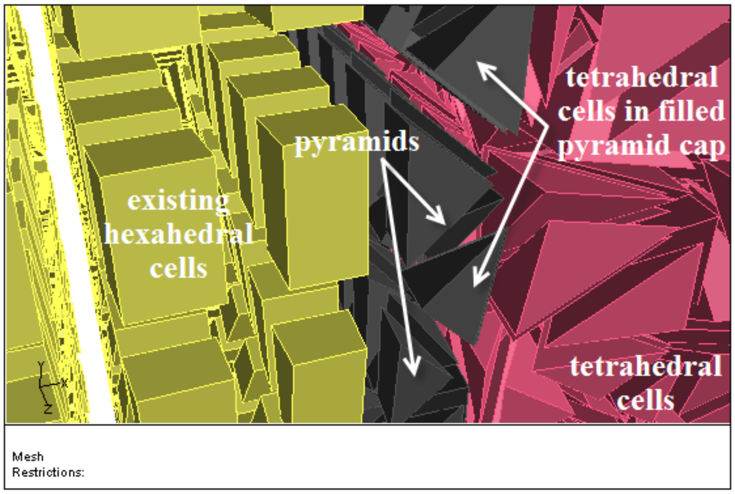
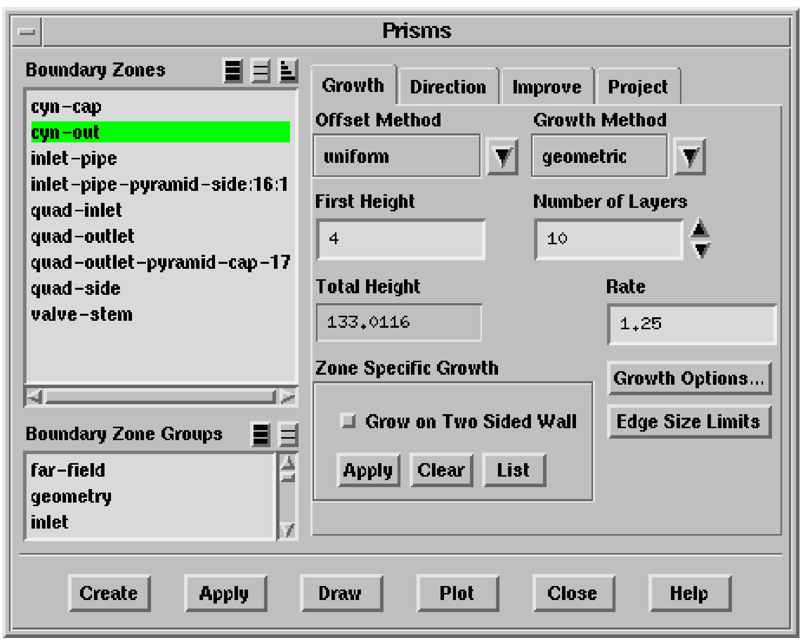


Figure 3.6: Pyramid Transition Between the Hexahedral and Tetrahedral Mesh

Step 5: Extend the Mesh Using Prisms

Mesh ⟶Prisms...



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

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

2. Set the parameters controlling prism growth.

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

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

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

   (c) Enter 10 for Number of Layers.

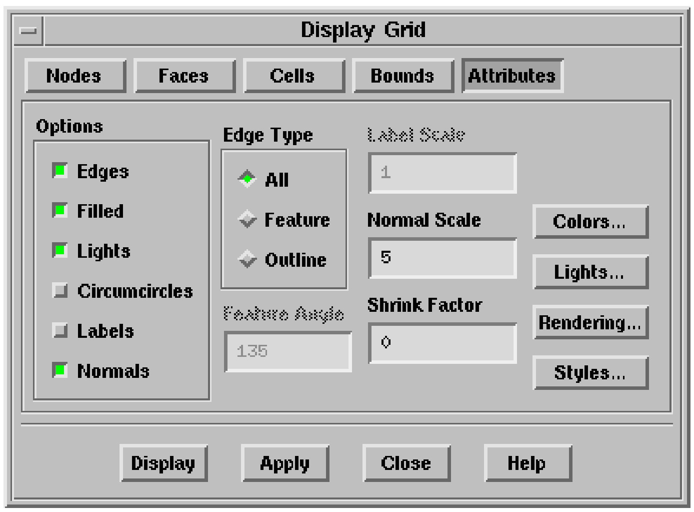
   The Total Height *added by the prisms is slightly more than 133.*

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

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

   (a) Enable the display of normals.

   $\boxed{\text{Display}}$ ⟶Grid...



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

   ii. Enter 5 for Normal Scale.

   Larger normals are easier to see in the grid display.

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

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

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

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

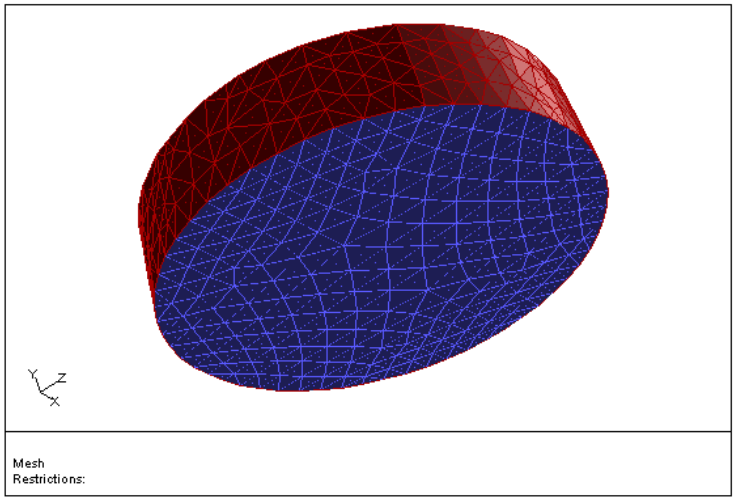


Figure 3.7: Cylinder Normals in Wrong Direction

(e) Click the Direction tab in the Prisms panel (to access the direction parameters), and then click Flip Normals.

(f) Click Display and close the Display Grid panel.

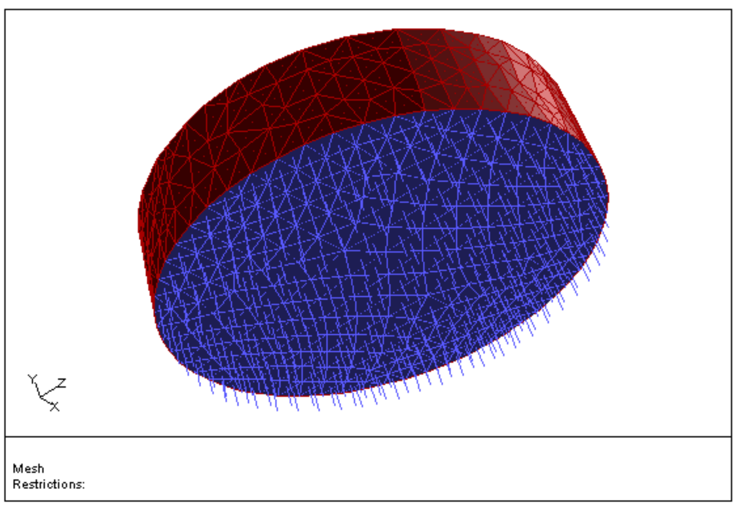
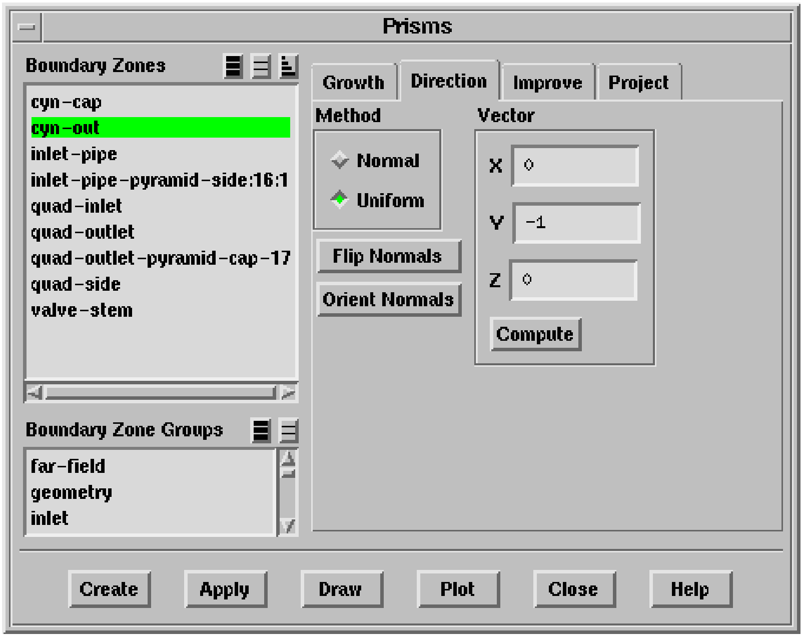


Figure 3.8: Cylinder with Normals in Correct Direction

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

4. Specify the growth direction for the prisms.



(a) 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...

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.

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

```
Prism Layer Summary:

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

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

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

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

1521 interior nodes created in new zone node-#.

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

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

The face and cell zones of interest are as follows:

interior-# *contains the wedge prism cells.*

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

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

8. Close the Prisms panel.

9. Display the new boundaries of the cylinder (Figure 3.9).

   Display ⟶Grid...

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

   (b) Click Display.

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

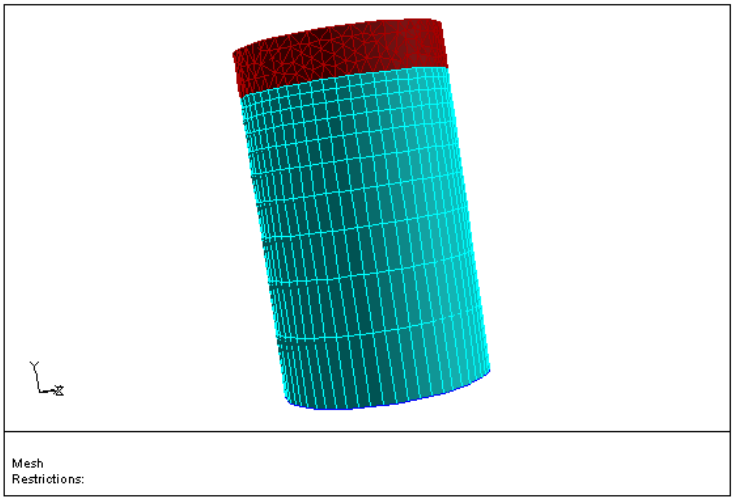


Figure 3.9: Cylinder Extended Using Prisms

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

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

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

    Boundary ⟶Manage...

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

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

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

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

    iii. Click Apply.

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

    i. Select prism-cap-# in the Face Zones list.

    ii. Select pressure-outlet in the Type list.

    iii. Click Apply.

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

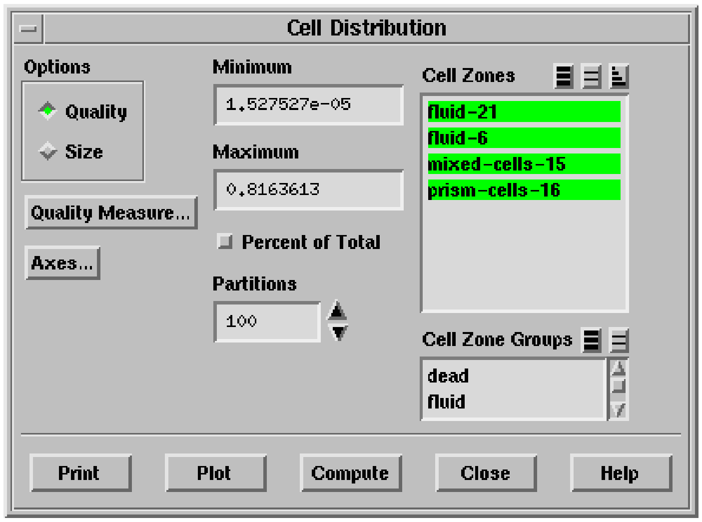
(c) Close the Manage Face Zones panel.

Step 6: Check and Save the Volume Mesh

1. Check the skewness of the entire volume mesh.

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

   Display ⟶ Plot ⟶ Cell Distribution...



   i. Select all the zones in the Cell Zones selection list.

   ii. Click Compute.

   iii. Click Plot.

   iv. Close the Cell Distribution panel.

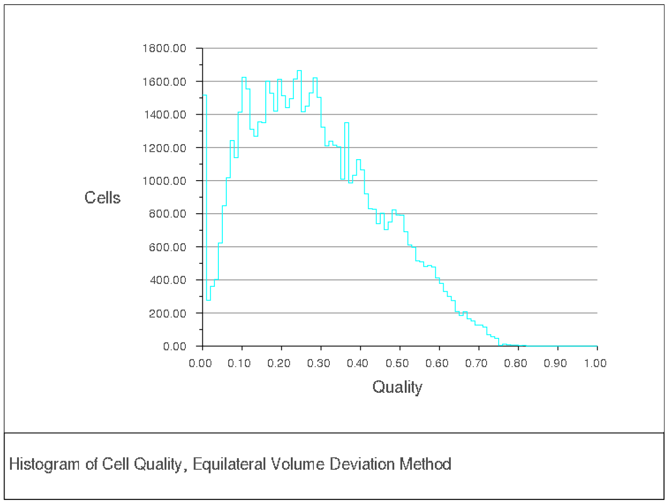
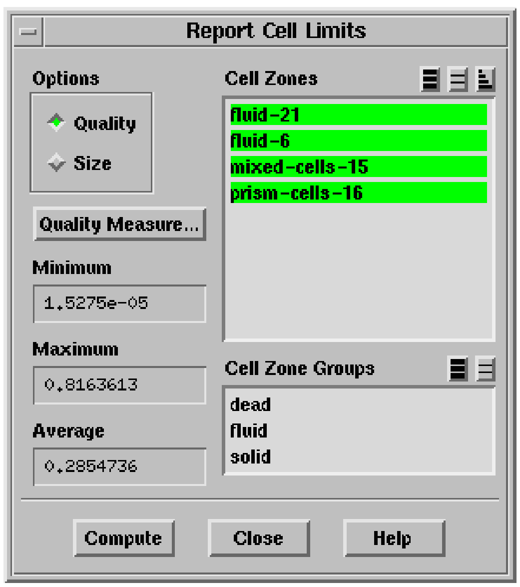


Figure 3.10: Cell Skewness Distribution

(b) Report the worst cell skewness.

Report ⟶Cell Limits...



i. Select all the zones in the Cell Zones selection list.

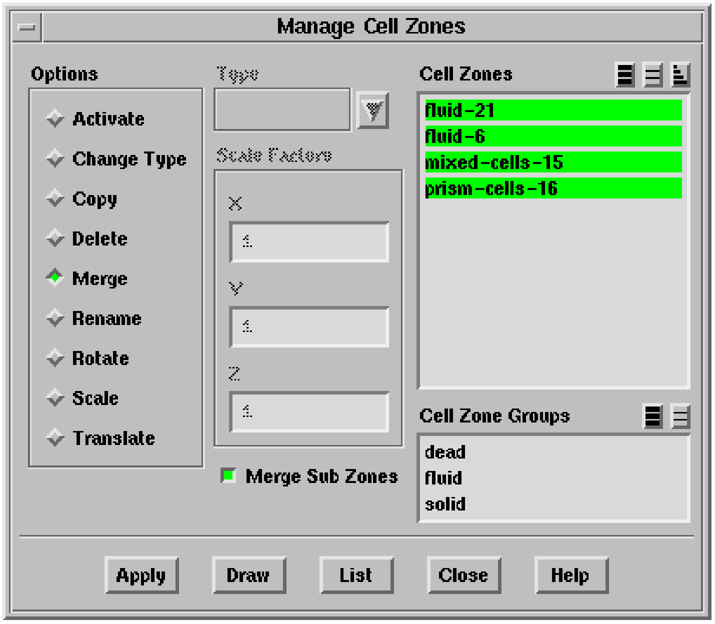
ii. Click Compute.

iii. Close the Report Cell Limits panel.

2. Merge the four cell zones.

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

   Mesh ⟶Manage...



   (a) Select all the zones in the Cell Zones selection list.

   (b) Select Merge in the Options list.

   (c) Enable Merge Sub Zones.

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

   (d) Click Apply.

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

   (e) Close the Manage Cell Zones panel.

3. Check the volume mesh.

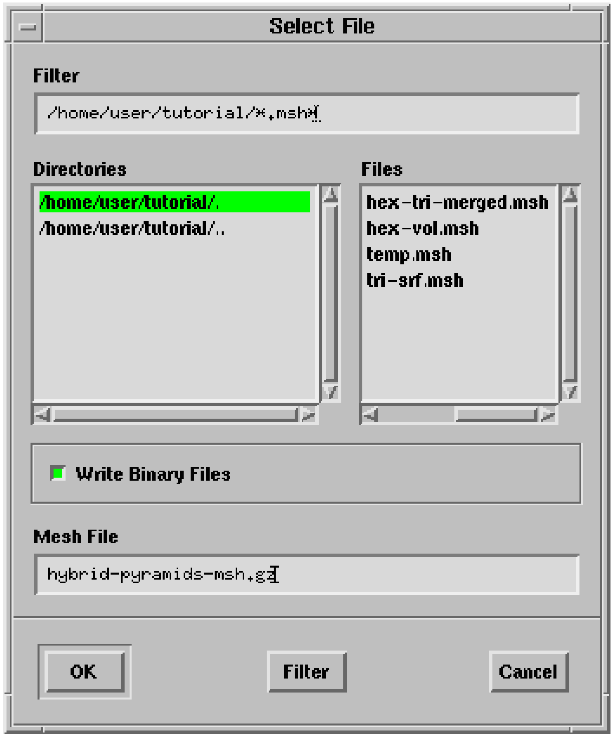
   Before saving the mesh file, check it to ensure that it has no negative cell volumes or left-handed faces.

   Mesh ⟶Check

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

4. Save the mesh.

File ⟶ Write ⟶ Mesh...



(a) Enter `hybrid-pyramids.msh.gz` for Mesh File.

(b) Click OK to save the volume mesh.

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

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

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

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

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

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

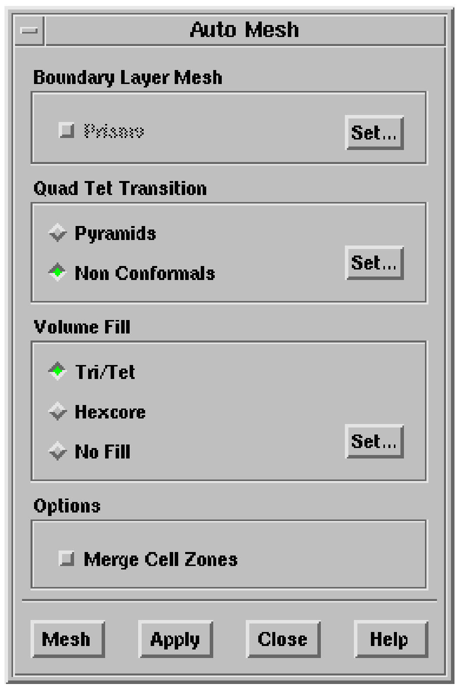
   File ⟶ Read ⟶ Mesh...

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

   Boundary ⟶ Manage...

3. Set the meshing parameters.

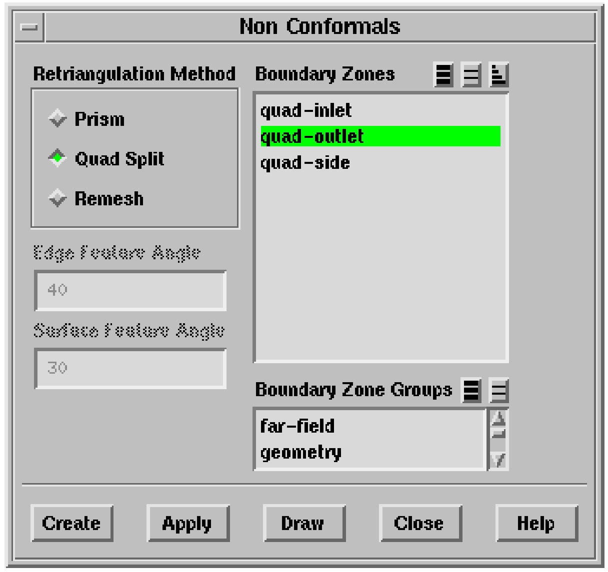
   Mesh ⟶ Auto Mesh...



   (a) Select Non Conformals in the Quad Tet Transition list and click the Set... button to open the Non Conformals panel.

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

      ii. Select Quad Split in the Retriangulation Method list.



iii. Click Apply and close the Non Conformals panel.

(b) Retain the selection of Tri/Tet in the Volume Fill list and click the Set... button to open the Tri/Tet panel.

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

ii. Retain the default settings in the Refinement tab and click Apply.

iii. Close the Tri/Tet panel.

(c) Click Apply in the Auto Mesh panel.

(d) Preserve the existing hexahedral mesh.

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

(e) Click Mesh in the Auto Mesh panel.

The maximum and average skewness values reported at the end of the meshing are approximately 0.886 and 0.344, respectively.

(f) Close the Auto Mesh panel.

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

Display ⟶Grid...

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

(b) Disable Free in the Options group box in the Faces tab.

(c) Click the Cells tab and select all the zones in the Cell Zones selection list.

(d) Enable All in the Options group box.

(e) Click the Attributes tab and enable Filled and Lights in the Options group box.

(f) Enter `0.4` for Shrink Factor.

(g) Click the Colors... button to open the Grid Colors panel.

i. Select Color by ID in the Options list.

ii. Close the Grid Colors panel.

(h) Click Display and zoom in close to the boundary between the hexahedral and tetrahedral mesh.

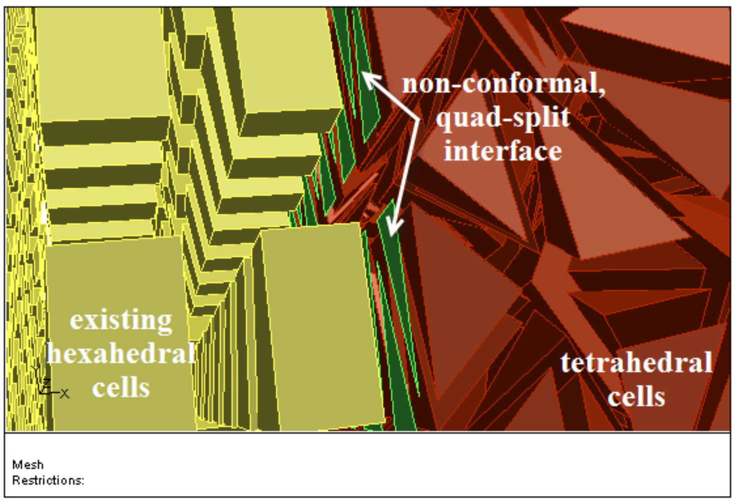


Figure 3.11: Non Conformal Transition Between Hexahedral and Tetrahedral Mesh

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

5. Check the skewness of the entire volume mesh.

(a) Plot the cell skewness distribution (Figure 3.12).

Display ⟶ Plot ⟶ Cell Distribution...

(b) Report the worst cell skewness.

Report ⟶ Cell Limits...

The worst cell skewness is approximately 0.886.

6. Merge the cell zones.

Mesh ⟶ Manage...

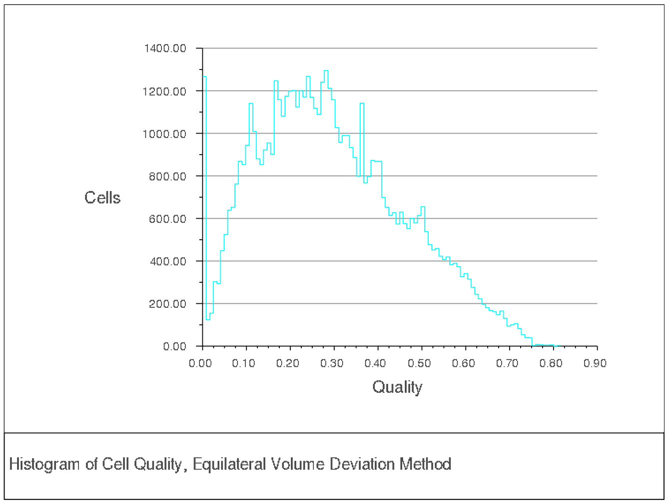


Figure 3.12: Cell Skewness Distribution

7. Check the volume mesh.

   Mesh ⟶Check

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

   File ⟶ Write ⟶Mesh...

9. Exit TGrid.

   File ⟶Exit

Summary

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

Tutorial 4. Viscous Hybrid Mesh Generation

Introduction

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.

TGrid allows you to create a viscous hybrid mesh by growing prisms from the faces on the surface mesh. It creates high quality prism elements near the boundary and tetrahedral elements in the rest of the domain. TGrid also supports automatic proximity detection and height adjustment while growing prisms in a narrow gap.

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:

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

Prerequisites

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

Preparation

1. Download `prisms.zip` from the FLUENT User Services Center to your working directory. This file can be found from the Documentation link on the TGrid product page.

   OR

   Copy `prisms.zip` from the TGrid documentation CD to your working directory.

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

     cdrom`/tgrid5.0/help/tutfiles`

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

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

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

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

2. Unzip `prisms.zip`.

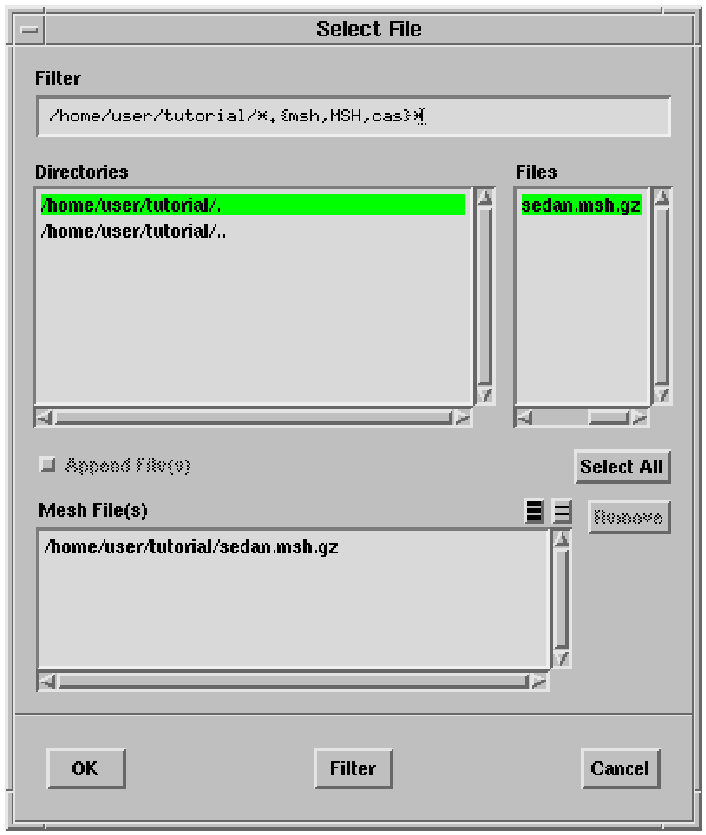
   The file, `sedan.msh.gz` *can be found in the* `prisms` *folder created on unzipping the file.*

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

Step 1: Read and Display the Boundary Mesh

1. Read the mesh file.

   File ⟶ Read ⟶ Boundary Mesh...

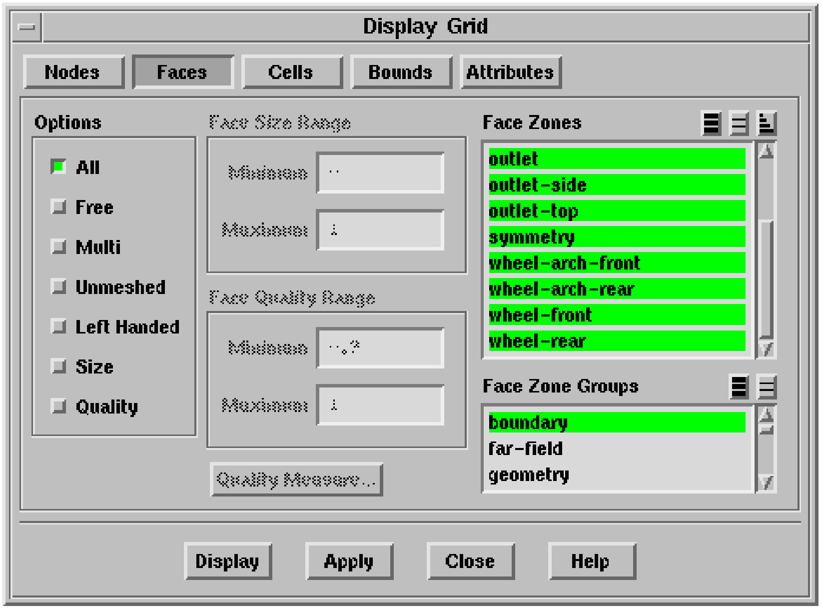


   (a) Select `sedan.msh.gz` in the Files list.

   (b) Click OK.

2. Display the boundary mesh (Figure 4.1).

   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.

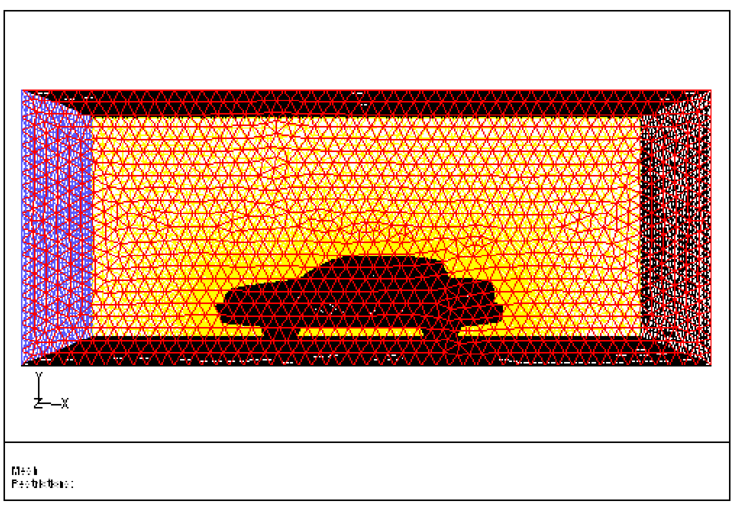


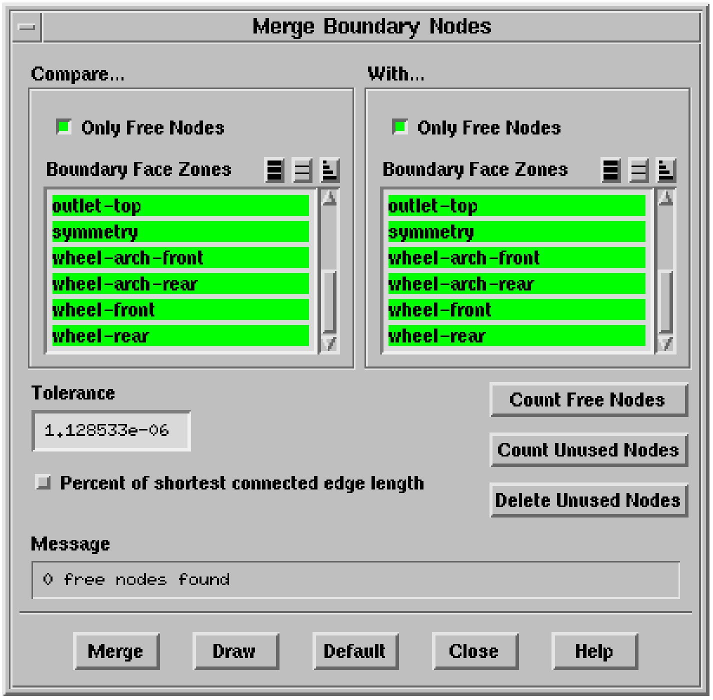
Figure 4.1: Boundary Mesh for the Sedan

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 panel.

Step 2: Check for Free and Unused Nodes

Boundary ⟶Merge Nodes...



1. Click Count Free Nodes.

   TGrid *reports the number of free nodes in the* Message *box. Click* Merge *to remove free nodes, if any.*

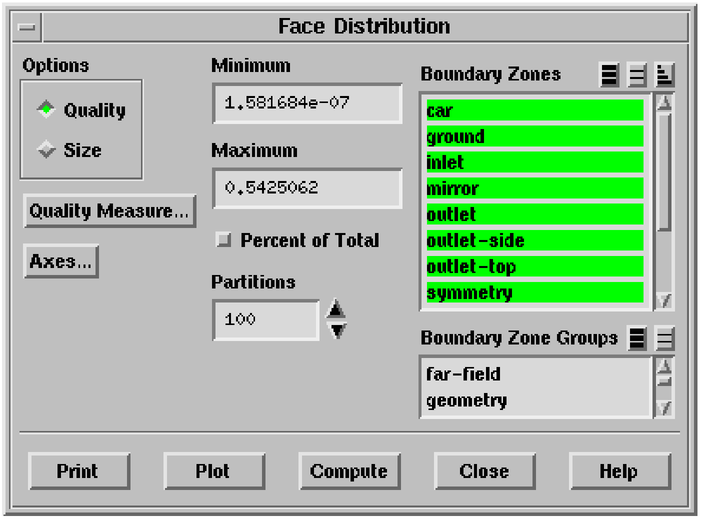
2. Click Count Unused Nodes.

   TGrid *reports the number of unused nodes 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,* TGrid *will be unable to project to and retriangulate the adjacent zone.*

3. Close the Merge Boundary Nodes panel.

Step 3: 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 4.2).
4. Close the Face Distribution panel.

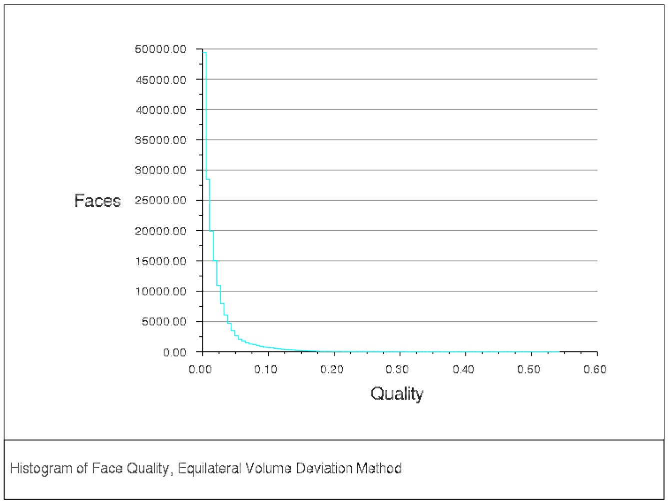
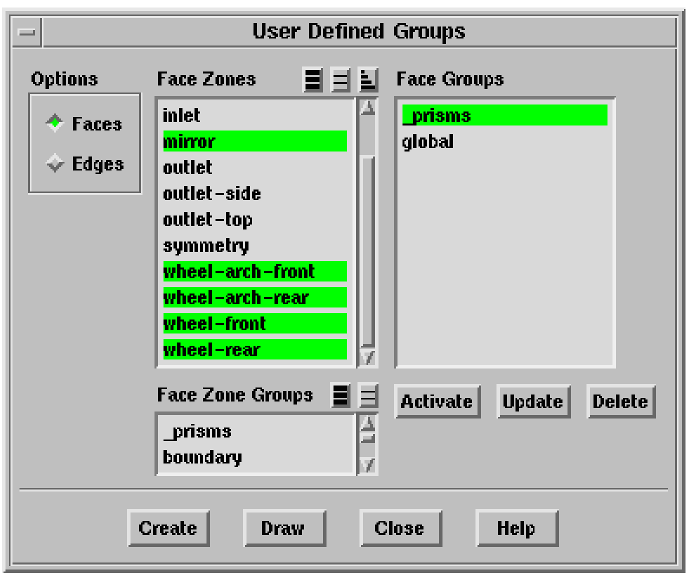


Figure 4.2: Surface Mesh Quality

Step 4: Set Parameters for Prism Layer Shrinkage and Manual Tetrahedral Meshing

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

   Boundary ⟶ Zone ⟶ Group...



   (a) Select car, ground, mirror, wheel-arch-front, wheel-arch-rear, wheel-front, and wheel-rear in the Face Zones selection list.

(b) Click Draw and verify that the zones selected are appropriate.

(c) Click Create.

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

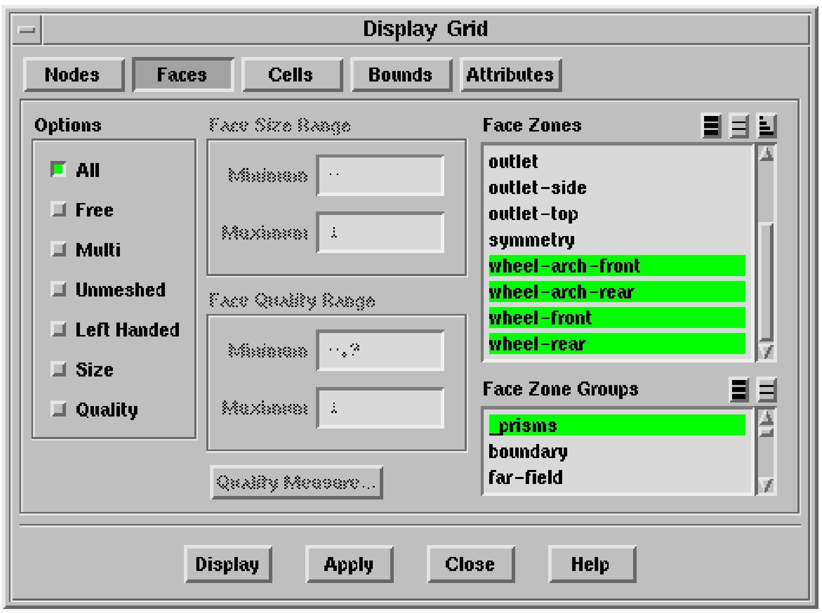
(d) Enter `_prisms` for Name and click OK.

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

(e) Close the User Defined Groups panel.

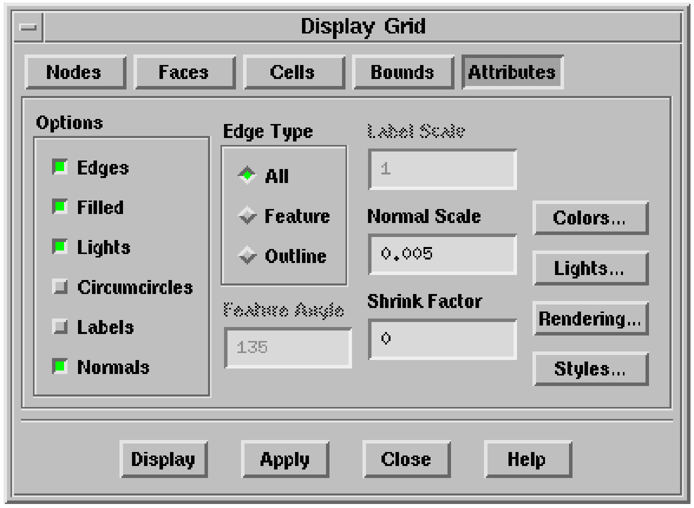
2. Verify that the normals are correctly oriented.

Display ⟶Grid...



(a) Deselect the previous selection of boundary and select _prisms in the Face Zone Groups selection list in the Faces tab.

(b) Click the Attributes tab and enable Normals.



(c) Enter 0.005 for Normal Scale.

(d) Enable Filled and Lights in the Options group box in the Attributes tab of the Display Grid panel.

(e) Click the Colors... button to open the Grid Colors panel.

(f) Select Color by ID in the Options list and close the Grid Colors panel.

(g) Click Display (Figure 4.3).

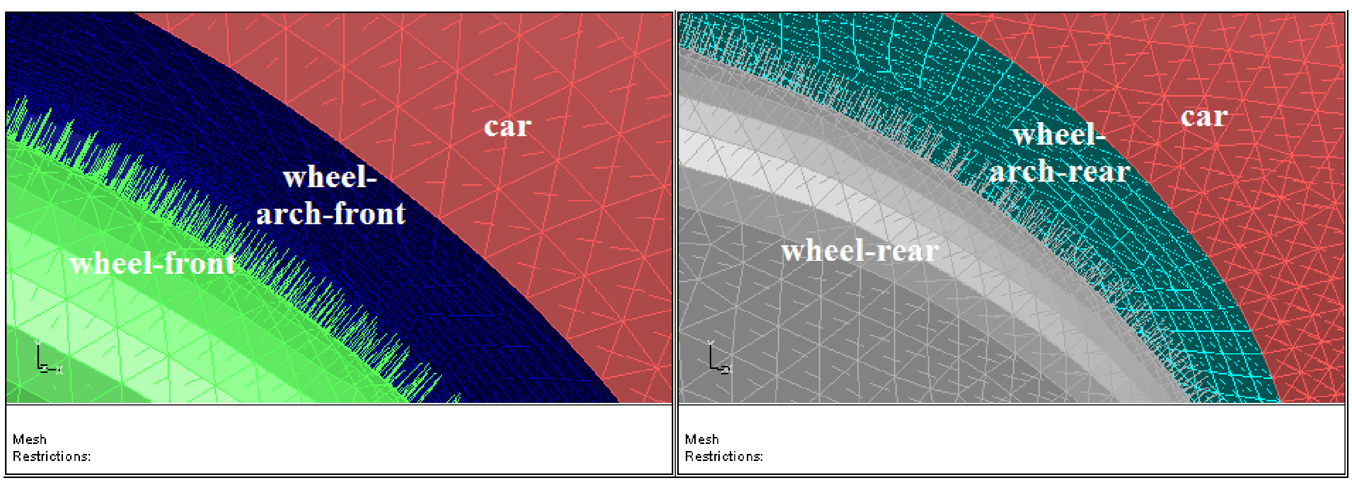


Figure 4.3: Normals on the Wheel and Wheel Arch Zones

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. Figure 4.3 shows the normals on the wheels and wheel arches. You will need to flip the normals on the* wheel-arch-front *and* wheel-arch-rear *zones for the prisms to be grown in the correct direction.*

In TGrid*, 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.*

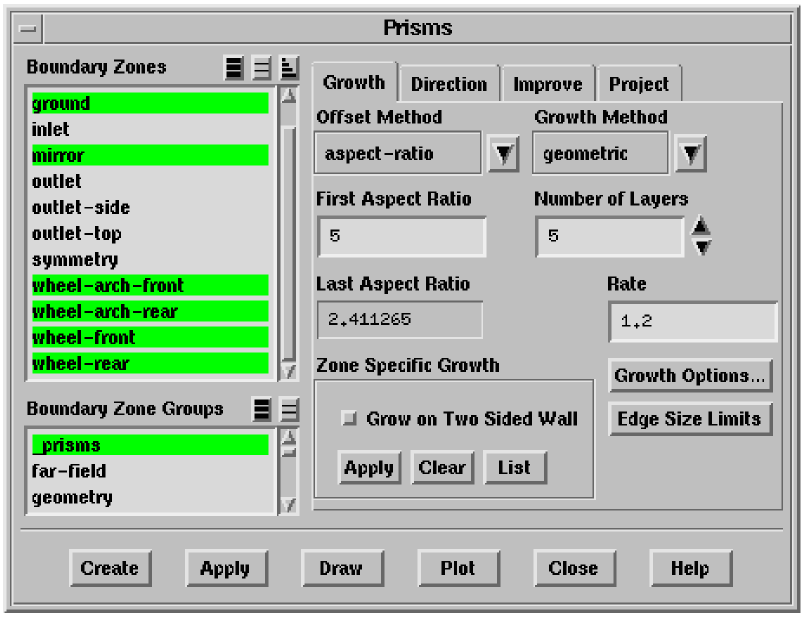
(h) Close the Display Grid panel.

3. Set the prism meshing parameters.

Mesh ⟶Prisms...

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

(b) Select aspect-ratio in the Offset Method drop-down list and enter 5 for First Aspect Ratio.



(c) Select geometric in the Growth Method drop-down list and enter 1.2 for Rate.

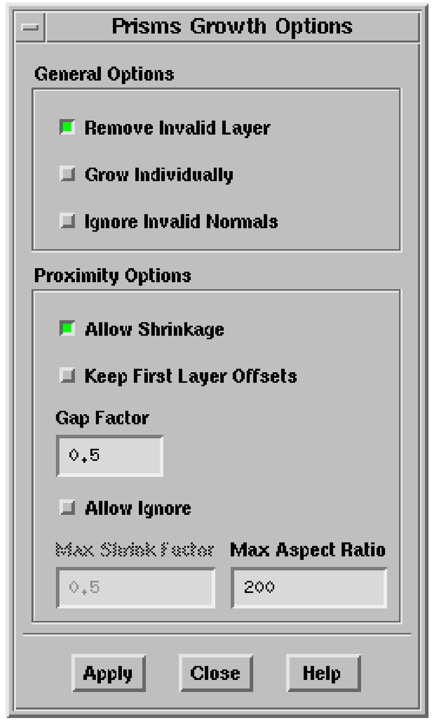
(d) Set the Number of Layers to 5.

(e) Click Apply (next to Create).

(f) Click the Growth Options... button to open the Prisms Growth Options panel.

i. Make sure Allow Shrinkage is enabled.

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

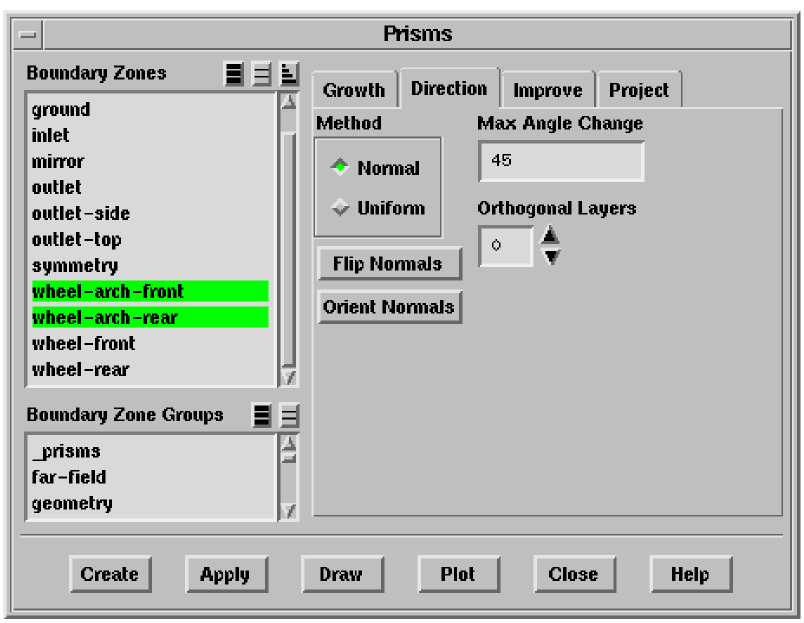


ii. Enter 200 for Max Aspect Ratio and click Apply.

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 panel.

(g) Click the Direction tab in the Prisms panel.



i. Deselect all the zones selected in the Boundary Zones selection list and select only wheel-arch-front and wheel-arch-rear.

ii. Click Flip Normals to orient the normals correctly (Figure 4.4).

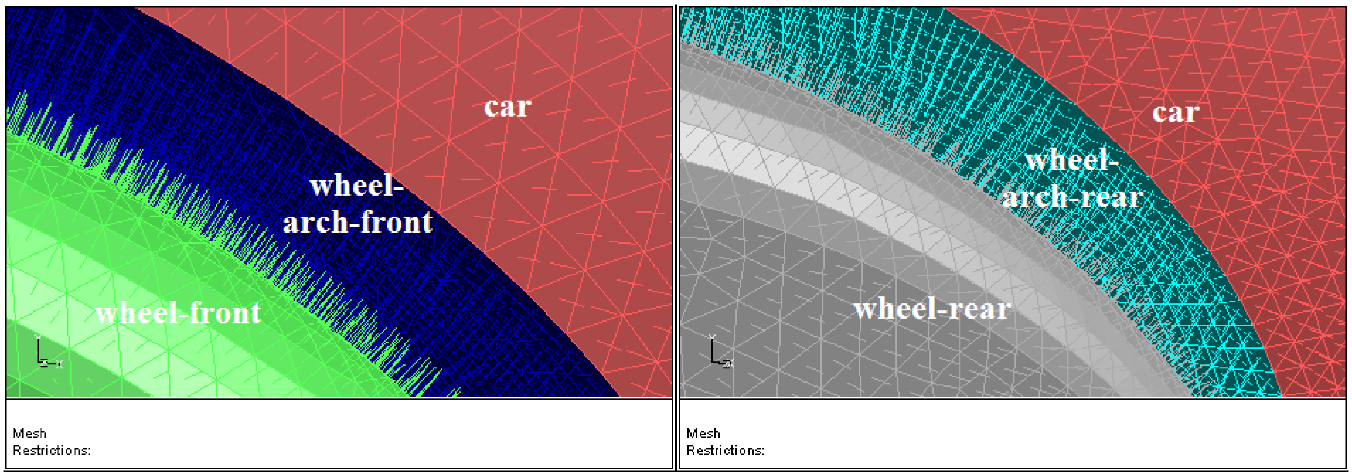


Figure 4.4: Normals on Wheels and Wheel Arches After Flipping

(h) Deselect the zones in the Boundary Zones selection list and select _prisms in the Boundary Zone Groups selection list.

(i) Click Create.

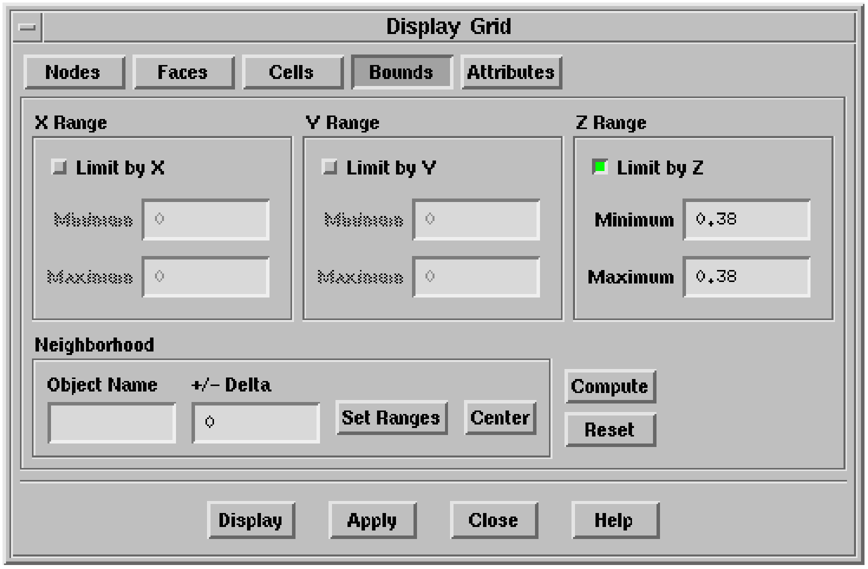
(j) Close the Prisms panel.

4. Examine the prisms generated by displaying a slide of cells at z = 0.38.

   Display ⟶Grid...

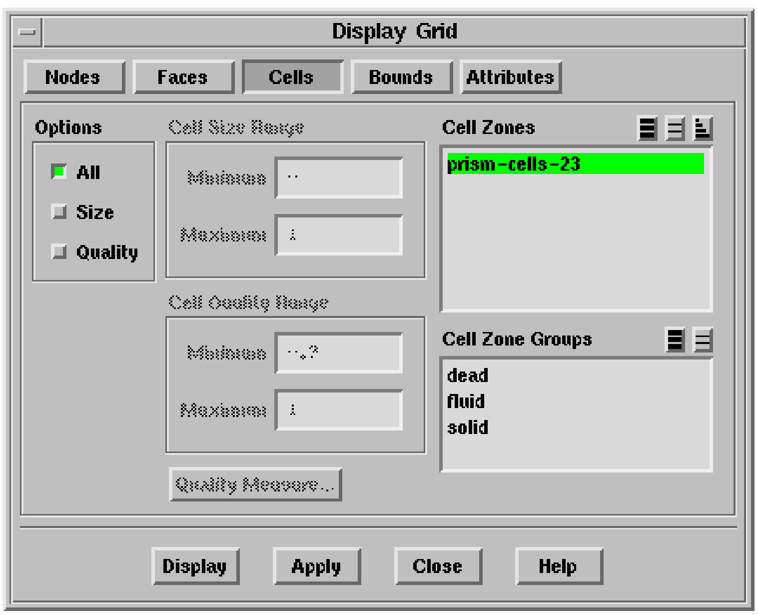
   (a) Retain the selection of the previously selected zones in the Face Zones selection list.

   (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 4.5).

(g) Close the Display Grid panel.

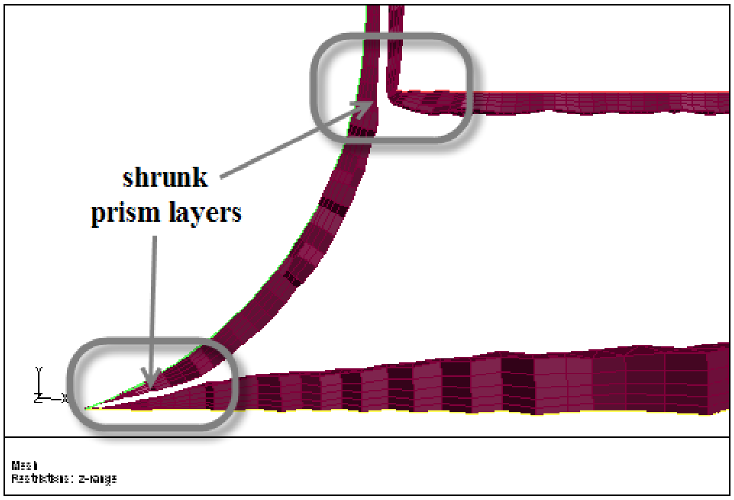
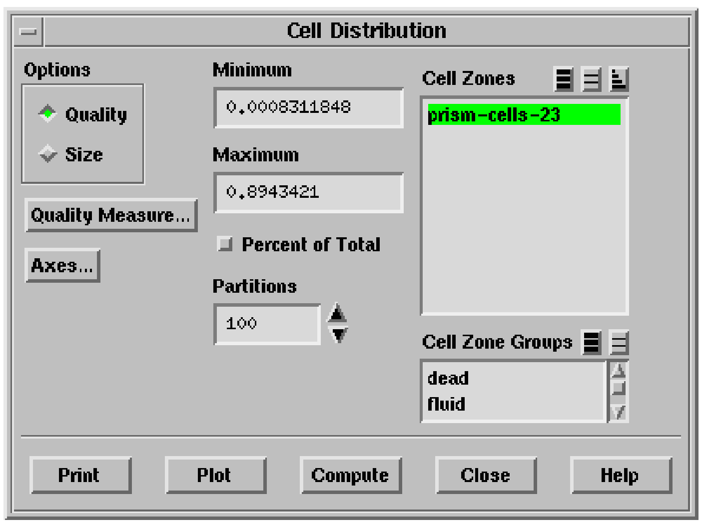


Figure 4.5: Prisms Shrunk in Areas of Proximity and Sharp Corners

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.

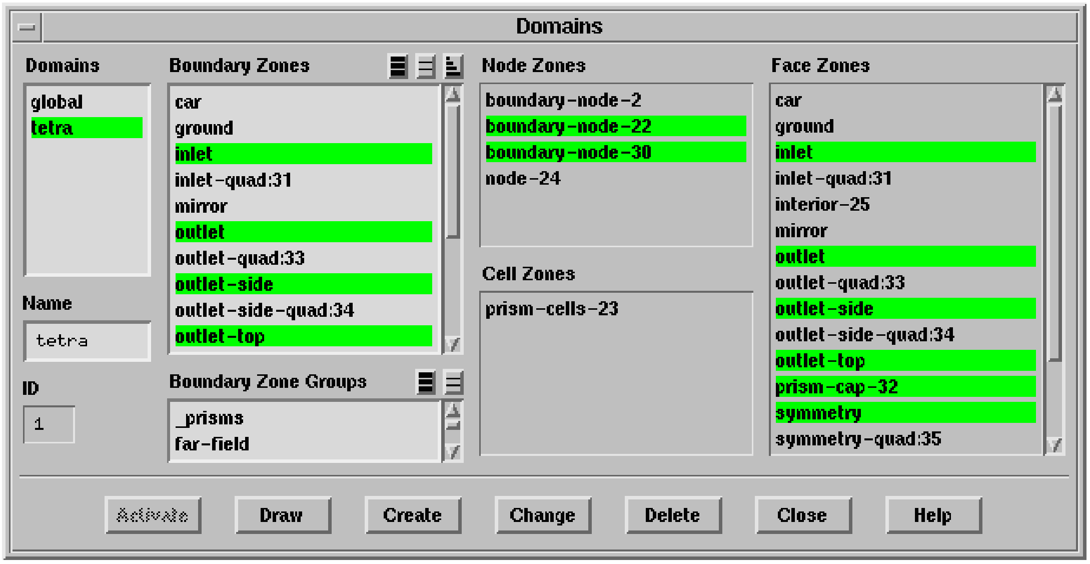
      A maximum skewness value below 0.9 is acceptable.

      Extra: *You may use the command* `/mesh/prism/improve/improve-prism-cells` *to improve the prism cell quality, if required.*

   (b) Close the Cell Distribution panel.

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.

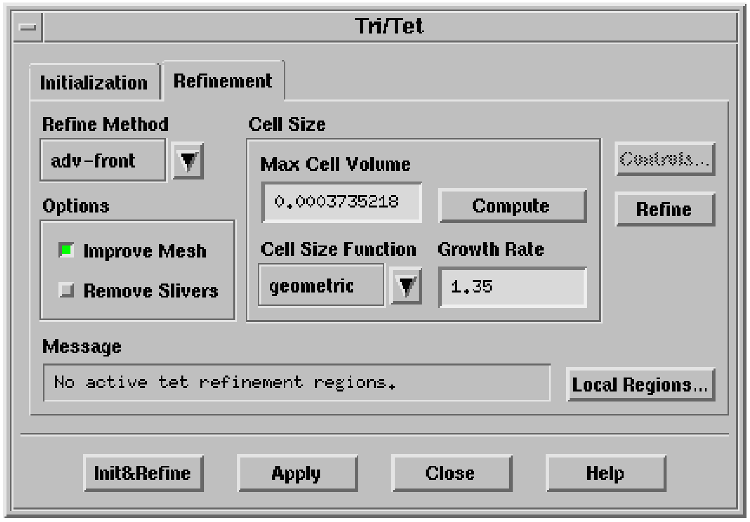
      v. Close the Display Grid panel.

   (d) Close the Domains panel.

7. Set the parameters for tetrahedral meshing.

   Mesh ⟶Tri/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.

(d) Click Apply and Init&Refine.

(e) Close the Tri/Tet panel.

8. Check the quality of the mesh.

   Display ⟶ Plot ⟶ Cell Distribution...

   The maximum skewness reported is around 0.95.

9. Activate the global domain.

   Mesh ⟶ Domains...

   (a) Select global in the Domains list and click Activate.

   (b) Close the Domains panel.

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 4.6).

    (d) Examine the areas of proximity (Figure 4.7).

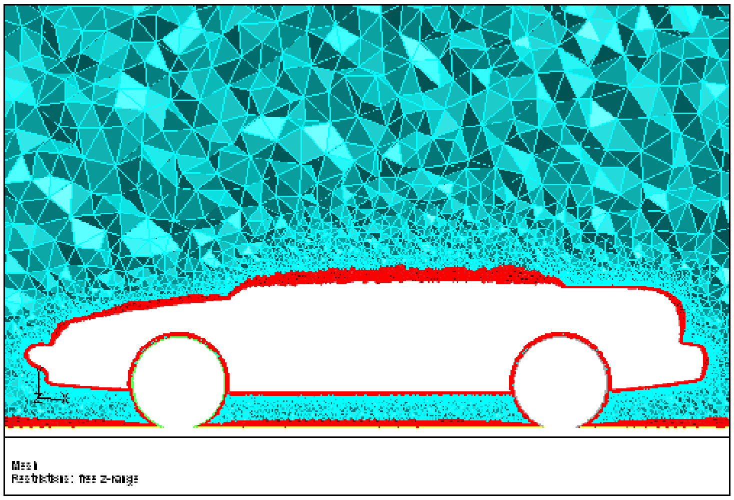


Figure 4.6: Slide of Cells at z = 0.38

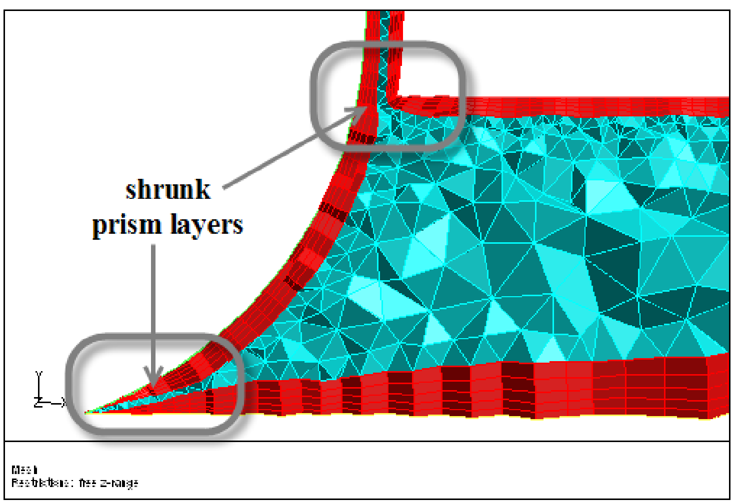
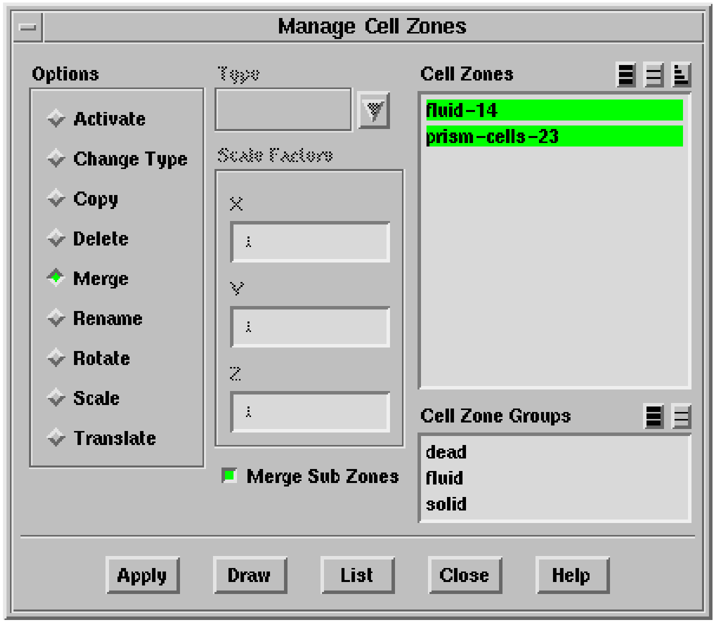


Figure 4.7: Prisms Shrunk in Areas of Proximity

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.

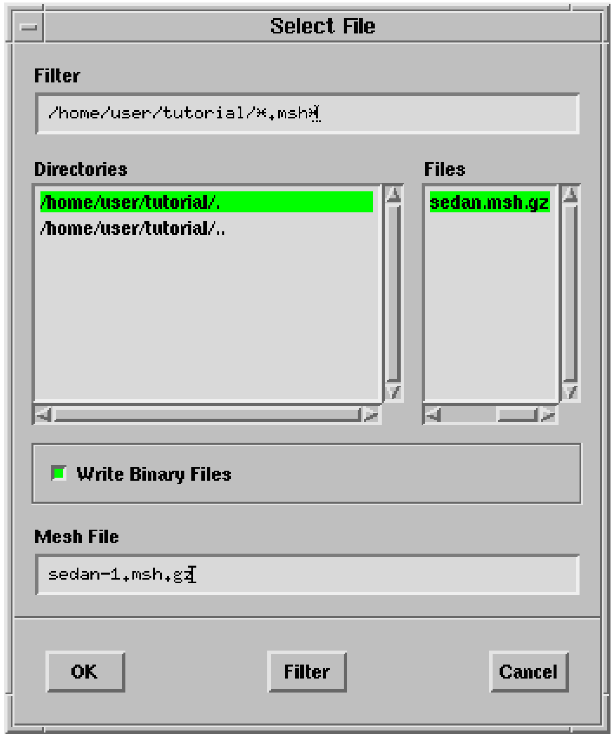
    (d) Close the Manage Cell Zones panel.

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.

Step 5: Set Parameters for Ignoring Prism Layers and Automatic Meshing

The previous sections described the set up and manual meshing procedure for a viscous mesh with prisms and tetrahedra. This demonstrated how TGrid *works with domains. In this step, you will use 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...

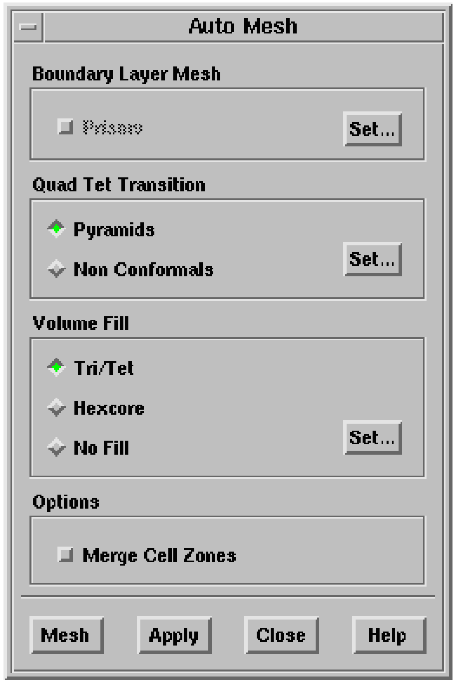
2. Display the grid and verify that the normals are correctly oriented.

   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 flip 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.*

(a) Click the Set... button in the Boundary Layer Mesh group box to open the Prisms panel.

   i. Click the Direction tab in the Prisms panel.

      A. Select wheel-arch-front and wheel-arch-rear and click Flip Normals.

   ii. Click the Growth tab and set the prism growth parameters.

      A. Select car, ground, mirror, wheel-arch-front, wheel-arch-rear, wheel-front, and wheel-rear in the Boundary Zones selection list.

      B. Select aspect-ratio in the Offset Method drop-down list and enter 5 for First Aspect Ratio.

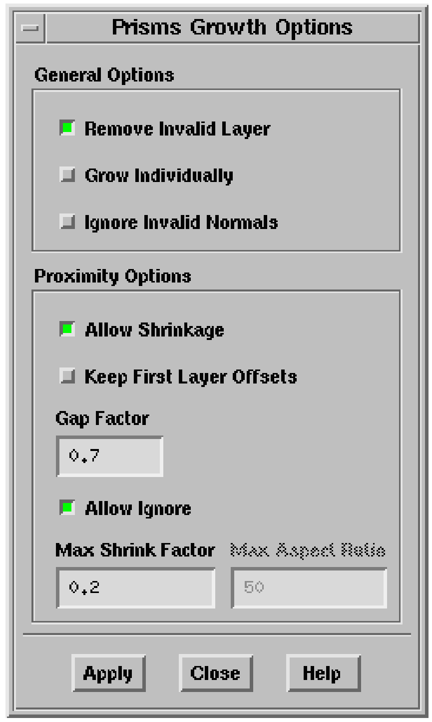
      C. Select geometric in the Growth Method drop-down list and enter 1.2 for Rate.

      D. Set Number of Layers to 5.

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

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

iii. Click the Growth Options... button to open the Prisms Growth Options panel.



A. Retain the Allow Shrinkage option and enable Allow Ignore.

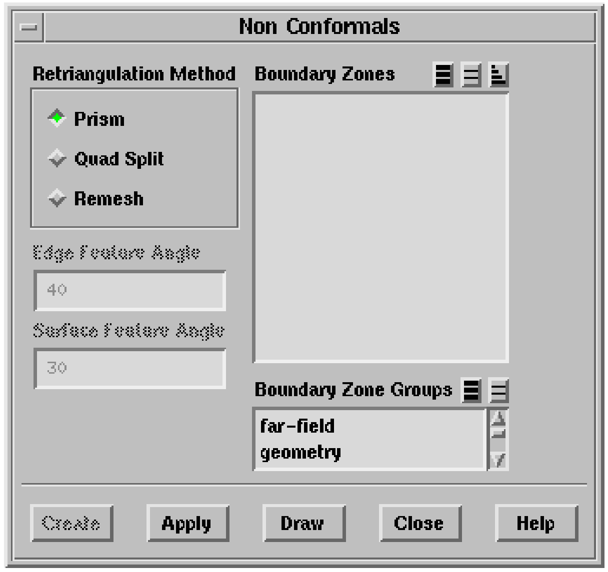
B. Enter 0.7 for Gap Factor and 0.2 for Max Shrink Factor, respectively.

C. Click Apply and close the Prisms Growth Options panel.

iv. Click Apply and close the Prisms panel.

(b) Enable Prisms in the Auto Mesh panel.

(c) Select Non Conformals in the Quad Tet Transition group box and click the Set... button to open the Non Conformals panel.



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 panel.

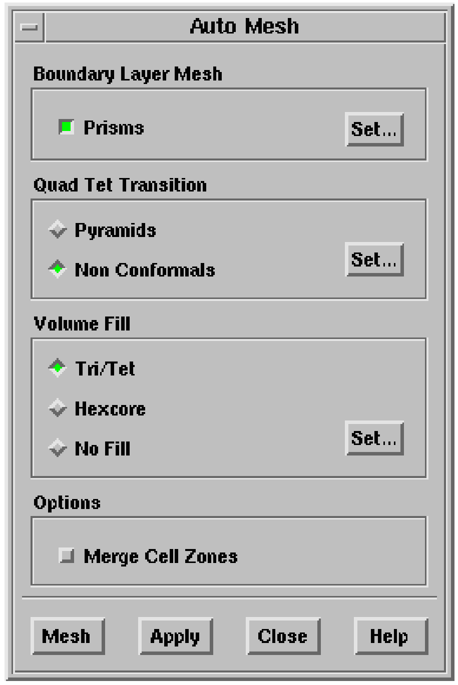
(d) Retain the selection of Tri/Tet in the Volume Fill group box and click the Set... button to open the Tri/Tet panel.

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 Tri/Tet panel.



(e) Click Mesh.

(f) Close the Auto Mesh panel.

4. Examine the mesh by displaying a slide of cells at z = 0.38.

   Display ⟶Grid...

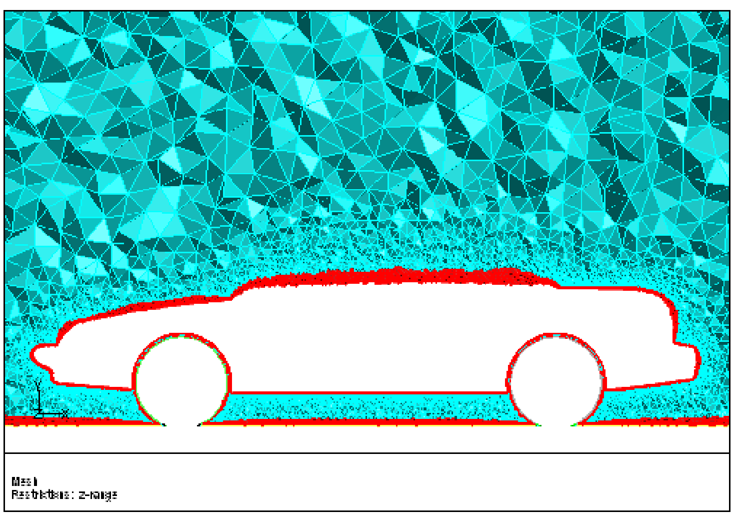


Figure 4.8: Slide of Cells at z = 0.38

Figure 4.9 shows that prism layer growth was ignored in areas of proximity such as sharp corners.

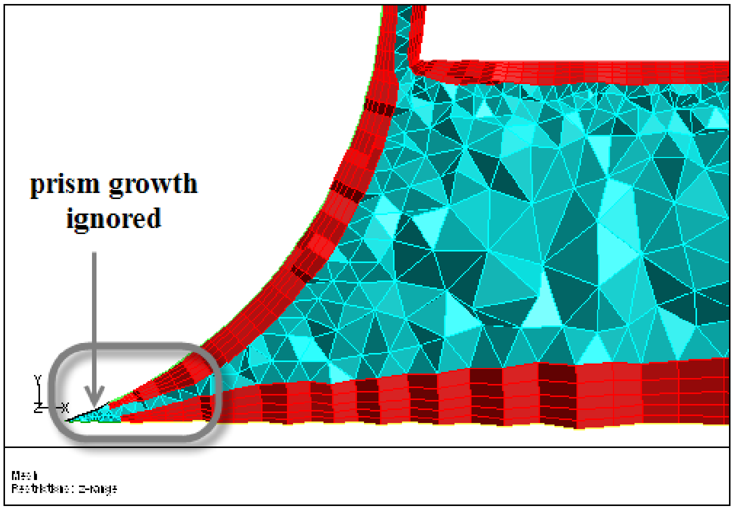


Figure 4.9: Prisms Ignored in Areas of Proximity

5. Merge the cell zones.

   Mesh ⟶Manage...

6. Check the quality of the mesh.

   Display ⟶ Plot ⟶Cell Distribution...

   The maximum skewness reported is around 0.91.

7. Check the mesh.

   Mesh ⟶Check

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

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.

Tutorial 5. Hexcore Mesh Generation

Introduction

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

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

This tutorial demonstrates how to do the following:

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

Prerequisites

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

Preparation

1. Download `hexcore.zip` from the FLUENT User Services Center to your working directory. This file can be found from the Documentation link on the TGrid product page.

   OR

   Copy `hexcore.zip` from the TGrid documentation CD to your working directory.

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

     cdrom`/tgrid5.0/help/tutfiles`

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

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

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

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

2. Unzip `hexcore.zip`.

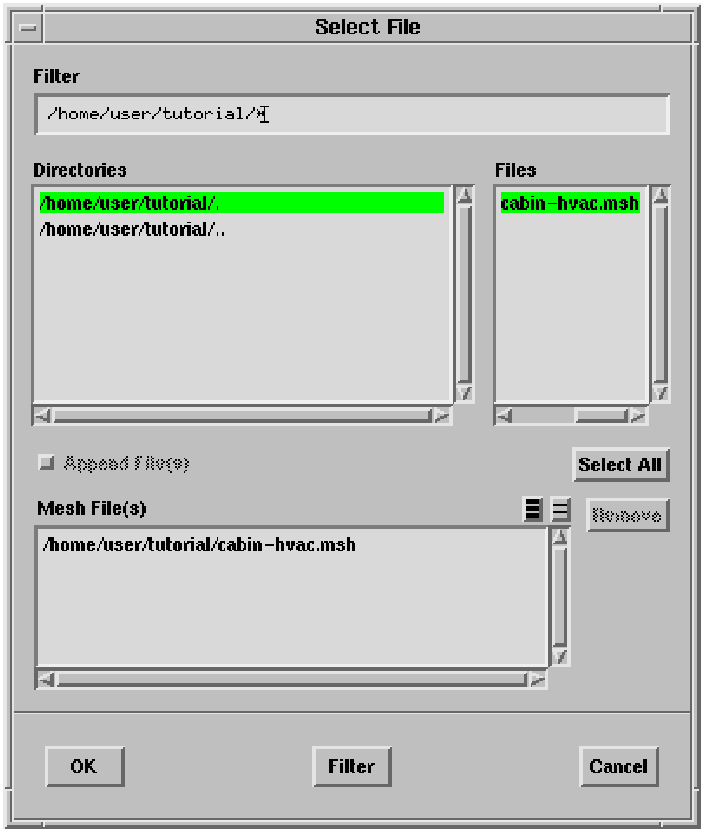
   The file, `cabin-hvac.msh` *can be found in the* `hexcore` *folder created on unzipping the file.*

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

Step 1: Read and Display the Mesh

1. Read the mesh.

   File ⟶ Read ⟶ Mesh...

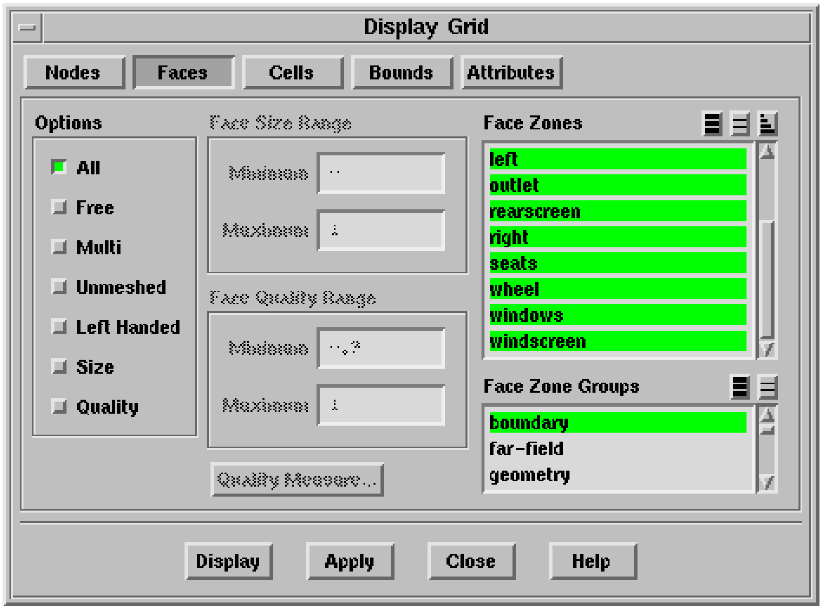


   (a) Select `cabin-hvac.msh` in the Files list.

   (b) Click OK.

2. Display the mesh (Figure 5.1).

   Display ⟶Grid...



   (a) Select boundary in the Face Zone Groups selection list to select all the boundary zones in the Face Zones selection list.

   (b) Click Display.

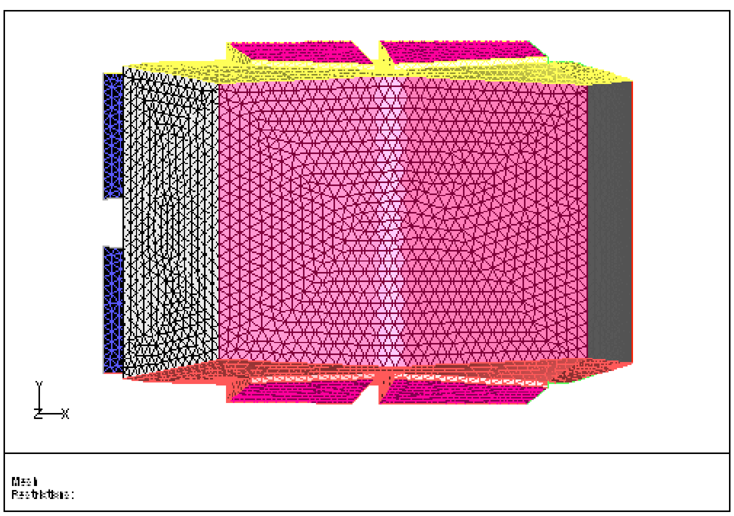


Figure 5.1: Grid Display

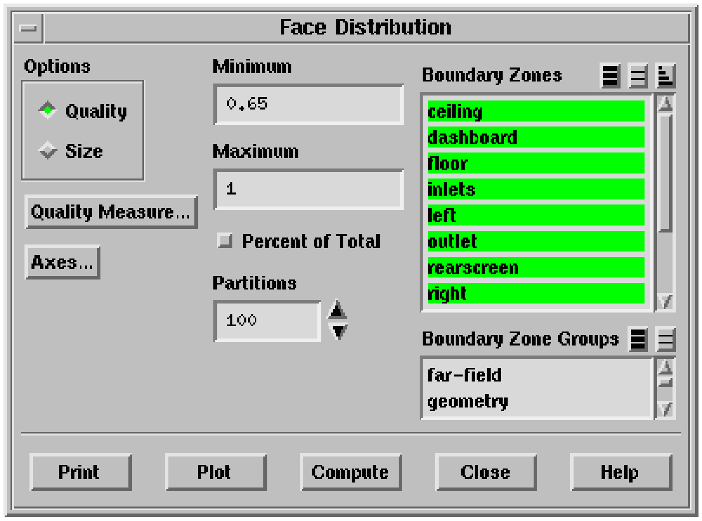
   (c) Close the Display Grid panel.

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

1. Display the surfaces with higher skewness.

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

   Display ⟶ Plot ⟶ Face Distribution...



   i. Select all the zones in the Boundary Zones selection list.

   ii. Enter 0.65 for Minimum.

   iii. Click Plot (Figure 5.2).

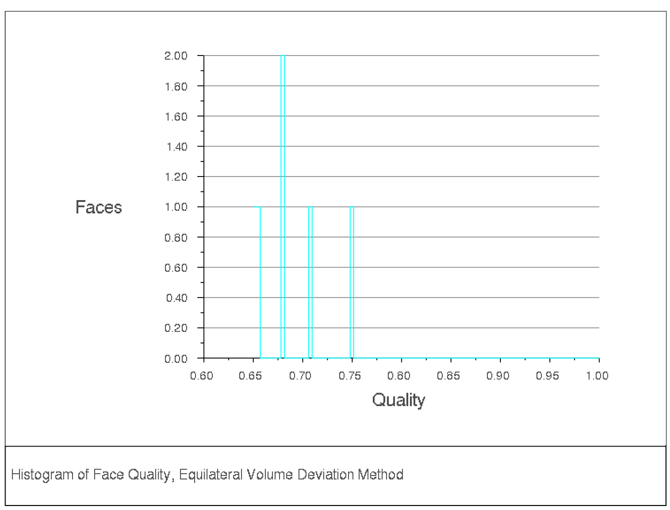


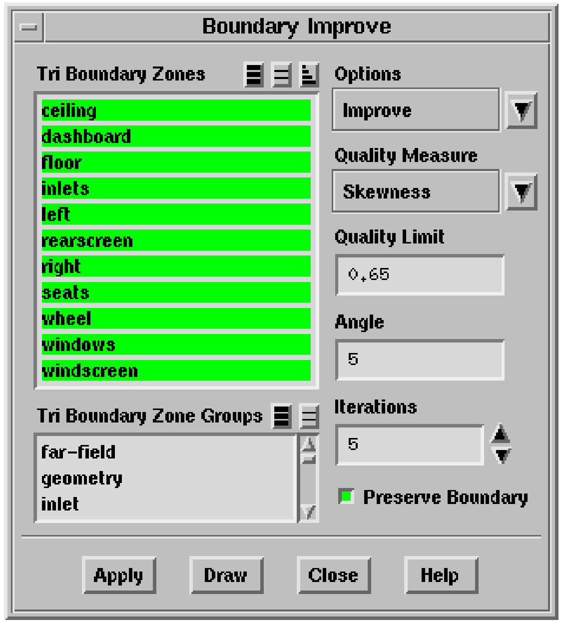
Figure 5.2: Skewness Distribution Above 0.65

   iv. Close the Face Distribution panel.

Step 3: Repair the Boundary Mesh

1. Modify the faces having skewness greater than 0.65.

   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.

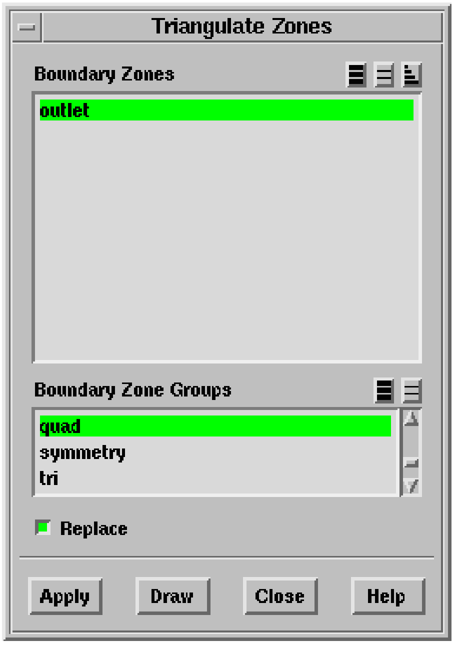
   TGrid *will fix the 6 faces above the specified maximum quality (0.65) and report that the current maximum quality is approximately 0.649.*

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

   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 5.3).

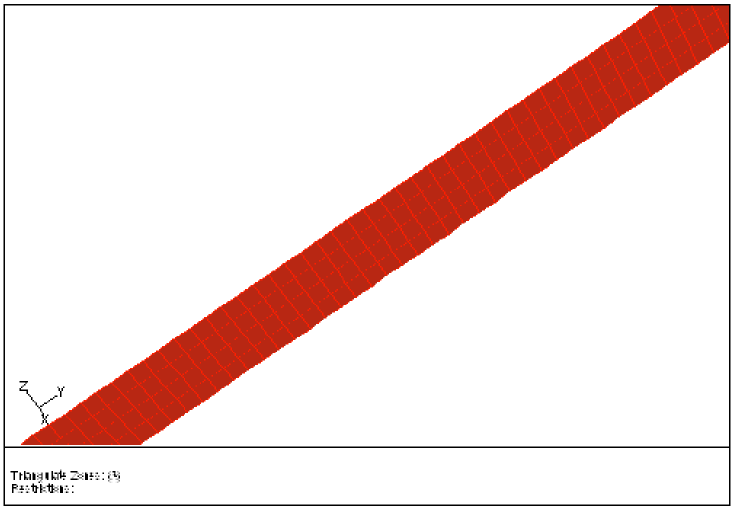


Figure 5.3: Quad Faces in the outlet Zone

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

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

(d) Close the Triangulate Zones panel.

Extra: *You can also create a non-conformal interface to allow quad faces during the hexcore mesh initialization. In this case,* TGrid *will automatically create the triangular mesh on all quad faces available in the domain. All the surfaces* *having quad elements will be copied and remeshed with triangular cells. The free nodes of the triangular mesh will be merged with the original surface mesh.*

3. Check the quality of the surface mesh.

   (a) Select all the zones in the Tri Boundary Zones selection list in the Boundary Improve panel.

   (b) Retain the selection of Skewness in the Quality Measure drop-down list and 0.65 for Quality Limit.

   (c) Click Apply.

   The maximum skewness reported is still less than 0.65, hence the mesh is acceptable.

   (d) Close the Boundary Improve panel.

4. Save the mesh file.

   File ⟶ Write ⟶ Mesh...

Step 4: Generate the Tetrahedral Mesh

1. Generate a tetrahedral mesh with the default settings.

   Mesh ⟶ Tri/Tet...

   (a) Enable Delete Dead Zones in the Tri Tet Zones group box.

   (b) Retain the default settings for the remaining parameters and click Init&Refine.

   The number of cells generated is approximately 204763 while the maximum skewness reported is around 0.856. The exact number may vary slightly on different platforms.

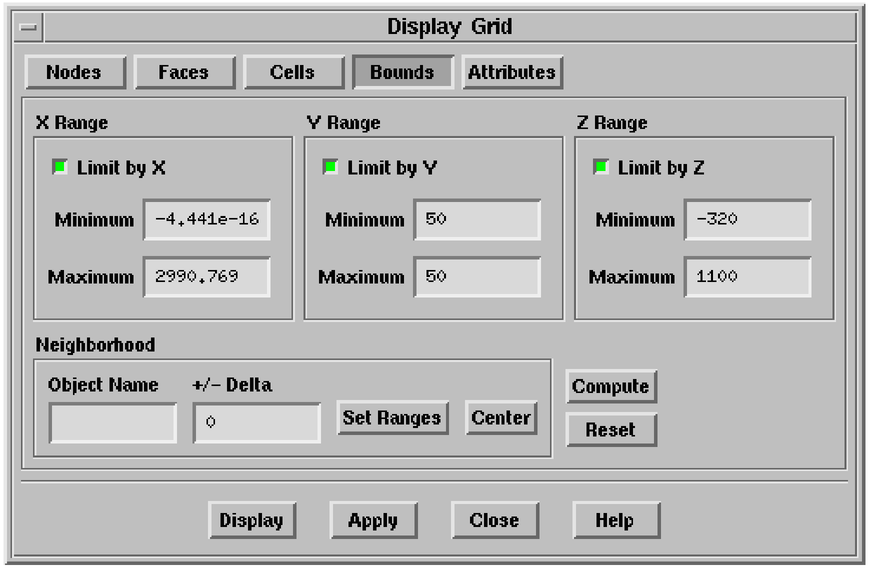
   (c) Close the Tri/Tet panel.

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

   Display ⟶ Grid...

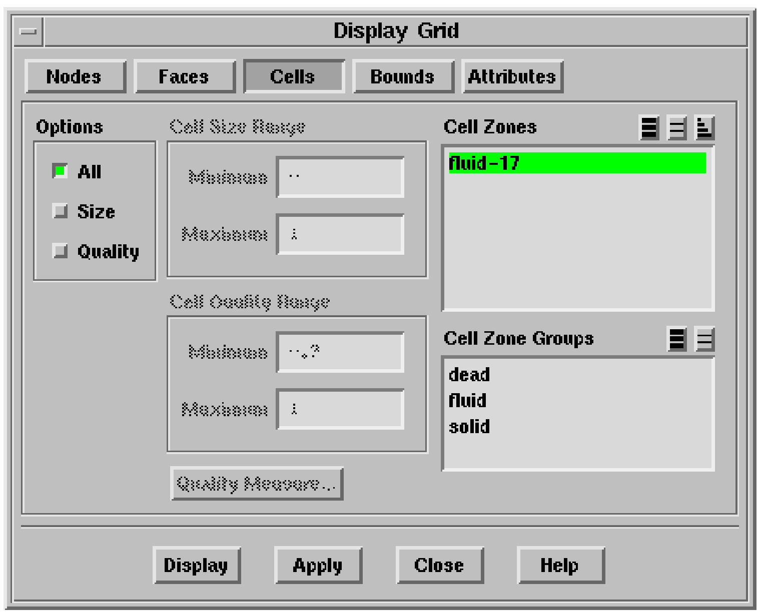
   (a) Retain the selection of all the boundary zones in the Face Zones selection list.

   (b) Click the Bounds tab and click Compute.



(c) Enter 50 for Minimum and Maximum in the Y Range group box.

(d) Click the Cells tab and enable All in the Options group box.



(e) Select the fluid zone (fluid-#) in the Cell Zones selection list and click Display.

(f) Display the top view (Figure 5.4).

Display ⟶Views...

i. Select top in the Views list.

ii. Click Apply and close the Views panel.

(g) Similarly, display the cells for a value of y = -525 (Figure 5.5).

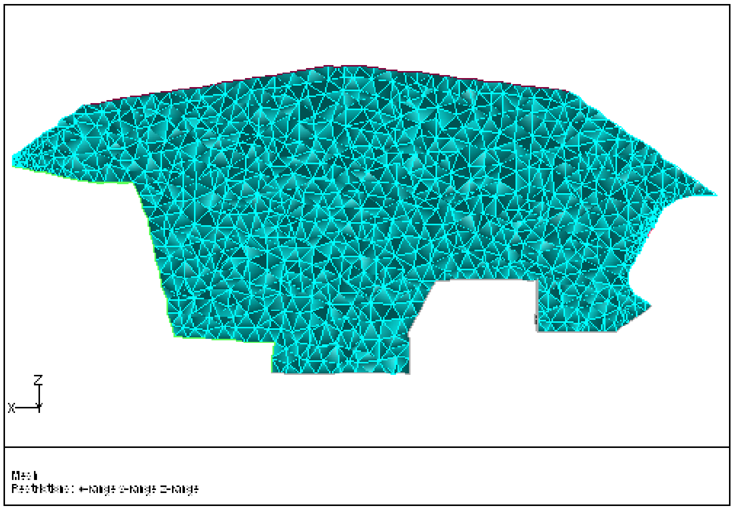


Figure 5.4: Slide of Cells at y = 50 for the Tetrahedral Mesh

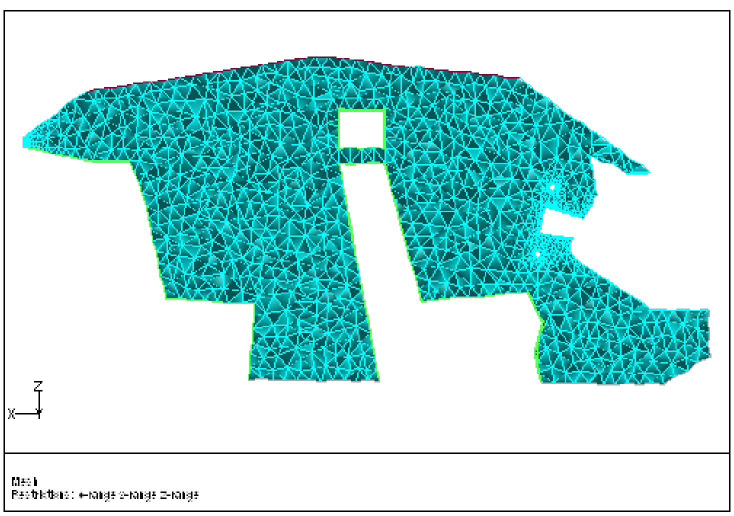


Figure 5.5: Slide of Cells at y = -525 for the Tetrahedral Mesh

3. Clear the mesh.

   Mesh ⟶Clear

Step 5: Generate the Hexcore Mesh

1. Generate the hexcore mesh using the default parameters.

   Mesh ⟶Hexcore...

   (a) Enable Delete Dead Zones in the Zones group box.

   (b) Retain the default settings for the remaining parameters and click Create.

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

   (c) Display the parent face.

      i. Click Reset in the Bounds tab.

      ii. Click the Faces tab and select parent-face-# in the Face Zones selection list.

      iii. Click Display (Figure 5.6).

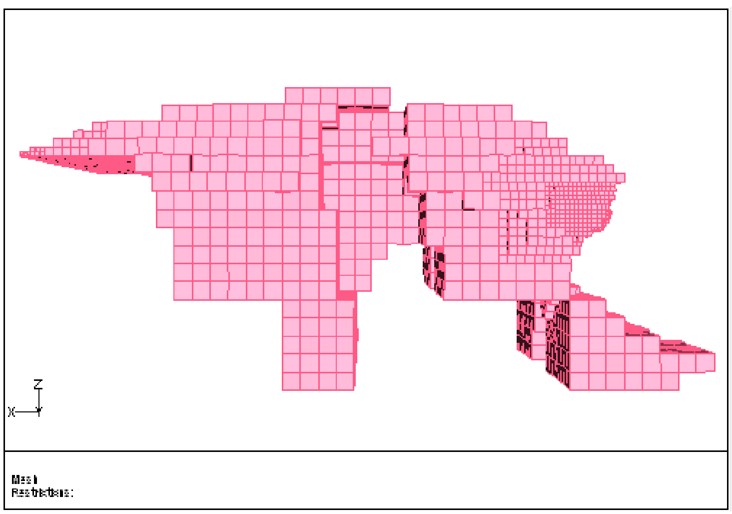


Figure 5.6: Parent Face for the Hexcore Mesh

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

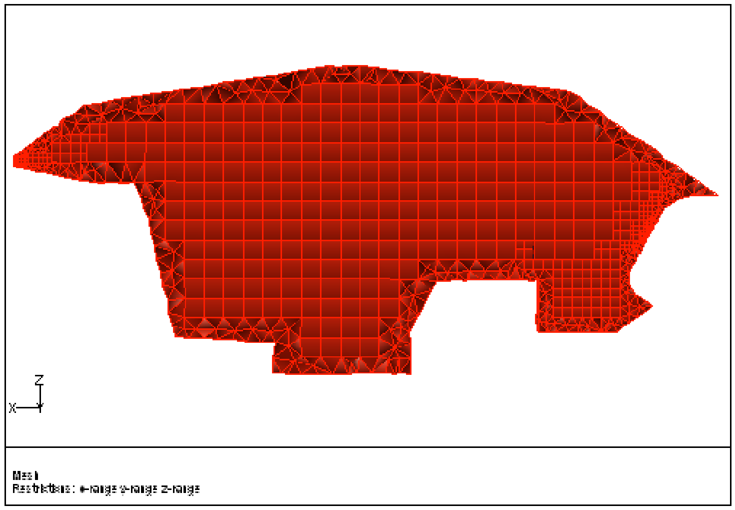


Figure 5.7: Slide of Cells at y = 50 for the Hexcore Mesh With Buffer Layers = 1

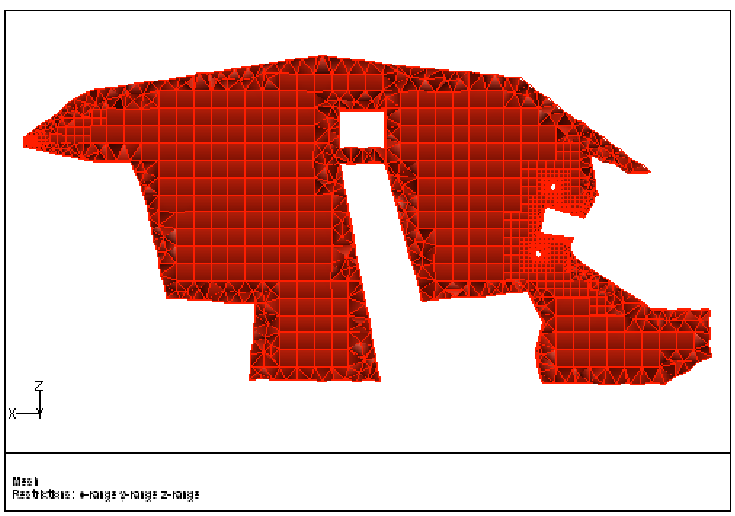


Figure 5.8: Slide of Cells at y = -525 for the Hexcore Mesh With Buffer Layers = 1

2. Check the number of cells generated.

   Report ⟶Mesh Size...

   (a) Click Update.

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

   (b) Close the Report Mesh Size panel.

3. Check the maximum skewness.

   Report ⟶Cell Limits...

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

   The maximum skewness reported is around 0.884.

   (b) Close the Report Cell Limits panel.

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

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

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

1. Clear the mesh.

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

   Mesh ⟶Hexcore...

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

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

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

   (c) Click OK in the Warning dialog box.

   (d) Click Create.

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

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

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

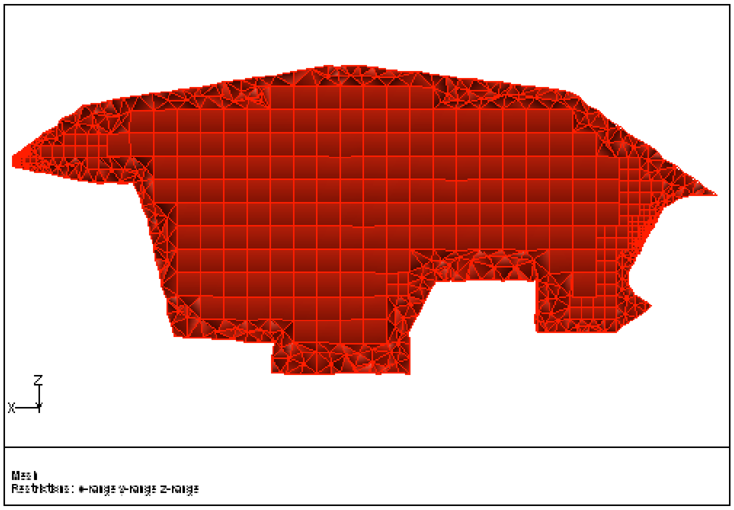


Figure 5.9: Slide of Cells at y = 50 for the Hexcore Mesh With Buffer Layers = 0

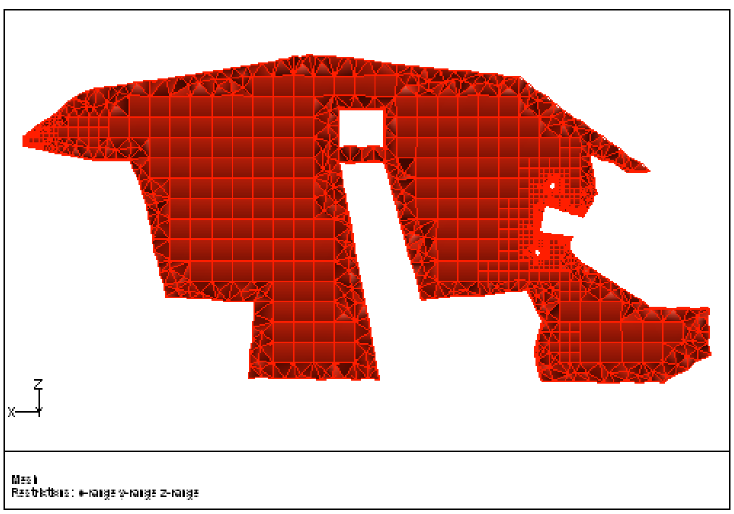


Figure 5.10: Slide of Cells at y = -525 for the Hexcore Mesh With Buffer Layers = 0

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

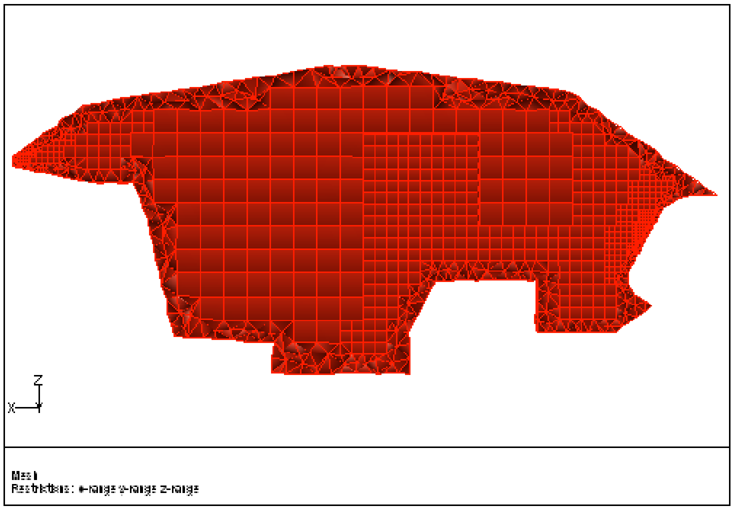


Figure 5.11: Slide of Cells at y = 50 for the Hexcore Mesh With Buffer Layers = 2

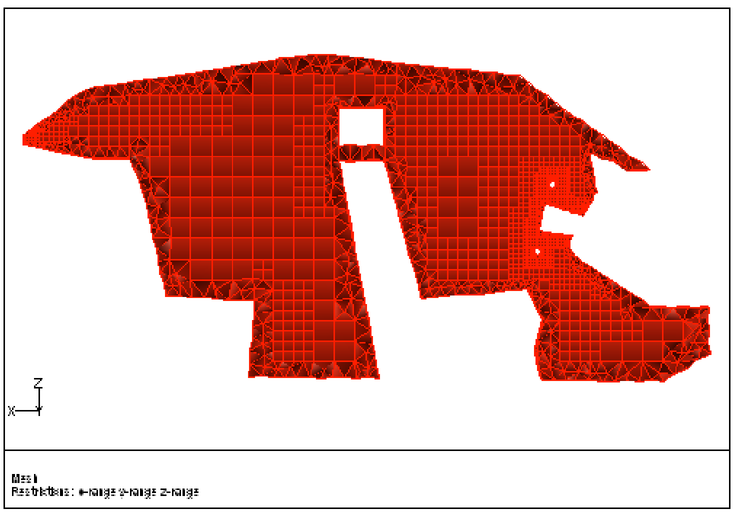


Figure 5.12: Slide of Cells at y = -525 for the Hexcore Mesh With Buffer Layers = 2

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

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

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

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

1. Clear the mesh.

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

   Display ⟶Grid...

   (a) Click Reset in the Bounds section of the Display Grid panel.

   (b) Select ceiling, outlet:#, rearscreen, and windscreen in the Face Zones selection list.

   (c) Click the Attributes tab.

   (d) Enable Normals in the Options group box and enter 10 for Normal Scale.

   Larger normals are easier to see in the grid display.

   (e) Click Display.

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

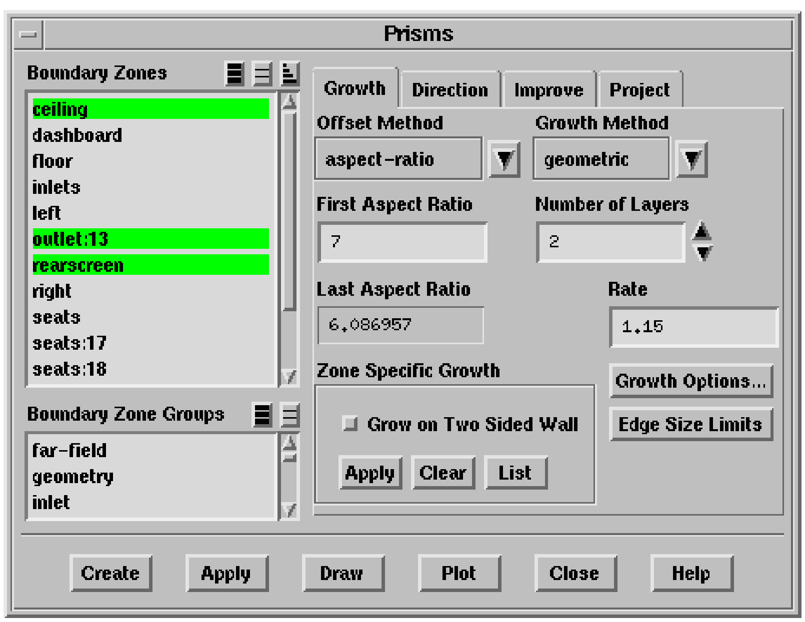
3. Define the meshing parameters.

   Mesh ⟶Auto Mesh...

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

   (a) Define the parameters for growing prism layers.

      i. Click the Set... button in the Boundary Layer Mesh group box to open the Prisms panel.



ii. Select ceiling, outlet:#, rearscreen, and windscreen in the Boundary Zones selection list.

iii. Select aspect-ratio in the Offset Method drop-down list and enter 7 for First Aspect Ratio.

iv. Select geometric in the Growth Method drop-down list and set the Number of Layers to 2.

v. Enter `1.15` for Rate.

vi. Click Apply in the Zone Specific Growth group box.

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

vii. Close the Prisms panel.

viii. Enable Prisms in the Boundary Layer Mesh group box in the Auto Mesh panel.

(b) Display all the surfaces except right using the Display Grid panel.

Make sure the display of normals is disabled in the Attributes *tab.*

(c) Define the parameters for hexcore meshing.

i. Select Hexcore in the Volume Fill list in the Auto Mesh panel and click the Set... button to open the Hexcore panel.

ii. Compute the average length of the faces on the ceiling zone.

A. Zoom in to the ceiling zone in the graphics display.

B. Use the hot-key Ctrl + N to enable the selection of nodes.

C. Select two adjacent nodes using the right mouse button.

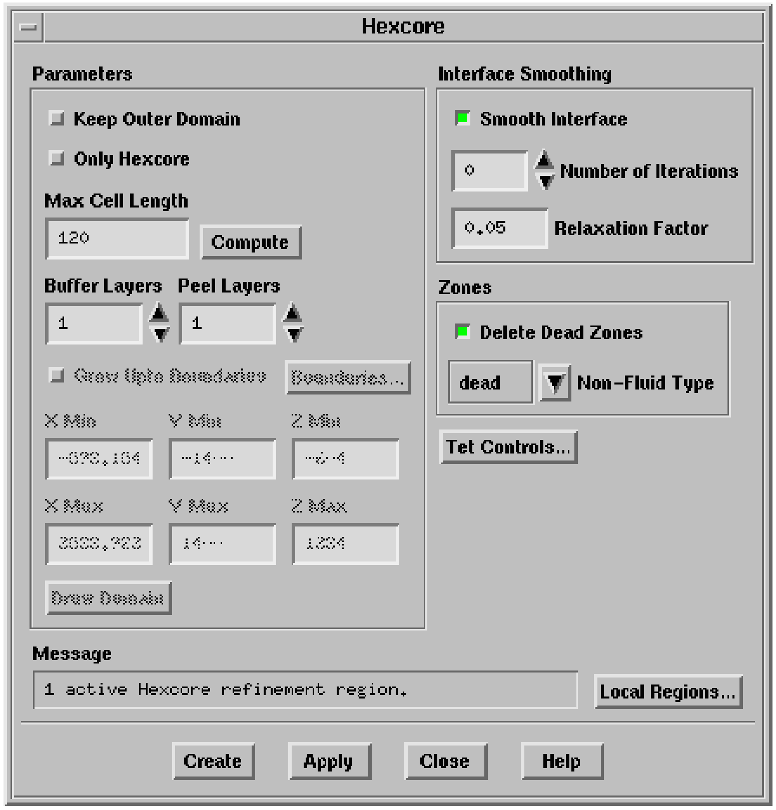
D. Use the hot-key Ctrl + D to calculate the distance between the selected nodes.

The distance between the nodes is computed to be approximately 75.

iii. Enter 120 for Max Cell Length.

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

iv. Set Buffer Layers to 1.



v. Enable Delete Dead Zones in the Zones group box.

vi. Click the Tet Controls... button to open the Tri/Tet panel.

A. Retain the settings in the Tri/Tet panel.

B. Close the Tri/Tet panel.

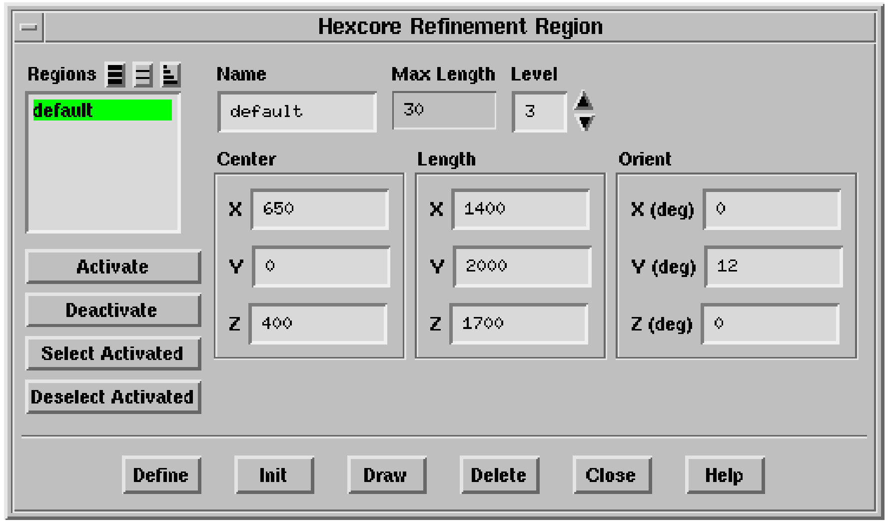
vii. Define and visualize the refinement region.

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

A. Click the Local Regions... button to open the Hexcore Refinement Region panel.

B. Retain `default` for Name.

C. Set Level to 3.



D. Enter the following parameters:

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

E. Click Define.

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

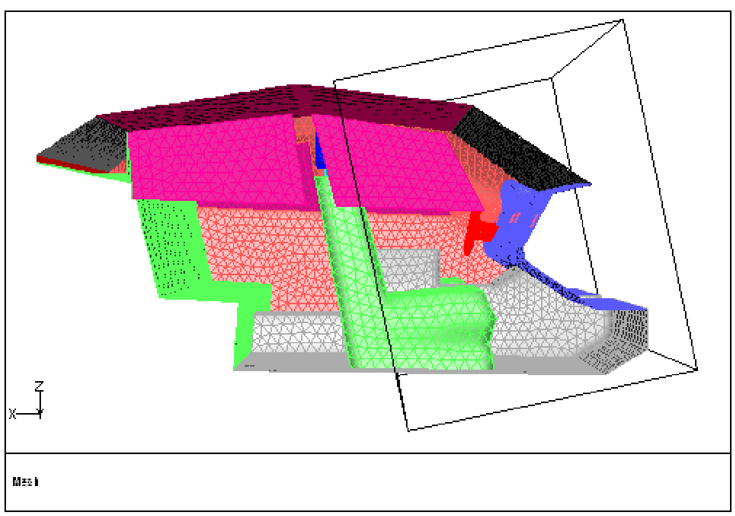


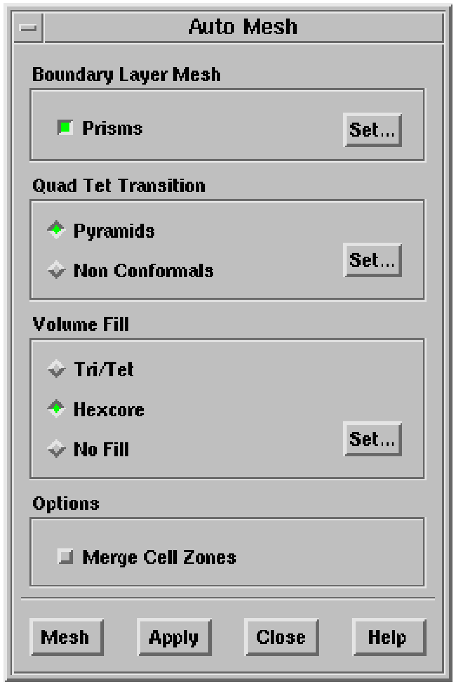
Figure 5.13: Refinement Region

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

H. Close the Hexcore Refinement Region panel.

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

(d) Click Apply and close the Hexcore panel.



4. Click Mesh in the Auto Mesh panel to generate the mesh.

5. Close the Auto Mesh panel.

6. Separate the hex cells from the tet cells for better visualization.

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

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

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

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

7. Examine the mesh by zone type.

   Mesh ⟶Manage...

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

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

8. Check the cell count and the maximum skewness.

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

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

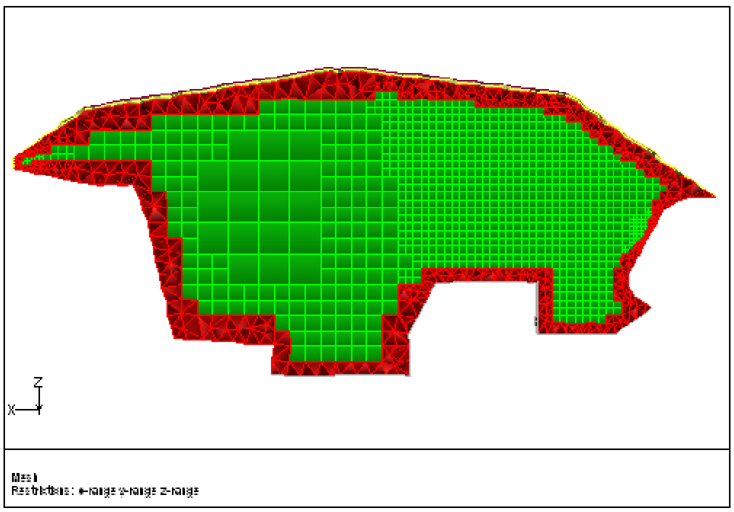


Figure 5.14: Slide of Cells at y = 50

10. Check the size of the largest hex cell.

    You should get a value close to 120*.*

Figure 5.15: Slide of Cells at y = -525

11. Save the mesh file.

    File ⟶ Write ⟶ Mesh...

12. Exit TGrid.

    File ⟶ Exit

Summary

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

Tutorial 6. Generating the Hexcore Mesh Upto Domain Boundaries

Introduction

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 until the domain boundaries, thereby also reducing the cell count. This tutorial demonstrates the generation of the hexcore mesh upto the domain boundaries for a sedan car.

This tutorial demonstrates how to do the following:

1. Read the mesh file and display the boundary mesh.
2. Set parameters for generating the hexcore mesh.
3. Set parameters for creating prism layers and the hexcore mesh using automatic meshing.
4. Check and save the volume mesh.

Prerequisites

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

Preparation

1. Download `hexcore-boundaries.zip` from the FLUENT User Services Center to your working directory. This file can be found from the Documentation link on the TGrid product page.

   OR

   Copy `hexcore-boundaries.zip` from the TGrid documentation CD to your working directory.

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

  cdrom`/tgrid5.0/help/tutfiles`

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

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

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

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

2. Unzip `hexcore-boundaries.zip`.

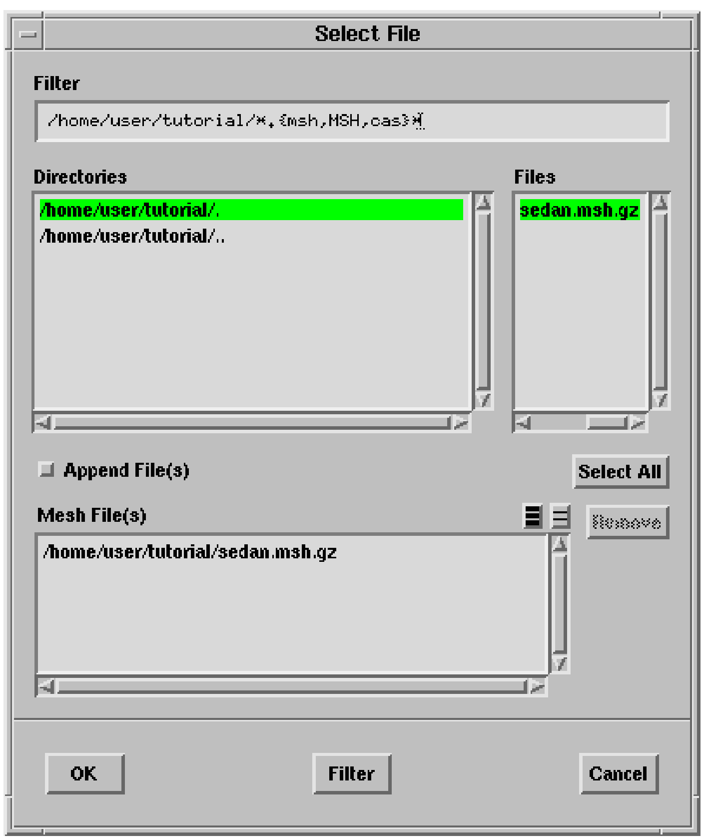
   The file, `sedan.msh.gz` *can be found in the* `hexcore-boundaries` *folder created on unzipping the file.*

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

Step 1: Read and Display the Boundary Mesh

1. Read the mesh file.

   File ⟶ Read ⟶ Boundary Mesh...

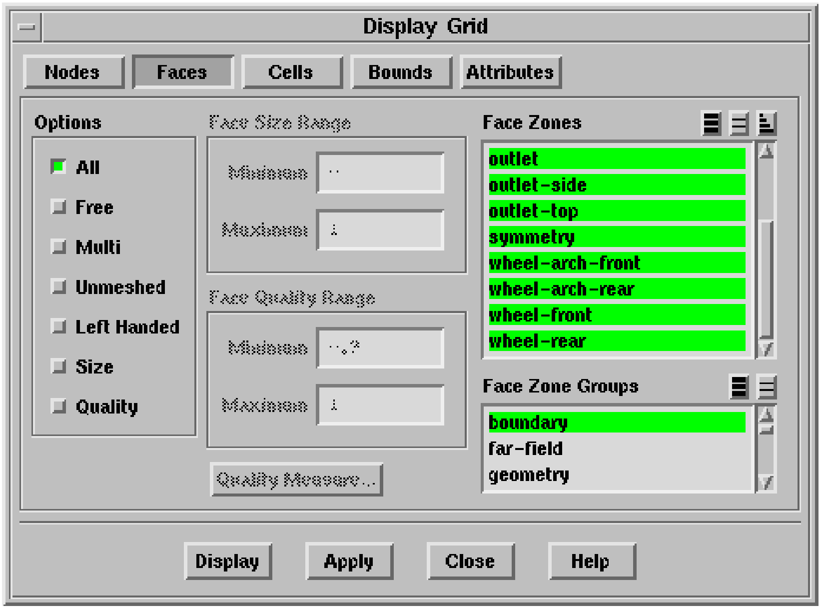


(a) Select sedan.msh.gz in the Files list.

(b) Click OK.

2. Display the boundary mesh (Figure 6.1).

Display →Grid...



(a) Select boundary in the Face Zone Groups selection list to select all the boundary zones in the Face Zones selection list.

(b) Click Display.

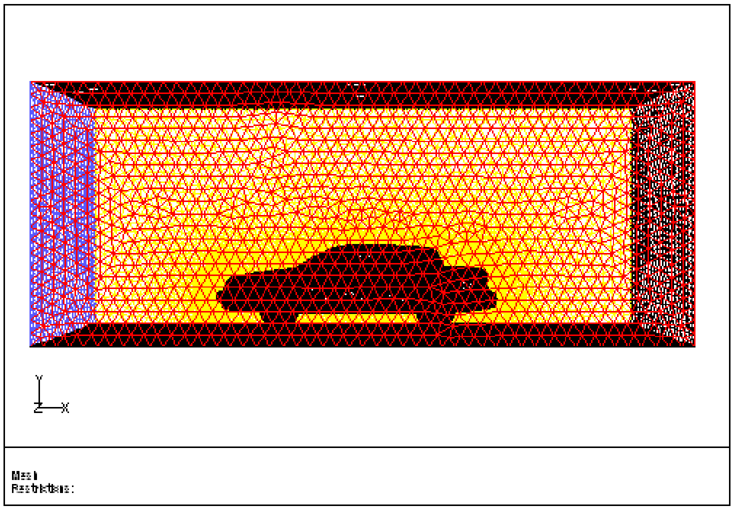


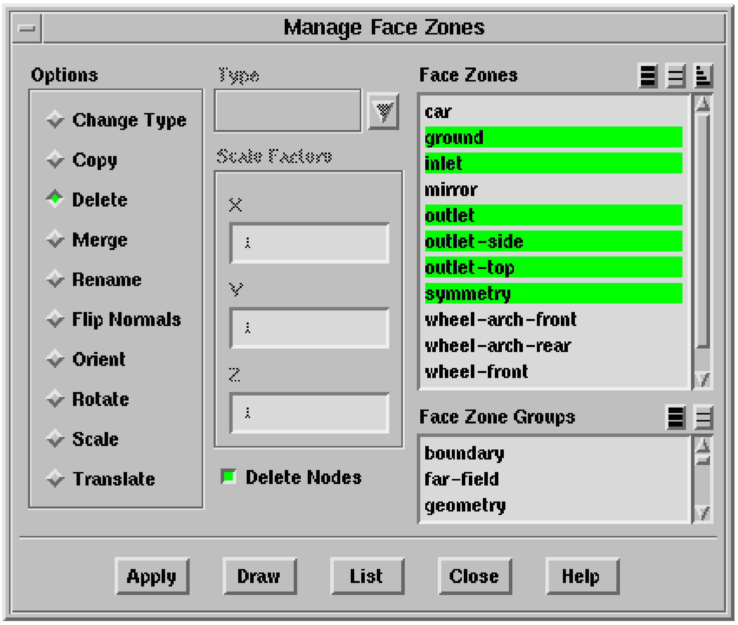
Figure 6.1: Boundary Mesh for the Sedan

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

(c) Close the Display Grid panel.

Step 2: Delete the Outer Box Boundaries

Boundary ⟶Manage...



1. 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 and click Apply.

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

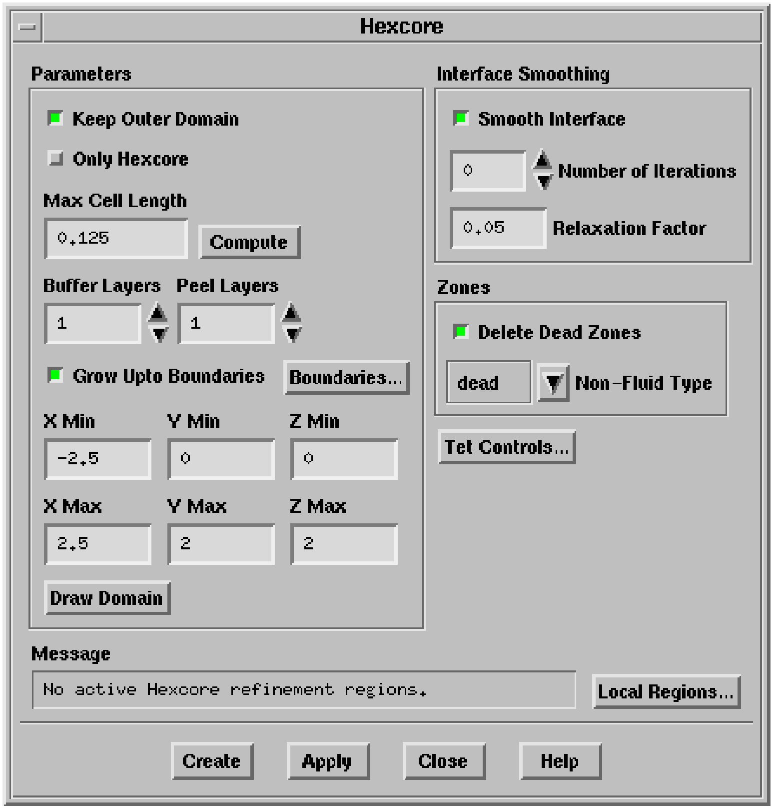
4. Click Yes in the Question dialog box.

5. Select all the zones in the Face Zones selection list and click Draw.

6. Close the Manage Face Zones panel.

Step 3: Set the Hexcore Meshing Parameters

Mesh ⟶Hexcore...



1. Enable Keep Outer Domain in the Parameters group box.
2. Retain the setting of both Buffer Layers and Peel Layers to 1.
3. Enable Grow Upto Boundaries.
4. 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
5. Click Draw Domain to view the size of the domain.
6. Enter 0.125 for Max Cell Length.
7. Enable Delete Dead Zones in the Zones group box.

   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.*

8. Click Create.

9. Close the Hexcore panel.

10. Examine the mesh generated.

    Display ⟶Grid...

    (a) Deselect the previous selections and select boundary in the Face Zone Groups selection list.

    (b) Click the Attributes tab and enable Filled and Lights in the Options group box.

    (c) Click the Colors... button to open the Grid Colors panel.

    (d) Select Color by ID in the Options list and close the Grid Colors panel.

    (e) Display the back view.

        Display ⟶Views...

        i. Select back from the Views list.

        ii. Click Apply and close the Views panel.

    (f) Click Display in the Display Grid panel (Figure 6.2).

        In Figure 6.2, you can see that the hex cells are generated upto the extents of the domain defined in the Hexcore *panel. For easy recognition, the newly created zones are named* wall-x-max, wall-x-min, wall-y-max, *etc.*

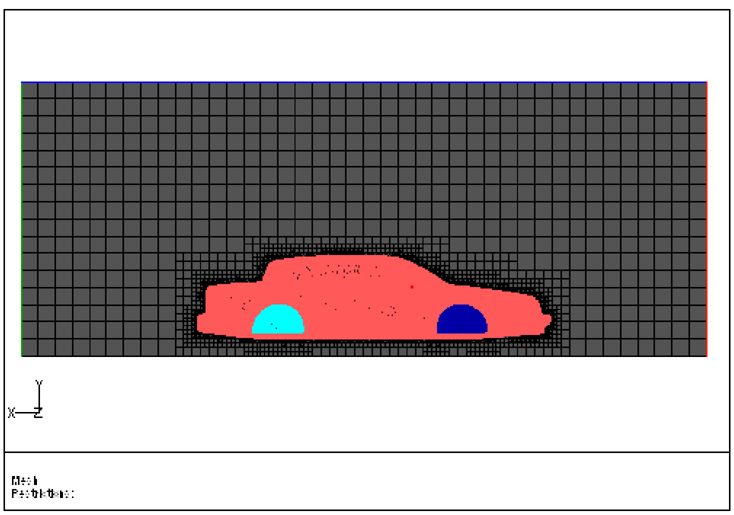


Figure 6.2: Hexcore Mesh Upto Domain Boundaries

    (g) Zoom in to the region shown in Figure 6.3.

        In Figure 6.3, you can see that the mesh on the boundaries is made up of quad cells, except near the sedan.

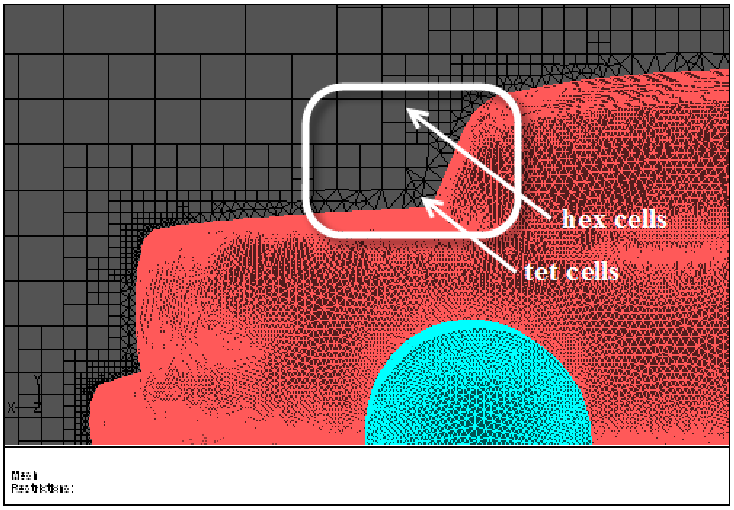


Figure 6.3: Hexcore Mesh Upto Domain Boundaries—Zoomed View

Step 4: Automatically Generate the Hexcore Mesh Upto the Boundaries with Prism Layers

In this step, you will set parameters for creating prism layers on the sedan body before 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.

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

   File ⟶ Read ⟶Boundary Mesh...

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

2. Display the grid.

   Display ⟶Grid...

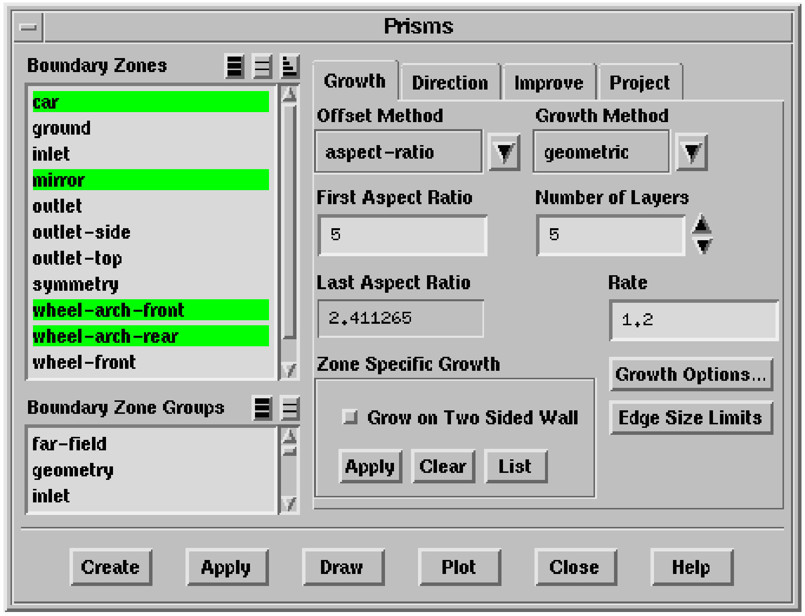
3. Set the meshing parameters.

   Mesh ⟶Auto Mesh...

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

   (a) Set the prism meshing parameters.

      i. Click the Set... button in the Boundary Layer Mesh group box to open the Prisms panel.



ii. Select car, mirror, wheel-arch-front, and wheel-arch-rear in the Boundary Zones selection list.

iii. Select aspect-ratio in the Offset Method drop-down list. and enter 5 for First Aspect Ratio.

iv. Select geometric in the Growth Method drop-down list and enter `1.2` for Rate.

v. Set Number of Layers to 5.

vi. Click Apply in the Zone Specific Growth group box.

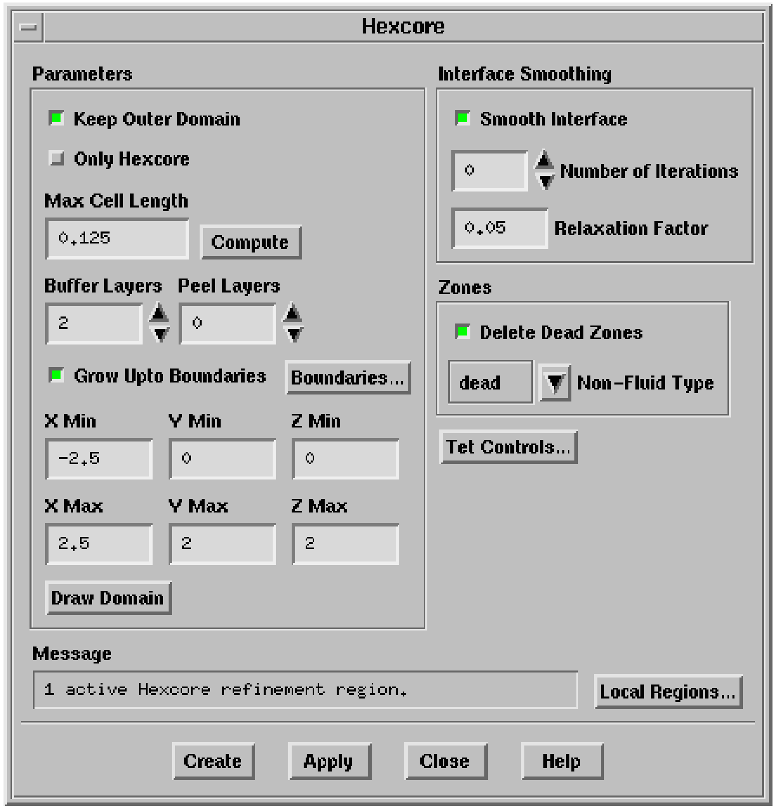
> ⚠ It is necessary to apply the prism growth parameters on specific zones for TGrid to retain the growth parameters in memory. The Prisms option in the Auto Mesh panel will be visible only after applying zone-specific growth.

vii. Close the Prisms panel.

viii. Enable Prisms in the Boundary Layer Mesh group box in the Auto Mesh panel.

(b) Set the hexcore meshing parameters.

i. Select Hexcore in the Volume Fill list and click the Set... button to open the Hexcore panel.

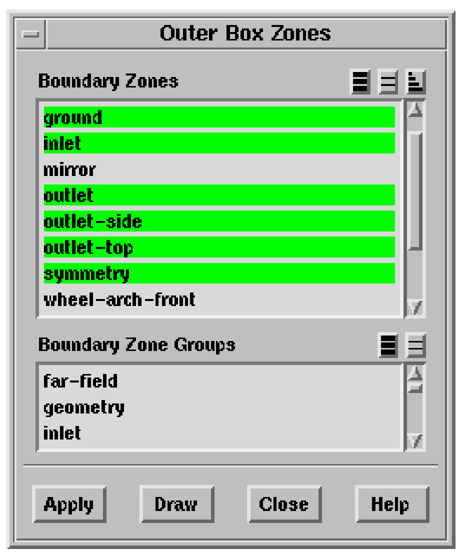


ii. Enable Keep Outer Domain in the Parameters group box.

iii. Enable Grow Upto Boundaries.

The outer box already exists, hence, you need to select the zones to be replaced.

iv. Click the Boundaries... button to open the Outer Box Zones panel.



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

B. Click Apply and close the Outer Box Zones panel.

The domain extents will automatically be set to the size of the bounding box. Make sure the domain extents are as follows:

X Min -2.5, Y Min 0, Z Min 0

X Max 2.5, Y Max 2, Z Max 2

v. Click Draw Domain to check the size of the domain to be meshed.

vi. Enter 0.125 for Max Cell Length.

vii. Set Buffer Layers to 2.

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

viii. Set Peel Layers to 0.

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.

ix. Enable Delete Dead Zones in the Zones group box.

x. Click the Tet Controls... button to open the Tri/Tet panel.

A. Retain the default settings in the Initialization tab.

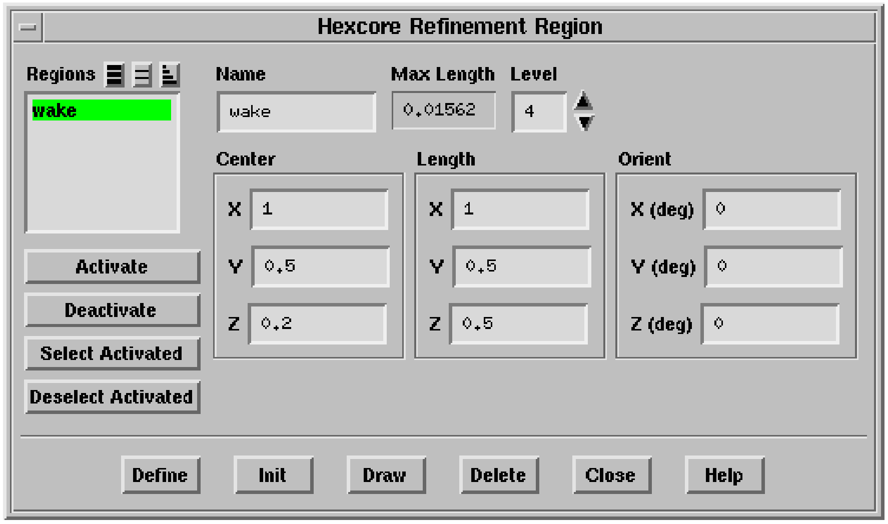
B. Click the Refinement tab and retain the selection of geometric in the Cell Size Function drop-down list.

C. Enter 1.3 for Growth Rate and click Apply.

D. Close the Tri/Tet panel.

xi. Define the local refinement region.

A. Click the Local Regions... button to open the Hexcore Refinement Region panel.



B. Enter `wake` for Name.

C. Set Level to `4`.

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

In this case, the Max Length *is equal to* $\frac{0.125}{2^{(4-1)}} = 0.01562$.

D. Enter (`1`, `0.5`, `0.2`) for Center and (`1`, `0.5`, `0.5`) for Length, respectively.

E. Retain the default orientation.

F. Click Draw and check the region extents and maximum length within the region.

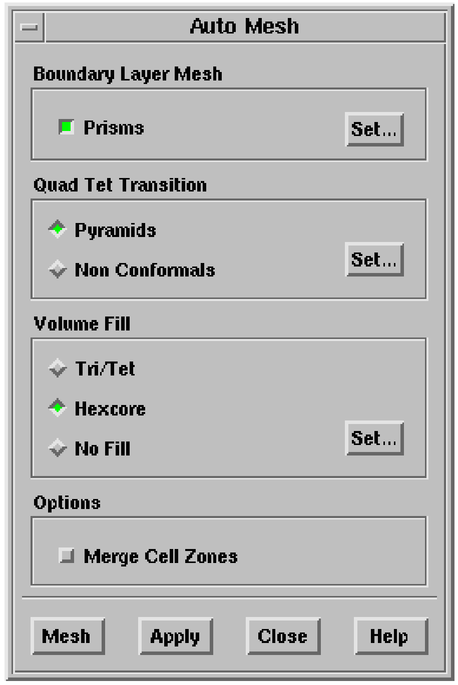
G. Click Define to create the region.

H. Retain the selection of wake in the Regions selection list and click Activate.

I. Close the Hexcore Refinement Region panel.

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

xii. Click Apply and close the Hexcore panel.



(c) Click Mesh.

(d) Close the Auto Mesh panel.

4. Examine the mesh.

Display ⟶Grid...

You can see that there are two sets of the outer box zones—the original triangular mesh zone (e.g., ground:old*) and the new zone created during hexcore meshing (e.g.,* ground*).*

(a) Deselect the previous selections and select car, ground, inlet, mirror, outlet, outlet-side, outlet-top, symmetry, wheel-arch-front, wheel-arch-rear, wheel-front, and wheel-rear in the Face Zone Groups selection list.

(b) Click the Attributes tab and enable Filled and Lights in the Options group box.

(c) Click the Colors... button to open the Grid Colors panel.

(d) Select Color by ID in the Options list and close the Grid Colors panel.

(e) Display the back view.

Display ⟶Views...

(f) Click Display in the Display Grid panel.

(g) Display the refinement region along with the mesh.

i. Click the Local Regions... button in the Hexcore panel to open the Hexcore Refinement Region panel.

Mesh ⟶Hexcore...

ii. Make sure wake is selected in the Regions selection list and click Draw.

iii. Close the Hexcore Refinement Region panel.

In Figure 6.4, you can see that the hex cells are generated upto the boundaries defining the outer box. You can also see the refinement region.

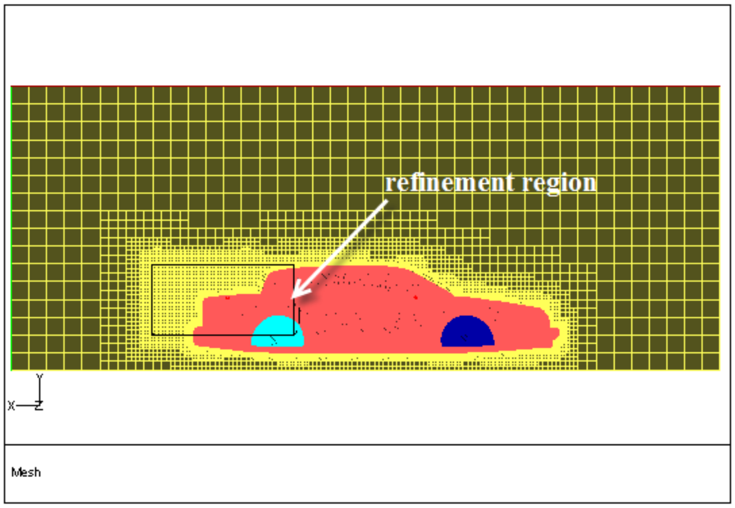


Figure 6.4: Hexcore Mesh Upto Outer Box Boundaries

(h) Separate the hex cells from the tet cells for better visualization.

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

Moved 595067 cells from fluid-# to fluid-#:#
Moved 1060007 triangular faces from interior-# to new zone interior-#:#.
Moved 134450 triangular faces from interior-# to new zone interior-#:#.
Moved 2559 mixed faces from ground to new zone ground:#.
Moved 1935 quadrilateral faces from symmetry to new zone symmetry:#.
Moved 1270 mixed faces from symmetry to new zone symmetry:#.

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

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

(i) Click the Bounds tab in the Display Grid panel and enable Limit by Z.

(j) Enter 0.37 for Minimum and Maximum in the Z Range group box.

(k) Click the Cells tab in the Display Grid panel and select all the zones in the Cell Zones selection list.

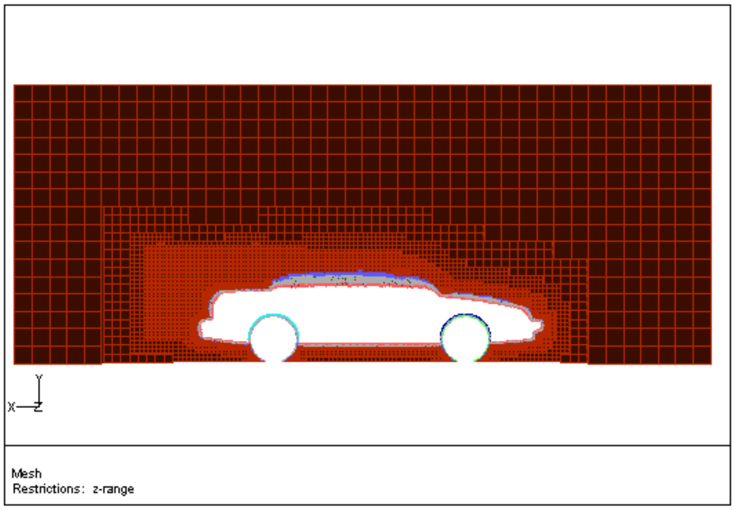


Figure 6.5: Slide of Cells at z = 0.37

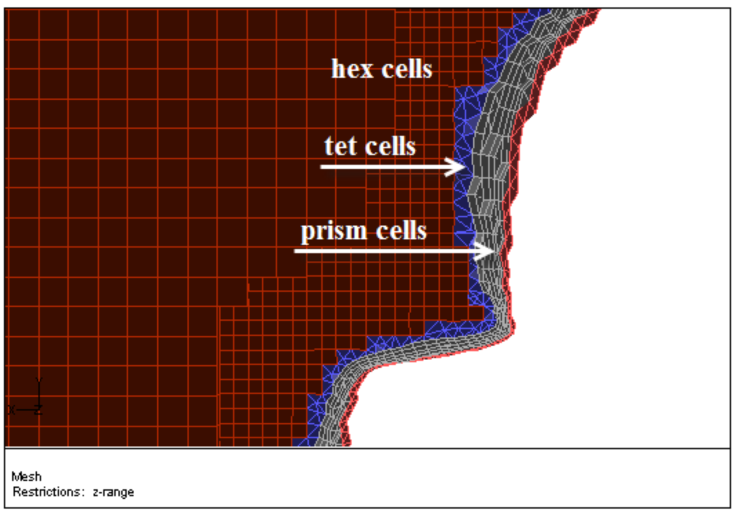


Figure 6.6: Hexcore Mesh Upto Domain Boundaries—Zoomed View

(l) Enable All in the Options group box and click Display.

(m) Zoom in to the region shown in Figure 6.6.

In Figure 6.6, you can see that the mesh on the boundaries is made up of quad cells, except near the sedan.

5. Delete the old outer box zones.

$\boxed{\text{Boundary}}$ ⟶Manage...

(a) Select ground:old, inlet:old, outlet-side:old, outlet-top:old, outlet:old, and symmetry: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 will appear, asking you to confirm if you want to delete the selected zones.

(d) Click Yes in the Question dialog box.

(e) Close the Manage Face Zones panel.

6. Check the mesh.

   $\boxed{\text{Mesh}}$ ⟶Check

7. Save the mesh file.

   $\boxed{\text{File}}$ ⟶ $\boxed{\text{Write}}$ ⟶Mesh...

8. Exit TGrid.

   $\boxed{\text{File}}$ ⟶Exit

Summary

This tutorial demonstrated the generation of a hexcore mesh upto 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.

Tutorial 7. Using the Boundary Wrapper

Introduction

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 in TGrid 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:

1. Read and display the mesh.
2. Perform pre-wrapping operations to close holes in the geometry.
3. Initialize the wrapper.
4. Check the region to be wrapped.
5. Refine the Cartesian grid using the local size function.
6. Wrap the surface and imprint necessary features.
7. Check the deviation of the wrapper surface from the original geometry.
8. Perform post-wrapping operations to improve wrapper surface quality.
9. Check and save the mesh.

The V-8 engine geometry used in this tutorial is provided by Platinum Pictures Multimedia Inc., www.3dcafe.com, and Michael Barthels.

Prerequisites

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

Preparation

1. Download `wrapper.zip` from the FLUENT User Services Center

   This file can be found from the Documentation link on the TGrid product page.

   OR

   Copy `wrapper.zip` from the TGrid documentation CD to your working directory.

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

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

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

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

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

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

2. Unzip `wrapper.zip`.

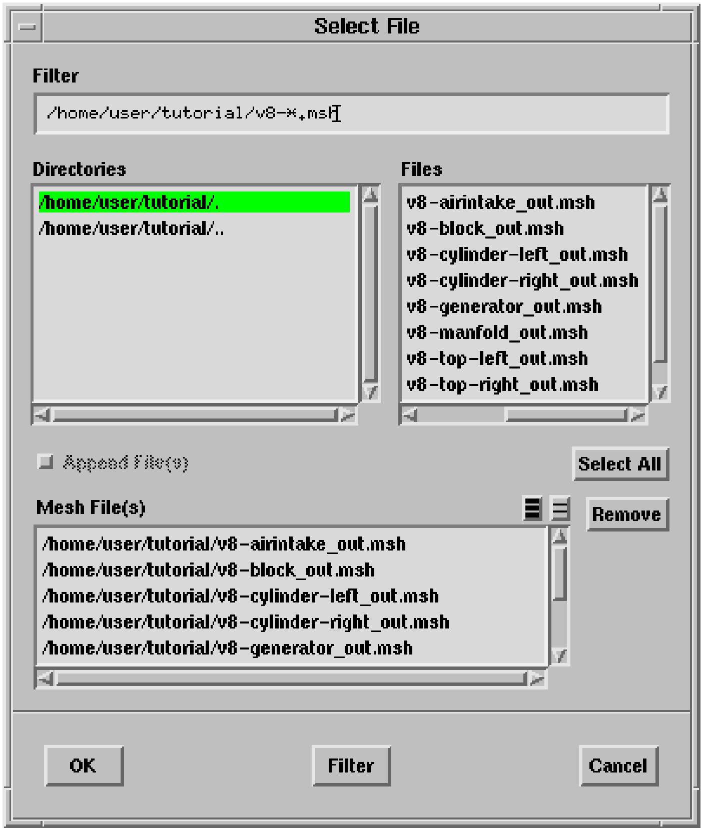
   The files `v8-airintake_out.msh`*,* `v8-block_out.msh`*,* `v8-cylinder-left_out.msh`*,* `v8-cylinder-right_out.msh`*,* `v8-generator_out.msh`*,* `v8-manfold_out.msh`*,* `v8-top-left_out.msh`*,* `v8-top-right_out.msh`*, and* `v8-wheels_out.msh` *can be found in the* `v8` *folder created on unzipping the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

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

Step 1: 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 text box and click Filter.

   (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 7.1).

   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 panel.

      i. Select Color by ID in the Options list.

      ii. Close the Grid Colors panel.

(d) Click the Rendering... button to open the Display Options panel.

i. Enable Double Buffering and Hidden Line Removal.

ii. Select Software Z-buffer in the Hidden Surface Method drop-down list.

iii. Click Apply and close the Display Options panel.

(e) Click Display in the Display Grid panel and rotate the geometry about the x-axis to obtain the view shown in Figure 7.1.

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

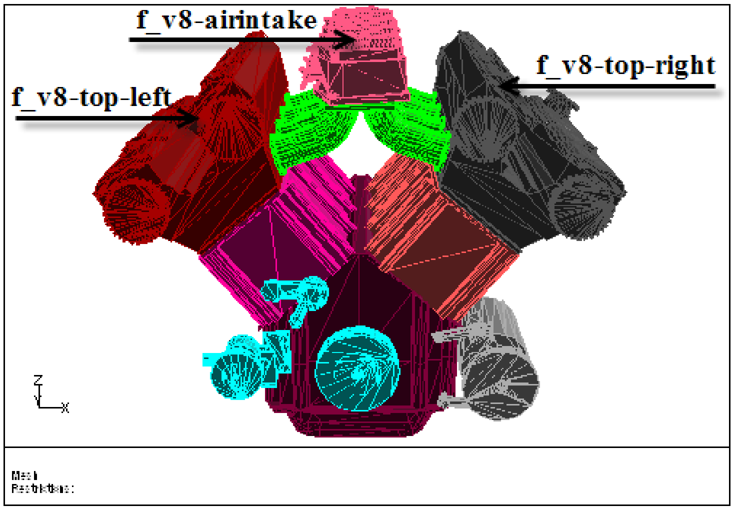


Figure 7.1: Grid Display

Step 2: 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. TGrid *provides a variety of options for closing arbitrary openings between zones, arbitrary openings in the same zone, coplanar openings, etc.*

In this step, you will close the holes in the following zones:

- f_v8-airintake *(see Figure 7.2)*
- f_v8-top-right *(see Figure 7.5)*
- f_v8-top-left *(see Figure 7.8)*

1. Close the holes in the f_v8-airintake zone.

   (a) Select the f_v8-airintake zone in the display window (click on it with the right mouse button).

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

   OR

   Use the hot-key, Ctrl + Z to select zone as the Filter.

   (b) Click the Bounds tab in the Grid Display panel.

   The selected zone, f_v8-airintake *will be added in the* Object Name *field in the* Neighborhood *group box.*

   (c) Click Set Ranges to update the X Range, Y Range, and Z Range fields.

   (d) Click Display and rotate the geometry to obtain the view shown in Figure 7.2.

   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.*

   ⚠ The remaining hole in the f_v8-airintake zone will be closed in **Step 4** after demonstrating the use of the Pan Regions and Trace Path features to detect and trace a leak in the geometry.

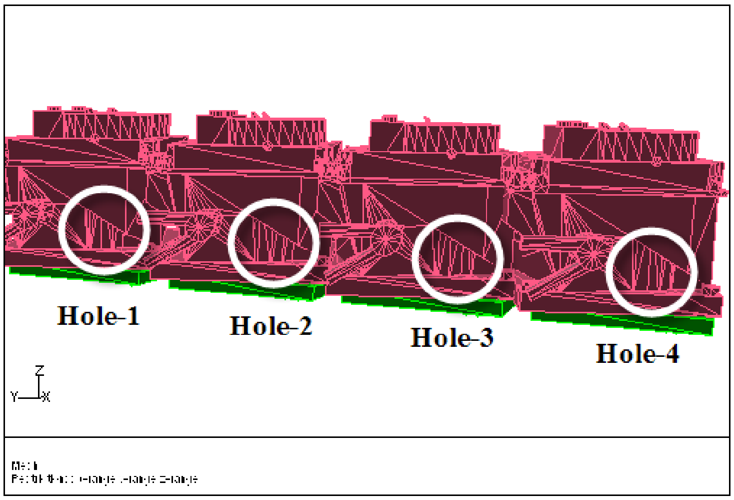


Figure 7.2: Holes to be Closed in the f_v8-airintake Zone

   (e) Zoom in to the region shown in Figure 7.3.

   (f) Select node in the Filter list in the Modify Boundary panel.

   Boundary ⟶Modify...

   OR

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

(g) Select the three nodes surrounding the hole using the right mouse button (Figure 7.3).

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.*

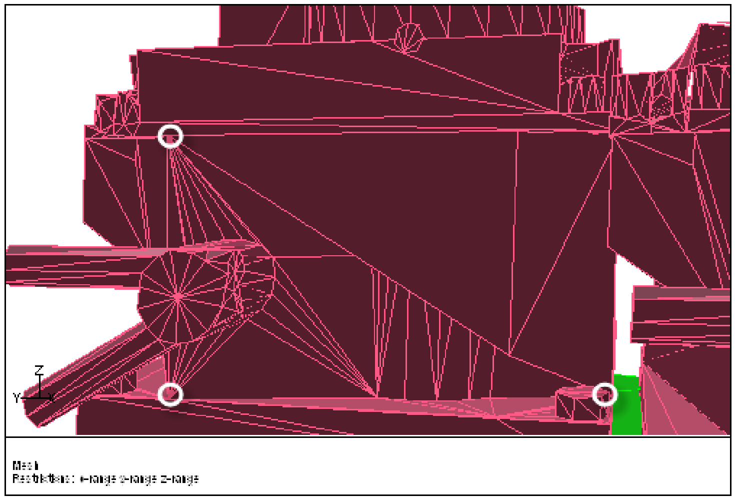


Figure 7.3: Nodes Selected to Close Hole-1 in f_v8-airintake

(h) Click Create in the Operation group box.

OR

Use F5.

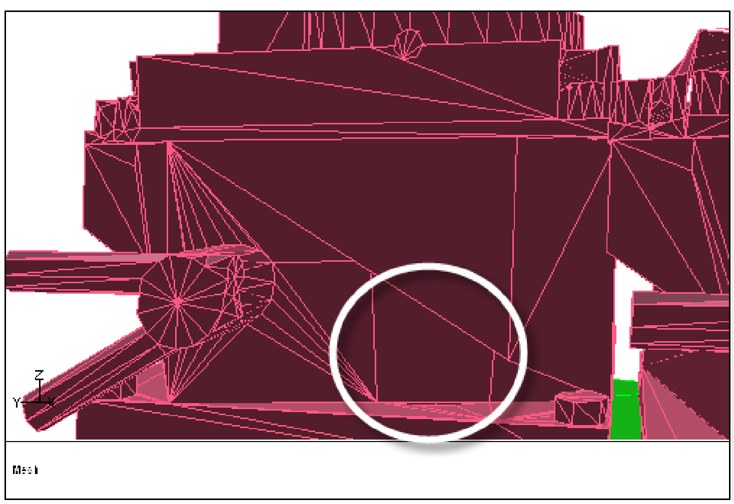


Figure 7.4: Closed Hole (Hole-1) in the f_v8-airintake Zone

TGrid *creates a face partially covering the hole, making the hole smaller than the mesh size used in this tutorial (see Figure 7.4).*

(i) Similarly, close the second and third holes (Hole-2 and Hole-3) in the f_v8-airintake zone.

> ⚠ You will close the remaining hole in **Step 4**.

2. Close the holes in the f_v8-top-right zone.

(a) Click Reset in the Bounds tab of the Display Grid panel.

(b) Display the grid.

(c) Select zone in the Filter list in the Modify Boundary panel (or use the hot-key, Ctrl + Z).

(d) Select the f_v8-top-right zone and click the Set Ranges button in the Bounds tab of the Display Grid panel.

(e) Click Display and pan to the region shown in Figure 7.5.

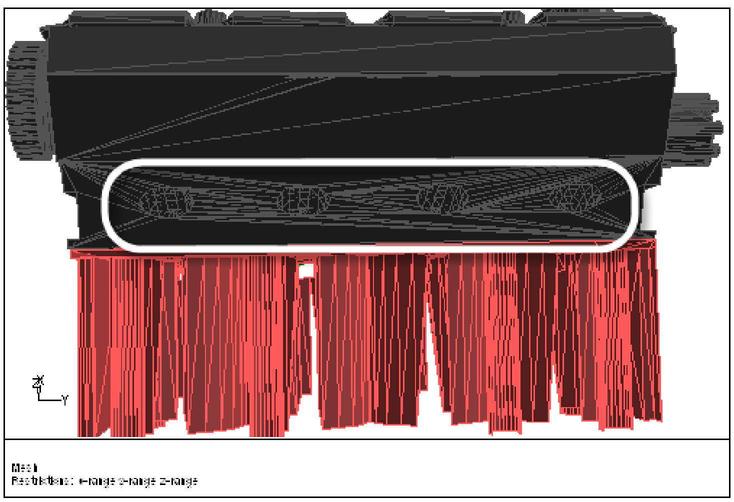


Figure 7.5: Coplanar Holes in the f_v8-top-right Zone

(f) Clear the Selections list in the Modify Boundary panel 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.

(g) Select the six positions shown in Figure 7.6 using the right mouse button.

The positions should be selected in either clockwise or anticlockwise order.

(h) Click Create in the Operation group box (or use F5).

TGrid *creates six boundary nodes at the selected positions. These nodes are automatically selected in the* Selections *list.*

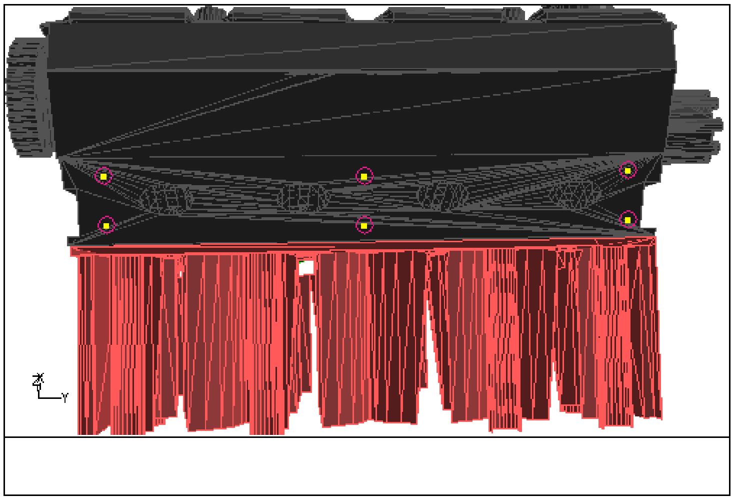


Figure 7.6: Nodes Selected to Close Holes in the f_v8-top-right Zone

(i) Retain the selection of the newly created nodes in the Selections list.

(j) Select zone in the Filter list and select the f_v8-top-right zone (or use the hot-key, Ctrl + Z).

(k) Select node in the Filter list (hot-key, Ctrl + N) and click Create (or use F5).

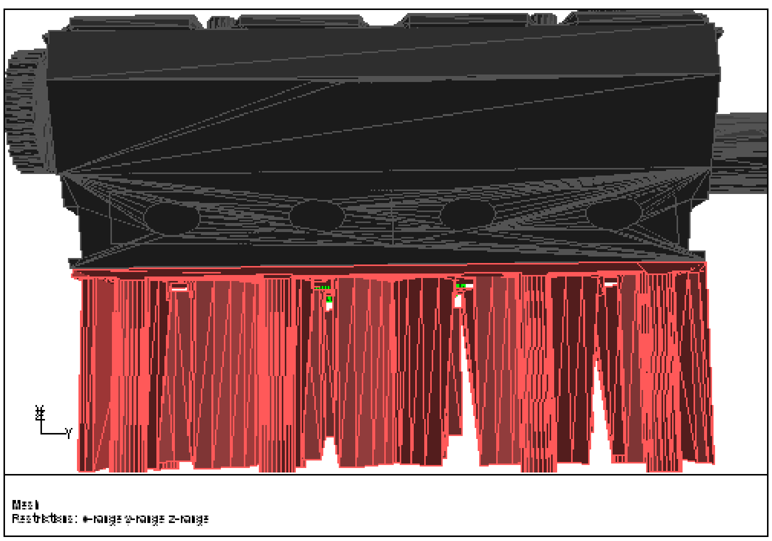


Figure 7.7: Holes Closed in the f_v8-top-right Zone

3. Close the holes in the f_v8-top-left zone by creating a plane surface.

   (a) Click Reset in the Bounds tab in the Display Grid panel.

   (b) Display the grid.

(c) Select zone in the Filter list in the Modify Boundary panel (or use the hot-key, Ctrl + Z).

(d) Select the f_v8-top-left zone and click the Set Ranges button in the Bounds tab of the Display Grid panel.

(e) Click Display and pan to the region shown in Figure 7.8.

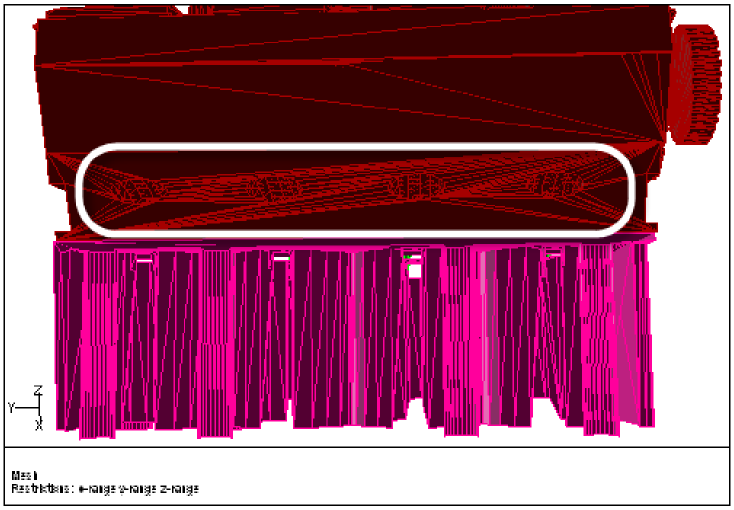
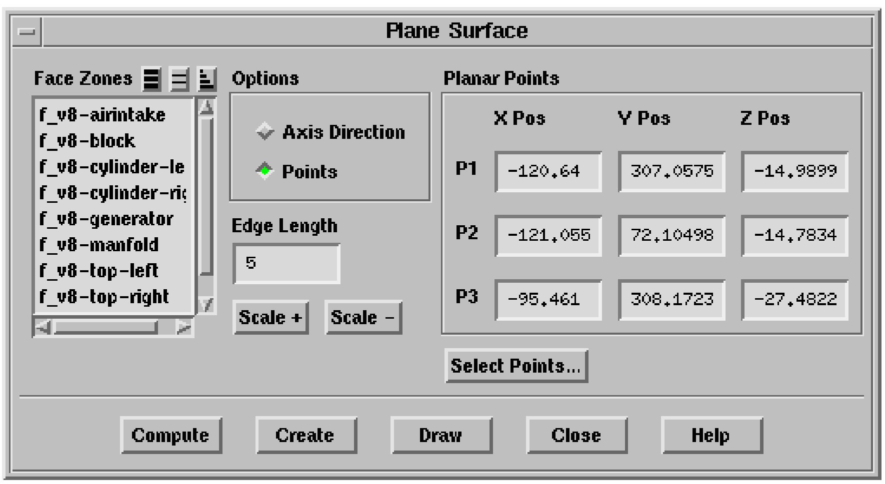


Figure 7.8: Coplanar Holes in the f_v8-top-left Zone

(f) Create a plane surface to cover the coplanar holes.

Boundary ⟶ Create ⟶ Plane Surface...



i. 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 7.9).

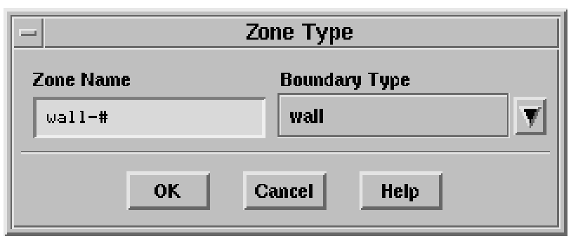
Select the two points along the longest side first. The coordinates of the three selected points will be updated in the Planar Points *group box.*

Figure 7.9: Selection of Points for Creating the Plane Surface

iv. Enter 5 for Edge Length.

v. Click Create.

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



vi. Click OK in the Zone Type panel.

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

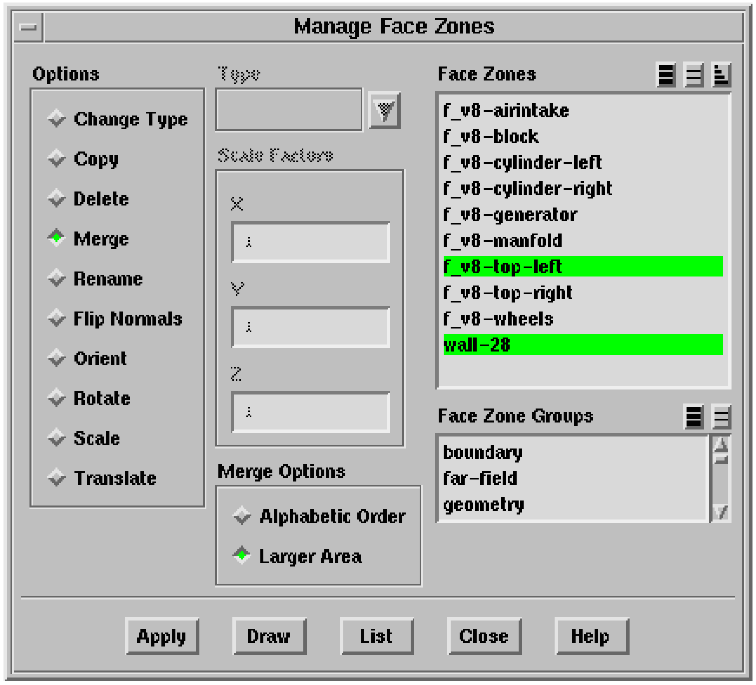
vii. Close the Plane Surface panel.

(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...

i. Select f_v8-top-left and wall-# in the Face Zones selection list.



ii. Select Merge in the Options list.

iii. Select Larger Area in the Merge Options list and click Apply.

The wall-# *zone will be merged with the* f_v8-top-left *zone.*

iv. Close the Manage Face Zones panel.

(h) Click Display in the Display Grid panel to see the recently closed holes (Figure 7.10).

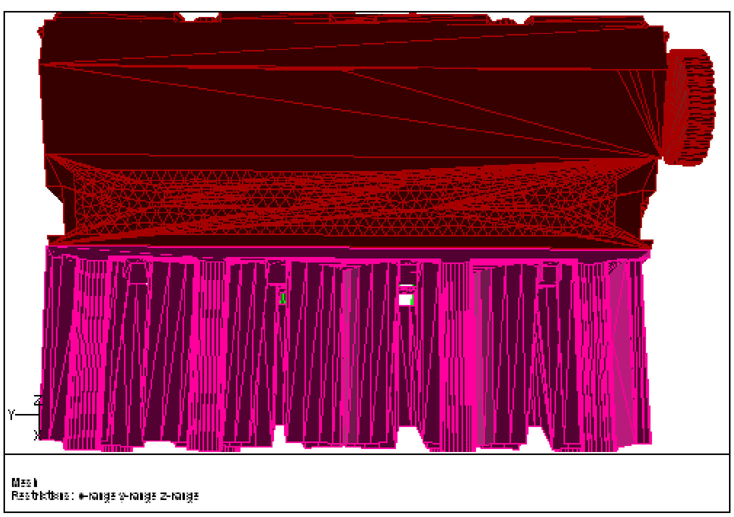


Figure 7.10: Holes Closed in the f_v8-top-left Zone Using a Plane Surface

4. Save the mesh (`engine.msh.gz`).

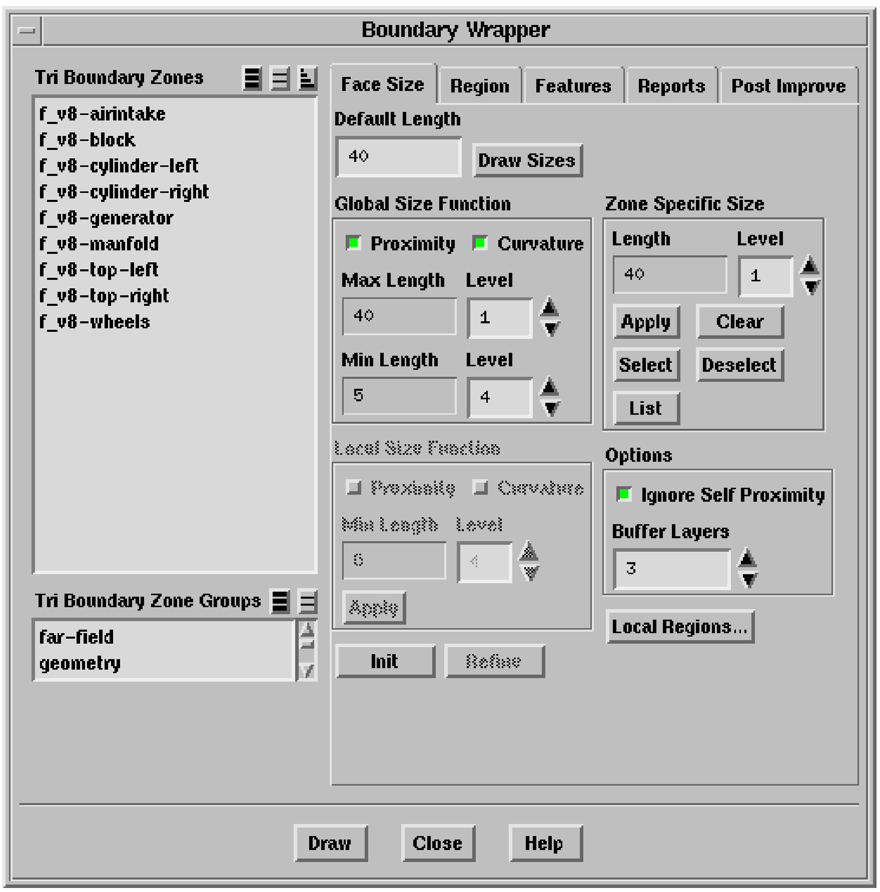
   File ⟶ Write ⟶ Mesh...

Step 3: Initialize the Surface Wrapper

The accuracy of the wrapper depends on the cell size distribution of the Cartesian grid. TGrid *allows you to 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.

   TGrid *will create a Cartesian hanging node grid and refine it based on the defined size functions. The grid will then be intersected with the geometry, thereby creating a number of Cartesian closed regions.* TGrid *will report the regions created.*

Step 4: Check the Region to be Wrapped

1. Click Reset in the Bounds tab of the Display Grid panel.

2. Click the Region tab in the Boundary Wrapper panel.

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

   Figure 7.11 shows the region to be wrapped.

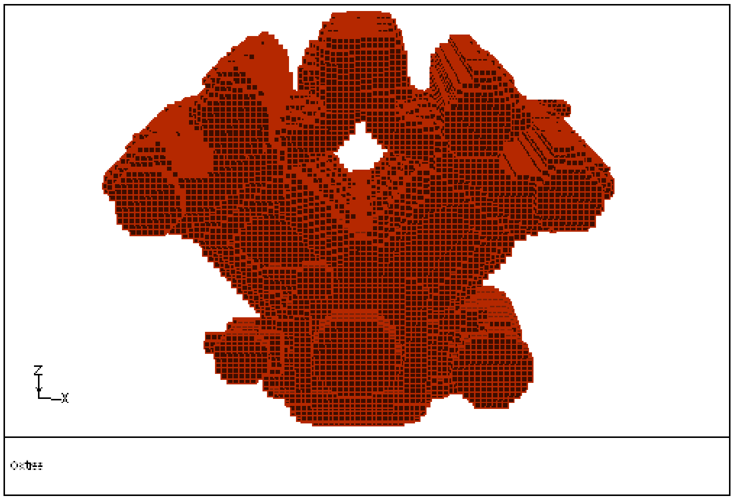


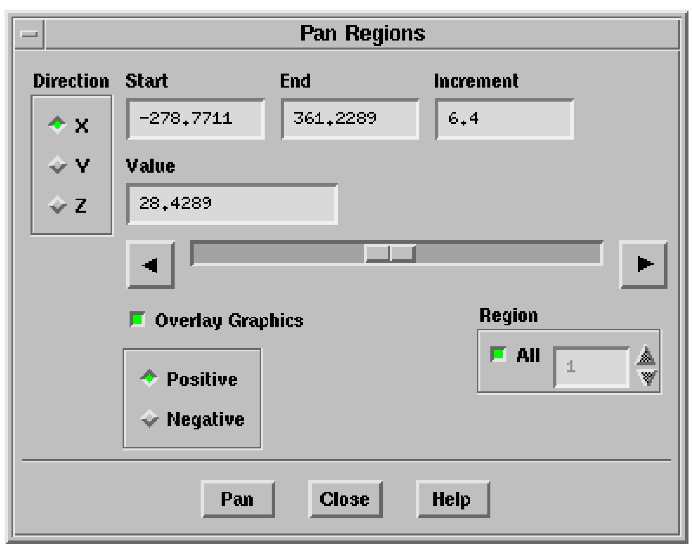
Figure 7.11: Region to be Wrapped

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

   (a) Display the entire geometry.

   $\boxed{\text{Display}}\longrightarrow$Grid...

   (b) Click the Pan Regions... button in the Region tab of the Boundary Wrapper panel to open the Pan Regions panel.



   i. Enable Overlay Graphics.

   ii. Move the slider bar to position the pan plane as shown in Figure 7.12 (Value $\approx$ 28).

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

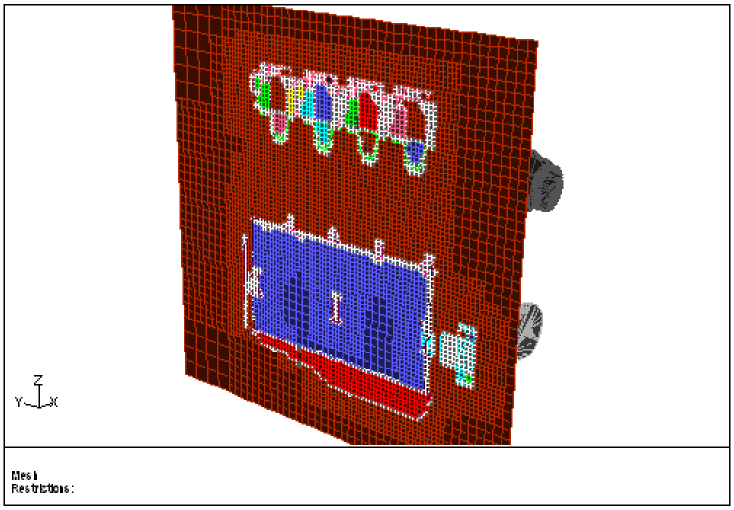


Figure 7.12: Pan Plane with the Overlaid Geometry

iii. Zoom in to the region of the leak (Figure 7.13).

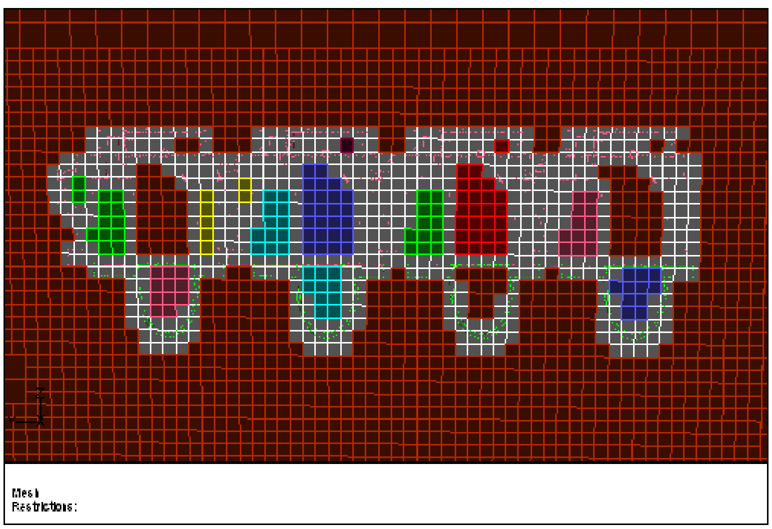


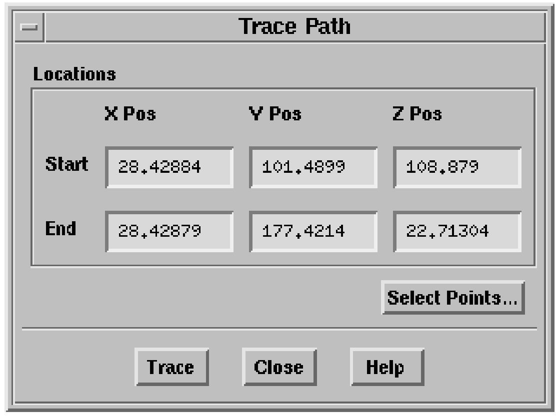
Figure 7.13: Region of the Leak

The presence of the main region color on the inside (Figure 7.12) 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 (Figure 7.14).

(c) Use the Trace Path feature to detect the leakage.

i. Click the Trace Path... button in the Region tab of the Boundary Wrapper panel to open the Trace Path panel.



ii. Click the Select Points... button and select two points as shown in Figure 7.14.

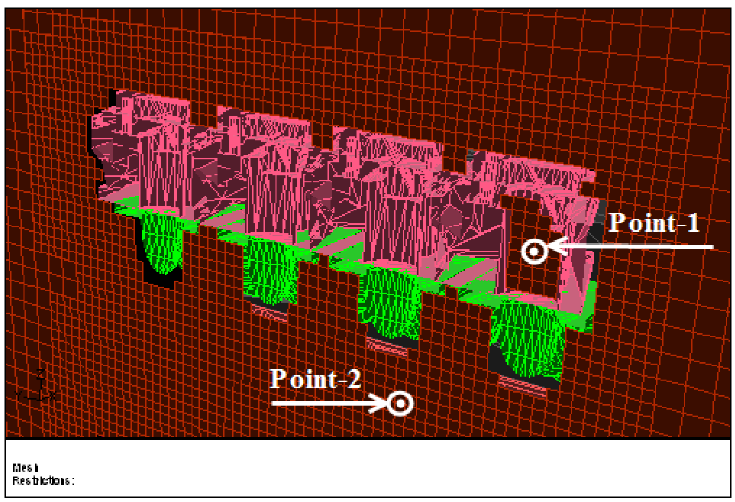


Figure 7.14: Detecting the Leakage using the Trace Path Panel

iii. Click Trace (Figure 7.15).

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

You may need to switch between Positive *and* Negative *in the* Pan Regions *panel to obtain the display in Figure 7.15.*

iv. Zoom in to the hole and select three nodes as shown in Figure 7.16.

v. Click Create in the Modify Boundary panel or use the hot-key F5 to close the hole.

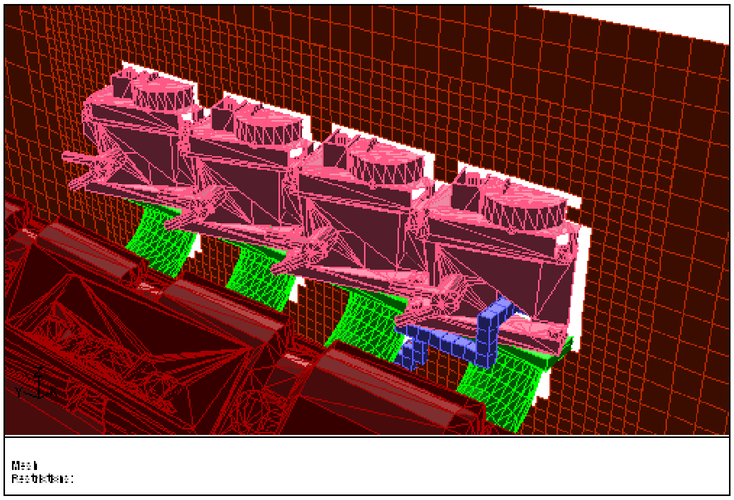


Figure 7.15: Display of Geometry with the Traced Path

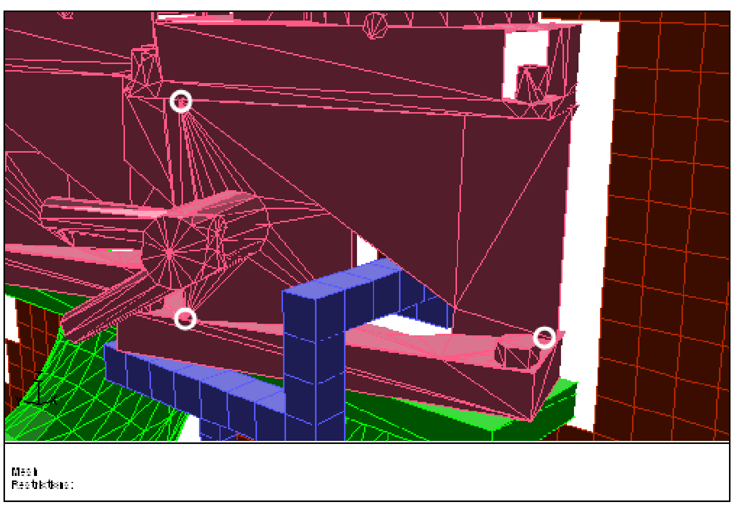


Figure 7.16: Nodes Selected to Close the Hole in the f_v8-airintake Zone

5. Select all the geometry in the Tri Boundary Zones selection list in the Boundary Wrapper panel.

6. Click Update Regions in the Region tab to update the region based on the newly added face (see Figure 7.17).

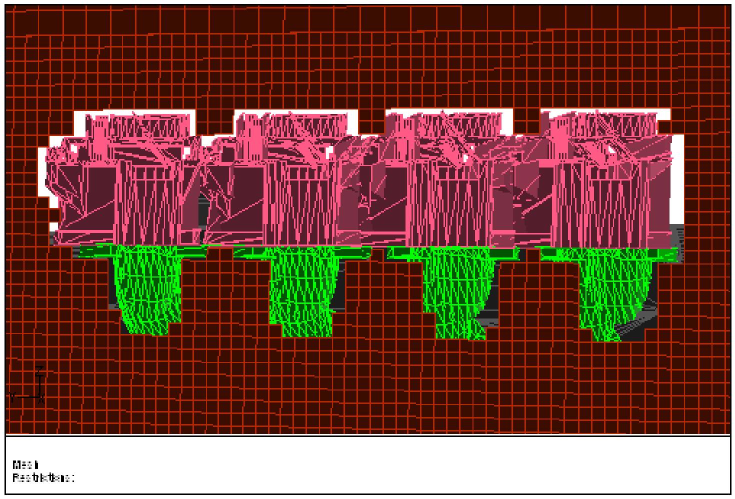


Figure 7.17: Region Updated for the Closed Hole

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.

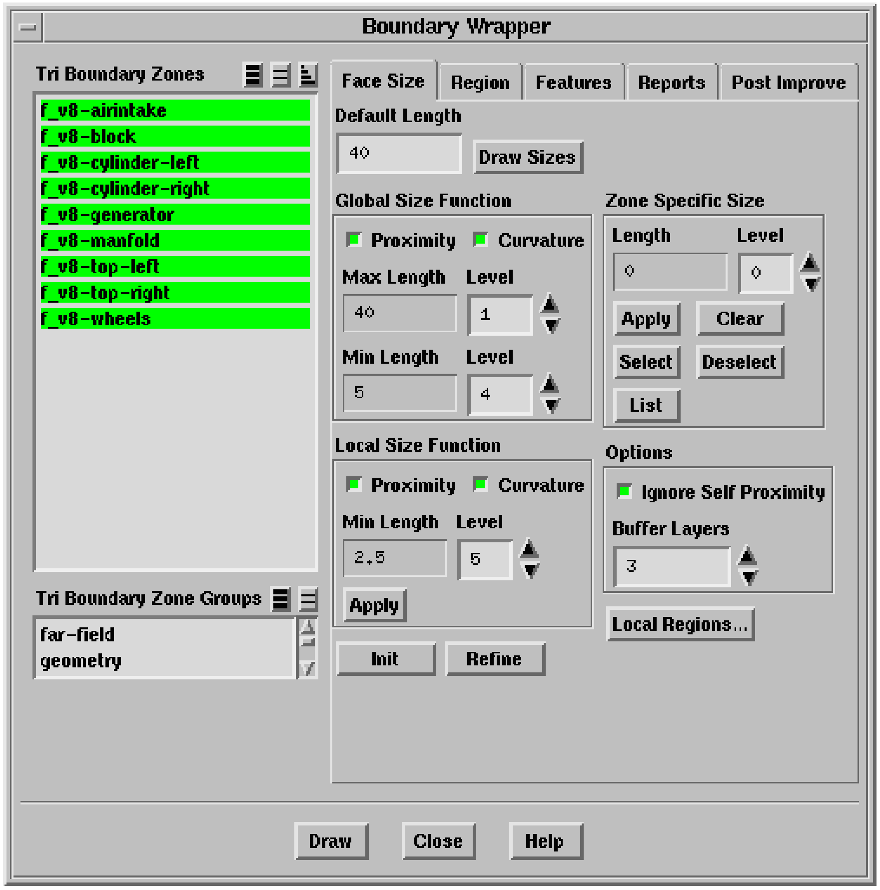
7. Close the Pan Regions and Trace Path panels.

Step 5: Refine the Main Region

1. Refine the Cartesian grid using the Local Size Function.

   (a) Enable Proximity and Curvature in the Local Size Function group box in the Face Size tab of the Boundary Wrapper panel.

   (b) Set Level to 5 in the Local Size Function group box.



(c) Select all the surfaces in the Tri Boundary Zones selection list.

(d) Click Apply in the Local Size Function group box.

(e) Click Refine.

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".

After refining the region, region:1 *is now much finer as seen in Figure 7.18.*

Step 6: Close Small Holes Automatically

In this step, you will retain and automatically fix all the potential holes for region:1*.*

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

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

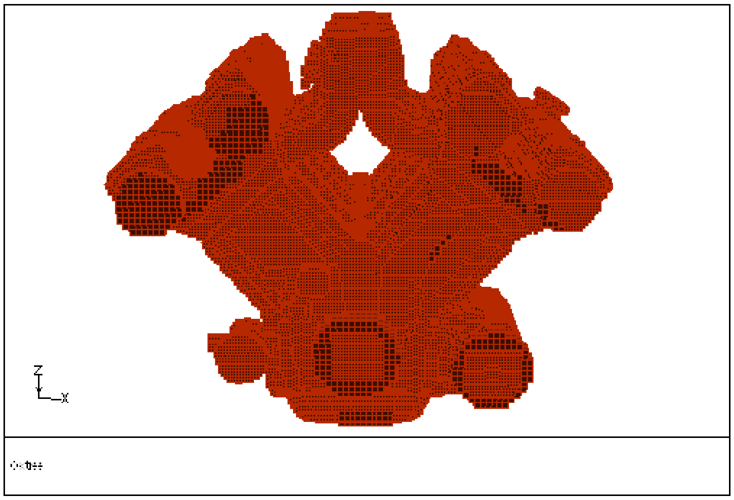
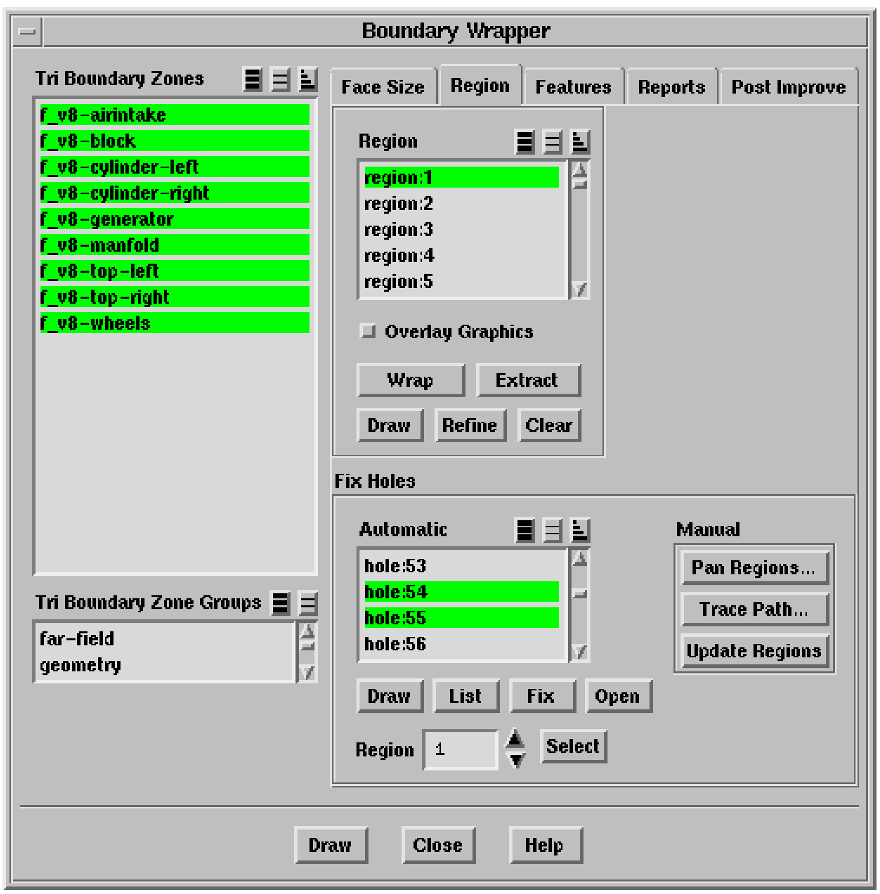


Figure 7.18: Region After Refining Using Local Size Function



2. Deselect the previously selected zones in the Face Zones selection list in the Faces tab of the Display Grid panel.

   Display ⟶Grid...

3. Click Display.

   This will allow you to display only the potential holes.

4. Click Draw in the Fix Holes group box in the Region tab of the Boundary Wrapper panel (Figure 7.19).

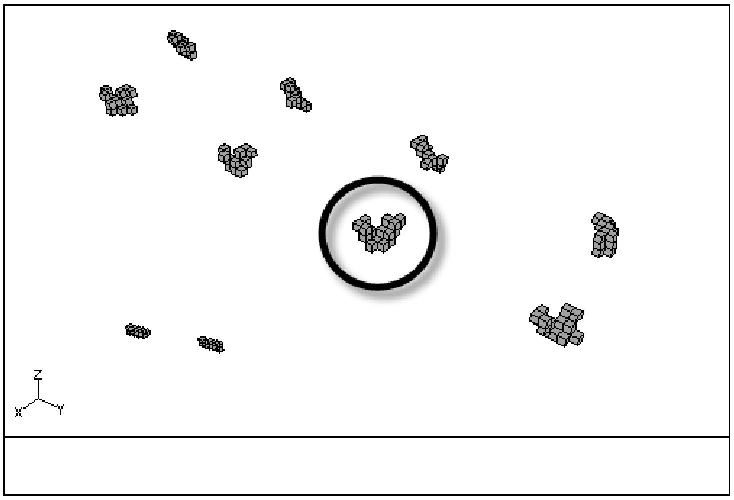


Figure 7.19: Potential Holes in region:1

5. Examine the potential holes.

   (a) Select position in the Filter list in the Modify Boundary panel (or use the hot-key Ctrl + X).

   Boundary ⟶Modify...

   (b) Click on the hole highlighted in Figure 7.19 using the right mouse button.

   The position of the hole will be added in the Object Name *field in the* Neighborhood *group box in the* Display Grid *panel.*

   (c) Enter 20 for +/- Delta in the Neighborhood group box in the Display Grid panel.

   (d) Click Set Ranges.

   The X Range, Y Range, *and* Z Range *fields will be updated accordingly.*

   (e) Select all the zones in the Face Zones selection list in the Faces tab of the Display Grid panel and click Display.

   (f) Click Draw in the Fix Holes group box in the Region tab of the Boundary Wrapper panel (Figure 7.20).

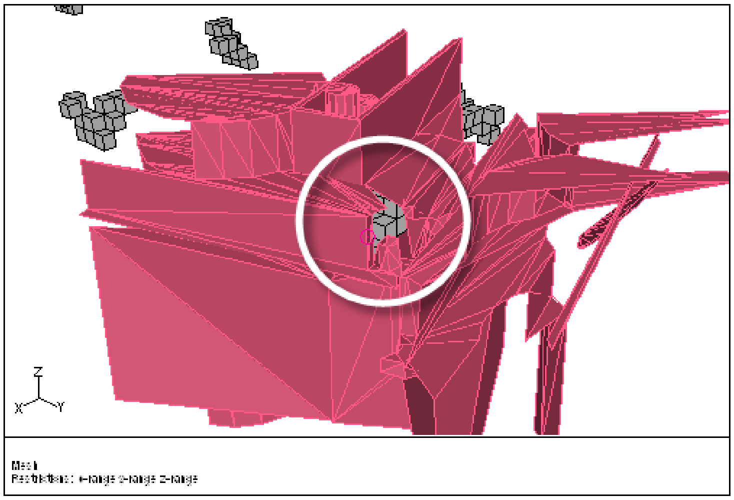


Figure 7.20: Bounds Set Around a Potential Hole

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 panel.

    (b) Click Fix in the Fix Holes group box in the Region tab in the Boundary Wrapper panel.

    (c) Select all the zones in the Face Zones selection list in the Display Grid panel.

    (d) Click Display (Figure 7.21).

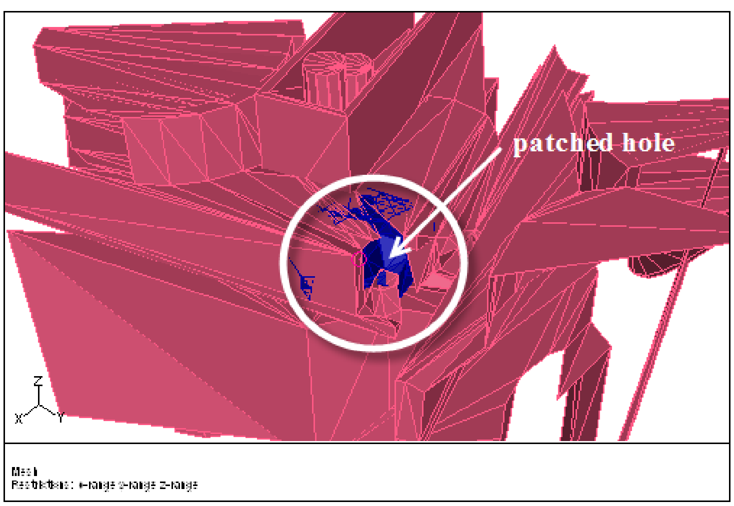


Figure 7.21: Hole Fixed Using the Automatic Hole Fixing Option

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.*

Step 7: Wrap the Main Region

1. Select region:1 in the Region list in the Region tab of the Boundary Wrapper panel.

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.

   TGrid *will create the wrapper surface for* region:1, wrapper-surf-#, *which will be available in the* Tri Boundary Zones *selection list.*

4. Display the wrapper surface (Figure 7.22).

   You may need to click Reset *in the* Bounds *tab of the* Display Grid *panel and manipulate the display in the graphics window to obtain the view shown in Figure 7.22.*

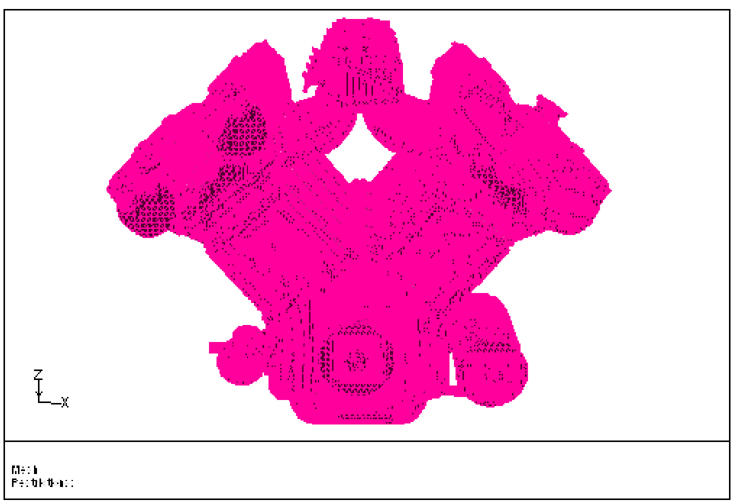


Figure 7.22: Wrapper Surface

5. Save the mesh file (`engine00.msh.gz`).

Step 8: Capture Features

1. Click the Features tab in the Boundary Wrapper panel.

2. 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.*

3. Click Extract Features.

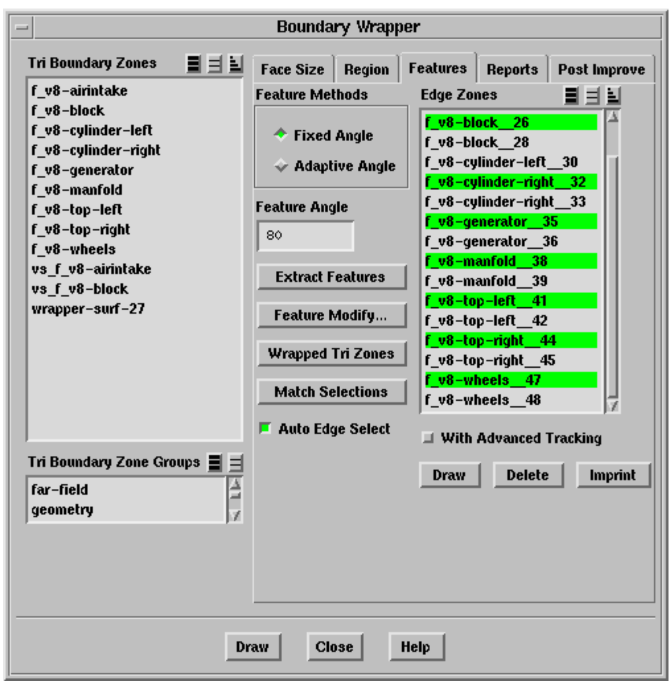
   The extracted features will now be available in the Edge Zones *selection list.*

4. 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 panel.



7. Click the Draw button below the Edge Zones selection list (Figure 7.23).

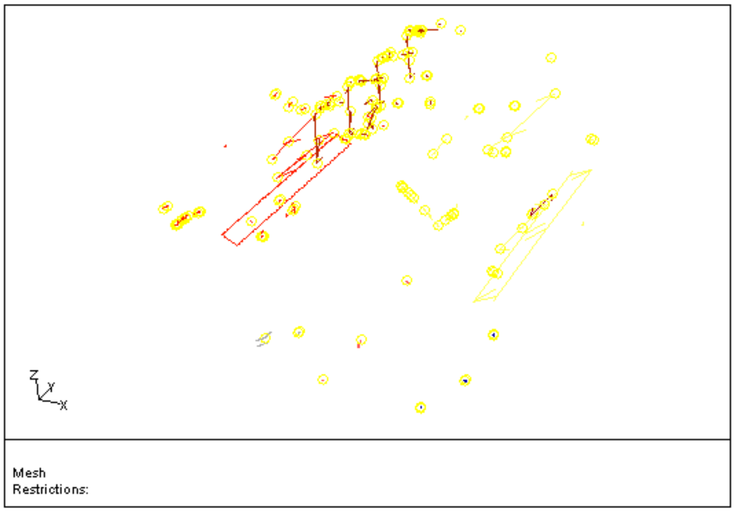


Figure 7.23: Insignificant Features to be Deleted

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.

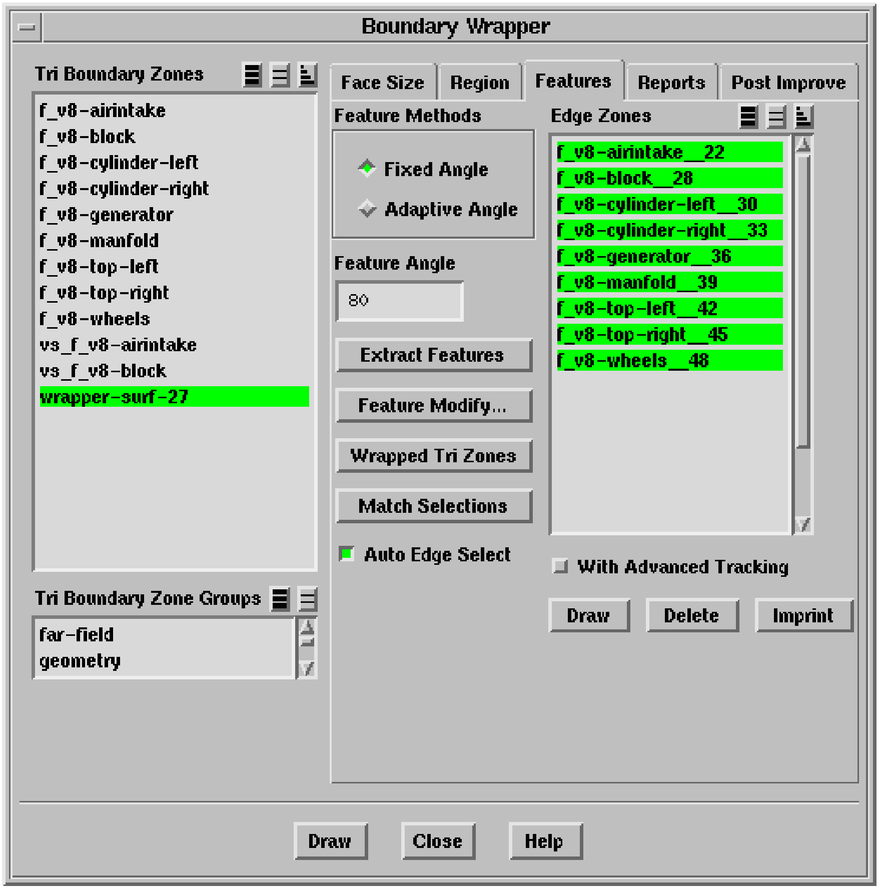
   Zoom in to the region shown in Figure 7.24.

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 panel and click Draw Contours.

    In this case, you will use the contours of wrapped surface quality to verify the imprinting of features.

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 panel.

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. Enable With Advanced Tracking and click Imprint again.

    Figure 7.24 shows the wrapper surface and contours of distance before and after imprinting.

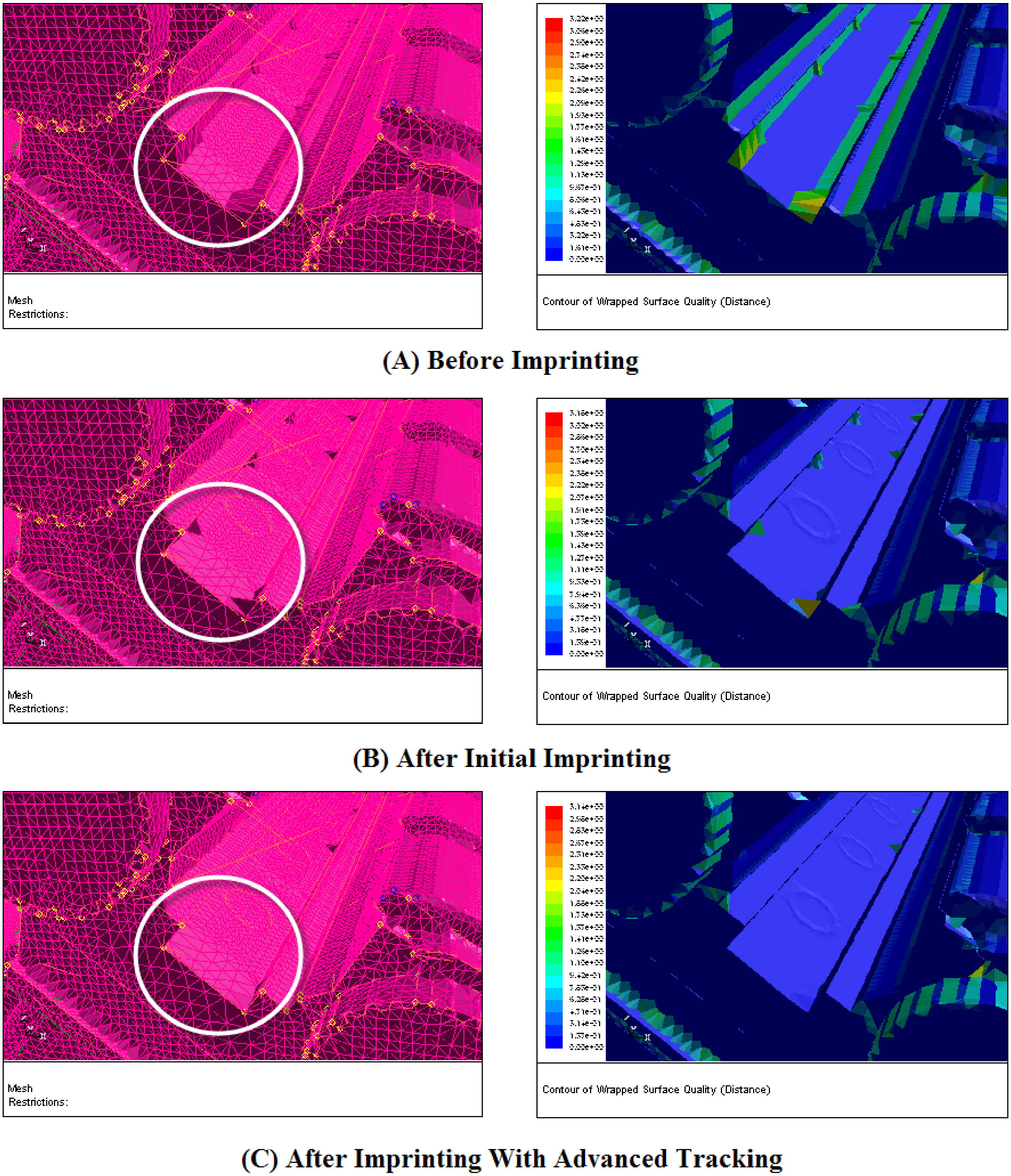
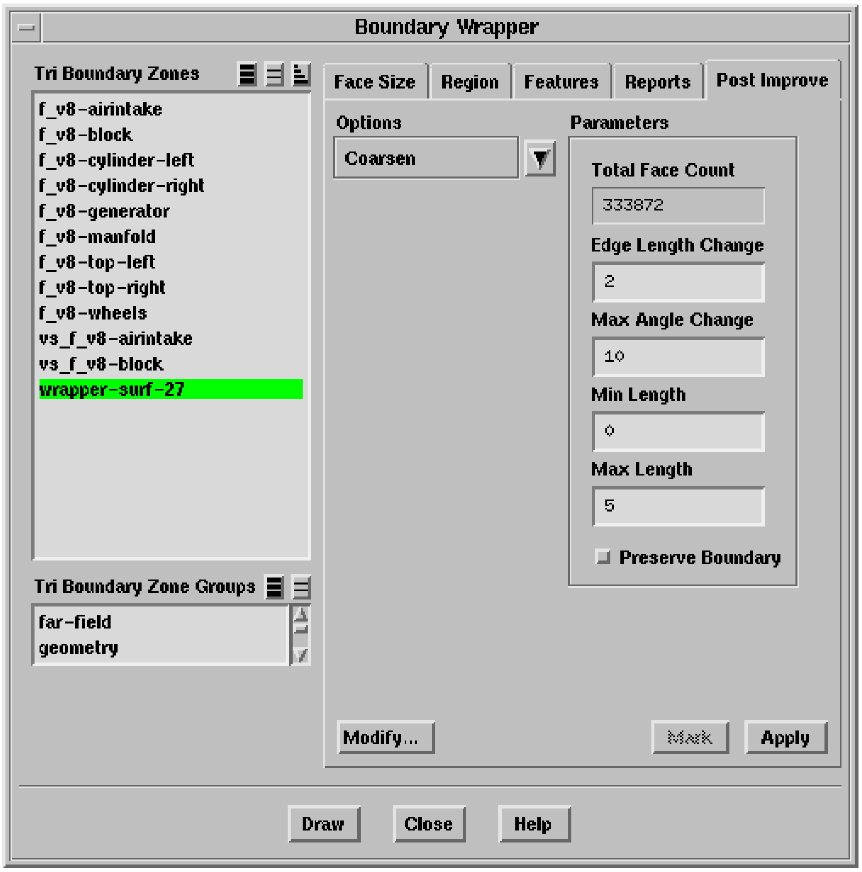


Figure 7.24: Wrapper Surface and Contours of Distance During Imprinting

Step 9: Post Wrapping Operations

1. Click the Post Improve tab of the Boundary Wrapper panel.

2. Coarsen the wrapper surface.



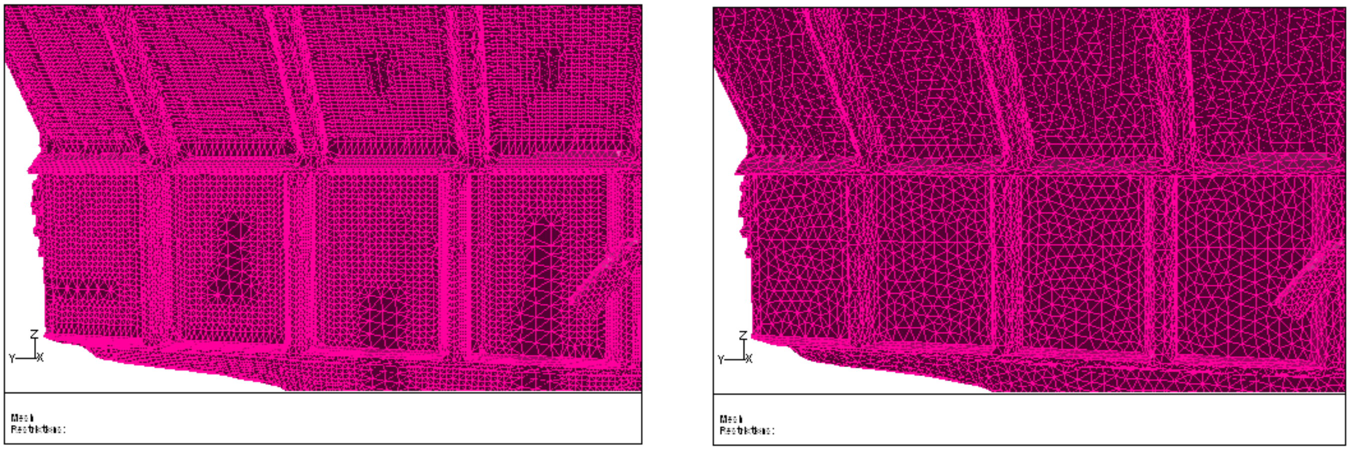
(a) Retain the selection of Coarsen in the Options drop-down list.

(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.

(e) Save the mesh (`engine-01.msh.gz`).



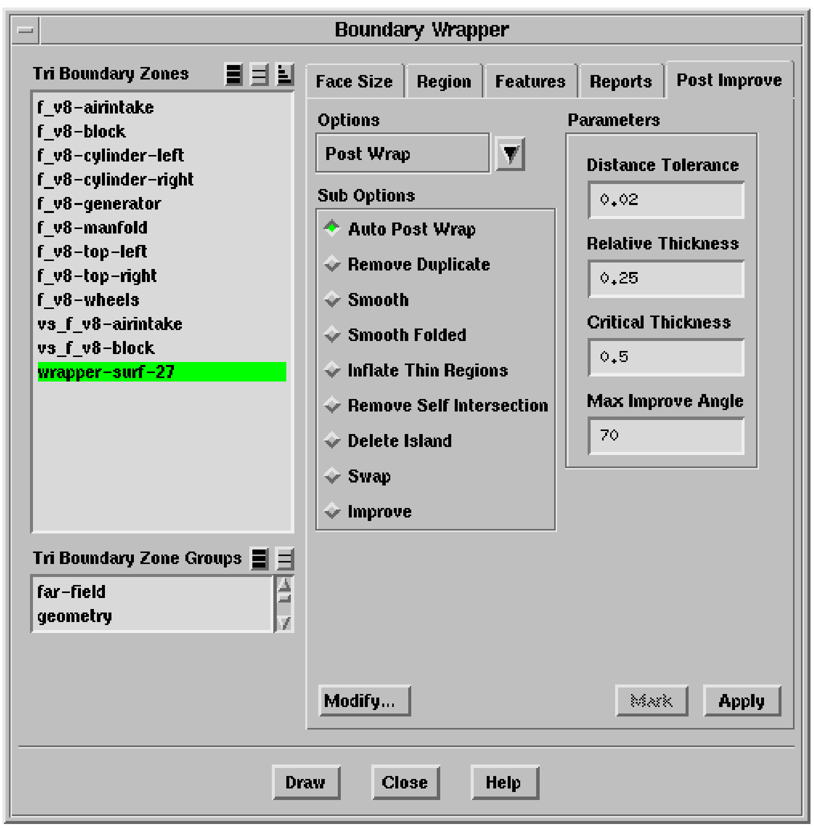
(A) (B)

Figure 7.25: Wrapper Surface (A) Before and (B) After Coarsening

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).

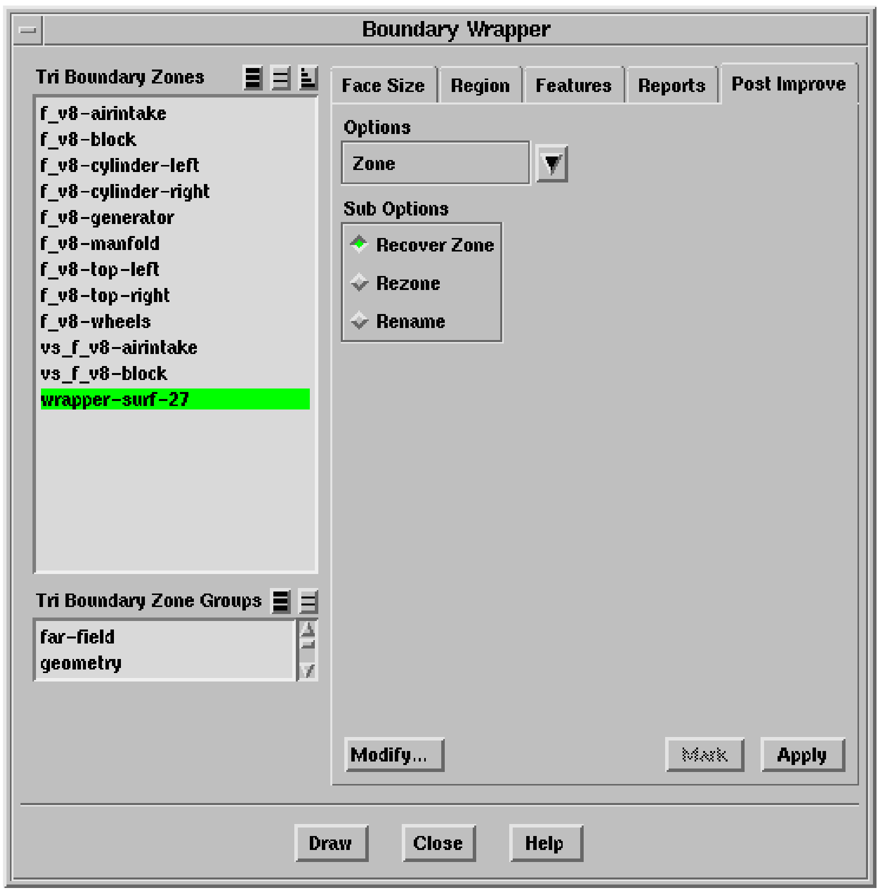
(d) Retain the default settings for the remaining parameters and click Apply.

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 7.26).

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-*.*

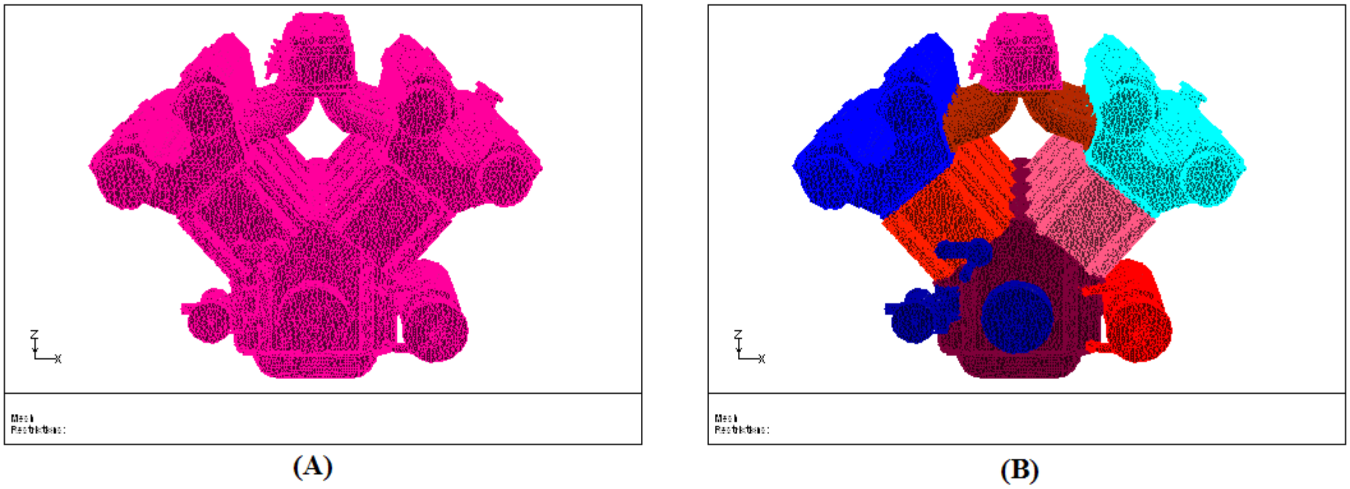
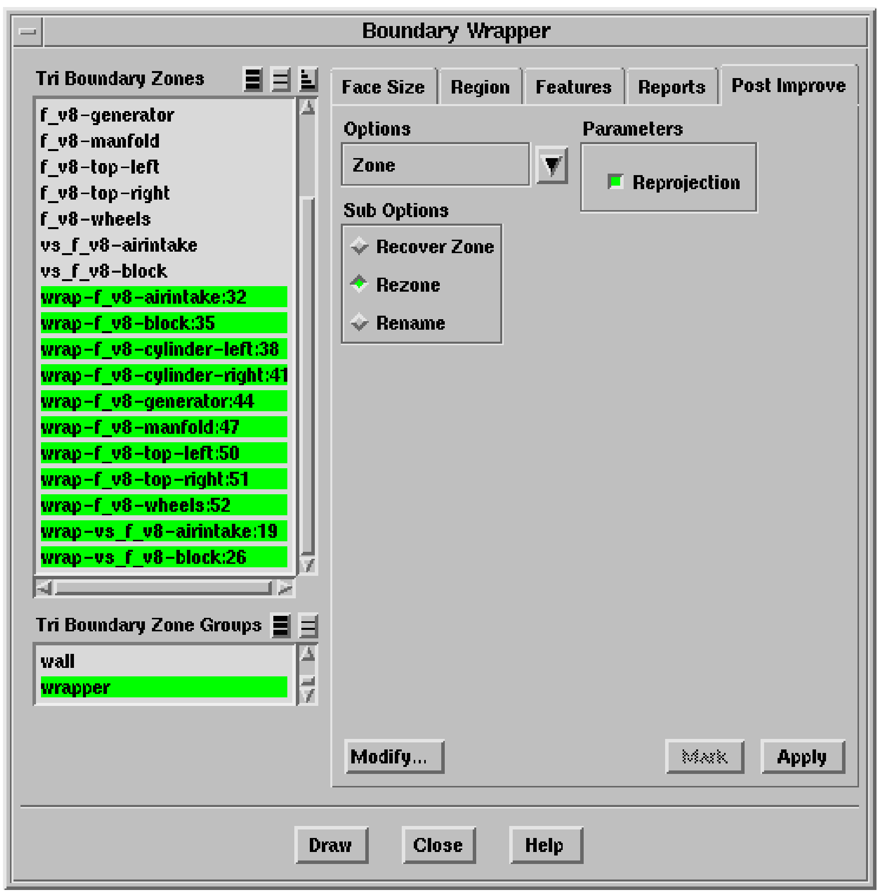


Figure 7.26: Wrapper Surface (A) Before and (B) After Recovering Zones

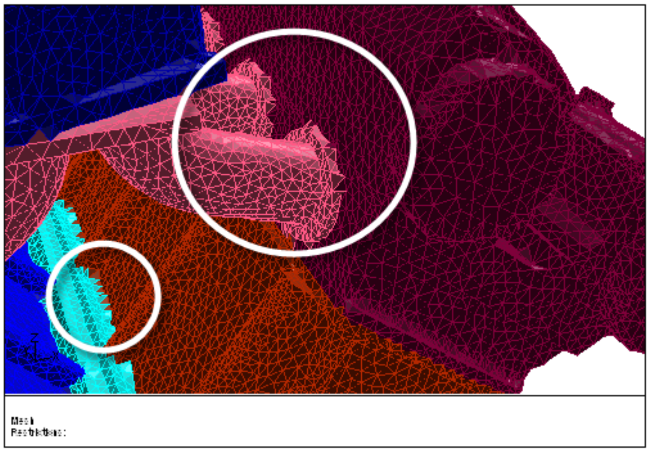
(d) Select the extracted wrapper surfaces in the Tri Boundary Zones selection list.



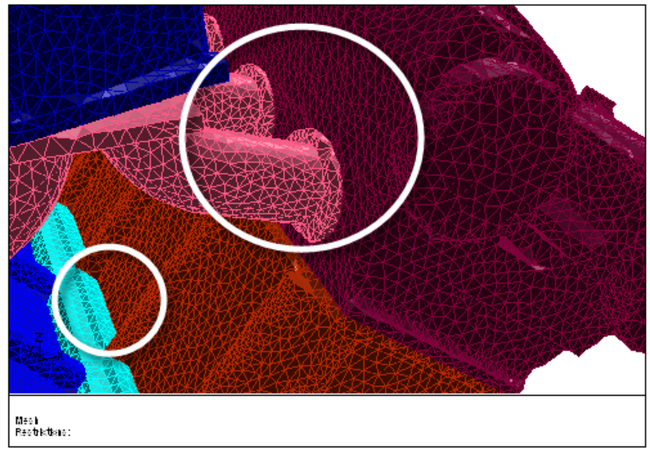
(e) Retain the selection of Zone in the Options drop-down list and select Rezone in the Sub Options list.

(f) Retain the Reprojection option in the Parameters group box.

(g) Click Apply.



(A)



(B)

Figure 7.27: 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:

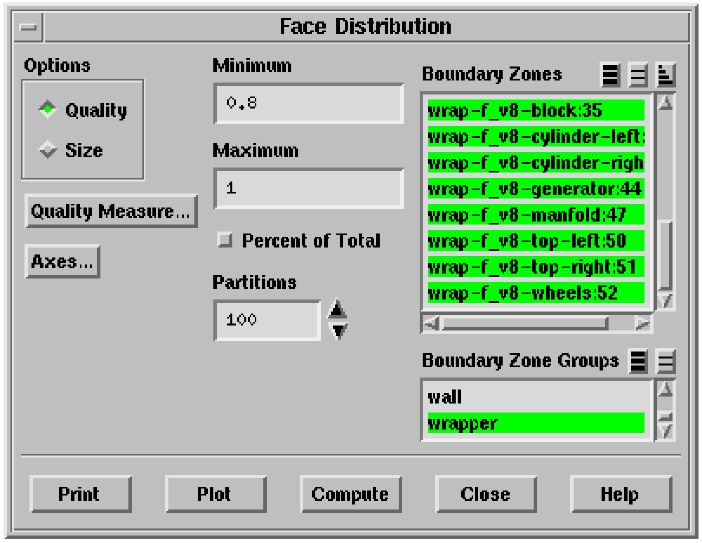
```
> boundary/merge-small-face-zones
Minimum area [0.01] 500

Merged wrap-vs_f_v8-block:# with v8-block:#
Merged wrap-vs_f_v8-airintake:# with v8-airintake:#
Merged 2 zones
```

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

Display ⟶ Plot ⟶Face Distribution...

(a) Select wrapper in the Boundary Zone Groups selection list to select all the wrapper zones in the Boundary Zones selection list.

(b) Enter 0.8 for Minimum.



(c) Click Plot (Figure 7.28).

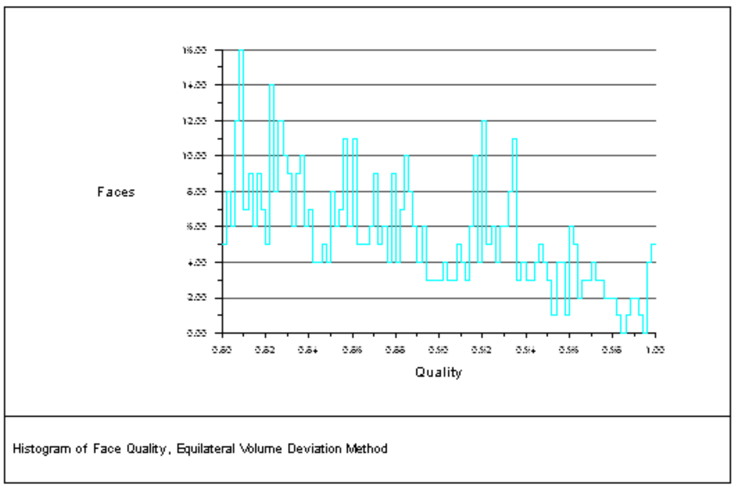
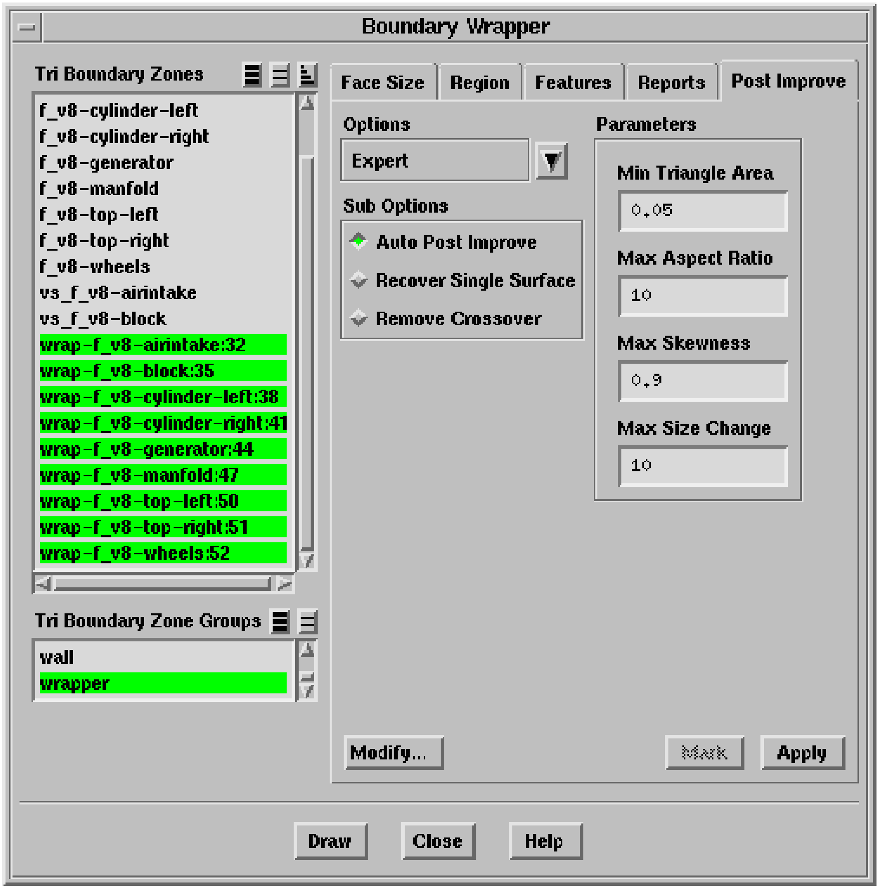


Figure 7.28: Face Skewness Distribution Between 0.8 and 1.0 (Before Auto Post Improve)

(d) Close the Face Distribution panel.

8. Improve the wrapper surfaces using the Auto Post Improve option.

   (a) Select Expert in the Options drop-down list and Auto Post Improve in the Sub Options list.



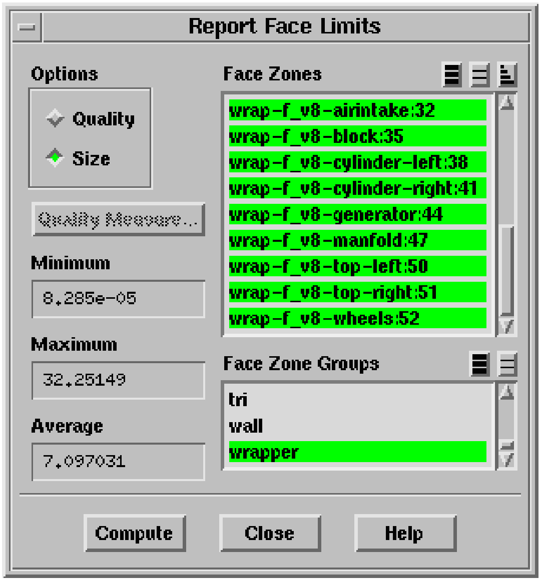
(b) Enter `0.05` for Min Triangle Area.

Note: *All cells 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...

i. Select wrapper *in the* Face Zone Groups *list to select all the wrapper zones.*

ii. Select Size *in the* Options *list.*



iii. *Click* Compute.

iv. *Close the* Report Face Limits *panel.*

The value specified for Min Triangle Area *is approximately 10 times the minimum size reported.*

(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 0.9 to 1.0 (Figure 7.29).

Display ⟶ Plot ⟶ Face Distribution...

In Figure 7.29, you can see that there are only four faces above 0.9.

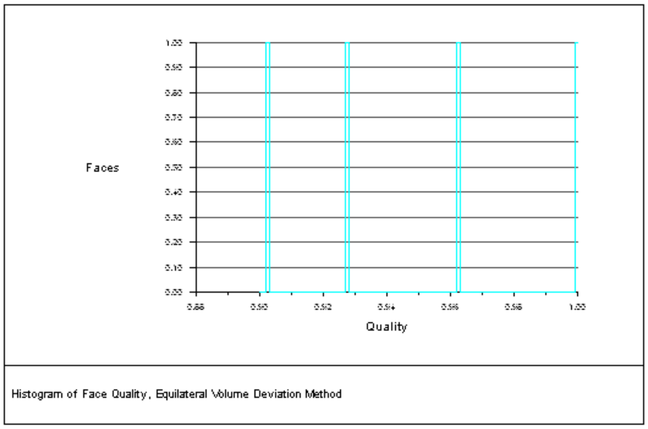


Figure 7.29: Face Skewness Distribution Between 0.9 and 1.0 (After First Auto Post Improve)

(f) Enter `0.1` for Min Triangle Area and `0.8` for Max Skewness, respectively.

(g) Click Apply.

(h) Plot the face skewness distribution in the range 0.8 to 1.0 (Figure 7.30).

Display ⟶ Plot ⟶ Face Distribution...

In Figure 7.30, you can see that there is only one face above 0.8.

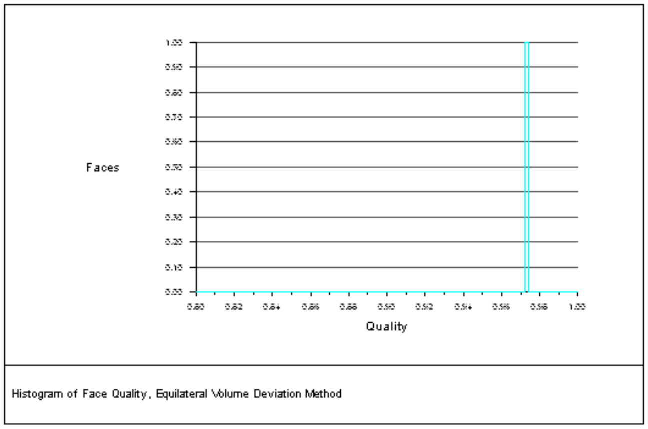


Figure 7.30: Face Skewness Distribution Between 0.8 and 1.0 (After Auto Post Improve)

9. Save the mesh (`engine-04.msh.gz`).

10. Delete the original geometry.

    Boundary ⟶ Manage...

    (a) Select the original geometry and patched holes in the Face Zones selection list.

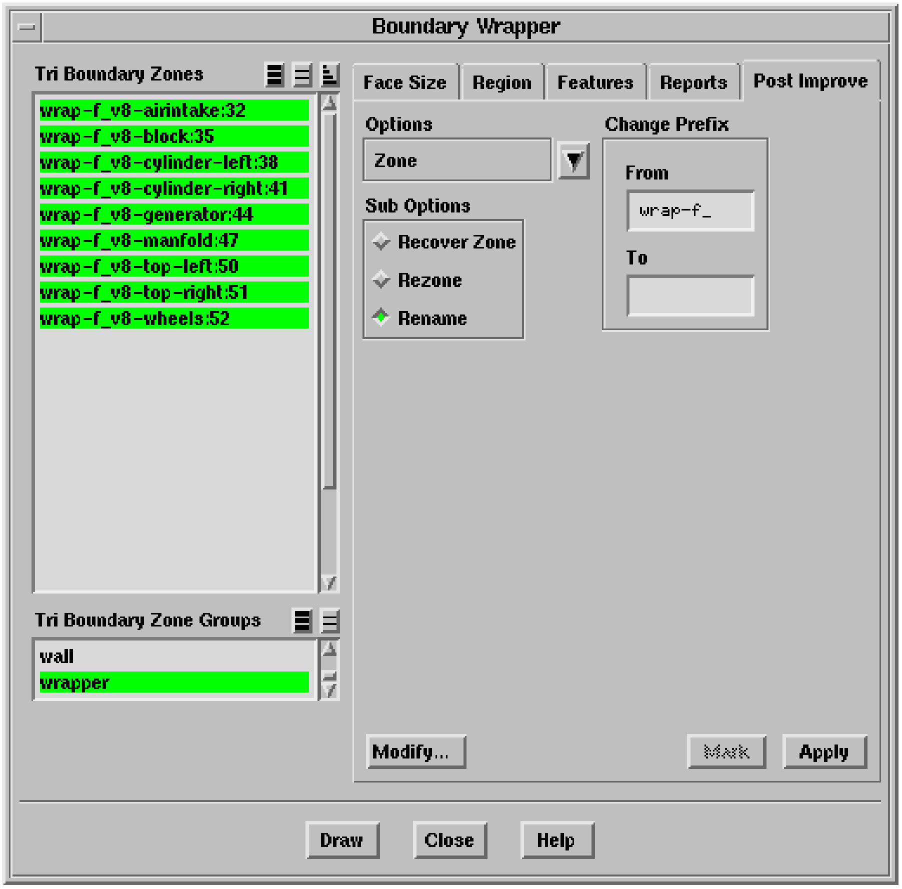
    (b) Select Delete in the Options list and click Apply.

    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 panel.

11. Rename the wrapper surfaces.



(a) Select the wrapper zones in the Tri Boundary Zones selection list in the Boundary Wrapper panel.

(b) Select Zone from the Options drop-down list and Rename in the Sub Options list.

(c) Enter `wrap-f_` in the From field.

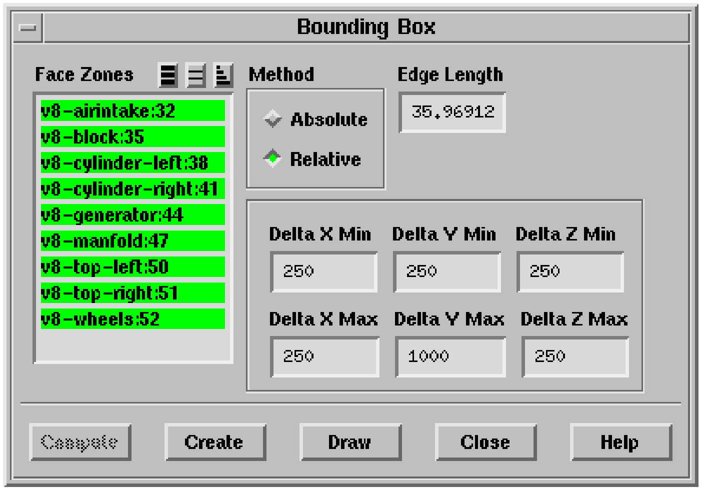
Leave the To *field blank so that the prefix will be removed.*

(d) Click Apply.

12. Close the Boundary Wrapper panel.

Step 11: 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 panel.
4. Click Create.

   The Zone Name *panel will open.*
5. Enter `tunnel` for Zone Name and click OK.
6. Close the Bounding Box panel.
7. Display the boundary mesh (Figure 7.31).

   Disable Filled *in the* Attributes *tab of the* Display Grid *panel and* Hidden Line Removal *in the* Display Options *panel to obtain the display shown in Figure 7.31.*

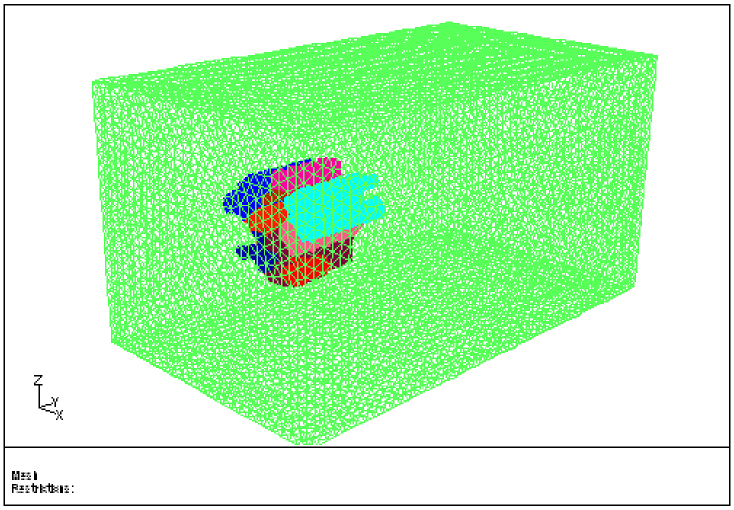
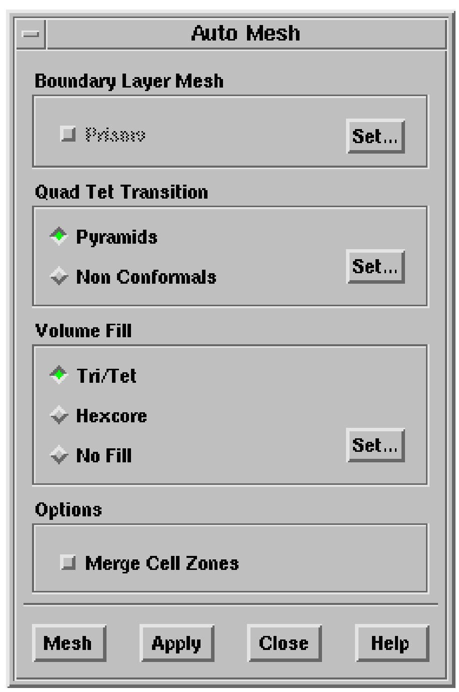


Figure 7.31: Mesh with the Tunnel

Step 12: Generate the Volume Mesh

Mesh ⟶Auto Mesh...



1. Retain the selection of Tri/Tet in the Volume Fill group box and click the Set... button to open the Tri/Tet panel.

   (a) Enable Delete Dead Zones in the Tri Tet Zones group box.

(b) Click Apply and close the Tri/Tet panel.

2. Click Mesh in the Auto Mesh panel.

3. Close the Auto Mesh panel.

4. Check the quality of the volume mesh.

   Display ⟶ Plot ⟶ Cell Distribution...

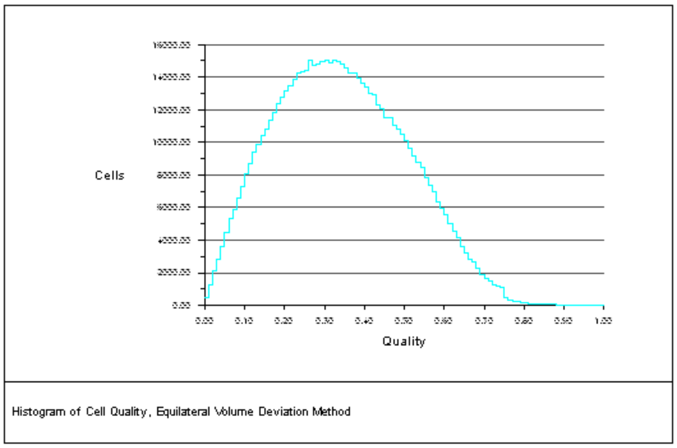


Figure 7.32: Cell Quality Distribution

5. Plot the skewness distribution above 0.95 (Figure 7.33).

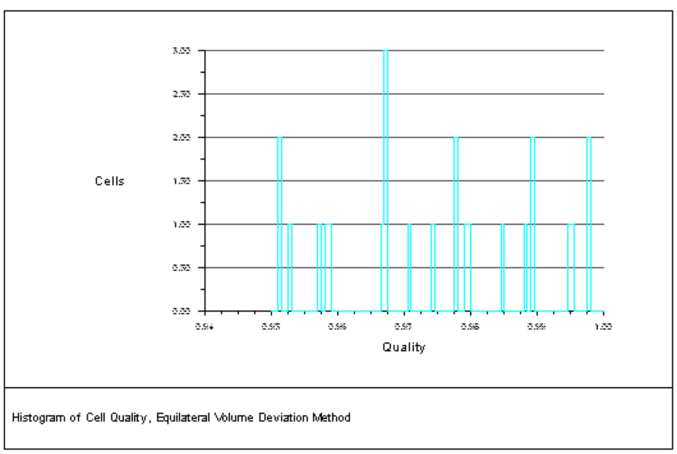
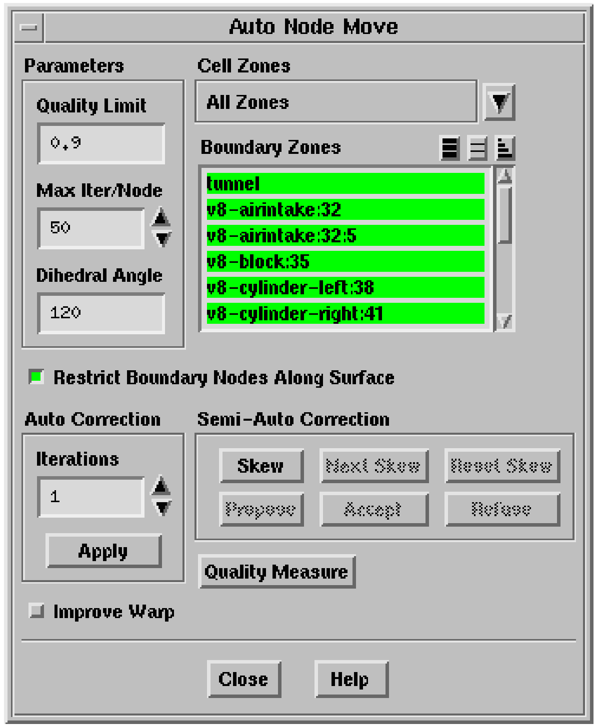


Figure 7.33: Cell Quality Distribution Above 0.95

Step 13: Improve the Volume Mesh

In this step, you will attempt to improve the volume mesh such that the maximum skewness is around 0.95.

Mesh ⟶ Tools ⟶Auto Node Move...

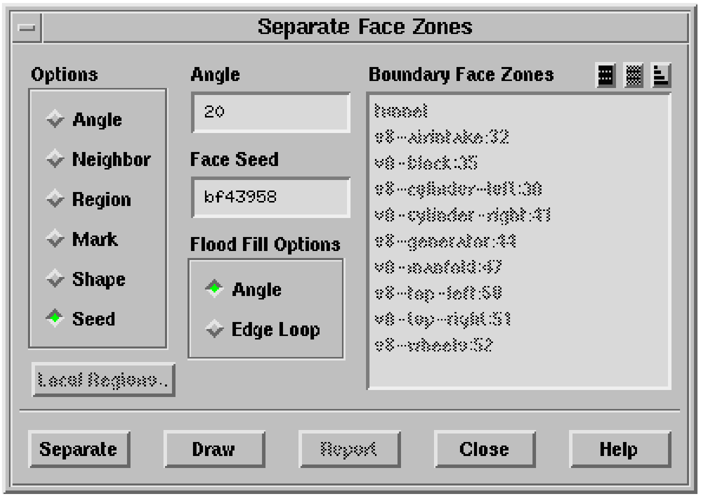


1. Select all the zones in the Boundary Zones selection list.
2. Click Apply in the Auto Correction group box.
3. Check the skewness distribution above 0.95.
4. Enter `0.95` for Quality Limit and `0` for Dihedral Angle, respectively.
5. Disable Restrict Boundary Nodes Along Surface.
6. Click Apply in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

   The maximum skewness reported is around 0.95, which is acceptable.
7. Close the Auto Node Move panel.

Step 14: 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 7.34.

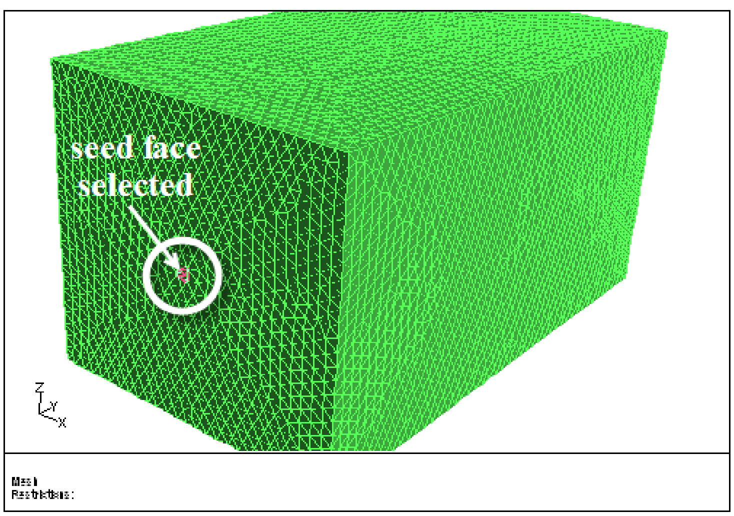


Figure 7.34: Seed Face for Separating the Tunnel Inlet

4. Click Separate.

   TGrid *will create the zone* tunnel-#, *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 TGrid.

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 demonstarted the use of the automatic hole fixing functionality to close small 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.

Tutorial 8. Cavity Remeshing

Introduction

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:

1. Create and remesh a cavity to replace the mirror in a tetrahedral mesh.
2. Create and remesh a cavity in a hybrid mesh (tetrahedra and prisms) having a single fluid zone.
3. Create and remesh a cavity in a hybrid mesh (tetrahedra and prisms) having multiple fluid zones.
4. Create and remesh a cavity in a hexcore mesh.

Prerequisites

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

Preparation

1. Download `cavity.zip` from the FLUENT User Services Center to your working directory. This file can be found from the Documentation link on the TGrid product page.

   OR

   Copy `cavity.zip` from the TGrid documentation CD to your working directory.

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

     cdrom`/tgrid5.0/help/tutfiles`

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

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

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

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

2. Unzip `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 the 3D (3d) version of TGrid.

Case A. For a Tetrahedral Mesh

Step 1: Read and Display Mesh

1. Read the mesh file.

   File ⟶ Read ⟶Mesh...

   (a) Select sedan_tetra.msh.gz in the Files list.

   (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 8.1).

   (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 Display.

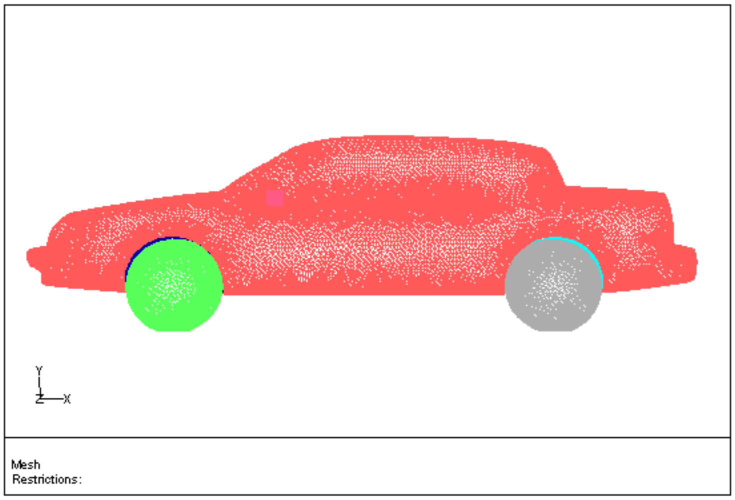


Figure 8.1: Boundary Mesh

(h) Display the left view.

Display ⟶Views...

i. Select left in the Views list.

ii. Click Apply and close the Views panel.

(i) Zoom in and examine the mesh around the mirror (Figure 8.2).

The mesh comprises tetrahedral cells.

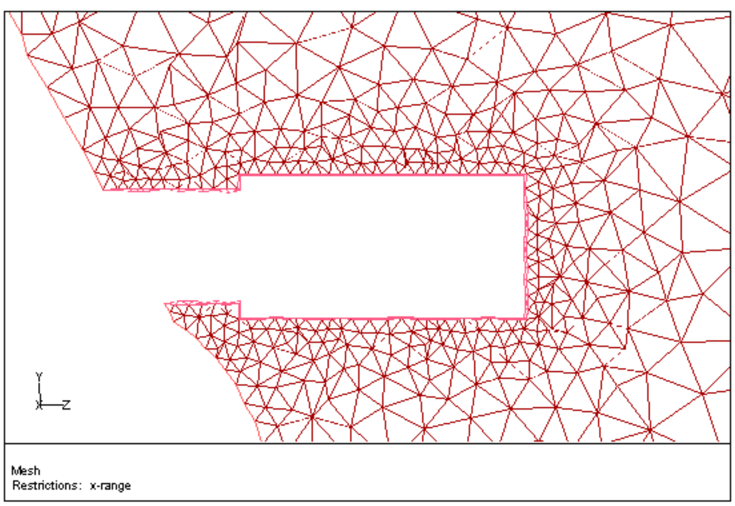


Figure 8.2: Tetrahedral Mesh Near the Mirror

Step 2: Import and Connect the New Mirror

*In this step, you will replace 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) Select mirror.msh.gz in the Files list.

   (b) Enable Append File(s) 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 8.3).

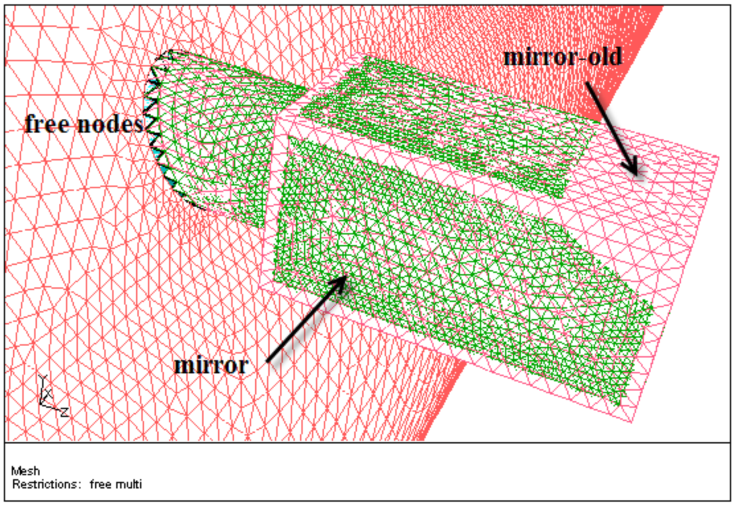
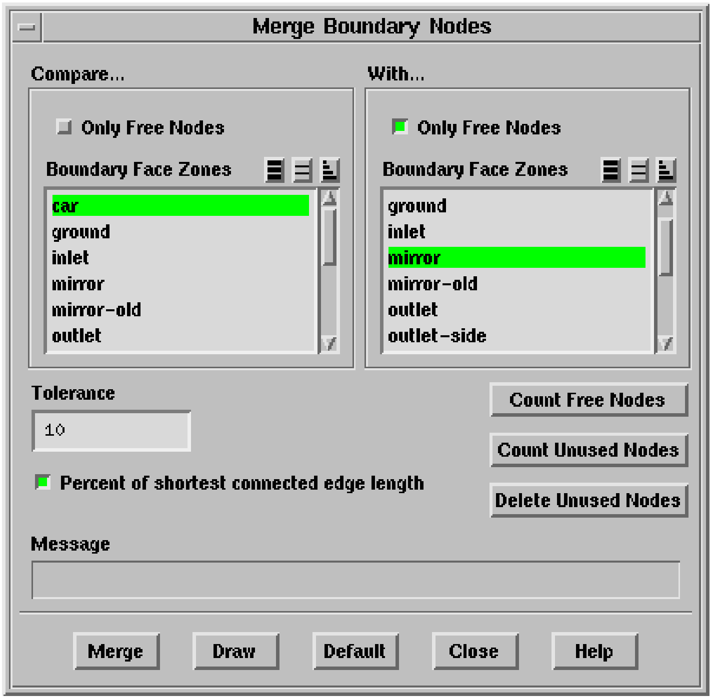


Figure 8.3: Free Nodes on the Mirror

In Figure 8.3, 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 panel.

   In Figure 8.4, you can see that there is a multiple connection between the mirrors and the car.

   (f) Close the Merge Boundary Nodes panel.

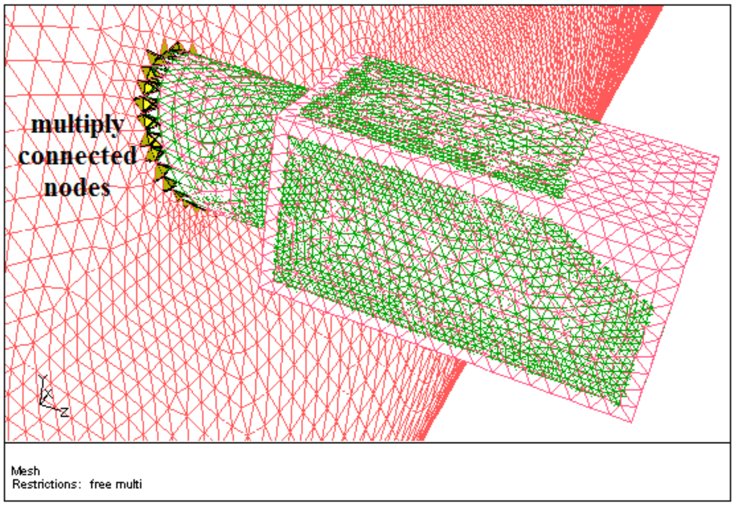


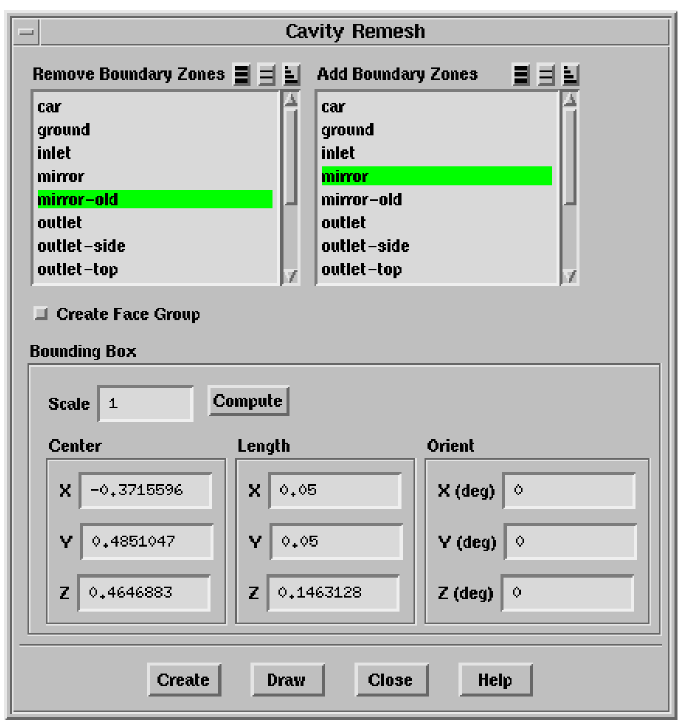
Figure 8.4: Multiply Connected Nodes on the Mirror

Step 3: Replace the Mirror

In this step, you will replace 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) Retain the value of 1 for Scale.

(d) Click Compute.

(e) Click Draw to verify that the cavity defined fits the mirror to be replaced.

(f) Click Create.

TGrid *separates the car zone into two—one inside (*car:#*) and the other outside the cavity region, creates an interior surface (*interior-#:#*) at the boundary of the cavity, and defines and activates a domain for the cavity.*

(g) Select car:# and mirror in the Face Zones list in the Display Grid panel.

(h) Click Display and close the Display Grid panel.

(i) Click Draw in the Cavity Remesh panel.

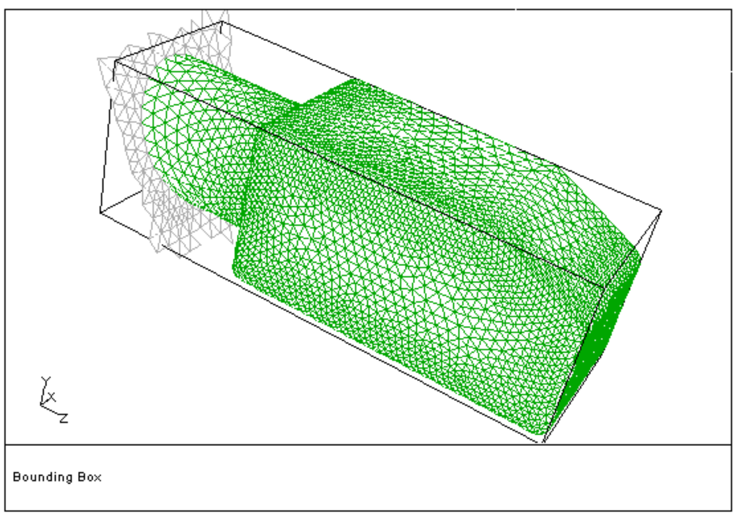


Figure 8.5: Cavity Domain Before Meshing

(j) Close the Cavity Remesh panel.

2. Mesh the cavity with tetrahedral cells.

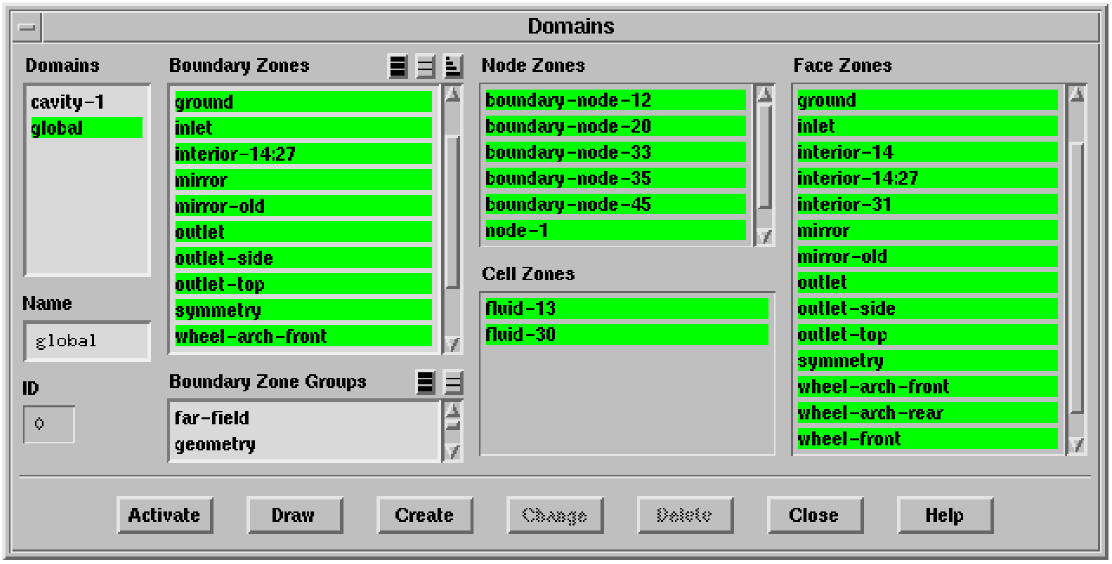
$\boxed{\text{Mesh}}\longrightarrow$Tri/Tet...

(a) Retain the default settings and click Init&Refine.

(b) Close the Tri/Tet panel.

3. Activate the global domain.

   Mesh ⟶Domains...



   (a) Select global in the Domains list.

   (b) Click Activate and close the Domains panel.

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 panel.

   (b) Disable Free and Multi in the Options group box.

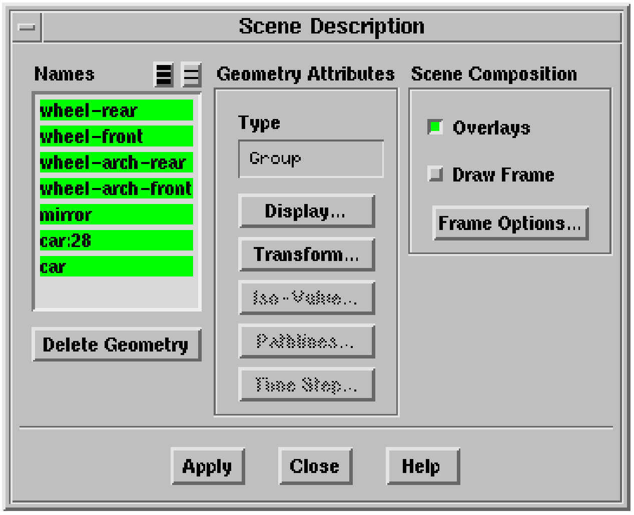
   (c) Click Display.

   (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 panel.

(e) Click the Cells tab in the Display Grid panel 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 panel (Figure 8.6).

Mesh ⟶ Tools ⟶ Cavity Remesh...

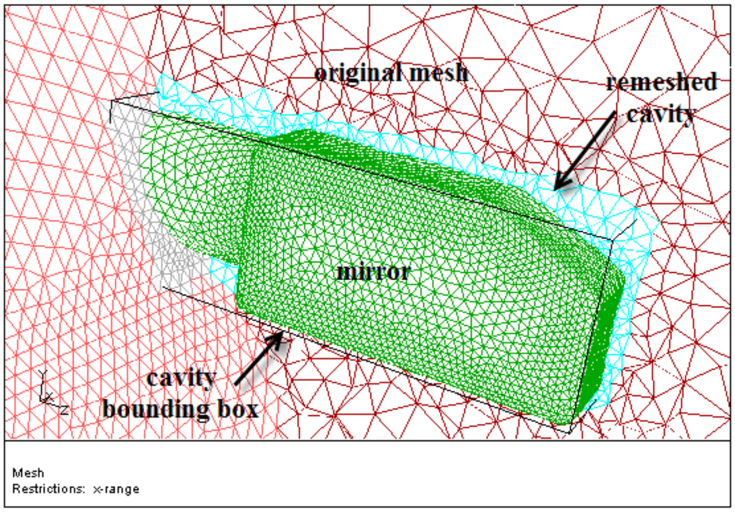
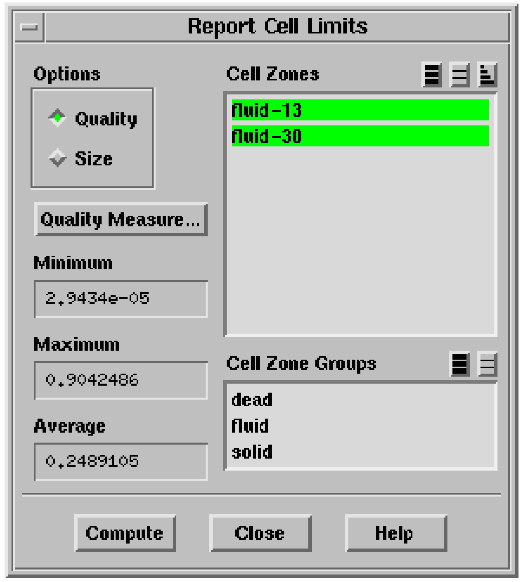


Figure 8.6: Cavity Remeshed With Tetrahedral Cells

5. Check the mesh quality.

   Report ⟶Cell Limits...



   (a) Select all zones in the Cell Zones selection list.

   (b) Click Compute.

   The maximum skewness reported is around 0.904. The exact value may vary slightly on different platforms.

6. Merge the cavity domain with the original mesh.

```
> /mesh/cavity/merge-cavity
Insert domain name/id [cavity-1] cavity-1 <Enter>
Into domain name/id [global] global <Enter>
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 appear, asking you to confirm if you want to delete the selected zone(s).*

(d) Click Yes in the Question dialog box.

(e) Close the Manage Face Zones panel.

8. Check the mesh.

   Mesh ⟶Check

9. Save the mesh (sedan-tet-cavity.msh.gz).

   File ⟶ Write ⟶Mesh...

Case B. 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 Display.

   (f) Display the left view.

   Display ⟶Views...

   (g) Zoom in and examine the mesh around the mirror (Figure 8.7).

3. Import and connect the mirror.

   (a) Read the mesh file mirror.msh.gz with the Append File(s) option enabled.

   File ⟶ Read ⟶Mesh...

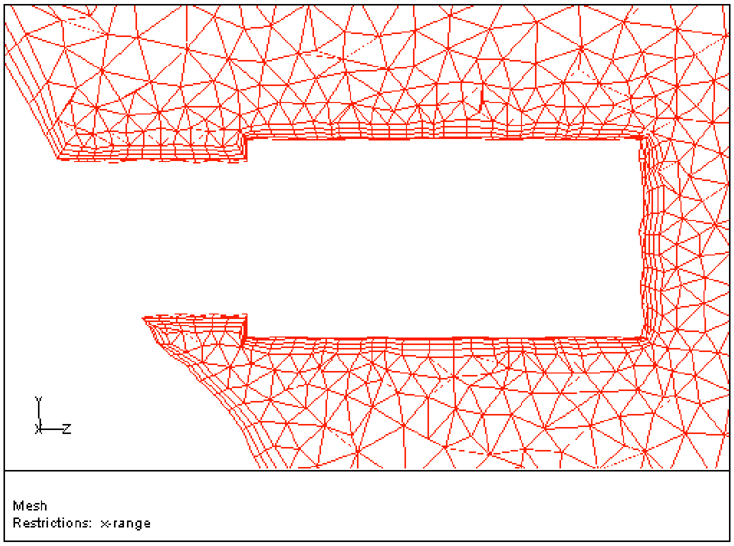


Figure 8.7: Hybrid Mesh Near the Mirror

(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 panel.

  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.

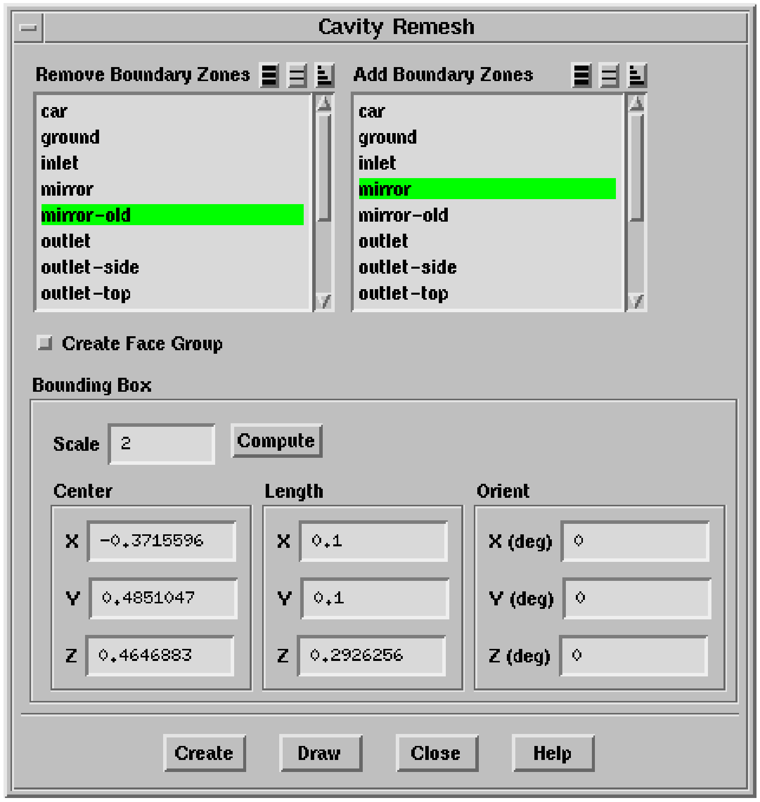
  v. Click Display in the Display Grid panel.

  Both mirrors are now connected to the car.

  vi. Close the Merge Boundary Nodes panel.

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.

   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.

   TGrid *separates the car zone into two—one inside (*car:#*) and the other outside the cavity region, creates an interior surface (*interior:#:#*) at the boundary of the cavity, and defines and activates a domain for the cavity.*

(h) Select all the zones in the Cavity Remesh panel.

(i) Click Draw to view the domain (Figure 8.8).

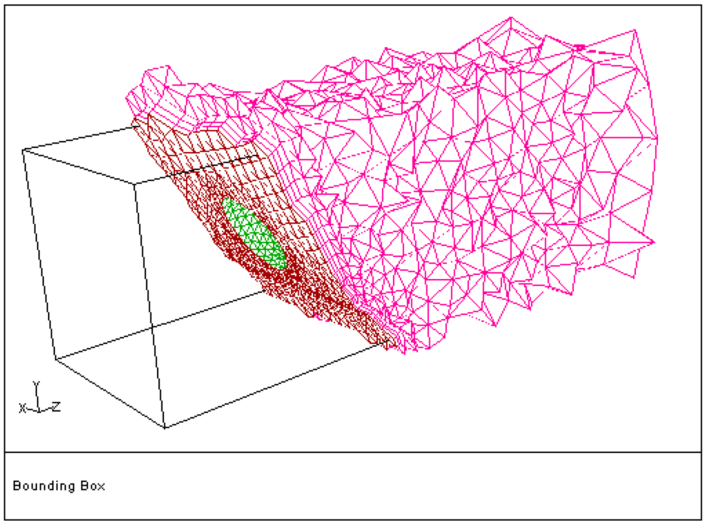


Figure 8.8: Cavity Domain Before Meshing

Zoom in to the interior zone (as shown in Figure 8.9), to see the quad faces. When you remesh the cavity, the prisms created during the remeshing will connect to these quad faces.

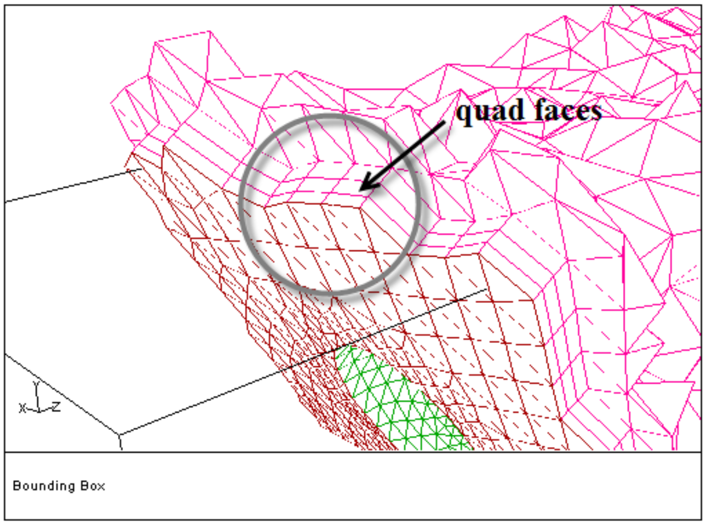
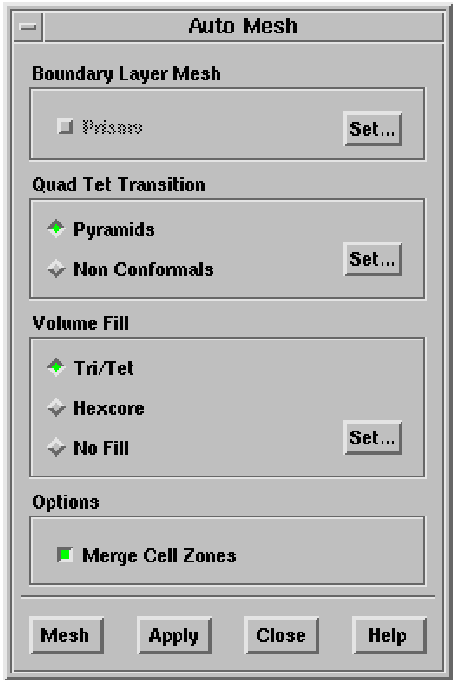


Figure 8.9: Quad Faces in the Interior

(j) Close the Cavity Remesh panel.

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.

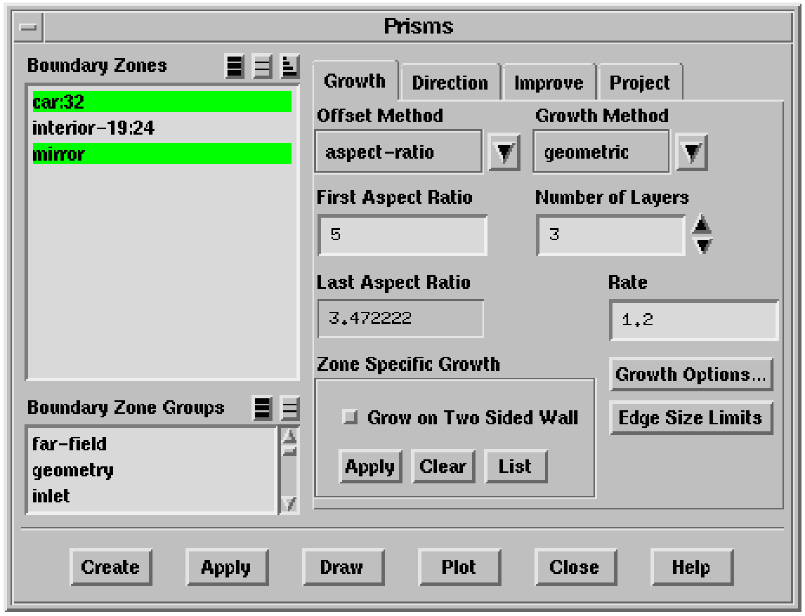
            The normals point outward and hence, are correctly oriented.

         D. Disable Normals and close the Display Grid panel.

      ii. Click the Set... button in the Boundary Layer Mesh group box in the Auto Mesh panel to open the Prisms panel.

      iii. Select car:# and mirror in the Boundary Zones selection list.

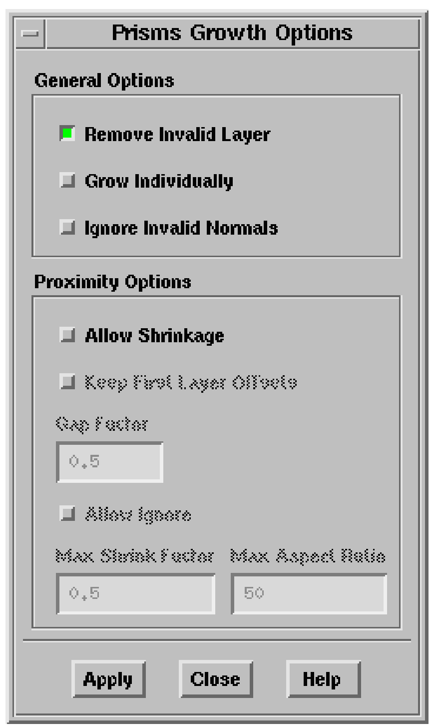
      iv. Select aspect-ratio in the Offset Method drop-down list and enter 5 for First Aspect Ratio.



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 panel.



A. Make sure that Allow Shrinkage is disabled in the Proximity Options group box.

B. Click Apply and close the Prisms Growth Options panel.

viii. Click Apply in the Zone Specific Growth group box.

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

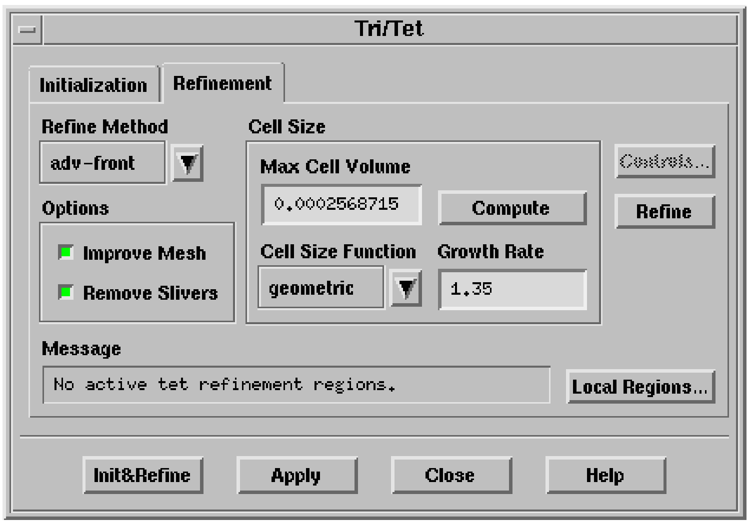
ix. Close the Prisms panel.

x. Enable Prisms in the Auto Mesh panel.

(b) Specify the tetrahedral meshing parameters.

i. Retain the selection of Tri/Tet click the Set.... button in the Volume Fill group box to open the Tri/Tet panel.

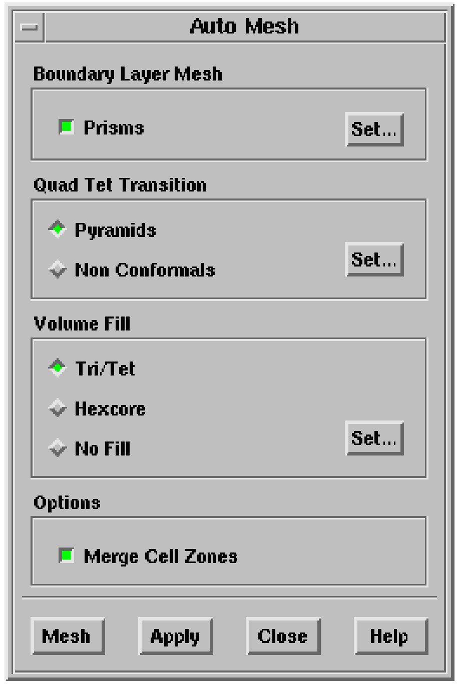
A. Click the Refinement tab and retain the selection of adv-front in the Refine Method drop-down list.



B. Retain the default value for Max Cell Volume.

C. Select geometric in the Cell Size Function drop-down list and enter `1.35` for Growth Rate.

D. Click Apply and close the Tri/Tet panel.



(c) Enable Merge Cell Zones.

(d) Click Mesh.

TGrid *will generate prism layers around the mirror and the car according to the defined parameters, automatically create a domain and fill it with tetrahedra, and finally merge the zones together.*

(e) Close the Auto Mesh panel.

(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 8.10).

In Figure 8.10, 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 panel.

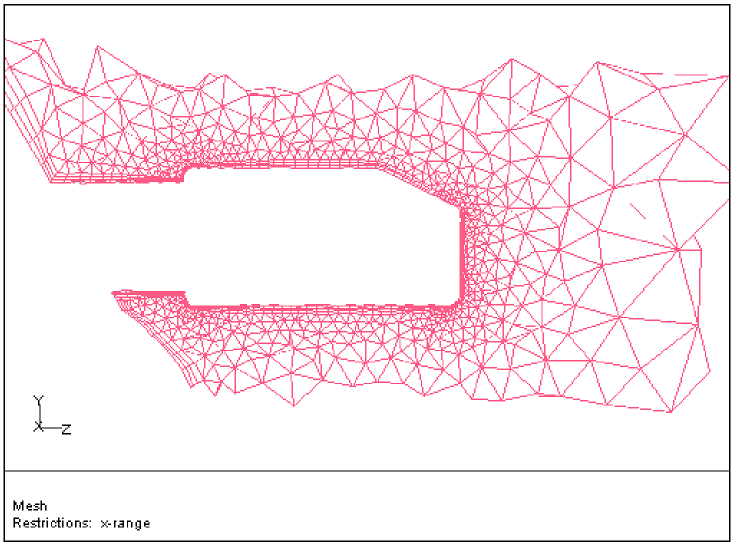


Figure 8.10: Remeshed Cavity

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 panel.

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 Compostition group box.

F. Click Apply and close the Scene Description panel.

G. Select the fluid zones in the Cell Zones selection list in the Cells tab of the Display Grid panel.

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 panel (Figure 8.11).

Mesh ⟶ Tools ⟶Cavity Remesh...

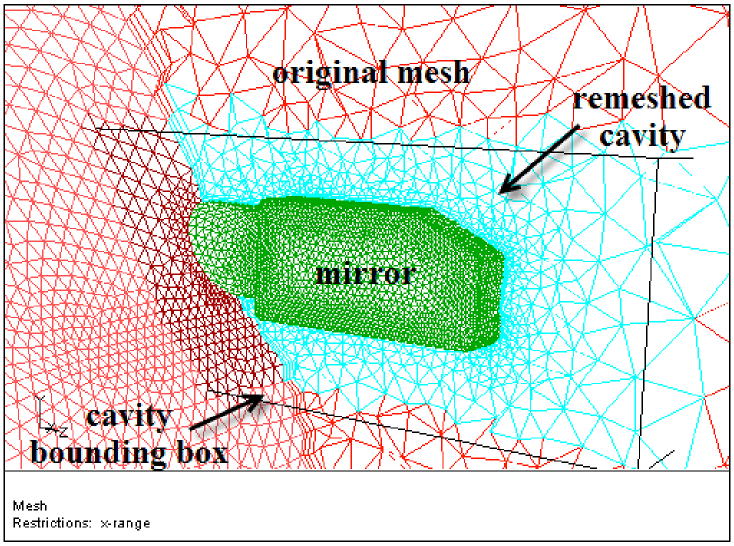


Figure 8.11: Cavity Remeshed With Hybrid Mesh

In Figure 8.10, 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 <Enter>
Into domain name/id [cavity-2] global <Enter>
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 panel.

8. Check the mesh quality.

   Report ⟶Cell Limits...

   The maximum skewness reported is around 0.92. 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...

Case C. 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 semi-colon (;) 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-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/tritet/controls/refine-method adv-front
/mesh/tritet/controls/cell-size-function geometric 1.35
/mesh/auto-mesh yes pyramids tritet 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 TGrid session.

3. Read the journal file `sedan_hyb-2zones-cavity.jou`.

   File ⟶ Read ⟶ Journal...

   Figures 8.12—8.14 show the mesh at intermediate stages.

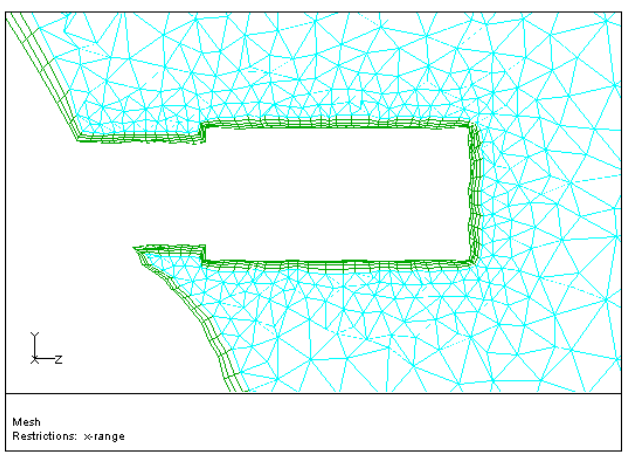


Figure 8.12: Hybrid Mesh Near the Mirror

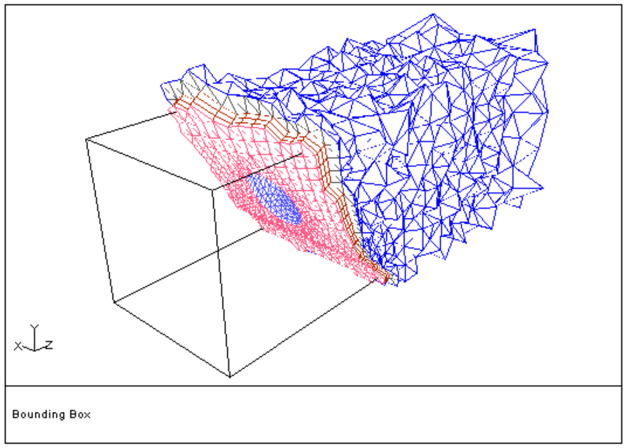


Figure 8.13: Cavity Domain Before Meshing

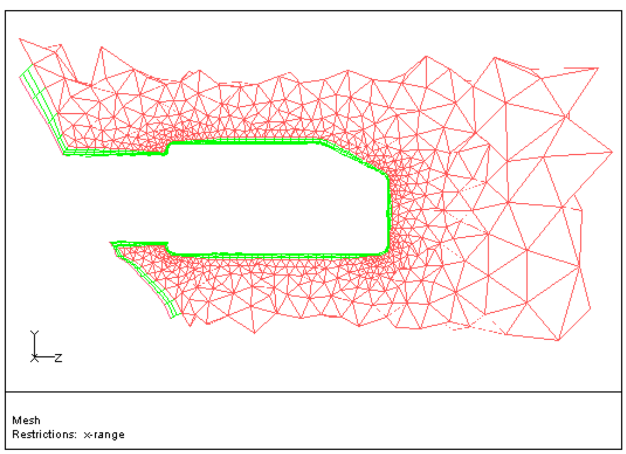


Figure 8.14: Remeshed Cavity

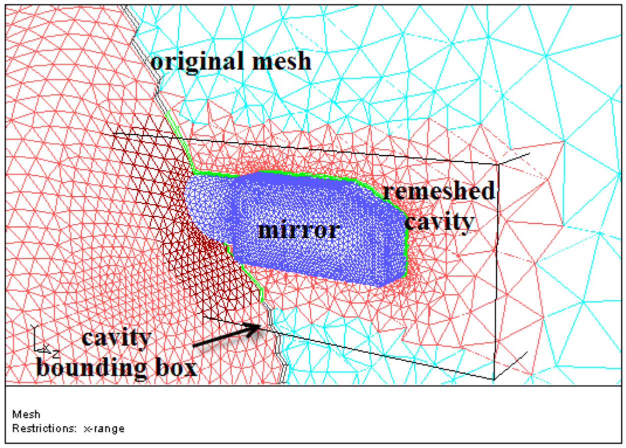


Figure 8.15: Cavity Remeshed With Hybrid Mesh

Case D. For a Hexcore Mesh

The steps in this section are similar to those described in previous sections, and hence are 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 Display.

   (d) Display the left view.

   Display ⟶ Views...

   (e) Zoom in and examine the mesh around the mirror (Figure 8.16).

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.

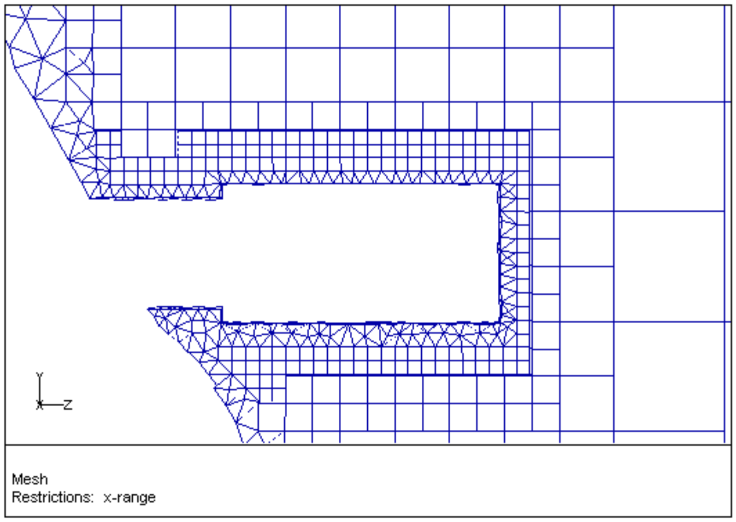


Figure 8.16: Hexcore Mesh Near the Mirror

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 panel.

5. Replace the old mirror with the new one.

   Mesh ⟶ Tools ⟶Cavity Remesh...

   Only the tetrahedral cells of the Hexcore mesh will be deleted. Hence, the new geometry must not be larger than the original one.

   (a) Select mirror-old in the Remove Boundary Zones selection list and mirror in the Add Boundary Zones selection list, respectively.

   (b) Enter `1.5` for Scale and click Compute.

   (c) Click Create.

   (d) Select all the zones in the Cavity Remesh panel and click Draw to display the cavity boundaries (Figure 8.17).

   (e) Close the Cavity Remesh panel.

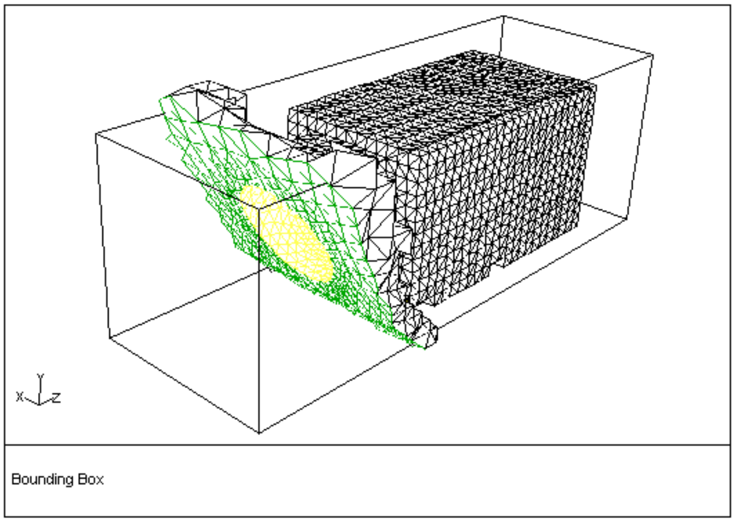
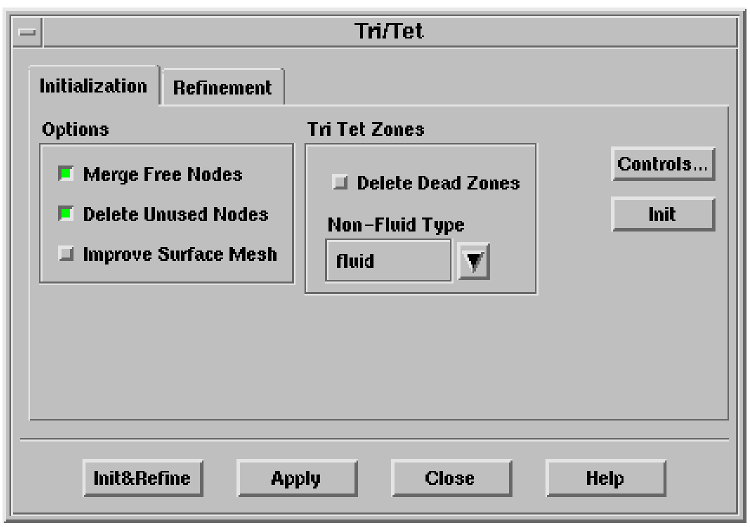


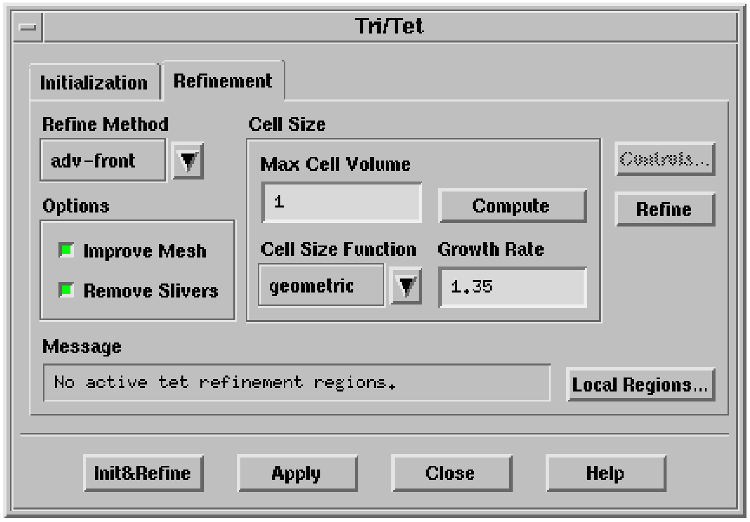
Figure 8.17: Cavity Domain Before Meshing

6. Mesh the cavity using tetrahedra.

   Mesh ⟶Tri/Tet...



   (a) Select fluid in the Non-Fluid Type drop-down list in the Tri Tet Zones group box in the Initialization tab.



(b) Click the Refinement tab and enter 1.35 for Growth Rate.

(c) Click Init&Refine.

(d) Close the Tri/Tet panel.

7. Check the mesh quality.

   Report ⟶Cell Limits...

   The maximum and average skewness values are approximately 0.817 and 0.308, respectively.

8. Examine the remeshed cavity by displaying the cells in the plane x = -0.37 (Figure 8.18).

   Display ⟶Grid...

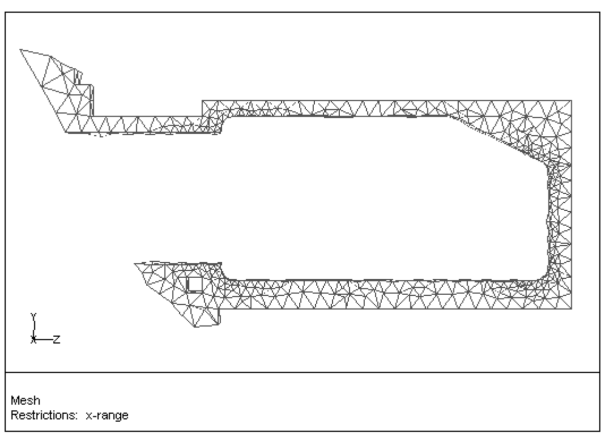


Figure 8.18: Remeshed Cavity

In Figure 8.18, you can see that the cavity has been remeshed with tetrahedra.

9. Activate the global domain.

   Mesh ⟶Domains...

10. Display the original mesh along with the remeshed cavity (Figure 8.19).

    (a) 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 panel.

    (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 in the Scene Description panel.

    Display ⟶Scene...

    (e) Select the fluid zones in the Cell Zones selection list and enable All in the Options group box in the Cells tab of the Display Grid panel.

    (f) Enable Limit by X and enter `-0.37` for both Minimum and Maximum in the X Range group box in the Bounds tab.

    (g) Click Display.

    (h) Click Draw in the Cavity Remesh panel.

    Mesh ⟶ Tools ⟶Cavity Remesh...

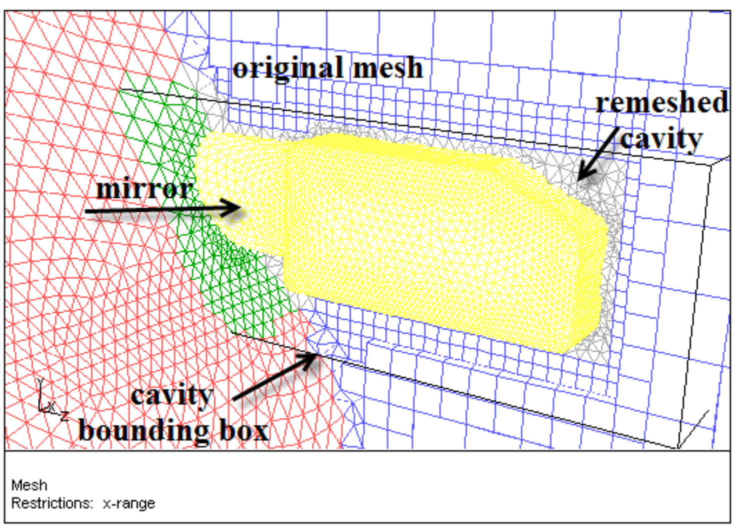


Figure 8.19: Cavity Remeshed With Tetrahedral Cells

11. Merge the cavity with the global domain.

```
> /mesh/cavity/merge-cavity
Insert domain name/id [cavity-1] <Enter>
Into domain name/id [global] <Enter>
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 IDs.

12. Delete the old mirror.

    Boundary ⟶Manage...

13. Check the mesh.

    Mesh ⟶Check

14. Save the mesh (`sedan-hexcore-cavity.msh.gz`).

    File ⟶ Write ⟶Mesh...

15. Exit TGrid.

    File ⟶Exit

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 (Case A), hybrid mesh with a single fluid zone (Case B), hybrid mesh with multiple fluid zones (Case C), and a hexcore mesh (Case D).