
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`.

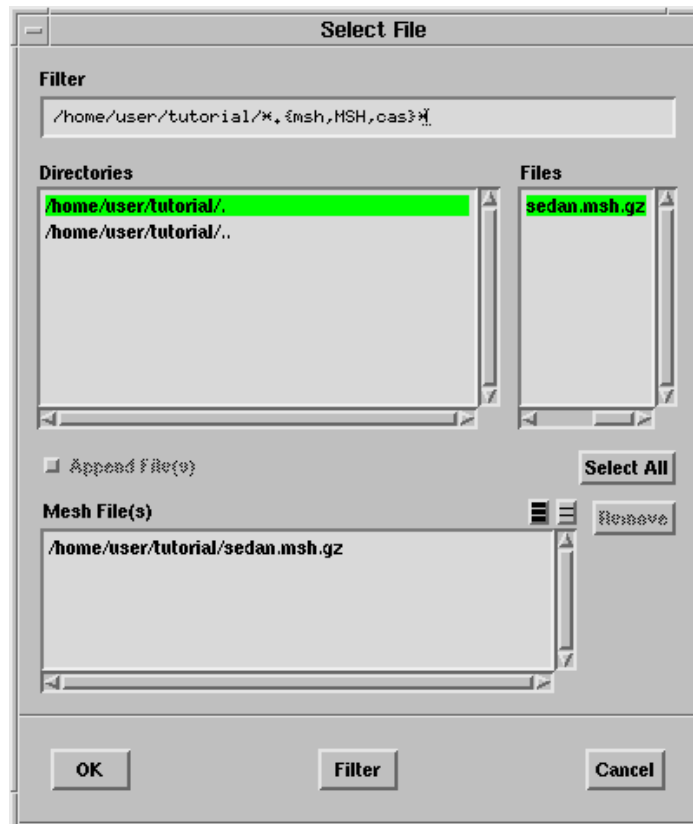
The file, `sedan.msh.gz` can be found in the `prisms` folder created on unzipping the file.

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

Step 1: Read and Display the Boundary Mesh

1. Read the mesh file.

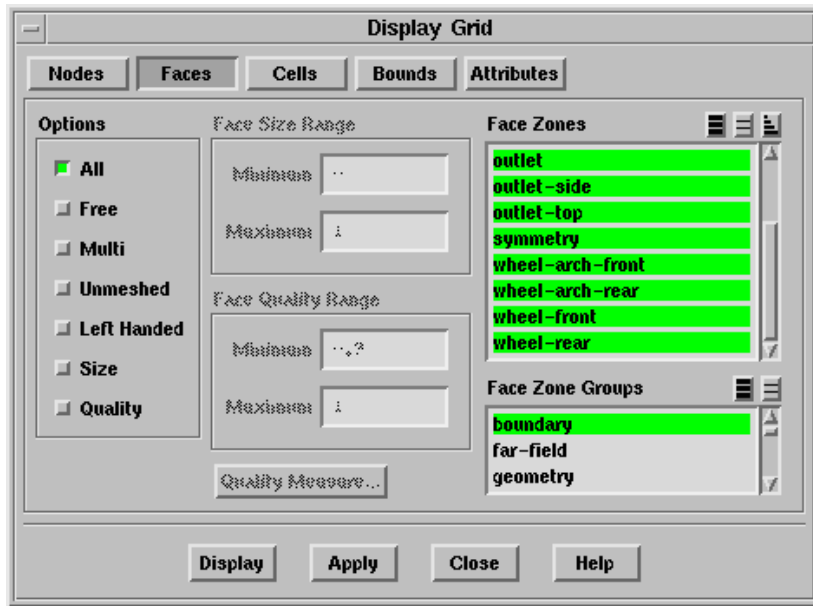
File → Read → Boundary Mesh...



- (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.

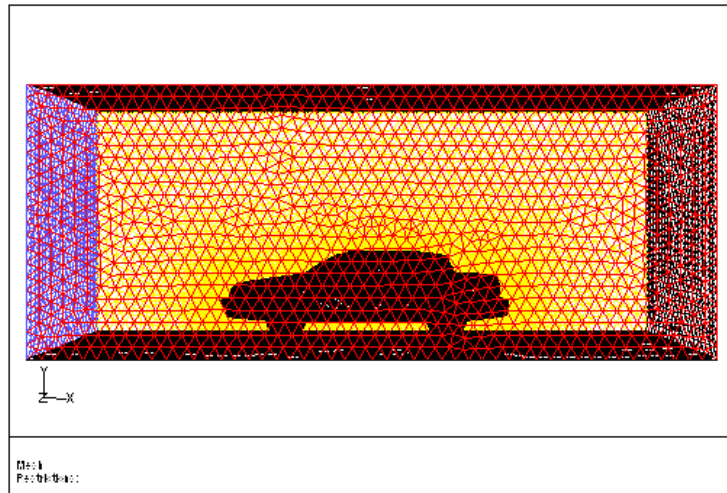


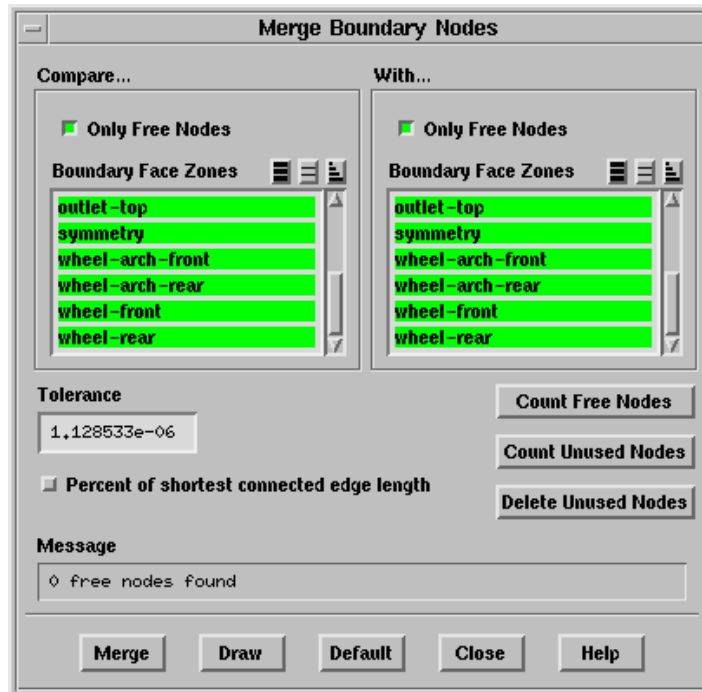
Figure 4.1: Boundary Mesh for the Sedan

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.

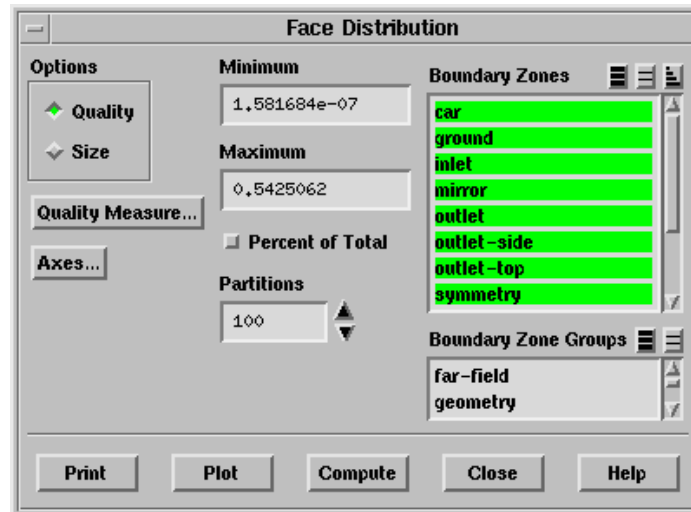
2. Click Count Unused Nodes.

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.

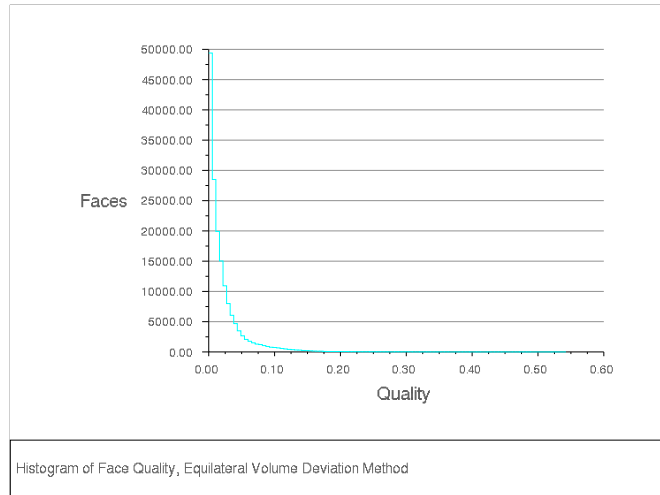
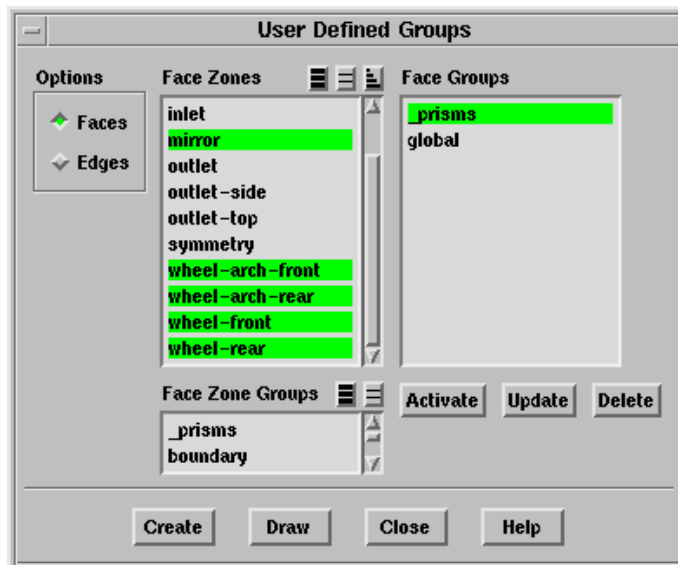
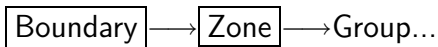


Figure 4.2: Surface Mesh Quality

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.



- (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.



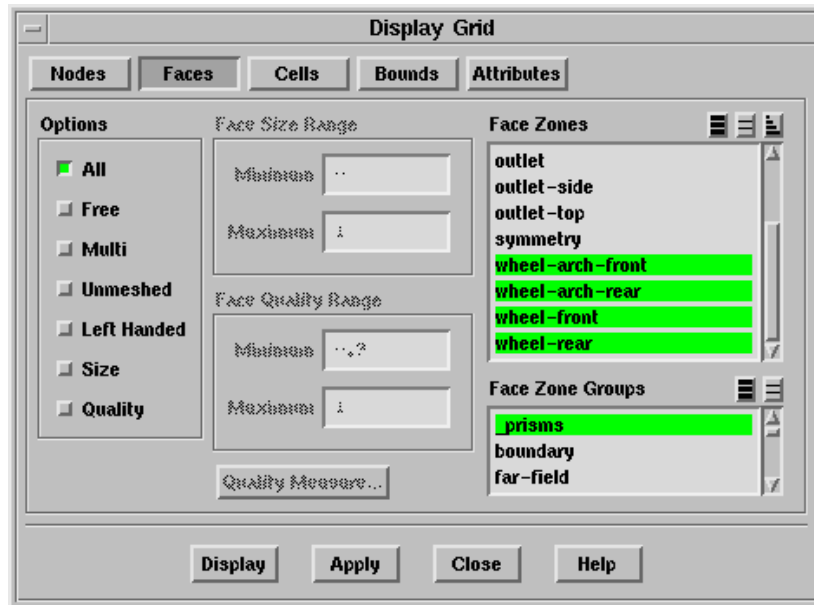
- (d) Enter `_prisms` for Name and click OK.

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.

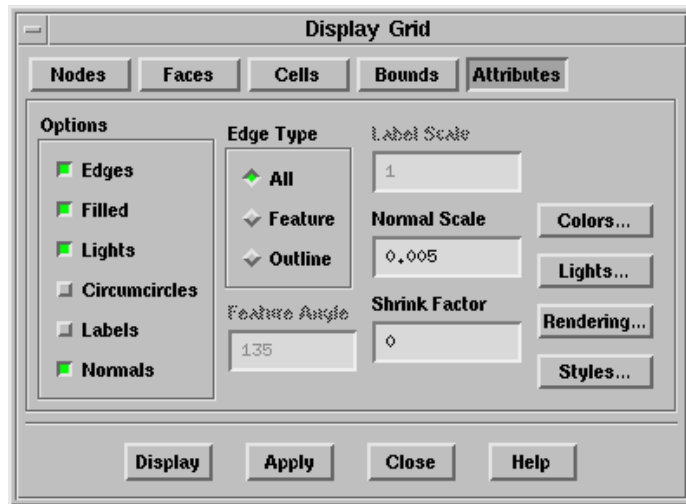
- 2. Verify that the normals are correctly oriented.

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).

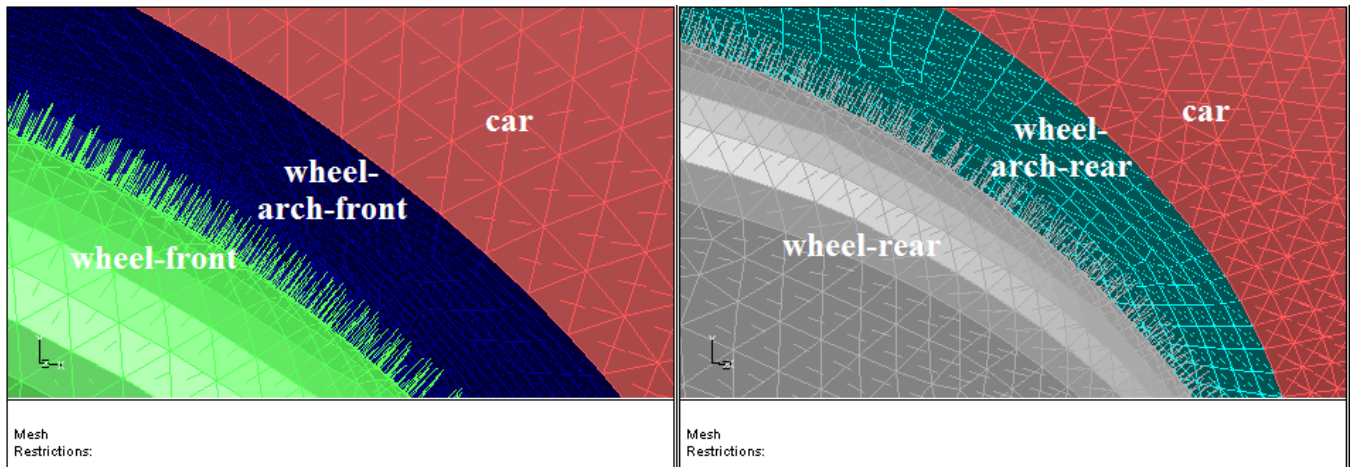


Figure 4.3: Normals on the Wheel and Wheel Arch Zones

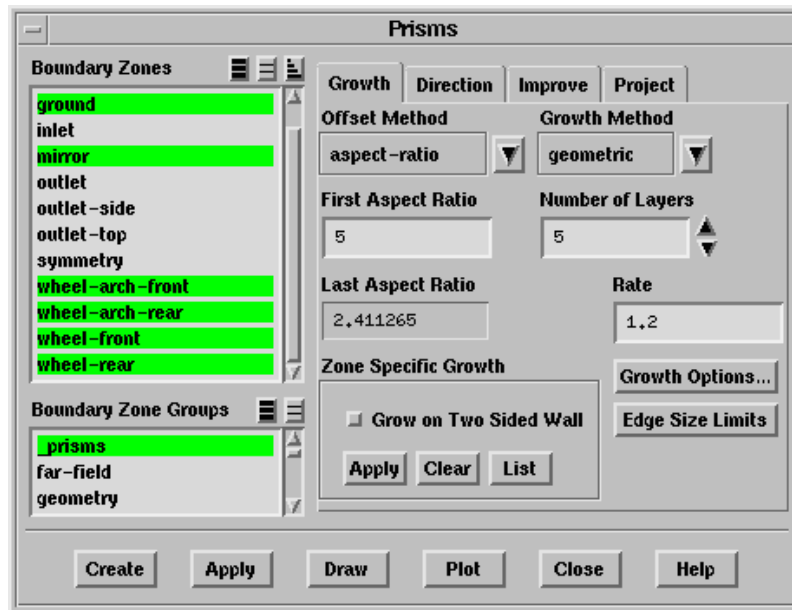
The normals on the car, mirror, wheel-arch-front, wheel-arch-rear, wheel-front, and wheel-rear point outward, while those on the ground point upward. Figure 4.3 shows the normals on the wheels and wheel arches. You will need to flip the normals on the wheel-arch-front and wheel-arch-rear zones for the 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.

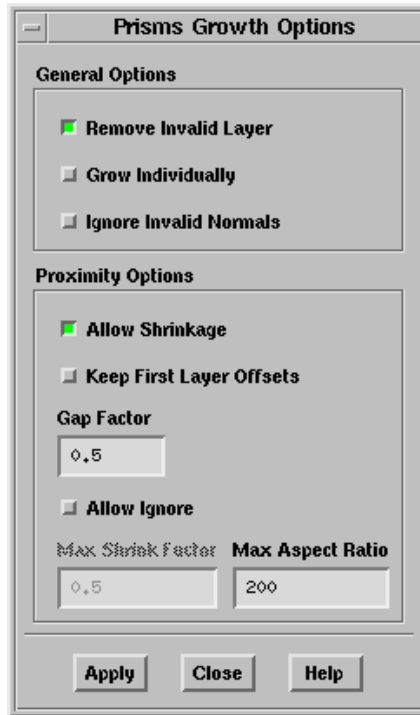
Mesh → Prisms...

- (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.

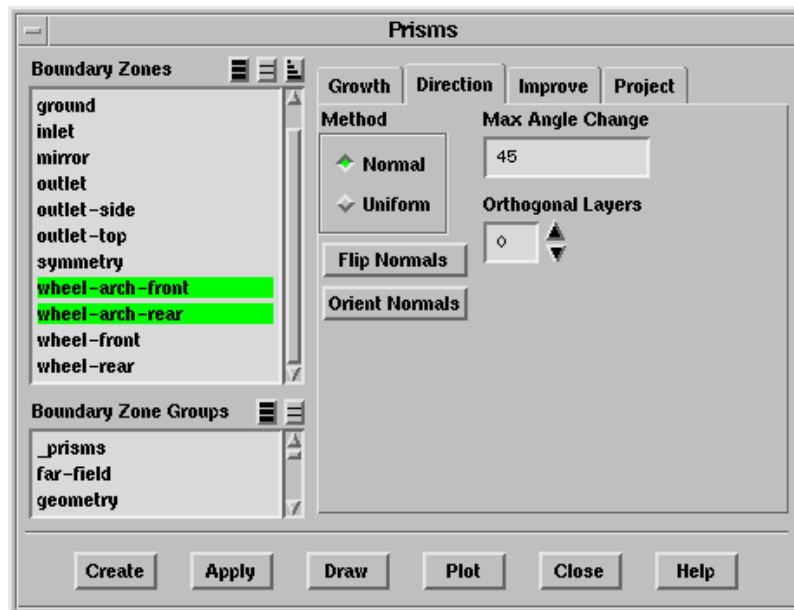


- ii. Enter 200 for Max Aspect Ratio and click Apply.

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).

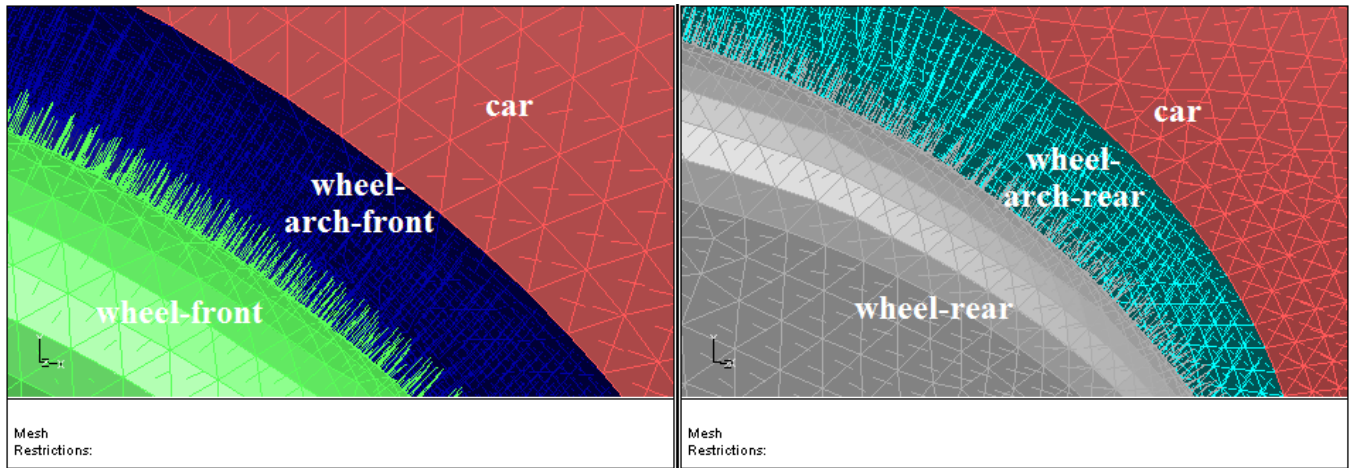
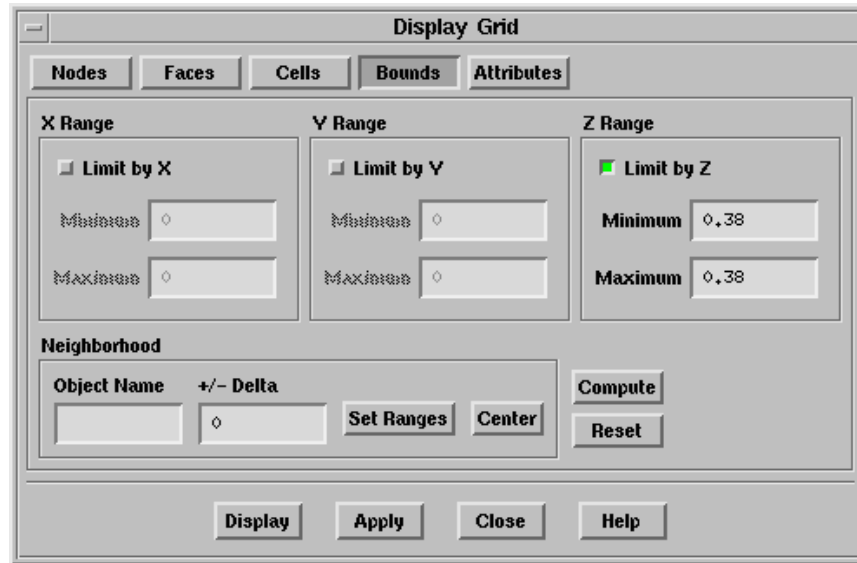
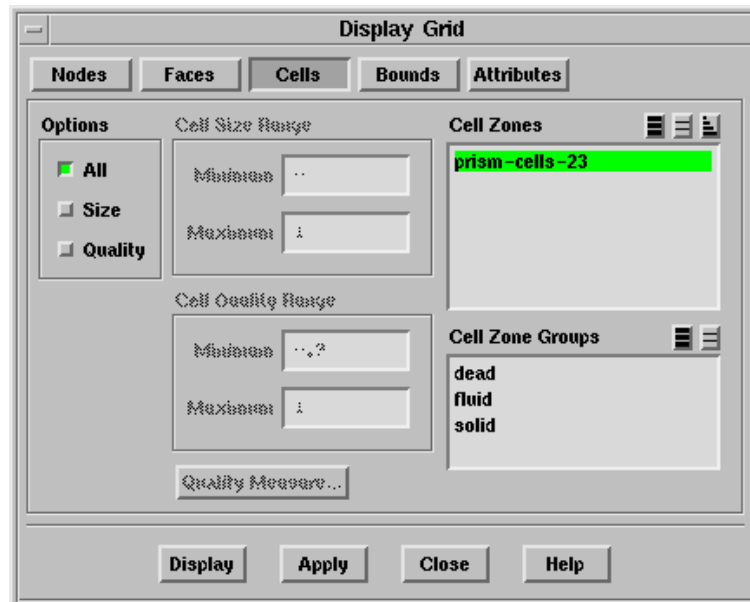


Figure 4.4: Normals on Wheels and Wheel Arches After Flipping

- (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 $z = 0.38$.
- **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.

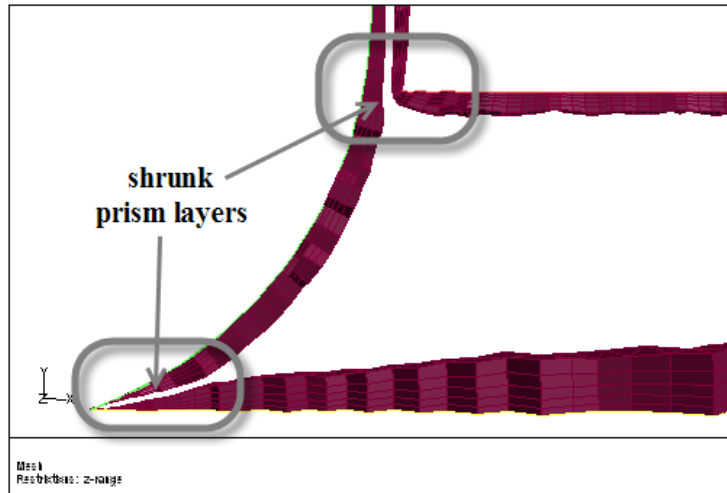
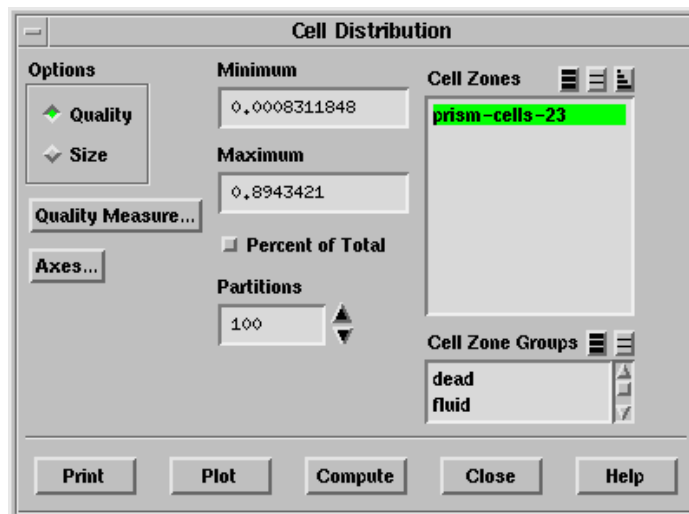


Figure 4.5: Prisms Shrunk in Areas of Proximity and Sharp Corners

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.

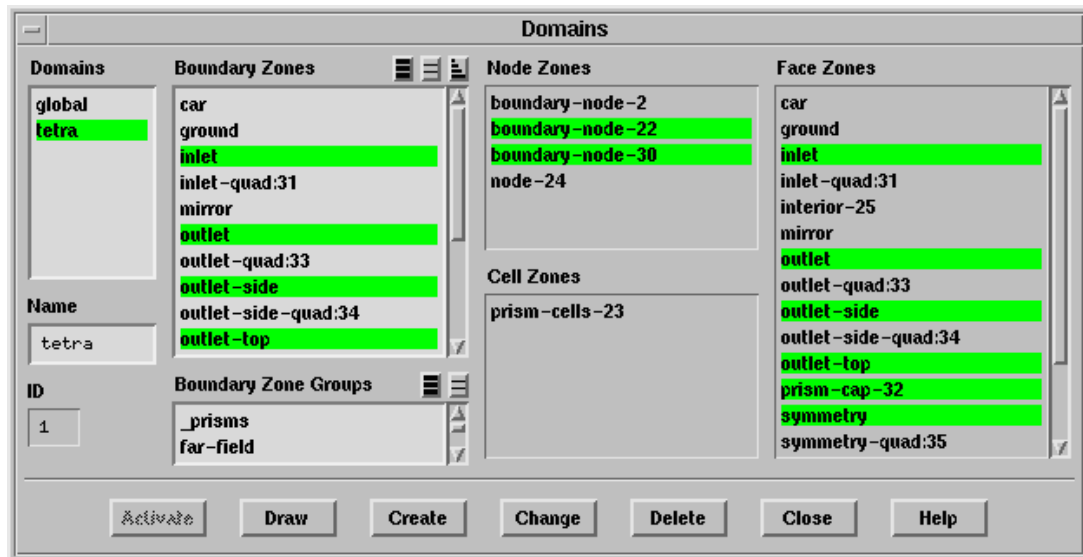
A maximum skewness value below 0.9 is acceptable.

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...



- Select inlet, outlet, outlet-side, outlet-top, prism-cap-#, and symmetry in the Boundary Zones selection list.
- Enter tetra for Name and click Create.
The domain tetra will be created and activated.
- Verify that all the zones required are included in the defined domain.

Display → Grid...

- Click the Bounds tab and click Reset.
- Select all the zones in the Face Zones selection list in the Faces tab.
- 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.

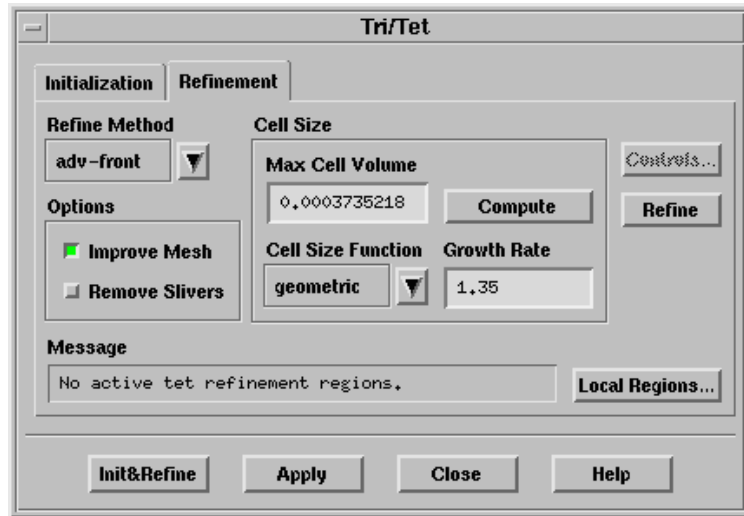
- Click Display.
- Close the Display Grid panel.

- Close the Domains panel.

7. Set the parameters for tetrahedral meshing.

Mesh → Tri/Tet...

- 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).

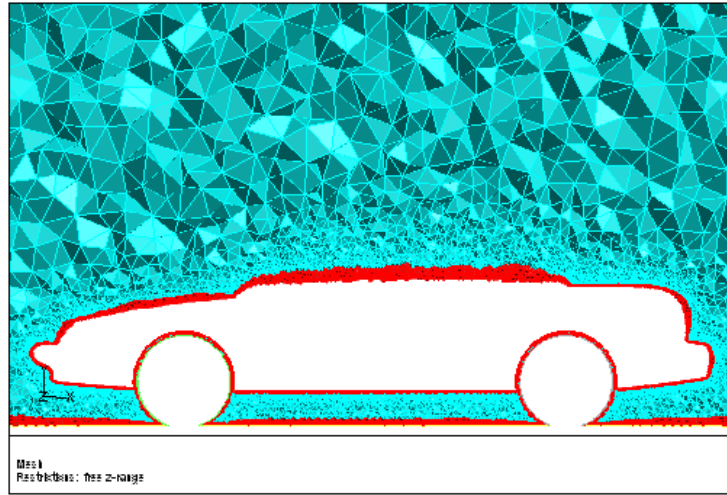


Figure 4.6: Slide of Cells at $z = 0.38$

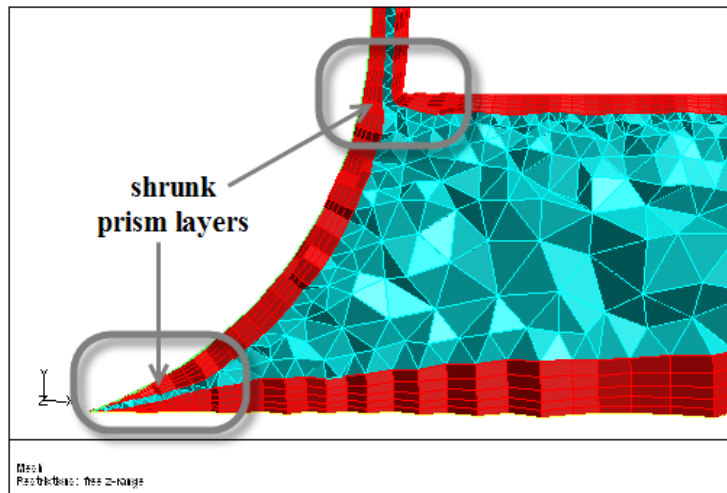
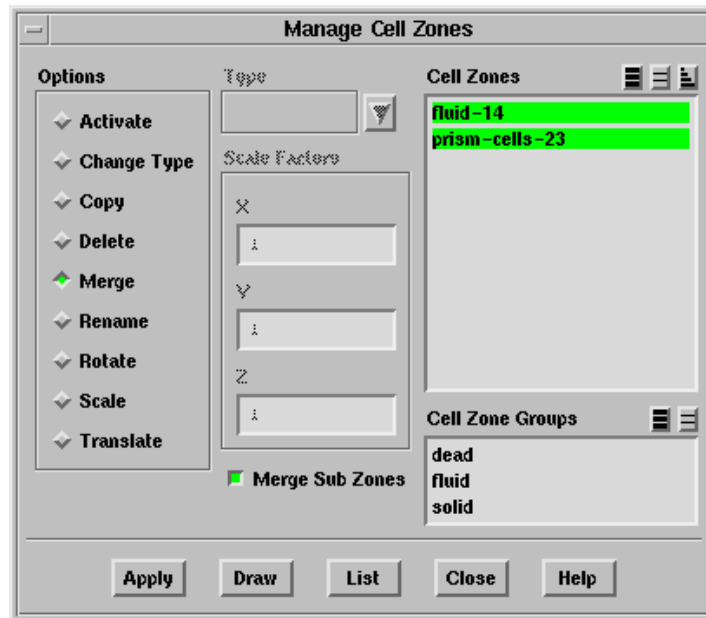


Figure 4.7: Prisms Shrunken in Areas of Proximity

11. Merge the cell zones generated.

Mesh → Manage...

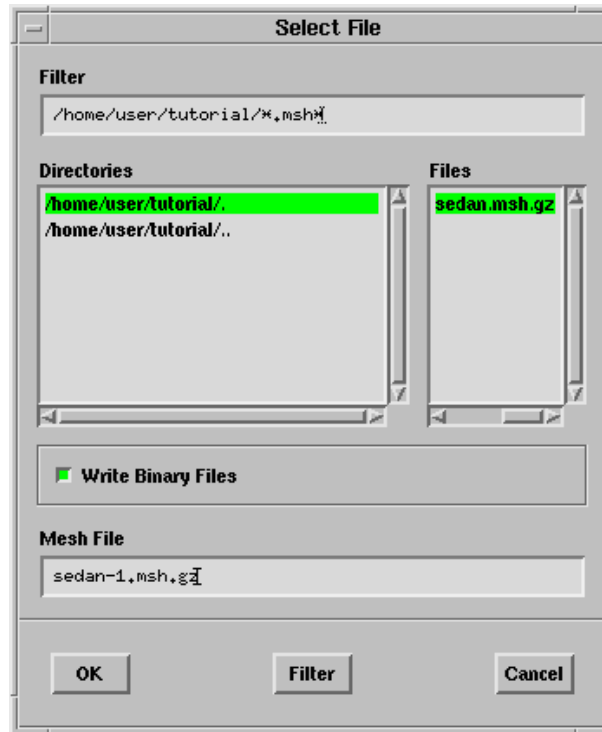


- (a) Select the fluid and prism cell zones in the Cell Zones selection list.
- (b) Select Merge in the Options list.
- (c) Enable Merge Sub Zones and click Apply.
- (d) Close the Manage Cell Zones panel.
12. Check the mesh.

Mesh → Check

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

File → Write → Mesh...



- Enter sedan-1.msh.gz for Mesh File.
- 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.

- Read the mesh file (sedan.msh.gz).

File → Read → Mesh...

- 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...



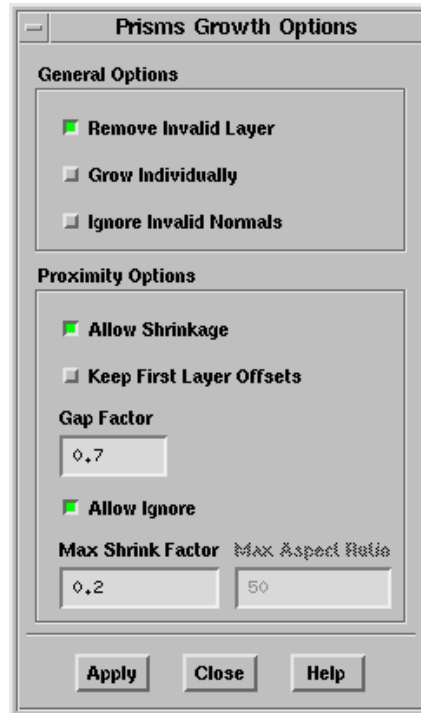
The Prisms option is greyed out as no prism parameters have been set.

- (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.

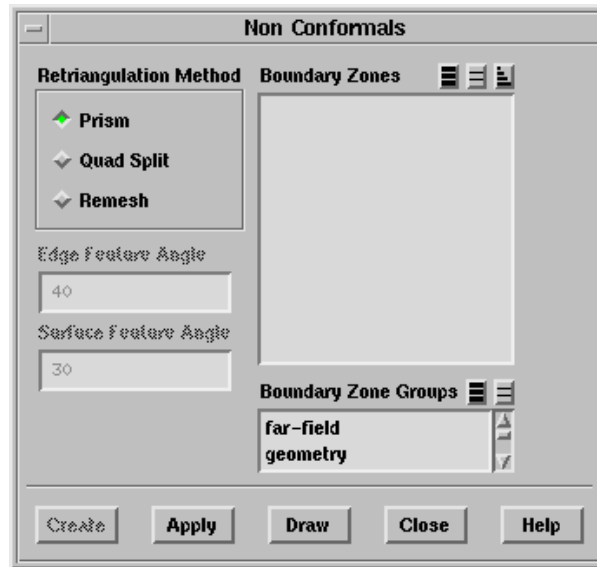


- A. Retain the Allow Shrinkage option and enable Allow Ignore.
- B. Enter 0.7 for Gap Factor and 0.2 for Max Shrink Factor, respectively.
- C. Click Apply and close the Prisms Growth Options panel.

- 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.
- Retain the settings in the Initialization tab.
 - Click the Refinement tab and retain the selection of adv-front in the Refine Method drop-down list.
 - Select geometric in the Cell Size Function drop-down list and enter 1.35 for Growth Rate.
 - 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...

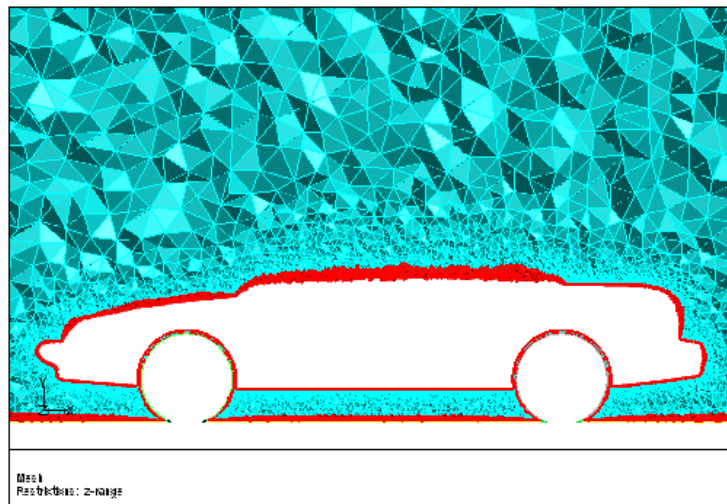


Figure 4.8: Slide of Cells at $z = 0.38$

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

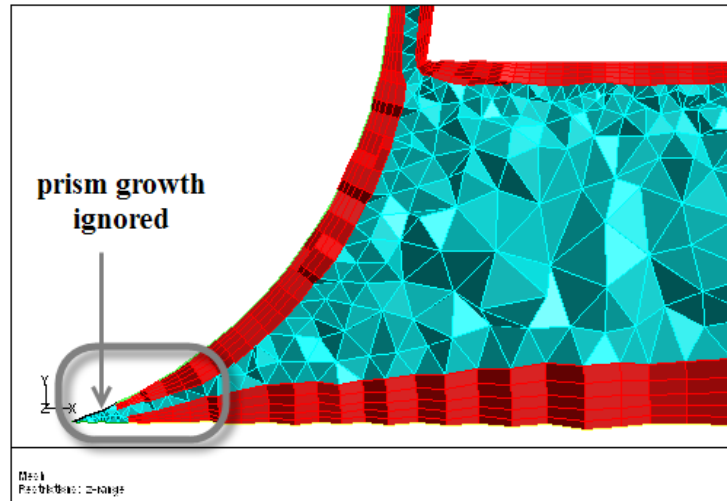


Figure 4.9: Prisms Ignored in Areas of Proximity

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.