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.

<div align="center">OR</div>

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).
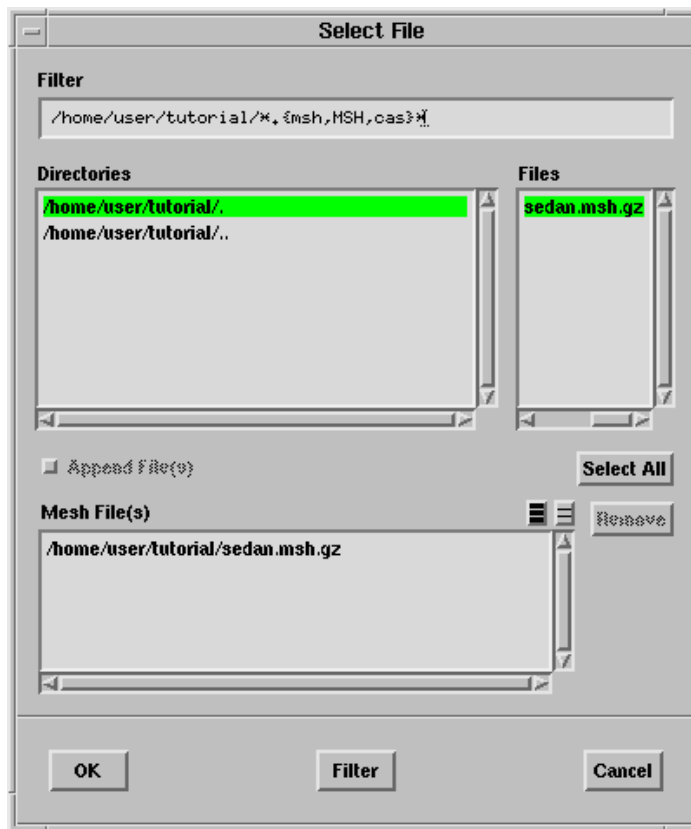
2. Unzip tet-mesh.zip.

   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.
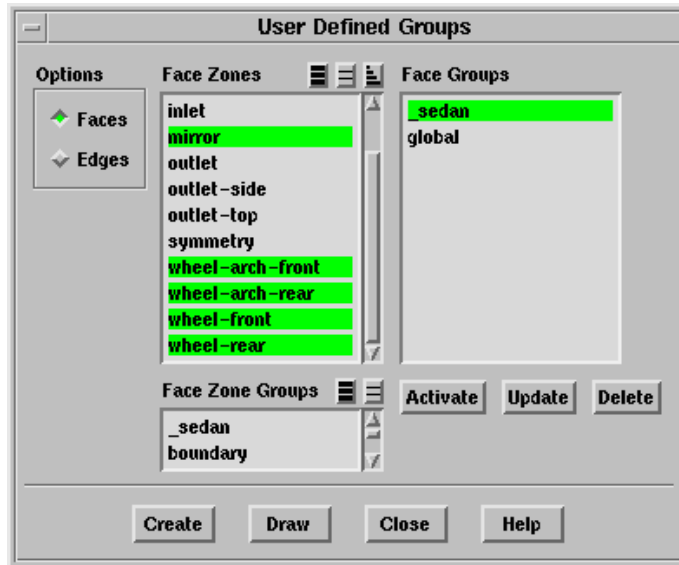
Step 1: Read and Display the Boundary Mesh

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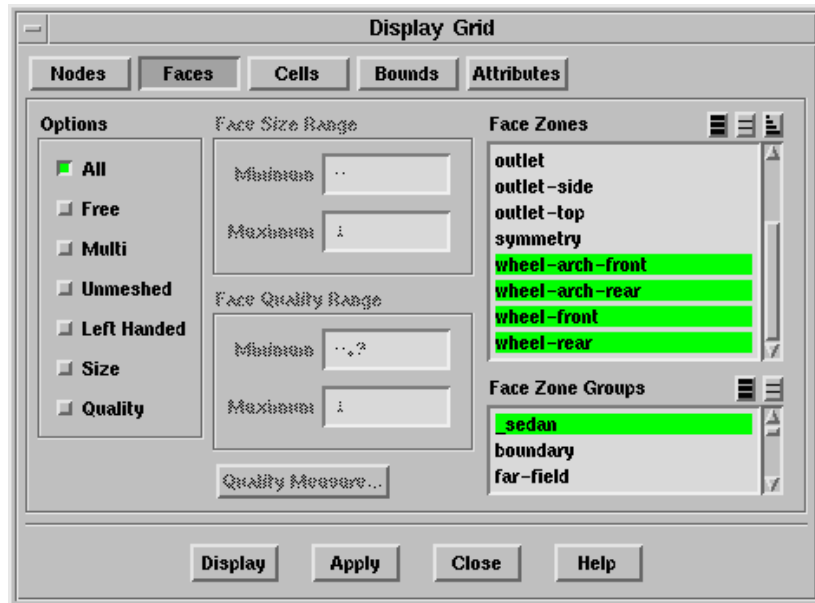
(a) Select _sedan in the **Face Zone Groups** selection list to select all the boundary zones defining the car in the **Face Zones** selection list.

(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...
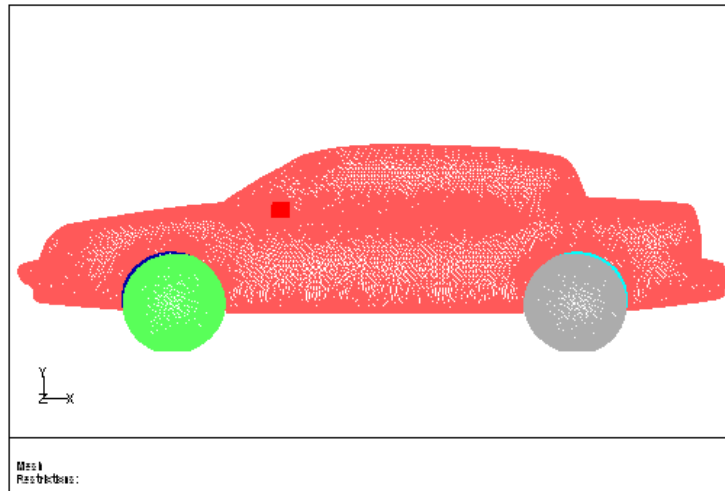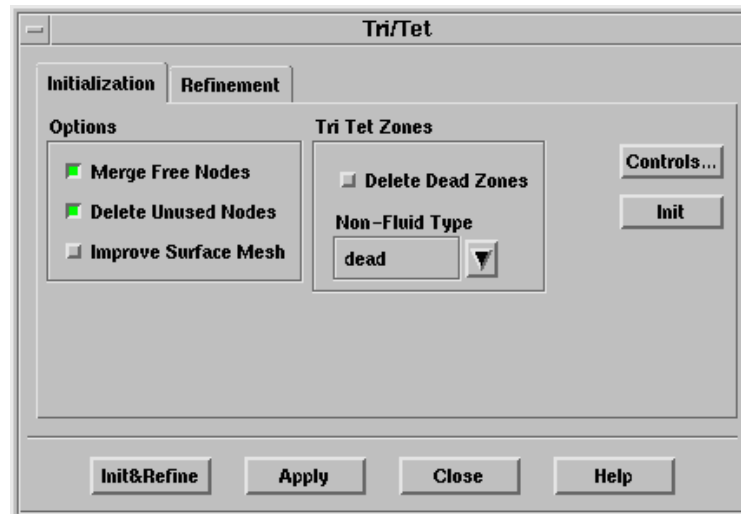
(e) Click **Display**.

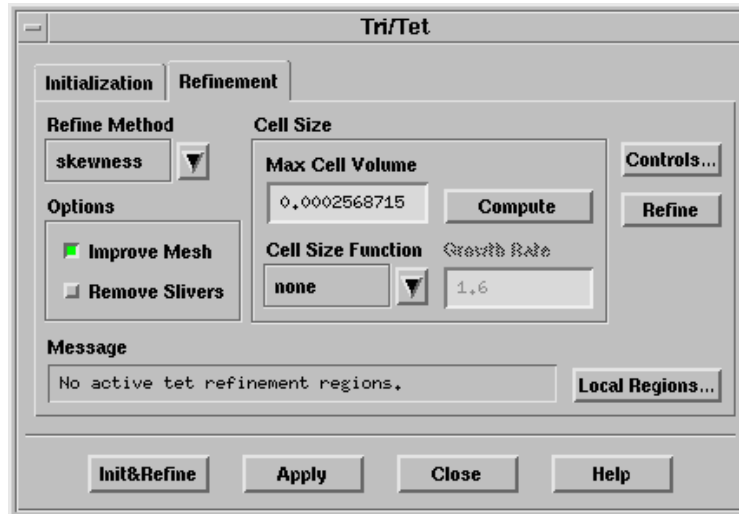(f) Close the **Display Grid** panel.

Figure 2.1: Grid Display

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

1. Specify the meshing parameters.

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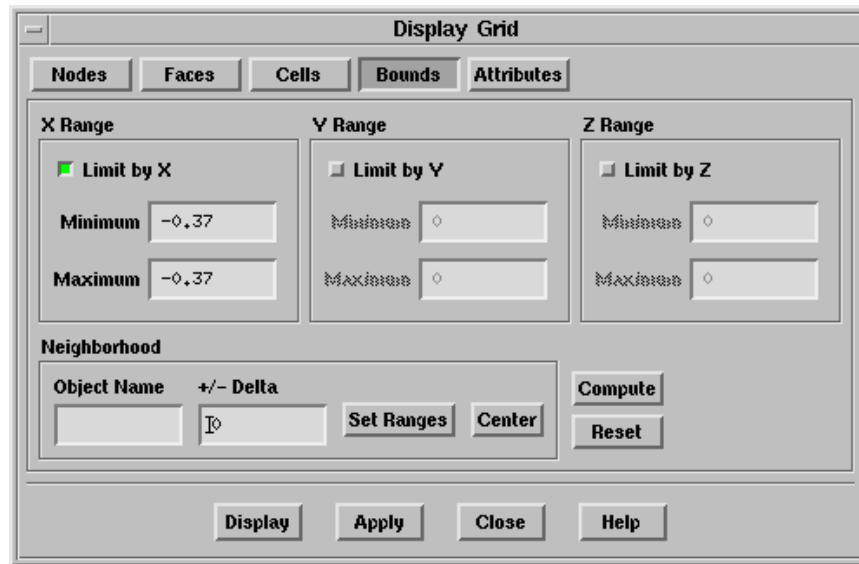
In this case, an edge length of 0.1 would seem appropriate (giving a volume of 1.18e-4), but the mesh would be larger and slower to generate. Hence, you will use the default value for the maximum cell volume.

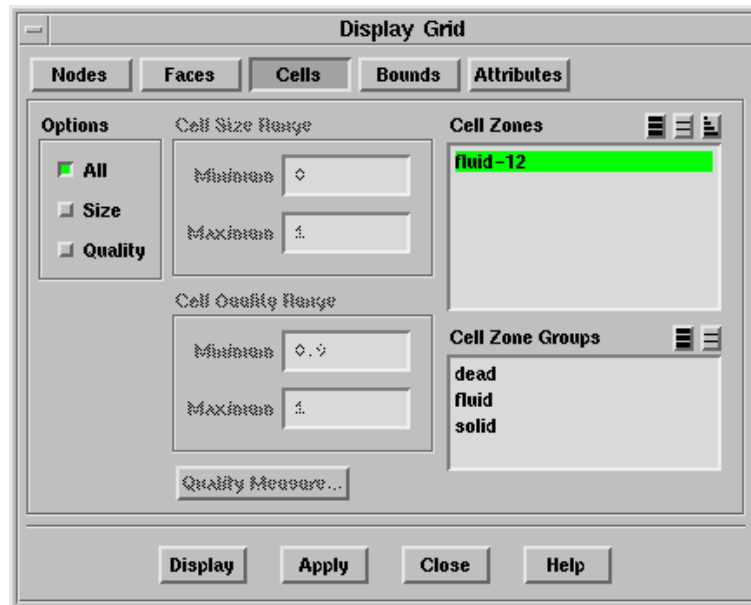(c) Click Apply and Init&Refine.

(d) Close the Tri/Tet panel.

2. Examine the mesh.

   Display ⟶Grid...

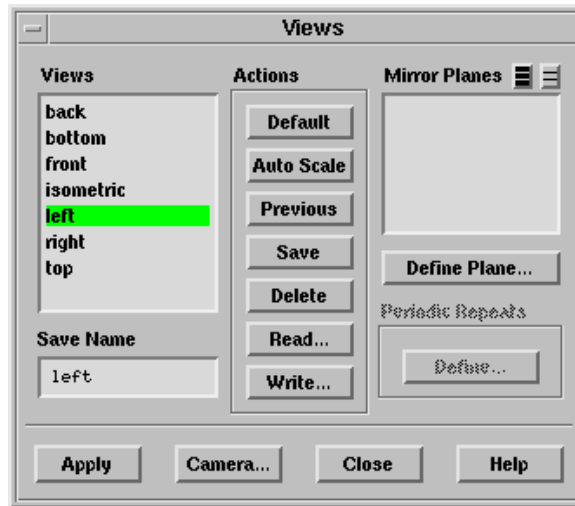   (a) Display the mesh on a slide through the mirror and the car (Figure 2.2).

   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.

$\boxed{\text{Display}}$ ⟶ Views...

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

B. Click Auto Scale.

C. Close the Views panel.
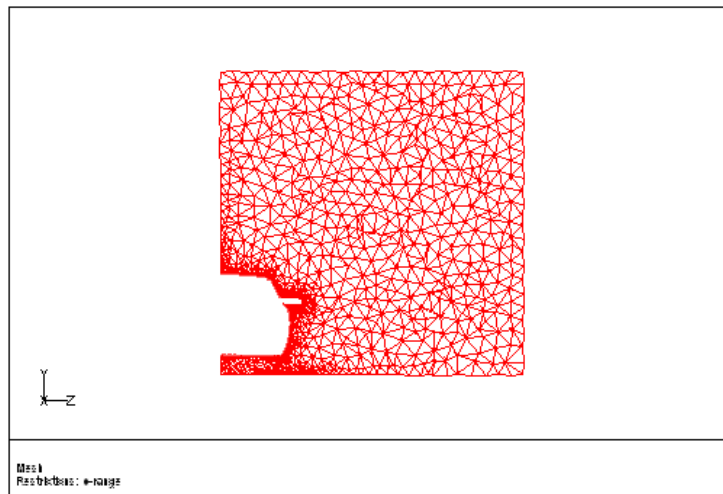

Figure 2.2: Slide of Cells at X = -0.37

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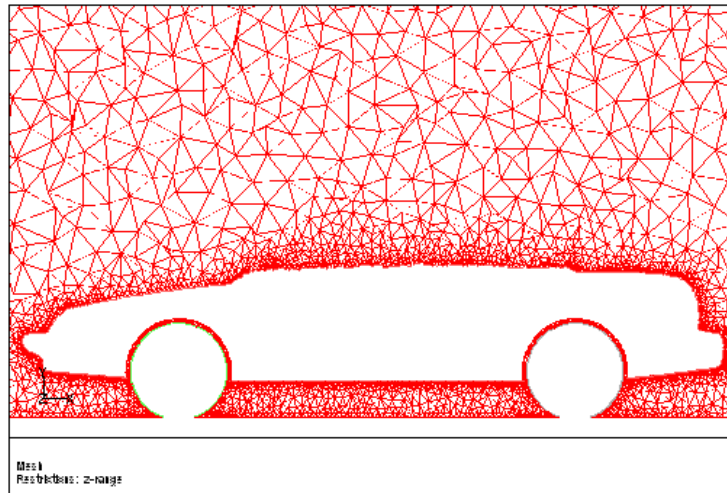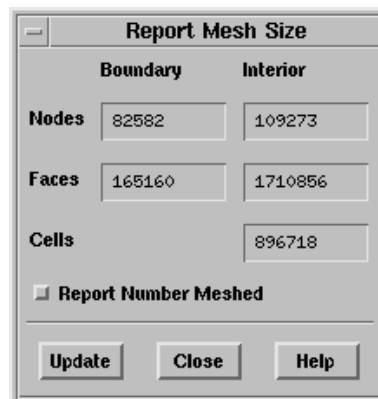
    iv. Zoom in to sedan to examine the cell growth.

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.
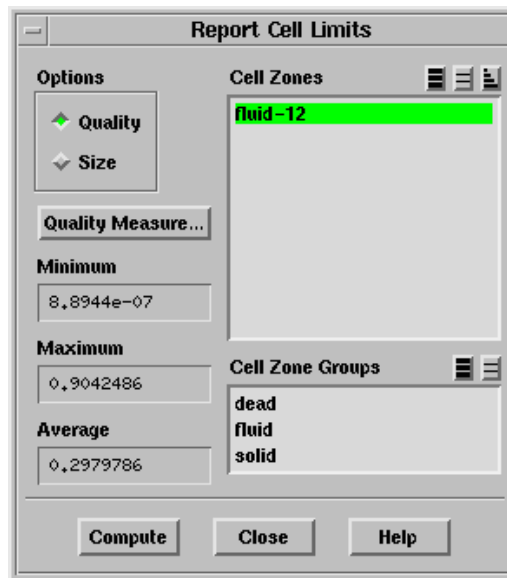
(c) Check the number of cells.

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.

$\boxed{\text{Report}}\longrightarrow\text{Cell Limits...}$

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.

   $\boxed{\text{Mesh}}\longrightarrow\text{Clear}$

2. Specify the meshing parameters.

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

   (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...

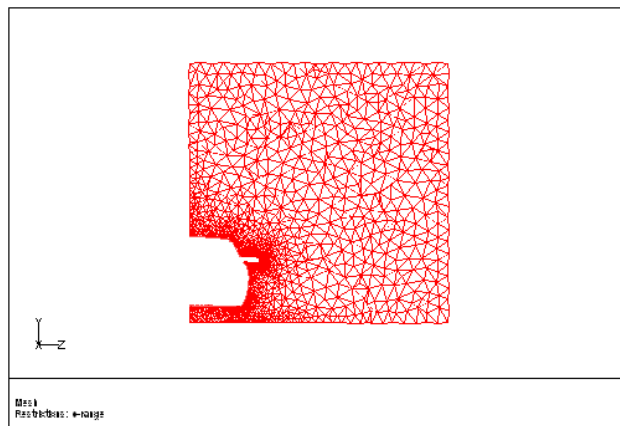   i. Display the mesh on a slide through the mirror and the car (x = -0.37). See Figure 2.4.

Figure 2.4: Slide of Cells at X = -0.37

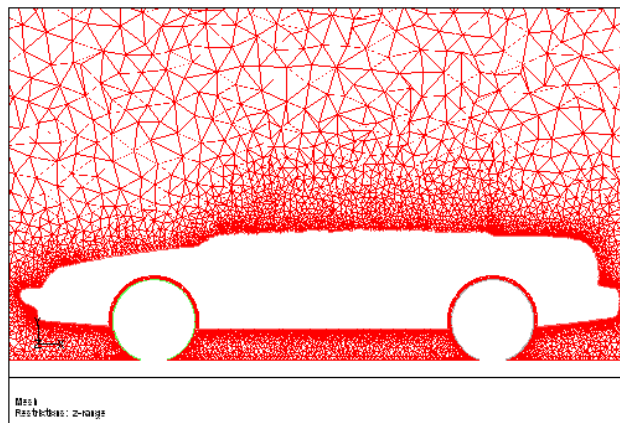   ii. Display the mesh on a slide through the wheels (z = 0.38). See Figure 2.5.

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.

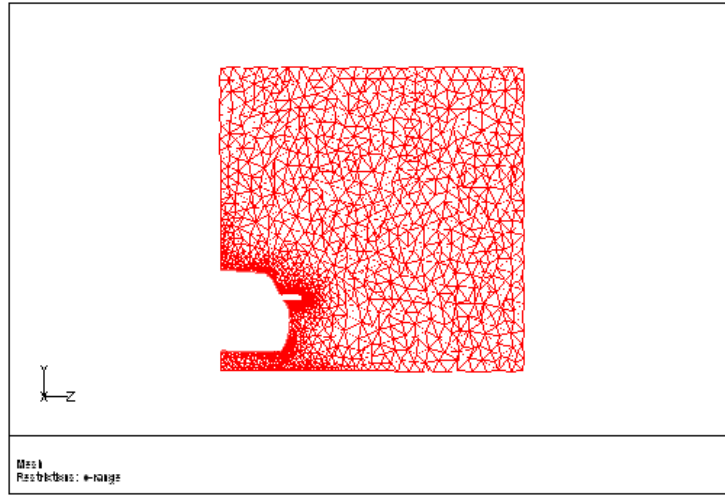 ii. Display the mesh on a slide through the wheels (z = 0.38). See Figure 2.7.

Figure 2.6: Slide of Cells at X = -0.37

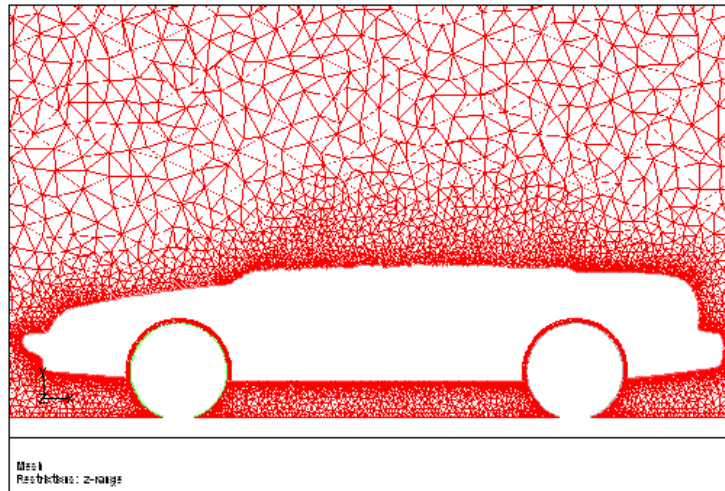Figure 2.7: Slide of Cells at Z = 0.38

(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.

Notes:

- *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.

   $\boxed{\text{Mesh}}\longrightarrow$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.

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

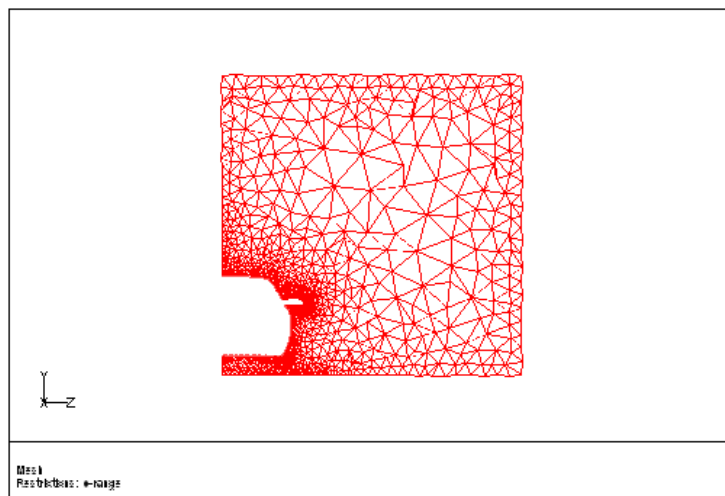   (a) Display the mesh on a slide through the mirror and the car (x = -0.37). See Figure 2.8.

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.

   $\boxed{\text{Report}}\longrightarrow$Mesh Size...

   The number of cells is 1339210. The exact number may differ on different platforms.

   (d) Check the maximum skewness.

   $\boxed{\text{Report}}\longrightarrow$Cell Limits....

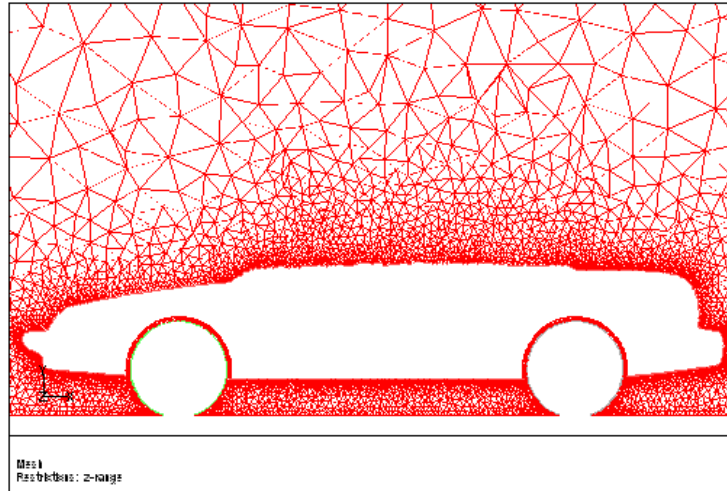   *The maximum skewness is 0.904, while the average skewness is 0.240.*

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.

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

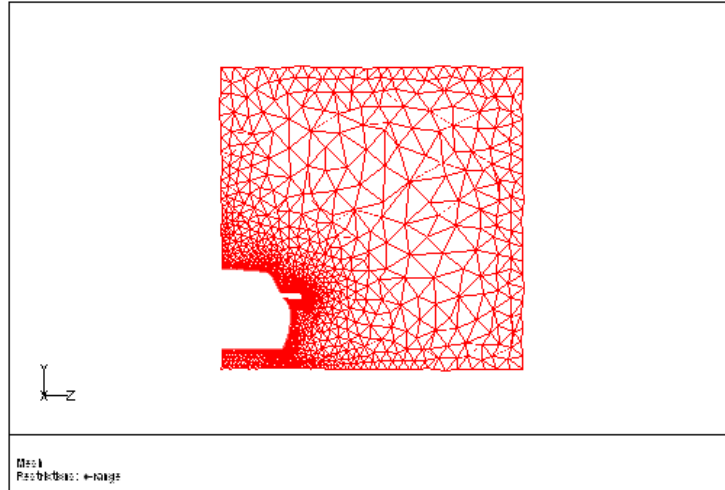(a) Display the mesh on a slide through the mirror and the car (x = -0.37).

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.
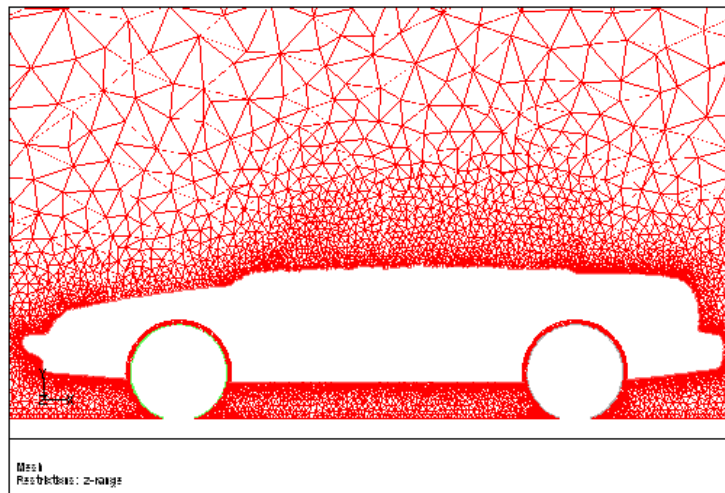

Figure 2.11: Slide of Cells at Z = 0.38

(c) Check the number of cells.

$\boxed{\text{Report}}\longrightarrow$Mesh Size...

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

(d) Check the maximum skewness.

$\boxed{\text{Report}} \longrightarrow \text{Cell Limits...}$

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.
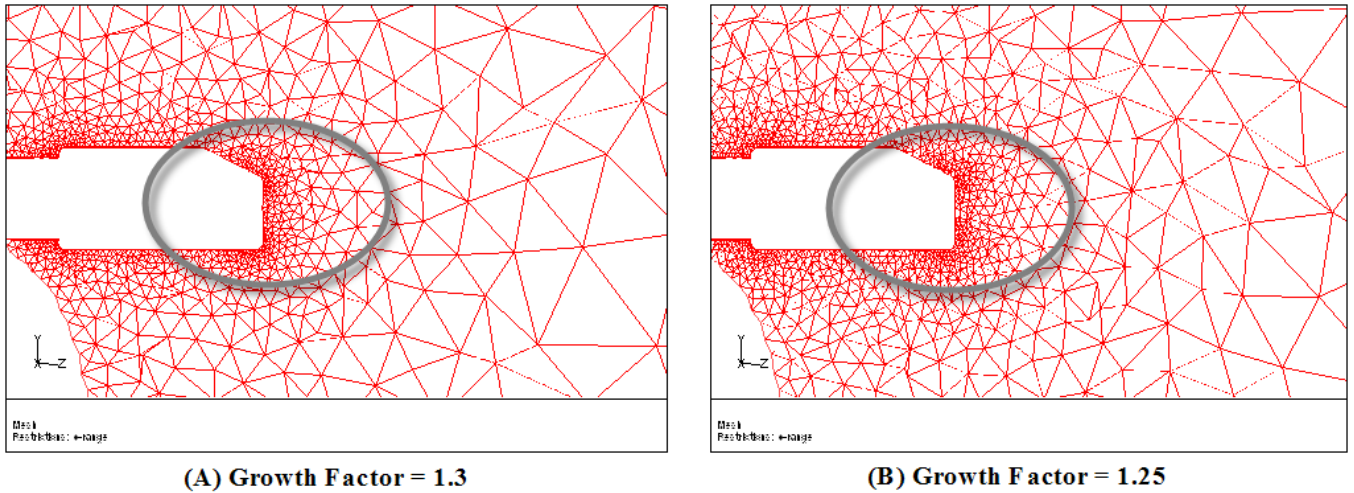

(A) Growth Factor = 1.3   **(B) Growth Factor = 1.25**

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.

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

   (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.
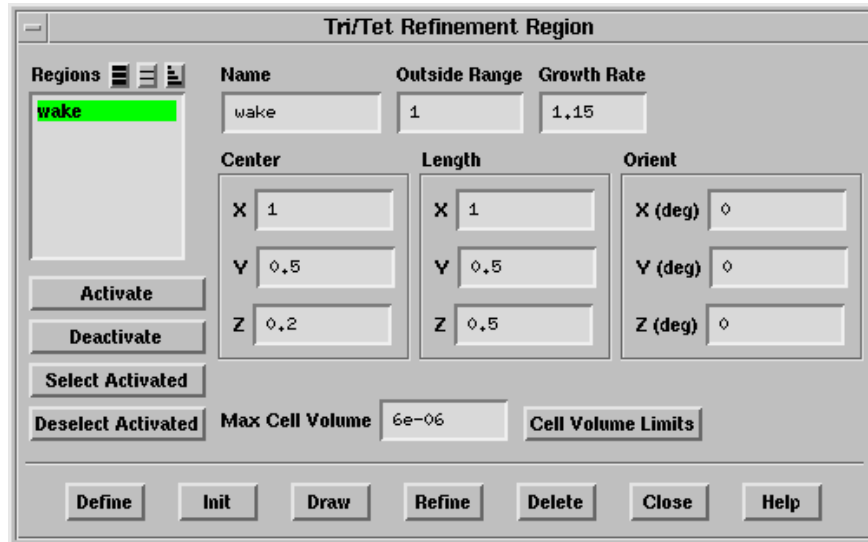
i. Enter wake for Name.

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.

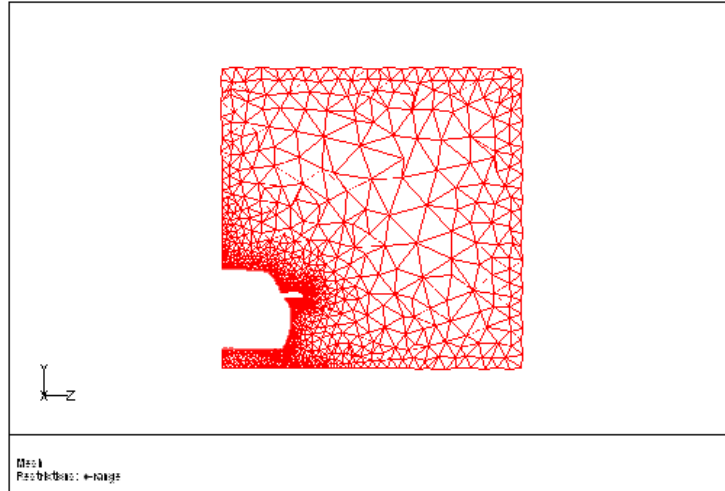

Figure 2.13: Slide of Cells at X = -0.37

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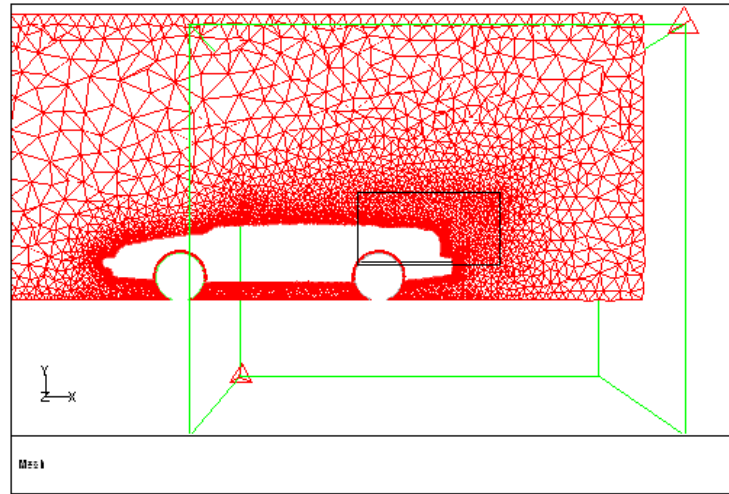
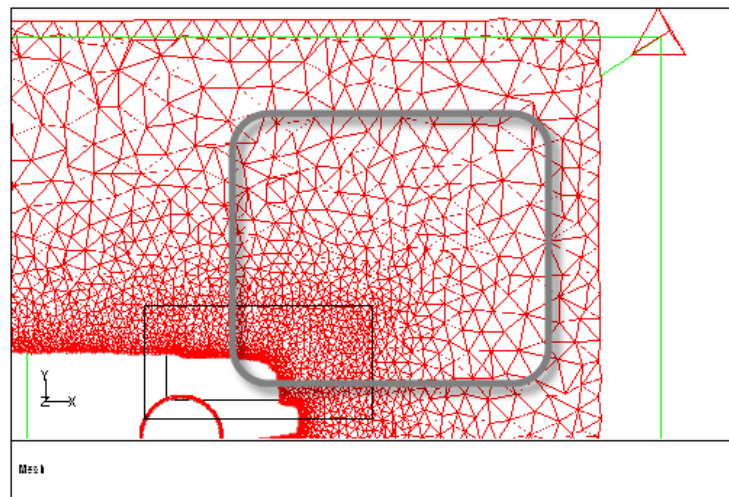Figure 2.14: Slide of Cells at Z = 0.38 with the Refinement Region

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.