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`.

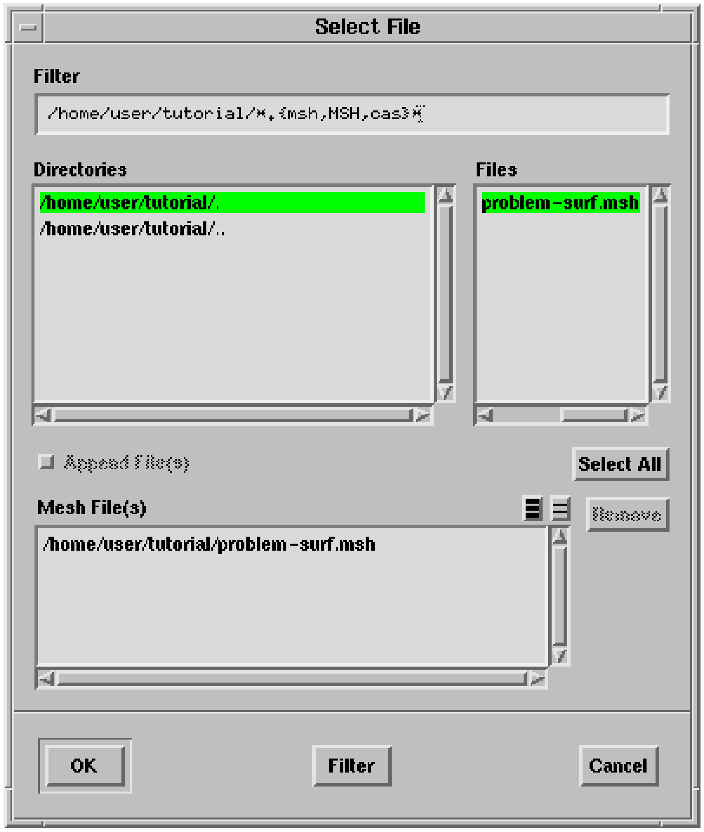
   The file, `problem-surf.msh` *can be found in the* `mesh-repair` *folder created on unzipping the file.*

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...

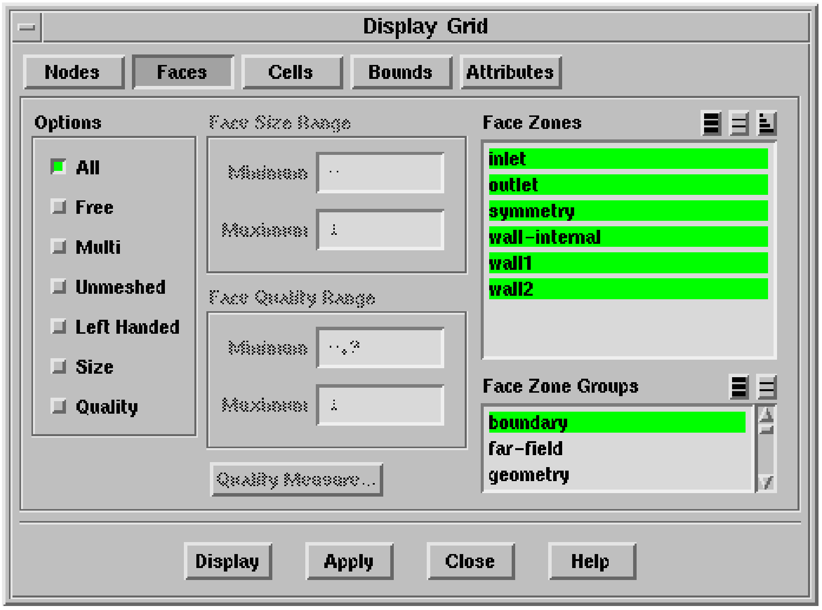


   (a) Select problem-surf.msh in the Files list.

   (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).

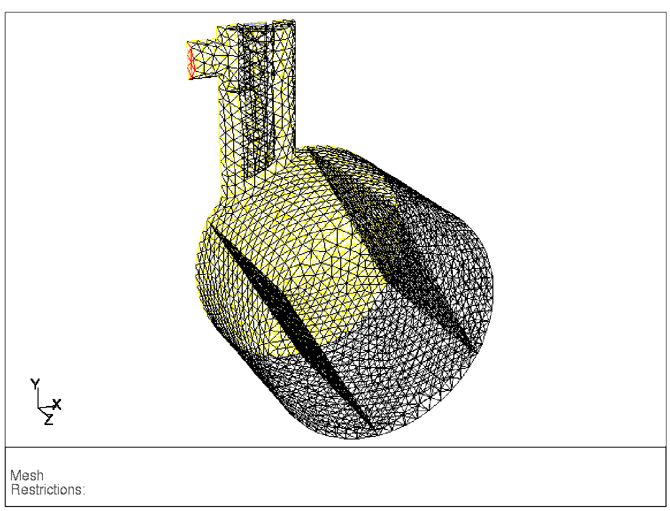


Figure 1.1: Boundary Mesh

   (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.

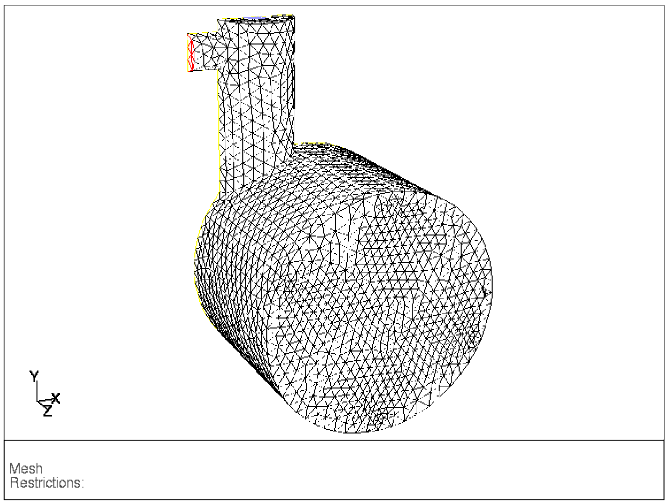


Figure 1.2: Boundary Mesh With Hidden Lines Removed

Step 2: Check for Free and Unused Nodes

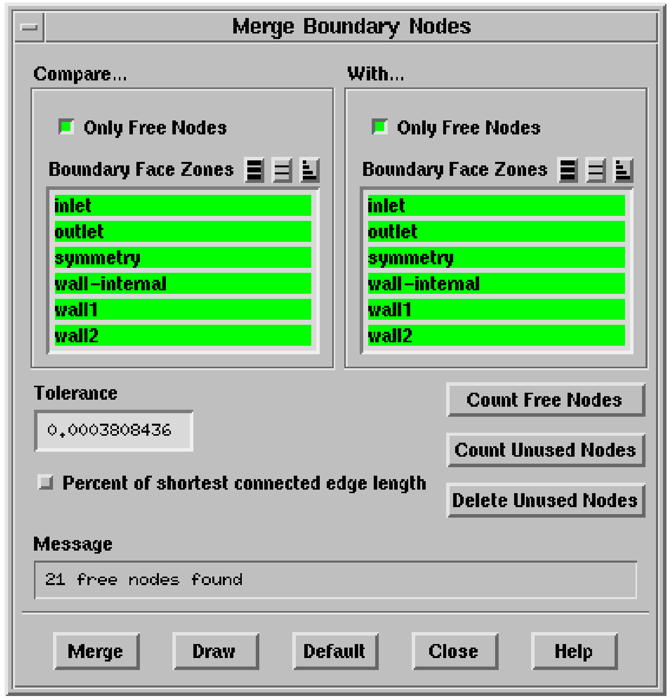
After reading the boundary mesh, check it for topological problems such as free and multiply-connected nodes and faces.

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

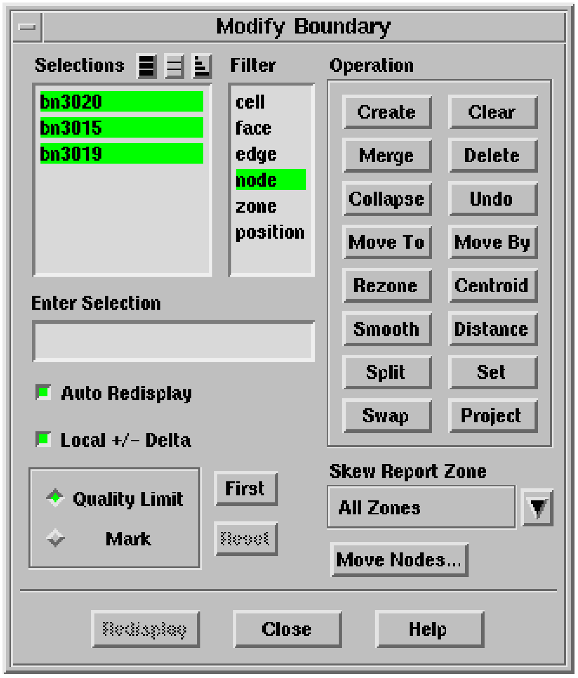
In this step, you will recreate the missing faces to repair the boundary mesh.

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).

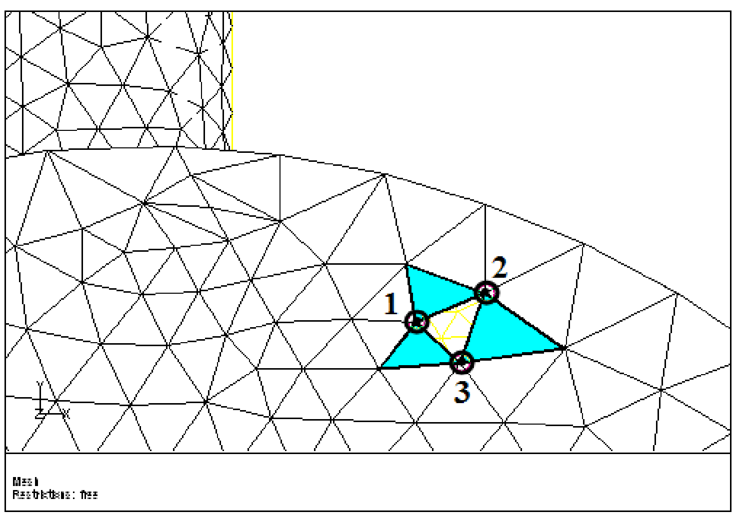


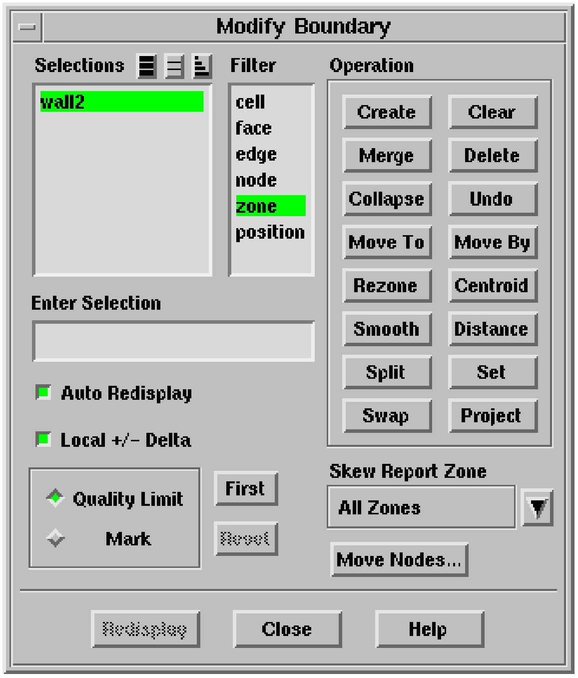
Figure 1.3: Recreating the Missing Face

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).*

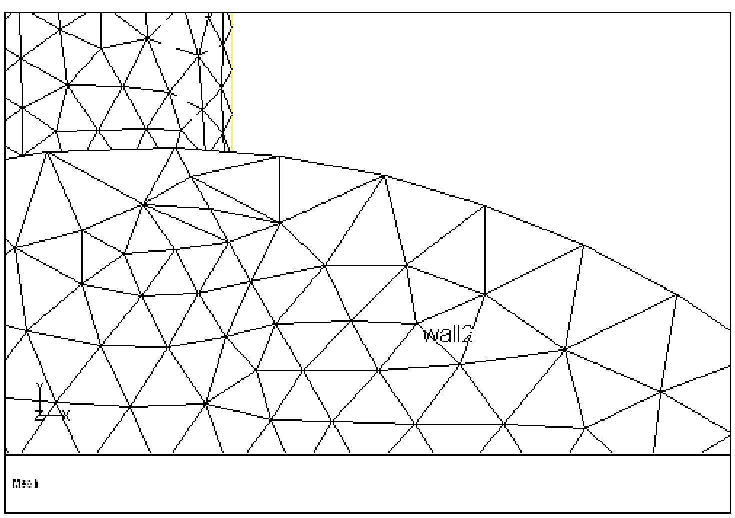


Figure 1.4: Verifying the Zone of the New Face

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).

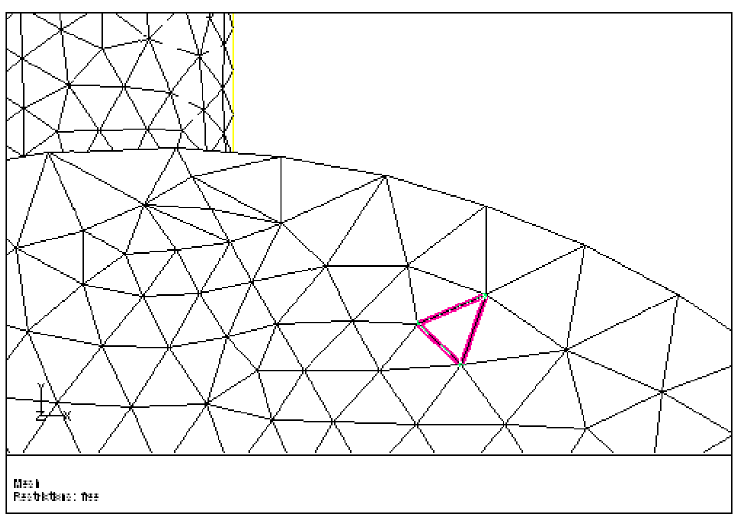
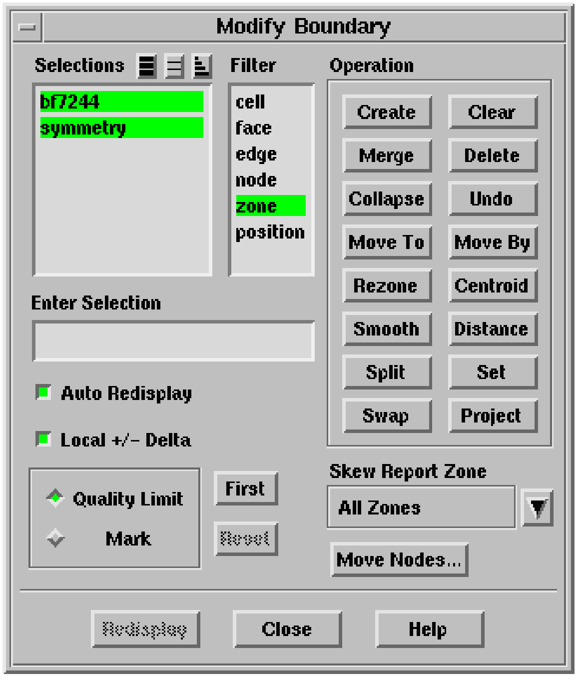


Figure 1.5: Face Selected to be Rezoned

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.

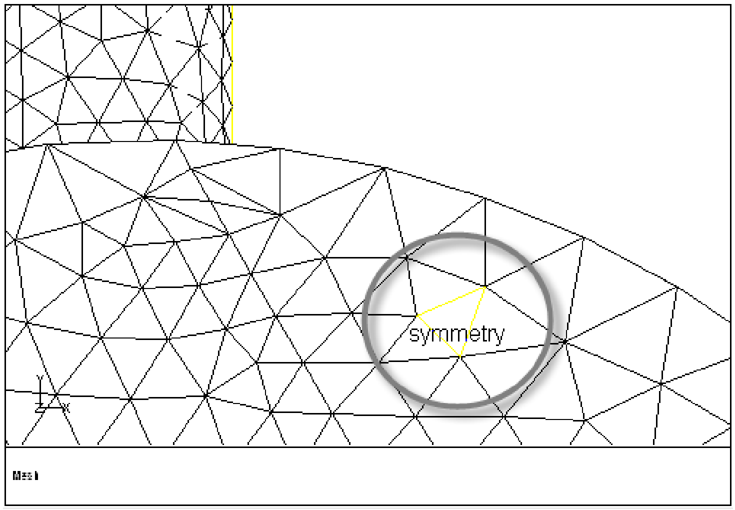
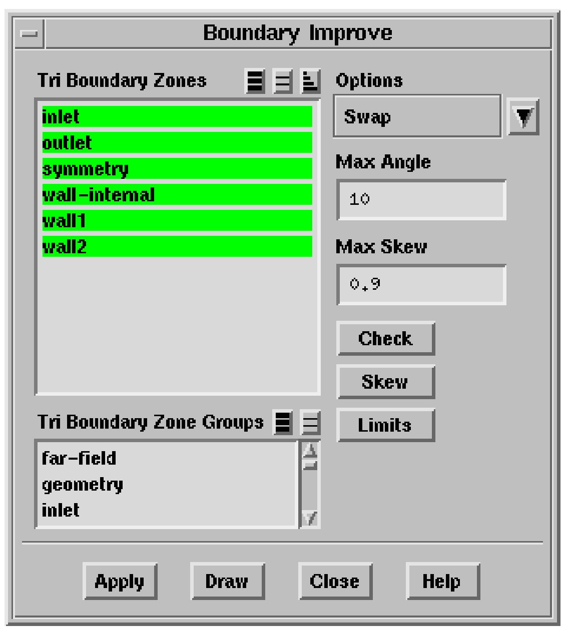


Figure 1.6: Face Rezoned to Symmetry Boundary

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).

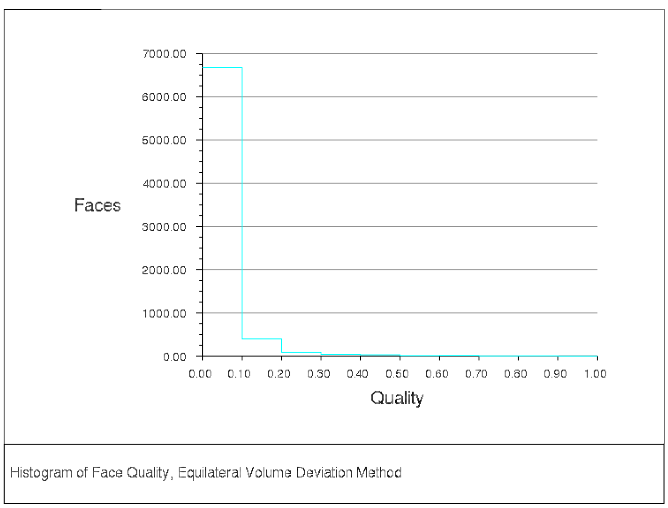


Figure 1.7: Histogram Plot of Face Skewness

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.

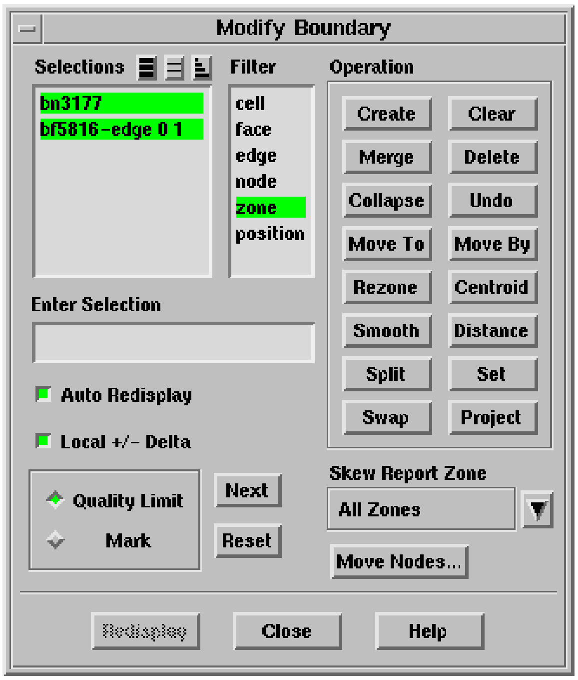
Extra: *This tutorial also aims at reducing the maximum face skewness below 0.6. This tutorial exposes you to some of the mesh repair tools. Then, it is up to you to try and get the maximum face skewness below 0.6.*

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.

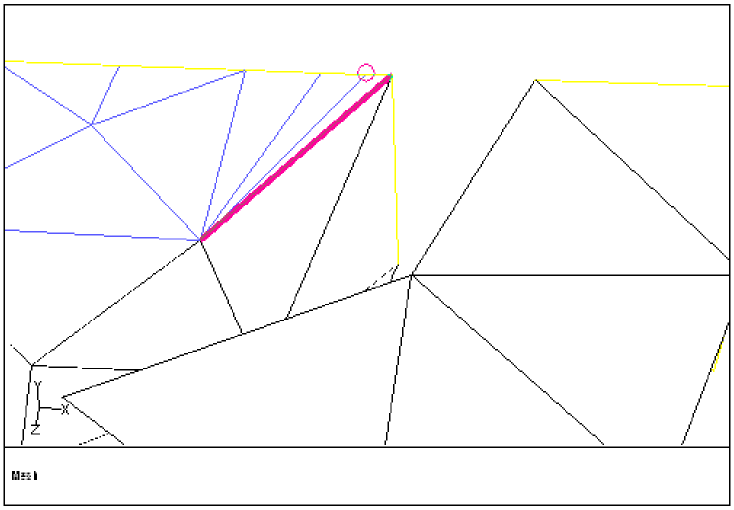


Figure 1.8: Face with the Greatest Skewness

(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.

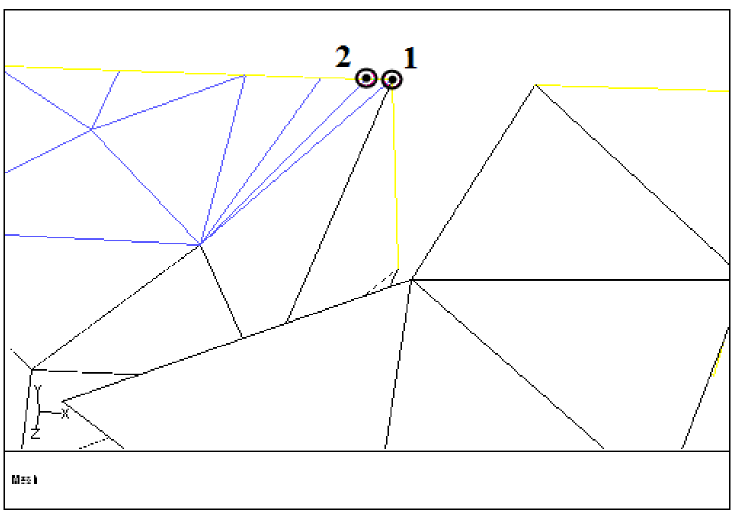


Figure 1.9: Nodes to be Merged

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

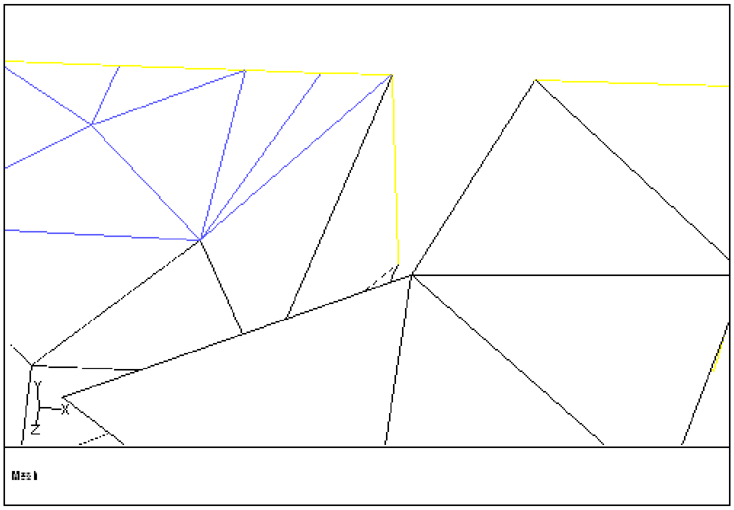


Figure 1.10: Surface Mesh After Merging Nodes

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.*

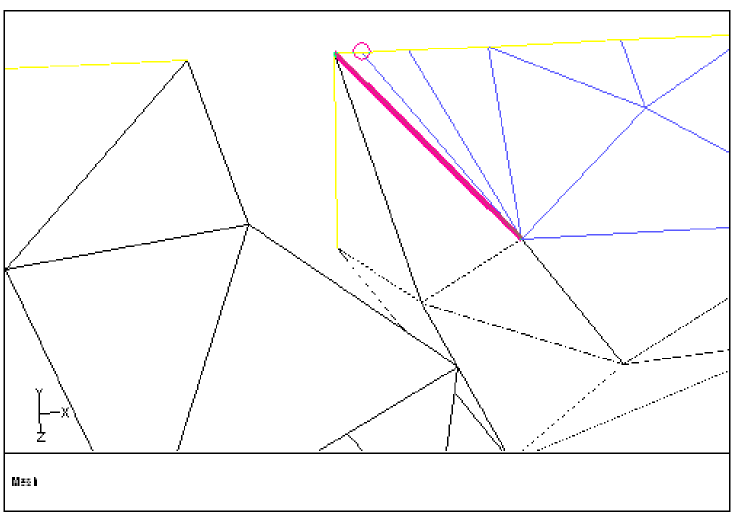


Figure 1.11: Face with the Next Greatest Skewness

   (b) Clear the Selections list.

   (c) Select node in the Filter list.

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

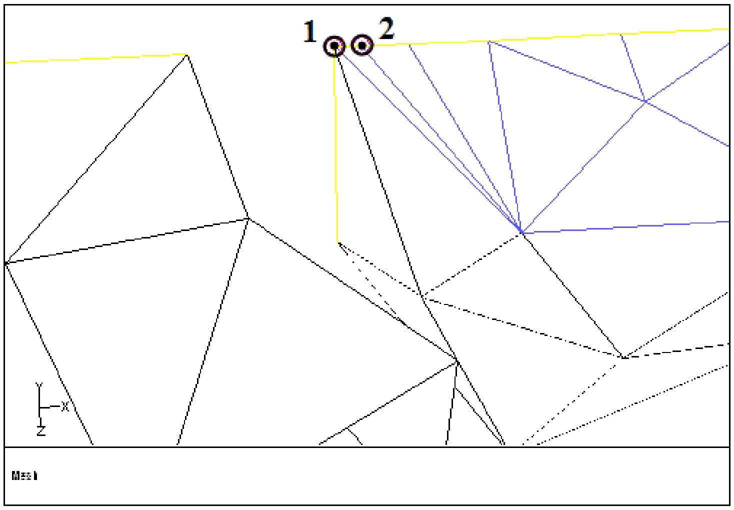


Figure 1.12: Nodes to be Merged

(e) Click Merge.

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

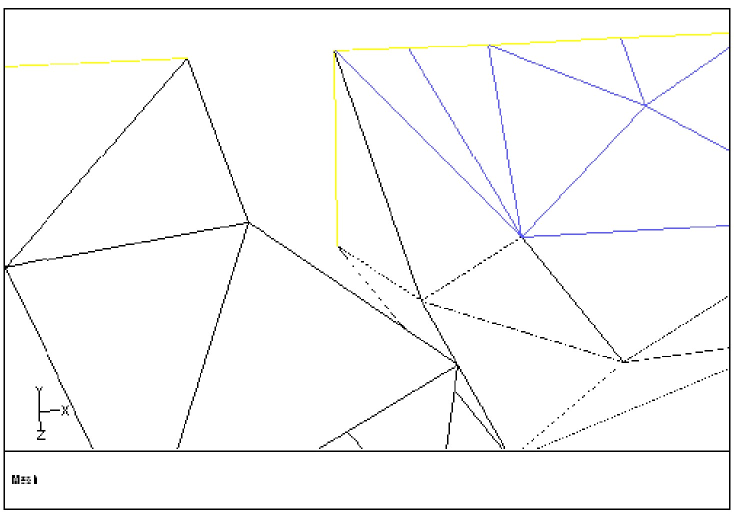


Figure 1.13: Surface Mesh After Merging Nodes

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).

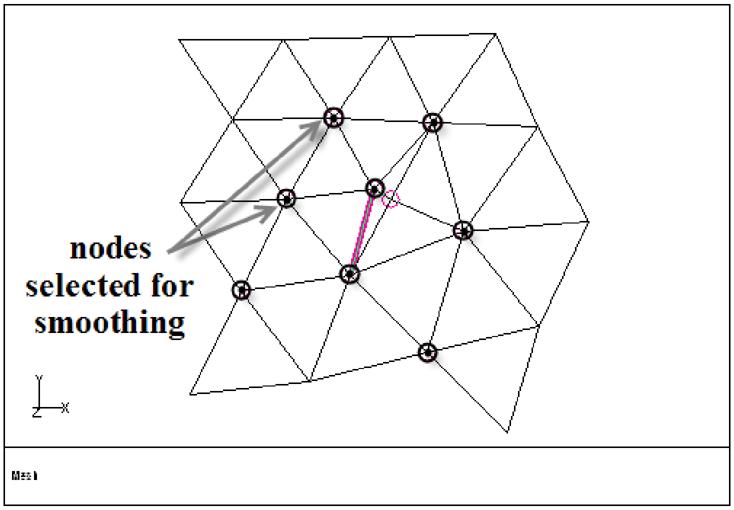


Figure 1.14: Face to be Modified with Node Smoothing

(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).*

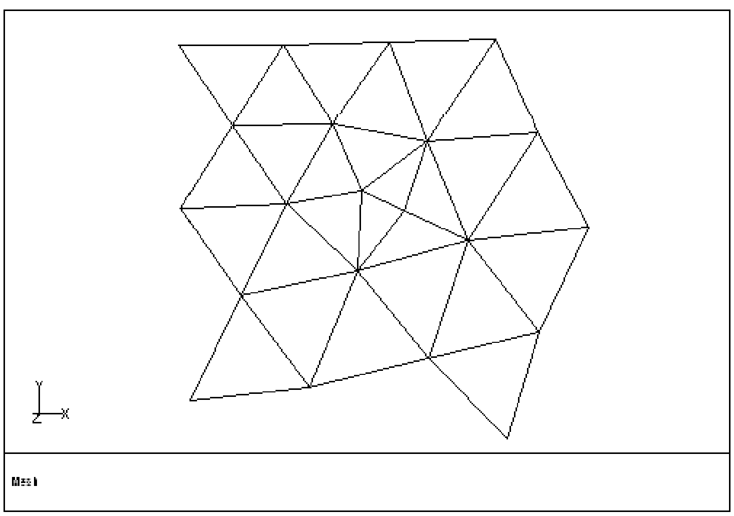


Figure 1.15: Surface Mesh After Node Smoothing

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).

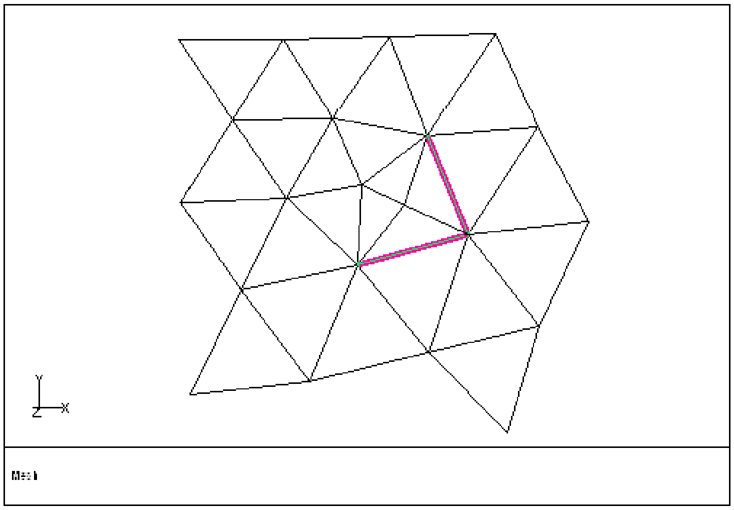


Figure 1.16: Edges Selected for Swapping

   (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.*

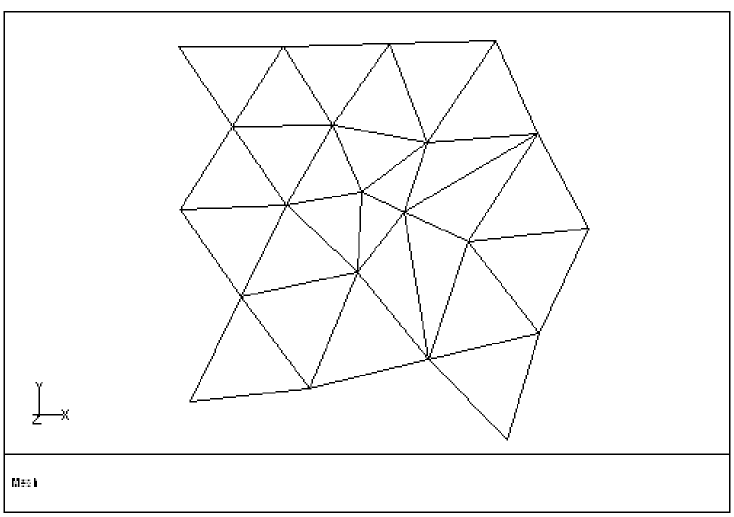


Figure 1.17: Surface Mesh After Edge Swapping

   (d) Select node in the Filter list.

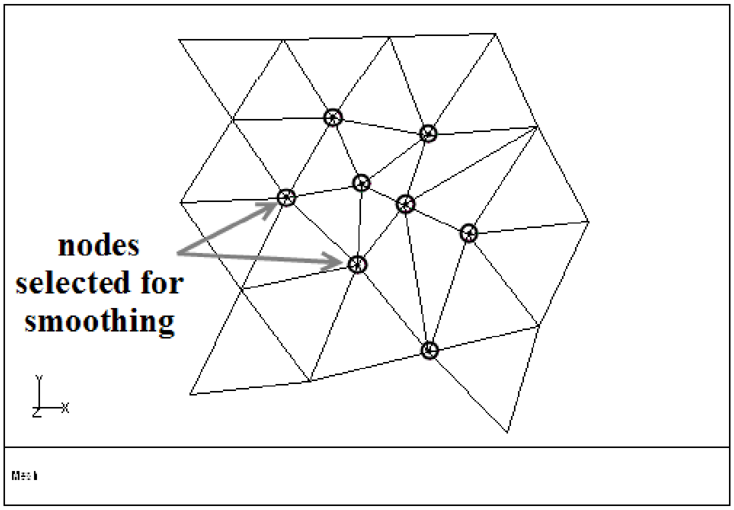


Figure 1.18: Nodes Selected for Smoothing

(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).

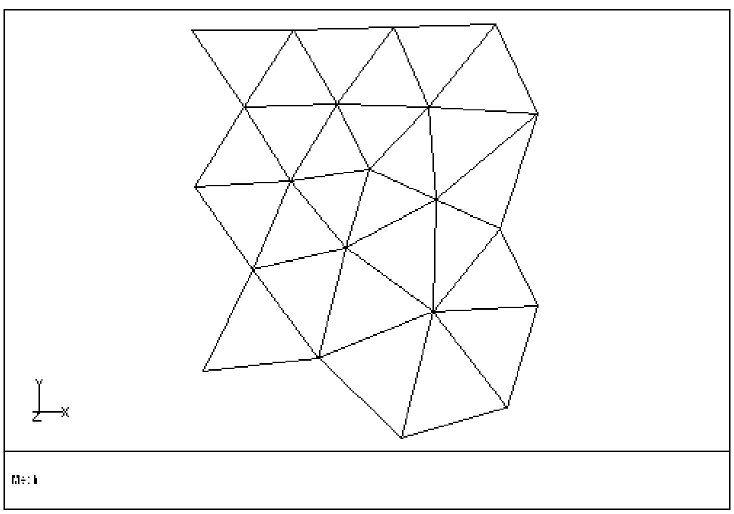


Figure 1.19: Surface Mesh After Node Smoothing

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).

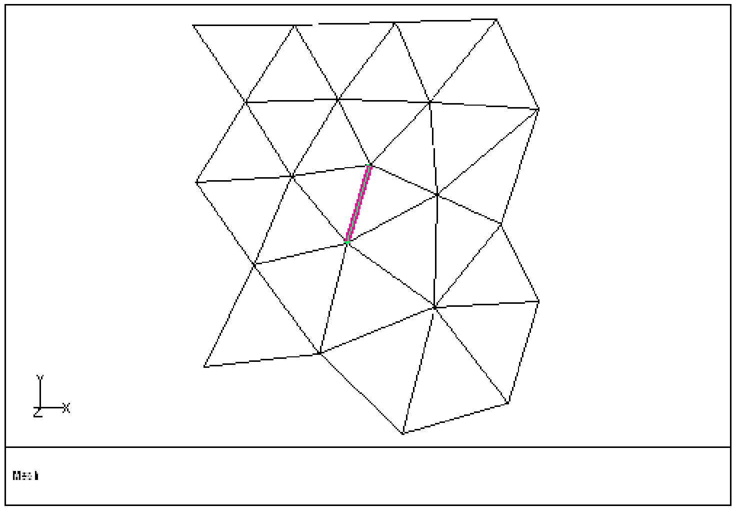


Figure 1.20: Edge Selected for Splitting

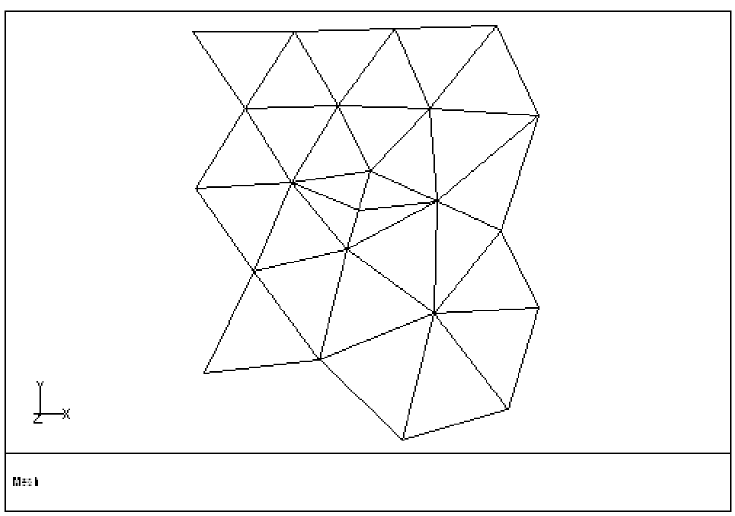


Figure 1.21: Surface Mesh After Edge Splitting

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

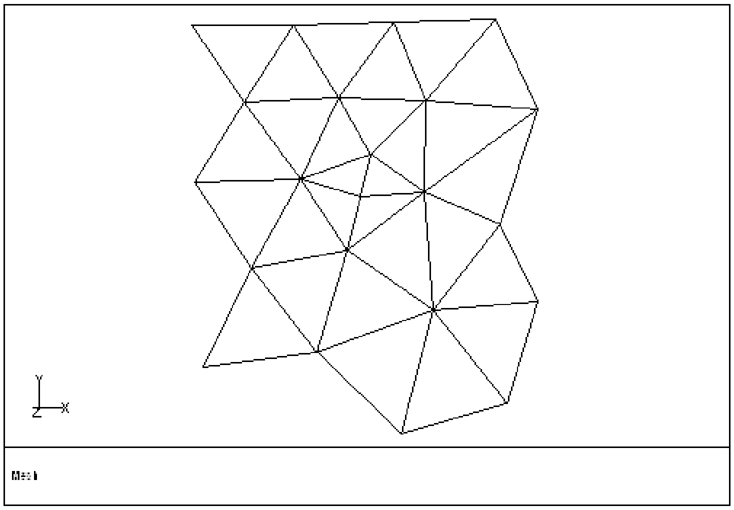


Figure 1.22: Surface Mesh After Node Smoothing

6. Modify the mesh by splitting faces.

   (a) Select face in the Filter list.

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

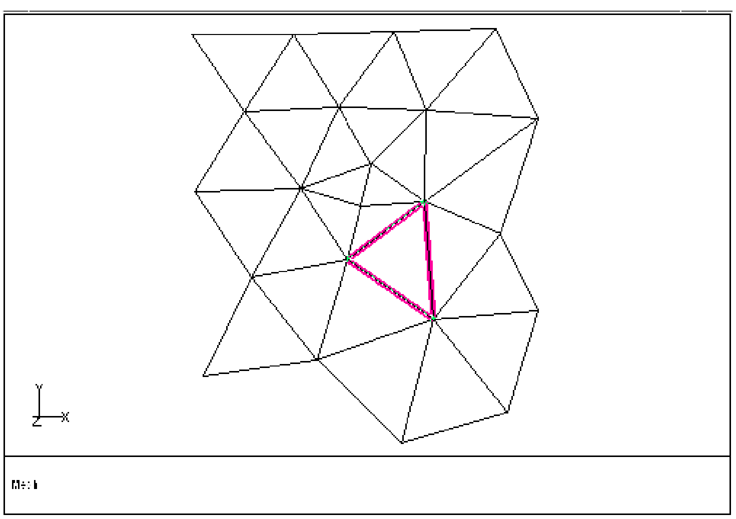


Figure 1.23: Face Selected for Splitting

   (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).

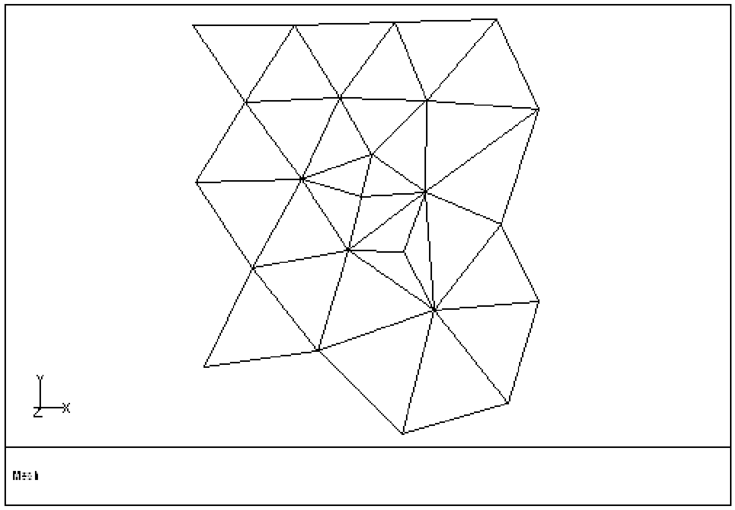


Figure 1.24: Surface Mesh After Splitting the Face

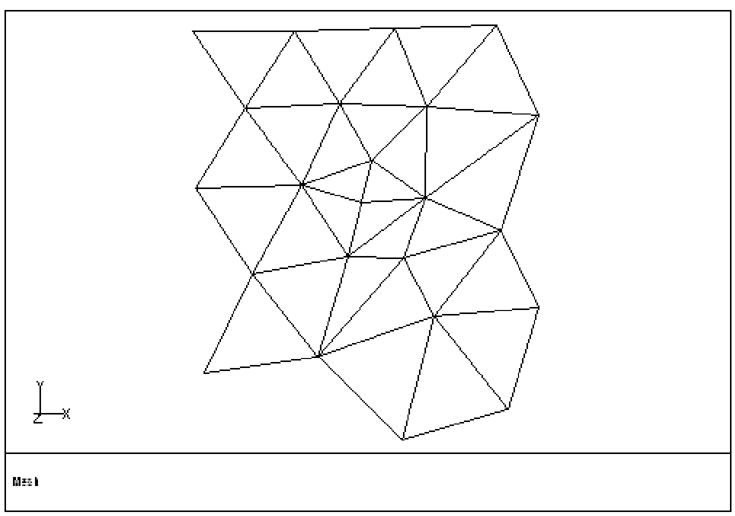


Figure 1.25: Surface Mesh After Edge Swapping

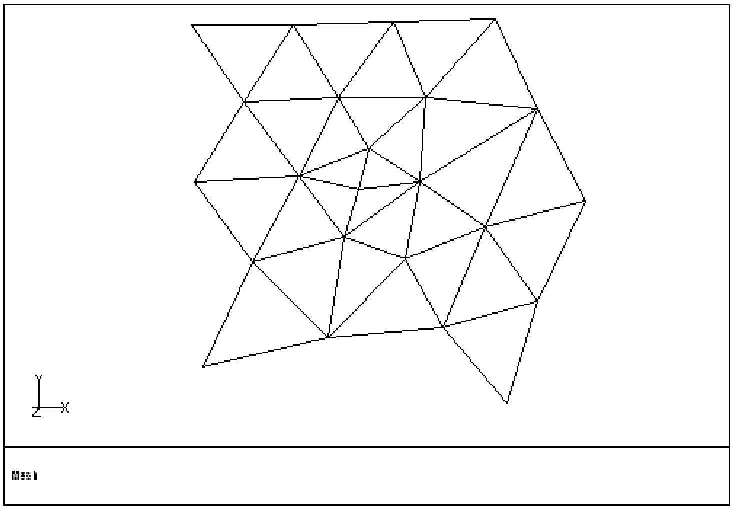
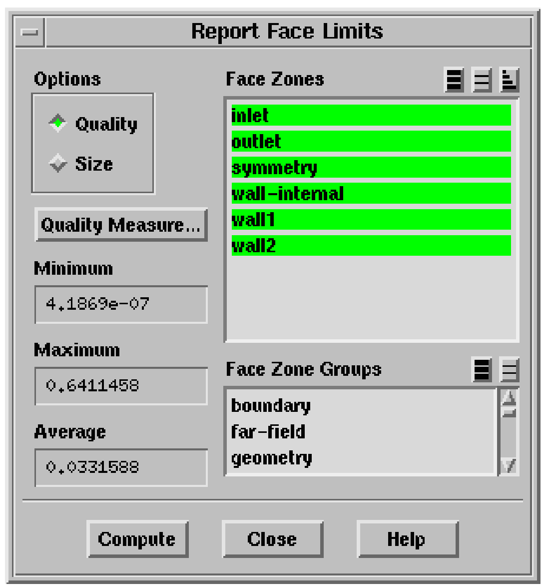


Figure 1.26: Surface Mesh After Node Smoothing

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.

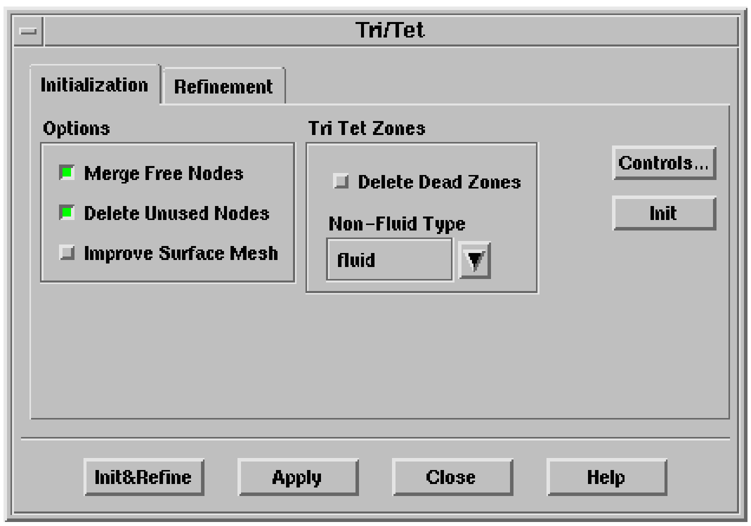
The maximum face skewness at this point in the tutorial is less than 0.65. There are nine faces with a skewness greater than 0.6 (this information was obtained from the Face Distribution *panel). You can try and reduce the maximum face skewness to a value less than 0.6 using the face modification tools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

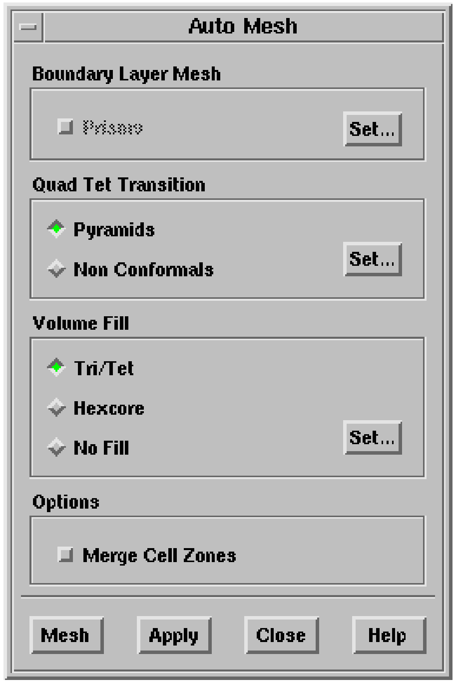
   By default, TGrid *automatically makes the cell zone with the largest volume the active fluid zone.* TGrid *treats the remaining cell zones (non-fluid zones) as dead zones and does not refine them. Hence, if you want to mesh multiple zones, change the* Non-Fluid Type *to* solid *or* fluid *depending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

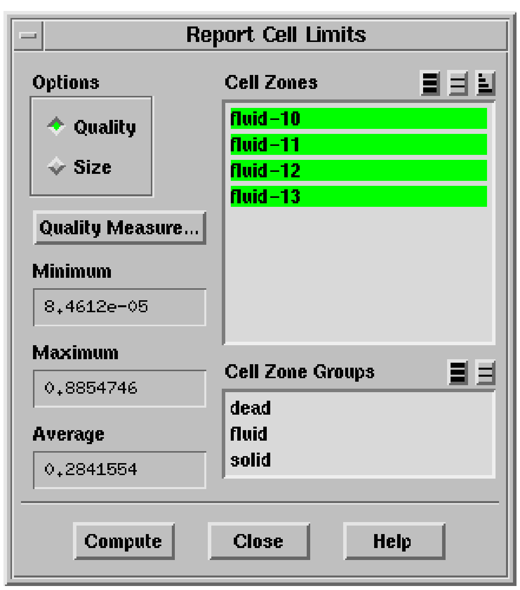


   (a) Retain the default settings and click Mesh.

   (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.

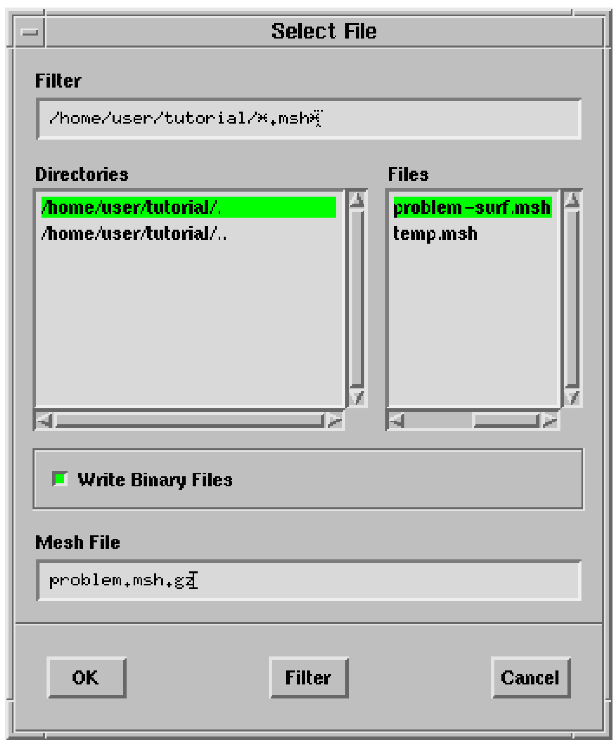
   Check the mesh to ensure it has no negative cell volumes or left-handed faces before saving the mesh file.

   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.