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`.

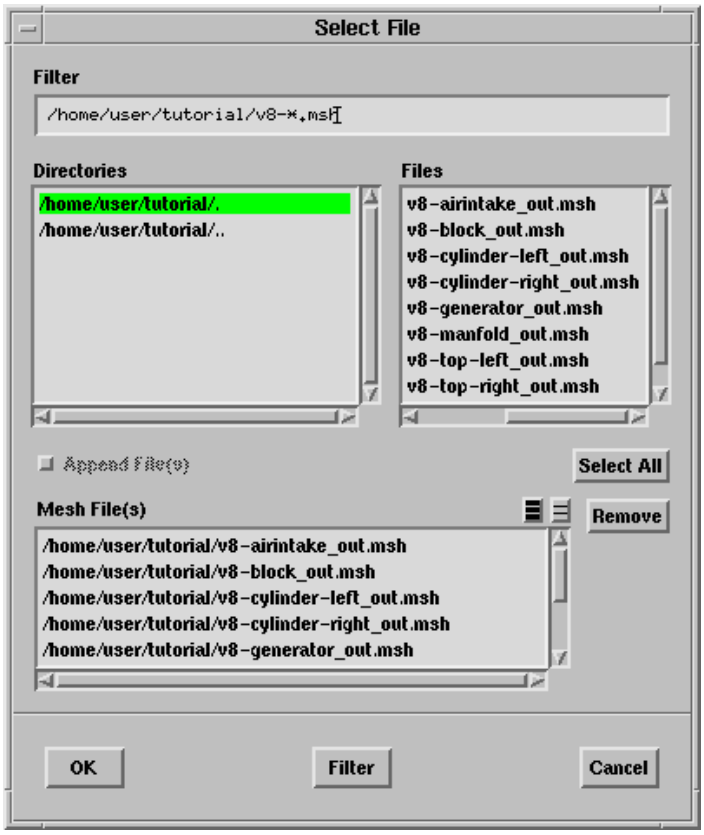
   The files `v8-airintake_out.msh`, `v8-block_out.msh`, `v8-cylinder-left_out.msh`, `v8-cylinder-right_out.msh`, `v8-generator_out.msh`, `v8-manfold_out.msh`, `v8-top-left_out.msh`, `v8-top-right_out.msh`, *and* `v8-wheels_out.msh` *can be found in the* `v8` *folder created on unzipping the file.*

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.*

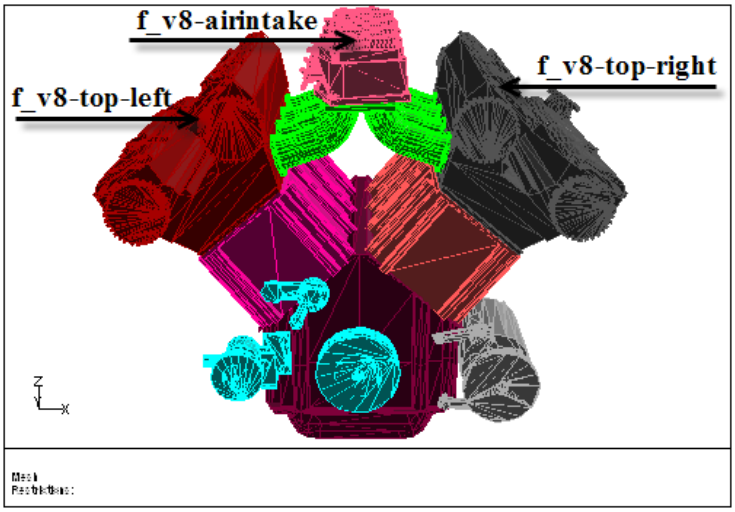


Figure 7.1: Grid Display

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.

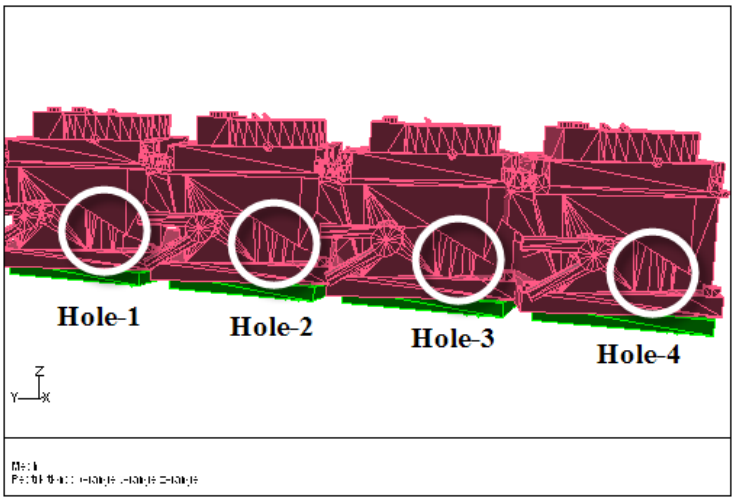


Figure 7.2: Holes to be Closed in the f_v8-airintake Zone

   (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.*

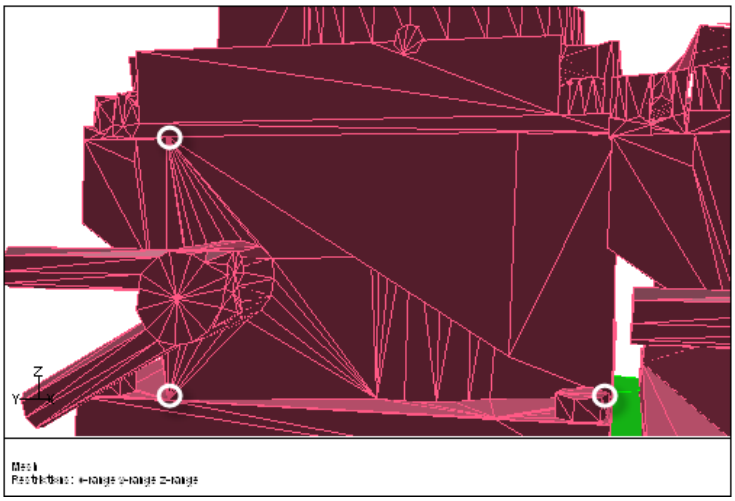


Figure 7.3: Nodes Selected to Close Hole-1 in f_v8-airintake

(h) Click Create in the Operation group box.

OR

Use F5.

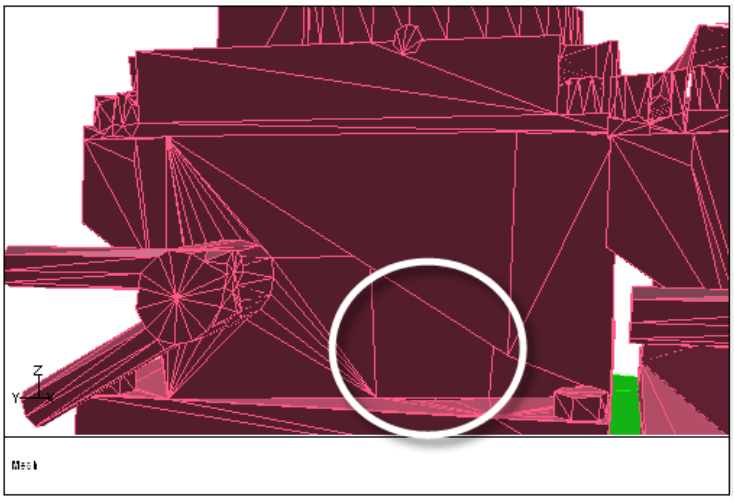


Figure 7.4: Closed Hole (Hole-1) in the f_v8-airintake Zone

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.

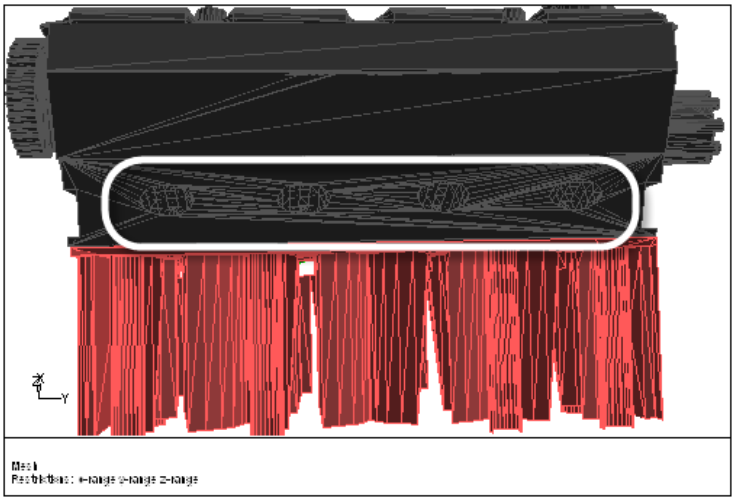


Figure 7.5: Coplanar Holes in the f_v8-top-right Zone

   (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.*

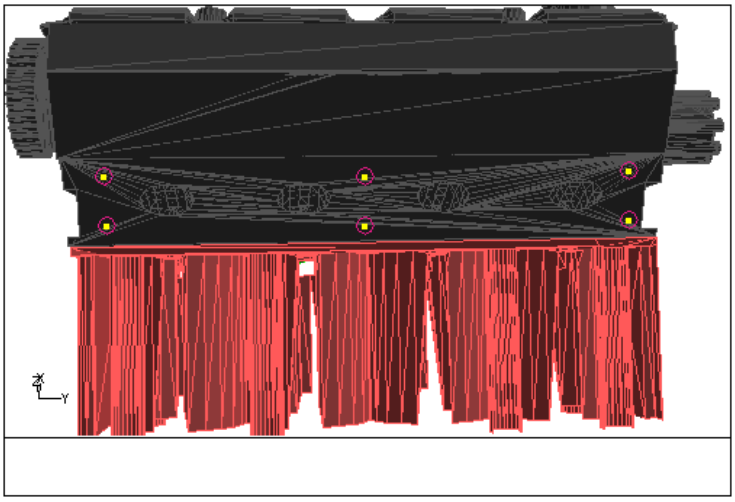


Figure 7.6: Nodes Selected to Close Holes in the f_v8-top-right Zone

(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).

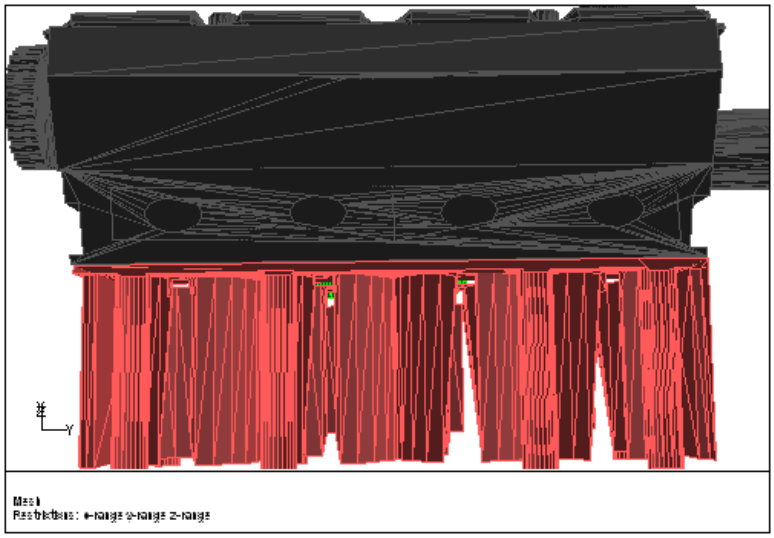


Figure 7.7: Holes Closed in the f_v8-top-right Zone

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.

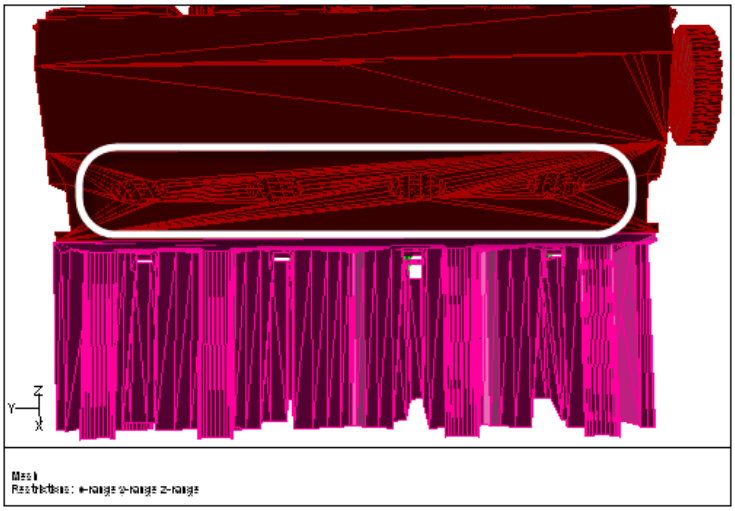
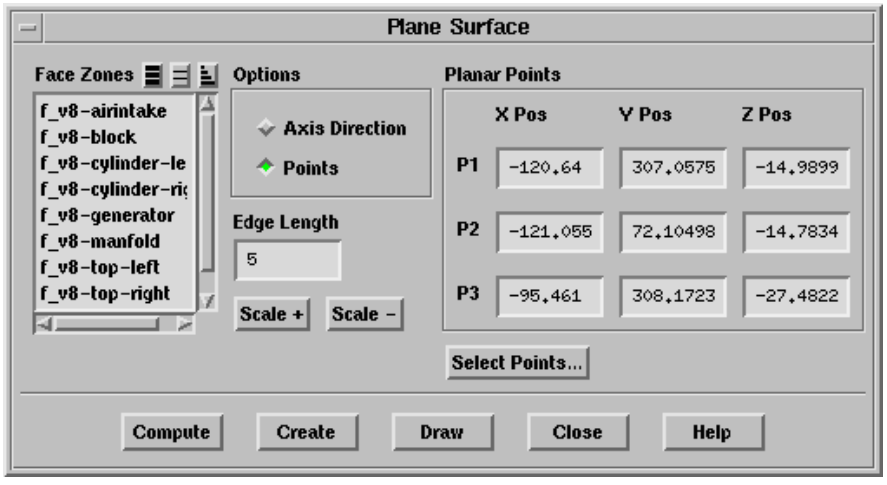


Figure 7.8: Coplanar Holes in the f_v8-top-left Zone

(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).

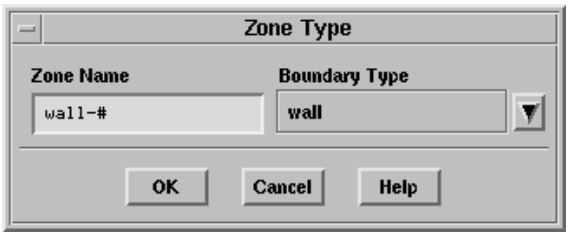
Select the two points along the longest side first. The coordinates of the three selected points will be updated in the Planar Points *group box.*

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.*

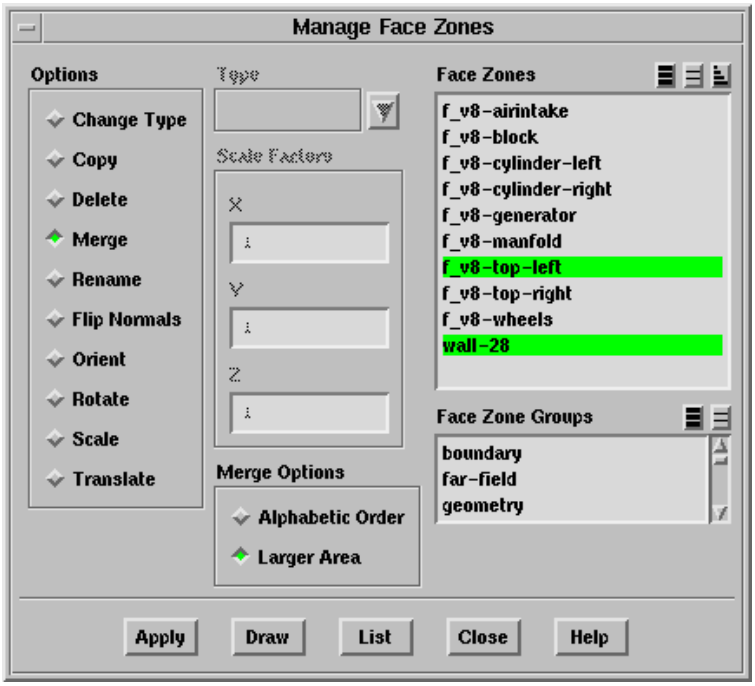
vii. Close the Plane Surface panel.

(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).

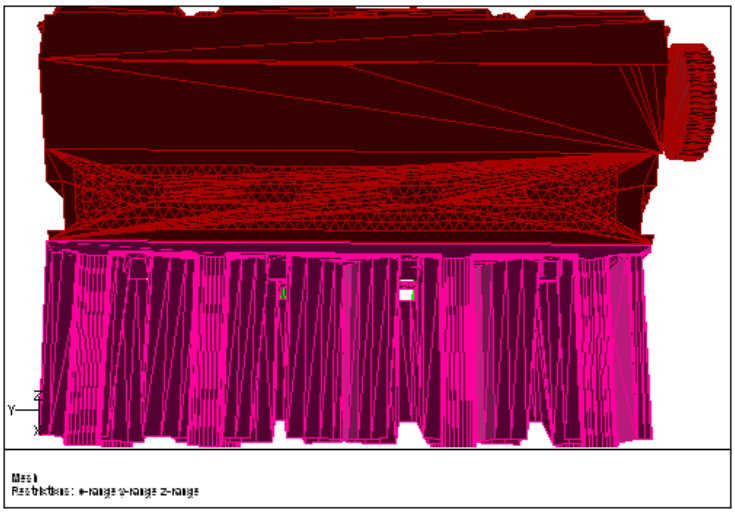


Figure 7.10: Holes Closed in the f_v8-top-left Zone Using a Plane Surface

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

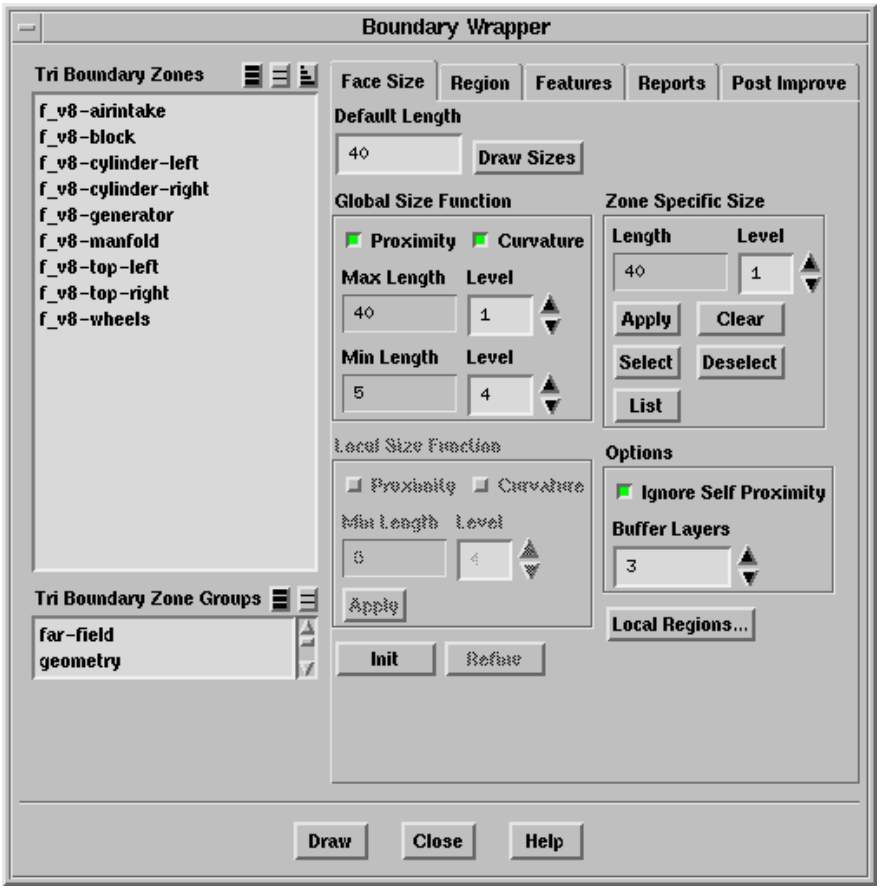
   File ⟶ Write ⟶ Mesh...

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.

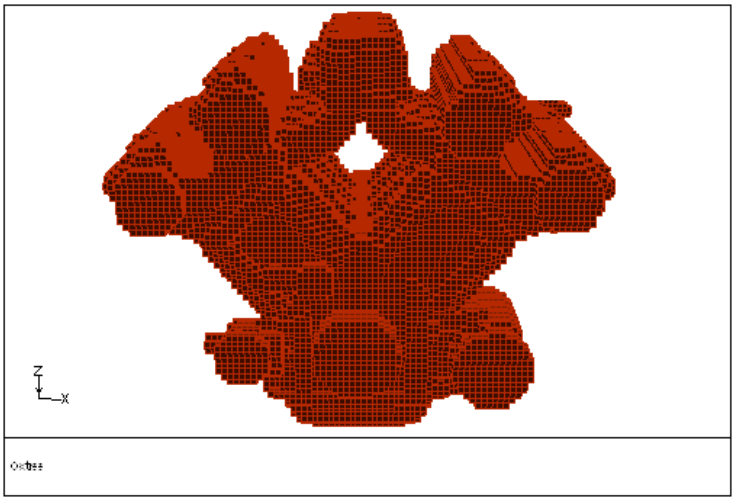
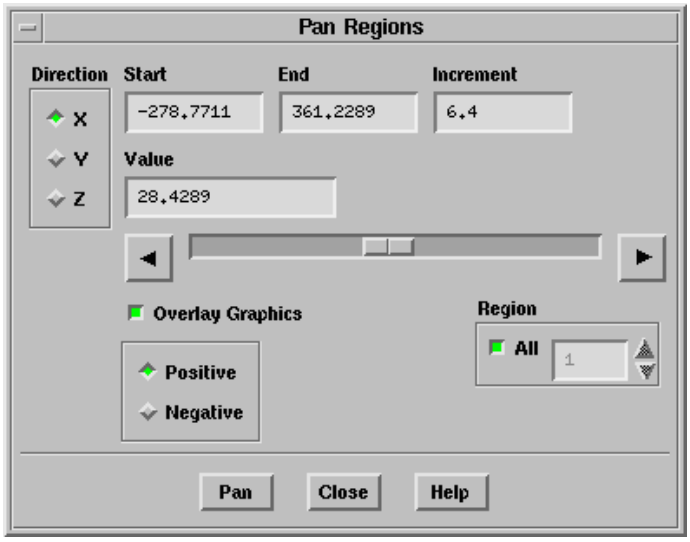


Figure 7.11: Region to be Wrapped

4. Pan the region to examine grid distribution and to search for holes.
   (a) Display the entire geometry.

   Display ⟶ 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 ≈ 28).

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

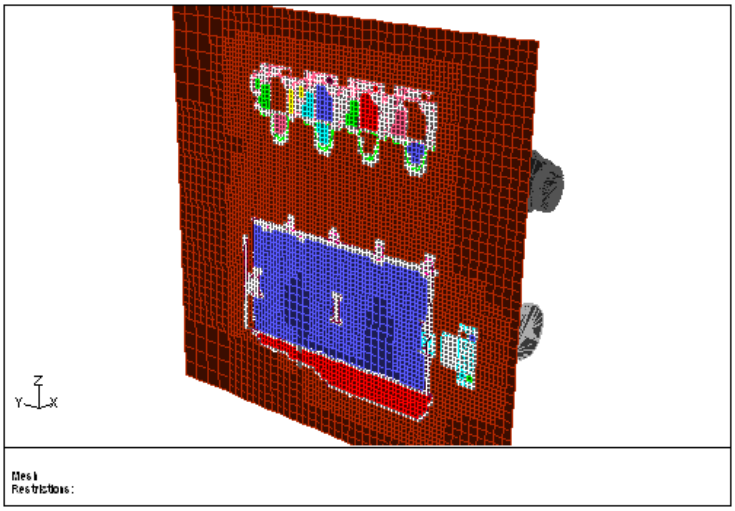


Figure 7.12: Pan Plane with the Overlaid Geometry

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

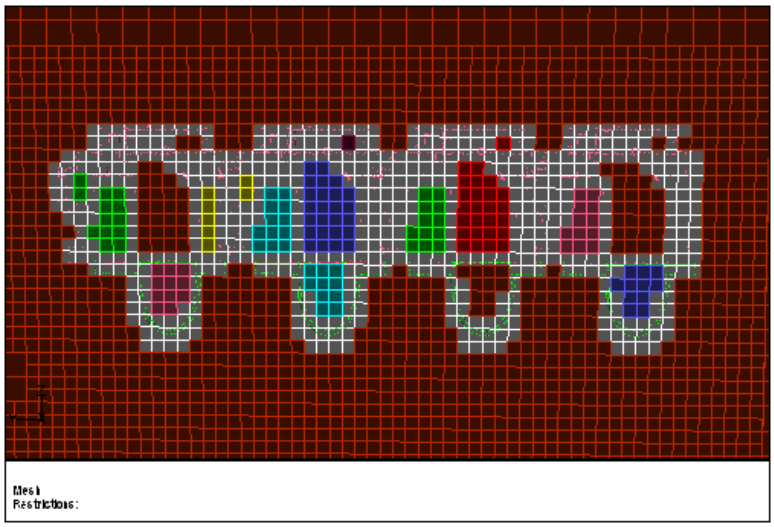


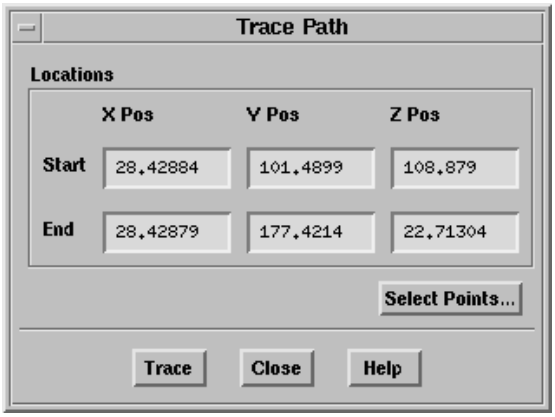
Figure 7.13: Region of the Leak

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.

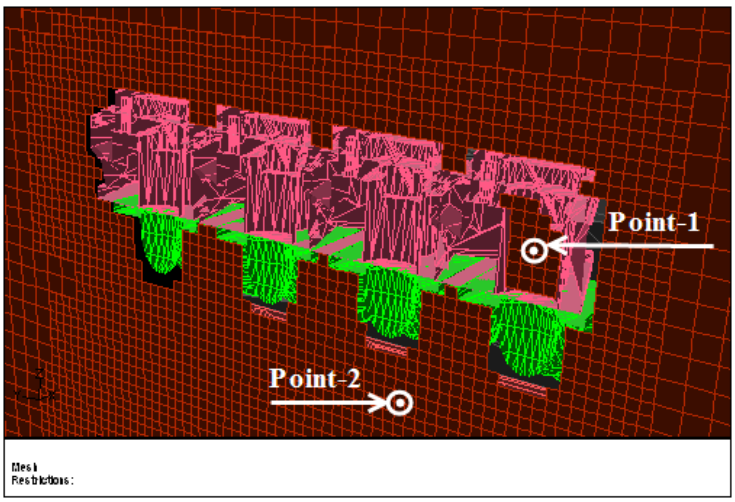


Figure 7.14: Detecting the Leakage using the Trace Path Panel

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.

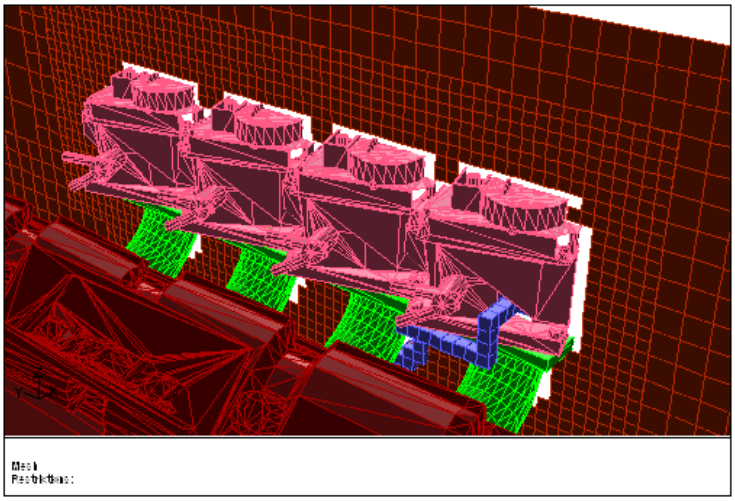


Figure 7.15: Display of Geometry with the Traced Path

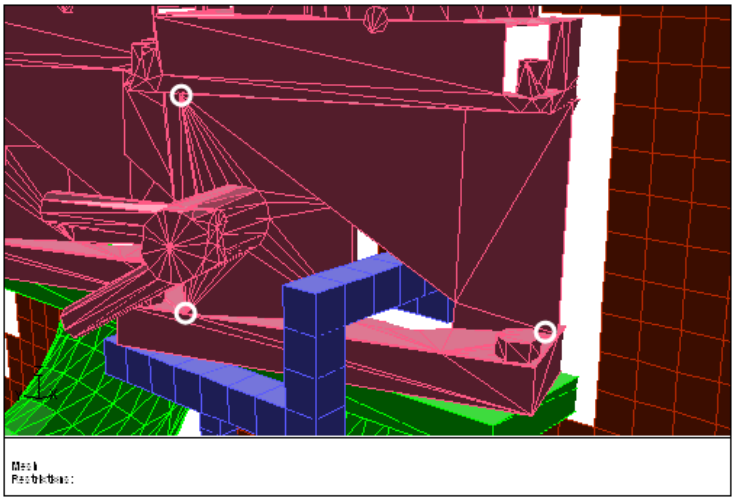


Figure 7.16: Nodes Selected to Close the Hole in the f_v8-airintake Zone

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).

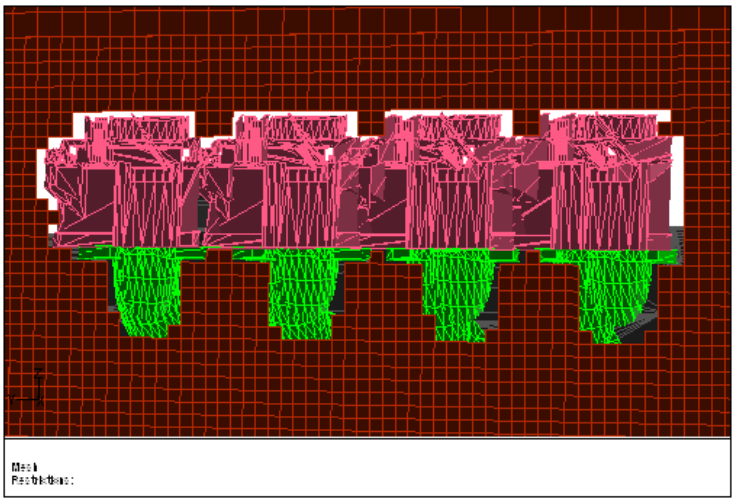


Figure 7.17: Region Updated for the Closed Hole

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.

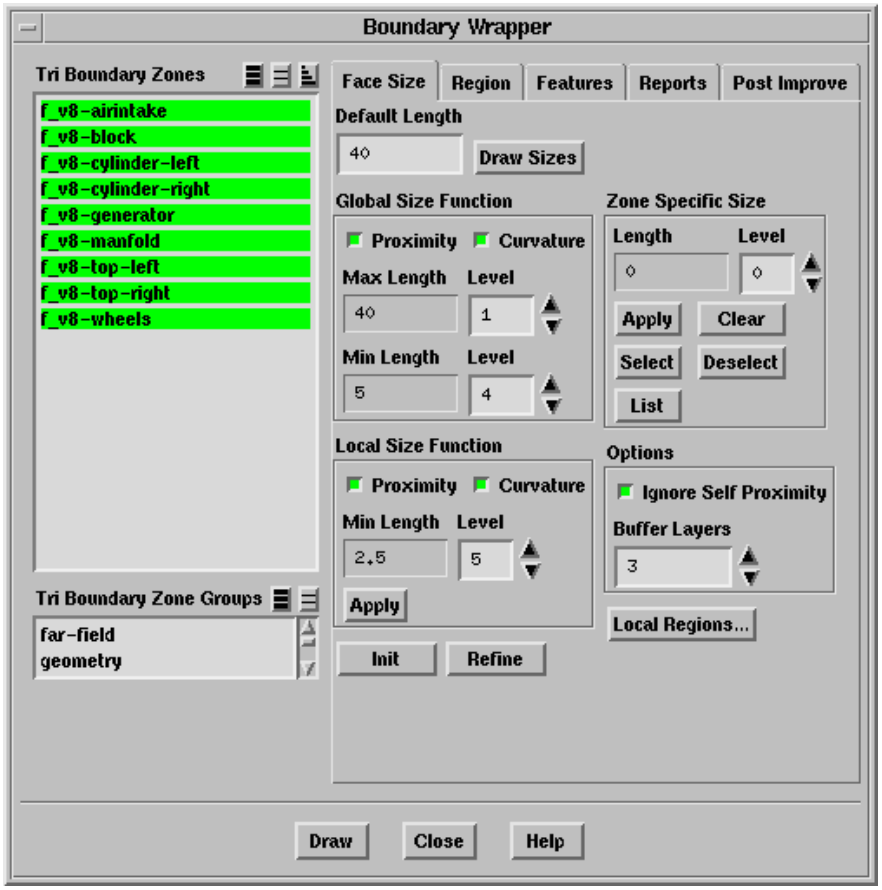
7. Close the Pan Regions and Trace Path panels.

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.*

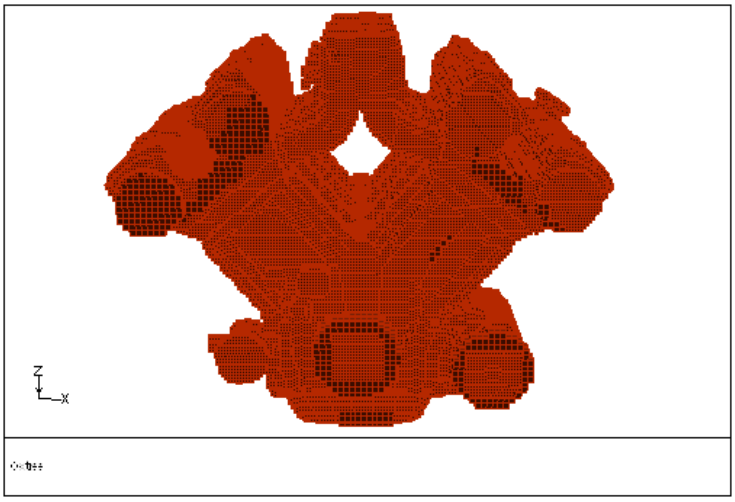
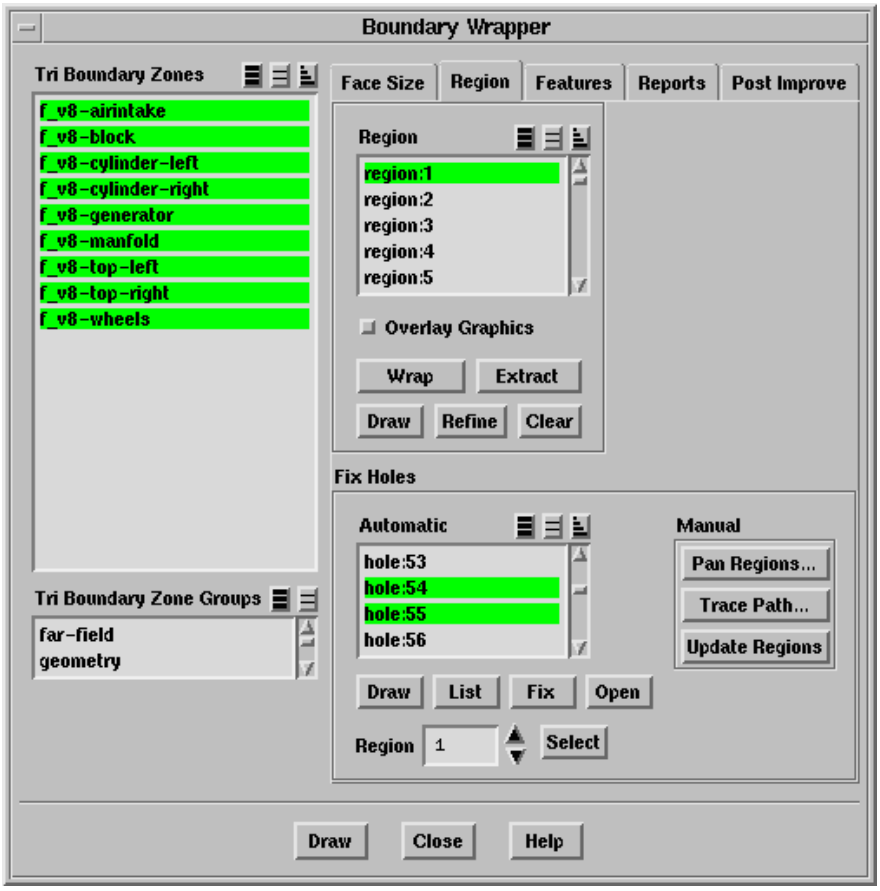


Figure 7.18: Region After Refining Using Local Size Function



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).

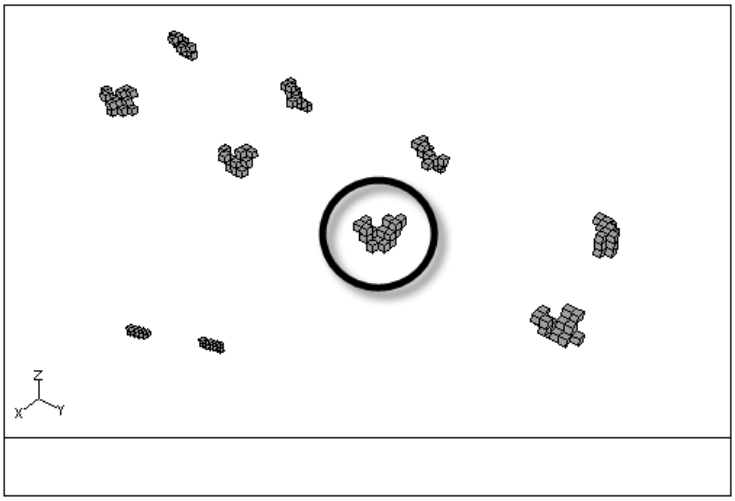


Figure 7.19: Potential Holes in region:1

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).

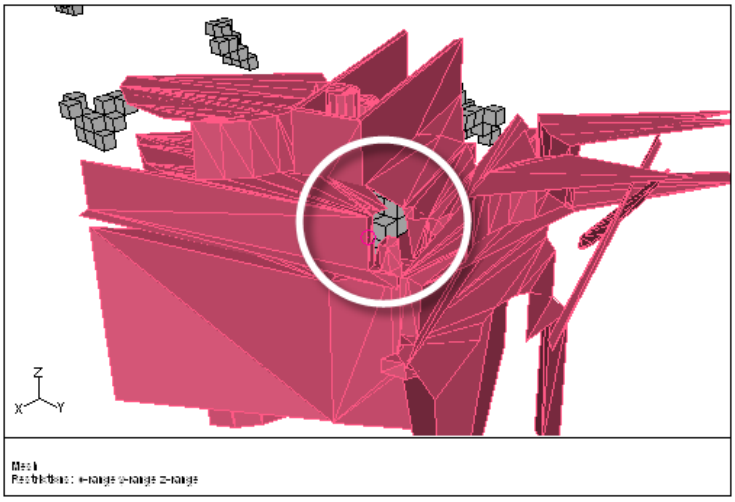


Figure 7.20: Bounds Set Around a Potential Hole

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).

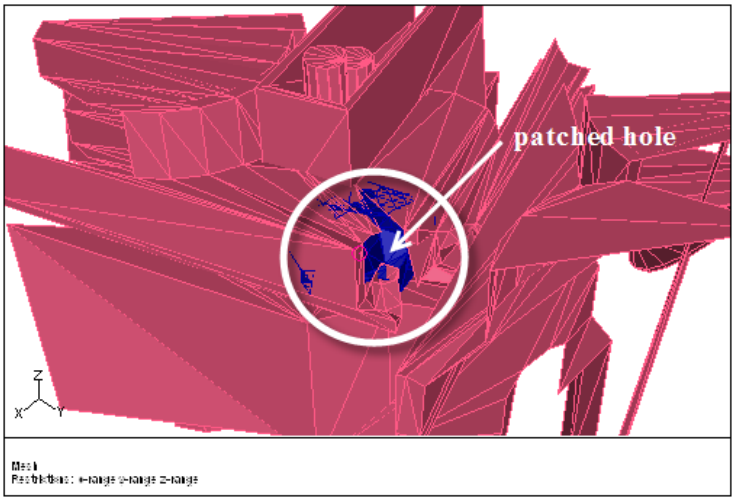


Figure 7.21: Hole Fixed Using the Automatic Hole Fixing Option

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.*

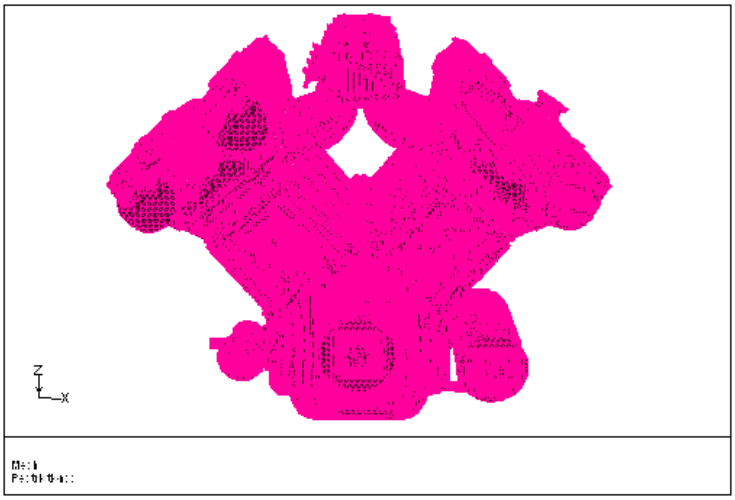


Figure 7.22: Wrapper Surface

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).

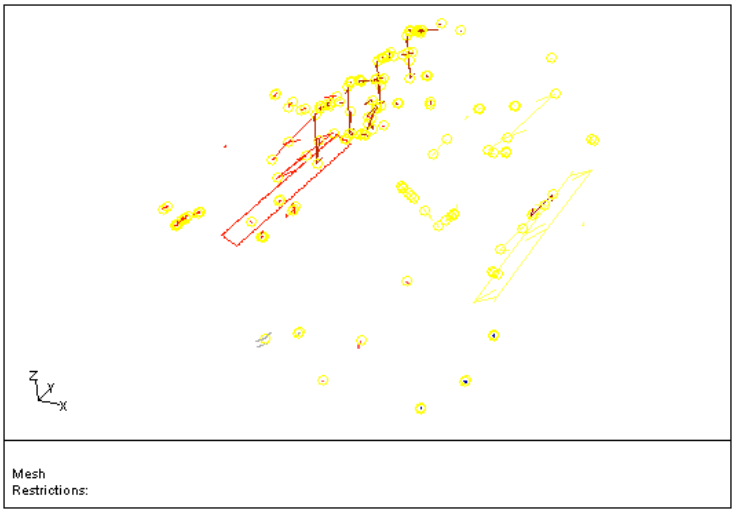


Figure 7.23: Insignificant Features to be Deleted

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.

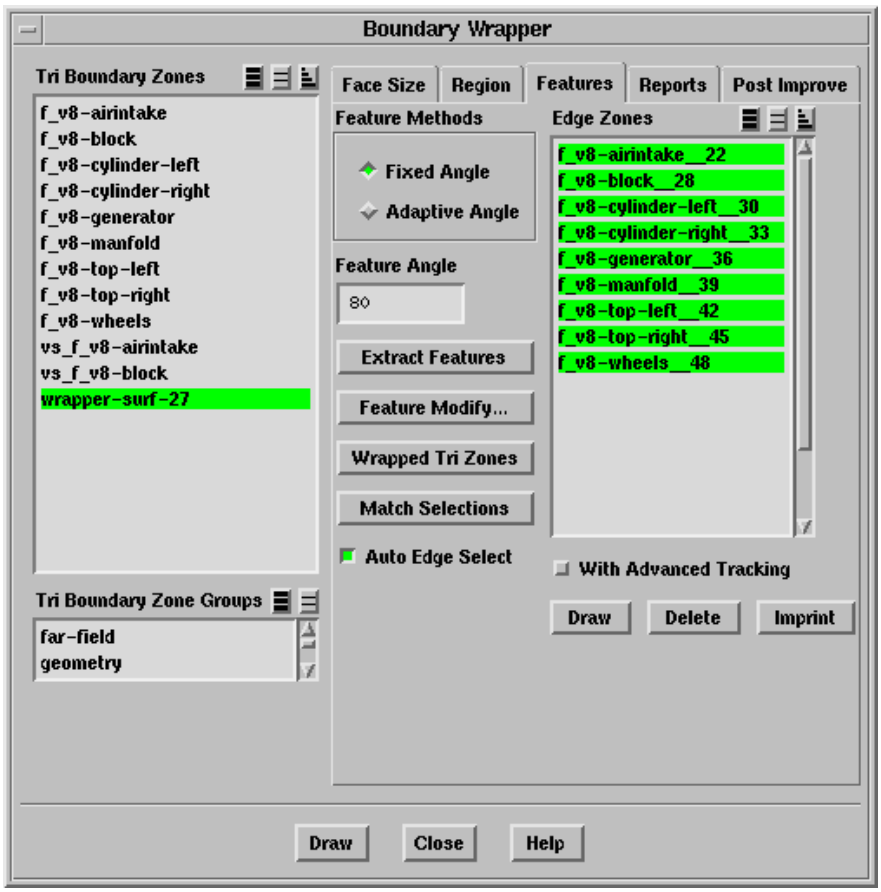
   Zoom in to the region shown in Figure 7.24.

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.

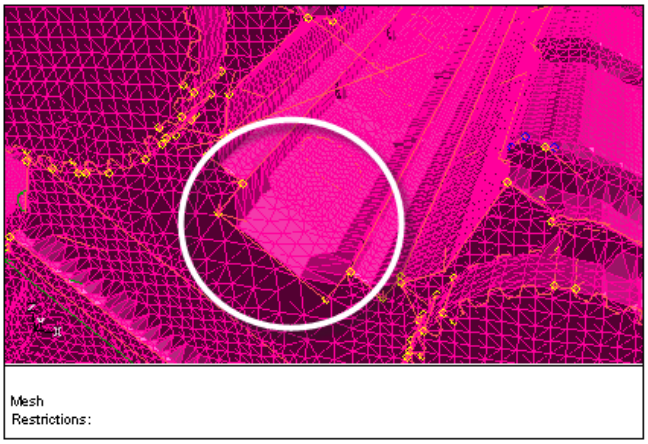
14. Display the wrapper surface to see the effect of imprinting.

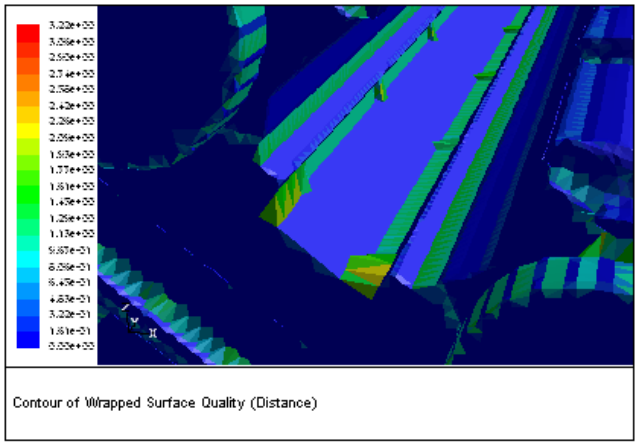
15. Click Draw Contours in the Reports tab of the Boundary Wrapper panel.

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.

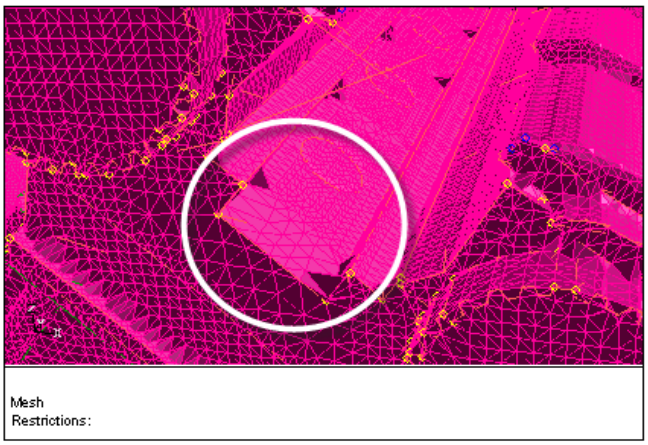
17. Enable With Advanced Tracking and click Imprint again.

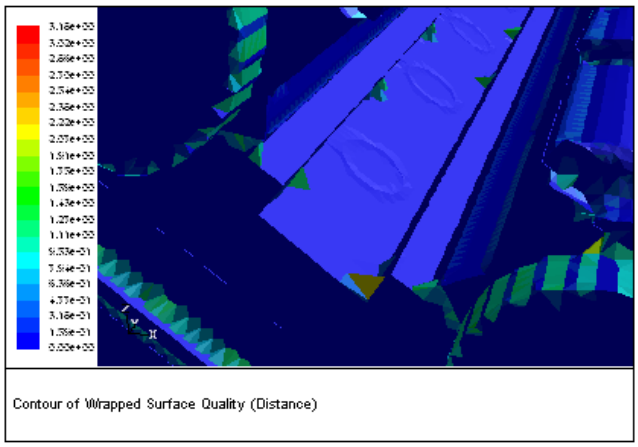
    Figure 7.24 shows the wrapper surface and contours of distance before and after imprinting.



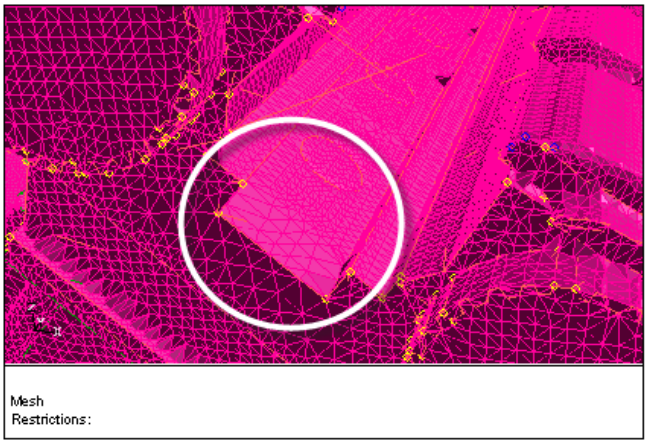


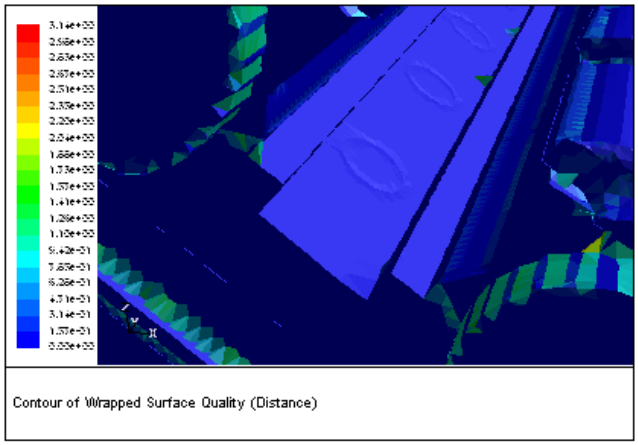
(A) Before Imprinting





(B) After Initial Imprinting





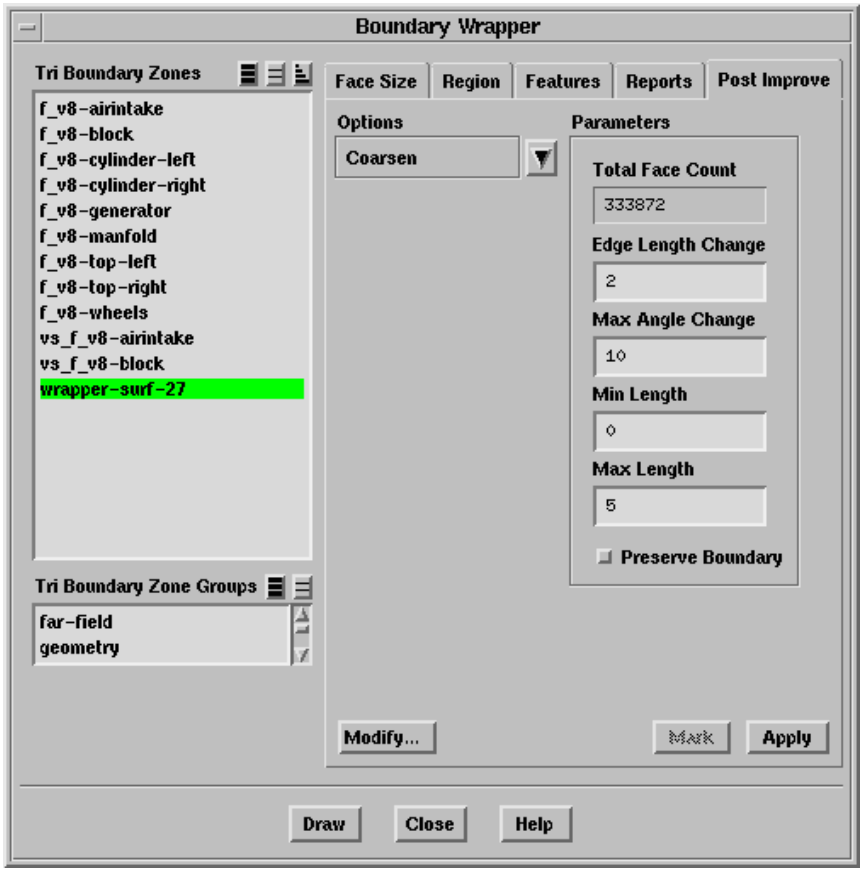
(C) After Imprinting With Advanced Tracking

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`).

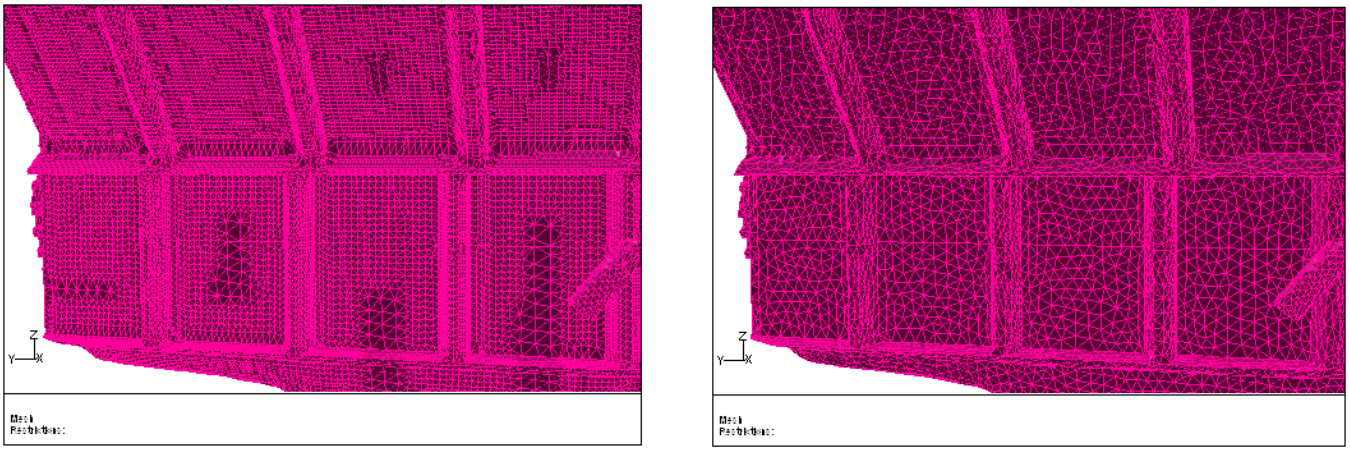
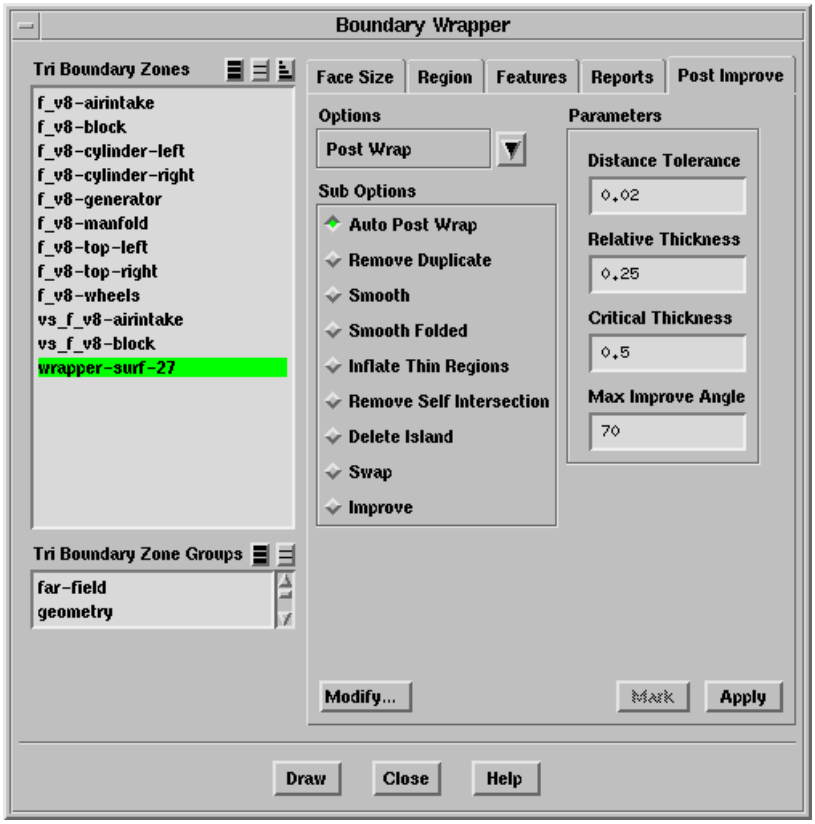


Figure 7.25: Wrapper Surface (A) Before and (B) After Coarsening

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-.

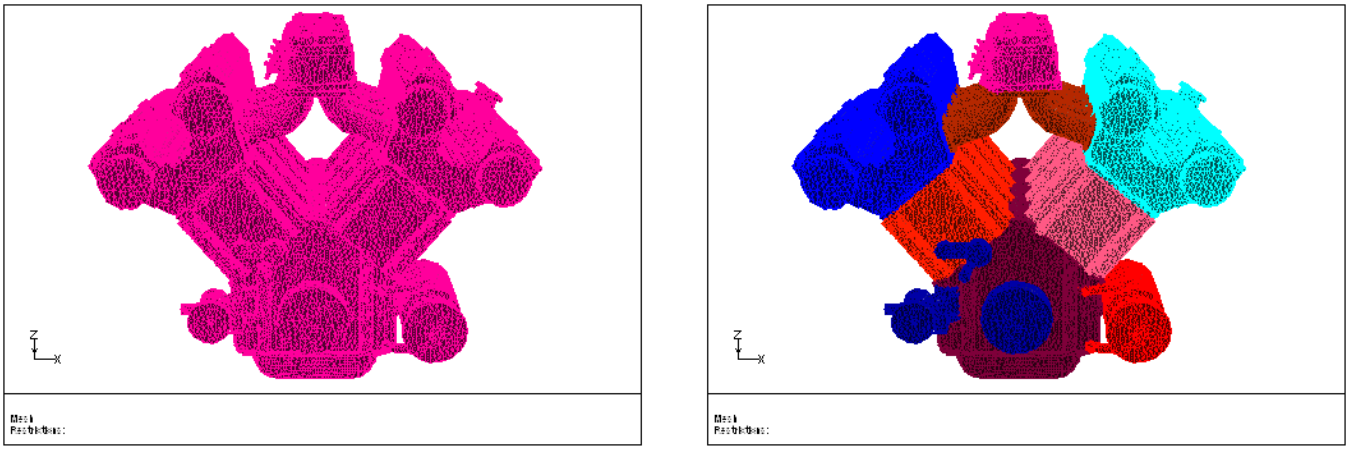
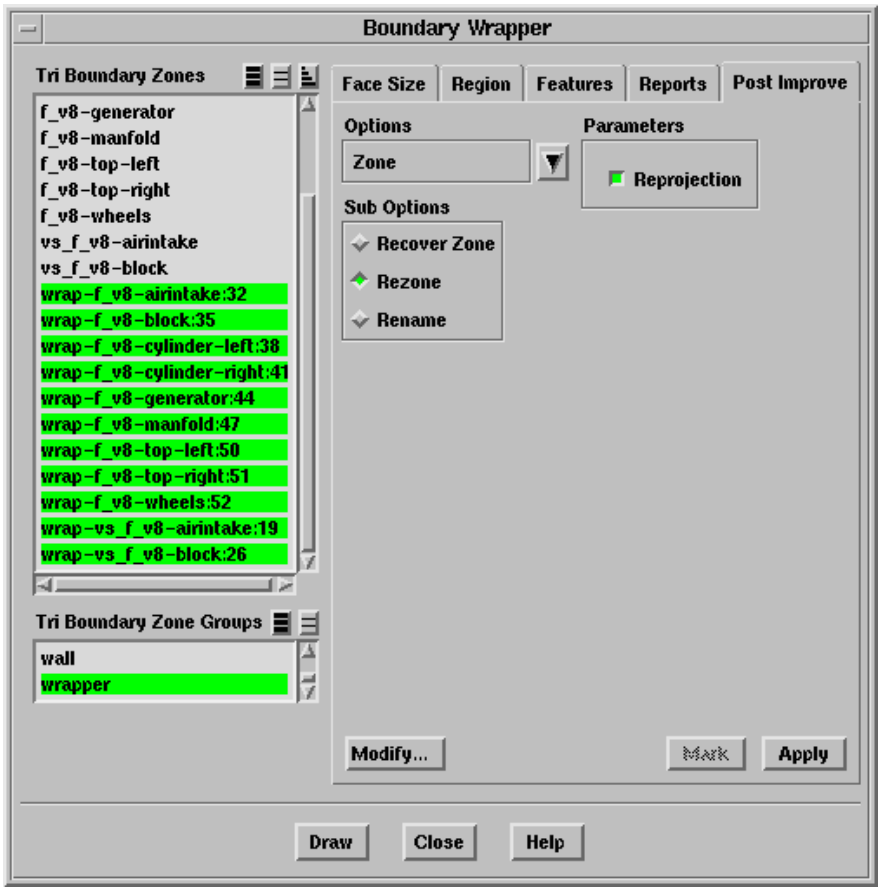


Figure 7.26: Wrapper Surface (A) Before and (B) After Recovering Zones

(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.

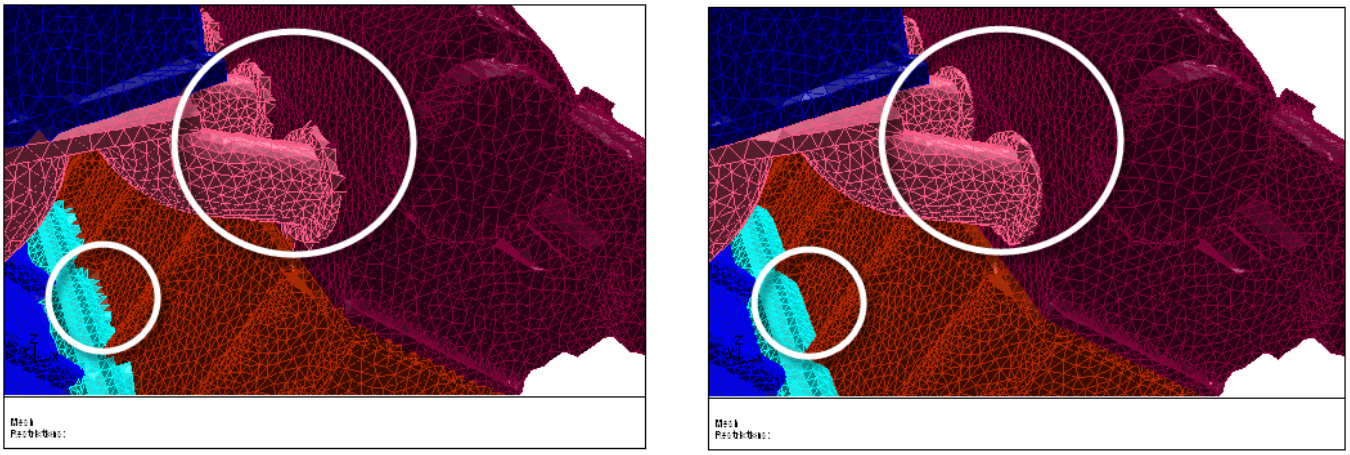


Figure 7.27: Wrapper Surface (A) Before and (B) After Rezoning

(h) Save the mesh (`engine-03.msh.gz`).

6. Merge the small area face zones with the neighboring zones using TUI commands:

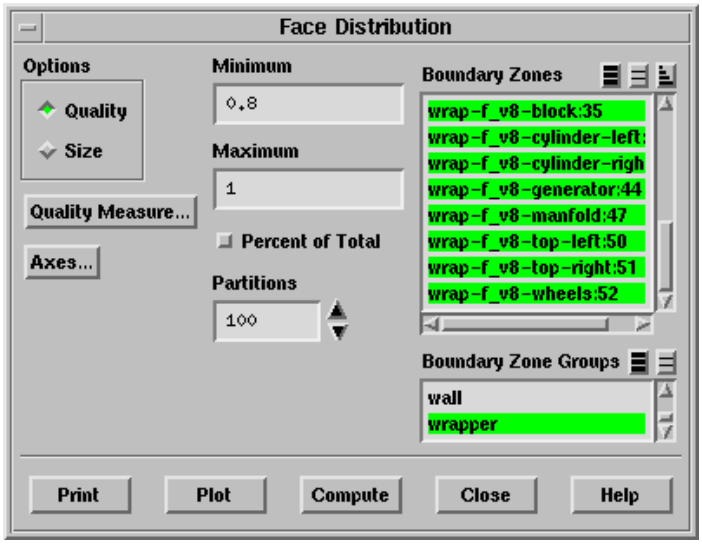
```
> boundary/merge-small-face-zones
Minimum area [0.01] 500

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).

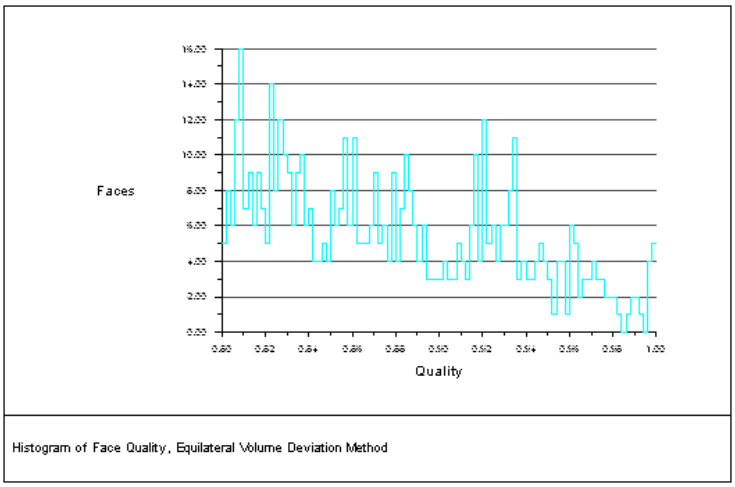
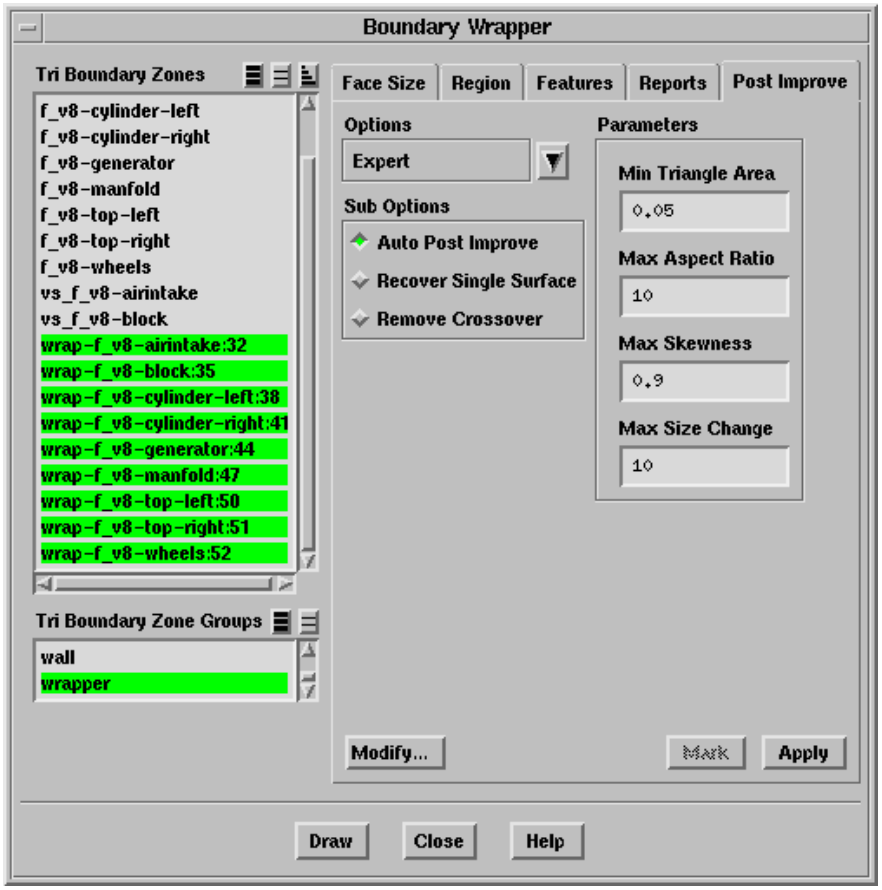


Figure 7.28: Face Skewness Distribution Between 0.8 and 1.0 (Before Auto Post Improve)

(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.



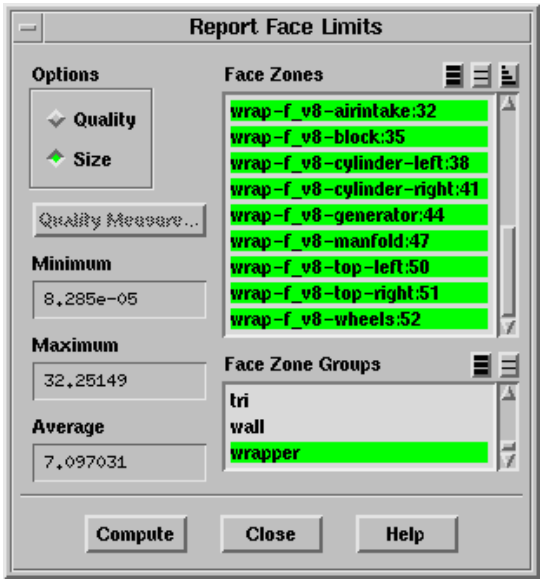
(b) Enter `0.05` for Min Triangle Area.

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.

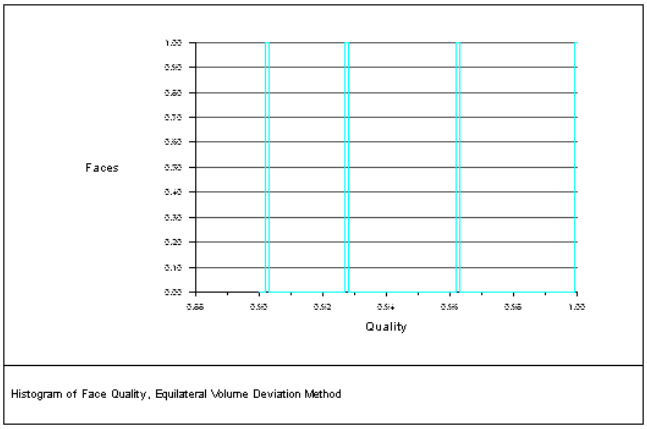


Figure 7.29: Face Skewness Distribution Between 0.9 and 1.0 (After First Auto Post Improve)

(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.

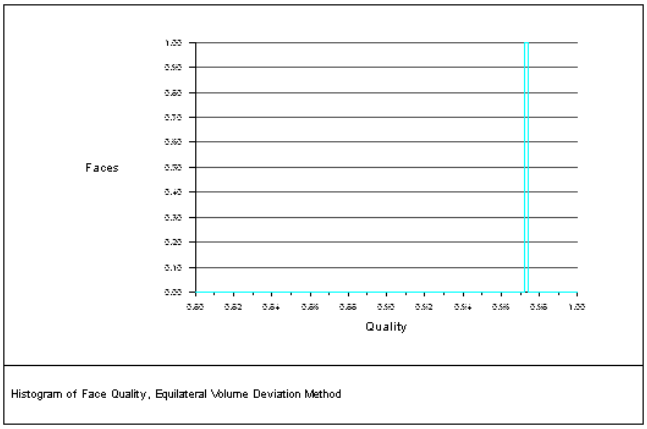


Figure 7.30: Face Skewness Distribution Between 0.8 and 1.0 (After Auto Post Improve)

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.

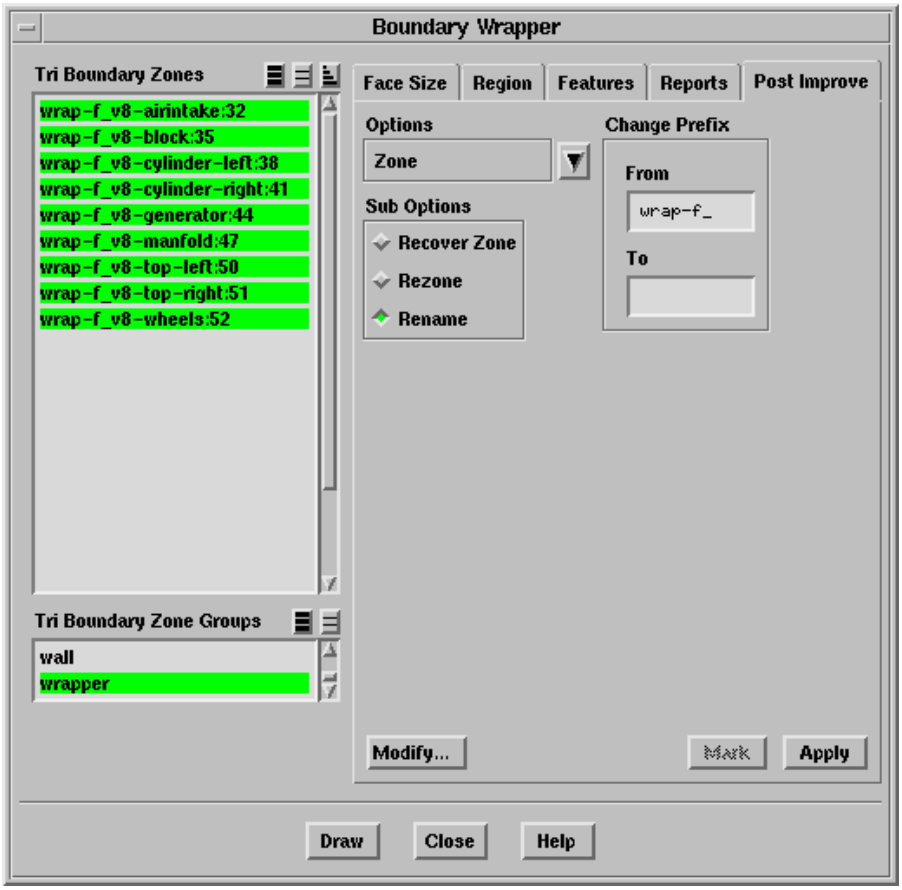
(b) Select Delete in the Options list and click Apply.

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.

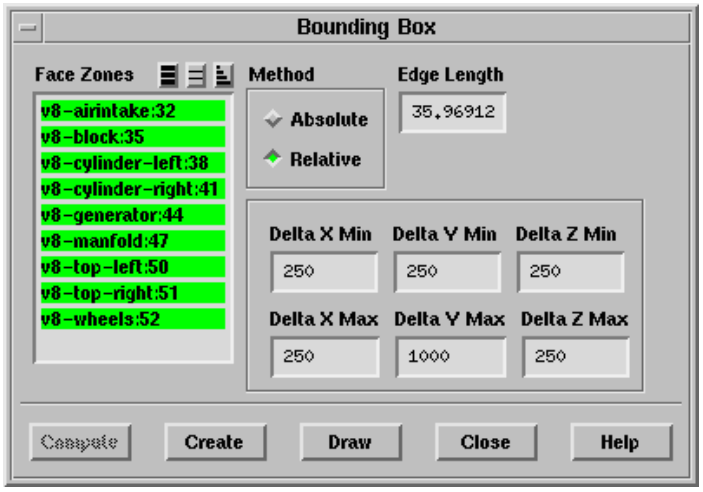
Leave the To *field blank so that the prefix will be removed.*

(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.*

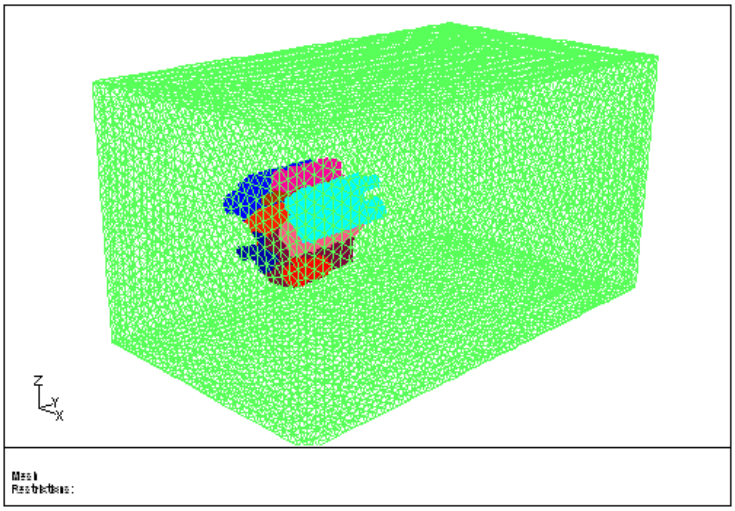
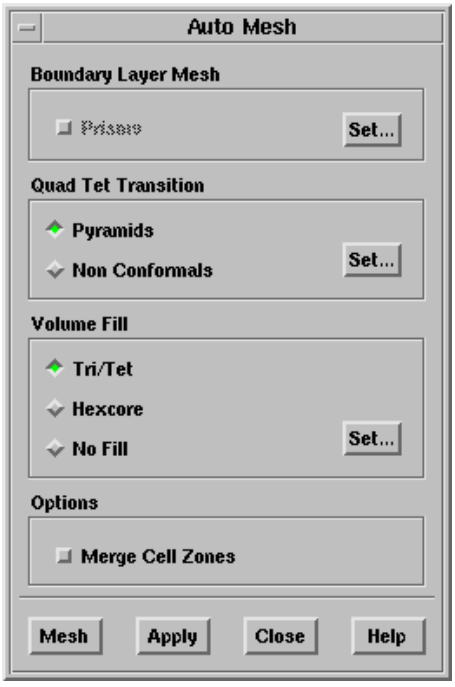


Figure 7.31: Mesh with the Tunnel

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...

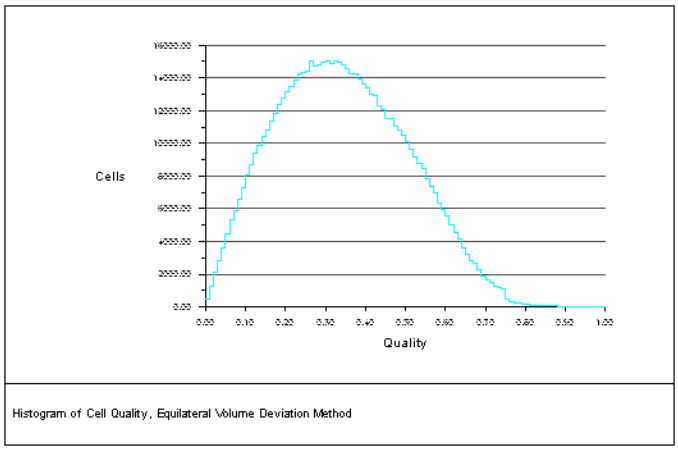


Figure 7.32: Cell Quality Distribution

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

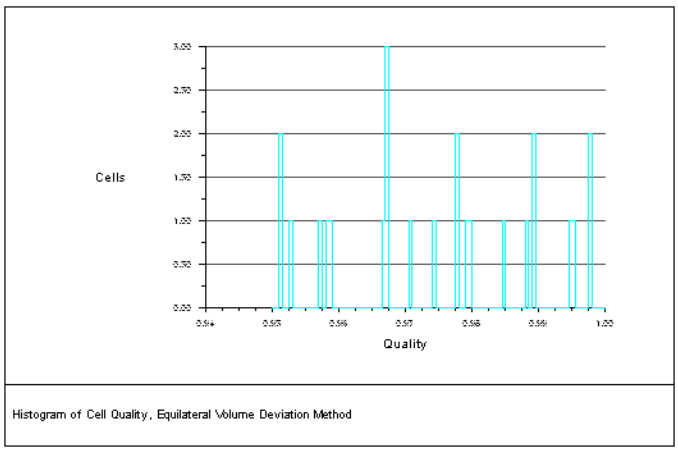
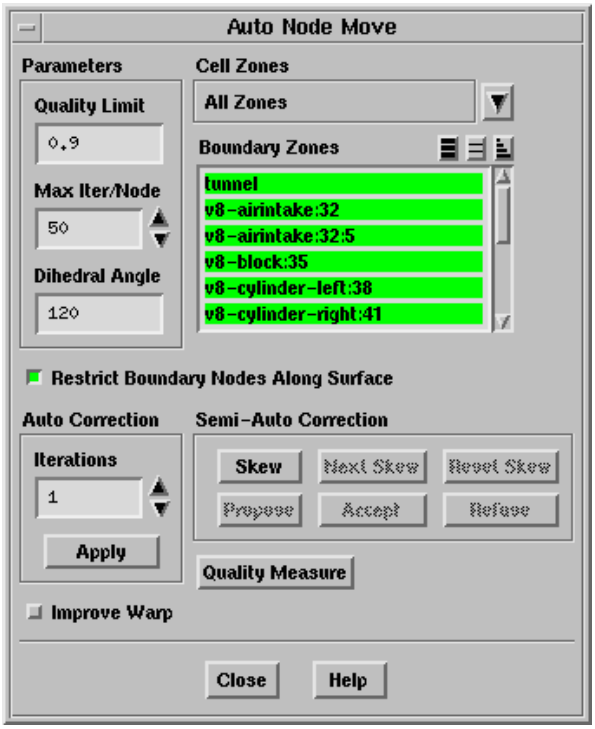


Figure 7.33: Cell Quality Distribution Above 0.95

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...

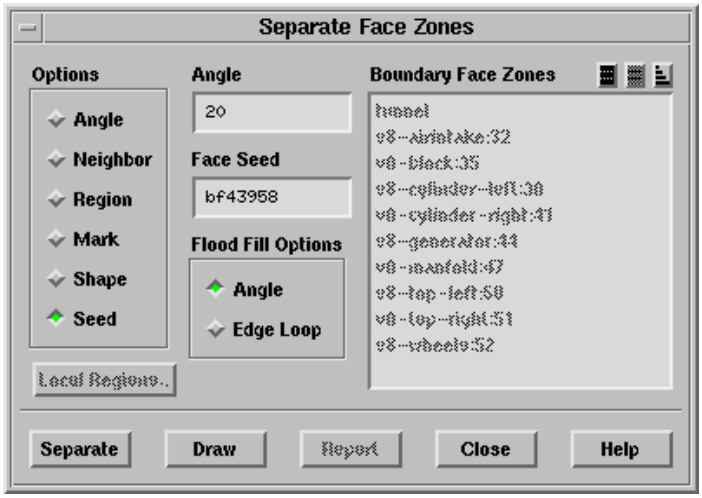


1. Select all the zones in the Boundary Zones selection list.
2. Click Apply in the Auto Correction group box.
3. Check the skewness distribution above 0.95.
4. Enter `0.95` for Quality Limit and `0` for Dihedral Angle, respectively.
5. Disable Restrict Boundary Nodes Along Surface.
6. Click Apply in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

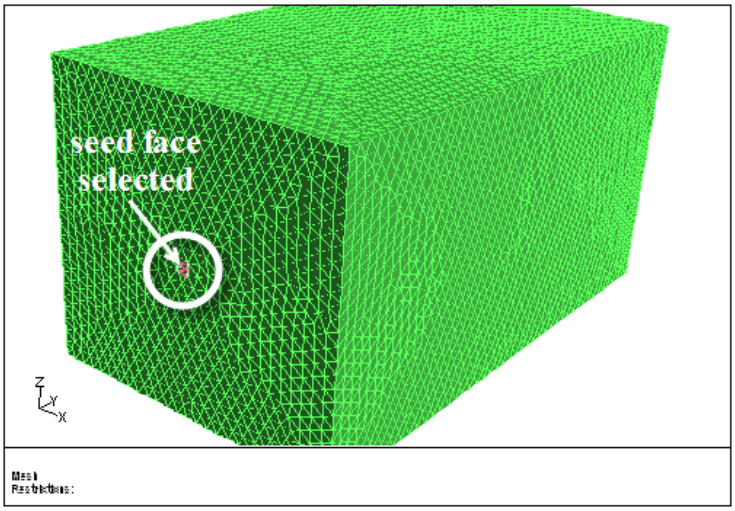


Figure 7.34: Seed Face for Separating the Tunnel Inlet

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.