Chapter 13. Improving the Mesh

A volume mesh created from a high quality surface mesh may contain some high skewness cells. The poor cells may result from unsuitable mesh size distribution over the domain or more often caused by constraints imposed by the boundaries. This chapter describes the options available in TGrid to improve the mesh quality by removing highly skewed cells.

- Section 13.1: Smoothing Nodes
- Section 13.2: Swapping
- Section 13.3: Improving the Mesh
- Section 13.4: Removing Slivers from a Tetrahedral Mesh
- Section 13.5: The Tri/Tet Improve Panel
- Section 13.6: Modifying Cells
- Section 13.7: Moving Nodes
- Section 13.8: Cavity Remeshing
- Section 13.9: Manipulating Cell Zones
- Section 13.10: Using Domains to Group and Mesh Boundary Faces
- Section 13.11: Checking the Mesh
- Section 13.12: Clearing the Mesh

After creating a triangular, tetrahedral, or hybrid mesh, you can improve the quality of the mesh by smoothing nodes and swapping faces. Smoothing and face swapping are tools that help to improve the quality of the final numerical mesh. You can also use the improve command which combines operations like collapsing cells, node smoothing, face swapping, and inserting nodes.

Tetrahedral mesh improvement options are available in the Tri/Tet Improve panel (see Section 13.5: The Tri/Tet Improve Panel).

13.1 Smoothing Nodes

Smoothing repositions the nodes to improve the mesh quality. The smoothing methods available in TGrid are:

- Laplace smoothing
- Variational smoothing
- Skewness-based smoothing

13.1.1 Laplacian Smoothing

Laplace smoothing is used to improve (reduce) the average skewness of the mesh. In this method, a Laplacian smoothing operator is applied to the unstructured grid to reposition nodes. The new node position is the average of the positions of its node neighbors.

The relaxation factor (a number between 0.0 and 1.0) multiplies the computed node position increment. A value of zero results in no movement of the node and a value of unity results in movement equivalent to the entire computed increment.

This node repositioning strategy improves the skewness of the mesh, but usually relaxes theclustering of node points. In extreme circumstances, the unchecked operator may create grid lines that cross over the boundary, creating negative cell volumes. To prevent such cross-overs, the skewness of the resulting cells is checked before the node is repositioned. This makes the smoothing operation time-consuming.

The smoothing operator can also be applied repeatedly, but as the number of smoothing iterations increases, the node points have a tendency to pull away from boundaries and the grid tends to lose any clustering characteristics.

13.1.2 Variational Smoothing

Variational smoothing is available only for triangular and tetahedral meshes. It can be considered as a variant of Laplace smoothing. The new node position is computed as a weighted average of the circumcenters of the cells containing the node. The variational smoothing method is provided as a complement to Laplace smoothing.

13.1.3 Skewness-Based Smoothing

Skewness-based smoothing is available only for triangular and tetrahedral grids. When you use skewness-based smoothing, TGrid applies a smoothing operator to the grid, repositioning interior nodes to lower the maximum skewness of the grid. TGrid will try to move interior nodes to improve the skewness of cells with skewness greater than the specified minimum skewness. You can also specify an appropriate value for the minimum improvement, if required. This allows you to stop performing the smoothing iterations when the maximum change in cell skewness is less than or equal to the value specified for minimum improvement.

TGrid will compare the maximum change in cell skewness with the specified value for minimum improvement. When the maximum change in cell skewness is less than or equal to the value specified for minimum improvement, further smoothing iterations will no longer yield appreciable improvement in the mesh. TGrid will stop the smoothing at this point even if the requested number of iterations has not been completed.

This skewness-based smoothing process can be very time-consuming, so it is advisable to perform smoothing only on cells with high skewness. Improved results can be obtained by smoothing the nodes several times. There are internal checks that will prevent a node from being moved if moving it causes the maximum skewness to increase, but it is common for the skewness of some cells to increase when a cell with a higher skewness is being improved. Hence, you may see the average skewness increase while the maximum skewness is decreasing.

You should consider whether the improvements to the mesh due to a decrease in the maximum skewness are worth the potential increase in the average skewness. Performing smoothing only on cells with very high skewness (e.g., 0.8 or 0.9) may decrease the adverse effects on the average skewness.

13.1.4 Text Commands for Smoothing

Text commands for smoothing the grid are as follows:

`/mesh/laplace-smooth-nodes` applies a Laplacian smoothing operator to the grid nodes. This command can be used for smoothing of all cell types, including prismatic cells.

`/mesh/tritet/improve/skew-smooth-nodes` applies skewness-based smoothing to nodes on the tetrahedral cell zones to improve the mesh quality.

`/mesh/tritet/improve/smooth-nodes` allows you to apply either Laplacian or variational smoothing to nodes on the tetrahedral cell zones to improve the mesh quality.

13.2 Swapping

Swapping can be used to improve the quality of triangular or tetrahedral grids.

13.2.1 Triangular Grids

The approach for triangular grids is to use the Delaunay circle test to decide if a face shared by two triangular cells should be swapped. A pair of cells sharing a face satisfies the circle test if the circumcircle of one cell does not contain the unshared node of the second cell.

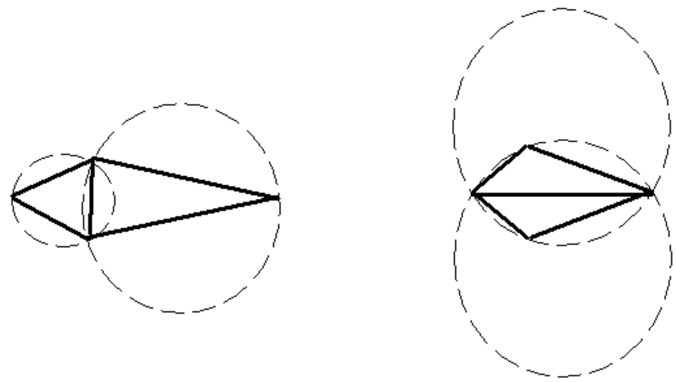


Figure 13.2.1: Cell Configurations—Satisfying and Not Satisfying the Circle Test

Figure 13.2.1 illustrates cell neighbors that satisfy and do not satisfy the circle test.In cases where the circle test is not satisfied, the diagonal or face is swapped, as illustrated in Figure 13.2.2.

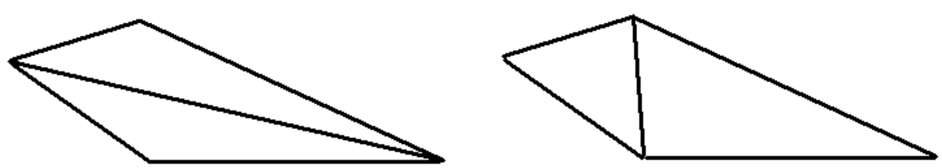


Figure 13.2.2: Swapped Face—Satisfying the Delaunay Circle Test

Repeated application of the swapping technique will produce a constrained Delaunay mesh. A Delaunay grid is a unique triangulation that maximizes the minimum angles in the mesh. Thus, the triangulation tends towards equilateral cells, providing the most equilateral grid for the given node distribution.

13.2.2 Tetrahedral Grids

For tetrahedral grids, swapping involves searching for a specific configuration of cells and replacing it by an alternative configuration. The default option is a 3–2 swap configuration where three tetrahedra are replaced by two tetrahedra after swapping. The other possible combinations are the 2–3 swap configuration (replacing two tetrahedra by three) and the 4–4 swap configuration (replacing four existing tetrahedra with four alternate tetrahedra).

Figure 13.2.3 shows the 2–3 and 3–2 swap configurations. The two tetrahedra (on the left) having a common interior face can be replaced by three tetrahedra having a common interior edge. The common face will be replaced by three interior faces and an interior edge during swapping. Conversely, the three tetrahedra (on the right) have two interior faces each and share a common interior edge. During swapping for a 3–2 configuration, three interior faces and the common interior edge will be replaced by a single face. This results in two tetrahedra having a common interior face.

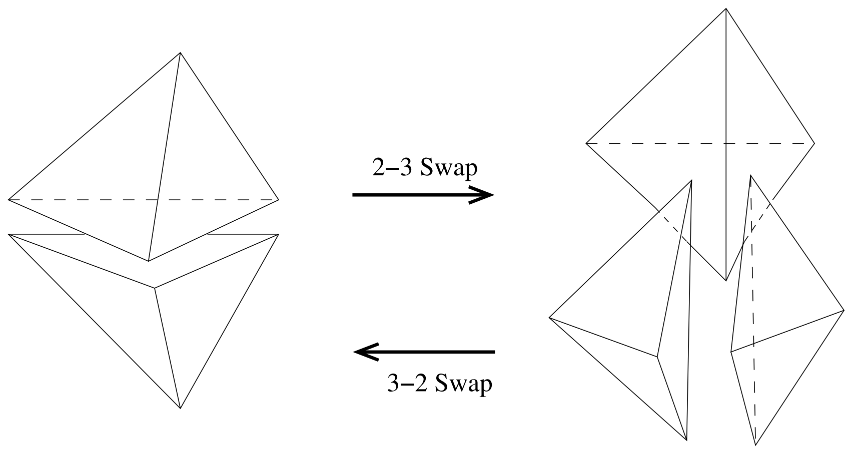


Figure 13.2.3: 2–3 and 3–2 Swap Configurations

Another possible swap configuration is the 4–4 swap confguration where four tetrahedra will be replaced by four alternate tetrahedra. In Figure 13.2.4, either the common interior edge or two common faces can be replaced, resulting in four alternate tetrahedra.

13.2.3 Text Interface for Smoothing and Swapping

Text command for smoothing and swapping on the grid is as follows:

`/mesh/tritet/improve/swap-faces` performs interior face swapping to improve cell skewness.

13.3 Improving the Mesh

The Improve operation is an automated procedure for sliver removal or for reducing the maximum skewness in the mesh. The improvement is carried out by removing cells above the specified skewness threshold by collapsing cells, swapping faces, smoothing nodes, and inserting new nodes iteratively. Each operation is specialized, and the mesh will be modified only if the mesh is noticeably improved. The skewness before and after

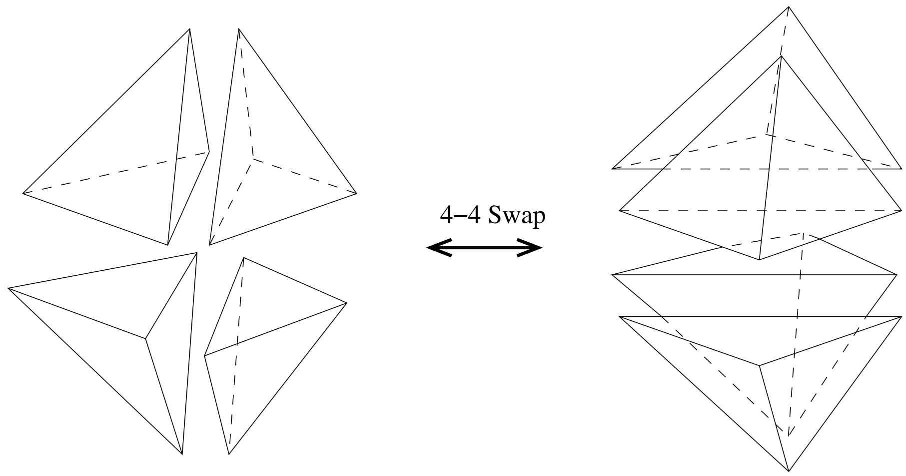


Figure 13.2.4: 4–4 Swap Configuration

an operation is taken into account to determine the improvement. Hence, the lower the skewness of the cells involved, the larger the improvement will have to be in order for the mesh to be modified. The improve operation is a more elaborate version of the Remove Slivers option invoked from the Refinement tab in the Tri/Tet panel (see Section 11.5: The Tri/Tet Panel).

13.4 Removing Slivers from a Tetrahedral Mesh

A sliver typically denotes a flat tetrahedral cell. Figure 13.4.1 shows an acceptable tetrahedron.

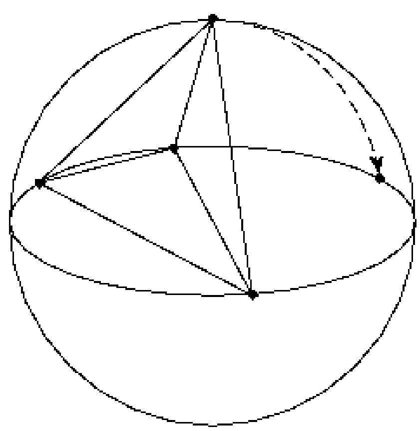


Figure 13.4.1: Sliver Formation

If the top node of the tetrahedron were to travel along the path of the dotted line in the direction of the arrow, as it approached the end of the line the resulting cell would be a degenerate tetrahedron, or a sliver.

In the following sections, the term sliver is used to denote all types of poorly shaped cells. There are several commands in TGrid to remove slivers or to reduce the maximum skewness of the mesh.

13.4.1 Automatic Sliver Removal

Sliver removal operations can be invoked during the tri/tet mesh refinement process. The Remove Slivers option in the Refinement tab of the Tri/Tet panel (see Section 11.5: The Tri/Tet Panel) controls the removal of slivers during the meshing process. The Remove Slivers option comprises operations such as collapsing cells (to remove nodes), face swapping, smoothing, and point insertion, which are invoked iteratively.

Usually the sliver cells will be removed during refinement, but occasionally a few might be left behind. In such cases, you can use the options in the Slivers tab in the Tri/Tet Improve panel (see Section 13.5: The Tri/Tet Improve Panel) to remove the slivers manually. The Improve command uses an automated procedure to remove slivers or to improve the mesh quality in general.

13.4.2 Removing Slivers Manually

The Tri/Tet Improve panel contains options for removing slivers manually. The operations available for sliver removal are:

Smoothing Boundary Slivers

This operation involves smoothing nodes on sliver cells having atleast one node on the boundary. During smoothing, the nodes will be repositioned so long as the skewness of the surrounding cells is improved. The nodes on features will also be smoothed, but will not be projected on to the original geometry. However, nodes at branch points (more than two feature edges at the node) and end points (one feature edge at the node) will be fixed. The nodes will be smoothed until the skewness value is less than the specified value. The default values for the skewness threshold, minimum dihedral angle between boundary faces, and feature angle are `0.985`, `10`, and `30`, respectively.

Smoothing Interior Slivers

This operation involves smoothing non-boundary nodes on sliver cells having skewness greater than the specified threshold value. The default value for the skewness threshold is `0.985`.

Swapping Boundary Slivers

A flat boundary cell containing two boundary faces can be removed by moving the boundary to exclude the cell from the zone in which it is located, effecting a minor change in the geometry. However, if there is another live zone on the other side of the boundary, this operation will result in the cell being moved to the other zone. In such cases (e.g., conjugate heat transfer problems), you can decide which live zone is least critical, and then move the boundary sliver to that zone.

The default values for the skewness threshold and the minimum dihedral angle between faces are `0.95` and `10`, respectively.

Refining Boundary Slivers

This operation attempts to increase the volume of boundary slivers to create a valid tet cell. TGrid identifies tetrahedra having one or two faces on the boundary and then splits the edge opposite the boundary face(s). The edge opposite the face pair with the largest dihedral angle will be split for a tet with one boundary face, while the edge opposite the boundary faces will be split for a tet having two boundary faces. The split node is then smoothed such that the volume of the tetrahedron increases, thereby creating a valid tet cell.

Refining Interior Slivers

This operation attempts to remove the sliver by placing a node at or near the centroid of the sliver cell. TGrid then performs swapping and smoothing to improve the skewness.

Collapsing Slivers

This operation attempts to collapse the edge of a skewed sliver cell on any one of its neighbors. The default skewness threshold is `0.985`.

Notes:

- *If you are not using a two-sided wall condition for the boundary, you can slit the face zone containing the sliver (using the* `/boundary/slit-boundary-face` *command) and then perform the sliver removal operation.*
- *Multiple slivers may exist on top of each other thus, requiring multiple operations to remove them all.*

13.4.3 Text Interface for Sliver Removal

The text interface commands for removing boundary slivers are:

`/mesh/tritet/improve/collapse-slivers` attempts to collapse the nodes of a skewed sliver cell on any one of its neighbors.

`/mesh/tritet/improve/refine-boundary-slivers` attempts to increase the volume of boundary slivers to create a valid tet cell. TGrid identifies tetrahedra having one or two faces on the boundary and then splits the appropriate edge. The split node is then smoothed such that the volume of the tetrahedron increases, thereby creating a valid tet cell.

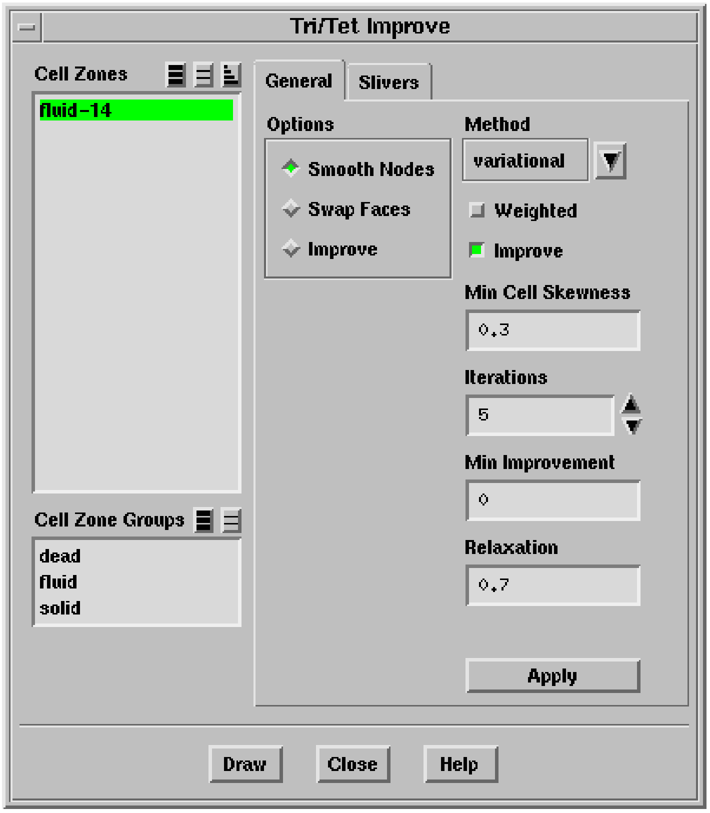
`/mesh/tritet/improve/refine-slivers` attempts to remove the sliver by placing a node at or near the centroid of the sliver cell. TGrid then performs swapping and smoothing to improve the skewness. You can also specify whether boundary cells are to be refined. Refining the boundary cells may allow you to carry out further improvement options such as smoothing, swapping, and collapsing slivers.

`/mesh/tritet/improve/sliver-boundary-swap` removes boundary slivers by moving the boundary to exclude the cells from the zone.

`/mesh/tritet/improve/smooth-boundary-sliver` smoothes nodes on sliver cells having all four nodes on the boundary until the skewness value is less than the specified value. The default values for the skewness threshold, minimum dihedral angle between boundary faces, and feature angle are `0.985`, `10`, and `30`, respectively.

`/mesh/tritet/improve/smooth-interior-sliver` smoothes non-boundary nodes on sliver cells having skewness greater than the specified threshold value. The default value for the skewness threshold is `0.985`.

13.5 The Tri/Tet Improve Panel



Controls

Cell Zones contains a list from which you can select the cell zones to be improved.

Cell Zone Groups contains a list of the cell zone types. If you select a cell type from the list, (e.g., fluid), all cell zones of that type will be selected in the Cell Zones list.

General contains optios for smoothing nodes, swapping faces, and improving cells.

Options contains a list of the options available for improving the mesh.

Smooth Nodes smoothes the nodes to improve the mesh quality.

Method contains a list of the smoothing methods available in TGrid.

Weighted implies that the cell size (as given by the cell size function) at the neighboring nodes (for the Laplace method) or the circumcenter (for the variational method) is used to weight the influence of each node (or circumcenter) when computing the average node location. This option is available only when variational or laplace is selected in the Method drop-dowm list.

Improve implies that the node will be repositioned only when the maximum skewness of the cells connected to the node is reduced. This option is available only when variational or laplace is selected in the Method drop-dowm list.

Min Cell Skewness specifies the skewness above which the nodes will be smoothed.

Iterations specifies the number of smoothing iterations.

Min Improvement specifies the convergence criterion for smoothing. For skewness based smoothing, smoothing will be stopped when the largest change in skewness is below the limit specified. For Laplace and variational smoothing, the convergence criterion is the root mean square of the distance the nodes were moved during smoothing.

Relaxation specifies the factor used for computing the smoothing distance. The value varies between 0 and 1. A value of 0 results in no movement, while a value of 1 results in movement equivalent to the computed distance. A smaller value will result in a more gradual change in the mesh. A reduced value may also result in a higher quality mesh, but at the expense of more smoothing iterations. This option is available only when variational or laplace is selected in the Method drop-dowm list.

Swap Faces swaps the faces of the cells to improve the mesh quality.

Swap Configuration contains a list of the possible configurations for swapping.

Skewness specifies the skewness above which cells will be considered for face swapping.

Iterations specifies the number of swapping iterations.

Improve improves the mesh quality based on the specified parameters.

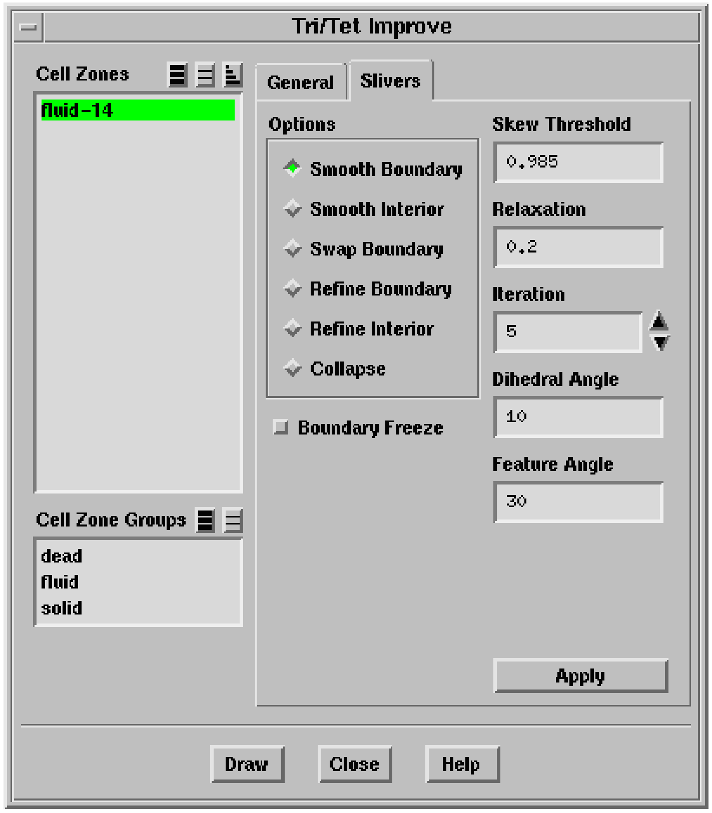
Skew Threshold specifies the skewness threshold for improving the mesh quality.

Dihedral Angle specifies the threshold dihedral angle. If the dihedral angle between two boundary faces is below the specified value, the cell is considered to be a valid boundary sliver (which cannot be improved).

Max Attempts specifies the number of improvement attempts.

Apply performs the selected operation.

Slivers contains the options available and parameters required for sliver removal.



Options contains a list of the operations available for sliver removal.

Smooth Boundary smoothes nodes on sliver cells having any node on the boundary such that skewness of the surrounding cells is improved.

Smooth Interior smoothes non-boundary nodes on sliver cells having skewness greater than the specified threshold value.

Swap Boundary removes boundary slivers by moving the boundary to exclude the cells from the zone.

Refine Boundary refines the boundary slivers by edge splitting.

Refine Interior remove the sliver by placing a node at or near the centroid of the sliver cell.

Collapse attempts to collapse the nodes of a skewed sliver cell on any one of its neighbors.

Skew Threshold specifies the skewness threshold above which sliver cells will be removed.

Relaxation specifies the relaxation factor for smoothing boundary and interior slivers.

Iteration specifies the number of iterations for the smoothing of boundary and interior slivers.

Min Skewness specifies the skewness above which the boundary slivers will be swapped.

Dihedral Angle specifies the threshold dihedral angle. If the dihedral angle between two boundary faces is below the specified value, the cell is considered to be a valid boundary sliver (which cannot be improved).

Feature Angle specifies the angle defining the features for smoothing of boundary slivers.

Boundary Freeze allows you to fix the boundary during the sliver removal operations.

Apply performs the selected operation.

13.6 Modifying Cells

TGrid provides tools enabling you to perform primitive operations on the cells such as smoothing nodes, swapping cells, splitting cells, etc. This section describes the generic procedure for modifying the cells using the Modify Cells panel. You will also use the Display Grid panel (see Section 14.1.3: The Display Grid Panel) during the modification of the cells.

13.6.1 Using the Modify Cells Panel

1. Display the cell(s) or cell zone(s) to be modified using the options in the Display Grid panel.
2. Select the type of entity (cell, face, node, etc.) you want to select in the Filter list in the Modify Cells panel.
3. Select the objects to be modified in the graphics window.
4. Click the appropriate button in the Operation group box.
5. Repeat the procedure to perform different operations on the cells.

⚠ Save the mesh periodically since not all operations are reversible.

The operations available in the Modify Cells panel for modifying cells are:

Smoothing Nodes

During node smoothing, the selected node will be repositioned based on the average of the surrounding nodes. Do the following to smooth a node:

1. Select the node(s) to be smoothed.
2. Click Smooth in the Operation group box.

Splitting Cells

During splitting, the selected cell will be refined by the addition of a node at the centroid of the cell.

Moving Nodes

You can move the selected node either to a specified position or by a specified magnitude.

Do the following to move a node to a particular position:

1. Select node in the Filter list and select the node to be moved.
2. Select position in the Filter list and select the appropriate poistion.
3. Click Move To in the Operation group box.

Do the following to move a node by a specified magnitude:

1. Select node in the Filter list and select the node to be moved.
2. Enter the magnitude by which you want to move the node in the Enter Selection field and press `<Enter>`.

   The incerement will now be selected in the Selections *list.*
3. Click Move By in the Operation group box.

Swapping Cells

You can perform either a 3–2 configuration swap or a 2–3 configuration swap. Refer to Section 13.2.2: Tetrahedral Grids for details on the swap configurations. Do the following to swap cells:

1. Select the appropriate option in the Filter list and select the entities to be swapped.
   - Select 3 cells or the common edge in order to perform a 3–2 swap.
   - Select 2 cells or the common face in order to perform a 2–3 swap.
2. Click Swap in the Operation group box.

Determining the Coordinates of the Centroid

Do the following to determine the centroid:

1. Select the appropriate option in the Filter list (cell, face, edge, or node) and select the required entity.
2. Click Centroid in the Operation group box.

The centroid coordinates will be printed in the TGrid console.

Determining the Distance Between Entities

Do the following to determine the distance between entities:

1. Select the appropriate option in the Filter list and select the two entities between which the distance is to be determined.
2. Click Distance in the Operation group box.

The distance between the selected entities will be printed in the TGrid console.

Projecting Nodes

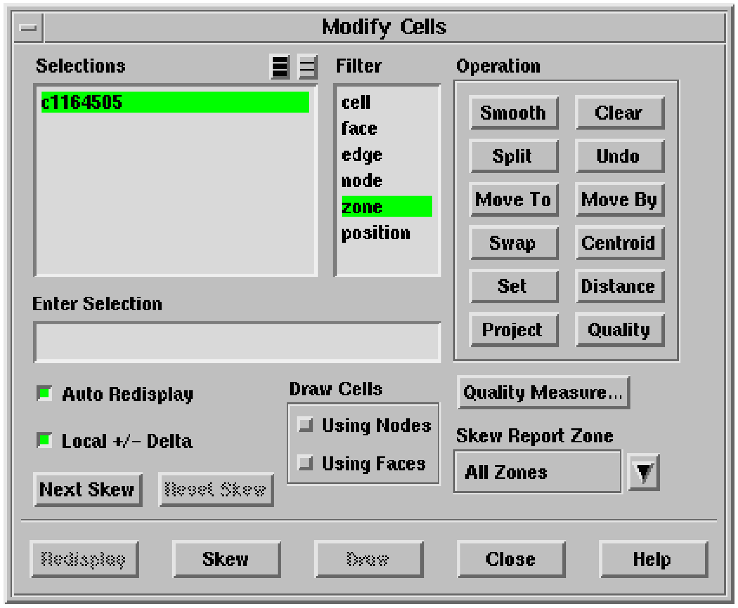
You can project nodes onto a specified projection line or plane. Do the following to project nodes:

1. Define the projection line or projection plane, as appropriate.
   (a) Select the appropriate option in the Filter list.
   (b) Select two entities to define the projection line or three entities to define the projection planne.
   (c) Click Set in the Operation group box to define the projection line or plane.
   The line coordinates or plane position will be repoted in the TGrid *console.*
2. Select the node(s) to be projected.
   The projection line/plane will be highlighted in the display window.
3. Click Project in the Operation group box.

13.6.2 The Modify Cells Panel

The Modify Cells panel allows you to perform operation such as smoothing nodes, moving nodes, swapping cells, etc.

Mesh ⟶ Tools ⟶ Cell Modify...



Controls

Selections contains a list of the selected entities. You need to ensure the appropriate entities are selected in the Selections list before clicking the required button in the Operation group box.

Filter lists the types of entities that can be selected. Only entities of the type selected in the Filter list can be selected. To select entities of more than one type, select the appropriate option in the Filter list and select the required entities of the same type. Then, change the selection in the Filter list and continue to select the entities as required.

Operation contains buttons for performing cell modification operations.

> Smooth repositions the selected node based on the average of the surrounding nodes.
>
> Clear removes the selected entities from the Selections list.
>
> Split refines the selected cell by adding a node at the centroid and splitting the cell into four cells.
>
> Undo undoes the previous operation.

Move To moves the selected node to the specified position.

Move By moves the selected node by the specified increment.

Swap swaps the cell configuration (3–2 or 2–3) according to the entities selected.

Centroid reports the centroid for the entity selected in the Selections list.

Set defines the projection line or plane determined by the entities selected in the Selections list.

Distance reports the distance between the two entities selected in the Selections list.

Project projects the selected node(s) onto to the defined projection line or plane.

Quality reports the quality of the selected entitiy according to the Measure selected in the Quality Measure panel (see Section 15.5.2: The Quality Measure Panel).

Enter Selection allows you to enter the name of an entity or the coordinates of a particular position as an alternative to selecting the entity from the graphics window.

Auto Redisplay toggles the automatic update of display after a cell modification operation has been performed. When Auto Redisplay is enabled, TGrid will automatically update the display, allowing you to see the change effected by the operation.

Local +/- Delta toggles the limiting of the grid display to a neighborhood around the selected entity. When Local +/- Delta is enabled, TGrid will display the selected entity and some faces around it.

Draw Cells contains options for drawing cells.

Using Nodes enables the display of cells comprising the selected node(s).

Using Faces enables the display of cells comprising the selected face(s).

Quality Measure... opens the Quality Measure panel (see Section 15.5.2: The Quality Measure Panel).

Skew Report Zone contains a drop-down list of the cell zones for which you can locate skewed cells. This allows you to select the cell zone of interest instead of locating the skewed cells for all zones.

Next Skew displays the cell having the next highest skewness value after the currently displayed cell. Clicking the Next Skew button repeatedly will display the skewed cells in the descending order of skewness. The Next Skewbutton will be activated only after the Skew button has been clicked in the Modiy Cells panel.

Reset Skew resets the skewness values. When you click Reset Skew, both the Next Skew and Reset Skew buttons will be deactivated. Click Skew and then Next Skew to display the skewed cells again.

Redisplay updates the display in the display window. This button is activated only when the Auto Redisplay option is disabled.

> You can disable Auto Redisplay for large meshes or when the graphics performance of your computer is slow. You can then use the Redisplay button to view the updated display.

Skew displays the cell with the worst (highest) skewness. The cell will also be added to the Selections list.

Draw displays the cells connected to the selected node(s) or face(s), according to the options enabled in the Draw Cells group box.

13.6.3 Text Commands for Modifying Cell Zones

Text interface commands for modifying cell zones are as follows:

`/mesh/modify/clear-selections` deseclects the selected entities.

`/mesh/modify/deselect-last` deseclects the last selected entity.

`/mesh/modify/extract-unused-nodes` extracts the unused nodes into a separate interior node zone.

`/mesh/modify/list-selections` lists the selected entities in the TGrid console.

`/mesh/modify/list-skewed-cells` lists the cells which are within the specified skewness limits.

`/mesh/modify/mesh-node` introduces a new node into the existing mesh.

`/mesh/modify/mesh-nodes-on-zone` inserts the nodes associated with a particular face or node thread into the volume mesh.

For a specified face thread, TGrid *will delete the faces before introducing the nodes into the mesh.*

`/mesh/modify/neighboring-skew` reports the skewness of all cells using the specified node.

`/mesh/modify/refine-cell` refines the cells specifed.

`/mesh/modify/select-entity` selects the specified entity.

`/mesh/modify/smooth-node` performs Laplace smoothing on the specified nodes.

13.7 Moving Nodes

Highly skewed meshes can be improved by moving the nodes of the cells. Moving nodes manually is a time consuming process. TGrid has a tool which automates the node movement process for improving the mesh quality. You can specify quality improvement based on the skewness (triangular/tetrahedral elements) or the warp (quadrilateral elements). You can also choose between the automatic correction process and the semi-automatic correction process for the skewness-based correction.

13.7.1 Automatic Correction

The automatic correction allows you to improve all the cells in the selected fluid zone based on the specified criteria. You can improve the cells based on the skewness or the warp values.

Skewness-Based Improvement

For the skewness-based correction, you can specify the skewness threshold, feature angle, and the number of iterations per node to be moved. TGrid will select the cells which have a skewness greater than the specified maximum skewness. For boundary nodes, you can restrict the node movement in the plane containing each of the boundary faces sharing the node being moved. The node to be moved for a particular cell will be selected based on the selection of zones in the Boundary Zones selection list and an alternative position for the node will be determined. The node will be moved to the new position only if the skewness of the cell and its neighbors is improved by the change in node position. The procedure is repeated for the specified iterations per node. You can also set the number of repetitions through the automatic correction procedure as required. By default, TGrid performs the correction procedure only once.

Warp-Based Improvement

For the warp-based correction, you can specify the maximum warp and the number of iterations per face to be improved. TGrid will select the quadrilateral faces which have a warp value greater than the specified maximum warp. The ideal position for the node to be moved will be determined based on the remaining three nodes. The node will be moved to the new position only if the warp of the face decreases and the cell skewness does not increase by the change in node position. The procedure is repeated for the specified iterations per face. You can also set the number of repetitions through the automatic correction procedure as required. By default, TGrid performs the correction procedure twice.

13.7.2 Semi-Automatic Correction

The semi-automatic correction is available only for skewness-based improvement. The generic procedure for using the semi-automatic correction is as follows:

1. Select the appopriate zones in the Cell Zones drop-down list and the Boundary Zones selection list.

2. Specify values for Max Skew, Iterations/Node, and Feature Angle as appropriate.

3. Enable Restrict Boundary Nodes Along Surface if required. When this option is enabled, the movement of the boundary node will be limited to the plane containing the boundary faces sharing the boundary node (see Figure 13.7.1).

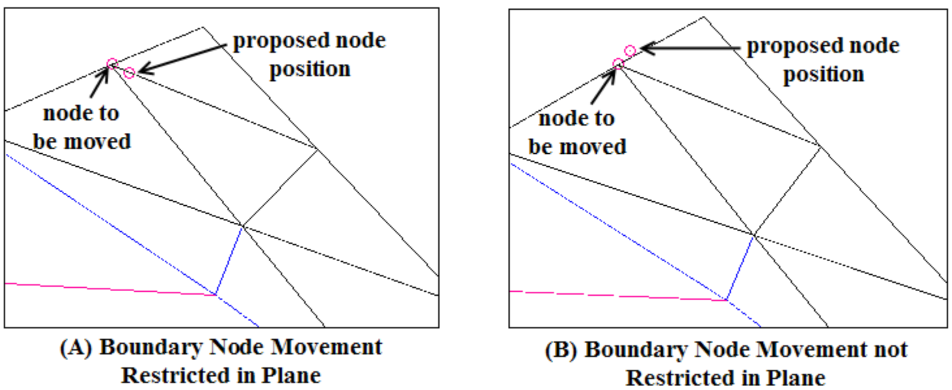


Figure 13.7.1: Movement of Boundary Nodes

4. Click Skew to display the cell with the maximum skewness depending on the skewness threshold specified. TGrid will display the cell having maximum skewness and cells/faces within a pre-defined range of the cell.

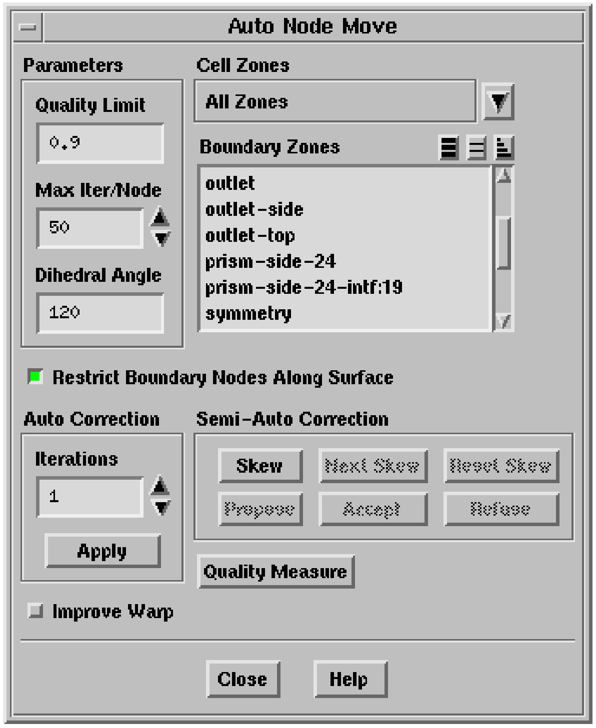
5. Click Propose. TGrid will highlight the node to be moved and the alternative position. The improvement in the skewness will also be reported in the TGrid console.

6. Click Accept if the proposed position is appropriate. Else, click Refuse and then Propose to obtain the next suggestion.

7. Click Next Skew to proceed with the node correction for the next highly skewed cell.

13.7.3 The Auto Node Move Panel

The Auto Node Move panel contains options for moving nodes to improve the mesh quality.



Controls

Parameters contains the parameters for node movement.

- Quality Limit specifies the quality limit for triangular/tetrahedral cells. All cells above the specified limit will be considered for improvement.
- Max Warp specifies the maximum allowable warp for quadrilateral faces. All quad cells having faces with warp greater than this value will be considered for improvement. This option is available only when the Improve Warp option is enabled.
- Max Iter/Node specifies the number of attempts to improve the skewness by moving a particular node.
- Max Iter/Face specifies the number of attempts to improve the warp of a face. This option is available only when the Improve Warp option is enabled.
- Dihedral Angle specifies the feature angle to be considered when moving boundary nodes and allows you to maintain features of the geometry. A boundary node will not be moved if the angle between the faces sharing the boundary node is less than the specified value.

Cell Zones contains a drop-down list of the cell zones from which you can select those to be considered for improvement.

Boundary Zones contains a list of the boundary zones from which you can select those to be considered for improvement.

Restrict Boundary Nodes Along Surface allows you to restrict the movement of boundary nodes within the plane caontaining the faces sharing the node. This option is available only for the skewness-based improvement.

Auto Correction contains options for using the automatic correction procedure for skewness-based improvement.

Iterations specifies the number of repetitions through the automatic correction procedure.

Apply performs the automatic correction procedure according to the number of iterations specified.

Semi-Auto Correction contains options for using the semi-automatic correction procedure for skewness-based improvement.

Skew displays the cell with the maximum skewness and cells/faces within a predefined range of the cell.

Next Skew displays the cell with the next highest skewness.

Reset Skew allows you to reset the skewness values.

Propose highlights the node to be moved and the alternative node position. The improvement in the skewness will also be reported in the TGrid console.

Accept allows you to accept the proposed node movement.

Refuse allows you to reject the proposed node movement.

Quality Measure opens the Quality Measure panel (see Section 15.5.2: The Quality Measure Panel).

Improve Warp contains options for using the automatic correction procedure for warp-based improvement.

13.7.4 Text Commands for Moving Nodes

The text commands for improving the mesh by moving nodes are as follows:

`/mesh/modify/auto-node-move` allows you to improve the mesh quality by node movement.

`/mesh/modify/auto-improve-warp` allows you to improve face warp by node movement.

13.8 Cavity Remeshing

Cavity remeshing is useful in parametric studies since it allows you add, remove, and replace different parts of the existing mesh. You can compare alternative designs by creating a cavity around the object to be replaced and then, inserting the new object and connecting it to the existing mesh. Prisms can be grown using appropriate parameters and the cavity can be filled with tet cells. You can also improve the quality of the volume mesh by creating an appropriate cavity around the skewed cells and remeshing it.

TGrid allows you to create a cavity in an existing mesh by removing tet cells in a defined bounded region. The cells intersecting the bounded region will be marked and TGrid will extract the cavity boundaries from the marked cells. The marked cells will then be deleted to create the cavity.

Figure 13.8.1 shows a cavity created around the rear-view mirror of a car. You can see the original mesh and the remeshed cavity created around the mirror. The bounding box defined for the cavity is also shown.

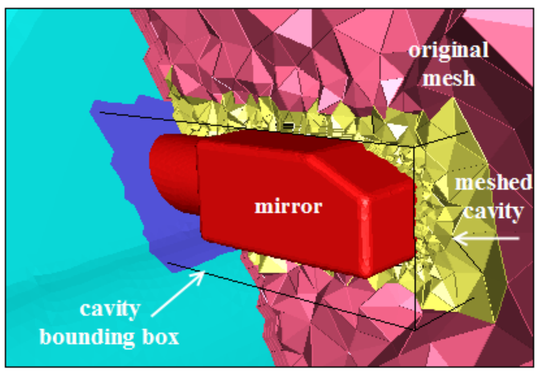


Figure 13.8.1: Remeshed Cavity Around a Mirror

The various options available in TGrid are:

- Removing zones: This option allows you to specify zones to be removed from the existing volume mesh.
- Adding zones: This option allows you to add new zones to the existing volume mesh.
- Replacing zones: This option allows you to remove a set of zones and replace them with a new set of zones.
- Improving a region: This option allows you to define a cavity around skewed cells in the existing mesh. You can modify the mesh as appropriate and then remesh the cavity.

Remeshing a Cavity

The generic procedure for remeshing a cavity using the Cavity Remesh panel is as follows:

1. Select the appropriate zones from the Remove Boundary Zones and Add Boundary Zones selection lists.

   - Removing Zones: Select the zone(s) to be removed in the Remove Boundary Zones selection list. Make sure that no zones have been selected in the Add Boundary Zones list.
   - Adding Zones: Select the zone(s) to be added in the Add Boundary Zones selection list. Make sure that no zones have been selected in the Remove Boundary Zones list.
   - Replacing Zones: Select the zone(s) to be removed in the Remove Boundary Zones selection list and the zone(s) to be added in the Add Boundary Zones selection list.
   - Improving a Region: Make sure that no zones have been selected in the Remove Boundary Zones and Add Boundary Zones selection lists.

2. Enable Create Face Group if you want to create a user-defined group (UDG) comprising the zones defining the cavity domain.

   TGrid *will create the* cavity *UDG and the corresponding* cavity *domain, but retain the* global *domain as active when the* Create Face Group *option is enabled. When this option is disabled, the* cavity *domain will be activated when the cavity is created.*

   i The Create Face Group option is useful when using the cavity remeshing feature for large cases. For such cases, you can avoid frequent switching between domains by enabling the Create Face Group option when creating the cavity. The basic procedure is as follows:

   (a) Create a UDG for the cavity and save the boundary mesh for the cavity group defined using the File/Write/Boundaries... option (see Section 6.1.5: Writing Boundary Mesh Files).
   (b) Read this boundary mesh in a separate TGrid session and create the volume mesh in the cavity as appropriate.
   (c) Save the volume mesh and read it back into the previous TGrid session using the Append File(s) option.
   (d) Connect the meshed cavity to the parent mesh and merge the cavity domain with the parent domain.

3. Enter an appropriate value for Scale and click Compute. The extents of the bounding box will be computed (based on the zones selected in the Remove Boundary Zones and Add Boundary Zones selection lists, and the scale factor specified) and

reported in the Cavity Remesh panel. Alternatively, you can specify the extents of the bounding box as required.

⚠ You need to manually specify the extents of the bounding box when using the cavity remeshing feature to improve a region of skewed cells.

4. Specify the orientation of the bounding box in the Orient group box.

5. Click Draw to preview the cavity domain to be created.

6. Click Create to create the cavity domain. The cavity domain will comprise the new zone(s)to be added and the boundary zones touching the domain defined by the bounding box. Any zone(s) to be removed will not be included in the cavity domain. The existing volume mesh in the cavity will be removed and boundary zone(s) extracted from the interior zone(s) (if any) will be changed to wall type and included in the cavity domain.

7. Connect the new zone(s) with the boundaries of the cavity domain using node merging or intersect operations when removing, adding, or replacing zones in the volume mesh. Refer to Sections 7.1 and 7.2 for details. Modify the boundary mesh (if required) when improving a region in the volume mesh. Refer to Chapter 7: Manipulating the Boundary Mesh for details on the various mesh improvement options.

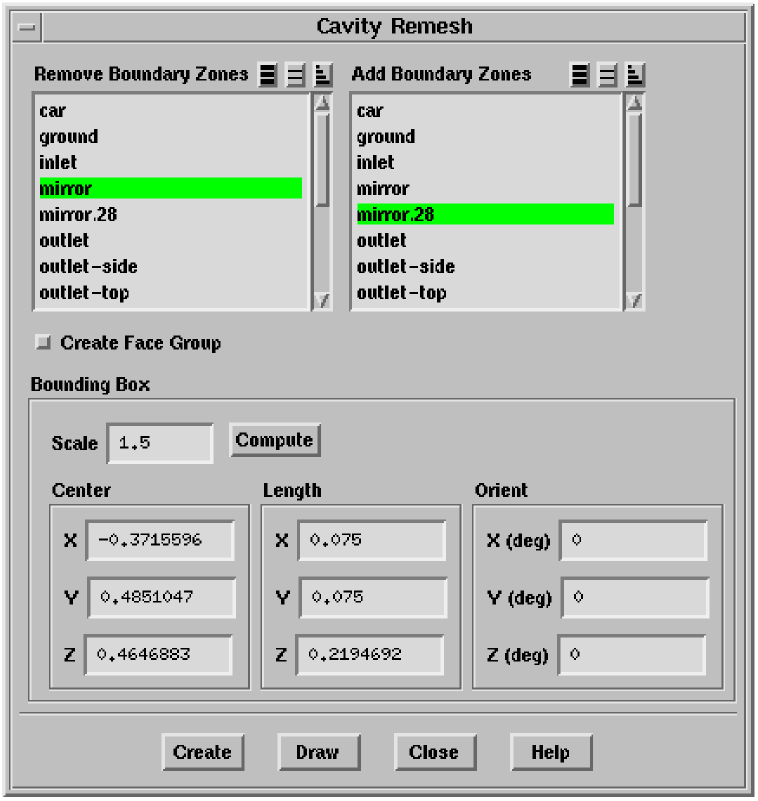
8. Activate the cavity domain and create the volume mesh as appropriate.

9. Activate the global domain and delete the boundary zone(s) which are no longer required. Merge the cavity domain with the parent domain using the command `/mesh/cavity/merge-cavity`.

   During the merging operation, the cavity cell zone(s) will be merged with the cell zone(s) in the parent domain. The wall boundaries extracted from the interior zones will be converted to interior type and merged with the corresponding zone(s) in the parent domain. Other boundary zones included in the cavity domain will be merged with the parent face zones.

13.8.1 The Cavity Remesh Panel

The Cavity Remesh panel contains options for creating a cavity in an existing volume mesh.



Controls

Remove Boundary Zones contains a selection list of available boundary zones from which you can select the zones to be removed from the volume mesh.

Add Boundary Zones contains a selection list of available boundary zones from which you can select the zones to be added to the volume mesh.

Create Face Group allows you to create a UDG comprising the zone(s) defining the cavity domain.

Bounding Box defines the extents of the cavity to be created.

Scale specifies the scale factor to be applied while calculating the bounding box for the zones selected in the Remove Boundary Zones and Add Boundary Zones selection lists.

Compute computes the bounding box extents according to the zones selected in the Remove Boundary Zones and Add Boundary Zones selection lists, and the scale factor specified.

Center specifies the coordinates of the center of the cavity.

Length specifies the length of the cavity bounding box along the X, Y, and Z axes.

Orient specifies the orientation of the cavity bounding box about the center of the cavity.

Create creates a cavity according to the bounding box defined.

Draw allows you to preview the cavity to be created.

13.8.2 Text Commands for Cavity Remeshing

The text interface commands for cavity remeshing are as follows:

`/mesh/cavity/add-zones` allows you to create a cavity for adding new zone(s) to the existing volume mesh.

`/mesh/cavity/merge-cavity` merges the specified cavity domain with the specified domain.

`/mesh/cavity/region` allows you to create a cavity to improve the existing volume mesh in the specified region.

`/mesh/cavity/remove-zones` allows you to create a cavity for removing zone(s) from an existing volume mesh.

`/mesh/cavity/replace-zones` allows you create a cavity for removing a set of zones from an existing volume mesh and replacing them with a new set of zones.

13.9 Manipulating Cell Zones

When TGrid creates a volume mesh (of any cell shape) from a boundary mesh, these cells will be grouped into cell zones (contiguous zones separated by boundaries). You can manipulate these zones to control further mesh generation or to duplicate an existing volume mesh to model a repeated geometry.

13.9.1 Active Zones and Cell Types

After the initial mesh is generated, all the cells are grouped into contiguous zones separated by boundaries. An artifact of the meshing algorithm is that a virtual zone is created outside the outer boundary, and it is always given a cell type of dead. This zone is automatically deleted upon completion of the initial mesh generation. If the initial mesh generation is interrupted for some reason, this zone will remain in the mesh until the initialization is completed. Other zone types available are fluid and solid.

The zone just inside the outer boundary is automatically set to be active and labeled a fluid zone, although you can change this type later. When refining the mesh, only the active zones are refined. By toggling the zones between active and inactive, you can refine different groups of zones independently, using different mesh parameters for the different groups.

If you plan to refine all cell zones using the same refinement parameters, change the Non-Fluid Type in the Tri/Tet panel (see Section 11.5: The Tri/Tet Panel) before initializing the mesh. If you change the Non-Fluid Type to any type other than dead, all zones will be set active automatically after the initialization occurs. This eliminates the need for you to set all zones to be active in the Cell Zones panel. See also the automatic activation method available through the text interface, described in Section 13.9.4: Text Commands for Manipulating Cell Zones.

13.9.2 Copying and Moving Cell Zones

If you are creating a mesh for a geometry that repeats periodically, you can simplify the meshing tasks. To do this, create the boundary and volume mesh for just one of the repeated sections. Copy the appropriate cell zone(s) to the required location(s). If the copy shares a boundary with the original zone (i.e., if the two zones are connected), ensure that the distribution of nodes is the same on the two overlaid boundaries.

A simplified case (copying a triangular cell zone for a 2D rectangular geometry) is illustrated in Figure 13.9.1. Here, the volume mesh was created for zone 1, and then copied and translated to create zone 2. The node distribution on the left boundary of zones 1 and 2 is the same as the distribution on the right boundary. Since the left boundary of zone 2 is overlaid on the right boundary of zone 1, there will be duplicate nodes. It is important that you merge these duplicate nodes.

The procedure for doing this is as follows:

1. Open the Merge Boundary Nodes panel (see Section 7.1.2: The Merge Boundary Nodes Panel).
2. Compare all nodes on both boundaries. To do this, select the two zones in both the Compare... and With... group boxes.
3. Disable Only Free Nodes for both the zones.
4. Click the Merge button to merge the duplicate nodes.

After the duplicate nodes on the two boundaries are merged, one of the two boundary face zones will be deleted automatically. Since duplicate faces are merged when the duplicate nodes are merged, one zone will no longer have any faces.

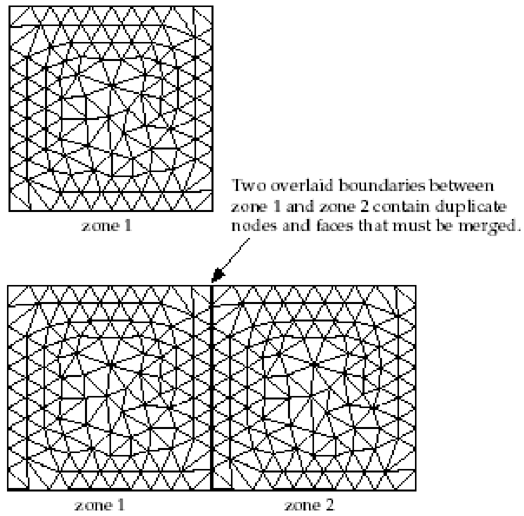
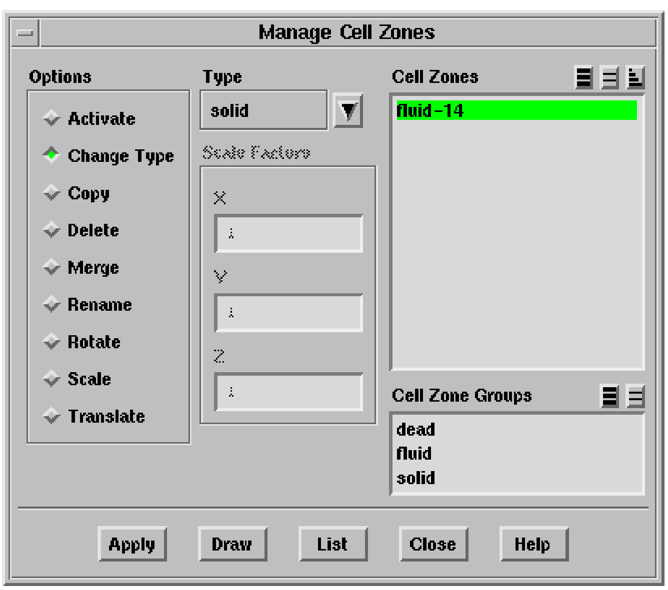


Figure 13.9.1: Copying and Translating a Cell Zone

13.9.3 The Manage Cell Zones Panel

The Manage Cell Zones panel allows you to change the cell type of a zone, set active cell zones, delete zones, and merge two or more zones into one. It also allows you to rename, rotate, scale, or translate one or more cell zones. The default zone name consists of a cell type (dead, fluid, or solid) and a zone ID number.

Mesh ⟶Manage...



Options contains the following zone manipulation options:

Activate activates the selected zone(s).

- When you click Apply, the selected zone(s) will be made active, and the zone(s) that are not selected will be made inactive.
- When you refine the mesh only cells in active zones will be refined.
- When you display the grid, the cells (including boundary cells), interior nodes, and interior faces that are displayed are those in the active zones. See Section 13.9.1: Active Zones and Cell Types for details.

Change Type sets the type of the selected cell zone to the type selected in the Type drop-down list.

Type contains a drop-down list of available cell zone types: dead, fluid, and solid.

FLUENT will read in the two live cell zone types (fluid and solid) and assign default conditions as described in the FLUENT User's Guide. Dead cell zones are regions that are not part of the computational domain. They will not be read by FLUENT.

Copy copies all nodes, faces, and cells of the selected zone (or zones), creating a new zone of the same type at the same location.

Delete deletes the selected cell zone(s, along with the associated faces and nodes. You may want to delete dead zones to free up some memoryin your computer. Commands for checking and deleting these faces are described in Section 7.15: Additional Boundary Mesh Text Commands.

This operation is irreversible. After deleting cell zones, check to see if there are any boundary faces located between two deleted zones. Such faces must be removed if you are generating a mesh for FLUENT.

Merge merges two or more selected zones into one. The resulting cell zone will have the name, type, and ID of the first selected zone that appears in the Cell Zones list. This operation is irreversible.

Merge Sub Zones controls whether or not same-type face zones and same-type node zones associated with the cell zones being merged should also be merged.

For example, if you merge two cell zones that are bounded by wall zones, the wall zone bounding one cell zone will be merged with the adjacent wall zone bounding the other cell zone.

Rename allows you to change the name of the selected zone.

Name specifies the new name for the zone selected in the Cell Zones list.

Rotate rotates the selected zone(s) through the specified angle. See Section 13.9.2: Copying and Moving Cell Zones for details.

Angle specifies the angle of rotation through which you want to rotate the selected zone(s).

axis allows you to specify the axis about which you want to rotate the selected zone(s).

pivot allows you to specify a pivot point about which you want to rotate the selected zone(s).

Copy Zone(s) allows you to copy the nodes of the selected zone(s), thereby creating new zone(s) of the same type before positioning the copied zone(s) per the specified Angle and axis (or pivot).

Scale scales the selected zone(s) by multiplying each of the node coordinates by the specified Scale Factors. The cell sizes will increase or decrease accordingly.

Scale Factors specifies the scale factors applied to the grid in each of the Cartesian coordinate directions (X, Y, (and in 3D) Z.

Copy Zone(s) allows you to copy the nodes of the selected zone(s), thereby creating new zone(s) of the same type before scaling the copied zone(s) per the specified scale factors.

Translate translates the selected zone(s) by the specified translation offsets. See Section 13.9.2: Copying and Moving Cell Zones for details.

Translation specifies the translation offsets (X, Y, Z) to be added to the Cartesian coordinate of every node in the selected zone(s).

Copy Zone(s) allows you to copy the nodes of the selected zone(s), thereby creating new zone(s) of the same type before positioning the copied zone(s) per the specified translation offsets.

Cell Zones contains a list from which you can select the zone(s) to be modified.

Cell Zone Groups contains a list of cell zone types. If you select a cell type from this list (e.g., fluid), all cell zones of that type will be selected in the Cell Zones list. This method allows you to easily select all cell zones of a certain type without having to select each zone individually. You can select multiple cell types in the Cell Zone Groups list to select all zones of several different types (e.g., solid and fluid).

Apply performs the selected option on the selected zones.

Draw displays the boundaries that define the selected cell zones. If any faces of a boundary are used in defining the zone, then the entire boundary is displayed.

List reports (in the console) the ID, name, type, and number of cells for each selected zone.

13.9.4 Text Commands for Manipulating Cell Zones

Text interface commands for manipulating cell zones are:

`/mesh/manage/active-list` lists all active zones.

`/mesh/manage/adjacent-face-zones` lists all face zones that refer to the specified cell zone.

`/mesh/manage/auto-set-active` sets the active zone(s) based on points that are defined in an external file. For each zone you want to activate, specify the coordinates of a point in the zone, the zone type (e.g., fluid), and a new name. This command is valid only for tri/tet meshes.

A sample file is as follows:

```
((1550.50 -466.58 896.41) fluid heater-#)
((1535.83 -643.14 874.71) fluid below-heater-#)
((1538.73 -444.28 952.69) fluid above-heater-#)
((1389.18 -775.51 825.97) fluid plenum-#)
```

Here, four fluid zones are identified, renamed, and activated. Any zone that you identify in the file will automatically be activated. The `#` (hash sign) indicates that TGrid should append the appropriate ID number for the zone.

`/mesh/manage/change-prefix` allows you to change the prefix of the cell zone.

`/mesh/manage/copy` copies all nodes and faces of specified cell zones.

`/mesh/manage/delete` deletes a cell zone, along with its associated nodes and faces.

`/mesh/manage/id` specifies a new cell zone ID. If a conflict is detected, the change is ignored.

`/mesh/manage/list` prints information on all cell zones.

`/mesh/manage/merge` merges two or more cell zones.

`/mesh/manage/name` allows you to rename a cell zone.

`/mesh/manage/origin` specifies a new origin for the mesh, to be used for cell zone rotation. The default origin is (0,0) or (0,0,0).

`/mesh/manage/revolve-face-zone` generates cells by revolving a face thread.

`/mesh/manage/rotate` rotates all nodes of specified cell zones by a specified angle.

`/mesh/manage/scale` scales all nodes of specified cell zones by a specified factor.

`/mesh/manage/set-active` sets the specified cell zones to be active.

`/mesh/manage/translate` translates all nodes of specified cell zones by a specified vector.

`/mesh/manage/type` allows you to changes the type of the cell zone.

`/mesh/separate/separate-cell-by-face` separates cells that are connected to a specified face zone into another cell zone. This separation method applies only to prism cells.

`/mesh/separate/local-regions/define` allows you to define the local region.

`/mesh/separate/local-regions/delete` deletes the specified local region.

`/mesh/separate/local-regions/init` creates a region encompassing the entire geometry.

`/mesh/separate/local-regions/list-all-regions` lists all the local regions defined.

`/mesh/separate/separate-cell-by-mark` separates cells within a specified local region into another cell zone.

`/mesh/separate/separate-cell-by-region` separates contiguous regions within a cell zone into separate cell zones.

`/mesh/separate/separate-cell-by-shape` separates cells with different shapes (pyramids, tetrahedra, etc.) into separate cell zones.

`/mesh/separate/separate-cell-by-size` separates cells based on the specified minimum and maximum cell sizes.

`/mesh/separate/separate-cell-by-skew` separates cells based on the specified cell skewness.

13.10 Using Domains to Group and Mesh Boundary Faces

Domains allow you to group different boundary zones together so that you can create tetrahedral meshes in the region they enclose, or you can limit the zones available for a display or report to only those zones in a selected subset of the domain, rather than the entire domain.

13.10.1 Using Domains

If you are generating a hybrid mesh containing quads and tris (in 2D) or hexas, tets, and pyramids (in 3D), identify a domain of the global mesh as the region in which you want to generate triangular or tetrahedral cells. You can also use domains to group boundary zones so that you can perform diagnostics on them or display them. When you display the grid, the zones available for display will be only those zones that are included in the active domain. Similarly, diagnostic reports will report information about only those zones.

- If you want to check a subset of the global domain, you can create and activate a domain that includes the desired zones, and then proceed with the display or report.
- If you want your grid display or report to include all zones in the mesh, be sure to activate the global domain in the Domains panel.

13.10.2 Defining Domains

The procedure for defining a new domain is as follows:

1. Deselect all zones in the Boundary Zones list and click New. It is quicker to create an empty domain and then add the zones you want, instead of creating a domain with many zones and then removing those you do not want.
2. In the Boundary Zones list, select the zones you want to include in the new domain. If you are not sure about the zones, click Draw to display the zones that are currently selected in the Boundary Zones list.

   It is possible to select all triangular or quadrilateral boundary face zones by choosing tri or quad in the Boundary Zone Groups list.

   - If you are creating a domain within a hybrid mesh to create triangles or tetrahedra, make sure that the domain contains all zones required to enclose the region that is to be meshed with tris/tets, and *only* those zones.
   - If the zones you select do not completely enclose the region, or if you include additional zones that do not bound this region, the tri/tet meshing is likely to fail or be incorrect.
3. Click on the Change button. TGrid will update the panel so that the node, face, and cell zones highlighted in their respective lists are those that are affiliated with the boundary zones in the domain.
4. Select the domain that you want to mesh (or display or report on) in the Domains list, and then click the Activate button. This domain is then considered to be the active domain, and the Activate button is disabled until you select another domain.

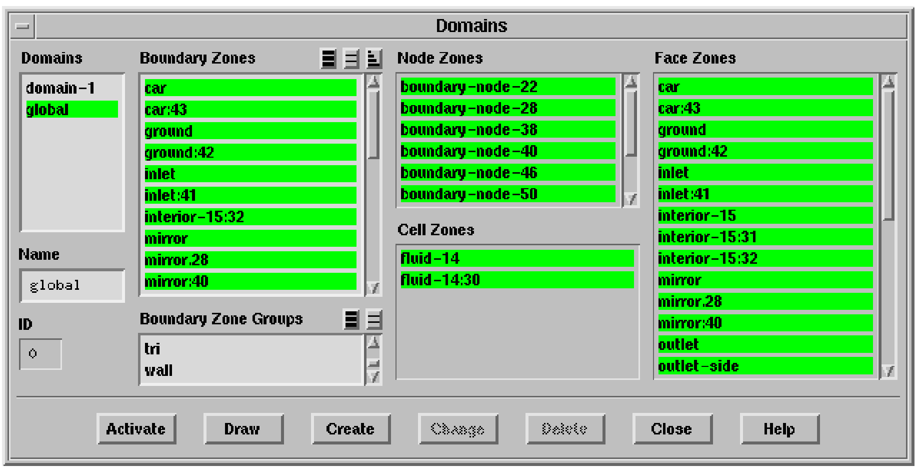
By default, the most recently created domain is automatically set to be the active domain, so you only need to explicitly set the active domain if it is not the one you just created.

Note: *If you are unsatisfied with a domain definition, delete it using the* Delete *button. Then either start over, or modify it by selecting it and performing steps 2 and 3.*

13.10.3 The Domains Panel

The Domains panel allows you to create domains encompassing the region(s) of a hybrid mesh that are to be meshed with tetrahedra or triangles, or domains containing boundary zones that you wish to group for other reasons. See Sections 13.10.1 and 13.10.2 for details.

Mesh ⟶Domains...



Controls

Domains contains a selectable list of all domains that have been defined, including the global domain (containing all zones in the mesh).

Name sets the name of the new domain you are about to create by clicking on the New button, or the name of the selected domain that you are about to change by clicking on the Change button.

ID shows the number of the selected domain. The global domain has an ID of 0, the next domain to be created has an ID of 1, and so on.

Boundary Zones contains a list from which you can select the zone(s) to be included in the domain.

Boundary Zone Groups contains a list of boundary zone types. If you select a boundary type from this list (e.g., inlet), all boundary zones of that type (for this example, all pressure-inlet and velocity-inlet boundaries) will be selected in the Boundary Zones list. This allows you to easily select all boundary zones of a certain type without having to select each zone individually. It is also possible to select all triangular or quadrilateral face zones, by choosing tri or quad in the Boundary Zone Groups list. You can select multiple boundary types in the Boundary Zone Groups list to select all zones of several different types (e.g., inlet and outlet).

Node Zones shows the names of all node zones in the domain. Those that are included in the currently selected domain (based on your selections in the Boundary Zones panel) will be highlighted. The highlighting will be updated when you click on the Change button after modifying your selections in the Boundary Zones list.

Cell Zones shows the names of all cell zones in the global domain. Those that are included in the currently selected domain (based on your selections in the Boundary Zones panel) will be highlighted. The highlighting will be updated when you click on the Change button after modifying your selections in the Boundary Zones list.

Face Zones shows the names of all face zones in the domain. Those that are included in the currently selected domain (based on your selections in the Boundary Zones panel) will be highlighted. The highlighting will be updated when you click on the Change button after modifying your selections in the Boundary Zones list.

For Node Zones, Cell Zones, and Face Zones you cannot modify the face zones in a given domain by changing this list. This list shows (by highlighting) the node zone, cell zones, and face zones respectively, that are contained in the domain as you have defined it.

Activate sets the domain selected in the Domains list to be the "active" domain. When you create the volume mesh, only those zones that are in the active domain will be meshed. When you display the grid, the zones available for display will be only those zones that are included in the active domain.

Similarly, diagnostic reports will report information about only those zones. If you want to check a subset of the global domain, you can create and activate a domain that includes the desired zones, and then proceed with the display or report. If you want your grid display or report to include all zones in the domain, be sure to activate the global domain.

The Activate button will be disabled as soon as a domain is activated. It will become available again if you select a different domain. When a new domain is created, it will automatically be identified as the active domain and the Activate button will be disabled.

Draw displays the grid for the selected Boundary Zones, based on the current settings in the Display Grid panel (see Section 14.1.3: The Display Grid Panel).

Create creates a new domain.

Change changes the definition of the selected domain to the definition currently shown in the panel.

Delete deletes the selected domain definition. The individual zones that were in the domain are not affected. You can only delete the *grouping* of these zones that was defined by the domain.

13.10.4 Text Commands for Domains

Text commands for domain creation and activation are as follows:

`/mesh/domains/activate` activates the specified domain for meshing/reporting operations.

`/mesh/domains/create` creates a new domain based on the specified boundary face zones.

`/mesh/domains/create-from` creates a new domain based on the specified cell zones.

`mesh/domains/delete` deletes the specified domain.

`mesh/domains/draw` displays the boundary face zones of the specified domain.

`mesh/domains/print` prints the zones comprising the specified domain.

13.11 Checking the Mesh

When you have completed the mesh generation process, you need to check the mesh before saving it.

Mesh ⟶Check

Alternatively, you can use the command `/mesh/check` to check the mesh.

When you select the Mesh/Check menu item, TGrid will check the mesh connectivity and the orientation of the faces (face handedness, which should be right-handed for all faces because the solvers use a right-handed system).

TGrid will report any problems it finds, along with the domain extents and the maximum and minimum cell volumes (which must not be negative).

The sample output is as follows:

```
Domain extents.
  x-coordinate: min = -1.016026e+02, max = 2.621100e+02.
  y-coordinate: min = -3.513216e+02, max = 2.000000e+02.
  z-coordinate: min = -1.016057e+02, max = 1.016057e+02.
Volume statistics.
  minimum volume: 6.357072e-03.
  maximum volume: 1.990373e+03.
    total volume: 7.620820e+06.
Face area statistics.
   minimum face area: 5.638886e-02.
   maximum face area: 2.900312e+02.
   average face area: 1.963704e+01.
Checking number of nodes per cell.
Checking number of faces per cell.
Checking cell faces.
Checking face handedness.
Done.
```

If any problems are reported, use the Display Grid panel (see Section 14.1.3: The Display Grid Panel) to find out where the problem faces or cells are located. If you have used domains to generate the mesh or group zones for reporting (as described in Section 13.10: Using Domains to Group and Mesh Boundary Faces), only the mesh in the active domain will be checked.

13.12 Clearing the Mesh

If you are dissatisfied with the mesh generated, you can choose to clear the mesh and start again from the boundary mesh. When TGrid clears the mesh, it deletes all interior nodes and faces, and all cells both live and dead. Only the boundary nodes and faces will be left. After the mesh is cleared, you can generate a new mesh.

This feature is available with the Clear menu item in the Mesh pull-down menu.

Mesh ⟶Clear

You can also use the text command `/mesh/clear-mesh`. To delete the boundary mesh, use the text command `mesh/reset-mesh`. When you use either of these commands TGrid will ask you to confirm that you want to clear or reset the mesh.

i If you have used domains to generate the mesh or group zones for reporting (as described in Section 13.10: Using Domains to Group and Mesh Boundary Faces), only the mesh in the active domain will be cleared.