Chapter 9. Creating a Mesh

After reading the boundary mesh and performing the necessary modifications (e.g., duplicate node merging, edge swapping) you will create the volume mesh.

Depending on the type of mesh you are starting from (boundary mesh only, boundary mesh with hexahedral cells in one or more regions, etc.), you can automatically generate the mesh, manually generate the mesh step by step, or use a combination of manual and automatic commands.

TGrid *allows you to create several types of meshes. The meshing strategy and the use of the* Auto Mesh *tool are described in this chapter. The quad-tet transition elements (pyramid and non-conformal meshing) and creation of a heat exchanger zone are also described. Advanced meshing techniques such as prism meshing, triangular/tetrahedral and hexcore meshing options, etc. are described in subsequent chapters.*

- Section 9.1: Choosing the Meshing Strategy
- Section 9.2: Using the Auto Mesh Option
- Section 9.3: Generating Pyramids
- Section 9.4: Creating a Non-Conformal Interface
- Section 9.5: Creating a Heat Exchanger Zone

9.1 Choosing the Meshing Strategy

TGrid offers a variety of cell shapes that can be used to mesh the domain, e.g., tetrahedra, prisms, hexahedra, and pyramids in 3D and triangles and quadrilaterals (2D prisms) in 2D. These shapes are shown in Figure 9.1.1.

A mesh consisting entirely of triangular elements is referred to as a triangular mesh, a mesh with only tetrahedral elements is a tetrahedral mesh, and a mesh with any combination of cell shapes is referred to as a hybrid mesh.

Before generating a volume mesh, determine the shapes that are appropriate for the case you are solving. Then follow the instructions for creating the required cell types. Most cases will fall into one of the following categories:

- Starting from a 3D boundary mesh containing only triangular faces (see Section 9.1.1: 3D Boundary Mesh Containing Only Triangular Faces)

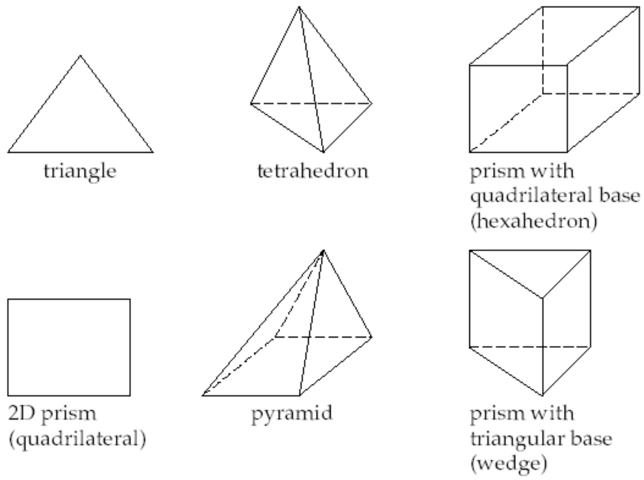


Figure 9.1.1: Different Cell Shapes

- Starting from a 3D boundary mesh (see Section 9.1.2: 3D Boundary Mesh)
- Starting from a 2D boundary mesh (see Section 9.1.3: 2D Boundary Mesh)
- Starting from a 2D boundary mesh with some quadrilateral cells (see Section 9.1.4: 2D Boundary Mesh with Some Quadrilateral Cells)
- Generating a hexcore volume mesh (see Section 9.1.5: Generating a Hexcore Volume Mesh)

9.1.1 3D Boundary Mesh Containing Only Triangular Faces

If you require a high mesh resolution in some portion of the domain, such as a boundary layer, you can obtain an efficient and better quality mesh by meshing that portion with prisms (wedges) and then meshing the rest of the domain with tetrahedra (tets). The resulting mesh is referred to as a viscous hybrid mesh.

The procedure is as follows:

1. Build one or more layers of prisms, starting from the appropriate boundary (or boundaries). See Chapter 10: Generating Prisms.
2. Create a domain encompassing the region to be meshed with tets.
3. Generate the tets in the selected domain using either automatic, or manual tet mesh generation, or a combination of the two. See Chapter 11: Generating Triangular/Tetrahedral Meshes.

Figure 9.1.2 shows several layers of prisms in a portion of a mesh created in this manner. The prisms extends throughout the entire region bounded by the quadrilateral faces, but only a few of them are shown here.

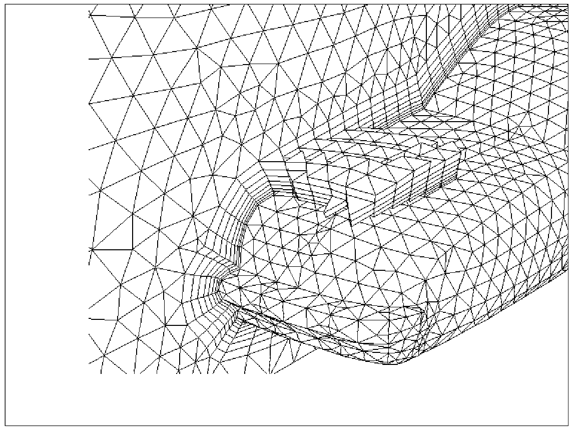


Figure 9.1.2: Grid with Prisms in a Boundary Layer Region

The surface mesh originally contained only triangular faces. The quadrilateral faces are created automatically when the prisms are built on the triangular faces.

If the quadrilateral faces of the prisms do not lie on the external boundary of the domain (i.e., if the prism region begins and/or ends in the interior of the domain), create a layer of transitional pyramids between steps 1 and 2. See Section 9.3: Generating Pyramids for details.

If you have no special boundary layer resolution requirements, you can generate a mesh consisting entirely of tets (see Figure 9.1.3). You can use the automatic tet mesh generation procedure, the manual procedure, or a combination of both. See Chapter 11: Generating Triangular/Tetrahedral Meshes for details.

9.1.2 3D Boundary Mesh

Start from a 3D boundary mesh containing triangular and quadrilateral faces, as well as hexahedral cells in the quadrilateral face regions. The resulting mesh is referred to as a zonal hybrid mesh.

1. Add a layer of pyramids to the quadrilateral boundary face zone that lies between the hexahedral region and the adjacent region to be meshed with tets. This creates the triangular boundary face zone that is required to create tets in the adjacent region. See Section 9.3: Generating Pyramids for details.

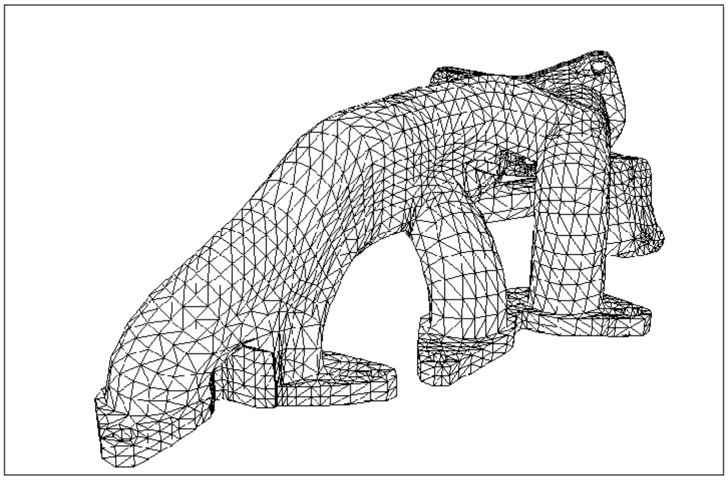


Figure 9.1.3: Surface Mesh for a Grid Containing Only Tetrahedra

2. Create a domain encompassing the region to be meshed with tets. See Section 13.10: Using Domains to Group and Mesh Boundary Faces for details.

3. Generate the tets in the selected domain using either automatic or manual tet mesh generation, or a combination of both. See Chapter 11: Generating Triangular/Tetrahedral Meshes for details.

Figure 9.1.4 shows the surface mesh for a portion of a grid containing hexahedra, pyramids, tetrahedra, and prisms that was created on a plenum feeding a valve-port cylinder.

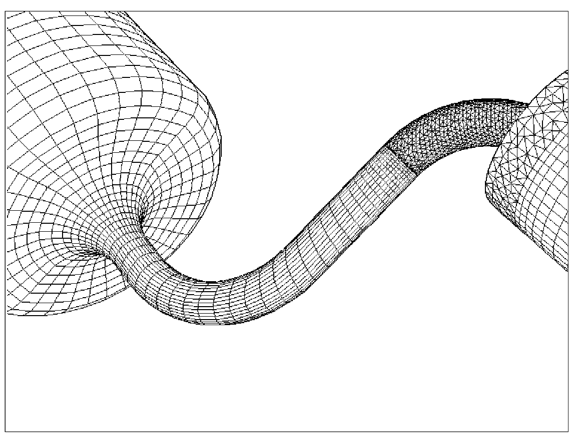


Figure 9.1.4: Surface Mesh

- The less complicated plenum pipe on the left of the figure is meshed using hexahedral cells.
- The more complex valve port (valve not visible because it is inside the surrounding pipe) is meshed using tetrahedra.
- Pyramids are used as a transition between the hexahedral grid for the plenum and the tetrahedral grid for the valve port.

This transition occurs where the triangular and quadrilateral faces meet in the middle of the figure. Additionally, the quadrilateral faces produced by extending triangular faces in the cylinder (i.e., the quadrilateral sides of the resulting prism wedges) can be seen in the far right of the figure.

9.1.3 2D Boundary Mesh

If you want to have a high mesh resolution in some portion of the domain, such as a boundary layer, you can obtain a better-quality and more efficient mesh by meshing that portion with prisms and then meshing the rest of the domain with triangles (tris). Figure 9.1.5 shows a grid created in this manner.

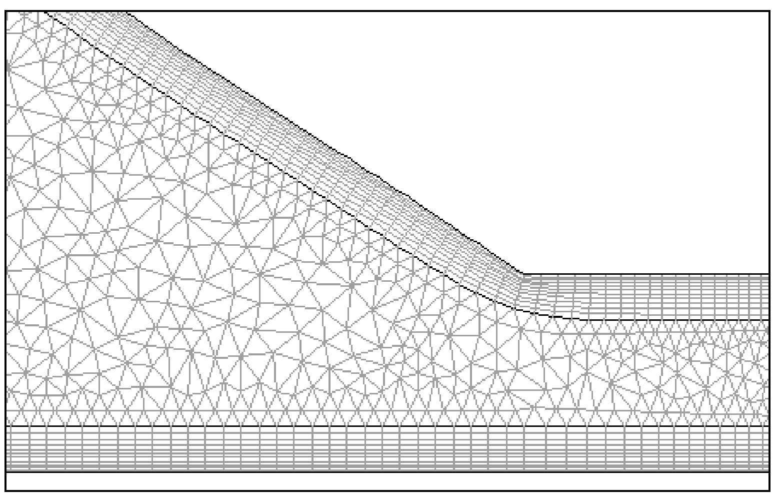


Figure 9.1.5: 2D Mesh Containing Prisms (Quadrilaterals) and Triangles

1. Build one or more layers of prisms (since it is a 2D domain, the prisms will be quadrilateral cells), starting from the required boundary/boundaries. See Chapter 10: Generating Prisms for details.
2. Create a domain encompassing the region to be meshed with tris. See Section 13.10: Using Domains to Group and Mesh Boundary Faces for details.

3. Generate the tris in the selected domain, using either automatic or manual tri mesh generation, or a combination of both. See Chapter 11: Generating Triangular/Tetrahedral Meshes for details.

If you have no special boundary layer resolution requirements, you can generate a mesh consisting entirely of tris (Figure 9.1.6). Use the automatic tri mesh generation procedure, the manual procedure, or a combination of both.

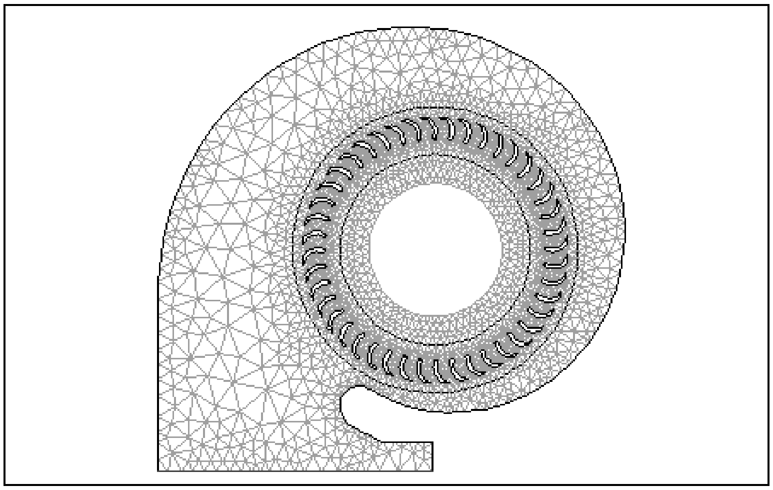


Figure 9.1.6: 2D Mesh Containing Only Triangles

9.1.4 2D Boundary Mesh with Some Quadrilateral Cells

1. Create a domain encompassing the region to be meshed with tris (Section 13.10: Using Domains to Group and Mesh Boundary Faces).
2. Generate the tris in the selected domain, using either automatic or manual tri mesh generation, or a combination of both (Chapter 11: Generating Triangular/Tetrahedral Meshes).

The result of this procedure is similar to the example shown in Figure 9.1.5. The only difference is that the quadrilateral cells in Figure 9.1.5 are created in TGrid, while here they are considered to be created elsewhere and imported.

9.1.5 Generating a Hexcore Volume Mesh

The *hexcore* mesh features a tet/hybrid mesh adjacent to walls and a Cartesian mesh in the core flow region. Figure 9.1.7 shows the typical hexcore mesh. The hexcore meshing scheme creates a mesh consisting of two regions:

- An inner region composed of regular Cartesian cells.
- An outer region consisting of tetrahedral elements.

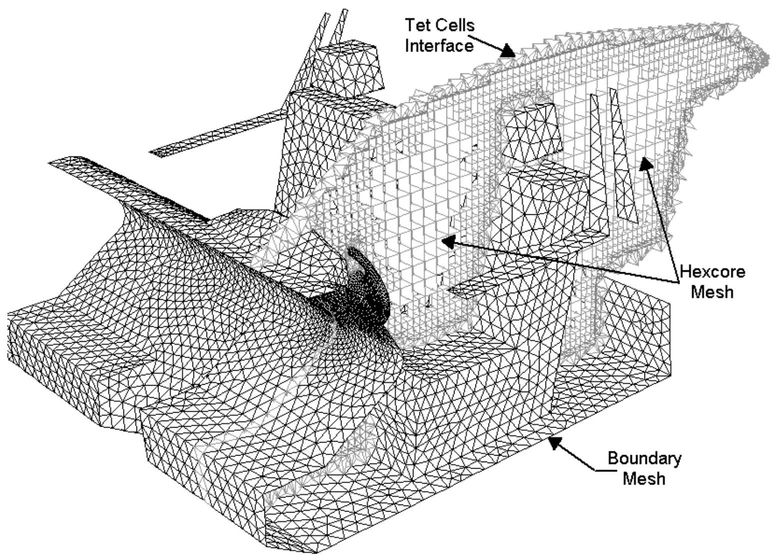


Figure 9.1.7: Hexcore Mesh

Wedge elements are created only when boundary layers are attached on faces pre-meshed with triangular elements. It combines the automation and geometric flexibility of tet/hybrid meshes with greatly reduced cell counts in many applications. Hexcore is most beneficial in geometries with large open spaces as in automotive, aerospace, and HVAC applications. See Chapter 12: Generating the Hexcore Mesh for details.

9.1.6 Additional Meshing Tasks

Additional meshing tasks that TGrid can handle are:

- If you have a complete volume mesh and want to extend some portion of the domain (e.g., increase the length of an inlet pipe), you can *grow* one or more layers of prisms from the current external (quadrilateral or triangular) boundary to be extended.

  Figure 9.1.8 shows a region of prisms (wedges) extended from the triangular face zone that bounds a tetrahedral region.

- Unless there is a reason to use hexahedral cells in the quad regions, it is preferable to convert a mixed tri/quad boundary mesh to an all-tri boundary mesh and then create a tet mesh for a 3D boundary mesh consisting of only triangular faces.

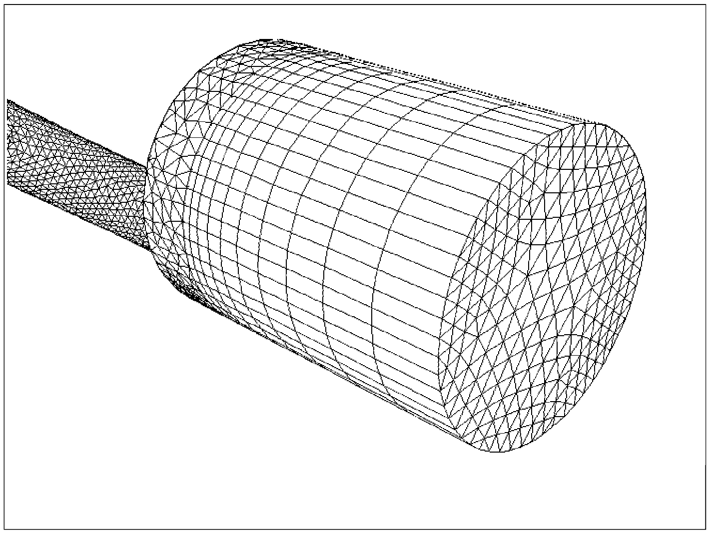


Figure 9.1.8: Extending an Existing Tetrahedral Mesh Using Prisms

Use the Triangulate Zones panel (see Section 7.9.1: The Triangulate Zones Panel) or the command `/boundary/remesh/triangulate` to convert quad face zones to tri face zones:

You can then use the Boundary Improve panel to improve the skewness of the triangular boundary zone created.

- You may choose to use a non-conformal interface to define the computational domain. This type of interface allows you to relax the requirement for point-to-point matching at the interface between the grids, as illustrated in Figure 9.1.9.

  This feature of relaxing the requirement for point-to-point matching at the interface between the grids is particularly useful in parametric studies where you want to change an isolated region of the domain without changing the entire mesh. The procedure is as follows:

  1. Prepare two separate grids. These grids should not share nodes, edges, faces, or cells.
  2. Read both the grids into TGrid.
  3. For surface grids, define a domain for each independent grid region and mesh the individual domains using an appropriate meshing strategy.

     Mesh ⟶Domains...

     For volume grids, change the face zone type of the two surfaces that will be treated nonconformally to interface.

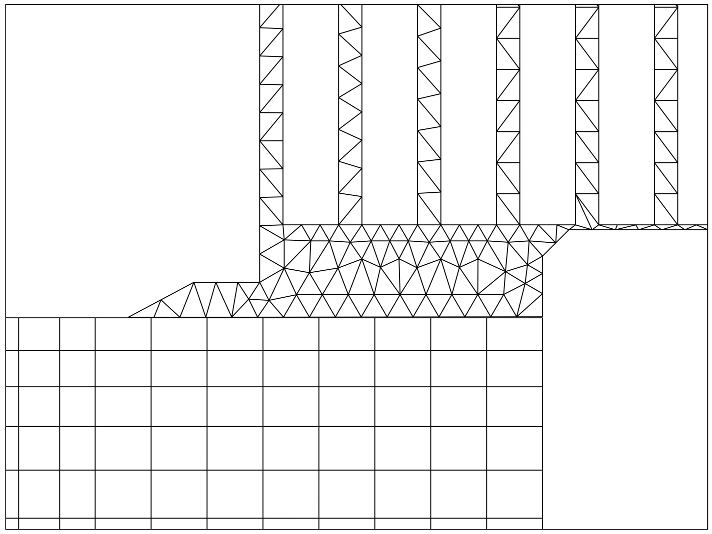


Figure 9.1.9: Example of a Non-Conformal Interface

Boundary ⟶Manage...

4. Write the grid to a file.
5. Read the combined grid file into FLUENT and create the non-conformal interface using the Grid Interfaces option.

9.1.7 Inserting Isolated Nodes into a Tri or Tet Mesh

To add nodes to the mesh without also specifying the faces, you can create a boundary zone in the boundary mesh (in the program that created it) just for this purpose. Then in TGrid you can introduce the nodes associated with these faces.

This feature is useful for clustering nodes (and therefore cells) in a controlled manner. Figure 9.1.10 shows a grid that was generated using this method to cluster the nodes behind the wedge. A grid that was generated for the same geometry without clustering is shown in Figure 9.1.11. You can follow either of the two procedures to insert isolated nodes into a mesh:

- Delete the face zone before meshing.
- Introduce the nodes using face zones after meshing.

Delete the Face Zone Before Meshing

This procedure is as follows:

1. Delete the face zone, but leave the associated nodes. Disable Delete Nodes in the Manage Face Zones panel to retain these nodes.

   i By default, the unused nodes are deleted when the faces of the zone are deleted.

2. Disable Delete Unused Nodes in the Tri/Tet panel and generate the volume mesh.

   i By default, unused nodes will be deleted during the automatic meshing.

   The nodes will be introduced when you initialize the mesh. When you use this procedure, *all* nodes must be inserted into the mesh or the initialization will fail.

   ⚠ Do not place isolated nodes too close to the boundary or to other nodes.

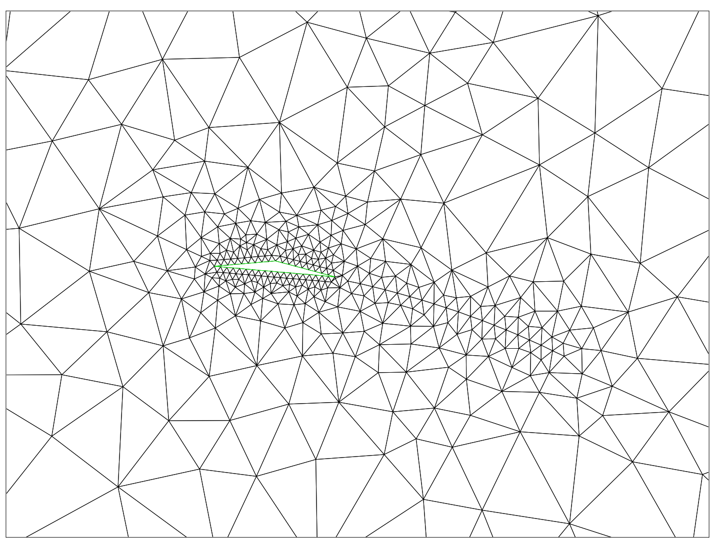


Figure 9.1.10: Mesh Generated Using Isolated Nodes to Concentrate Cells

Introduce the Nodes Using Face Zones After Meshing

This procedure is as follows:

1. Create a subdomain that does not include the face zones used to control the mesh density.

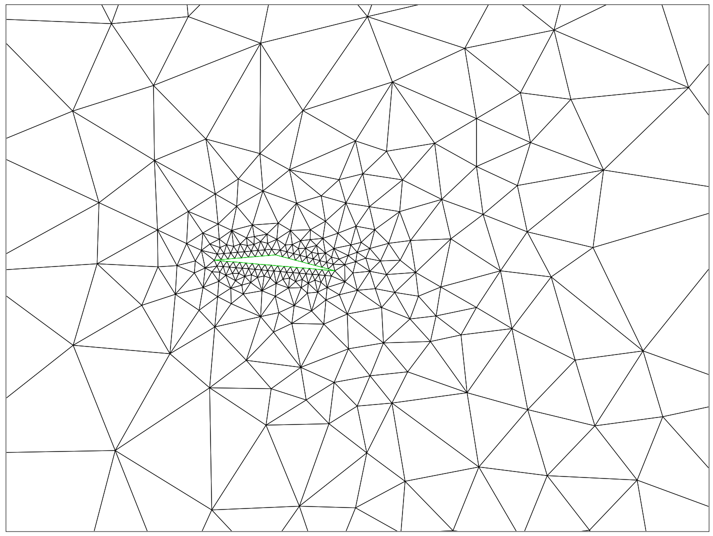


Figure 9.1.11: Mesh Generated Without Using Isolated Nodes

2. Create the volume mesh using the subdomain.

3. Activate the global domain.

4. Introduce the additional nodes using the text command: `mesh/modify/mesh-nodes-on-zone`.

   i The `mesh/modify/mesh-nodes-on-zone` command will delete the faces associated with the face zone.

9.2 Using the Auto Mesh Option

The Auto Mesh panel (see Section 9.2.1: The Auto Mesh Panel) allows you to automatically create the volume mesh in TGrid using the different mesh elements available. The generic procedure for using the Auto Mesh panel for creating the mesh comprises the following steps:

1. Determine the mesh elements required for the particular case. Refer to Sections 9.1.1—9.1.5 for details on choosing the meshing strategy.

2. If you need to grow prisms for the geometry under consideration, click the Set... button in the Boundary Layer Mesh group box to open the Prisms panel. Set the appropriate prism parameters and click Apply in the Prisms panel. Refer to Section 10.3: Prism Meshing Options for details on the prism meshing options available

in TGrid. Verify that the Prisms check box is enabled in the Boundary Layer Mesh group box in the Auto Mesh panel.

3. Select the appropriate quad-tet transition elements from the Quad Tet Transition list. Click the Set... button to open the Pyramids panel (see Section 9.3.3: The Pyramids Panel) or the Non Conformals panel (see Section 9.4.1: The Non Conformals Panel) (depending on the selection) and specify the appropriate parameters. Refer to Sections 9.3.1 and 9.4 for details.

4. Select the appropriate option from the Volume Fill list. Click the Set... button to open the Tri/Tet panel (see Section 11.5: The Tri/Tet Panel) or the Hexcore panel (see Section 12.4: The Hexcore Panel) (depending on the selection) and specify the appropriate parameters. Refer to Sections 11.3, 11.4, and 12.3 for details.

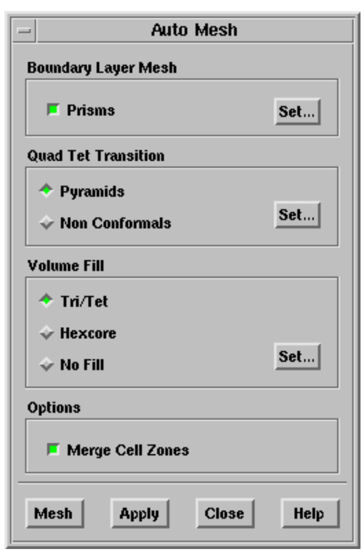
5. Enable Merge Cell Zones, if desired.

6. Click Mesh to automatically create the mesh.

Alternatively, you can use the command `/mesh/auto-mesh` to generate the mesh automatically. You need to specify the meshing parameters for the respective mesh elements (prisms, pyramids or non-conformals, tri/tet or hex) using either the respective panels or the associated text commands prior to using the `auto-mesh` command. TGrid will prompt you to specify the mesh elements required when the `auto-mesh` command is invoked. The previously set parameters will be used to create the mesh.

9.2.1 The Auto Mesh Panel

The Auto Mesh panel allows you to automatically create the volume mesh. You can specify the mesh elements to be used and set appropriate parameters for the same.



Controls

Boundary Layer Mesh contains options for creating the boundary layer (prism) mesh.

Prisms allows you to create prism layers in the geometry.

Set... opens the Prisms panel.

Note: *The* Prisms *option in the* Auto Mesh *panel will be visible only after applying zone-specific growth (by clicking* Apply *in the* Zone Specific Growth *group box in the* Prisms *panel (see Section 10.10: The* Prisms *Panel)).*

Quad Tet Transition contains options for specifying the quad-tet transition elements for a hybrid mesh.

Pyramids specifies that pyramids will be created as transitional elements between the quadrilateral and triangular cells.

Non Conformals specifies that a non-conformal interface will be created as a transition between the quadrilateral and triagular cells.

Set... opens the Pyramids panel (see Section 9.3.3: The Pyramids Panel) or the Non Conformals panel (see Section 9.4.1: The Non Conformals Panel), as appropriate.

Volume Fill contains options for volume meshing.

Tri/Tet allows you to mesh the geometry with triangular/tetrahedral/hybrid mesh.

Hexcore allows you to mesh the geometry with tetrahedral/hybrid mesh adjacent to the walls and hexahedral mesh in the core flow region.

No Fill allows you to mesh the boundary layers with prisms and pyramids (or non-conformals) as specified, without creating tetrahedral/hexahedral cells.

Set... opens the Tri/Tet panel (see Section 11.5: The Tri/Tet Panel) or the Hexcore panel (see Section 12.4: The Hexcore Panel), as appropriate.

Options contains additional meshing options.

Merge Cell Zones toggles the merging of cell zones during meshing.

9.3 Generating Pyramids

A pyramid has a quadrilateral face as its base and four triangular faces extending from the sides of the quadrilateral up to a single point above the base. See Figure 9.3.1.

To generate a conformal mesh with a region of tetrahedral cells adjacent to a region of hexahedral cells in TGrid, you will first create a layer of pyramids as a transition from quadrilateral faces to triangular faces. After creating a single layer of pyramids, TGrid will use the resulting triangular faces to create tetrahedra.

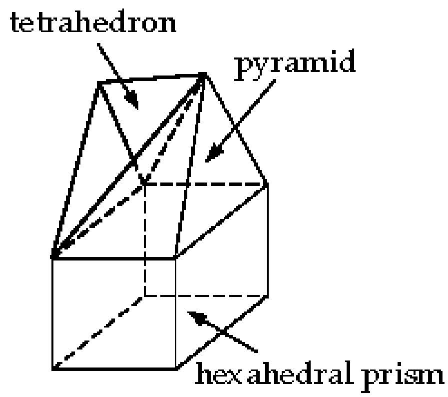


Figure 9.3.1: Pyramid Cell—Transition from a Hexahedron to a Tetrahedron

To create pyramids, you need to specify the boundary from which the pyramids will be built, the method for determining the top vertex of each pyramid, and the pyramid height.

9.3.1 Creating Pyramids

The procedure for creating a layer of pyramids from a quadrilateral boundary zone is as follows:

1. Check the aspect ratio limits of the boundary face zones on which you need to build pyramids.

   Report ⟶Face Limits...

   The use of a high-aspect-ratio quadrilateral for the base of a pyramid produces skewed triangular faces that can cause problems during the tetrahedral mesh generation. If the maximum quadrilateral face aspect ratio is much greater than 10, you will need to regenerate them.

   - If the faces were created in a different preprocessor, return to that application and try to reduce the aspect ratio of the faces in question.
   - If the faces were created by TGrid during the building of prism layers, rebuild the prisms using a more gradual growth rate.

2. Select the appropriate quadrilateral boundary zone(s) in the Boundary Zones selection list in the Pyramids panel (see Section 9.3.3: The Pyramids Panel).

   Mesh ⟶Pyramids...

   Click Draw to view the selected zones.

   If the quadrilateral zone you require does not appear in the list, use the `boundary/reset-element-type` text command to update the type of the zone.

It is possible that TGrid may not recognize the quadrilateral zone due to changes made to the boundary mesh.

For example, if you separate a mixed (tri and quad) face zone into a tri face zone and a quad face zone, TGrid will continue to identify each of these as a mixed zone. You need to reset the element type for the quad zone for TGrid to recognize it and include it in the Boundary Zones selection list.

3. Select the appropriate method for determining the pyramid vertex location in the Options list. The skewness method is selected by default, and is appropriate for most cases.

4. Specify the height of the pyramids by setting the Offset Scaling value.

5. Enable Fill Cap, if required. The Fill Cap option inserts a tetrahedral cell between the exposed pyramid faces satisfying the Face Angle criterion (see Figure 9.3.2).

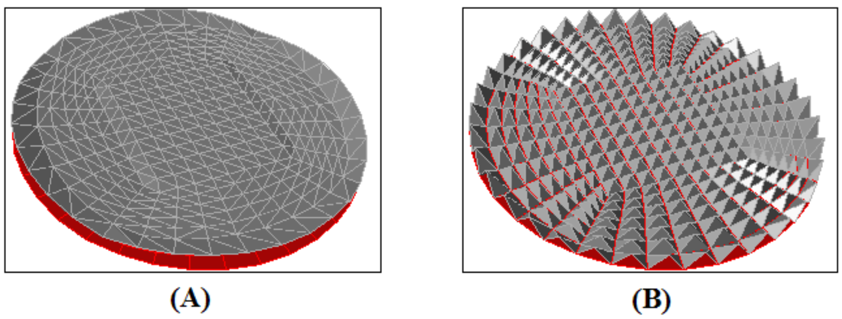


Figure 9.3.2: Pyramid Cap (A) Using the Fill Cap Option (B) Without the Fill Cap Option

6. Click Create. TGrid will report the new pyramid cell zone and the new face zones in the console window. You can then use the Display Grid panel (see Section 14.1.3: The Display Grid Panel) to view these new zones.

Display —→Grid...

7. Change the boundary type of the quadrilateral base zone to the appropriate type (if necessary).

Boundary —→Manage...

Since you built cells next to the quadrilateral base zone, its original boundary type may no longer be correct.

The pyramids should be automatically created on the appropriate side of the specified boundary zone(s). If the pyramids are on the wrong side, do the following:

1. Delete the newly created zones related to the pyramids.
2. Reverse the normal direction on the quadrilateral boundary where the pyramids are being built (using the Flip Normals option in the Manage Face Zones panel).
3. Recreate the pyramids.

9.3.2 Zones Created During Pyramid Generation

When TGrid generates pyramids, it will create at least two new zones: a cell zone containing the pyramid cells and a face zone containing the triangular faces of the pyramids.

TGrid will create the following zones:

- The cell zone containing the pyramids (`pyramid-cells-n`).
- The face zone containing the triangular faces of the pyramid cells (`base-zone-pyramid-cap-n`).

  For example, if the pyramids were built from the quadrilateral face zone wall-4, they will be placed in a new zone called wall-4-pyramid-cap-9 (where the `9` is the zone number assigned by TGrid).
- The face zones containing the pyramid sides which use existing faces from the original boundary mesh (`base-zone-pyramid-side:n`), where, `n` is the zone number assigned by TGrid.

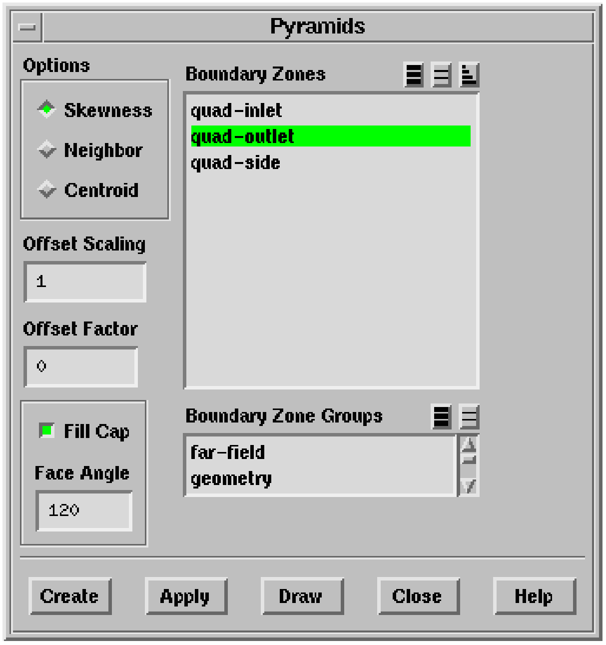
  For example, if TGrid uses triangular boundary faces from the zone wall-3, they will be placed in a new zone called wall-3-pyramid-side-6 (where the `6` is the zone number assigned by TGrid).

i If you include the `pyramid-side` boundary zone(s) when defining the domain in which you are going to generate a tetrahedral mesh, the tetrahedral meshing will fail.

9.3.3 The Pyramids Panel

The Pyramids panel allows you to specify all the parameters required for pyramid creation and to create the pyramids. See Section 9.3.1: Creating Pyramids for details about using the options in the Pyramids panel.

Mesh →Pyramids...



Controls

Options contains the options for determining the location of the new vertex of the pyramid.

Skewness specifies that the new vertex of a pyramid should be placed just above the centroid of the quadrilateral base, or the pyramid should share the vertex of a neighboring pyramid or a node on an adjacent boundary face, whichever will yield the best skewness (see Figures 9.3.3 and 9.3.4). This is the default option, and is the best choice for most cases.

Neighbor specifies that if the pyramid is adjacent to a triangular boundary face and the angle formed by the quadrilateral base and the triangular boundary face is not greater than 115 degrees. The pyramid should not use a node on that boundary as its new vertex, (Figure 9.3.3).

Otherwise, the new vertex should be placed just above the centroid of the quadrilateral base. If the node of a neighboring boundary face is used as the vertex, the height specification will be ignored for that pyramid.

Centroid specifies that the new vertex of the pyramid should always be placed just above the centroid of the quadrilateral base of the pyramid. See Figure 9.3.4.

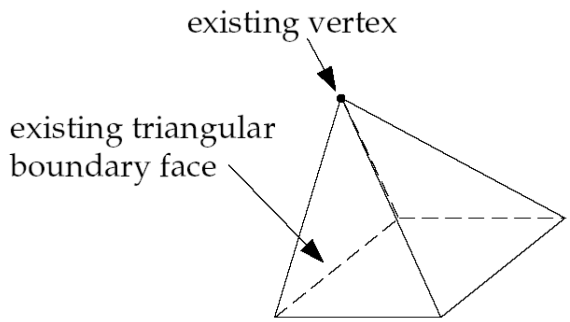


Figure 9.3.3: Using an Existing Node as the Pyramid Vertex

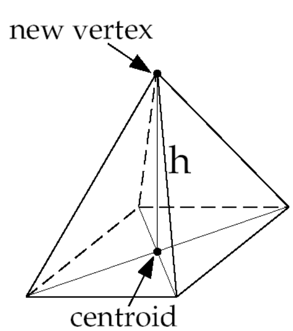


Figure 9.3.4: Pyramid Vertex Placed Above the Centroid

Offset Scaling specifies the scaling (s) to be used to determine the height of the pyramid. The height (h) is computed from the following equation:

$$h = \frac{sL}{\sqrt{2}} \tag{9.3-1}$$

where

$$L = \sqrt{A}$$

A = area of the quadrilateral base of the pyramid.

Offset Factor specifies the fraction of the computed pyramid height (offset) by which the pyramid heights will be randomly adjusted. The default value is 0, indicating that all pyramids will have the exact height computed by TGrid. A value of 0.1, for example, will limit each adjustment to $+/-10\%$ of the computed height.

Fill Cap specifies that the space between the pyramids will be automatically filled with tetrahedra. This option is enabled by default, since filling the pyramid cap with tetrahedra makes it easy to initialize the tetrahedral mesh based on these faces.

Face Angle specifies the dihedral angle between adjacent pyramid faces for which tetrahedral cells should be created (when the Fill Cap option is on). The default value of 120^o is acceptable for most cases. With a larger value for Face Angle, TGrid will create more tetrahedral cells in the pyramid cap. With a smaller value, fewer tetrahedral cells will be created.

Boundary Zones contains a list from which you can select the zone(s) to be operated on.

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. 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).

Create creates the pyramids you have defined.

Draw displays the boundary zone(s) selected in the Boundary Zones list. All current settings in the Display Grid panel will be in effect in the resulting grid display.

9.3.4 Text Interface for Generating Pyramids

Text commands for creating pyramid cells are as follows:

`/mesh/pyramid/controls/fill-angle` specifies the dihedral angle between adjacent pyramid faces for which tetrahedral cells should be created (when the `fill-cap?` option is enabled).

`/mesh/pyramid/controls/fill-cap?` enables the filling of the space between pyramids with tetrahedra.

`/mesh/pyramid/controls/neighbor-angle` sets the threshold dihedral angle used to limit the neighboring faces considered for pyramid creation. For example, if the value is set to 110 degrees and the angle between a given quadrilateral face and the neighboring triangular face is greater than 110 degrees, the resulting pyramid will not include the triangular face.

`/mesh/pyramid/controls/offset-factor` specifies the fraction of the computed pyramid height (offset) by which the pyramid heights will be randomly adjusted. The default value is 0, indicating that all pyramids will have the exact height computed by TGrid. A value of 0.1, for example, will limit each adjustment to $+/-10\%$ of the computed height.

`/mesh/pyramid/controls/offset-method` specifies the method by which offset distances are determined.

`mesh/pyramid/controls/offset-scaling` specifies the scaling (s) to be used to determine the height of the pyramid. See Equation 9.3-1.

`/mesh/pyramid/create` creates a layer of pyramid cells on the specified quadrilateral face zone.

9.3.5 Pyramid Meshing Problems

Most problems associated with creating pyramid layers manifest themselves in the subsequent process of generating the tetrahedral mesh.

Rapid Changes in Volume

Rapid changes in the sizes of cells have a negative influence on the convergence and accuracy of the numerical solution. The pyramid layer creation can produce rapid variations in cell volume. In any of the following situations:

- If the quadrilateral surface mesh has faces with rapid changes in size.

- If there is great disparity between the sizes of the quadrilateral faces and the neighboring triangular faces used in the pyramid creation.

You can avoid the rapid variation in volume by creating quadrilateral and neighboring triangular grids with smooth variations in face size.

Intersecting Faces

If the quadrilateral surface used to create pyramids has highly concave corners, the resulting pyramids may pierce each other and/or neighboring boundary faces (see Figure 9.3.5).

In such cases, you can either increase the resolution to prevent the intersections or alter the meshing strategy. An alternative is to separate the quadrilaterals in the concave corner into another zone (using the Separate Face Zones panel or the Modify Boundary panel), create triangular faces from the quadrilateral faces using the `boundary/create-tri-surface` text command, and then create pyramids.

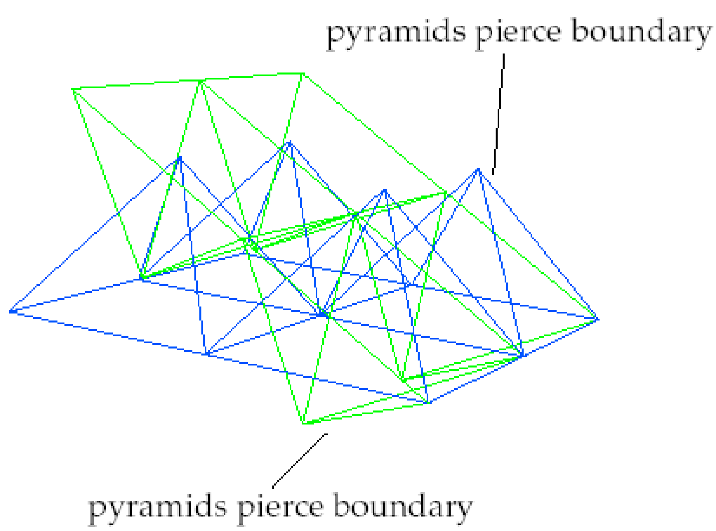


Figure 9.3.5: Pyramid Cells Intersecting Each Other and Boundary

The skewness-based pyramid creation will use the existing triangular faces and avoid the intersection problem (see Figure 9.3.6).

High Aspect Ratio

Creating pyramids on quadrilateral faces with very high aspect ratios results in highly skewed triangular faces. Subsequent attempts to create a tetrahedral mesh from these elements will produce a poor-quality mesh.

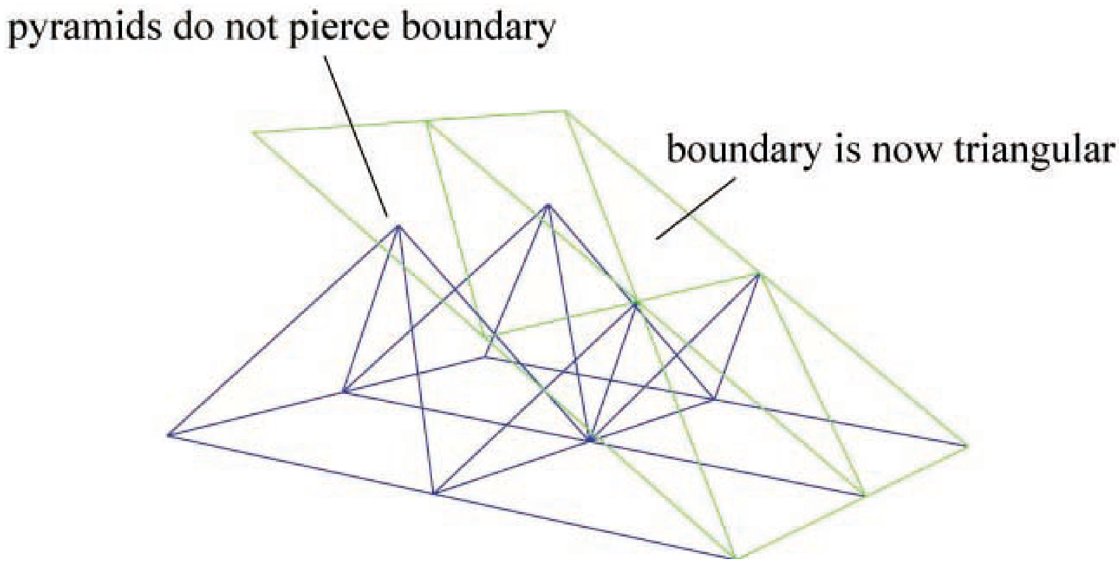


Figure 9.3.6: Fixed Intersecting Pyramid Cells Using Triangular Faces

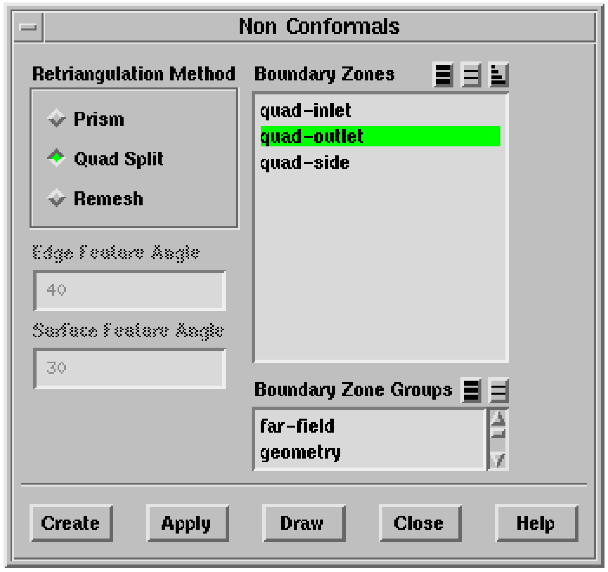
Irrespective of the method used to generate the quadrilaterals, modify the meshing strategy to reduce the aspect ratio using an external grid generation package or the prism layer capability.

9.4 Creating a Non-Conformal Interface

For meshes containing both hexahedral and tetrahedral elements, you can generate a non-conformal interface to avoid creating intermediate pyramids as transition elements between the quadrilateral and triangular surfaces. You can also choose to create a non-conformal interface when growing prisms from a boundary on a surface mesh, to avoid quad faces in the domain to be meshed. TGrid will copy the surfaces containing quad elements and triangulate them while keeping the original surfaces intact. The free nodes of the triangulated surface will then be merged with the nodes on the original surface mesh. Both surfaces will then be converted to interface type.

9.4.1 The Non Conformals Panel

The Non Conformals panel allows you to specify the method to be used for retriangulating the quad faces on the non-conformal zones.



Controls

Retriangulation Method contains a list of methods available for retriangulating the quadrilateral faces on the non-conformal zones.

- **Prism** remeshes all quad zones named prism-side-n.
- **Quad Split** splits the quad faces diagonally into tri faces.
- **Remesh** remeshes all the zones comprising quad faces based on the edge and surface feature angle specified.

Boundary Zones contains a list of the boundary zones.

Boundary Zone Groups contains a list of the available boundary zone groups, including the user-defined groups.

Edge Feature Angle specifies the edge feature angle for remeshing when the Remesh method is selected.

Surface Feature Angle specifies the surface feature angle for remeshing when the Remesh method is selected.

Create creates the non-conformal interface according to the specified parameters.

Apply applies the specified parameters to the zones selected.

Draw displays the zones selected in the Boundary Zones selection list in the graphics window.

9.4.2 Text Interface for Creating a Non-Conformal Interface

The text commands for creating a non-conformal interface are as follows:

`/mesh/non-conformals/controls/enable?` toggles the creation of a non-conformal interface.

`/mesh/non-conformals/controls/retri-method` specifies the method to be used for retriangulating the quad faces on the non-conformal zones.

- `prism` remeshes all the quad zones named prism-side-n.
- `quad-split` splits the quad faces diagonally into tri faces.
- `remesh` remeshes all the quad faces based on the edge and surface feature angle specified.

`/mesh/non-conformals/create` creates the non-conformal interface on the specified face zone(s) using the specified retriangulation method.

9.5 Creating a Heat Exchanger Zone

Many engineering systems, including power plants, climate control, and engine cooling systems typically contain heat exchangers. However, for most engineering problems, it is impractical to model individual fins and tubes of the heat exchanger core.

TGrid allows you to create a heat exchanger volume mesh using the Mesh/Create/Heat Exchanger... menu item.

The heat exchanger mesh in TGrid comprises prisms generated from a quad split surface mesh. You need to specify four points (either by selecting the locations or nodes) and the required intervals between the first selected point and the each of the remaining points to create the heat exchanger mesh. TGrid creates a meshed plane using the first three specified points and the corresponding intervals. Prisms are created on the meshed plane using the fourth point and the corresponding interval.

> ⚠ The order of selection of the points is important since the heat exchanger zone is created based on the intervals specified between the first selected point and each of the remaining points. If the points are not specified in the correct order, you will get a heat exchanger zone that is different from the required one.

Alternatively, you can use the TUI command `/mesh/create-neat-exchanger` to create the heat exchanger mesh. You need to specify the method for selecting the points (by location or by nodes), the points, the intervals, and the zone name prefix, as required.

You can preview the heat exchanger zones and modify the parameters if you are dissatisfied with the results. TGrid creates the heat exchanger zones (prefixed by hxc-) as shown in Figure 9.5.1. You can specify the prefix for the zones as required in the Zone Prefix panel (see Section 7.14.3: The Zone Prefix Panel).

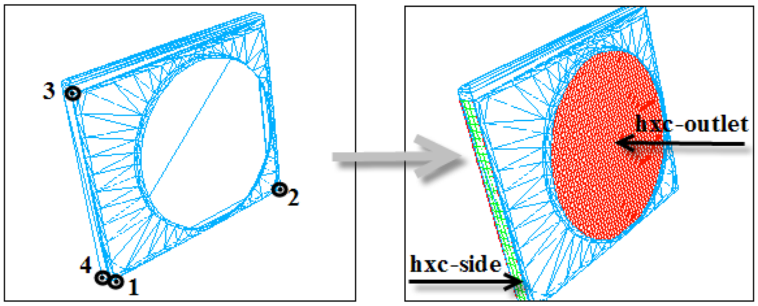
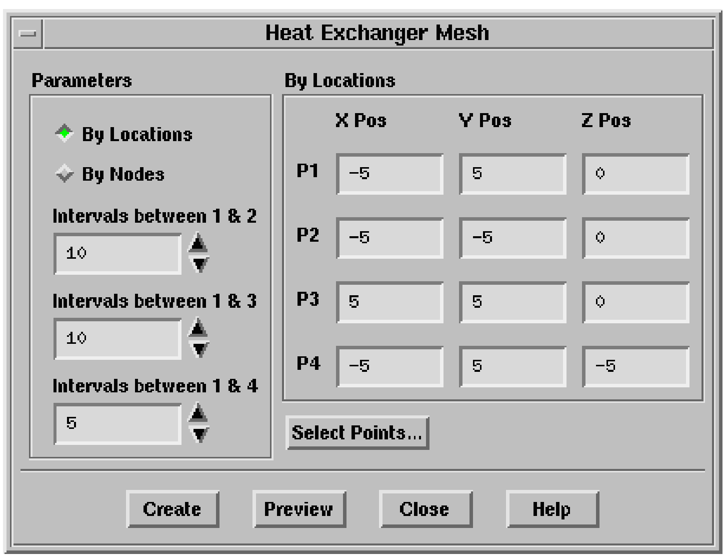


Figure 9.5.1: Creating the Heat Exchanger Mesh

9.5.1 The Heat Exchanger Mesh Panel

The Heat Exchanger Mesh panel allows you to specify the parameters required to create the heat exchanger mesh.

Mesh ⟶ Create ⟶ Heat Exchanger...



Controls

Parameters contains inputs to be specified for heat exchanger mesh creation.

By Locations allows you to specify the locations (X Pos, Y Pos, Z Pos) of the four points P1, P2, P3, and P4. You can either enter the position values or select the points using the mouse.

By Nodes allows you to specify the nodes to be used. You can either enter the **Node ID** or select the points using the mouse.

Intervals between 1 & 2 specifies the number of intervals between points 1 and 2.

Intervals between 1 & 3 specifies the number of intervals between points 1 and 3.

Intervals between 1 & 4 specifies the number of intervals between points 1 and 4.

Select Points... allows you to select the points (by selecting the locations or nodes) using the mouse. When you click this button, a **Working** dialog box will open, prompting you to select 3 points to define the plane, and the 4th point to grow prisms.

Create opens the **Zone Prefix** panel (see Section 7.14.3: The Zone Prefix Panel) where you can specify the prefix for the heat exchanger zones to be created.

Preview allows you to preview the heat exchanger to be created.