
Chapter 11. Generating Triangular/Tetrahedral Meshes

This chapter describes the procedure to create the triangular/tetrahedral mesh in the domain. It also describes some of the common problems faced during tri/tet meshing. It contains the following sections:

- [Section 11.2: Manually Creating a Tri or Tet Mesh](#)
- [Section 11.3: Initializing the Tri/Tet Mesh](#)
- [Section 11.4: Refining the Tri/Tet Mesh](#)
- [Section 11.5: The Tri/Tet Panel](#)
- [Section 11.6: Additional Text Commands for Tri/Tet Mesh Generation](#)
- [Section 11.7: Common Tri/Tet Meshing Problems](#)

You can use one of the following techniques to generate triangular or tetrahedral meshes:

- [Automatic mesh generation \(Section 11.1: Automatically Creating a Tri or Tet Mesh\)](#)
- [Manual mesh generation \(Section 11.2: Manually Creating a Tri or Tet Mesh\)](#)
- [A combination of manual and automatic commands \(Sections 11.3, 11.4\)](#)

If the cells (e.g., prisms or pyramids) have already been created in some portion of the computational domain, you need to create a domain encompassing the region to be meshed with triangular or tetrahedral cells before starting with the tri/tet mesh generation. See [Section 13.10: Using Domains to Group and Mesh Boundary Faces](#) for details. In such cases, all automatic and/or manual meshing actions will apply only to the active domain.

Alternatively, instead of creating a new domain, you can specify the cell zone(s) to be preserved while generating the tet mesh, before proceeding with the mesh generation. You can use the command `/mesh/tritet/preserve-cell-zone` to specify the cell zone(s) to be preserved.

11.1 Automatically Creating a Tri or Tet Mesh

The Auto Mesh submenu in the Mesh menu contains commands that encapsulate the recommended volume mesh generation strategy. The starting point for this procedure is a valid surface mesh.

11.1.1 Automatic Meshing Procedure for Tri/Tet Meshes

The automatic tri/tet mesh generation process is divided into two fundamental tasks: initialization and refinement. The automatic initialization procedure comprises the following steps:

1. Merging free nodes.
2. Deleting unused nodes.
3. Improving the surface mesh.
4. Initializing the mesh.
5. Generating and separating cell regions.



If the mesh fails to initialize, typically this indicates a problem with the surface mesh. In some rare cases, the sliver size (which is automatically computed) may need to be changed. This may happen for domains in which the minimum boundary face size is very small as compared to the domain extent. When changing the sliver size, the rule of thumb is that the specified value should be of the order $1e-12$ times the minimum face area.

The automatic refinement procedure comprises the following steps:

1. Sorting the boundary faces by size (3D only).
2. Refining the boundary cells.
3. Reverse sorting cells by skewness.
4. Refining the active cell zone(s).
5. Swapping cells based on skewness.
6. Reverse sorting cells by skewness.
7. Smoothing the mesh.
8. Removing boundary slivers.

You can select either the advancing front refinement, or the skewness-based refinement of the tetrahedral mesh. The refinement procedure is repeated a number of times.

You can control the number of repetitions for skewness-based refinement by setting the **Number of Levels** in the **Tri/Tet Refine Controls** panel (see Section 11.5.2: [The Tri/Tet Refine Controls Panel](#)). For each subsequent level of refinement, the cell skewness thresholds are lowered. Additional refinement levels increase the grid resolution (and the number of cells) and decrease the average cell skewness.

Each refinement level actually consists of two sweeps through the refinement procedure:

- One sweep at the appropriate skewness threshold for that particular level of refinement.
- Another sweep with a high skewness threshold and a relaxed **Min Boundary Closeness** set in the **Tri/Tet Refine Controls** panel (see Section 11.5.2: [The Tri/Tet Refine Controls Panel](#)).

This sweep attempts to reduce highly skewed cells in confined regions of the geometry.

Boundary slivers will be removed after the refinement sweeps are complete if the **Remove Slivers** option is enabled in the **Refinement** tab of the **Tri/Tet** panel.

In some rare cases, it may happen that the first sweep of the first refinement level will never finish because more skewed cells are formed due to refinement, which will be further refined to create more skewed cells, and so on. Such a problem may occur when there is a problem with the boundary mesh, such as rapid mesh transition in small gaps, etc. This problem can be avoided by using the incremental improvement option for the refinement parameters or by increasing the **Min Node Closeness** value in the **Tri/Tet Refine Controls** panel (see Section 11.5.2: [The Tri/Tet Refine Controls Panel](#)).

You can select the **fast transition** option for the refinement parameters in order to generate a mesh with a smaller number of cells. When this option is selected, TGrid will use the appropriate refinement parameters to generate fewer cells during meshing. The rate of change of the cell size is increased in order to reduce the number of cells generated. Alternatively, you can specify an appropriate cell size function with a sufficiently large value for the growth rate.



Various steps in the automatic meshing process can be added or eliminated using the **Tri/Tet** panel (see Section 11.5: [The Tri/Tet Panel](#)). In addition, most aspects of the automatic meshing process can be changed dynamically through the **Scheme** interface. Contact your Fluent support engineer for details.

11.1.2 Using the Auto Mesh Tool

 → Auto Mesh...

The Auto Mesh tool can be used to automatically do either of the following functions:

- Generate a triangular or tetrahedral volume mesh starting from a boundary mesh.
- Generate a triangular or tetrahedral volume mesh in an unmeshed domain of a mesh that contains other cell shapes.

The Auto Mesh tool allows you generate the tri/tetvolume mesh automatically. When you select Tri/Tet in the Volume Fill list and click the Mesh button in the Auto Mesh panel:

- If the volume mesh does not exist, TGrid will perform all the steps listed in Section 11.1.1: [Automatic Meshing Procedure for Tri/Tet Meshes](#).
- If a volume mesh already exists, a Question dialog box will appear, asking if you want to clear the existing mesh. If you click Yes, TGrid will clear the volume mesh and perform the active steps in the automatic mesh generation process. If you click No, the operation will be canceled.



You can use the command `/mesh/tritet/preserve-cell-zone` to specify the cell zone(s) to be preserved during mesh generation and click the Mesh button to proceed with the automatic mesh generation.

11.1.3 Automatic Meshing of Multiple Cell Zones

The Auto Mesh tool allows you to mesh multiple cell zones automatically. This is useful when the mesh has multiple cell zones (e.g., the problem requires a fluid zone and one or more solid zones). You can refine the mesh using refinement parameters specific to individual zones, if required. The procedure for meshing multiple cell zones comprises the following steps:

1. Initialize the mesh.
2. Activate the appropriate zones using the Manage Cell Zones panel (see Section 13.9.3: [The Manage Cell Zones Panel](#)).
3. Set the zone-specific refinement parameters and refine the active zones.

If you want to mesh *all* the zones included in the mesh with the same refinement parameters, modify the Non-Fluid Type in the Tri/Tet panel (see Section 11.5: [The Tri/Tet Panel](#)).



By default, the **Non-Fluid Type** is set to **dead** as TGrid considers the grid to comprise a single fluid region and one or more dead regions. The active zone is considered to be the fluid zone and only this fluid zone will be considered for refinement during the automatic meshing process.

When the **Non-Fluid Type** is set to a type other than **dead** (e.g., **solid**), all the zones will be active after the initialization is complete. Hence, all the zones will be considered for refinement. If required, you can change the zone type using the **Manage Cell Zones** panel (see Section 13.9.3: [The Manage Cell Zones Panel](#)).

Select the appropriate type from the **Non-Fluid Type** drop-down list in the **Tri Tet Zones** group box in the **Tri/Tet** panel before initializing the mesh.

11.1.4 Automatic Meshing for Hybrid Meshes

The **Auto Mesh** tool in TGrid can also create prisms and pyramids automatically, allowing automatic generation of hybrid meshes.

- If a mix of surface mesh types (quadrilateral and triangular) is present in the domain, TGrid will create pyramid cell zones on the quadrilateral boundary faces before meshing the tetrahedral domain.
- If prism growth parameters have been attached to a boundary zone (using the **Prisms** panel (see Section 10.10: [The Prisms Panel](#))), TGrid will automatically extrude the prism layers before meshing the tetrahedral domain.

When pyramids or prisms are generated using the automatic mesh generation feature, the intermediate boundary zones will be merged automatically after the tetrahedral mesh generation is complete. This enhancement avoids the creation of additional boundary zones such as **prism-side** and **pyramid-cap** (Sections 10.4 and 9.3.2).

- TGrid also allows you to transition from quad to tri faces using the **Non Conformals** option. This option is useful when you want to avoid pyramids on quad faces when growing prisms from a boundary. For mix surface mesh types (quadrilateral and triangular), you can select **Non Conformal** in the **Quad Tet Transition** list in the **Auto Mesh** panel. TGrid will copy the surfaces containing quad cells and triangulate them keeping the original faces intact. The free nodes of the triangulated surface will then be merged with the nodes of the original surface mesh and both the surfaces will be converted to interface type.

11.1.5 Further Mesh Improvements

Examine the following after generating the mesh using the automatic mesh generation process:

- The cell distribution (see Section 15.4: Reporting Boundary Cell Limits)
- The cell limit reports (see Sections 15.3 and 15.4)
- The mesh size (see Section 15.1: Reporting the Mesh Size)

For 2D problems, it is possible to reduce the maximum skewness to 0.4. In 3D, it is typically possible to reduce the maximum skewness to the range 0.8–0.9. Skewness values higher than 0.9 are typically obtained due to constraints imposed by the surface meshes. If you still have highly skewed cells apply additional swapping and smoothing to improve the quality (see Sections 13.1 and 13.2). To increase the density of the grid locally, use refinement regions (see Section 11.4.1: Using Local Refinement Regions).



Do not over-refine the grid in TGrid. The volume mesh produced in TGrid should be of sufficient density to resolve the shape of the geometry. The mesh can be improved more effectively using the solution-adaptive mesh capability provided in FLUENT.

11.2 Manually Creating a Tri or Tet Mesh

In addition to the Auto Mesh tool, TGrid allows you to control the tri/tet mesh generation process by modifying parameters at each step. The basic operations are described here, and the methods for modifying the associated parameters are described in Sections 11.3 and 11.4.

11.2.1 Manual Meshing Procedure for Tri/Tet Meshes

The basic components of the manual meshing process are examining and repairing the surface mesh, creating an initial mesh, refining the mesh, and improving the mesh.

The following steps describe the recommended meshing procedure.

Step 1: Examining the Surface Mesh

The first step in the mesh generation process is to examine the validity and quality of the surface mesh. The recommended approach includes the following steps:

- Checking for free and isolated nodes (see Section 7.1.1: Free and Isolated Nodes).

- Checking and improving the surface mesh quality (see Section 7.4: [Improving Boundary Surfaces](#)).
- Visually examining the surface mesh for free, multiply-connected, skewed, and/or close-proximity faces (see Section 14.1: [Displaying the Grid](#)).
- Making local repairs if required (see Section 7.3: [Modifying the Boundary Mesh](#)).

After obtaining a valid, (ideally) high-quality surface mesh, you can proceed to create the volume mesh.

Step 2: Creating the Initial Mesh

The first step in generating the volume grid is creating the initial mesh. This process first creates a pre-meshed box encompassing the entire geometry, and then sequentially introduces each boundary node into the mesh. As the nodes of a boundary face are inserted into the mesh, any necessary mesh modifications needed to insert the face are performed, effectively inserting both boundary nodes and faces into the mesh simultaneously. For more information see Section 11.3: [Initializing the Tri/Tet Mesh](#).

Activating Multiple Zones (for Multi-zone Grids Only)

If the mesh has multiple regions (e.g., the problem requires a fluid zone and one or more solid zones), you can refine the mesh using refinement parameters specific to individual zones, if required. The procedure for meshing multiple cell zones comprises the following steps:

1. Initialize the mesh.
2. Activate the appropriate zones using the **Manage Cell Zones** panel.
3. Set the zone-specific refinement parameters and refine the active zones.

If you want to mesh *all* the zones included in the mesh with the same refinement parameters, modify the **Non-Fluid Type** in the **Tri/Tet** panel (see Section 11.5: [The Tri/Tet Panel](#)).

Select the appropriate type from the **Non-Fluid Type** drop-down list in the **Tri Tet Zones** group box in the **Tri/Tet** panel.



By default, the **Non-Fluid Type** is set to **dead** as TGrid considers the grid to comprise a single fluid region and one or more dead regions. The active zone is considered to be the fluid zone and only this fluid zone will be considered for refinement during the automatic meshing process.

When the **Non-Fluid Type** is set to a type other than **dead** (e.g., **solid**), all the zones will be active after the initialization is complete. Hence, all the zones will be considered for refinement. If required, you can change the zone type using the **Manage Cell Zones** panel (see Section 13.9.3: [The Manage Cell Zones Panel](#)).

Step 3: Refining the Mesh

You will refine the initial mesh by adding cells at the boundary and in the interior. The refinement methods available are the skewness-based method and the advancing front method. You can also define local refinement regions using the **Tri/Tet Refinement Region** panel. You can modify the parameters in the **Refinement** tab of the **Tri/Tet** panel and select additional refinement options for skewness-based refinement in the **Tri/Tet Refine Controls** panel if, required.

Step 4: Improving the Mesh

You can improve the skewness by smoothing, swapping, and refining the mesh further to improve the quality of the volume mesh. At any point in the mesh generation process, you can compute and/or plot the cell skewness distribution using the **Cell Distribution** panel (see Section 14.3.1: [The Cell Distribution Panel](#)). This will give you an idea of the improvement required. For 2D problems, it is usually possible to reduce the maximum skewness to 0.4. In 3D, however, it is typically possible to reduce it to 0.8–0.85 (for simple geometries) or 0.9–0.95 (for more complex geometries).

Step 4a: Swapping and Smoothing without Refining

Swapping and smoothing improve the mesh by manipulating the nodes and faces without increasing the total number of cells. Refinement, on the other hand, improves the mesh by adding nodes, which typically increases the number of cells. To get the best possible mesh with the minimum number of cells, perform only smoothing and swapping before refining the mesh any further.

Smoothing repositions interior nodes to lower the maximum skewness of the grid. For swapping, given $n + 2$ nodes in dimension n , there are at most two triangulations of the nodes depending on the configuration of the nodes. In cases where two triangulations exist, TGrid examines the two alternatives, selecting the one that has the lowest maximum skewness. TGrid will automatically perform smoothing and swapping to improve the mesh when the **Improve Mesh** option is enabled in the **Refinement** tab of the **Tri/Tet** panel.

Refer to Sections 13.1 and 13.2 for details on improving the mesh by swapping and smoothing.

Step 4b: Further Refinement

You can further refine the cells in the active zones by changing parameters such as Max Cell Volume, and Max Cell Skew and Max Boundary Cell Skew (available only for skewness-based refinement).

i Changing mesh size controls other than Max Cell Volume will have no effect on a mesh that has already been refined. Alternately, use local refinement regions. Refer to Section 11.4.1: [Using Local Refinement Regions](#) for details.

Refining with skewness parameters reduced to values less than 0.5 will rarely improve a 3D mesh and will be very time-consuming. For a 2D mesh, you may try reducing these parameters further to see if any improvement results.

i Do not over-refine the grid in TGrid. The volume mesh produced in TGrid should be of sufficient density to resolve the shape and any intuitive flow features (e.g., wall boundary layers). Additional resolution can be more effectively produced using the solution-adaptive mesh capability provided in FLUENT.

At this step you will have an acceptable grid for most geometries.

Step 4c: Boundary Slivers and Other Sources of Lingering High Skewness

When viewing the cell distribution plot, if you find that there are a few cells with very high skewness, check to see where they are located (i.e., on the boundary or in the interior). To do so, use the Report Boundary Cell Limits panel (see Section 15.4.1: [The Report Boundary Cell Limits Panel](#)).

Refer to Sections 13.4 and 13.7 for details on removing slivers and other mesh improvement options.

You can initialize and refine the tri/tet mesh in a single step by clicking the Init&Refine button in the Tri/Tet panel after setting the appropriate parameters in the Initialization and Refinement tabs. Alternatively, you can set the appropriate parameters in the Initialization tab and click the Init button to create the initial mesh. You can then refine the initial mesh by setting the appropriate parameters in the Refinement tab and clicking the Refine button.

If you need to refine a particular region, you can define the region to be refined using the Tri/Tet Refinement Region panel (see Section 11.5.3: [The Tri/Tet Refinement Region Panel](#)).

11.3 Initializing the Tri/Tet Mesh

The first step in the volume mesh generation process is initialization. The initial mesh consists of the nodes and triangles of the boundary surface mesh. For some 3D geometries, it is not possible to create a tetrahedral mesh from the boundary nodes alone. In such cases, TGrid automatically adds a small number of nodes in the interior of the domain. Interior nodes may also have to be added to resolve numerical problems associated with the Delaunay criterion.

11.3.1 Using the Tri/Tet Panel

You can initialize the mesh using the options available in the **Initialization** tab of the **Tri/Tet** panel.

Mesh → Tri/Tet...

1. Select the appropriate actions to be performed during initialization from the **Options** group box.

The **Merge Free Nodes** and **Delete Unused Nodes** options are enabled by default. You can also include improving the surface mesh in the mesh initialization process by enabling the **Improve Surface Mesh** option.

2. Select the appropriate option from the **Non-Fluid Type** drop-down list. Enable **Delete Dead Zones**, if required.

When the initial mesh is generated, all the cells are grouped into contiguous zones separated by boundaries. TGrid considers the mesh to comprise a single fluid zone and one or more dead regions. The zone just inside the outer boundary is set to be active and is labeled a fluid zone. All other non-fluid zones will be inactive. Only active zones will be considered for refinement during the mesh generation process.

You can refine different groups of zones using different refinement parameters for each group by toggling the zones between active and inactive. If however, you need to use the same refinement parameters for all the zones, you can change the specification of **Non-Fluid Type** to a type other than **dead** (e.g., **solid**). When the **Non-Fluid Type** is set to a type other than **dead**, all the zones will be activated after initialization. Hence, you can set the appropriate refinement parameters without setting all the zones to be active.

3. Specify additional initialization parameters, if required. These parameters are available in the **Tri/Tet Init Controls** panel. Click the **Controls...** button to open the **Tri/Tet Init Controls** panel (see Section 11.5.1: [The Tri/Tet Init Controls Panel](#)).
4. Click **Init** to initialize the mesh.

A **Working** dialog box will appear, informing you that the initialization is in progress. Click the **Cancel** button in the **Working** dialog box to abort the mesh initialization process. Canceling the initialization will leave the mesh incomplete.

- If you try to initialize the mesh after canceling an initial attempt, TGrid will ask you (in a **Question** dialog box) if it is OK to clear the incomplete mesh. After you approve, TGrid will begin the initialization process again.
- If you try to initialize the mesh when duplicate nodes exist, the initialization will fail. You must clear the mesh and merge the duplicate nodes before attempting the initialization again.
- If you try to initialize the mesh when a volume mesh already exists, a **Question** dialog box will appear, asking if you want to clear the existing mesh. If you click **Yes**, TGrid will clear the volume mesh and proceed to initialize the mesh. If you click **No**, the operation will be canceled.
- If you need to preserve the existing mesh during the meshing process, use the command `/mesh/tritet/preserve-cell-zone` to specify the cell zone(s) to be preserved and click the **Init** button to proceed with the initialization.
- If the initial mesh generation fails, check the validity of the surface mesh. In some rare cases, you may need to change the sliver size parameter (see Section 11.5.1: [The Tri/Tet Init Controls Panel](#)).

Refer to Section 11.7: [Common Tri/Tet Meshing Problems](#) for more information on meshing problems.

11.3.2 Text Commands for Initializing the Mesh

The text commands for initializing the mesh have the same functionality as the initialization parameters available in the **Initialization** tab of the **Tri/Tet** panel.

`/mesh/tritet/init` meshes boundary nodes, generating the initial Delaunay mesh.

`/mesh/tritet/controls/delete-dead-zones?` toggles the automatic deleting of dead zones during mesh initialization.

`/mesh/tritet/controls/delete-unused-nodes?` toggles the deleting of unused nodes during mesh initialization.

`/mesh/tritet/controls/improve-surface-mesh?` toggles the improving of the surface mesh during mesh initialization.

`/mesh/tritet/controls/merge-free-nodes?` toggles the merging of free nodes during mesh initialization.

`/mesh/tritet/controls/non-fluid-type` selects the non-fluid cell zone type.

`/mesh/tritet/controls/advanced/circumsphere-tolerance` defines the thickness of the circumsphere boundary.

In a Delaunay meshing algorithm, the most basic operation is checking to see if a node is inside or outside of the circumsphere of a cell. Due to numerical errors, it is not possible to get a definitive answer. The tolerance is used to define a thickness for the boundary of the circumsphere. If a node is on this boundary, it is neither inside nor outside of the circumsphere and hence will be treated specially.

`/mesh/tritet/controls/advanced/defaults?` toggles the use of the default values for the initialization parameters.

`/mesh/tritet/controls/advanced/node-tolerance` defines the smallest distance between two distinct nodes. This command is available only when you disable the use of the default values for the initialization parameters.

`/mesh/tritet/controls/advanced/sliver-size` is the smallest cell whose size can be determined accurately. This command is available only when you disable the use of the default values for the initialization parameters.

11.4 Refining the Tri/Tet Mesh

After initializing the mesh, you can improve the quality and density of the grid using global and local refinement. Global refinement allows you to refine all cells in the active zones, while local refinement allows you to refine cells within a specified region.

In most applications, you will use only global refinement to create an acceptable discretization of the volume. Local refinement is used to modify the grading away from boundaries or increase the resolution of an interior region of the grid. You need to define and activate the regions to be refined before proceeding.

The refinement process comprises a series of sweeps through a sequence of sorting, refining, reverse sorting, and swapping and smoothing of the cells. You can specify the number of refinement levels to be performed for skewness-based refinement. Each refinement level consists of two iterations through the refinement procedure. For each subsequent level of refinement, the cell skewness thresholds are lowered. Additional refinement levels will increase the grid resolution (and the number of cells) and decrease the average cell skewness. The preset refinement control parameters are available in the **Tri/Tet Refine Controls** panel (see Section 11.5.2: [The Tri/Tet Refine Controls Panel](#)). Alternatively, you can select the advancing front refinement method.

Specify whether the mesh has to be improved or if the slivers have to be removed during refinement. The **Improve Mesh** option performs additional smoothing and swapping with lowered skewness thresholds, attempting to improve the average skewness of the mesh. The **Remove Slivers** option attempts to lower the maximum skewness by improving highly skewed (sliver) cells.

The boundary mesh is fixed during the mesh improvement and sliver removal operations.

Refinement Controls

The refinement controls for the skewness method are available in the Tri/Tet Refine Controls panel (see Section 11.5.2: [The Tri/Tet Refine Controls Panel](#)). Click the Controls... button in the Refinement tab of the Tri/Tet panel to open the Tri/Tet Refine Controls panel.

The refinement controls for the advancing front method are available in the `/mesh/tritet/controls/adv-front-method` menu.

The `/mesh/tritet/controls/adv-front-method/skew-improve/target?` command is important as it allows you to enable targeted skewness-based refinement for the advancing front method. The refinement process will attempt to reach the targeted skewness (specified by the command `/mesh/tritet/controls/adv-front-method/skew-improve/target-skewness`). Though the default values work well in most situations, it may be advantageous to increase the `target-skewness` value in some cases (e.g., cases where a combination of a poor surface mesh and narrow gaps will result in poor cells in the mesh) to increase the meshing speed. Refer to Section 11.4.3: [Text Commands for Setting Refinement Controls](#) for the list of commands available.

11.4.1 Using Local Refinement Regions

A refinement region limits refinement to specific cells inside the domain. When you use the Refine option in the Tri/Tet Refinement Region panel, TGrid refines the cells within the activated region(s). The primary use of refinement regions is to reduce the cell size in the region to less than it would be normally. Currently, the only possible region shape is a box, which can be oriented as required.

The region is defined by an x , y , and z range and its orientation about the x , y , and z axes. Any cell within the region or which intersects the region will be subjected to the refinement criteria. You can control the concentration of interior nodes by specifying the maximum cell volume in the refinement region. It is possible to save different values for the Maximum Cell Volume when you save each region. You can also specify an outer region with the required volume growth to have a smooth transition between the original and the refined cells.

Note: *In addition to the use of refinement regions for refinement during the mesh generation process, they can also be used as a postprocessing tool to refine a region of an existing mesh.*

You can create multiple regions that overlap each other and the geometry. To create additional regions, you can copy an existing region and then modify the parameters as required. The default region includes the entire geometry.

Figure 11.4.1 shows the refinement region and outer region. The maximum cell size for the refinement and outer regions is displayed based on the maximum cell volume and

outside volume growth specified. For information on refining triangular boundary faces in anticipation of local refinement, refer to Section 7.5: Refining the Boundary Mesh.

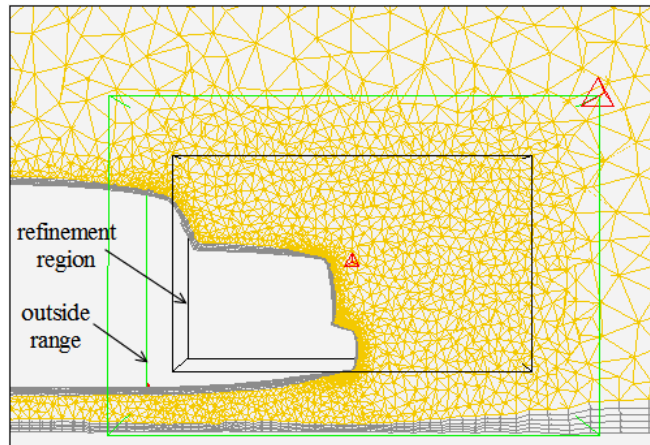


Figure 11.4.1: Local Refinement Region for the Tetrahedral Mesh

11.4.2 Using the Tri/Tet Panel

You can refine the mesh using the options available in the **Refinement** tab of the **Tri/Tet** panel.

→ Tri/Tet...

1. Select the appropriate refinement method from the **Refine Method** drop-down list.
2. Enable the appropriate options in the **Options** group box.
3. Specify an appropriate value for **Max Cell Volume**.
4. Select the appropriate option from the **Cell Size Function** drop-down list.

Ensure that none of the cells in the interior are larger than required by selecting the appropriate option in the **Cell Size Function** list. the following options are available:

none specifies that the cell size is determined based on the **Max Cell Volume** and skewness. This option is available only for the skewness-based refinement method.

linear specifies that the cell size in the interior is linearly interpolated from the size of the boundary triangles.

geometric specifies that the cell size in the interior of the domain is obtained by a geometric growth from the closest boundary according to the growth rate specified.

5. Modify the refinement control parameters, if required.
 - For the skewness method, click the **Controls...** button to open the **Tri/Tet Refine Controls** panel (see Section 11.5.2: [The Tri/Tet Refine Controls Panel](#)). Select the appropriate option from the **Preset Parameters** drop-down list. Verify the parameter values in the **Tri/Tet Refine Controls** panel and modify the values as appropriate.
 - For the advancing front method, use the commands available in the `/mesh/tritet/controls/adv-front-method` menu. You can also specify the targeted skewness for the refinement process (see Section 11.4: [Refinement Controls](#) for details).
6. To refine specific regions of the mesh, click the **Local Regions...** button to open the **Tri/Tet Refinement Region** panel (see Section 11.5.3: [The Tri/Tet Refinement Region Panel](#)). Define the refinement region(s) and activate them. The number of activated regions will be reported in the **Message** field in the **Refinement** tab of the **Tri/Tet** panel.
7. Click **Refine** to refine the mesh.

11.4.3 Text Commands for Setting Refinement Controls

The text interface commands for setting refinement parameters are as follows:

`/mesh/tritet/refine` refines the initialized mesh.

`/mesh/tritet/controls/adv-front-method/first-improve-params` defines the refining front improvement parameters for the advancing front method.

`/mesh/tritet/controls/adv-front-method/second-improve-params` defines the cell zone improvement parameters for the advancing front method.

`/mesh/tritet/controls/adv-front-method/refine-parameters` defines the refinement parameters for the advancing front method.

`/mesh/tritet/controls/adv-front-method/skew-improve/attempts` specifies the number of overall improvement attempts for the advancing front method.

`/mesh/tritet/controls/adv-front-method/skew-improve/boundary-sliver-skew` specifies the boundary sliver skewness for the advancing front method. This parameter is used for removing sliver cells along the boundary.

`/mesh/tritet/controls/adv-front-method/skew-improve/iterations` specifies the number of improvement iterations in each attempt for the advancing front method.

`/mesh/tritet/controls/adv-front-method/skew-improve/sliver-skew` specifies the sliver skewness for the advancing front method. This parameter is used for removing sliver cells in the interior.

`/mesh/tritet/controls/adv-front-method/skew-improve/target-skew` specifies the targeted skewness during improvement for the advancing front method.

`/mesh/tritet/controls/adv-front-method/skew-improve/target?` allows you to enable targeted skewness-based refinement for the advancing front method. This option allows you to improve the mesh until the targeted skewness value is achieved.

`/mesh/tritet/controls/advanced/max-cells` sets the maximum number of cells in the mesh.

`/mesh/tritet/controls/advanced/max-nodes` sets the maximum number of nodes in the mesh.



Be careful when you set upper limits on the number of cells or nodes. TGrid will stop adding nodes or cells when it reaches the prescribed limit, and may leave the mesh unevenly refined and less than adequate.

`/mesh/tritet/controls/cell-size-function` specifies the cell size function for refinement. You can select `geometric`, `linear`, or `none`, as appropriate.

`/mesh/tritet/controls/compute-max-cell-volume` computes the maximum cell volume for the current mesh.

`/mesh/tritet/controls/max-cell-volume` specifies the maximum allowable cell volume.

`/mesh/tritet/controls/refine-method` allows you to select the refinement method. You can select either skewness-based refinement or the advancing front method.

`/mesh/tritet/controls/skewness-method/levels` specifies the number of refinement levels for skewness-based refinement.

`/mesh/tritet/controls/skewness-method/max-skew-improve?` toggles the skewness-based improvement during refinement.

`/mesh/tritet/controls/skewness-method/must-improve-skewness?` toggles the modification of the default auto refinement parameters in order to improve skewness after refinement.

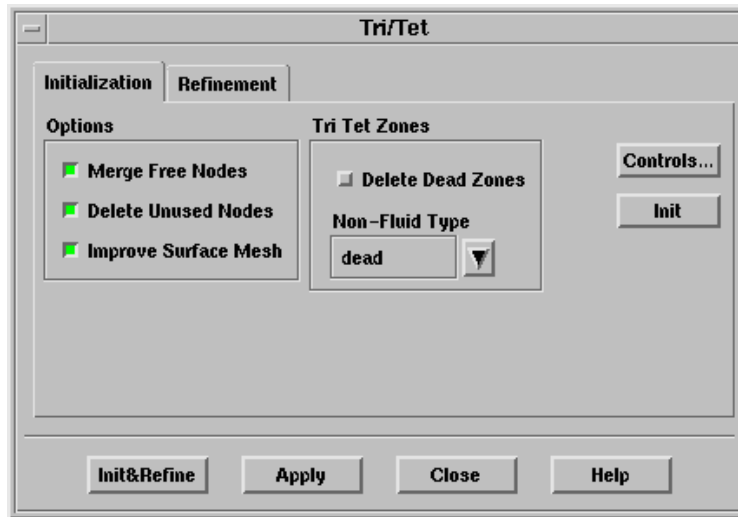
`/mesh/tritet/controls/skewness-method/refine-boundary-cells?` toggles the automatic refinement of boundary cells during refinement.

`/mesh/tritet/controls/skewness-method/refine-cells?` toggles the automatic refinement of cells during refinement.

- `/mesh/tritet/controls/skewness-method/smooth-mesh?` toggles the automatic smoothing of the mesh during refinement.
- `/mesh/tritet/controls/skewness-method/sort-boundary-faces?` toggles the automatic sorting of boundary faces by size during refinement.
- `/mesh/tritet/controls/skewness-method/sort-cells?` toggles the automatic reverse sorting of cells by skewness during refinement.
- `/mesh/tritet/controls/skewness-method/swap-faces?` toggles the automatic swapping of faces during refinement.
- `/mesh/tritet/controls/skewness-method/type` allows you to select the appropriate pre-defined skewness refinement parameters. You can select `default`, `fast-transition`, or `incremental-improve` as required.
- `/mesh/tritet/local-regions/activate` activates the specified region(s) for refinement.
- `/mesh/tritet/local-regions/deactivate` deactivate the specified region(s) for refinement.
- `/mesh/tritet/local-regions/define` defines the refinement region according to the specified parameters.
- `/mesh/tritet/local-regions/delete` deletes the specified refinement region.
- `/mesh/tritet/local-regions/ideal-vol` reports the volume of an ideal tetrahedron for the edge length specified.
- `/mesh/tritet/local-regions/init` defines the default refinement region encompassing the entire geometry.
- `/mesh/tritet/local-regions/list-all-regions` lists all refinement region parameters and the activated regions in the TGrid console.
- `/mesh/tritet/local-regions/refine` refines live cells inside the specified region based on refinement parameters.

11.5 The Tri/Tet Panel

The Tri/Tet panel allows you to set the parameters controlling tri/tet meshing.



Controls

Initialization contains options for initializing the mesh.

Options contains options to be included/excluded in the mesh initialization process.

Merge Free Nodes merges free nodes (see Section 7.1.1: [Free and Isolated Nodes](#)).

Delete Unused Nodes deletes nodes which are not referenced by any faces (see Section 7.1.1: [Free and Isolated Nodes](#)).

Improve Surface Mesh performs edge swapping based on the Delaunay criterion to improve the quality of the surface mesh.

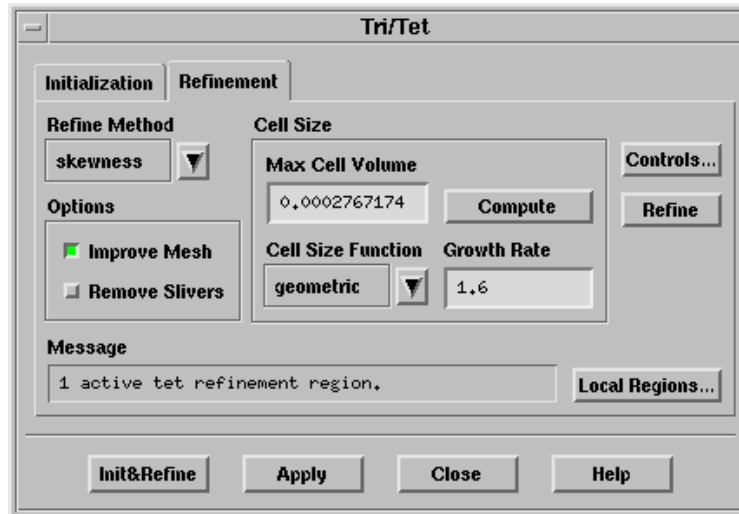
Tri Tet Zones contains parameters for tri/tet cell zone initialization.

Delete Dead Zones toggles the automatic deleting of dead zones.

Non-Fluid Type allows you to select the non-fluid cell type. TGrid will set the non-fluid zones to the selected type after initializing the mesh.

Controls... opens the Tri/Tet Init Controls panel (see Section 11.5.1: [The Tri/Tet Init Controls Panel](#)).

Init initializes the mesh.



Refinement contains options for refining the mesh.

Refine Method allows you to select the refinement method to be used.

Options contains options to be included/excluded in the refinement process.

Improve Mesh improves the mesh during refinement (see Sections 13.1 and 13.2).

Remove Slivers removes slivers (see Section 13.4: [Removing Slivers from a Tetrahedral Mesh](#)).

Cell Size contains parameters controlling the tri/tet cell size.

Max Cell Volume specifies the maximum acceptable volume for a cell. Any active cell with a size greater than the specified value will be refined.

Compute computes the maximum cell volume based on the longest edge in the boundary mesh.

Cell Size Function allows you to control the cell size distribution in the interior of the mesh.

geometric determines the cell size distribution by geometric growth from the closest boundary, according to the specified growth factor.

linear determines the cell size distribution by linear interpolation from the boundary cell size.

none determines the cell size distribution based on the **Max Cell Volume** and **skewness**. This option is available only for the skewness-based refinement.

Growth Rate specifies the growth factor for determining the cell size distribution when the **geometric** method is selected from the **Cell Size Function** drop-down list.

Controls... opens the Tri/Tet Refine Controls panel (see Section 11.5.2: [The Tri/Tet Refine Controls Panel](#)).

Refine refines the mesh based on the specified parameters and activated regions.

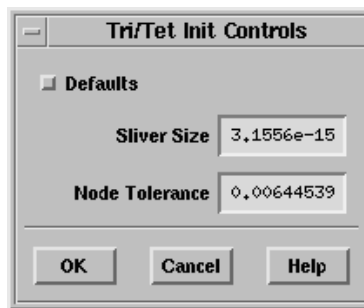
Local Region... opens the Tri/Tet Refinement Region panel (see Section 11.5.3: [The Tri/Tet Refinement Region Panel](#)).

Init&Refine initializes and refines the mesh.

Apply applies the parameters set in the Tri/Tet panel.

11.5.1 The Tri/Tet Init Controls Panel

The Tri/Tet Init Controls panel allows you to set additional parameters to control the initial mesh generation.



Controls

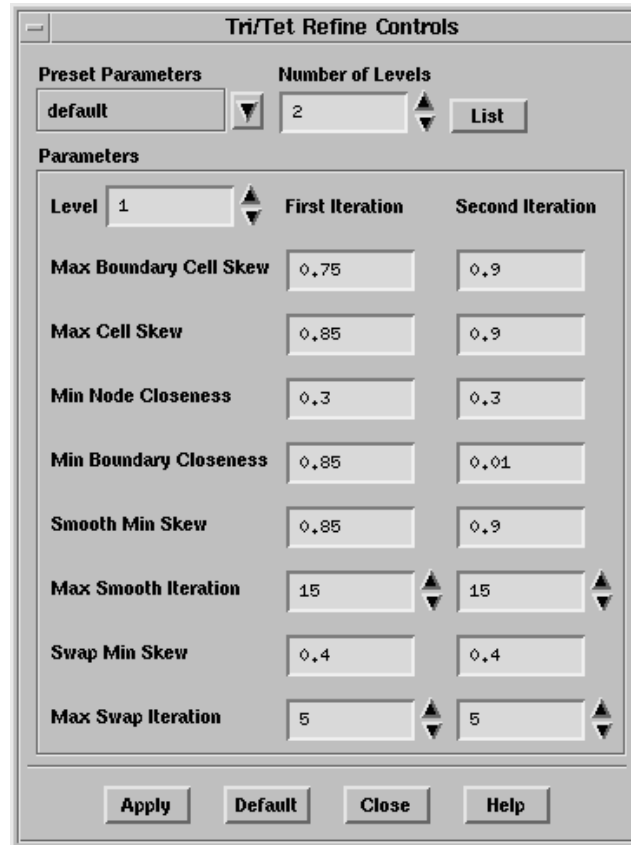
Defaults enables the use of the default parameter values for mesh initialization

Sliver Size is the smallest cell size which can be determined accurately. Cells having a calculated size less than the specified sliver size will be assigned a size of zero. This option is available only when the **Defaults** option is disabled.

Node Tolerance is the smallest distance between two distinct nodes. If the distance between two nodes is less than the specified node tolerance, the nodes are considered to be duplicates. This option is available only when the **Defaults** option is disabled.

11.5.2 The Tri/Tet Refine Controls Panel

The Tri/Tet Refine Controls panel allows you to set additional parameters to control the skewness-based refinement of the mesh.



Controls

Preset Parameters contains a drop-down list of the preset refinement control parameters available.

default specifies the default refinement parameters.

fast-transition specifies the refinement parameters which will result in quicker mesh gradation away from the boundaries and will generate fewer cells as compared to the **default** option.

incremental-improve specifies the refinement parameters which will result in incremental lowering of skewness.

Number of Levels specifies the number of refinement levels.

List displays the refinement parameters for the specified number of levels in the console.

Parameters contains input fields for the refinement parameters.

Level specifies the level of refinement for which refinement parameters are displayed.

Each refinement level consists of two sweeps through the refinement procedure.

Max Boundary Cell Skew specifies the maximum acceptable skewness for a boundary cell. All active boundary cells with skewness greater than this value will be refined

Max Cell Skew specifies the maximum acceptable skewness for a cell. All active cells with skewness greater than this value will be refined.

Min Node Closeness specifies the distance from an interior node (normalized by the local mesh size) within which it is appropriate to add a node. This restriction prevents a node from being added too close to another node.

Min Boundary Closeness specifies the distance from a boundary node (normalized by the local mesh size) within which it is appropriate to add a node. Do not add nodes too close to the boundary because it will be nearly impossible to remove a highly skewed cell, if created.

Smooth Min Skew specifies the minimum skewness required to perform smoothing.

Max Smooth Iteration specifies the maximum number of smoothing iterations.

Swap Min Skew specifies the minimum skewness required to perform swapping.

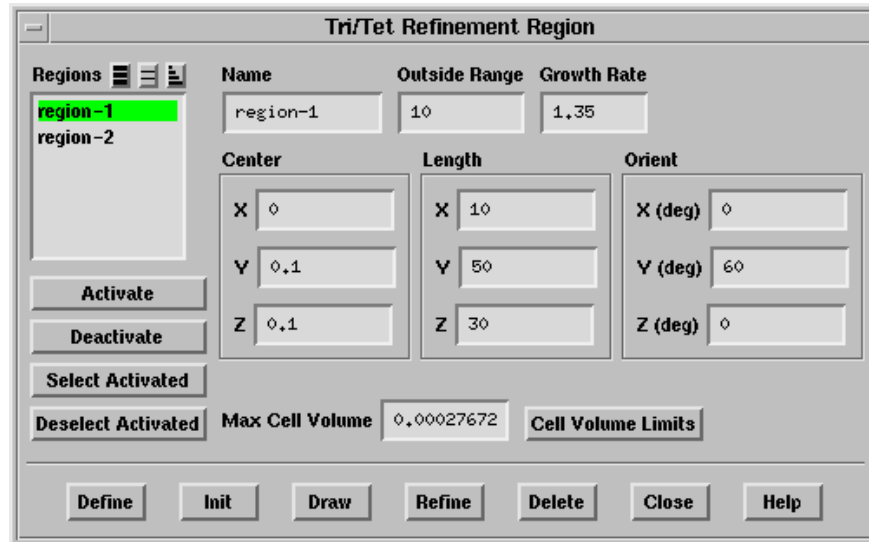
Max Swap Iteration specifies the maximum number of swapping iterations.

Apply applies the specified refinement parameters.

Default restores all parameters to their default values.

11.5.3 The Tri/Tet Refinement Region Panel

The Tri/Tet Refinement Region panel provides options for creating, modifying, and activating/deactivating refinement regions, for tri/tet mesh refinement.



Controls

Regions contains a list of the defined regions.

Activate activates the region selected in the **Regions** list.

Deactivate deactivates the region selected in the **Regions** list.

Select Activated selects the activated regions in the **Regions** list.

Deselect Activated deselects the activated regions in the **Regions** list.

Name reports the name of the selected region. You can specify a new name by entering it in this text entry box.

Outside Range defines the size of the outer transition region in which the geometric growth factor is applied to obtain a smooth transition between the original and the refined cells.

Growth Rate is the geometric growth factor applied in the outer transition region.

Center allows you to specify the coordinates of the center of the region you want to create. This is useful when you know the region in which refinement is required. Specify the coordinates of a point in that region as a center of the region to be created.

Length allows you to specify the absolute size of the new region in x, y, and z directions.

Orient allows you to specify the orientation of the new region.

Note: TGrid *orients the region by rotation first about the x-axis, then the y-axis, and finally the z-axis. You need to take this into account while specifying the orientation of the region as rotation in any other order will produce different results.*

Max Cell Volume sets the maximum cell volume for the selected region. You can change this value by entering a new value in this text entry box.

Cell Volume Limits reports (in an **Information** dialog box and the console) the number of cells in the selected region, and the minimum, maximum, and average size of these cells.

Define creates a new region according to the specified parameters. It also allows you to modify the selected region according to the specified changes.

Init creates a default region encompassing the entire geometry.

Draw draws the region(s) in the graphics window. The maximum cell size for the refinement and outer regions will also be displayed. If the boundary grid was displayed (using the **Display Grid** panel (see Section 14.1.3: [The Display Grid Panel](#))) before drawing the refinement region, the grid will also be included in the display.

Refine refines active cells inside the selected region, based on the specified refinement parameters.

Delete deletes the selected region.

11.6 Additional Text Commands for Tri/Tet Mesh Generation

`/mesh/tritet/init-refine` generates the triangular/tetrahedral mesh.

`/mesh/tritet/controls/advanced/freeze-boundary-cells` freezes the boundary cells during mesh improvement and sliver removal operations.

`/mesh/tritet/controls/advanced/keep-virtual-entities?` toggles the automatic deleting of virtual entities after mesh initialization.

`/mesh/tritet/controls/improve-mesh/improve?` toggles the automatic improvement of the mesh during the meshing process.

`/mesh/tritet/controls/improve-mesh/laplace-smooth` sets the parameters for Laplacian smoothing in order to improve mesh quality.

`/mesh/tritet/controls/improve-mesh/skewness-smooth` sets the parameters for skewness-based smoothing in order to improve mesh quality.

`/mesh/tritet/controls/improve-mesh/swap` sets the parameters for swapping in order to improve mesh quality.

`/mesh/tritet/controls/remove-slivers/angle` specifies the maximum dihedral angle for considering the cell to be a sliver

`/mesh/tritet/controls/remove-slivers/attempts` specifies the number of attempts overall to remove slivers.

`/mesh/tritet/controls/remove-slivers/iterations` specifies the number of iterations to be performed for the specific sliver removal operation.

`/mesh/tritet/controls/remove-slivers/remove?` toggles the automatic removal of slivers.

`/mesh/tritet/controls/remove-slivers/skew` specifies the skewness threshold for sliver removal.

`/mesh/tritet/delete-virtual-cells` deletes virtual cells created due to the use of the `keep-virtual-entities?` option.

`/mesh/tritet/preserve-cell-zone` allows you to specify the cell zones to be preserved during the meshing process.

11.7 Common Tri/Tet Meshing Problems

Most problems with the mesh generation process become manifest in the failure to generate an initial mesh. There are two sources of such problems:

- An invalid surface mesh
- Incorrect sliver size specified

Some of the common problems and suggestions for checking and fixing the mesh are described in this section.

To look at the nodes and faces that have not been meshed, enable `Unmeshed` in the `Display Grid` panel.

Duplicate Nodes or Faces

Some codes create multiple copies of the same node where two boundary curves or surfaces meet, resulting in a connectivity problem. This problem can be detected by reporting the number of free nodes or by drawing the free edges. If the geometry does not contain infinitely thin walls then there should not be any free nodes or edges. You can remove duplicate nodes using the `/boundary/delete-duplicate-nodes` command described in Section 7.15: [Additional Boundary Mesh Text Commands](#).

The presence of duplicate faces can be detected in one of two ways:

- If the nodes of the duplicate faces are not shared by the non-duplicate faces, the duplicate faces may have free edges that can be displayed.
- If the nodes are not distinct or have been merged, there will be multiply-connected edges.

Note: *Duplicate faces can be handled in TGrid and hence it is not necessary to remove them.*

Extra Nodes or Faces

Nodes that are not used by any face are called unused nodes. The unused nodes can easily be found and deleted using the operations described in Section 7.1.2: [The Merge Boundary Nodes Panel](#) if they are not required. However, there are cases where these additional nodes may be useful (see Section 9.1.7: [Inserting Isolated Nodes into a Tri or Tet Mesh](#)). Extra faces can be identified and removed using the method described for duplicate faces.

Intersecting Faces

If there are intersecting faces in the mesh, TGrid will not be able to generate a mesh until the problem faces are removed. Most intersecting faces can be located using the `/boundary/mark-face-intersection` command. Also, TGrid will report boundary face intersection if encountered during initialization.

After the meshing fails, the unmeshed faces can be drawn to see the problem area. Usually, since the intersecting face will have been meshed, it will not be drawn and some additional sleuthing will be required. Draw all the faces near the unmeshed faces by setting display bounds (see Section 14.1.3: [The Display Grid Panel](#)). Click the **Mark** button in the **Intersect Boundary Zones** panel (see Section 7.2.5: [The Intersect Boundary Zones Panel](#)) to highlight the face intersection. Alternatively, you can use the command `/boundary/mark-face-intersection` to highlight the face intersection.

Poor Boundary Node Distribution

Some meshing problems can be caused by a poor quality surface mesh. The surface mesh may have highly skewed faces, or two or more boundaries in close proximity may have very different face sizes. In the latter case, the boundaries can be connected (e.g., in a corner) or completely separate. Either way, the boundary mesh cannot be created without considering the impact of other nearby boundaries.

If the gap between two boundaries is D , the largest edge on a face should not exceed D . The mesh quality is extremely important when a coarse mesh is required in a small gap.

Non-Closed Boundaries

You can generate an initial mesh even if there is a *hole* in a boundary either due to missing faces or a gap between two zones. In this case, instead of having two separate zones on opposite sides of the boundary, you will have just one combined zone. This situation is evident because you will have very few zones.

- If the hole is in the outer boundary, the cells outside the boundary will be combined with the cells inside, resulting in an error. The edges around the hole will be marked as free edges and therefore can be displayed.
- If the problem is caused by a small gap between boundaries and the duplicate nodes have already been merged, increase the tolerance and perform additional merging.

The following TUI command can be used to detect holes in the geometry by tracing the path between the two specified cells:

```
/mesh/tritet/trace-path-between-cells
```

A path between the selected cells will be highlighted.

Interior Node Near Boundary

During refinement, a node may be placed too close to a boundary face, resulting in a highly skewed cell. You can detect this situation by displaying the highly skewed cells. You can then smooth the grid to eliminate the problem.

If the interior node is constrained by the boundary faces, increase the **Min Boundary Closeness** (in the Tri/Tet Refine Controls panel (see Section 11.5.2: [The Tri/Tet Refine Controls Panel](#))) and regenerate the mesh. Refer to Sections 13.5 and 13.7.3 for details.

