Appendix E. Tips

This appendix contains tips on the following topics:

E.1 Reading Files

- When TGrid reads a mesh file, it first searches for a file with the exact name you typed. If a file with that name is not found, it will search for a file with `.msh` appended to the name.
- You can read a TGrid mesh file using the File/Read/Mesh... menu item or the text command `file/read-mesh`.
- If the boundary mesh is contained in two or more separate files, you can read them together. TGrid will assemble the complete boundary mesh for you.
- To read a 2D mesh into the 3D version of TGrid, use the File/Import/Fluent 2D Mesh... menu item or the `file/import/fluent-2d-mesh` text command.

- To read a 3D mesh into the 2D version of TGrid, use the File/Import/Fluent 3D Mesh... menu item or the `file/import/fluent-3d-mesh` text command.
- Read a compressed file using the text interface by entering the file name. First, TGrid attempts to open a file with the input name. If it cannot find a file with that name, it attempts to locate files with default suffixes and extensions appended to the name.
- The compressed file read will fail if the `compress` or GNU `gzip` compression routines are not available on your platform.
- Scheme source files can be loaded in three ways:
  - through the menu system as a scheme file
  - through the menu system as a journal file
  - through Scheme itself.
- TGrid cannot read grids from the solvers that have been adapted using hanging nodes. If you wish to read one of these grids into TGrid, you will need to coarsen the mesh within the solver until you have recovered the original unadapted grid.
- Only one journal file can be open for recording at a time, but you can write a journal and a transcript file simultaneously. You can also read a journal file at any time.
- On Windowssystems, only `-env`, `-gu` (with restrictions), `-help`, `-i journal`, `-r`, `-rx`, `-v`, and `-vx` are available.
- Journal files are always loaded in the main (i.e., top-level) text menu, regardless of where you are in the text menu hierarchy when you invoke the read command.
- The `read-journal` command always loads the file in the main (i.e., top-level) menu, regardless of where you are in the menu hierarchy when you invoke it.
- To start TGrid and immediately read a journal file, type `tgrid -i jou`, replacing `jou` with the name of the journal file you want to read.

E.2 Writing Files

- When TGrid writes a mesh file, `.msh` will be added to the name you type unless the name already ends with `.msh`.
- To write a mesh file that can be read by TGrid, use the File/Write/Mesh... menu item or the text commands `file/write-mesh` or `file/write-case`.

  File ⟶ Write ⟶ Mesh...

- To write a mesh file in the format that can be read by FLUENT, use either the File/Write/Mesh... menu item or the text command `file/write-mesh`.

  File ⟶ Write ⟶ Mesh...

- The Filter Options panel allows you to change the extension (e.g., `.cas`, `.msh`, `.neu`) and arguments used with a specified filter. For example, if you save the PATRAN files with a .NEU extension instead of `.neu`, you can either substitute or add `.NEU` to the extension list.

- Only one journal file can be open for recording at a time, but you can write a journal and a transcript file simultaneously. You can also read a journal file at any time.

- Only one transcript file can be open for recording at a time, but you can write a transcript and a journal file simultaneously. You can also read a journal file while a transcript recording is in progress.

- The compressed file write will fail if the `compress` or GNU `gzip` compression routines not available on your platform.

E.3 Saving Hard Copy Files

- For the quickest print time, save vector files for simple 2D displays and raster files for complicated scenes.

- For raster hardcopy files, you can control the resolution of the hardcopy image by specifying the size (in pixels).

- Non-geometric entities such as text, titles, color bars, and orientation axis are not exported using the VRML option. In addition, most display or visibility characteristics set in TGrid, such as lighting, shading method, transparency, face and edge visibility, outer face culling, and hidden line removal, are not explicitly exported but are controlled by the software used to view the VRML file.

- Before saving a hardcopy file, you can have a preview of the saved image. Click Preview, to apply the the current settings to the active graphics window so that you can see the effects of different options interactively before saving the hardcopy.

- Raster supports IRIS image, JPEG, PostScript, EPS, and TIFF formats.

- Vector supports PostScript, EPS, and VRML formats.

- For the quickest print time, save vector files for simple 2D displays and raster files for complicated scenes. For raster hardcopy files, you can control the resolution of the hardcopy image by specifying the size (in pixels).

- Most monochrome PostScript devices will render Color images in shades of gray. To ensure that the color ramp is rendered as a linearly-increasing gray ramp, select Gray Scale.
- For all formats except the window dump you can specify which type of Coloring you want to use for the hardcopy file. For all hardcopy formats except the window dump, you can control two additional settings under Options.
- For PostScript, and EPS files, specify the resolution in dots per inch (DPI) instead of setting the width and height.
- Each time you create a new window dump, the value of `%n` will increase by one, so you there is no need to track numbers onto the hardcopy filenames manually.
- If you are planning to make an animation, save the window dumps into numbered files, using the `%n` variable. To do this, use the Window Dump Command. Enter `myfile%n.xwd` as the filename in the Select File dialog box.
- If you use the ImageMagick animate program, save the files in MIFF format (the native ImageMagick format). It is more efficient.
- When saving window dumps, remember that the window dump will capture the window exactly as it is displayed, including resolution, colors, transparency, etc. For this reason, all of the inputs that control these characteristics are disabled in the Hardcopy panel when you enable the Window Dump format.
- You can save your mesh to a file that can be read by HYPERMESH, NASTRAN, PATRAN, and ANSYS.

E.4 Importing Meshes

- A single filter, `fe2ram` allows you to convert files created by several finite-element packages to the grid file format used by TGrid. This filter allows you to convert surface or volume meshes from ANSYS, I-deas, NASTRAN, PATRAN, or other packages.

  The ARIES files can be converted if they are first saved as ANSYS Prep7 files, as described in Section A.3.5: ARIES Files.
- ARIES provides a filter to TGrid or you may write a Prep7 file from ARIES and use the `fe2ram` filter with arguments for an ANSYS file.

  For more information on importing ANSYS files, see Section A.3.4: ANSYS Files.
- When you read the mesh into the solver, it will ignore the dead cell zones. If you have a very large mesh, and there are many cells in a dead zone, delete them. To do so, use the Manage Cell Zones panel. This reduces the memory requirements.

E.5 Creating a Mesh

- If cells (e.g., prisms or pyramids) have already been created in some portion of your computational domain, create a "domain" encompassing the region to be meshed with triangular or tetrahedral cells before you can begin the tri/tet mesh generation.

  See Section 13.10: Using Domains to Group and Mesh Boundary Faces for details. All automatic or manual meshing actions will apply only to the active domain.

- To improve the speed and reduce the number of potential problems, introduce nodes in a scattered manner. It is more effective than consecutively introducing nodes in close proximity.

- After the mesh is initialized, TGrid will set any non-fluid zones to Non-Fluid Type. If your mesh includes multiple regions (e.g., the problem for which you are creating a grid includes a fluid zone and one or more solid zones), and you plan to refine all of them using the same refinement parameters, modify the Non-Fluid Type *before* initializing the mesh.

- If there are intersecting faces in the mesh, TGrid will not be able to generate a mesh until the problem faces are removed. If there are no free or multiply-connected edges then it will not be possible to identify the faces until meshing is attempted.

E.6 Grouping Elements

- All boundary types are considered as wall zones. You can set the appropriate boundary types in TGrid or in the solver.

- The `Group` command in I-deas is used to create the boundary zones needed by TGrid.

  – All faces grouped together are listed together in the output as a single zone. In FLUENT, boundary conditions are set on a per-zone basis.

  – All nodes grouped together are listed together in the output as a single zone. It is important *not* to group nodes of internal faces with nodes of boundary faces.

  – In PATRAN, named components are applied to the nodes to create groups of faces called zones. In FLUENT, boundary conditions are applied to each zone.

E.7 Deleting Duplicate Nodes

- I-deas often generates duplicate nodes in the process of creating triangular elements. They can be removed by using one of the following:

  – The `remove coincident node` command in I-deas.

  - The Merge button in the Merge Boundary Nodes panel (in TGrid the `boundary/merge-duplicates` text command).

- The node merging process is usually faster in TGrid, but more visual in I-deas.

- NASTRAN often generates duplicate nodes in the process of creating triangular elements. These can be removed by using the Merge button in the Merge Boundary Nodes panel (or the `boundary/merge-duplicates` text command) in TGrid.

E.8 Manipulating the Boundary Mesh File

- In TGrid, modify the boundary mesh to improve its quality.

  The quality of the boundary mesh influences the interior mesh generation and the numerical analysis. Quality is measured based on four criteria: clustering, smoothness, skewness, and aspect ratio.

- Smoothness is particularly critical in regions where surfaces intersect or where they are in close proximity.

- Skewness of the cells can have a dramatic influence on the accuracy and convergence characteristics of the numerical integration technique.

  Highly skewed cells decrease accuracy and slow convergence. A detailed discussion of skewness can be found in Section 15.5: Mesh Quality.

- Free edges are acceptable if you are modeling a zero-thickness wall ("thin wall") in the geometry (e.g., Figure 7.1.2).

- Improving the boundary mesh not only makes mesh generation easier, but also improves the final mesh quality and provides a better solution. See Section 7.3: Modifying the Boundary Mesh for boundary mesh improvement utilities.

- You can either retain the isolated nodes (nodes that are not used by any boundary faces) to influence the generation of the interior mesh or delete them. See Section 9.1.7: Inserting Isolated Nodes into a Tri or Tet Mesh.

- To merge or collapse multiple pairs, select an even number of nodes and ensure that you select them in the correct order.

- Select a subset of all boundary zones for node comparison to save time. TGrid will be able to search for duplicate nodes and merge them more quickly if it does not need to check the nodes on all boundaries.

- If the initial mesh generation is not successful, the nodes that failed to be inserted are moved to the front of the node list so that they will be inserted first in the next attempt.

E.9 Examining the Mesh

- After reading the mesh file, check the boundary mesh for topological problems such as free and multiply-connected nodes and faces.
- When TGrid initializes the mesh, it automatically makes the cell zone with the largest volume (or the cell zone that contains a boundary zone with type pressure-outlet) the active fluid zone. TGrid treats the rest of the cell zones (non-fluid zones) as dead zones.
- If any unmeshed nodes or faces are displayed, you will need to regenerate the initial mesh.
- 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), you can report the face distribution only for those face zones that are in the active domain.
- When using the sliders and dial to manipulate the view, turn off Wireframe Animation in the Display Options panel (see Section 14.5.1: The Display Options Panel), to watch the display move interactively while you move the slider or the dial indicator.
- You can edit the text in the caption block of the graphics window by clicking your left mouse button in the desired location. A cursor will appear, and you can then type new text or delete the text that was originally there. Text in the caption block will *not* be deleted when you clear annotations.

E.10 Reporting Mesh Statistics

- If the quadrilateral face skewness is much greater than 8, regenerate them.
  - If they were created in a different preprocessor, return to that application and try to reduce the aspect ratio of the faces in question.
  - If they were created by TGrid during the building of prism layers, rebuild the prisms using a more gradual growth rate.
- For 2D, a maximum skewness less than 0.5 and an average of 0.1 are good. For 3D, a maximum skewness less than 0.9 and an average of 0.4 are good.
- To check the skewness of boundary cells, use the Report Boundary Cell Limits panel or the associated text commands.
- 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), the report will apply only to the active domain.

- The skewness method you choose in the Quality Measure panel will only affect reports and displays based on skewness. The triangular and tetrahedral meshing procedure will always be based on the equilateral volume skewness method.

E.11 Refining the Mesh

- The refinement procedure is repeated a number of times. You can control the number of refinement levels by setting the Number of Levels (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 (increase the number of cells) and lower the cell skewness average.

- Each refinement level actually consists of two sweeps through the refinement procedure where in the second sweep only refines a cell if the local skewness is improved. This sweep attempts to reduce highly skewed cells in confined regions of the geometry. After the refinement sweeps, you can try boundary sliver removal.

- Canceling the meshing process during refinement will cause the algorithm to exit at the next reasonable stopping place, leaving the mesh in an acceptable state.

- The default value of Minimum Skewness is 0.85. You may increase this value, but do not decrease it, since smoothing operations are very time-consuming, and the lower the value of the Minimum Skewness, the smoothing process will take more time.

- For some meshes the first sweep of the first refinement level will never finish because more skewed cells are formed due to refinement, which will then be further refined to create more skewed cells, and so on.

  This problem can be avoided by turning on the Incremental Improvement option.

- Various steps in the automatic meshing process can be added or eliminated, as required . In addition, most aspects of the automatic meshing process can be changed dynamically through the Scheme interface.

- 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.

- There are situations where it may be undesirable to introduce additional nodes to eliminate highly skewed cells. In these situations, face swapping is used to improve the grid quality.

- Smoothing is a very time-consuming process, so you should only perform smoothing on cells with high skewness. Improved results can be obtained by smoothing the nodes several times.
- Carefully 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 lessen the adverse effects on the average skewness.
- After obtaining an initial triangulation of the boundary nodes, the final volume mesh is produced by introducing additional nodes and swapping interior triangular faces. This step is separated into two parts: refining a cell with one or more boundary faces and refining non-boundary cells.
- It is undesirable to add a node too close to a boundary because if a highly skewed cell is created, it will be almost impossible to remove it.
- 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.
- Near the boundaries, there will often be cells of higher skewness, because of the limitations placed on node placement. These can be removed with face swapping.
- Of the three methods available for improving skewness, *swapping* and *smoothing* improves the mesh by manipulating the nodes and faces without increasing the total number of cells.

  Refinement improves the mesh by adding cells.
- To get the best possible mesh with the smallest number of cells, perform smoothing and swapping before refining the mesh any further.
- Since 2D meshes do not suffer from the problems that occur in 3D, swapping and smoothing have no effect in 2D, so they need not be performed.
- After initializing the mesh, you can improve the quality and density of the grid using global and local refinement. The global refinement visits all cells in the active zones, while the local refinement only visits cells within a specified region.

  In most applications, use only the global refinement to create an acceptable discretization of the volume. The local refinement is used to modify the grading away from boundaries or increase the resolution of an interior region of the grid.
- Swapping is *not* performed if it will significantly alter the geometry defined by the original boundary mesh.
- If the pyramids of the pyramid cell zone are on the wrong side, delete the newly created zones related to the pyramids.

- Each refinement level actually consists of two sweeps through the refinement procedure:
  - One sweep at the appropriate skewness thresholds for that particular level of refinement.
  - Another sweep with high skewness thresholds and a relaxed Min Boundary Closeness set in the Tri/Tet Refinement Controls panel (see Section 11.5.2: The Tri/Tet Refine Controls Panel).
- If it is desirable to have a small highly skewed cell than a large one, two modifications are introduced.
  - The face centroid is used (not the circumcenter).
  - The distance from the face is limited by the length of the longest face edge.
- The basic strategy is to refine the cells with the highest skewness, starting with the boundary cells, then interior cells, and then conclude each refinement pass with swapping and smoothing. In each subsequent refinement, the cell skewness refinement parameters are lowered to increase the mesh density and gradually improve the quality of the mesh.
- Sometimes there can be flat cells with two boundary faces that cannot be removed with face swapping. However by claiming the two interior faces to be boundary faces, the cell is effectively eliminated.

E.12 Using the GUI

- 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), you can report the face distribution only for those face zones that are in the active domain.
- The Wireframe Animationoption is turned on by default. It uses a wireframe representation of all geometry during mouse manipulation. Turn it off only if your computer has a graphics accelerator. Otherwise the mouse manipulation may be very slow.
- Double Buffering turns double buffering on or off, if it is supported by the driver. Double buffering drastically reduces screen flicker during graphics updates. If your display hardware does not support double buffering and you turn this option on, double buffering will be done in the software. Software double buffering uses extra memory.
- Hidden Line Removal turns hidden line removal on or off. If you do not use hidden line removal, TGrid will not try to determine which lines in the display are behind others; it will display all of them, and a cluttered display will result.

Turn this option off if you are working with a 2D problem or with geometries that do not overlap. Hidden Surface Removal works in a similar manner hiding/displaying the surfaces instead of lines.

- In the Views Panel, the Delete button removes the selected view name from the views list. Be careful not to delete any of the predefined views.
- When using the sliders and dial to manipulate the view, turn off Wireframe Animation in the Display Options panel (see Section 14.5.1: The Display Options Panel) to watch the display move interactively while you are moving the slider or the dial indicator.
- You can edit the text in the graphics window's caption block by clicking your left mouse button in the desired location. A cursor will appear, and you can then type new text or delete the text that was originally there. Text in the caption block will *not* be deleted when you clear annotations.
- Although the Zone Selection Helper panel is opened from the Display menu, it can be used with *all* panels that contain zone lists (e.g., Cell Zones and Boundary Zones panels).
- To look at the nodes and faces that have not been meshed, turn on the Unmeshed option in the Display Grid panel.
- If you use the Compute button to calculate the average normal direction, be sure that you do not change the sign of the displayed Vector. It is important for the uniform direction vector to have the same sign as the normal direction for the face zone.
- In the Prisms panel, lowering the value of layer below the number of layers to be grown can save computation time. But this should only be done when high skewness is not an issue.
- If any new cell has zero or negative volume or any left-handed faces, or if any skewness exceed the specified Max. Allowable Skewness, prism layer creation will be stopped.
- Face swapping and skewness-based smoothing apply only to triangular and tetrahedral cells. Laplace smoothing, however, can be applied to all the cell shapes.
- 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, so you should limit the smoothing to highly skewed cells.