ANSYS TurboGrid User's Guide

Table of Contents

Chapter 1: ANSYS TurboGrid Workspace

1.1. Introduction

The ANSYS TurboGrid interface is divided into several parts, as shown in *Figure 1.1* (p. 1). This chapter describes two main parts of the ANSYS TurboGrid interface: the *object selector* and the *object editor*.

Figure 1.1 Workspace
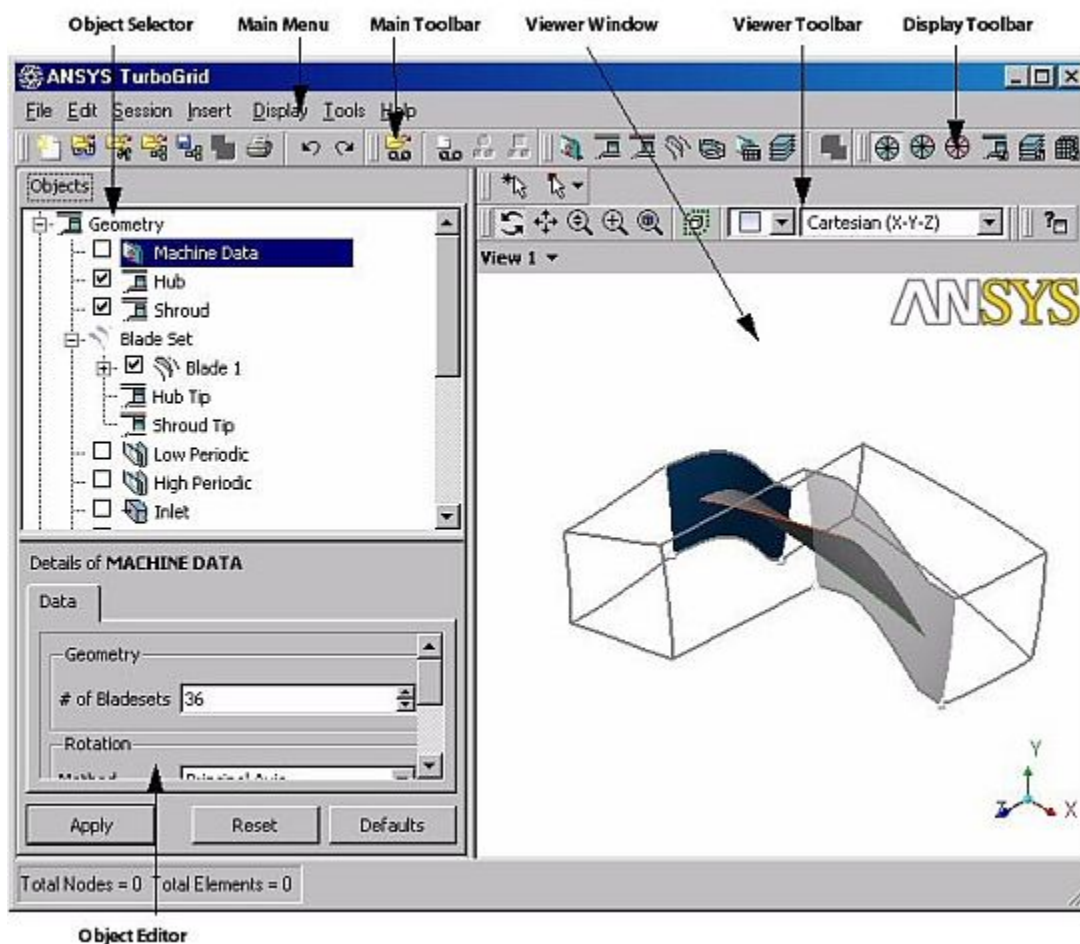
After reading this chapter, you will be familiar with the basic features of the object selector and the object editor.

Information about main menu options and the viewer is given in subsequent chapters in this guide.

ANSYS TurboGrid Workflow (p. 71) shows how to use ANSYS TurboGrid.

1.2. Object Selector

The object selector lists all existing objects in ANSYS TurboGrid.
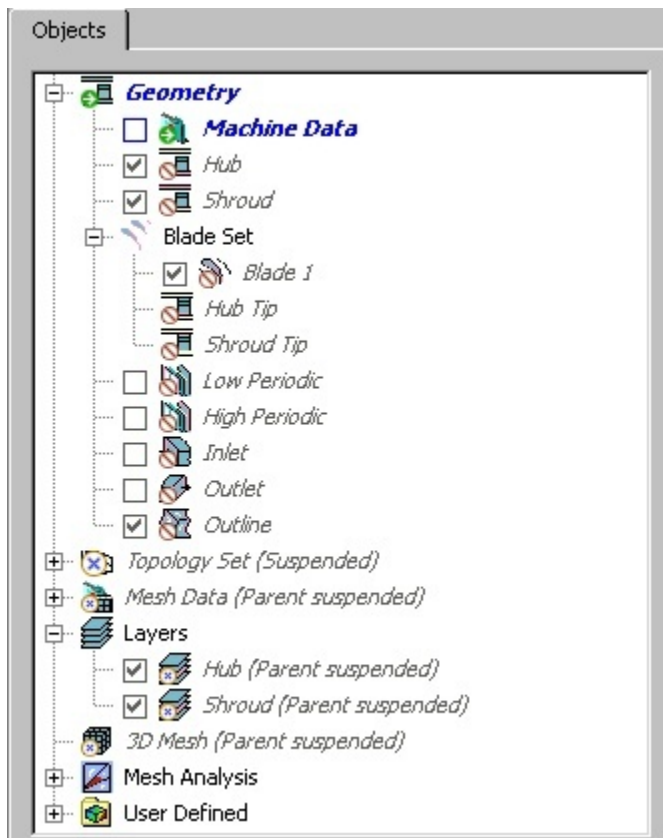
The object selector initially contains some objects in a tree format. *Objects* are data items used to drive aspects of mesh generation, visualization, and calculations. These objects must be defined in a certain order. For details, see *Steps to Create a Mesh* (p. 71).

To help guide you through the mesh creation process, the object names are listed with special icons and text fonts in the object selector.

Table 1.1 Icon Overlays and Text Styles

Icon Over-lay	Font	Appended Phrase	Description
	Grey Ital-ic		The object cannot be pro-cessed. Some other object(s) must be defined before this object can be processed.
	Blue Bold Ital-ic		The object is ready to be defined, if applicable, and then processed. The object must be processed before a mesh can be created.

Icon Over-lay	Font	Appended Phrase	Description
	Black		The object is complete and re-quires no more information before a mesh can be gener-ated. The object can, however, be edited.
⚠	Red Bold Ital-ic	(Error)	The object has a problem. In the case of the `Mesh Analys-is` object, a red font indicates that at least one mesh statistic falls outside the limits set in the `Mesh Limits` object.
✖		(Suspended)	The object will not be pro-cessed because it is suspended. You can control whether such an object is suspended from the shortcut menu. For details, see *Suspend Object Updates Command* (p. 142).
✖		(Parent suspen-ded)	The object is suspended be-cause a parent object is suspen-ded. For example, whenever the Topology Set object is sus-pended, the Mesh Data object will also be suspended.

The tree view reflects the structure of the object definitions. For example, there is a `Hub` object in both the `Geometry` and `3D Mesh` branches. To select the geometry `Hub` object, select the `Hub` object from the `Geometry` branch.

You can open the object editor for any object by:

- Double-clicking it.

  You may need to expand a tree branch to reach a particular object. This is accomplished by clicking on the plus symbol at the root of the branch.

- Right-clicking the object and using the shortcut menu.

  Shortcut menu items will be available according to the type of object. All shortcut menu commands are described in *Shortcut Menu Commands* (p. 136).

An alternative way to edit an object is by using the **Command Editor** dialog box. The **Edit in Command Editor** menu item is available by right-clicking an object in the object selector. This operation opens the **Command Editor** dialog box and displays the definition of the object and its parameter settings. Edit the CCL to change the object. For further details, see *Command Editor Command* (p. 66).

1.2.1. Visibility Check Box

In the object selector, each object that can be displayed in the viewer window has a check box to the left of it. The check box controls the visibility of the object in the viewer. Selecting the check box turns visibility on for that object, while clearing the check box turns the visibility off. For details, see *Common Options* (p. 5).

1.3. Object Editor

The object editor is used to define or edit the properties of an object. It contains a set of one or more tabs that depend on the type of object being edited.

Many properties can be set via a CEL expression. To enter an expression, click in the box for a property, then click the *Enter Expression* 🖩 icon which appears beside the field, then do one of the following:

- Enter an expression definition directly.
- Type the name of an existing expression.

    (This is a special case of the first point.)

You must ensure that the expression evaluates to a value having appropriate units for the property that uses the expression.

For details on CEL expressions, see ANSYS CFX Expression Language in the *TurboGrid Reference Guide*.

When using ANSYS TurboGrid in ANSYS Workbench, you can set any property that accepts a CEL expression via a parameterized CEL expression with the parameterized CEL expression being defined by a Workbench input parameter. You can do this in either of the following ways:

- Create an expression and parameterize it using the **Expression Editor** dialog box, then use that expression as the value of the property. For details, see *Expression Editor Dialog Box* (p. 62).

- In the object editor, click the *Set to Workbench input parameter* ☐ icon (which appears to the right of the property) and then, in the **Enter Parameter Name** dialog box, either enter a unique name for a new Workbench input parameter (a suggested name is entered by default) or select an existing Workbench input parameter.

    If you make changes in the object editor, be sure to apply them (click **Apply**) before clicking the *Set to Workbench input parameter* ☐ icon, otherwise your changes will be lost.

    If you cannot see the *Set to Workbench input parameter* ☐ icon, click the user interface away from the property that you are trying to parameterize.

You can click *Set to value* 🅿 to stop a property from being specified by a Workbench input parameter. However, the corresponding CEL expression persists, and can be managed by the **Expression Editor** dialog box. For details, see *Expression Editor Dialog Box* (p. 62).

A property that accepts only an integer value is also capable of being controlled by a Workbench input parameter. For such a property, the *Set to Workbench input parameter* ☐ and *Set to value* 🅿 icons are used in the same way as for a property that accepts a CEL expression. Integer-value properties

cannot take expressions. (No CEL expression is involved in transferring the Workbench input parameter's value from ANSYS Workbench to ANSYS TurboGrid, for a property that accepts only an integer value.)

1.3.1. Common Options

The options located at the bottom of the object editor are available from any of the tabs, and are common to most objects. A description of each option follows:

- The **Apply** button saves the changes made to all the tabs and updates the object in the viewer. Objects that depend on the edited object are also updated. For example, altering a plane would also change any object that uses that plane in its definition.

- The **Reset** button returns the settings for the object to those stored in the database for all the tabs. The settings are stored in the database each time you click the **Apply** button.

- The **Defaults** button restores the system default settings for all the tabs of the object. The system defaults are stored for each parameter, without regard for the types of object(s) in which the parameter may be used. For this reason, using the **Defaults** button may result in unsuitable changes to an object's settings, and is not recommended.

1.3.2. Data Tab

Most objects have one or more tabs which define the object. These are often called *data* tabs; e.g., the one for a hub is called **Data Hub**. The data tab displays the definition of the object currently being edited. Each of the objects are described in *ANSYS TurboGrid Workflow* (p. 71).

1.3.3. Color Tab

1.3.3.1. Mode

The **Color** tab controls the color of objects in the viewer. The color can be either constant or based on a variable. Select one of the two options for **Mode**.

- Select `Constant` to specify a single color for an object. To choose a color, click the ⬚ icon to the right of the **Color** box.

- Select `Variable` to plot a variable on an object (maximum face angle on a plane, for example).

- Select `Use Plot Variable` (available for some plots, including an isosurface) to color an object by the same variable used to define it.

1.3.3.2. Variable

Choose the variable to plot from the **Variable** drop-down list. The drop-down list of variables contains the most commonly used variables. For a full list of variables, click the ⬚ icon.
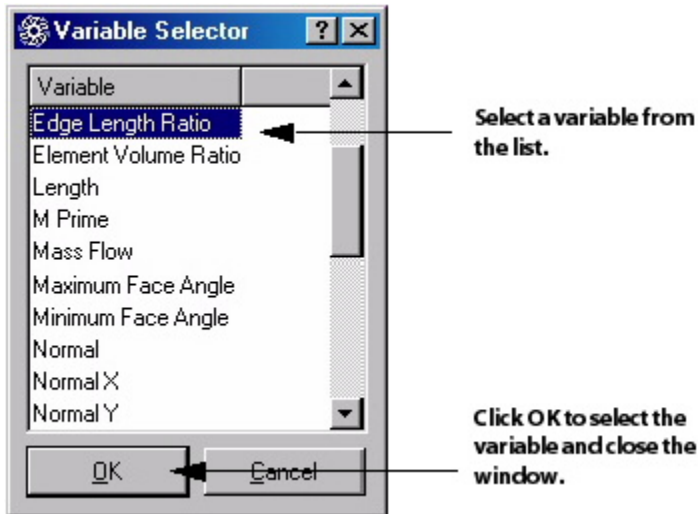
1.3.3.3. Range

Click ▼ next to the **Range** box to see the available methods for defining the range of the variable used to define the plot. This affects the variation of color used when plotting the object in the viewer. The lowest values of a variable in the selected range are shown in blue in the viewer, the highest values are shown in red.

- `Global` uses the range of the variable over all domains (regardless of the domains selected on the **Geometry** tab) to determine the minimum and maximum values.

- `Local` uses the range of the variable over the current object to determine the minimum and maximum values. This option is useful for utilizing the full color range on the object.

- Using `User Specified`, enter the minimum and maximum values for the contours. This option is useful to concentrate the full color range in a specific variable range. The variable values can be typed in, set using the embedded slider or, by clicking the 🔲 icon to the right of the **Units** box, entered as an expression. Click ▼ in the box to the right of the variable value to see the available units for the variable(s).

1.3.3.4. Hybrid and Conservative Values

Select whether the plotted object is based on hybrid or conservative values of the variable used for coloring.

1.3.3.5. Undefined Color

Any areas in which the variable is not defined (when a section of an object lies outside of the computational domain, for example) use the color specified in the **Undef. Color** box. Click the ⸱⸱⸱ icon to the right of this box, to change the undefined color.

1.3.4. Render Tab

The exact appearance of the **Render** tab depends on the type of object plotted in the viewer window.

1.3.4.1. Draw Faces

If the **Draw Faces** check box is selected, the faces which make up an object are drawn. The faces are colored using the settings on the **Color** tab.

To change the transparency of an object, type in the transparency value or use the embedded slider (which has a maximum value of 1 and a minimum value of 0). A transparency of 0 means the shading is opaque (or having no transparency) and a transparency of 1 means the shading is invisible (completely transparent).

Shading properties can be changed to either `None`, `Flat Shading` or `Smooth Shading`.

- Select `None` so that no shading is applied to the object; it appears black.
- Select `Flat Shading` so that each rendered element is colored a constant color. Color interpolation is not used across or between rendered elements.
- Select `Smooth Shading` so that color interpolation is applied which results in color variation across a rendered element based on the color of surrounding rendered elements.

Lighting can be turned on and off by selecting/clearing the **Lighting** check box.

Specular lighting can be turned on and off by selecting/clearing the **Specular** check box. When selected, objects appear to reflect light.

Face culling turns off visibility of rendered element faces of objects that either face the viewer or point away from the viewer. Domain boundaries always have a normal vector that points out of the domain. **Face Culling** options are:

- `Front Faces`

  Selecting `Front Faces` turns off visibility of all outward-facing rendered element faces (the faces on the same side as the normal vector). This would, for example, turn off visibility of one side of a plane or the outward facing rendered elements of a cylinder locator. When applied to a volume object, the first layer of rendered element faces that point outwards are rendered invisible.

- `Back Faces`

  Selecting `Back Faces` turns off visibility of inward-facing rendered element faces (the faces on the opposite side to the normal vector). When applied to volume objects, the effect of back culling is not always visible in the viewer, since the object-rendered elements that face the outward direction obscure the culled faces. It can, however, reduce the render time when further actions are performed on the object. The effect of this would be most noticeable for large volume objects. In the same way as for front face culling, it turns off visibility of one side of surface locators.

- `No Culling`

  `No Culling` turns on the visibility of all rendered element faces.

  Note

  Face culling affects printouts performed using the `Use Screen Capture` method.

1.3.4.2. Draw Lines

If the **Draw Lines** check box is selected, the lines which make up the object's surface are drawn. To change the line width, type in the line width value, increase or decrease the value by 1 by clicking the up and down arrows, or use the embedded slider (which has a maximum value of 10 and a minimum value of 1). Line color can be changed by clicking on the [...] icon to the right of the **Color** box.

The **Edge Angle** setting is used to limit the number of visible edges in a plot. The edge angle is considered to be the angle between two faceted faces of a surface which are connected by an edge. If the angle between two adjacent faces is greater than the **Edge Angle** setting, then the edge shared by the faces is drawn. If the edge angle is 0°, the entire surface is drawn. If the edge angle is large, then only the most significant corner edges of the surface are drawn.

A sensible setting for **Edge Angle** depends on the geometry. Experiment to get a value that clearly shows where the surface is located, without displaying too much of the surface mesh. Too many lines can make it confusing when more objects are added to the geometry.

Setting an edge angle defines a minimum angle for drawing parts of the surface mesh. For example, if an edge angle of 30° is chosen, any edges shared by faces with an angle between them of 30° or more are drawn.

Reducing the edge angle shows more of the surface mesh in the viewer. When the edge angle is 0°, all of the surface mesh is shown.

1.3.4.3. Applying Instance Transforms to Objects

Instance transforms are created separately using **Insert** > **User Defined** > **Instance Transform**. For details, see *Instance Transform Command* (p. 44). The **Apply Instancing** check box is selected by default, and `Default Transform` is selected. To apply a different transform, it must be created and then selected from the list of existing instance transforms.

Chapter 2: File Menu

The following sections describe the commands available in the **File** menu:

2.1. Introduction

ANSYS TurboGrid uses and produces the following file types:

Session File

`.tse` session files are produced by ANSYS TurboGrid and contain CCL commands. Session files record the commands executed to a file for playback at a later date. Use session files to run ANSYS TurboGrid in batch mode. See *Session Menu* (p. 27) and Batch Mode in the *TurboGrid Reference Guide* for details.

> **Note**
>
> Because the session file is a text file of CCL commands, you can write your own session files using a text editor.

State File

`.tst` state files are produced by ANSYS TurboGrid and contain CCL commands. They differ from session files in that only a snap-shot of the current state is saved to a file. Using state files, you can close ANSYS TurboGrid and continue working later from the same point. See *Load State Command (Import State Command)* (p. 14) and *Save State Command (Export State Command)* (p. 15) for details.

> **Note**
>
> Because the state file is a text file of CCL commands, you can write your own state files using a text editor.

> **Note**
>
> State files previously had the extension `.cst`.

Topology File

`.tgt` topology files are produced by ANSYS TurboGrid and define the topology. Using topology files, you can use the same topology for various cases without having to redefine it each time.

> **Note**
>
> Because the topology file is a text file, you can write your own topology files using a text editor.

Grid and Boundary Condition Files

`.grd` and `.bcf` grid and boundary condition files are produced by ANSYS TurboGrid and contain the mesh in a format which can be read by CFX-TASCflow. For details, see *Save Mesh Command* (p. 17).

Mesh File

`.gtm` mesh files are produced by ANSYS TurboGrid and contain the mesh in a format which can be read by ANSYS CFX. For details, see *Save Mesh Command* (p. 17).

BladeGen.inf File

`BladeGen.inf` information files contain machine data and curve file data. By loading a `BladeGen.inf` file, you can set up the machine data, hub, shroud, and blade curves.

When loading a `BladeGen.inf` file, the curve type of the `Hub` and `Shroud` geometry objects will be set to `Piecewise Linear`.

Curve File

`.crv` and `.curve` curve files are used by ANSYS TurboGrid to define machine geometry. These files contain points in free-format ASCII style and can be created using a text editor, ANSYS BladeGen or by saving a modified blade geometry in ANSYS TurboGrid.

TurboGrid 1.6 cfg File

`.cfg` simulation setup files contain machine data. By loading a `.cfg` file, you can set up the machine data, the hub, shroud, blade curves, and the tip geometry.

Tetin File

`.tin` Tetin files are produced by ANSYS TurboGrid and describe the geometry in a format which can be read by ICEM CFD products. For details, see *Export Geometry Command* (p. 20).

Picture Files

PNG (`.png`), CFD Viewer State (3D), JPEG (`.jpg`), Windows Bitmap (`.bmp`), PPM (`.ppm`), PostScript (`.ps`), Encapsulated PS (`.eps`), and VRML (3D) (`.wrl`) files can be saved. For details, see *Save Picture Command* (p. 20).

2.2. New Case Command

To display the object selector and begin a new ANSYS TurboGrid simulation, select **File** > **New Case** from the main menu or click *New Case* ⬜.

2.3. Load BladeGen Command

To load a BladeGen `inf` file, select **File** > **Load BladeGen**. The **Open BladeGen File** dialog box is displayed. Select a file to load, then click **Open**. The `inf` is structured as follows:

```
!======  CFX-BladeGen Export  ========
Axis of Rotation: Z
Number of Blade Sets: 9
Number of Blades Per Set: 2
Geometry Units: MM        <---- Unknown|IN|MM|FT|MI|M|KM|MIL|UM|CM|UIN
Blade 0 LE: EllipseEnd    <---- EllipseEnd|CutoffEnd|SquareEnd
Blade 0 TE: CutOffEnd
Blade 1 LE: EllipseEnd
Blade 1 TE: CutOffEnd
Hub Data File: hub3.curve
Shroud Data File: shroud3.curve
Profile Data File: profile3.curve
```

(The statements following "<----" are comments that show possible values. They are not part of the format.)

If the `inf` file does not specify the curve files, you may turn on an option in the dialog box (called **Guess missing curve files**) in order to guess the names of the missing curve files. The algorithm for guessing curve files looks for files in the working directory with a `.curve` or `.crv` extension, with "hub", "shroud", or "profile" in the name (using a case-insensitive search). If the curve file names are unusual, you should verify that the correct curve files were selected by opening the appropriate `Geometry` object (for example, the `Hub` object) in the object editor, or by selecting **File** > **Load Curves** and examining the curve file names.

If a `BladeGen.inf` file specifies multiple blades in the blade set, multiple blades will be generated.

Note

After loading a `BladeGen.inf` file, the **Curve Type** settings for the `Hub` and `Shroud` objects will be set to `Piece-wise linear` instead of the default (`Bspline`).

2.4. Load Curves Command

To load new geometry curves, select **File** > **Load Curves** from the main menu or click *Load Curves* ![icon] . The **Load TurboGrid Curves** dialog box is displayed.
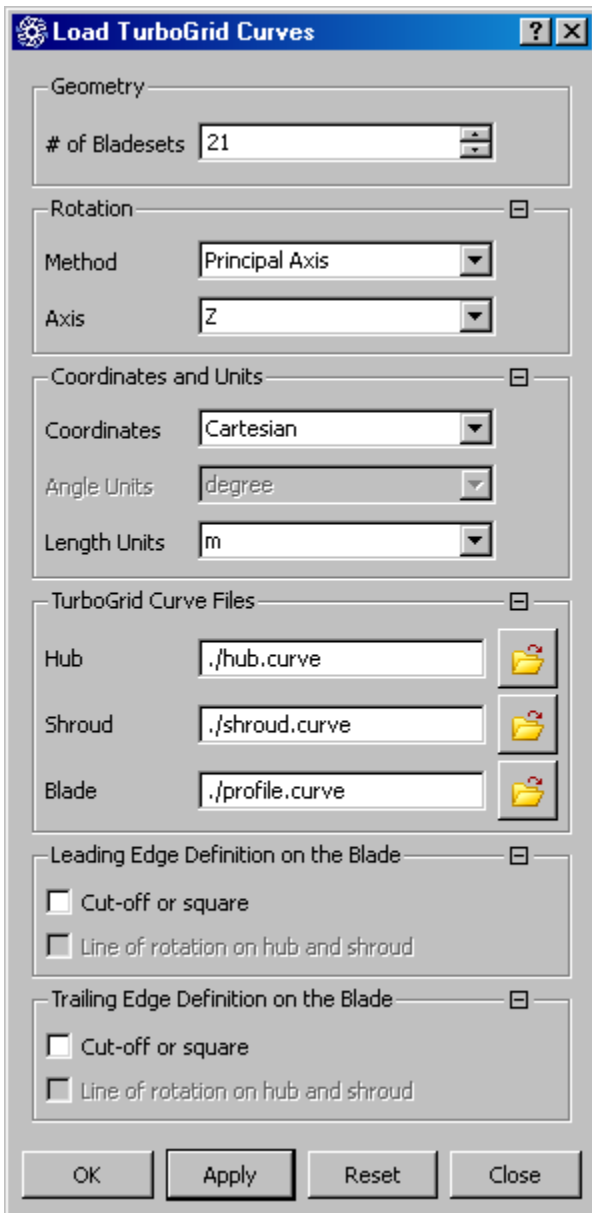
This dialog box provides a convenient way to define multiple geometry objects, compared to editing each geometry object separately using the object selector. It is useful when you first open ANSYS TurboGrid and when you want to use the same settings on a slightly modified geometry.

The information specified on the **Load TurboGrid Curves** dialog box will be used to overwrite the corresponding information in the appropriate geometry objects. No other previously-defined settings are affected. For example, if a control angle is defined for the Hub object, it will not be changed by using the **Load TurboGrid Curves** dialog box.

If a curve file that specifies multiple blades in the blade set is specified for the Blade Set object, multiple blades will be generated. If a curve file that specifies multiple blades in the blade set is specified

for a particular blade object (stored under the `Blade Set` object in the object selector), only the first blade in the file is used. If blade names are not included in the curve file, then names will be generated in the form Blade *n*, where *n* is an integer.

2.4.1. Geometry

A rotating machine component is made up of adjacent blades which are equally spaced around the circumference of the machine. The Theta extent of one blade set is calculated as 360 degrees divided by the number of main blades. Many rotating machine components have secondary and tertiary blades which are placed between the main blades. These are often called splitter blades. A blade set contains one main blade and optional splitter blades which repeat cyclically around the axis of the rotating machine component.

For rotating machine components without any splitter blades, the number of blade sets equals the total number of blades.

ANSYS TurboGrid creates a mesh for one blade set only. The mesh can be copied and rotated using an ANSYS CFX Pre-processor, if necessary, before it is solved in an ANSYS CFX Solver.

2.4.2. Rotation

Click ⏷ next to the method box to see the available methods for defining the machine's rotation axis.

- Using `Principal Axis`, select the rotation axis from the X, Y or Z axes using the **Axis** drop-down list.
- Using `Rotation Axis`, define a custom rotation axis by entering the Cartesian coordinates of 2 points on the axis. The coordinates of the points can be typed in or set using the embedded sliders.

2.4.3. Coordinates and Units

Click ⏷ next to the corresponding boxes to select the coordinates, angle units, and length units used in the hub, shroud, and blade files. If any of these are not the same for all three geometry files, use the object selector to individually define them.

2.4.4. TurboGrid Curve Files

To specify a curve file, set a file name using a path relative to the working directory. You can click the corresponding *Browse* 📂 icon to select a curve file using a browser.

2.4.5. Leading/Trailing Edge Definition on the Blade

Define whether the trailing edge is `cut-off or square`. Optionally specify a line-of-rotation division of the mesh that stems from the trailing edge at the hub and shroud. The same applies for the leading edge.

2.5. Load CFG Command

To load ANSYS TurboGrid 1.6 simulation setup (CFG) files, select **File** > **Load CFG** from the main menu to access the **Load CFG File** dialog box.

The **Length Units** setting in the dialog box must be set correctly. The units that you specify will be used to interpret the data stored in the curve files specified by the CFG file.

The following data is loaded from a CFG file:

- # of passages
- Axis of rotation
- Existence of tip and location
- Coordinate frame for curve files
- Hub, shroud and blade curve file names
- Existence of inlet and outlet regions.

2.6. Reread Curves Command

To re-load the curve files that have previously been loaded into memory, select **File** > **Reread Curves**. This command is useful if you are updating the files in, for example, ANSYS BladeGen/Blade Modeler, and you want to re-load the updated files into ANSYS TurboGrid.

2.7. Load State Command (Import State Command)

A previously saved state file can be loaded into ANSYS TurboGrid. To load a state file in stand-alone mode, select **File** > **Load State** from the main menu or click *Load State* 🖳 on the toolbar; when running in ANSYS Workbench, select **File** > **Import State**. The **Load State File** dialog box is displayed.

If any file specified in a state or session file cannot be found, ANSYS TurboGrid will automatically search the state or session file directory for a file of the same name. If this search fails, the current working directory will be searched. As a result, state and session files will not have to be edited to change the path when state, session and curve files are moved from one directory to another.

Select the **Load as new simulation** radio button to delete all existing objects and create new objects which are defined in the state file.

Select the **Append to current simulation** radio button to add all objects defined in the state file to the existing objects. Existing objects are not removed unless they have the same name as an object in the state file, in which case they are replaced. Loading a state file in this way allows the use of a number of state files as building blocks for commonly used objects.

> **Note**
>
> If you are running ANSYS TurboGrid in ANSYS Workbench, and there is a geometry provided to ANSYS TurboGrid from an upstream cell, then that geometry is used and the geometry data in the state file is ignored.

> **Note**
>
> Because the `TOPOLOGY SET` object is now suspended by default, session and state files from version 11 or earlier may not play or load correctly. To support older session and state files, the **Play Session File** and **Load State** dialog boxes have an option named **Unsuspend TOPOLOGY SET before loading**. Selecting this option causes the `TOPOLOGY SET` object to be unsuspended before playing/loading a session/state file. When starting ANSYS TurboGrid from the command line, adding the command line parameter `-u` causes the `TOPOLOGY SET` object to be initially unsuspended.

2.8. Save State Command (Export State Command)

If you have not saved a state file during the current ANSYS TurboGrid session, selecting **File** > **Save State** from the main menu (**File** > **Export State** when running in ANSYS Workbench) opens the **Save State** dialog box where you can type a file name for your state file. For details, see *Save State As Command* (p. 15).

If you have already saved a state file during the current ANSYS TurboGrid session, selecting **File** > **Save State** from the main menu overwrites that file. To save a state to a different file name, select **File** > **Save State As...** from the main menu. For details, see *Save State As Command* (p. 15).

2.9. Save State As Command

Saving a state file produces a text file containing CCL commands for the current ANSYS TurboGrid state. To save a state to a new file, select **File** > **Save State As...** from the main menu or click *Save State As...*
. The **Save State** dialog box is displayed.
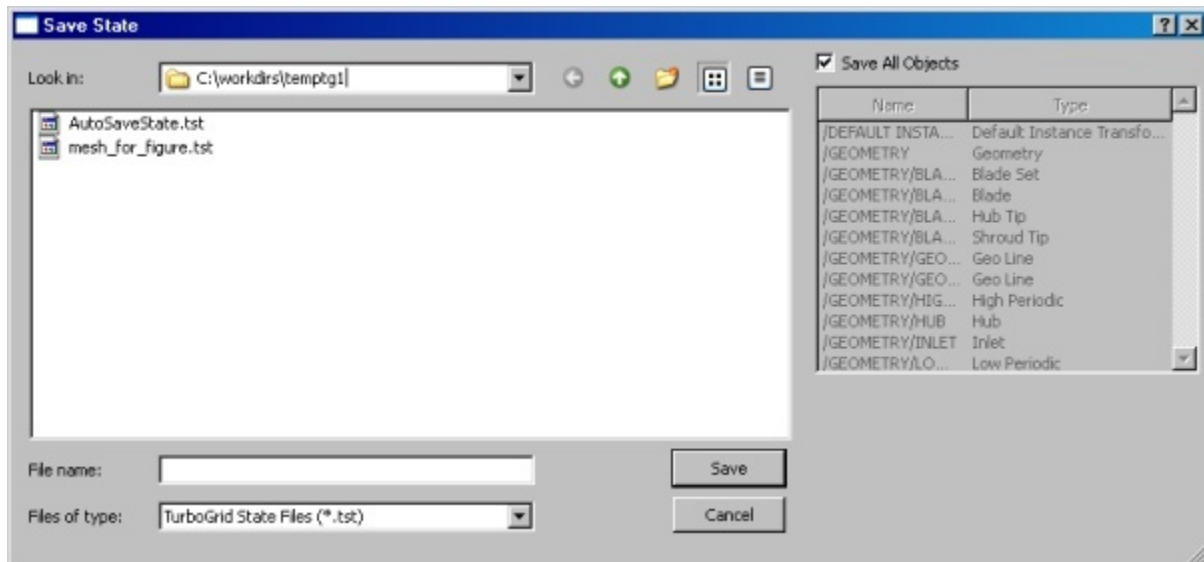
Set **Look in** to the directory in which you want to create the state file.

In the list on the right side, select the objects to include in the state file, or select **Save All Objects**. If **Save All Objects** is selected, the current state of all objects is written to the state file. If **Save All Objects** is cleared, select the objects to save to the state file by clicking on each object. The current state of all selected objects is written to the state file.

Enter (or select) a file name, then click **Save** to save the state file.

A state file is linked to the geometry files from which it was created by an absolute path; therefore, the location of the geometry files should not be changed. This also applies to topology files if the **From File** option is selected for a `Topology Set` object.

State files are automatically saved with a `.tst` file extension.

2.10. Save Project

When running in ANSYS Workbench, you can select **File** > **Save Project** to save the entire ANSYS Workbench project.

2.11. Refresh

When running in ANSYS Workbench, you can select **File** > **Refresh** to refresh the associated Turbo Mesh cell.

2.12. Save Submenu

Many objects that can be saved or saved under a different name are listed under the **Save** submenu.

2.12.1. Save Blade Command

If you have not saved a blade file during the current ANSYS TurboGrid session, selecting **File** > **Save** > **Blade** from the main menu opens the **Save Blade** dialog box where you can type a file name for your blade file. For details, see *Save Blade As Command* (p. 16).

If you have already saved a blade file during the current ANSYS TurboGrid session, selecting **File** > **Save** > **Blade** from the main menu overwrites that file. To save a blade to a different file name, select **File** > **Save** > **Blade As...** from the main menu. For details, see *Save Blade As Command* (p. 16).

2.12.2. Save Blade As Command

Saving a blade file produces a text file defining the current blade, which can then be used to define the blade for future meshes. If the same changes to the blade would otherwise be repeated, this eliminates wasted time. To save a blade to a new file, select **File** > **Save** > **Blade As** from the main menu. The **Save Blade** dialog box is displayed.

Blade files are saved with a `.crv` file extension if the file type is `All Blade Files (*.crv *.curve)`.

2.12.3. Save Periodic/Interface Surfaces Command

Saving the periodic/interface surfaces produces a text file describing the location and shape of the interfaces between adjacent blades. You can optionally set a base name for constructing the data file name. You can optionally change the units in which to save the data.

2.12.4. Save Inlet Command

Saving the inlet produces a text file describing the location and shape of the inlet region. This includes any added inlet points.

Inlet files are saved with a `.crv` file extension if the file type is `All Inlet Files (*.crv *.curve)`.

2.12.5. Save Inlet As Command

Save an inlet (see above) to an alternative filename.

2.12.6. Save Outlet Command

Saving the outlet produces a text file describing the location and shape of the outlet region. This includes any added inlet points.

Outlet files are saved with a `.crv` file extension if the file type is `All Outlet Files (*.crv *.curve)`.

2.12.7. Save Outlet As Command

Save an outlet (see above) to an alternative filename.

2.12.8. Save Topology Command

If you have not saved a topology file during the current ANSYS TurboGrid session, selecting **File** > **Save** > **Save Topology** from the main menu opens the Save Topology dialog box where you can type a file name for the topology file. For details, see *Save Topology As Command* (p. 17).

If you have already saved a topology file during the current ANSYS TurboGrid session, selecting **File** > **Save** > **Save Topology** from the main menu overwrites that file. To save a topology to a different file name, select **File** > **Save** > **Save Topology As** from the main menu. For details, see *Save Topology As Command* (p. 17).

2.12.9. Save Topology As Command

Saving a topology file produces a text file defining the current topology. This file can then be used to define the topology for other geometries, which may be necessary for certain analyses. To save a topology to a new file, select **File** > **Save** > **Save Topology As** from the main menu or click *Save Topology As* ![icon]. The **Save Topology** window is displayed.

Topology files are saved with a `.tgt` file extension automatically.

2.13. Save Mesh Command

If you have not saved a mesh during the current ANSYS TurboGrid session, selecting **File** > **Save Mesh** from the main menu opens the **Save Mesh** dialog box where you can type a file name for the mesh file(s). For details, see *Save Mesh As Command* (p. 18).

If you have already saved a mesh during the current ANSYS TurboGrid session, selecting **File** > **Save Mesh** from the main menu overwrites the file(s). To save a mesh to a different name, select **File** > **Save Mesh As** from the main menu. For details, see *Save Mesh As Command* (p. 18).

2.14. Save Mesh As Command

Saving a mesh file produces input files for ANSYS CFX or CFX-TASCflow, or produces a CGNS file. To save a mesh to a new file, select **File** > **Save** > **Save Mesh As** from the main menu or click *Save Mesh As* 🏢. The **Save Mesh** dialog box is displayed.

2.14.1. Save Mesh Dialog Box
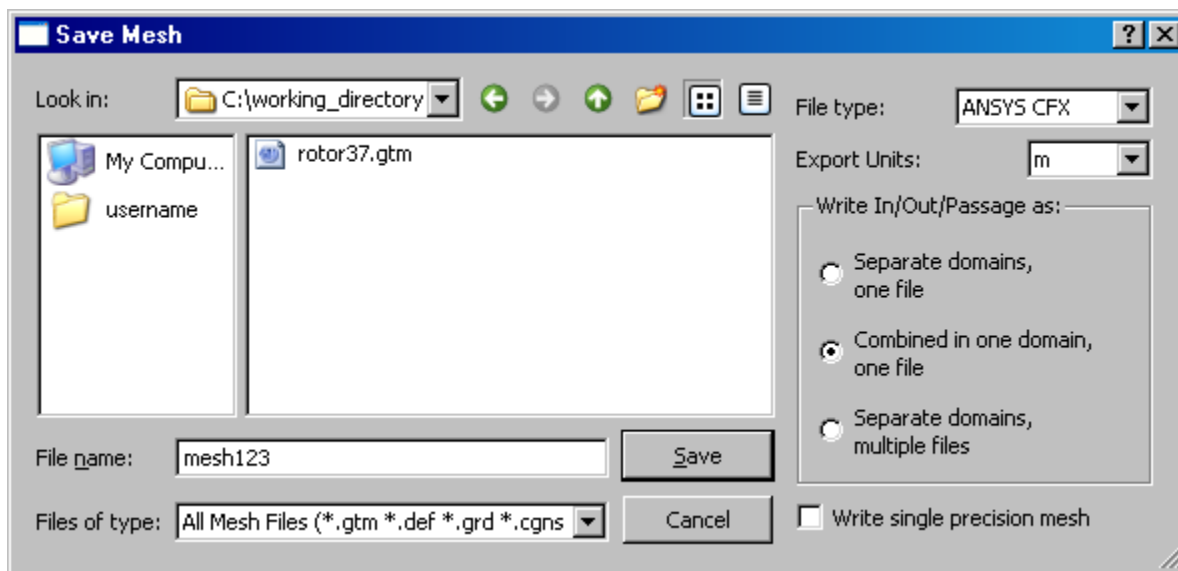
Set **Look in** to the directory in which you want to create the mesh file.

2.14.1.1. File Type

If **File type** is set to `ANSYS CFX`, a file of type `.gtm` is saved by default (that is, if no extension is specified). You can also save a file of type `.def` by adding the `.def` extension to the specified filename. Both `.gtm` and `.def` files contain regions that can be used in CFX-Pre to set up a CFD problem.

If **File type** is set to `CFX-TASCflow`, four types of file are saved: `.grd`, `.bcf`, `.gci` and `.lun`. A `.grd` file contains a number of regions that can be used in CFX-TASCflow to set up the CFD problem. If you specify the filename `mycustomname` or `mycustomname.grd`, ANSYS TurboGrid saves files named `mycustomname.grd`, `mycustomname.bcf`, `mycustomname.gci`, and `mycustomname.lun`. If you specify a filename of `grd`, ANSYS TurboGrid saves files named `grd`, `bcf`, `gci`, and `name.lun`. If **File type** is switched to `CFX-TASCflow` when the specified file name is blank, ANSYS TurboGrid sets the file name to `grd`.

If **File type** is set to `CGNS`, a `.cgns` file is saved. The `.cgns` file can be used by, for example, CFX-Pre, CFD-Post, and FLUENT.

If you change **File type**, any existing file extension (for example, `.def` or `.grd`) is changed automatically in the specified file name.

2.14.1.2. Export Units

Set **Export Units** to the length unit for the exported mesh.

2.14.1.3. Write In/Out/Passage As

If **File type** is set to `ANSYS CFX`, the following options are available:

- **Separate domains, one file** The inlet and outlet blocks remain separate from the passage block. Three separate assemblies appear in CFX-Pre. This choice is ideal when you want to place the inlet and outlet blocks in a different frame of reference from the passage.

- **Combined in one domain, one file**

  The inlet and outlet blocks are combined with (merged with) the passage block. One combined assembly appears in CFX-Pre. This choice is ideal when you want to keep the inlet and outlet blocks in the same frame of reference as the passage.

- **Separate domains, multiple files**

  The inlet and outlet blocks are combined with (merged with) the passage block. One combined assembly appears in CFX-Pre. This choice is ideal when you want to keep the inlet and outlet blocks in the same frame of reference as the passage.

If **File type** is set to `CFX-TASCflow`, three sets of files are saved using the same naming convention as for `CFX`. This means that there will be a `FILENAME_Inlet`, `FILENAME_Outlet`, and a `FILENAME` all of which will each have a file with a `.grd`, `.bcf`, `.gci` and `.lun` extension.

2.14.1.4. Write Single Precision Mesh

If **File type** is set to `ANSYS CFX`, you can select the **Write single precision mesh** check box to cause a single-precision mesh file to be written instead of a double-precision file. The default is double-precision. There is little benefit to using single-precision other than to reduce the size of the mesh file.

2.14.1.5. Generate Periodic and GGI Interfaces

This option is available if **File type** is set to `CFX-TASCflow`. This option causes the export to carry out the following steps:

- Create a `.gci` file for each mesh which defines macros for generating periodic interfaces in CFX-TASCbob3D.

- Run CFX-TASCbob3D in batch (if CFX-TASCflow is installed with a valid license), execute the required macros to generate the interfaces between blades (passage and periodic interfaces), and update the `.grd` and `.bcf` files as required.

2.14.2. Region Naming

The regions saved to the `.gtm`, `.def`, and `.grd` files are BLADE, PER1, PER2, HUB, SHROUD, INFLOW and OUTFLOW.

If the mesh has an inlet or outlet 3D region, additional regions will be created in the files for these domains with the suffix **Inlet** or **Outlet**. For example, the regions for a case with an inlet 3D region are PER1 Inlet, PER2 Inlet, HUB Inlet, SHROUD Inlet, INFLOW and OUTFLOW Inlet. The same applies for an

outlet 3D region and for the Passage Region. For the parts specific to a region (BLADE for Passage, IN-FLOW for Inlet, and OUTFLOW for Outlet), no suffix is added.

2.15. Export Geometry Command

The geometry in ANSYS TurboGrid can be exported to a tetin file format which can then be read by ICEM CFD products. To export a geometry to a tetin file, select **File** > **Export Geometry** from the main menu. The **Export Geometry** dialog box is displayed.

Geometry files are saved with a `.tin` file extension automatically.

2.16. Save Picture Command

To save the current contents of the viewer window to a file, select **File** > **Save Picture** from the main menu or click *Save Picture* 📷 . The **Save Picture** dialog box is displayed.

2.16.1. Save Picture Dialog Box
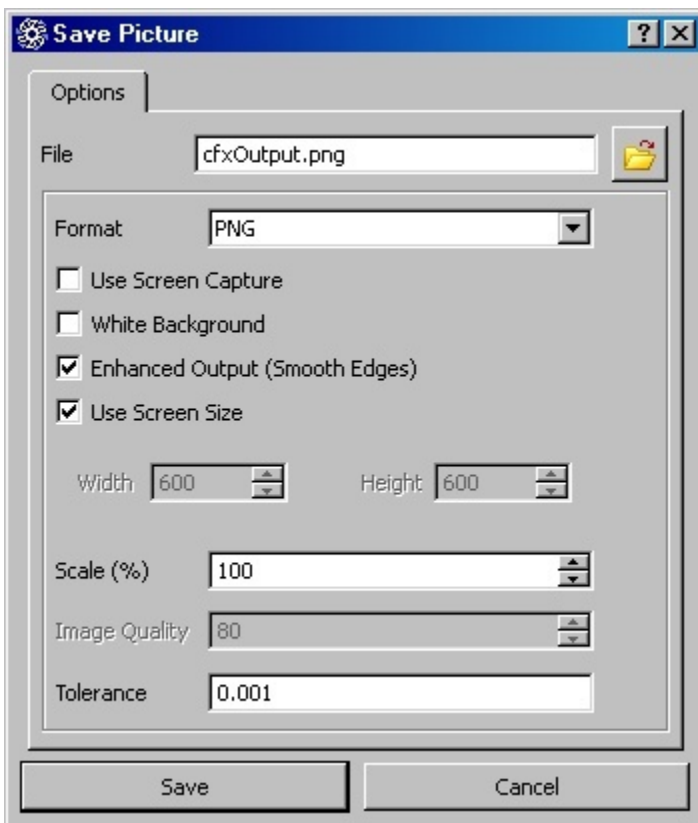
Type a file name and path into the **File** box, or click *Browse* 📁 to open a dialog box that helps you to select a directory and file name.

Files are always saved with the file extension corresponding to the selected graphics format.

2.16.1.1. Format

Click ▼ next to the **Format** box to see the available file formats.

- `Portable Network Graphics` (*.png) is a file format intended to replace the GIF format. It was designed for use on the WorldWide Web and retains many of the features of GIF with new features added.

- CFD Viewer State (3D) (*.cvf) is a format that can be read by the stand-alone CFD viewer.

- `JPEG` (*.jpg) is a compressed file format developed for compressing raw digital information. File sizes are small but it is not recommended for line drawings.

- `Bitmap` (*.bmp) files are usually large and do not adjust well to resizing or editing. They do retain all of the quality of the original image and can be easily converted to other formats.

- `Portable Pixel Map` (*.ppm) is similar to the bitmap format.

- PostScript (*.ps) and Encapsulated PS (*.eps) are generally recommended for output to a printer or line drawings. However, labels[1], the ruler [2] and transparency[3] will cause the PS/EPS to output as a very large bitmap file, in which case a PNG file would be a more efficient alternative. Note that the ANSYS logo and the axis do not cause the PS/EPS output to become a bitmap.

- `Virtual Reality Modeling Language` (*.wrl) is used to present interactive three-dimensional views and can be delivered across the World Wide Web.

2.16.1.2. Use Screen Capture Check Box

When **Use Screen Capture** is selected, a screen capture of the viewer is saved to the output file.

Note

Face culling only affects printouts made using the screen capture method.

2.16.1.3. White Background Check Box

You can save the current image with a white background by selecting **White Background**.

When the White Background check box is selected, certain white objects are colored black and certain black objects are colored white in the image file (except VRML). Objects that are not affected can usually be manually colored by editing them.

2.16.1.4. Enhanced Output (Smooth Edges) Check Box

When **Enhanced Output (Smooth Edges)** is selected, the image is processed by antialiasing.

2.16.1.5. Use Screen Size Check Box

When **Use Screen Size** is selected, the output has the same width and height, measured in pixels, as shown in the viewer. You can clear the check box to specify the width and height manually.

2.16.1.6. Width and Height Settings

You can specify the width and height of the image in pixels by entering values for **Width** and **Height**. In order to use these settings, the **Use Screen Size** check box must be cleared.

---

[1]You can hide labels from **Edit** > **Options** > **CFX-Pre** > **Labels and Markers** > **Labels**.

[2]You can hide the ruler from **Edit** > **Options** > **CFX-Pre** > **Graphics Style** > **Visibility** > **Ruler Visibility**.

[3]You can control transparency from **Edit** > **Render** > **Transparency**.

2.16.1.7. Scale (%)

Scale (%) is used to scale the size of bitmap images to a fraction (in percent) of the current viewer window size. This option is disabled with the clearing of the **Use Screen Size** check box.

2.16.1.8. Image Quality

Image Quality is only available for the JPEG format. Type in the value for the image quality, increase or decrease the value by 1 by clicking ▲ or ▼ respectively, or use the embedded slider (which has a maximum value of 100, which is the highest image quality, and a minimum value of 1, which is the lowest image quality).

2.16.1.9. Tolerance

Tolerance is a non-dimensional value used in face sorting when generating pictures. Larger values result in faster generation times, but may cause defects in the resulting output.

2.16.1.10. Save Button

Click **Save** to save the current viewer contents to an image file.

2.17. Recent State Files Submenu

ANSYS TurboGrid saves the file paths of the last several state files opened. To re-open a recently used state file, select **File** > **Recent State Files** from the main menu and then select the file from the **Recent State Files** submenu.

2.18. Recent Session Files Submenu

ANSYS TurboGrid saves the file paths of the last several session files opened. To re-open a recently used session file, select **File** > **Recent Session Files** from the main menu and then select the file from the **Recent Session Files** submenu.

2.19. Quit Command

To exit from ANSYS TurboGrid select **Quit** from the file menu. Objects created during the ANSYS TurboGrid session are not automatically saved. If you want to save the objects before quitting, create a state file. For details, see *Save State Command (Export State Command)* (p. 15).

Chapter 3: Edit Menu

The following sections describe the commands available in the **Edit** menu:

3.1. Undo and Redo Commands

ANSYS TurboGrid includes an infinite Undo feature, limited only by the available memory on the machine and a few other restrictions described below. Select **Edit** > **Undo** from the main menu or click *Undo* ↰ on the toolbar to return to the state immediately prior to when the last **Apply** action was executed.

Select **Edit** > **Redo** from the main menu or click *Redo* ↱ on the toolbar to reapply changes that were undone. A **Redo** command must follow an undo or Redo command.

The undo function has a few limitations:

- When a mesh is created, the undo stack is cleared, meaning that the mesh creation process itself, and all commands before it, cannot be undone using the **Undo** command.

- While creating a session file, the undo/redo commands are not available.

- Any action which does not affect the state cannot be undone. For example, creating a mesh cannot be undone, nor can saving a topology file since neither of these actions changes the state.

- The undo function cannot return to the initial state of a default object. For example, after defining the Hub object for the first time, you cannot click *Undo* to return to the undefined state of the Hub object.

- Undo also reverses geometry manipulation when the named view icons located at the top of the Viewer window have been used. Rotation, zoom and translation actions performed using the mouse are not affected by selecting **Edit** > **Undo** from the main menu or by clicking on the ↰ icon on the toolbar. You can, however, undo some viewer manipulations. For details, see *Viewer Hotkeys* (p. 53).

The redo feature is used to reverse an undo action. Selecting **Edit** > **Redo** from the main menu or clicking on the ↱ icon on the toolbar can be done repeatedly to reverse as many undo actions as have been applied.

3.2. Options Command

Select **Edit** > **Options** from the main menu to set various viewer and appearance options in ANSYS TurboGrid.

3.2.1. TurboGrid Options

The **Enable Beta Features** check box will make the beta features of ANSYS TurboGrid available.

3.2.1.1. Viewer

Highlight Type controls how an object is highlighted in the viewer window while in picking mode when highlighting is on. For details, see *Viewer Toolbar* (p. 52).

- If `Bounding Box` is selected, the object is always highlighted with a red box surrounding the Object.

- If `Wireframe` is selected, the object is traced with a red line if the object contains surfaces.

3.2.1.1.1. Background

The following background options are available:

- `Color`: A constant color can be chosen.

- `Image`: One of a list of `Predefined` images or a `Custom` image can be selected. When setting a custom image, you must choose an image file and a type of mapping. The image types that are supported are: `bmp`, `jpg`, `png`, `ppm`. Mapping options are `Flat` and `Spherical`.

3.2.1.1.2. Text Color and Edge Color

In the stand-alone version, set text color and edge color as appropriate.

3.2.1.1.3. Axis Visibility

If the **Axis Visibility** check box is selected, the axis appears in the lower left corner of the viewer window. The axis is useful for reference when the geometry is rotated. The axis labels change when the viewer coordinates are transformed.

3.2.1.1.4. Ruler Visibility

If the **Ruler Visibility** check box is selected, a ruler appears in the viewer to show the length scale.

3.2.1.1.5. Stereo

See *Stereo Viewer* (p. 56).

3.2.2. Common Options

ANSYS TurboGrid includes an auto-save function which backs up work at set time intervals by saving a state file. Select the frequency of the auto save by picking a value from the drop-down list. **Auto Save** can be disabled by selecting `Never` from the list.

To change the directory in which auto-saved state files are saved, type a file name and path into the **Temporary Dir** box, or click on the browse icon beside the **Temporary Dir** box to open the **Temporary Directory** dialog box.

3.2.2.1. Appearance

Click on `Appearance` in the options box to control the appearance of the GUI.
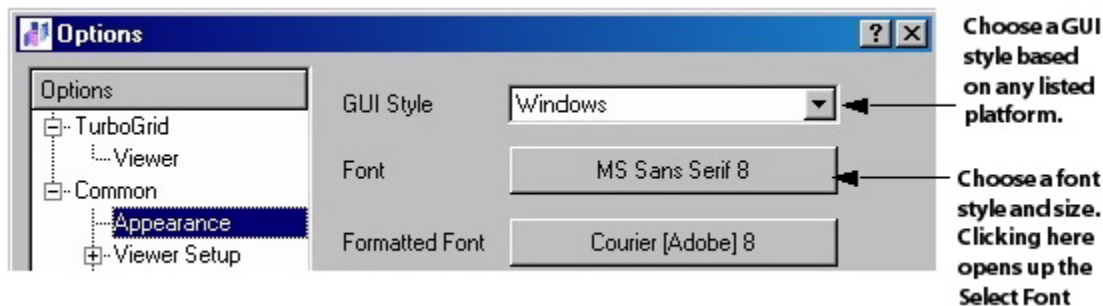
ANSYS TurboGrid sets the GUI Style to that of the machine's platform, by default. For example, on Windows the GUI has a Windows look to it. If you prefer a Linux appearance to the GUI, then select Motif from the drop-down list. Any of the following appearances can be used on any platform:

- `Windows`
- `Motif`
- `CDE`
- `Plastique`
- `Cleanlooks`

The font used within the GUI can be changed by clicking on the font button to open the **Select Font** window.

3.2.2.2. Viewer Setup: General

Viewer options under the `Common` branch are `Double Buffering` and `Unlimited Zoom` toggles.

Note

Using Unlimited Zoom will allow the exceeding of the depth buffer accuracy, resulting in rendering artifacts.

3.2.2.2.1. Double Buffering

`Double Buffering` is a feature supported by most OpenGL implementations. It provides two complete color buffers that swap between each other to animate graphics smoothly. If your implementation of OpenGL does not support double buffering, you can clear this check box.

3.2.2.2.2. Unlimited Zoom

By default, zoom is restricted to prevent graphics problems related to depth sorting. Selecting `Unlimited Zoom` allows an unrestricted zoom.

3.2.2.3. Viewer Setup: Mouse Mapping

The mouse mapping options enable you to assign viewer actions to mouse actions and keyboard/mouse combined actions.

A description of each action follows.

- Rotate: rotate the view about the screen X and Y axes.

- Object Zoom: drag the mouse up to zoom out and down to zoom in.

- Camera Zoom: drag the mouse up to zoom in and down to zoom out.

- Translate: drag the mouse to translate the view in the plane of the screen.

- Zoom Box: drag a rectangle around the area of interest. The selected area will fill the viewer when the mouse button is released.

- Zoom In: click the mouse button to zoom in step-by-step centered on the location of the mouse pointer.

- Zoom Out: click the mouse button to zoom out step-by-step centered on the location of the mouse pointer.

- Rotate Z: drag the mouse up to rotate the view clockwise about the screen Z axis, and down to rotate the view counterclockwise.

- Set Pivot Point: click on an object to set the point about which the Rotate and Rotate Z actions pivot.

- Move Light: drag to move the angle of the virtual light source in the viewer. Drag the mouse left or right to move the horizontal lighting source and up or down to move the vertical lighting source. The lighting angle holds two angular values, each between 0° and 180°.

3.2.2.4. Units

Click on `Units` in the options box to control the units which are presented in the GUI for each Quantity Type.

Select a pre-defined Units System such as `SI`, `English Engineering` or `British Technical`. The predefined Units for any quantity types in these units systems cannot be changed. Select the **Custom** units system to specify any valid units for each quantity type. For example, you may want to display **Length** in `mm` (an SI unit), but **Angle** in `radians` (not an SI unit). Click on **More Quantity Types** to display the **Custom Quantity Types** form and set custom units for more quantity types.

The units set on this panel define the units used in the GUI for all quantity types. The units you select for a quantity type appear wherever that quantity type is used. For example, if you choose `mm` as the unit of **Length** and create a plot in ANSYS TurboGrid colored by **Length**, you must specify a user specified length range in units of mm. If you created a Legend for the plot, the values on the Legend would be in units of mm.

If the **Always convert units to Preferred Units** check box is selected, all quantities entered in the GUI are converted into the preferred units.

Note

Setting units in the variable editor will override the actual setting for that quantity type.

3.2.2.5. Advanced

The `cmd timeout` value is an advanced feature. It sets the delay between a mouse control action and its execution.

Chapter 4: Session Menu

The following sections describe the commands available in the **Session** menu:

4.1. Introduction

Session files contain a record of the commands issued during an ANSYS TurboGrid session. Actions that cause commands to be written to a session file include:

- Creation of new objects and changes to existing objects committed by clicking **Apply** in the object editor.

- Creation of a mesh.

- Commands issued in the **Command Editor** dialog box.

- Calculations performed in the built-in calculator.

- Viewer manipulation performed using the icons located at the top of the viewer window. (Viewer manipulation performed using the mouse and keyboard are not recorded to a session file.)

- Creation of new cameras and selecting a camera view.

- All actions available from the **File** menu.

- Creation of expressions and user variables.

- Creation of chart lines and viewing charts.

Session files can be used to run ANSYS TurboGrid in batch mode. For details, see Batch Mode in the *TurboGrid Reference Guide*.

Note

Since the session file is a text file of CCL commands, you can write your own session files using a text editor.

Note

Do not end your session file with the `quit` command.

4.2. Play Session Command

A previously recorded session file can be played in ANSYS TurboGrid. To play a session file, select **Session** > **Play Session** from the main menu or click *Play Session* 📁 on the toolbar. The **Play Session File** dialog box is displayed.

The commands listed in the selected session file are executed. Existing objects with the same name as objects defined in the session file are replaced by those in the session file.

For any file specified in a state or session file, if the file cannot be found ANSYS TurboGrid will automatically search the state or session file directory for a file of the same name. If this procedure fails, the current working directory will be searched. As a result, state and session files will not have to be edited to change the path when state, session and curve files are moved from one directory to another.

Note

A session file cannot be played if it contains the `Undo` command. To run a session file which contains the `Undo` command, edit the session file first to remove the command.

Note

Since the `TOPOLOGY SET` object is now suspended by default, session and state files from version 11 or earlier may not play or load correctly. To support older session and state files, the **Play Session File** and **Load State** dialog boxes have an option named **Unsuspend TOPOLOGY SET before loading**. Selecting this option causes the `TOPOLOGY SET` object to be unsuspended before playing/loading a session/state file. When starting ANSYS TurboGrid from the command line, adding the command line parameter `-u` causes the `TOPOLOGY SET` object to be initially unsuspended.

Note

Workbench journal files for TurboGrid and TurboGrid session files from 12.0/12.1 may not work with 14.0. If the recorded file does not explicitly set the topology type, the following CCL block needs to be added:

```
TOPOLOGY SET:
ATM Topology Optimizer = off
END
```

For Workbench journal files, this CCL block should be added immediately before the following line occurs:

```
> um mode=normal, object=/TOPOLOGY SET
```

The line above appears only in journal files that were recorded while the `Topology Set` object was processed (unsuspended).

For session files, this CCL block should be added to the beginning of the session file.

For information about using ANSYS Workbench journaling and scripting with ANSYS TurboGrid, see Using ANSYS Workbench Journaling and Scripting with TurboSystem in the *TurboSystem User Guide*.

4.3. New Session Command

To create a new Session file, select **Session** > **New Session** from the main menu or click *New Session* on the toolbar. The **Set Session File** dialog box is displayed.

Upon clicking **Save** in the dialog window, that file becomes the current session file. Commands are not written to the file until recording begins. For details, see *Start Recording Command* (p. 29).

Session files should be saved with a `.tse` file extension. The extension is added to a file name if `TG Session Files (*.tse)` is selected as the file type.

If you create more than one session file during a ANSYS TurboGrid session, the most recently created file is by default the current session file. To set a different file to be the current session file, select an existing file from the Set Session File dialog box and then click **Save**. The following message then appears:
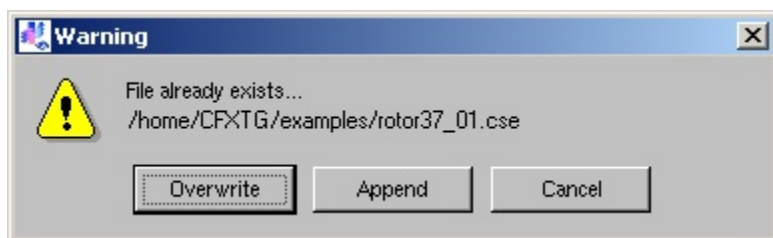
Click **Overwrite** to delete the existing session file and create a new file in its place. Click **Append** to add selected commands to the end of the existing session file when recording begins.

4.4. Start Recording Command

To start recording a session file, select **Session** > **Start Recording** from the main menu or click *Start Recording* on the toolbar. This activates recording of CCL commands issued to the current session file. A session file must be set before recording can begin. For details, see *New Session Command* (p. 29).

While recording a session file, the *Undo* and *Redo* icons are disabled.

4.5. Stop Recording Command

To stop recording a session file, select **Session** > **Stop Recording** from the main menu or click *Stop Recording* on the toolbar. This terminates recording of CCL commands issued to the current session file. Start and stop recording to a session file as many times as necessary.

Chapter 5: Insert Menu

The following sections describe the commands available in the **Insert** menu.

Note

Some of the commands in the **Insert** menu are not available until you create a mesh.

5.1. Introduction

The **Insert** menu contains a list of objects that can be used as tools to analyze the mesh and geometry (although they are not necessary for mesh generation).

When you select any of the objects from the **Insert** menu, a dialog box appears in which you can either accept the default name or type a new one for the object (in this case a plane). The name should be different from any current object of the same type to avoid overwriting the existing object. ANSYS TurboGrid does not let you create objects with the same name but different types.
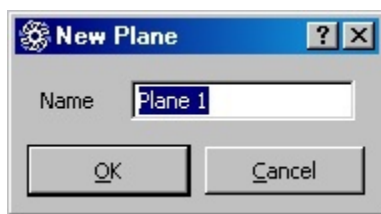
Click **OK** or press **Enter** to open the relevant object editor. The object does not exist in the database until you click **Apply** in the object editor. For information on how to create objects from the command line see Object Creation and Deletion in the *TurboGrid Reference Guide*

5.2. Mesh Command

The flow path through the rotating machine is divided into small but discrete volumes. These volumes are hexahedral elements, which have six sides and eight corners. There are nodes placed on each corner and the assembly of these nodes forms the mesh. The mesh fills the entire flow path and is used by ANSYS CFX solvers. The quality of the solution depends partly on the quality of the Mesh. To create a mesh, select **Insert** > **Mesh** from the main menu, click *Insert Mesh* 🔲 on the toolbar, or right-click anywhere in the viewer or tree view and select **Insert Mesh** from the shortcut menu.

ANSYS TurboGrid creates the mesh using the current state of the `Topology Set` and `Mesh Data` objects. The time it takes to create the mesh depends on its size and the number of smoothing iterations chosen. Displayed in the status bar at the bottom left corner of the application window is an estimate

of the total number of nodes and the total number of elements in the mesh. After the mesh is created, the color and rendering settings in the viewer window can be controlled for each individual mesh surface.

If changes are made to any of the `Geometry`, `Topology Set`, `Mesh Data`, or `Layers` objects, the mesh must be recreated in order for these changes to be included in the mesh.

5.3. User Defined Submenu

The **User Defined** submenu lists commands that create new objects.

Each command invokes a dialog box to prompt you for the name of the object to create. The name of a new object must be different from any existing object. When you click **OK** or press **Enter** to accept the name of the object, the relevant object editor opens, but the object is not created until you click **Apply** on the object editor.

5.3.1. Point Command

A point can exist anywhere within or outside the domain. To create a new point, select **Insert** > **User Defined** > **Point** from the main menu.

5.3.1.1. Point: Geometry Tab

5.3.1.1.1. Point Definition

The available options for defining a point are described next:

- `XYZ`

  Set the point coordinates. To set the coordinates, first click one of the point coordinate boxes. The point coordinates will be displayed with a yellow background, and the viewer will switch to picking mode. You can then pick a point directly by clicking a visible object in the viewer. The point can lie outside of the domain.

  You can alternatively set the coordinates one-at-a-time by typing in the coordinates and/or using the embedded slider that appears beneath the coordinate boxes.

- `Node Number`

  Create a point at a nodal location. Use the **Domains** drop-down list to select the domain(s) in which the locator exists. After choosing the domain(s) in which to select the node, enter the node number. Type in the node number or set it using the embedded sliders. When more than one domain is selected, a point is created for the specified node number in each domain (if the node number exists). If the node number does not exist in one domain but exists in another, select only the domain in which the node exists or an error message is displayed.

- `Variable Minimum` and `Variable Maximum`

  Create the point where a variable is at its maximum or minimum value on any named locator. The **Domains** drop-down list is used to select the domain(s) in which the locator exists. After choosing the domain(s), select the locator name and the variable of interest. When more than one domain is selected, a point is created for the maximum/minimum value of the variable within each domain.

5.3.1.1.2. Symbol Definition

The **Symbol Size** must be between `0` and `10` (`10` being a similar scale to the geometry). Type in the value or set it using the embedded slider.

Set **Symbol** to one of the available symbols.

Picking Mode can be used to select and/or translate points in the viewer. For details, see *Viewer Toolbar* (p. 52).

Note

You cannot move points that have been defined using `Node Number`, `Variable Minimum`, or `Variable Maximum`.

5.3.1.2. Point: Color Tab

See *Color Tab* (p. 5) for details.

5.3.1.3. Point: Render Tab

See *Render Tab* (p. 6) for details.

5.3.2. Line Command

A line locator can exist between two points anywhere within or outside the domain. To create a new line, select **Insert** > **User Defined** > **Line** from the main menu.

5.3.2.1. Line: Geometry Tab

5.3.2.1.1. Domains

Select the domain(s) in which the line will exist.

5.3.2.1.2. Line Definition

Lines are created by defining two points. To set the coordinates, first click one of the point coordinate boxes. The point coordinates will be displayed with a yellow background, and the viewer will switch to picking mode. You can then pick a point directly by clicking a visible object in the viewer. The point can lie outside of the domain.

You can alternatively set the coordinates one-at-a-time by typing in the coordinates and/or using the embedded slider that appears beneath the coordinate boxes.

5.3.2.1.3. Line Type

Set the **Line Type** to either **Cut** or **Sample**.

- **Cut** extends the line in both directions until it reaches the edge of the domain. Points on the line correspond to points where the line intersects a mesh element face. As a result, the number of points on the line is indirectly proportional to the mesh spacing.

- **Sample** creates the line between the two specified points. The sample line is a set of evenly-spaced sampling points that are independent of the mesh spacing. The number of points along the line corresponds to the value in the **Samples** box.

Picking Mode can be used to select and/or translate lines in the viewer. For details, see *Viewer Toolbar* (p. 52).

5.3.2.2. Line: Color Tab

See *Color Tab* (p. 5) for details.

5.3.2.3. Line: Render Tab

See *Render Tab* (p. 6) for details.

5.3.3. Plane Command

A plane locator is a two-dimensional area that exists only within the boundaries of the geometry. To create a new plane, select **Insert** > **User Defined** > **Plane** from the main menu.

5.3.3.1. Plane: Geometry Tab

5.3.3.1.1. Domains

Select the domain(s) in which the plane will exist.

5.3.3.1.2. Plane Definition

The available methods for defining a plane are described next:

- YZ Plane

  Create a plane normal to the X axis and at a specific X value. Type in a value for **X**, set it using the embedded slider, or click in the box beside **X**, then click *Enter Expression* 📊 to the right of the **X** setting and enter its value as an expression. For details, see *Expressions Command* (p. 61).

- ZX Plane

  Create a plane normal to the Y axis and at a specific Y value. Type in a value for **Y**, set it using the embedded slider, or click in the box beside **Y**, then click *Enter Expression* 📊 to the right of the **Y** setting and enter its value as an expression. For details, see *Expressions Command* (p. 61).

- XY Plane

  Create a plane normal to the Z axis and at a specific Z value. Type in a value for **Z**, set it using the embedded slider, or click in the box beside **Z**, then click *Enter Expression* 📊 to the right of the **Z** setting and enter its value as an expression. For details, see *Expressions Command* (p. 61).

- Point and Normal

  Create a plane using a single point on the plane and a vector normal to the plane.

  To set the point coordinates, first click one of the point coordinate boxes. The point coordinates will be displayed with a yellow background and the viewer will switch to picking mode. You can

then pick a point directly by clicking a visible object in the viewer. The point can lie outside of the domain.

You can alternatively set the coordinates one-at-a-time by typing in the coordinates and/or using the embedded slider that appears beneath the coordinate boxes.

When picking a point from the viewer for the normal vector, the vector is taken from the origin to the point picked.

- `Three Points`

  Create a plane using three points. To set the point coordinates, first click one of the point coordinate boxes. The point coordinates will be displayed with a yellow background, and the viewer will switch to picking mode. You can then pick a point directly by clicking a visible object in the viewer. The point can lie outside of the domain.

  You can alternatively set the coordinates one-at-a-time by typing in the coordinates and/or using the embedded slider that appears beneath the coordinate boxes.

  The normal vector to the plane is calculated using the right-hand rule. The first vector is from Point 1 to Point 2 and the second is from Point 1 to Point 3 as shown in the diagram below.
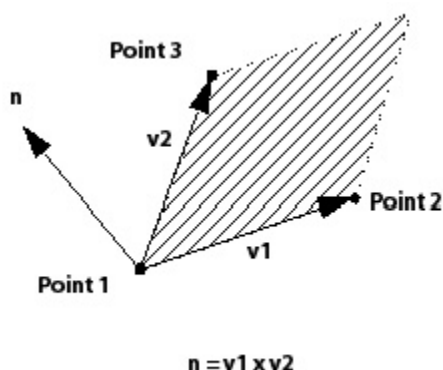
5.3.3.1.3. Plane Bounds

The available types of bounds are described next:

- When `None` is selected the plane cuts through a complete cross-section of each domain specified in the **Domains** list. The plane is only bounded by the limits of the domains.

- The `Circular` option defines the bounds of a plane as a circle centered at the point used in the **Plane Definition**. Enter the value of the radius of the circle or click in the box beside **Radius**, then click *Enter Expression* 🔲 to the right of the **Radius** setting to enter an expression for the radius of the circle. The plane is undefined in areas where the circle extends outside of the domains specified in the **Domains** list.

- For the `Rectangular` option, the plane bounds are defined by a rectangle centered about the point selected in the **Plane Definition** with lengths in the x and y-directions of **X Size** and **Y Size**, respectively (You might have to specify lengths in other directions, depending on the **Plane Definition**.) . The size is determined with reference to the plane center (that is, the plane is resized around its center). The **X Angle** value rotates the plane counter-clockwise about its normal by the specified number of degrees. The plane is undefined in areas where the rectangle extends outside of the domains specified in the **Domains** list.

Both the circular and rectangular options have an **Invert Plane Bound** check box. If this check box is selected, the area defined by the rectangle or circle is used as a cut-out area from a slice plane that is bounded only by the domain(s). The area inside the bounds of the rectangle or circle does not form part of the plane, but everything on the slice plane outside of these bounds is included.

5.3.3.1.4. Plane Type

Set **Plane Type** to either **Slice** or **Sample**.

Slice extends the plane in all directions until it reaches the edge of the domain. Points on the plane correspond to points where the plane intersects an edge of the mesh. As a result, the number of points in a slice plane is indirectly proportional to the mesh spacing.

Sample creates the plane with either circular or rectangular bounds, depending on the plane bounds selected. For the `Circular` option, the density of points on the plane corresponds to the radius of the plane specified in the **Plane Bounds** frame, and the values for the **Radial** and **Circumferential** settings in the **Plane Type** frame. For rectangular bounds, the density of points on the plane corresponds to the size of the bounds for the plane in each of the plane directions, and the sample values provided in the **Plane Type** frame. A sample plane is a set of evenly-spaced points that are independent of the mesh spacing.

Picking Mode can be used to select and/or translate planes in the viewer. For details, see *Viewer Toolbar* (p. 52).

5.3.3.2. Plane: Color Tab

See *Color Tab* (p. 5) for details.

5.3.3.3. Plane: Render Tab

See *Render Tab* (p. 6) for details.

5.3.4. Turbo Surface Command

A *turbo surface* is a surface that exists only within the boundaries of the geometry and is defined using variables specific to rotating machinery. To create a new turbo surface, select **Insert** > **User Defined** > **Turbo Surface** from the main menu.

5.3.4.1. Turbo Surface: Geometry Tab

5.3.4.1.1. Domains

Select the domain(s) in which the turbo surface will exist.

5.3.4.1.2. Turbo Surface Definition

ANSYS TurboGrid creates a turbo surface using a defined variable and its value. Set the domain to either `All Domains` or `DOMAIN:Passage`. Select a variable and set a value to define the location of the turbo surface (The variable is "K" by default, which is the mesh plane index that varies from hub to shroud for the H-Grid and J-Grid topologies.). You may enter the value directly, or, for the "K" variable, use the up/down arrows, or, for any other variable, click in the **Value** box, then click *Enter Expression* (to the right of the **Value** box) to enter an expression for the value, or use the slider.

5.3.4.2. Turbo Surface: Color Tab

See *Color Tab* (p. 5) for details.

5.3.4.3. Turbo Surface: Render Tab

See *Render Tab* (p. 6) for details.

5.3.5. Volume Command

A *volume* object is a collection of mesh elements; it can be created anywhere within the domain. To create a new volume, select **Insert** > **User Defined** > **Volume** from the main menu.

5.3.5.1. Volume: Geometry Tab

5.3.5.1.1. Domains

Select the domain(s) in which the volume will exist.

5.3.5.1.2. Volume Definition

The available methods for defining a volume are described next:

- Sphere

  Create a volume in the shape of a sphere by defining the center point and radius for the sphere.

  To set the coordinates of the center points, first click one of the point coordinate boxes. The point coordinates will be displayed with a yellow background, and the viewer will switch to picking mode. You can then pick a point directly by clicking a visible object in the viewer. The point can lie outside of the domain.

  You can alternatively set the coordinates one-at-a-time by typing in the coordinates and/or using the embedded slider that appears beneath the coordinate boxes.

  Type in the radius value or set it using the embedded slider. Ensure that the units are set correctly. When **Mode** is set to Intersection, the volume consists of all elements that intersect the sphere surface. When **Mode** is set to Below Intersection or Above Intersection, the volume consists of all elements that are "below" or "above" the sphere surface, respectively. If the **Inclusive** check box is selected, all elements that intersect the sphere surface are added to the volume.

- From Surface

  Create a volume using an existing surface. Set **Location** to one or more of the available surfaces for defining the volume (Use **Ctrl** to multi-select.). When **Mode** is set to Intersection, the volume consists of all elements that intersect the sphere surface. When **Mode** is set to Below Intersection or Above Intersection, the volume consists of all elements that are "below" or "above" the sphere surface, respectively. If the **Inclusive** check box is selected, all elements that intersect the sphere surface are added to the volume.

- Isovolume

  Create a volume based on the value(s) of a variable. Click ▾ next to the variable box to see the available variables for defining the volume. Type in the value(s), set the value(s) using the embedded

slider, or click in the box, click *Enter Expression* 🔲 to the right of the box, then enter the value(s) as an expression. Set the units for the variable(s) appropriately. When **Mode** is set to `At Value`, the volume consists of all elements having the defined variable value. When **Mode** is set to `Below Value` or `Above Value`, the volume consists of all elements that are "below" or "above" the defined variable value, respectively. When **Mode** is set to `Between Values`, the volume consists of all elements that are "between" the defined variable values. If the **Inclusive** check box is selected, the elements having the defined variable value are added to the volume.

5.3.5.1.3. Hybrid/Conservative

For volumes made with **Method** set to `Isovolume`, the option of using hybrid or conservative values is available. Unless you are post-processing CFD results using ANSYS TurboGrid, this option can be ignored. For details, see *Hybrid and Conservative Variable Values* (p. 65).

> **Note**
>
> Volumes are not displayed as perfect shapes (for example, a perfect sphere) because mesh elements are either included or excluded from the volume.

5.3.5.2. Volume: Color Tab

See *Color Tab* (p. 5) for details.

5.3.5.3. Volume: Render Tab

See *Render Tab* (p. 6) for details.

5.3.6. Isosurface Command

An *isosurface* object is a surface upon which a particular variable has a constant value, called the "level". To create a new Isosurface, select **Insert** > **User Defined** > **Isosurface** from the main menu.

5.3.6.1. Isosurface: Geometry Tab

5.3.6.1.1. Domains

Select the domain(s) in which the isosurface will exist.

5.3.6.1.2. Isosurface Definition

Set **Variable** to one of the available variables for defining the isosurface. Type in the variable value, set it using the embedded slider, or click in the box, click *Enter Expression* 🔲 (to the right of the box), and enter an expression. Set appropriate units for the variable. The isosurface connects all locations with the specified variable value.

5.3.6.1.3. Hybrid/Conservative

The option of using hybrid or conservative values is available. Unless you are post-processing CFD results using ANSYS TurboGrid, this option can be ignored. For details, see *Hybrid and Conservative Variable Values* (p. 65).

5.3.6.2. Isosurface: Color Tab

See *Color Tab* (p. 5) for details.

> **Note**
>
> You may color the isosurface using any variable or choose a constant color. You should not select the `Local` range option when coloring an isosurface with the variable used to define it. In this case the local range is zero by definition, so a plot would be colored according to the effects of round-off error.

5.3.6.3. Isosurface: Render Tab

See *Render Tab* (p. 6) for details.

5.3.7. Polyline Command

5.3.7.1. Polyline: Geometry Tab

A *polyline* object is a set of connected line segments that connect a series of points. To create a polyline, select **Insert** > **User Defined** > **Polyline** from the main menu.

Available methods of creating a polyline are:

*   `From File`

*   `Boundary Intersection`

The file format for a polyline is shown below:

```
[Name]
Polyline 1
[Data]
X [ m ], Y [ m ], Z [ m ], Area [ m^2 ], Density [ kg m^-3 ]
-1.04539007e-01, 1.68649014e-02, 5.99999987e-02, 0.00000000e+00, ...
-9.89871025e-02, 3.27597000e-02, 5.99999987e-02, 0.00000000e+00, ...
.
.
.
[Lines]
0, 1
1, 2
.
.
.
[Name]
Polyline 2
.
.
.
```

In this example, the two lines containing data are shown word-wrapped onto the next line. In the actual file, all data for a given point must be on a single line.

The name of each locator is listed under the **Name** section. Point coordinates and the corresponding variable values are stored in the **Data** section. Line connectivity data is listed in the **Lines** section; this data makes reference to the points that are defined in the **Data** section. For this purpose, point numbering in the **Data** section is consecutive, starting at zero.

Comments in the file are preceded by # (or ## for the CFX-5.6 polyline format) and can appear anywhere in the file.

Blank lines are ignored and can appear anywhere in the file (except between the line "[Data]" and first line of data following it).

5.3.7.2. Polyline: Color Tab

See *Color Tab* (p. 5) for details.

5.3.7.3. Polyline: Render Tab

See *Render Tab* (p. 6) for details.

5.3.8. Surface Command

To create a user surface, select **Insert** > **User Defined** > **Surface** from the main menu.

5.3.8.1. User Surface: Geometry Tab

5.3.8.1.1. Method

Choose one of the available methods for defining a user surface:

*   `From File`

    Select the `From File` alternative when you are not able to create the required surface using the Boundary Intersection method. This more versatile option reads data describing the surface from a file. The points may have *path variables* (variables that are only defined on the surface) associated with them. For details on the required format for a surface data file, see *Surface Data Format* (p. 40). Click the browse icon to open an **Import** dialog box and browse to the surface data file. You can alternatively type the path and file name into the **Input File** box.

*   `Boundary Intersection`

    `Boundary Intersection` forms a user surface where a given locator intersects a specified list of boundaries. The **Domains** setting specifies the domains in which this user surface exists. To select multiple boundaries for the **Boundary List**, click [...] and hold **Ctrl** as you select each boundary. Set **Location** to one of the existing graphic objects.

    The user surface consists of mesh element faces on the specified boundaries, for the mesh elements that are intersected by the locator. The user surface is usually narrow with a varying width. The fluctuation in width is more noticeable in a coarser mesh.

5.3.8.1.2. Surface Data Format

When the `From File` method is selected, an external file must exist that defines the surface. A set of surfaces can be defined in a simple text file with the following format.

```
## Comment line - optional.
## List of path variables
# <varName1>
# <varName2>
# ...
## List of point locations with path variable values
```

```
## Each line in the following list is numbered 0,1,2...
<X> <Y> <Z> <Var1Value> <Var2Value> ...
<X> <Y> <Z> <Var1Value> <Var2Value> ...
...
## Next line is a keyword that starts the definition of faces
# Faces
## List of 3 - 6 point numbers to define faces.
<Point0> <Point1> <Point2>
<Point1> <Point2> <Point3> <Point4> <Point5> <Point6>
...
```

Comments in the file are preceded by `##` and can appear anywhere in the file. A single `#` does *not* indicate a comment; words appearing after a single `#` are keywords such as `Faces`.

The start of the file should begin with a list of path variables (up to 256 characters, spaces allowed). These are variables that are only defined on the user surface. Ensure that the names of these variables do not conflict with the names of existing variables. There is no need to define any path variables (if you just want to define the location of a user surface), in which case the file begins with the point location values.

The point location list in `X Y Z` format follows the optional path variable list. You must also include a value for each path variable that you have defined at the start of the file (if any). Surfaces are defined by typing `# Faces` followed by lists of 3 (triangle) to 6 (hexagon) points to define each surface. Each surface is automatically closed by connecting the last point to the first point. The list of point locations are numbered 0,1,2....n-1 where n is the number of points in the list. When defining faces, use these numbers to reference the points in the point location list. The faces specification is *not* optional.

Blank lines are ignored and can appear anywhere in the file.

The following example defines one quadrilateral face with two path variables at each point of the face:

```
# Time
# MyVar
1 1 1 1.2 500
1 2 1 2.1 200
2 2 1 3.4 300
2 1 1 4.65 400
# Faces
0 1 2 3
```

5.3.8.2. User Surface: Color Tab

See *Color Tab* (p. 5) for details.

5.3.8.3. User Surface: Render Tab

See *Render Tab* (p. 6) for details.

5.3.8.4. Surface Groups

Surface groups are produced automatically when a mesh is generated. They are found in the `3D Mesh` branch of the object selector. Each surface group shows a surface of the mesh. The available surface groups vary according to the number of blades and types of leading and trailing edges. A representative list follows:

- `HUB, SHROUD`

- `[blade name] LOWBLADE,[blade name] HIGHBLADE`

If there is one blade in the blade set, then the surface groups `LOWBLADE` and `HIGHBLADE` will be available after generating a mesh. If there are two or more blades in the blade set, the surface group names start with the blade name. For example, if there are two blades named `Main` and `Splitter`, then surface groups `Main LOWBLADE`, `Main HIGHBLADE`, `Splitter LOWBLADE` and `Splitter HIGHBLADE` will be available after generating a mesh.

- `[blade name] BLADE LE`, `[blade name] BLADE TE`

  If there is one blade in the blade set, then the surface groups `BLADE TE` and `BLADE LE` will be available, as applicable, after generating a mesh. These surface groups are applicable only for cut-off or square leading/trailing edges. If there are two or more blades in the blade set, the surface group names start with the blade name. For example, if there are two blades named `Main` and `Splitter`, and the trailing edge of `Main` is cut-off, and the leading and trailing edges of `Splitter` are cut-off, then surface groups `Main BLADE TE`, `Splitter BLADE LE` and `Splitter BLADE TE` will be available after generating a mesh.

- `LOWPERIODIC`, `HIGHPERIODIC`

- `INLET`, `OUTLET`

"LOW" refers to low Theta value and "HIGH" refers to high Theta value.

A visibility check box next to each surface group enables you to control which are displayed.

Besides the visibility, you may also change the color and render properties for each surface group.

5.3.8.4.1. Surface Group: Definition Tab

You may view the **Domains** and **Locations** information on the **Definition** tab.

5.3.8.4.2. Surface Group: Color Tab

See *Color Tab* (p. 5) for details.

5.3.8.4.3. Surface Group: Render Tab

See *Render Tab* (p. 6) for details.

5.3.9. Contour Command

A *contour plot* is a series of lines linking points with equal values of a given variable. For example, contours of height exist on geographical maps and give an impression of gradient and land shape. To create a contour, select **Insert** > **User Defined** > **Contour** from the main menu.

5.3.9.1. Contour Plot: Definition Tab

5.3.9.1.1. Domains

Select the domain(s) in which the contour object will exist.

5.3.9.1.2. Locations

Select the locator(s) on which to plot the contours. To select multiple locators, click the ⌷ icon, hold down **Ctrl**, and select each locator.

5.3.9.1.3. Variable

Choose the plot variable for the contour plot.

5.3.9.1.4. Range

Set **Range** to one of the available methods for defining the range of the contour plot. This affects the variation of color used when plotting the contours in the viewer. The lowest values of a variable in the selected range are shown in blue in the viewer, the highest values are shown in red.

- `Global` uses the range of the variable over all domains (regardless of the domains selected on the **Geometry** tab) to determine the minimum and maximum values for the contours.

- `Local` uses the range of the variable over the selected locator(s) to determine the minimum and maximum values for the contours.

- When using `User Specified`, enter the minimum and maximum values for the contours. Type in the variable values, set them using the embedded slider or, by clicking *Enter Expression* ▣ to the right of the **Units** box, enter them as an expression.

- Using `Value List`, a list separated by commas, specify the actual values at which contours should be plotted. For example, if plotting minimum face angle, try a value list of `5, 10, 15, 20, 25` degrees. It should be noted that entering a value list overrides the number specified in the **# of Contours** box (see below).

5.3.9.1.5. Hybrid/Conservative

The option of using hybrid or conservative values is available. Unless you are post-processing CFD results using ANSYS TurboGrid, this option can be ignored. For details, see *Hybrid and Conservative Variable Values* (p. 65).

5.3.9.1.6. Number of Contours

Set the **# of Contours** to appear in the plot. This is the number of bands plus one. This number is overridden if the **Range** setting is set to `Value List`.

5.3.9.2. Contour Plot: Labels Tab

The **Show Numbers** check box determines whether numbers corresponding to the number of contours are displayed on the plot. To view the values of the plotted variable at each contour, create a legend of the contour plot. See *Legend Command* (p. 45) for more details. To change the size of the text that appears on the contour plot, type a new value into the **Text Height** box or use the embedded slider (which has a maximum value of 1 and a minimum value of 0). The **Text Height** number is a fraction of the viewer height. **Text Font** controls which font is used for the numbers that appear in the contour plot. To change the text color, set **Color Mode** to `User Specified` then choose a color by clicking in the **Text Color** box or by clicking the ⋯ icon, which invokes a color selector.

5.3.9.3. Contour Plot: Render Tab

See *Render Tab* (p. 6) for details.

Note

When the **Draw Faces** check box is selected, the area between contour lines is shaded with a color that corresponds to a value midway between the upper and lower contour line value. For example, for a contour line at 1000 Pa and a contour line at 1200 Pa, the shaded area has a color that corresponds to 1100 Pa. If you have created a legend for the contour plot, the legend adopts flat shading between 2 contour levels. By referring to the legend, the variable values can quickly be associated with the shaded regions of the plot.

Note

To view the contour lines as a single color, select the **Constant Coloring** check box. With this option selected, you can either leave the color set to default or you can set **Color Mode** to `User Specified` then choose a color by clicking in the **Text Color** box or by clicking the ⊡ icon, which invokes a color selector.

5.3.10. Instance Transform Command

Due to rotational symmetry, only one blade passage needs to be meshed, reducing computing cost and time. *Instance transforms* are used to replicate sections of the computational domain in the viewer for viewing purposes. For example, you may use a rotational transform to copy a blade passage to produce a plot of an entire rotor or stator, or a fraction thereof. However, for most purposes, it is sufficient to make use of the special toolbar icons dedicated to viewing 1, 2, or all instances (⊕, ⊕, and ⊕ respectively). Instance transforms are more flexible than these special toolbar icons in that they can be applied to single objects rather than to only all (qualified) objects at the same time.

To create an instance transform, select **Insert** > **User Defined** > **Instance Transform** from the main menu.

5.3.10.1. Instance Transform: Definition Tab

Note

In this release of ANSYS TurboGrid, instancing is purely visual. This means that quantitative calculations can only be carried out for the original geometry.

5.3.10.1.1. Number of Copies

The **# of Copies** is the number of times the domain is replicated in the viewer. Type in the value, increase or decrease the value by 1 by clicking ▲ or ▼ respectively.

To see the whole rotating machine, the number of copies must equal the number of blade sets, as defined in the `Machine Data` object.

5.3.10.1.2. CCL Editing

If you edit an instance transform in the **Command Editor** dialog box, changes to settings, other than **Number of Copies**, will be lost the next time the **Apply** button is clicked in the instance transform

---

editor. This happens because the instance transform editor overwrites all of the ccl parameters, except **# of Copies**, with the values stored in the default instance transform object.

5.3.11. Legend Command

A legend can be created for any object that plots a variable. The legend gives an approximate quantitative value to the colors representing the variable on a locator. To create a legend, select **Insert** > **User Defined** > **Legend** from the main menu. After a legend object is created, it is listed in the object selector under `User Defined`.

5.3.11.1. Legend: Definition Tab

5.3.11.1.1. Plot

Set **Plot** to the object for which to create a legend.

> **Note**
>
> Any existing object can be selected, but if there is not a variable for which to create a legend, one is not created.

5.3.11.1.2. Location

The exact position of the legend in the viewer can be controlled using the **Location** settings. Set **X Justification** to `Left`, `Center`, `Right`, or `None`. Similarly, set **Y Justification** to `Top`, `Center`, `Bottom`, or `None`. In both cases, if `None` is selected, type in the position value(s) (or use the embedded sliders, which have a maximum value of 1 and a minimum value of 0) in the appropriate **Position** box (The left box is for **X Justification** and the right box is for **Y Justification**.). The position values represent a fraction of the viewer width from the left side for **X Position**, or a fraction of the viewer height from the bottom for **Y Position**. The position entered is the bottom-left corner of the legend.

5.3.11.2. Legend: Appearance Tab

5.3.11.2.1. Sizing Parameters

The size of the legend can be set as a fraction of the viewer window height. Increasing the value of the **Size** setting increases both the height and width of the legend. Type in the size value or use the embedded slider (which has a maximum value of 1 and a minimum value of 0).

The **Aspect** setting controls the width of the color range bar displayed in the legend. Type in the aspect value or use the embedded slider (which has a maximum value of 0.2 and a minimum value of 0).

5.3.11.2.2. Text Parameters

The **Precision** setting controls the number of digits after the decimal point, for numbers displayed in the legend. Set **Precision** to a format of either `Scientific` or `Fixed`.

The **Value Ticks** setting specifies the number of graduations (with labels) displayed in the legend. For example, for a scale ranging from 0 to 10, a setting of 3 ticks produces graduations at 0, 5 and 10. For a contour plot, each number assigned to a contour line is displayed on the legend, along with its associated variable value. The **Value Ticks** setting has no effect when the legend is for a contour plot.

5.3.12. Text Command

Text can be added to the viewer, for example for annotation or comments. To create text, select **Insert** > **User Defined** > **Text** from the main menu. After a text object is created, it is listed in the object selector under `User Defined`.

5.3.12.1. Text: Definition Tab

5.3.12.1.1. Location

Set **Position Mode** to one of the available methods for defining the location of the text.

- Using **Two Coords**, position the text at a fixed location in the viewer. Set **X Justification** to `Left`, `Center`, `Right`, or `None`. Similarly, set **Y Justification** to `Top`, `Center`, `Bottom`, or `None`. In both cases, if `None` is selected, type in the position value(s) (or use the embedded sliders, which have a maximum value of 1 and a minimum value of 0) in the appropriate **Position** box (The left box is for **X Justification** and the right box is for **Y Justification**.). The position values represent a fraction of the viewer width from the left side for **X Position**, or a fraction of the viewer height from the bottom for **Y Position**. The position entered is the bottom-left corner of the text.

- Using **Three Coords**, position the text using Cartesian coordinates attached to the geometry. The text moves with the geometry, but its orientation remains fixed. The X, Y, and Z coordinates are required to set the text location. Type in the position values or use the embedded sliders. Type in the rotation value, or set it using the embedded slider (which has a maximum value of 360 degrees and a minimum value of 0 degrees), or click in the **Rotation** box, click *Enter Expression* 🔲 (to the right of the **Rotation** setting), and enter an expression. A rotation angle of 0 degrees positions the text horizontally; a positive angle is measured counter-clockwise from that position.

5.3.12.2. Text: Appearance Tab

5.3.12.2.1. Text Properties

Type in the text height or use the embedded slider (which has a maximum value of 1 and a minimum value of 0).

To change the text color, change **Color Mode** to `User Specified` then choose a color by clicking in the **Color** box or by clicking the ⬛ icon, which invokes a color selector.

Chapter 6: Display Menu

The following sections describe the commands available in the **Display** menu:

6.1. Introduction

The **Display Menu** options are also available from a toolbar located above the viewer.
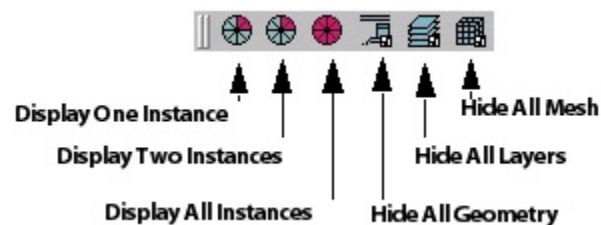
6.2. Display One Instance Command

The default setting, **Display One Instance**, will show one passage of the geometry in the viewer. Since only one instance must be solved, displaying only one instance will allow ANSYS TurboGrid to work more quickly as less rendering is required.

For further information, see *Instance Transform Command* (p. 44).

6.3. Display Two Instances Command

When **Display Two Instances** is set, two passages of the geometry are shown in the viewer. This can give a better idea of the full machine without having ANSYS TurboGrid slowed by rendering the full machine.

For further information, see *Instance Transform Command* (p. 44).

Note

When working with two instances displayed, it is important to know that topology changes, such as the movement of master control points, can only be done on the original instance, and not the second one displayed.

6.4. Display All Instances Command

When **Display All Instances** is set, the full machine is shown in the viewer. This setting shows the full geometry and the mesh can be seen on the entire geometry. It is not recommended to work in this setting as the constant rendering of the full machine will slow down processing speed.

For further information, see *Instance Transform Command* (p. 44).

6.5. Hide/Unhide Geometry Objects Commands

Select **Hide Geometry Objects** to turn off the visibility of all geometry objects (that is, hub, shroud, blade, and so on) and give a clear view of the remaining objects. Select **Unhide Geometry Objects** to do the opposite.

6.6. Hide/Unhide Layers Commands

Select **Hide Layers** to turn off the visibility of all of the layers. This is much more efficient than turning off the visibility for each individual layer in the object selector. Select **Unhide Layers** to do the opposite.

6.7. Hide/Unhide Mesh Objects Commands

Select **Hide Mesh Objects** to turn off the visibility of the `3D Mesh` objects. This can be useful for viewing the geometry after a mesh has been created. Select **Unhide Mesh Objects** to do the opposite.

6.8. Blade-to-Blade View Submenu

In the blade-to-blade view, some parts of the mesh might appear to be distorted, wavy, or overlapping[1]. You might be able to reduce the amount of distortion by selecting an appropriate command in the **Blade-to-Blade View** submenu. Each command affects the portion of the geometry on which the transform is based. The optimal choice depends on the blade geometry.

The **Blade-to-Blade View** submenu commands are:

- **Use Default Transform**

  The **Use Default Transform** command chooses the transform method automatically by effectively choosing either the **Use Full Transform** command or the **Use Passage Transform** command.

- **Use Full Transform**

  The **Use Full Transform** command causes the blade-to-blade coordinates to be calculated using the complete hub and shroud curves.

- **Use Passage Transform**

  The **Use Passage Transform** command causes the blade-to-blade coordinates to be calculated using the portion of the hub and shroud curves that fall within the passage mesh, truncated at the inlet and outlet (that is, excluding the portions of the hub and shroud curves that lie within the inlet and outlet mesh blocks).

- **Use Passage Excluding Tip Transform**

---

[1]By contrast, the Cartesian view does not typically exhibit distortion, except at extremely high zoom levels.

The **Use Passage Excluding Tip Transform** command is similar to the **Use Passage Transform** command, except that the tip regions are excluded in the spanwise direction. For example, if a blade has no hub tip and a profile-based shroud tip, the blade-to-blade coordinates are calculated using the portion of the hub curve that falls within the passage and the profile curve at the shroud tip. This transform may be the best choice if the hub/shroud tip is defined by a profile that varies significantly from a constant span when viewed in the other transforms.

The Passage and Passage Excluding Tip transforms:

- usually exhibit less distortion than the Full transform,

- are available only after the topology has been created,

- cause geometry objects to be omitted from the blade-to-blade view (even if these transforms are used indirectly via the default transform).

Chapter 7: Viewer

7.1. Introduction

The viewer in ANSYS TurboGrid plays a central role in the mesh creation process. Its interactive interface, including the mouse, toolbars and hotkeys, allows you to inspect and alter your work.

The viewer toolbar and hotkeys are described in this chapter. Mouse controls are described in *Viewer Setup: Mouse Mapping* (p. 25).

Note

In order to see correct colors and accurately displayed objects in the 3D Viewer, some combinations of ATI video cards and ATI graphics drivers on Windows XP require that you set the environment variable **VIEWER_CACHE_COLORS** to 0:

1. Right-click **My Computer** and select **Properties**. The **System Properties** dialog box appears.
2. Click the **Advanced** tab.
3. Click **Environment Variables**.
4. Under **System variables**, click **New**.
5. In the **Variable name** field, type: VIEWER_CACHE_COLORS
6. In the **Variable value** field, type the number: 0
7. Click **OK**.
8. To verify the setting, open a command window and enter: `set`

   The results should include the line:

   ```
   VIEWER_CACHE_COLORS=0
   ```

This setting will fix problems such as:
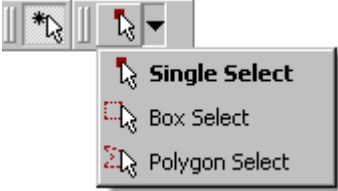
- Boundary condition markers placed incorrectly or rendered in white.
- Regions around the circles are incorrect (rendered as yellow areas marked with blue)
- Mesh lines not displayed properly and with dark patches showing.

Note
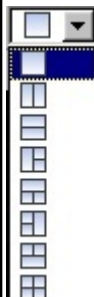
Depending on the graphics card and driver version, you may experience problems with the accuracy of mouse clicks in the 3D Viewer. For example, you may try to insert a control point at a given location by using the mouse, but the control point appears at a location far from where you clicked the mouse. If you experience such problems, try lowering the hardware acceleration setting of your graphics card.

7.2. Viewer Toolbar

The viewer has the following tools:

Tool	Description
	Enters picking mode. To choose the behavior of this mode, use the drop-down arrow on the adjacent toolbar icon (which becomes visible only after you click *Select*): Single Select Box Select Polygon Select
	This is the **Single Select** option for the *Select* tool. You can use it to select objects or drag certain objects to new locations, using the mouse. When a number of objects overlap, the one closest to the camera is picked. The text at the bottom of the viewer window shows which object would be picked at the mouse's current location. If you cannot pick the object you want because other objects overlap it, turn off the visibility of the overlapping objects, or adjust the camera to make it possible to pick the object.
	This is the **Box Select** option for the *Select* tool. You can use it to select objects using a box. Drag a box around the object(s) you want to select.
	This is the **Polygon Select** option for the *Select* tool. You can use it to select objects using an enclosed polygon. Click to drop points around the object(s). Double-click to complete the selection.
	Rotates the view as you drag with the mouse. Alternatively, hold down the middle mouse button to rotate the view.

Tool	Description
	Pans the view as you drag with the mouse. Alternatively, you can pan the view by holding down **Ctrl** and the middle mouse button.
	Adjusts the zoom level as you drag with the mouse vertically. Alternatively, you can zoom the view by holding down **Shift** and the middle mouse button.
	Zooms to the area enclosed in a box that you create by dragging with the mouse. Alternatively, you can drag and zoom the view by holding down the right mouse button.
	Centers all visible objects in the viewer.
	Toggles highlighting. Highlighting makes it easier to select the correct object from the viewer. While in picking mode, highlighting by default puts a red box around the object which would be selected at the mouse's current location. The style of highlighting is controlled by **Edit** > **Options** > **TurboGrid** > **Viewer** > **Highlight Type**.
	Selects the viewport arrangement; by default, each view displays a different transform. Independent zoom, rotation, and translate options can be carried out in each viewport.
	Displays the **Viewer Key Mapping** dialog box. See *Viewer Hotkeys* (p. 53) for details.

7.3. Viewer Hotkeys

A number of hotkeys are available to carry out common viewer tasks. Before using a viewer hotkey, place the mouse focus on the viewer window.

Key	Action
Space	Toggle between picking and viewing mode.
Up or **Down** or **Left** or	Rotate about horizontal and vertical axes.

Key	Action
Right	
Ctrl-**Up** or **Ctrl**-**Down**	Rotate about an axis normal to the screen.
Shift-**Up** or **Shift**-**Down** or **Shift**-**Left** or **Shift**-**Right**	Moves the light source.
Ctrl-**Shift**	When in viewing mode, you can hold **Ctrl**+**Shift** to select objects in the viewer. Releasing **Ctrl**+**Shift** returns you to viewing mode.
1	Switches to one viewport.
2	Switches to two viewports.
3	Switches to three viewports.
4	Switches to four viewports.
c	Centers the graphic object in the viewer window.
n	Toggles the projection between orthographic and perspective.
r	Resets the view to the initial orientation.
u	Undoes transformation.
U	Redoes transformation.
x	Sets view from +X axis.
X	Sets view from -X axis.
y	Sets view from +Y axis.
Y	Sets view from -Y axis.
z	Sets view from +Z axis.
Z	Sets view from -Z axis.

7.4. Multiple Viewports

Initially the viewer contains one single viewport. The viewer can be divided into more than one window or multiple viewports, ranging from one to four.

By default, the viewer shows the current objects in a 3D Cartesian view.

The currently active viewport layout is shown in the box to the right of the *Highlighting* icon on the viewer toolbar. Click to the right of the current viewport layout picture to select a new viewport layout from the drop-down list on the viewer toolbar.
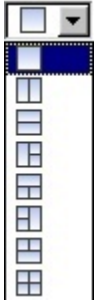
Change the active viewport by placing the mouse pointer over a viewport and clicking with any of the three mouse buttons. The viewport that contains the mouse pointer is then set as the active viewport.

Mouse-controlled transformations are applied automatically to the viewport which contains the mouse pointer. When the transformation is complete the viewport also becomes the active viewport.

Hardcopy plots (postscript or other image file formats) always show all visible viewports in the viewer (a verbatim copy of the viewer window).

If the viewer consists of four viewports, each with object(s), and the viewport layout is changed to have less than four viewports, the graphic in the now "hidden" viewports remains intact, but not visible. When the viewport layout returns to the former layout all graphic objects are present, unchanged.

Each viewport has a default coordinate system.

- Viewport 1: Cartesian
- Viewport 2: 2D Blade-to-blade
- Viewport 3: 2D Meridional
- Viewport 4: 3D Turbo

You can change the coordinate system for each viewport by right-clicking on a blank area in the viewer, and selecting one of the **Transformation** commands.

Note

If a viewport initially appears to be empty when it is used for the first time, try clicking *Fit View* . This will center the objects and reset the zoom level.

7.4.1. Selecting, Adding, and Deleting Views

Each viewport can use any of the 4 pre-defined views or user-defined views. To switch between views, use the drop-down menu in the upper-left corner of the viewport. To add a new view based on the current state of the viewer, right-click in the viewer and select **Create New View** from the shortcut menu. To delete an existing user-defined view, right-click in the viewer and select **Delete View** while the view is selected.

7.5. Selecting and Dragging Objects while in Viewing Mode

Picking mode allows you to select and move objects, such as points, curves, and planes.

55

To enter picking mode, click *Select* ![icon]. To leave picking mode (i.e., switch back to *viewing mode*), click *Rotate* ![icon] or *Pan* ![icon].

To temporarily enter picking mode when you are in viewing mode, hold **Ctrl**+**Shift**. When you release the **Ctrl**+**Shift** keys, you will return to viewing mode.

When you click *Select* ![icon] to enter picking mode, a toolbar icon appears to enable you to change the selection method:
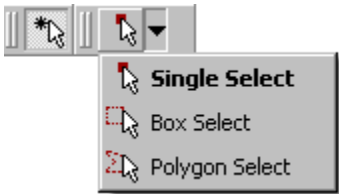
For details, see *Viewer Toolbar* (p. 52).

7.6. Stereo Viewer

If you:

1. Have a standard stereo display

2. Have a graphics card that supports quad buffering OpenGL output

3. Have set your graphics card to "Stereo"

4. Have set your view to Perspective mode (right-click in the Viewer and select **Projection** > **Perspective**)

...you can view output in stereo[1]. To enable this functionality:

1. Select **Edit** > **Options**.

2. In the **Options** dialog box, select **TurboGrid** > **Viewer**.

3. On the **Viewer** panel:

    a. Set the Stereo **Mode** to **Stereo**.

    b. Set the **Stereo Effect**. The value of the "stereo effect" that is required is related to the distance between the observer and the display. If the stereo effect is too strong, either move away from the display, or move the slider towards **Weaker**.

4. Click **OK** to save the settings.

---

[1]The stereo viewer feature has been tested on:

*   XP-64 and Vista-64 with an NVidia graphics card and a Planar Stereo Monitor

*   XP-64 with an ATI card and a Zalman Stereo Monitor.

Chapter 8: Tools Menu

The following sections describe the commands available in the **Tools** menu:

8.1. Calculator Command

The built-in calculator provides quantitative information about the geometry and mesh. To view the calculator, select **Tools** > **Calculator** from the main menu. The **Function Calculator** dialog box is displayed.

> **Note**
>
> In this release of ANSYS TurboGrid, quantitative calculations can only be carried out for the original geometry. Any applied instance transforms are purely visual and do not affect the calculation results.

8.1.1. Function Calculator Dialog Box

8.1.1.1. Function

Click ⯆ to select a function from the drop-down list. The table below outlines the available quantitative functions.

Function Name	Operation
area (p. 58)	Area of location
areaAve (p. 58)	Area-weighted average
areaInt (p. 58)	Area-weighted integral (can be projected to a direction)
ave (p. 59)	Arithmetic average
count (p. 59)	Number of calculation points
length (p. 59)	Length of a curve
lengthAve (p. 59)	Length-weighted average
length-Int (p. 59)	Length-weighted integration
maxVal (p. 60)	Maximum Value
minVal (p. 60)	Minimum Value
probe (p. 60)	Value at a point

Function Name	Operation
sum (p. 60)	Sum over the calculation points
volume (p. 60)	Volume of a 3-D location
volumeAve (p. 60)	Volume-weighted average
volumeInt (p. 61)	Volume-weighted integral

8.1.1.1.1. area

The `area` function is used to calculate the area of a 2-D location. The following example demonstrates use of the function.

- **Function:** `area`, **Location:** `Plane1`. This example calculates the total area of the locator `Plane1`.

8.1.1.1.2. areaAve

The `areaAve` function calculates the area-weighted average of an expression on a 2-D location. The area-weighted average of a variable is the average value of the variable on a location when the mesh element sizes are taken into account. Without the area weighting function, the average of all the nodal variable values would be biased towards variable values in regions of high mesh density. The following examples demonstrate use of the Function.

- **Function:** `areaAve`, **Location:** `Outlet`, **Variable:** `Velocity`. This example calculates the average magnitude of the velocity on the `outlet` location. Note that flow direction is not considered since the magnitude of a vector quantity at each node is calculated. Use the scalar components of velocity (e.g., `Velocity u`) to include a directional sign, for example:

- **Function:** `areaAve`, **Location:** `Outlet`, **Variable:** `max(Velocity u, 0.0[m s^-1])`. This example calculates the area-weighted average value of `Velocity u`, with negative values of the variable replaced by zero. Note that this is not the average positive value since zero values contribute to the average.

8.1.1.1.3. areaInt

The `areaInt` function integrates a variable over the specified 2-D location. To perform the integration over the total face area, the **None** option should be selected from the Direction drop-down list. If a direction is selected, the result is an integration over the projected area of each face onto a plane normal to that direction. Each point on a location has an associated area which is stored as a vector and therefore has direction. By selecting a direction in the calculator you are using only a single component of the vector in the area-weighting function. Since these components can be positive or negative, depending on the direction of the normal on the location, it is possible for areas to cancel out. An example of this would be on a closed surface where the projected area is always zero (the results returned are not in general zero since the variable values differ over the closed surface). On a flat surface the normal vectors always point in the same direction and never cancel out. The following examples demonstrate use of the function.

- **Function:** `areaInt`, **Location:** `Plane1`, **Variable:** `Pressure`, **Direction:** `None` This example integrates pressure over Plane1. The result returned is the total pressure force acting on Plane1. The magnitude of each area vector is used and so the direction of the vectors is not considered.

- **Function:** `areaInt`, **Location:** `Plane1`, **Variable:** `Pressure`, **Direction:** `Global X`. This example integrates pressure over the projected area of Plane1 onto a plane normal to the X-axis. The result is the pressure force acting in the X-direction on Plane1. This differs slightly from using the force function to calculate the X-directional force on Plane1 — the force function includes forces due to the advection

of momentum when calculating the force on an internal arbitrary plane or a non-wall boundary (inlets etc.).

8.1.1.1.4. ave

The `ave` function calculates the arithmetic average (the mean value) of a variable or expression on the specified location. This is the sum of the values at each node on the location divided by the number of nodes. Results are biased towards areas of high nodal density on the location. To obtain a mesh-independent result, use the `lengthAve`, `areaAve`, `volumeAve` or `massFlowAve` functions. The following example demonstrates use of the function.

The average of a vector value is calculated as an average of its magnitudes, not the magnitude of component averages. As an example, for velocity, $|v|_{\text{ave}} = \dfrac{|v_1| + |v_2|}{2}$

where $|v_i| = \sqrt{\left(v_{x\,i}^2 + v_{y\,i}^2 + v_{z\,i}^2 \right)}$

- **Function:** `ave`, **Location:** `MainDomain`, **Variable:** `Temperature`. This example calculates the mean temperature at all nodes in the selected domain.

8.1.1.1.5. count

The `count` function returns the number of nodes on the specified location. The following example demonstrates use of the function.

- **Function:** `count`, **Location:** `MainDomain`. This example returns the number of nodes in the specified domain.

8.1.1.1.6. length

Computes the length of the specified line as the sum of the distances between the points making up the line. The following example demonstrates use of the function.

- **Function:** `length`, **Location:** `Polyline1`. Calculates the length of the Polyline.

8.1.1.1.7. lengthAve

Computes the length-based average of the variable on the specified line. This is the 1-D equivalent of the areaAve function. The result is independent of the nodal distribution along the line since a weighting function assigns a higher weighting to areas of sparse nodal density. The following example demonstrates use of the function.

- **Function:** `lengthAve`, **Location:** `Polyline1`, **Variable:** `Velocity`. This calculates the average velocity on the location Polyline1 using a length-based weighting function to account for the distribution of points along the line.

8.1.1.1.8. lengthInt

Computes the length-based integral of the variable on the specified line. This is the 1-D equivalent of the areaInt function. The following example demonstrates use of the function.

8.1.1.1.9. maxVal

Returns the maximum value of the specified variable on the specified locator. Create a user variable if you want to find the maximum value of an expression. The following example demonstrates use of the function.

- **Function:** `maxVal`, **Location:** `Default`, **Variable:** `Yplus`. This returns the maximum `Yplus` value on the `Default` wall boundaries.

8.1.1.1.10. minVal

Returns the minimum value of the specified variable on the specified locator. Create a user variable if you want to find the minimum value of an expression. The following example demonstrates use of the function.

- **Function:** `minVal`, **Location:** `MainDomain`, **Variable:** `Temperature`. These settings return the minimum temperature in the domain.

8.1.1.1.11. probe

Returns the value of the specified variable on the specified point object. The following example demonstrates use of the function.

- **Function:** `probe`, **Location:** `Point1`, **Variable:** `Density`. Returns the density value at `Point1`.

> **Important**
>
> This calculation should only be performed for point locators described by single points. Incorrect solutions are produced for multiple point locators.

8.1.1.1.12. sum

Computes the sum of the specified variable values at each point on the specified location. The following example demonstrates use of the function.

- **Function:** `sum`, **Location:** `SubDomain1`, **Variable:** `Volume of Finite Volume` Returns the sum of the finite volumes assigned to each node in the location `SubDomain1`. In this case this sums to the volume of the subdomain.

8.1.1.1.13. volume

The `volume` function is used to calculate the volume of a 3-D location. The following example demonstrates use of the function.

- **Function:** `volume`, **Location:** `Volume1`. Returns the sum of the volumes of each mesh element included in the location `Volume1`.

8.1.1.1.14. volumeAve

The `volumeAve` function calculates the volume-weighted average of an expression on a 3-D location. This is the 3-D equivalent of the `areaAve` function. The volume-weighted average of a variable is the average value of the variable on a location weighted by the volume assigned to each point on a location.

Without the volume weighting function, the average of all the nodal variable values would be biased towards values in regions of high mesh density. The following example demonstrates use of the function.

- **Function:** `volumeAve`, **Location:** `Volume1`, **Variable:** `Density`. This example calculates the volume-weighted average value of density in the region enclosed by the location `Volume1`.

8.1.1.1.15. volumeInt

The `volumeInt` function integrates the specified variable over the volume location. This is the 3-D equivalent of the areaInt function. The following example demonstrates use of the function.

- **Function:** `volumeInt`, **Location:** `Volume1`, **Variable:** `Density`. This calculates the integral of density (the total mass) in `Volume1`.

8.1.1.2. Location

Click ▾ to select a location from the drop-down list. Only locations valid for the selected function are available.

8.1.1.3. Variable

Click ▾ to select a variable from the drop-down list. Only variables valid for the selected function are available.

For most functions, click in the **Variable** box and enter an expression to use as the variable. The expression can include other variables and any valid CEL (ANSYS CFX Expression Language) function (see CEL Functions, Constants and System Variables in the *TurboGrid Reference Guide*). For example, `abs(Velocity u)` could be entered so that the calculation is performed using the absolute values of the variable `Velocity u`.

8.1.1.4. Direction

The `areaInt` function requires a direction to be specified before the calculation can be performed. The `areaInt` function projects the location onto a plane normal to the specified direction (if the direction is not set to `None`), and then performs the calculation on the projected location (direction specification can also be `None`). The direction of the normal vectors for the location is important and cancels out for surfaces such as closed surfaces.

8.1.1.5. Hybrid and Conservative Variables

In **ANSYS TurboGrid** there is no difference between hybrid and conservative variables. Leave all controls for selecting between them at their default values.

8.2. Expressions Command

The expression editor is used to create new expressions and modify existing expressions in ANSYS TurboGrid. ANSYS TurboGrid can use expressions in place of any numeric value (as long as the correct units are returned by the expression). For example, you can use expressions to define object properties or new variables.

61

Note

- When a setting is defined by an expression, and the latter evaluates to a quantity that has no units, the software internally applies the default units for that setting.

- In an expression, a term that has no units can be added to a term that has angular units, in which case the software internally applies radians to the term that has no units.

To create or edit an expression, select **Tools** > **Expressions** from the main menu. The **Expression Editor** dialog box is displayed.

8.2.1. Expression Editor Dialog Box

The **Expression Editor** dialog box contains:

- A tree view for viewing and managing expressions (The tree view has a shortcut menu with commands for creating/deleting/modifying expressions, as described below.)

- A details view for creating/editing/displaying the definition of the new/selected expression

  (You can find a list of valid CEL expressions and constants in ANSYS CFX Expression Language in the *TurboGrid Reference Guide*.)

- A **Value** box to show the value of the selected expression when the expression does not contain variables and evaluates to a single value.

You can right-click an existing expression or a blank area in the tree view portion of the **Expression Editor** dialog box in order to access a shortcut menu with the following commands:

Table 8.1 Expression Editor Shortcut Menu Commands

Command	Description
Edit	Displays the definition of the selected **Expression** in the details view portion of the **Expression Editor** dialog box, and gives it the keyboard focus. An alternative way to do the same is to double-click an expression in the tree view.
Insert > Expression	Opens the **New Expression** dialog box where you can enter a name for a new expression. After you enter a name, you must provide and apply a definition in order to finish creating the new expression. When selecting an expression name: • You cannot create an expression with the same name as an object. • You cannot create an expression with the same name as a variable.
Edit in Command Editor	Invokes the **Command Editor** dialog box populated with the selected expression.
Delete	Deletes the selected expression.

Command	Description
Duplicate	Creates a duplicate of the selected expression after obtaining the name of the new expression from the **New Expression** dialog box.
Use as Workbench Input Parameter[1 (p. 63)]	Makes the selected expression a Workbench input parameter so that it is available in, and controllable by, ANSYS Workbench. The Workbench input parameter is initialized by the expression definition specified in ANSYS TurboGrid. The definition must be a function of only constants; units are permitted. It is not possible to edit the value of an expression in ANSYS TurboGrid while the expression is a Workbench input parameter.
Use as Workbench Output Parameter[1 (p. 63),2 (p. 63)]	Makes the selected expression a Workbench output parameter, causing it to be output from ANSYS TurboGrid for use in ANSYS Workbench.
Deselect as Workbench Parameter[1 (p. 63),2 (p. 63)]	Removes the ANSYS Workbench parameter status from the selected expression.

Footnote

1. You can enable/disable the use of a Workbench input parameter directly for many settings in the object editor. For details, see *Object Editor* (p. 4).

2. For details on Workbench parameters, see Working with Parameters and Design Points in the *Workbench User Guide*.

When defining an expression:

- You cannot use a user variable.

- Constants require units inside square brackets.

- You can right-click in the details view to access a shortcut menu for inserting variables, locators, expressions, and functions into your equation.

8.2.1.1. Expression Editor Example

To define an expression for distance from the X-axis:

1. Select **Tools** > **Expressions** from the main menu to open the **Expression Editor** dialog box.

2. When the **New Expression** dialog box appears, enter the name `radial` and click **OK**.

3. In the details view portion of the **Expression Editor** dialog box, enter the following expression definition:

```
sqrt(Y^2+Z^2)
```

4. Click **Apply** to create the expression.

Note that the **Value** box does not show a value for the expression because the expression does not evaluate to a single value.

The expression created in this example (`radial`) is used in the *Variable Editor Example* (p. 65).

8.3. Variables Command

The variable editor is used to create new user variables and modify existing variables in ANSYS TurboGrid.

To create or edit a variable, select **Tools** > **Variables** from the main menu. The **Variable Editor** dialog box is displayed.

8.3.1. Variable Editor Dialog Box

All icons become active when you select a variable from the **Variable** list. Alternatively, you can right-click on an existing variable in the list to see the same options. Each icon and its function is described in the following table.

Icon	Description
	New opens the **New Variable** dialog box where you can enter a name.
	Edit displays the selected **Variable** in the **Expression** and/or **Units** box where you can edit it.
	Copy makes a duplicate of the selected variable and opens the **New Variable** dialog box where you can enter a name for the copy.
	Delete removes the selected variable from the list.
	Use Hybrid Values sets all variables to hybrid values.
	Use Conservative Values sets all variables to conservative values.

When you click *New* to create a new variable, the **New Variable** dialog box is displayed.
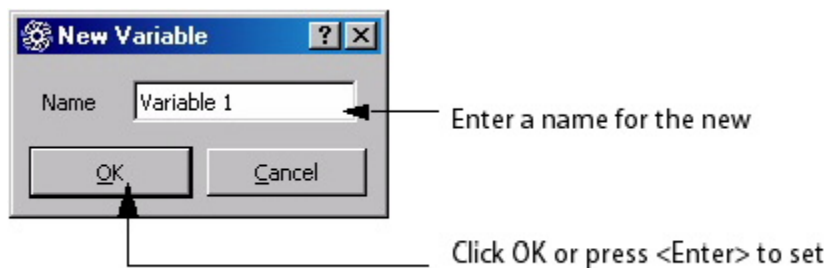
8.3.1.1. Name

There are a few guidelines to follow when selecting a variable name.

- You cannot create a variable with the same name as an object.

- You cannot create a variable with the same name as an expression. For example, if you have an expression named `Radius`, you must choose a different variable name for that expression.

- Within the ANSYS CFX Expression Language some variables are known by short names to save typing in the full variable name. For example, `p` refers to `Pressure`. Although it is possible to create a variable with the same name as an abbreviated variable, it is ignored. For example, if you use a variable with the name `p` in an expression, it returns the value `Pressure` in all cases, no matter what the definition of the variable is.

8.3.1.2. Type

Click *Edit* 📝 to edit both the fundamental and user variables. Select the expression used to define the variable from a list of existing expressions for user variables. The expressions available from this list are those which you have created in the expression editor. For details, see *Expressions Command* (p. 61).

For fundamental variables, the units are changeable. Click ▾ next to the Units box to see the available units for the selected variable. This means, for example, that you could create a legend which uses alternative angle units (such as degrees or radians) by clicking on the ▾ icon and selecting new units.

Note

These settings override the global units setting, defined in the Edit Menu. For details, see *Options Command* (p. 23).

The variable type used affects all quantitative calculations and plots in ANSYS TurboGrid.

8.3.1.3. Hybrid and Conservative Variable Values

This is useful only for advanced post-processing and is not relevant for ANSYS TurboGrid. It is included to maintain consistency with other ANSYS CFX products. Refer to the CFD-Post documentation for more information on the differences between hybrid and conservative variable values.

8.3.1.4. Variable Editor Example

In this example, an isosurface that has a fixed radial distance from the X-axis is created using the expression defined in the *Expression Editor Example* (p. 63).

1. Select **Tools** > **Variables** from the main menu to open the variable editor.

2. Click *New* 📋 to create a new variable. When the **New Variable** dialog box appears enter the name `Radial Distance` and click **OK**.

3. In the variable editor, use the **Expression** drop-down list to select the expression `radial` which you created earlier. Click **Apply** to create the new variable.

This variable now appears in the list of available variables and can be used like any other variable. Notice that the variable type is listed as **User**.

You can now create an isosurface using this variable. For details on creating isosurfaces, see *Isosurface Command* (p. 38).

1. Select **Insert** > **User Defined** > **Isosurface** from the main menu, enter a name, then click **OK** on the **New Isosurface** dialog box.

2. In the **Geometry** tab for the isosurface set **Variable** to `Radial Distance`.

3. Set **Value** to `20 m`. This is a suitable value for the Rotor 37 geometry used in Tutorial 1. You may need to alter this value to something more sensible depending on the geometry you are viewing.

4. Click the **Color** tab and set **Mode** to `Variable`.

5. Select a sensible variable (e.g., `Maximum Face Angle` or `Axial Distance`) with which to color the isosurface.

6. Set **Range** to `Local` so that the full color range is used on the isosurface.

7. Click **Apply** to create the isosurface object.

You should now see the isosurface in the viewer. All points on the isosurface are a distance of 20 m (or whatever value you used in the **Value** box) from the X-axis.

8.4. Command Editor Command

The **Command Editor** dialog box can be used to create or modify any of the objects in ANSYS TurboGrid using the CFX Command Language. For further details, see CFX Command Language in the *TurboGrid Reference Guide*.

Power Syntax can be entered and processed in the **Command Editor** dialog box. For details, see Power Syntax in the *TurboGrid Reference Guide*. Power Syntax commands should be proceeded by the `!` symbol.

In addition, any valid CCL command can be typed and processed in the **Command Editor** dialog box. For details, see Command Actions in the *TurboGrid Reference Guide*. Action Commands should be preceded by the `>` symbol.

To create an object using CCL, select **Tools** > **Command Editor** from the main menu. The **Command Editor** dialog box is displayed.

To edit an existing object using CCL, right-click the object in the object selector and select **Edit in Command Editor** from the shortcut menu. The CCL definition of that object is automatically displayed and can be edited to alter the object properties.

You can access the following basic editing tools by right-clicking in the **Command Editor** dialog box:

- **Undo**

  Undoes the last edit action.

- **Redo**

  Redoes the most recently undone edit action.

- **Cut**

  Cuts the selected text and places it on a clipboard.

- **Copy**

  Places the selected text on a clipboard.

- **Paste**

Pastes the clipboard text at the insertion point, or replaces the selection.

- **Clear**

    Clears all of the contents of the **Command Editor** dialog box.

- **Select All**

    Selects all of the contents of the **Command Editor** dialog box.

- **Find**

    Makes a search tool appear at the bottom of the **Command Editor** dialog box. Enter a search term and click either **Next** or **Previous** to search upwards or downwards from the insertion point or text selection. To hide the search tool, press <Esc>.

67

Chapter 9: Help Menu

The following sections describe the commands available in the **Help** menu:

9.1. On ANSYS TurboGrid Command

Click **Help** > **On ANSYS TurboGrid** to view the introductory documentation. This includes the new features for ANSYS TurboGrid.

9.2. Contents Command

The **Contents** command shows a table of contents for all of the online ANSYS TurboGrid documentation.

9.3. Index Command

The **Index** command shows how to access the index for the ANSYS TurboGrid documentation.

9.4. Tutorials Command

The **Tutorials** command provides access to the ANSYS TurboGrid tutorials.

9.5. Search Command

The **Search** command shows how to access the search feature for the ANSYS TurboGrid online documentation.

9.6. Installation and Licensing Command

Opens the installation and licensing documentation.

9.7. About ANSYS TurboGrid Command

The **About ANSYS TurboGrid** command describes the purpose of ANSYS TurboGrid, shows the version number, and provides an internet address.

9.8. About ICEM CFD Command

The **About ICEM CFD** command describes ICEM CFD, shows the version number, and provides an internet address.

9.9. About Qt Command

The **About Qt** command describes the purpose, shows the version number, and provides an internet address.

9.10. Help on Help Command

The **Help on Help** command provides additional information about ANSYS TurboGrid help and how to access it.

Chapter 10: ANSYS TurboGrid Workflow

10.1. Introduction

This chapter explains the steps required to create a mesh. A brief overview is given in the next section. The remaining sections of this chapter cover each major step in detail.

10.2. Steps to Create a Mesh

To create a mesh:

1. Define the geometry.

   For details, see *Geometry* (p. 73).

2. Define the topology object(s) by selecting a method and optionally changing other settings.

   For details, see *Topology* (p. 90).

3. Optionally modify the `Mesh Data` object to adjust the number of nodes.

   If you plan to make a fine (high-resolution) mesh, you can optionally set the mesh density at a later time in order to minimize processing time while establishing the topology. Keep in mind that changing the mesh density can affect the mesh quality.

   For details, see *Mesh Data* (p. 113).

4. Optionally modify the layer objects to improve mesh quality.

   For details, see *Layers* (p. 122).

5. Generate the 3D mesh using the **Insert** > **Mesh** menu item, the *Create Mesh* icon, or by clicking the **Generate** button in the `3D Mesh` object.

For details, see *3D Mesh* (p. 133).

6. Optionally investigate the mesh and refine any of the above objects as necessary.

    To help identify problem areas of the mesh, mesh analysis tools are available. For details, see *Mesh Analysis* (p. 133).

    If you change any objects that affect the mesh, you must generate the mesh again.

7. Save the mesh to a file.

    For details, see *Save Mesh Command* (p. 17).

In general, you should proceed from top to bottom in the object selector, or from left to right on the toolbar.

Figure 10.1 Object Selector
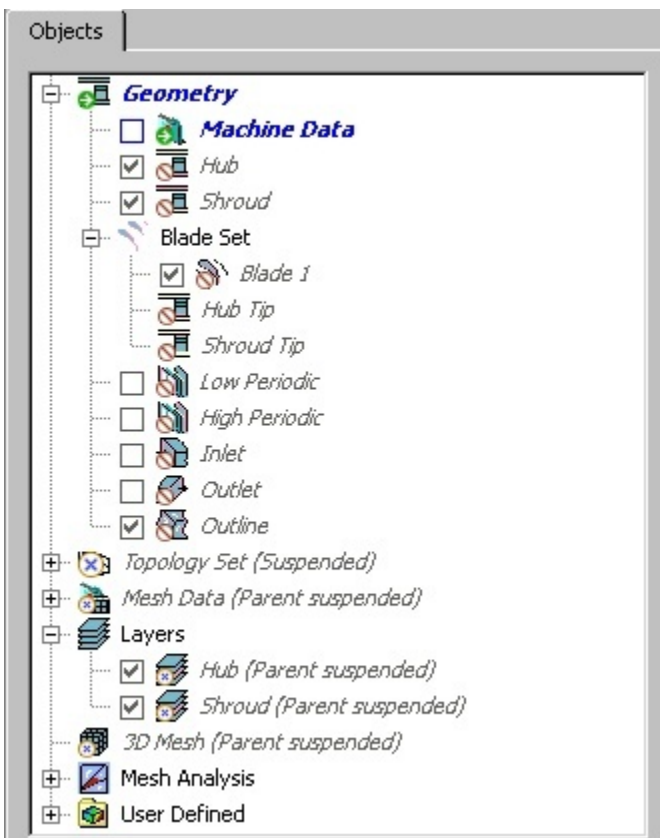
Figure 10.2 Part of Toolbar
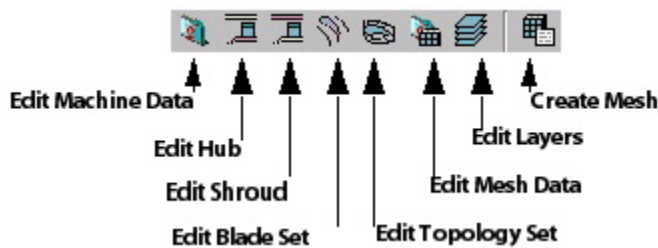
The other objects in the object selector are used for visualization and inspection purposes.

Not every object in the object selector needs to be defined before a mesh can be created. However, every object represented by an icon on the toolbar must be defined.

10.3. Geometry

To define the geometry, you can do any of the following:

- In ANSYS Workbench, attach a Geometry or Blade Design cell upstream of the Turbo Mesh cell.

  Note that, in this case, some of the geometry objects (such as Machine Data, Hub, Shroud, Blade Set, and Blade) have their settings taken from the upstream cell, and changes to these settings cannot be made in ANSYS TurboGrid. Other settings in geometry objects are taken from the upstream cell by default, but can be overridden. To override a particular setting, select the check box for the frame surrounding the setting (in the object editor). For example, although the machine rotation axis is, by default, taken from the upstream cell, you can change the rotation axis by selecting the **Rotation** check box in the **Machine Data** object editor.

- Load a BladeGen `inf` file by selecting **File** > **Load BladeGen** or clicking *Load BladeGen* .

  This is recommended if a `BladeGen.inf` file exists. For details, see *BladeGen.inf File* (p. 10).

- Specify multiple geometry objects at once by selecting **File** > **Load Curves** or clicking *Load Curves* .

  This information includes machine data, the names of geometry definition files, choice of coordinate system and leading/trailing edge settings. For details, see *Curve File* (p. 10).

- Load a `cfg` file by selecting **File** > **Load CFG**.

  For details, see *TurboGrid 1.6 cfg File* (p. 10).

- Define the `Machine Data` object, then load each of the remaining geometry objects separately.

The geometry objects are outlined below with a summary of the function performed by each.

The Machine Data Object (p. 74)	This object defines the geometry and rotation properties of the rotating machine. For details, see *The Machine Data Object* (p. 74).
The Hub and Shroud Objects (p. 76)	These objects define the hub of the rotating machine. For details, see *The Hub and Shroud Objects* (p. 76).
The Blade Set Object and Blade Objects (p. 77)	These objects define the blade(s) of the rotating machine. For details, see *The Blade Set Object and Blade Objects* (p. 77).
The Hub Tip and Shroud Tip Objects (p. 86)	These objects define the hub tip and shroud tip of the blade(s). For details, see *The Hub Tip and Shroud Tip Objects* (p. 86).
The Low Periodic and High Periodic Objects (p. 87)	These objects control the methods for creating the low and high periodic surfaces. For details, see *The Low Periodic and High Periodic Objects* (p. 87).

<table>
<tr><td>The Inlet and Outlet Objects (p. 88)</td><td>These objects define the locations and profiles of the inlet and outlet domains of the mesh. For details, see The Inlet and Outlet Objects (p. 88).</td></tr>
<tr><td>The Outline Object (p. 90)</td><td>This object controls the display of the outline of the geometry. For details, see The Outline Object (p. 90).</td></tr>
</table>

After you have finished creating the geometry, you may want to hide some or all of it to allow you to see the topology clearly. You can control the visibility of a geometry object by toggling the check box next to the object in the object selector. You can also right-click an object and use shortcut menu commands. The toolbar allows you to hide and unhide all geometry objects via the *Hide all geometry objects* and *Unhide_geometry_objects* icons.

10.3.1. The Machine Data Object

The `Machine Data` object contains geometric data that applies to the entire turbo machine (e.g., the location of the rotation axis). Defining the machine data is an essential step in creating a mesh in ANSYS TurboGrid, and can be accomplished by one of the following methods:

- Editing the `Machine Data` object.

- In ANSYS Workbench, attaching a Geometry or Blade Design cell upstream of the Turbo Mesh cell.

  Note that, in this case:

  – The **Pitch Angle** setting is taken from the upstream cell, and cannot be changed in ANSYS TurboGrid.

  – The **Rotation** settings are taken from the upstream cell by default, but can be overridden by selecting the **Rotation** check box.

  – The **Base Units** setting can be changed and is not affected by changes to the upstream cell.

- Loading a BladeGen `.inf` file by selecting **File** > **Load BladeGen** or clicking *Load BladeGen INF file* .

  You can define machine data, hub, shroud, and blade objects by loading a BladeGen `.inf` file. For details, see *Load BladeGen Command* (p. 11).

- Loading curve files by selecting **File** > **Load Curves** or clicking *Load Curves* .

  You can define the machine data, hub, shroud, and blade objects by loading curve files. For details, see *Load Curves Command* (p. 12).

- Loading a `cfg` file by selecting **File** > **Load CFG**.

  For details, see *TurboGrid 1.6 cfg File* (p. 10).

To edit the machine data, open the `Machine Data` object from the object selector or click *Edit Machine Data* .

10.3.1.1. Data Tab

10.3.1.1.1. Pitch Angle

A rotating machine component is made up of blade sets that are identical, adjacent to each other, and equally spaced around the entire circumference. In the context of ANSYS TurboGrid, the *pitch angle* is the Theta extent of one blade set. To calculate the pitch angle based on the number of blade sets per 360°, set **Method** to `Bladeset Count`; the pitch angle is then calculated as 360° divided by the number of blade sets. If you need to specify the pitch angle directly (as in the Deformed Turbine tutorial), set **Method** to `Specified Angle`.

ANSYS TurboGrid creates a mesh for one blade set only. The mesh can be copied and rotated using an ANSYS CFX Pre-processor, if necessary, before it is solved in an ANSYS CFX Solver.

10.3.1.1.2. Rotation

The **Method** for defining the rotation axis can be set to one of the following:

- **Principal Axis** - select the X, Y, Z, -X, -Y, or -Z axis.
- **Rotation Axis** - define a custom axis by entering the Cartesian coordinates of 2 points on the axis. The coordinates of the points can be typed in or selected using the ⬚ icon in the object editor.

10.3.1.1.3. Units

It is necessary to specify base units because the geometry kernel has an optimal range for storing numbers between 1 and 500. If, for example, the geometry data is given in mm, but the machine is on the order of 10 m in size, the numbers stored in mm would be outside of the optimal range. In this case, the base units should be cm (default), to reflect the scale of the machine.

The base units specified in the `Machine Data` object are used only to adjust the range of numbers to suit the geometry kernel; they are not used to interpret geometric data contained in files (e.g., the hub curve file), nor are they used to specify the units of an exported mesh.

10.3.1.2. Rotation Axis Visibility

By turning on the visibility of the `Machine Data` object, the rotation axis becomes visible in the viewer, along with a sample radial direction vector. An example is shown in *Figure 10.3* (p. 75).

Figure 10.3 Rotation Axis
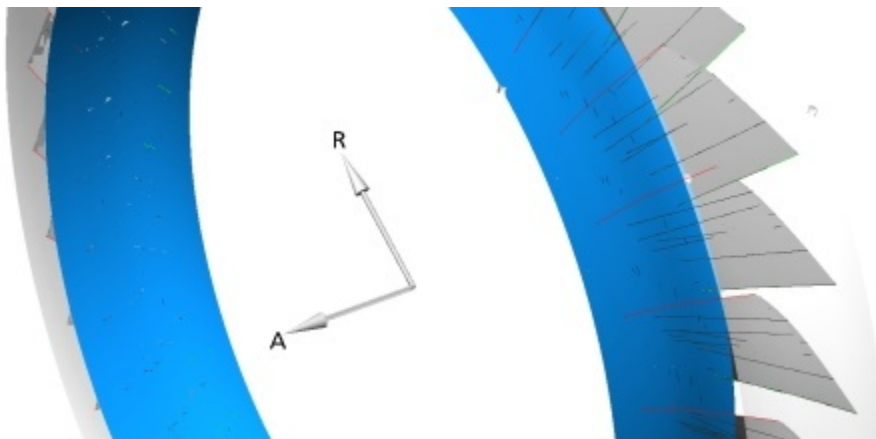
The visibility of the `Machine Data` object is off by default. For details on setting visibility, see *Visibility Check Box* (p. 4).

10.3.2. The Hub and Shroud Objects

The hub is the surface of the machine closest to the axis of rotation. It defines the inner fluid flow surface. To define the machine component's hub, edit the `Hub` object from the selector or click *Edit Hub* ⬛.

The shroud is the surface of the machine farthest from the axis of rotation. It defines the outer fluid flow surface. To define the machine component's shroud, edit the `Shroud` object from the selector or click *Edit Shroud* ⬛.

The hub and shroud can be defined only after the machine data has been defined, although all of these objects can be defined in one step. For details, see *Steps to Create a Mesh* (p. 71).

10.3.2.1. Data Hub and Data Shroud Tabs

10.3.2.1.1. Coordinate System and Hub File Definition

Set **Coordinates**, **Angle Units**, and **Length Units** according to the data in the hub/shroud file. The hub/shroud file must contain data points which define the hub/shroud curve. To specify a hub/shroud file, set **File Name** using a path relative to the working directory. You can click the *Browse* 📂 icon to select a curve file using a browser.

The hub/shroud curve runs upstream to downstream and must extend upstream of the blade leading edge and downstream of the blade trailing edge. The points must be listed in the file line by line, in free-format ASCII style, in order from upstream to downstream.

For example:

```
-1.0  0.0  1.0
 2.0  0.0  1.0
```

No names or labels may be present in the file.

Note

If the hub/shroud file was used in earlier versions of CFX-TurboGrid, you can use the file again in this release without any modifications. However, any data in the fourth field formerly used for parametric values is now ignored.

The **Curve Type** setting has the following options for defining the type of hub/shroud curve.

- `Bspline`, the default, means that a smooth curve is interpolated using the points listed in the hub/shroud file. This method may be necessary if the hub/shroud curve is defined with a small number of points.

- `Piece-Wise Linear` means that the points listed in the hub/shroud file are connected to one another with straight lines.

10.3.2.1.2. Curve or Surface Visibility

By default, the hub/shroud surface is visible while the hub/shroud curve is not visible. To see the curve, the following steps could be taken in the **Data Hub** tab:

1. Select the **Curve or Surface Visibility** > **Show Curve** check box.

2. Clear the **Curve or Surface Visibility** > **Show Surface** check box.

3. Click **Apply**.

4. Right-click the `Hub/Shroud` object and select **Render (Properties)** > **Edit Options**.

5. On the **Render** tab, select the **Draw Lines** check box.

6. Click **OK**.

10.3.2.1.3. Reread Button

If the file that defines the hub/shroud has been modified since it was last loaded, clicking the **Reread** button will cause the hub/shroud data to be reloaded. The `Blade Set` object and individual blade objects also have this feature.

10.3.2.2. Transform Tab

This tab contains specifications for rotational and translational transforms that adjust the geometry. The settings on this tab adjust the hub/shroud geometry only. The `Blade Set` object and individual blade objects can also have their coordinates transformed using the corresponding tabs of those objects.

10.3.2.2.1. General Rotation

Specify an axis via two points and a rotation angle about the axis.

10.3.2.2.2. Translation

Specify the direction and distance to translate the geometry. The direction may be any one of: `Machine Axis`, `X Axis`, `Y Axis`, `Z Axis`, or `Vector` (any specified direction).

10.3.2.2.3. Axial Rotation

Specify the angle to revolve the geometry about the machine axis.

10.3.3. The Blade Set Object and Blade Objects

To define the machine component's blades as a set, edit the `Blade Set` object from the object selector or click *Edit Blade Set* . To define or modify a single blade, edit the corresponding object stored under the `Blade Set` object. You can only define the blade set or individual blades after the hub and shroud have been defined, although all of these can be defined in one step. For details, see *Steps to Create a Mesh* (p. 71).

The count and naming of the individual geometry (and topology, and mesh data) objects for each blade in the blade set are managed automatically by ANSYS TurboGrid. If a profile file containing multiple blades is specified for the `Blade Set` object (whether directly or indirectly, e.g., by loading a BladeGen `.inf` file or `.cfg` file), then a blade object will be created for each set of data in the file and named according to the name specified in the file (unless a name is not specified, in which case a generic name of the form "Blade *n*" will be given, where *n* is an integer starting at 1).

The individual blade objects contain a profile curve file specification and a setting to indicate which single blade in the file applies; numbering starts at zero.

10.3.3.1. Blade Tab

10.3.3.1.1. Apply Settings To All Blades Check Box

When **Apply Settings To All Blades** is selected, `Blade Set` settings overwrite the corresponding blade object settings. If **Apply Settings To All Blades** is cleared, the settings that apply to all blades are disabled.

If you change a setting for a blade object, and the `Blade Set` object has a corresponding setting, then the **Apply Settings To All Blades** check box will be cleared to help prevent accidentally losing the blade-specific settings in case you reapply the `Blade Set` object.

10.3.3.1.2. Coordinate System and Blade File Definition

10.3.3.1.2.1. File Name

Set **File Name** to the name of the blade set file. To open a file selector, click *Browse* 📂.

The blade file must contain data points which define the blade profile (or rib) curves and should have the file extension `.curve` or `.crv`.

The profile points must be listed line by line, in free-format ASCII style in a closed-loop surrounding the blade.

A minimum of two profiles are required in the blade file: one which lies close to the hub surface and one which lies close to the shroud surface. The profile is not required to lie exactly on the surface. If it lies between the hub and shroud surfaces, it must be within 8% of the span from the surface. If a profile lies outside of the passage, its distance from the surface has no maximum limit.

Profiles can be used to define tips. For details, see *The Hub Tip and Shroud Tip Objects* (p. 86). Profiles are allowed, and sometimes required, beyond a tip.

The profiles must be listed in the file in order from hub to shroud. Individual profile datasets are separated by a line beginning with the "#" character. Any text following the "#" character is used as the name associated with the subsequent profile.

For example:

```
# Hub Profile
0.0   0.0   1.0 le
1.0   0.0   1.0
1.0   1.0   1.0 te
0.0   1.0   1.0
0.0   0.0   1.0
# Intermediate Blade Profile
0.0   0.0   2.0 le
1.0   0.0   2.0
1.0   1.0   2.0 te
0.0   1.0   2.0
0.0   0.0   2.0
# Shroud Tip Profile
0.0   0.0   2.75 le
1.0   0.0 2.75
1.0   1.0 2.75 te
0.0   1.0 2.75
```

```
0.0   0.0 2.75
# Shroud Profile
0.0   0.0   3.0 le
1.0   0.0   3.0
1.0   1.0   3.0 te
0.0   1.0   3.0
0.0   0.0   3.0
```

There is no restriction on the number of profiles or the number of data points in a profile dataset. However, as noted above, a minimum of two profiles is required.

10.3.3.1.2.2. Coordinates, Angle Units, and Length Units

Set **Coordinates**, **Angle Units**, and **Length Units** according to the data in the blade file.

10.3.3.1.3. Geometric Representation

ANSYS TurboGrid generates blade surfaces using a two-step process:

1. Curves are generated.

2. A surface is created by lofting across the set of curves (that is, sweeping from one curve to the next).

The **Geometric Representation** settings control how these steps are performed.

10.3.3.1.3.1. Method

The **Method** setting allows two preset methods, and one general method, for controlling the settings that govern the geometric representation of blade surfaces (that is, the **Lofting**, **Curve Type**, and **Surface Type** settings, described shortly).

The available **Method** options are:

- `BladeModeler`

  The `BladeModeler` option sets **Lofting** to `Spanwise`, **Curve Type** to `Bspline`, and **Surface Type** to `Ruled`.

- `Flank Milled`

  The `Flank Milled` option sets **Lofting** to `Streamwise`, **Curve Type** to `Piece-wise Linear`, **Surface Type** to `Ruled`.

- `Specify`

  The `Specify` option makes the **Lofting**, **Curve Type**, and **Surface Type** settings available for direct specification.

The following rules are followed by ANSYS TurboGrid for selecting the geometric representation method:

- If you load a blade from a BladeModeler `.inf` file, the `BladeModeler` option will be selected automatically.

- If the `BladeModeler` option is selected and there are only 2 blade profile curves (for the applicable blade), the selected option will change to `Flank Milled` (which is equivalent, in this case).

- If the `Flank Milled` option is selected and there are more than 2 blade profile curves (for the applicable blade), the selected option will change to `BladeModeler` and a warning message will be issued.

10.3.3.1.3.2. Lofting

The direction in which lofting occurs is set by the **Lofting** setting. The available **Lofting** options are:

*   `Streamwise`

    The curves that are swept in the process of streamwise lofting are formed by connecting corresponding points between adjacent blade profiles. For this method of lofting, you must ensure that:

    –   each blade profile has the same number of points

    –   the points are ordered in the same direction

    –   the first, nth, and last point of each blade profile are at similar locations around the blade

*   `Spanwise`

    The curves that are swept in the process of spanwise lofting are the blade profile curves.

Figure 10.4 (p. 81) shows how spanwise lofting and streamwise lofting differ.

Figure 10.4 Spanwise Lofting versus Streamwise Lofting
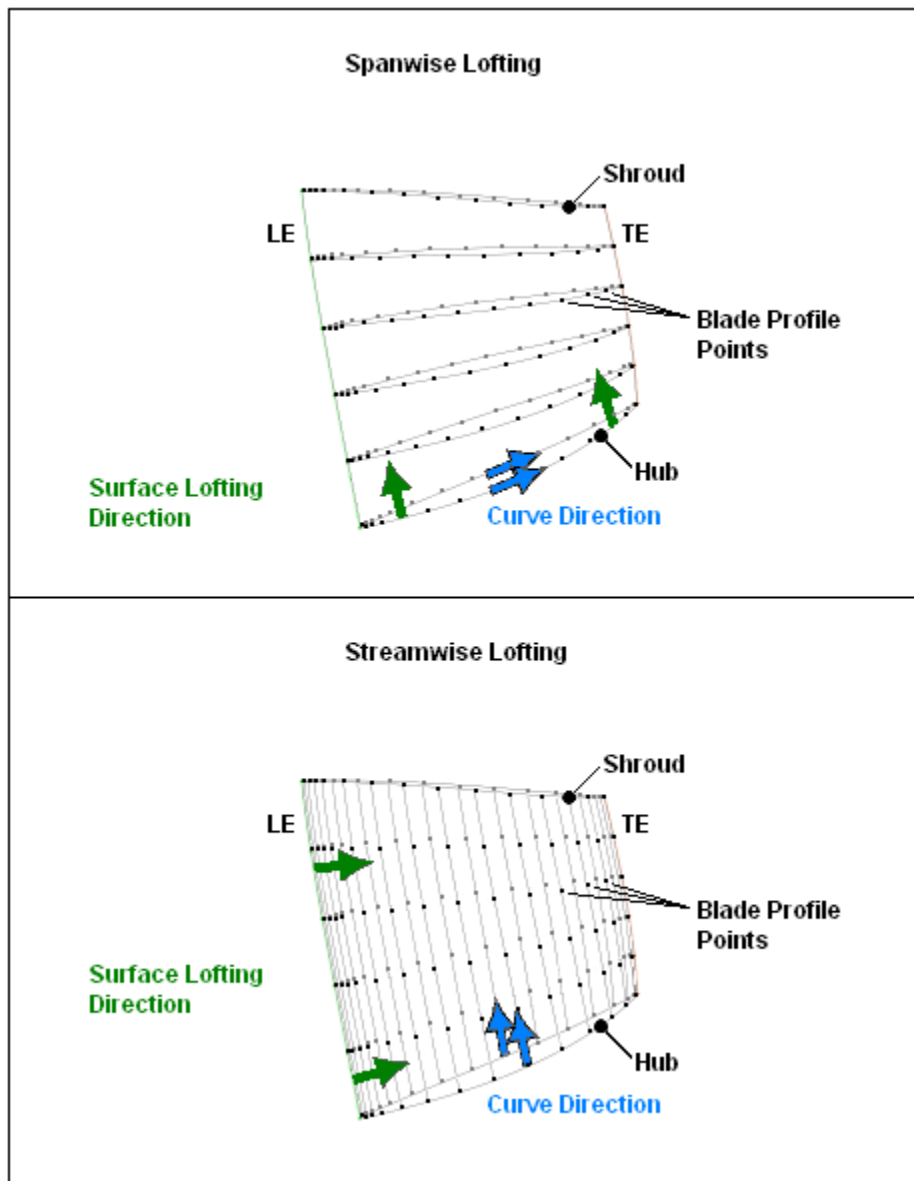
The appropriate lofting setting depends on how the geometry was produced. For example, Blade Modeler can produce blades via streamwise or spanwise lofting. The appropriate setting is stored in the `BladeGen.inf` file. As another example, a flank-milled blade should be generated and interpreted using two blade profiles (one at (or near) the hub and one at (or near) the shroud) with streamwise lofting.

Note

The method of "sweeping" through the curves in the lofting process is controllable via the **Surface Type** setting, which can be set to `Bspline` (default) or `Ruled`. For details, see *Surface Type* (p. 82). For streamwise lofting, the `Bspline` method may cause unexpected results. In this case, the workaround is to use the `Ruled` method.

10.3.3.1.3.3. Curve Type

The set of curves that are used to loft the blade surface may be constructed in one of two ways, as determined by the **Curve Type** setting. The available **Curve Type** settings are:

- `Piece-wise linear`

  The `Piece-wise linear` setting causes the points listed for each profile in the blade file to be connected to one another with straight lines. You should use this setting when you have a large number of points (1000 or more) in a profile. This setting is always used when **Method** is set to `Flank Milled`.

- `Bspline`

  The `Bspline` setting results in the interpolation of a smooth curve for each profile using points listed in the blade file. This setting may be necessary if the profile curves are defined with a small number of points. This setting is discouraged when you have a large number of points (1000 or more) in a profile, due to undesirable effects on the shape of the spline, and due to limitations of the software. The `Bspline` setting is the default when **Method** is set to `BladeModeler` or `Specify`.

10.3.3.1.3.4. Surface Type

The manner in which the set of curves is lofted is controlled by the **Surface Type** setting. The available **Surface Type** options are:

- `Ruled`

  A ruled surface is created by sweeping along linear paths that connect one curve to a corresponding place on the next curve. The `Ruled` method for surfaces is similar to the `Piece-wise linear` method for curves, but is done in 2 dimensions instead.

- `Bspline`

  A B-spline surface is created by sweeping along curvilinear paths that connect one curve to a corresponding place on the next curve. The `Bspline` method may be necessary if the blade is defined with a small number of profile curves.

10.3.3.1.4. Leading and Trailing Edge Definitions

ANSYS TurboGrid uses an algorithm to determine the location of the leading edge curve(s) and trailing edge curve(s) and in most cases gets the correct locations (double edges are possible in the case of cut-off or square edges). In some instances, the location of one or more of the points may need to be adjusted. See *Leading Edge* (p. 84) and *Trailing Edge* (p. 85) for information about adjusting the leading and trailing edges.

If you know which points on each profile should lay on the leading and trailing edge curves, you can optionally designate them in the blade file using a fourth field. This ensures the correct positioning of the leading and trailing edge curves in the beginning. The leading edge point on a profile should have **le1** (or **le2** for the second leading edge in the case of a double leading edge) in the fourth field and the trailing edge point on a profile should have **te1** (or **te2**) in the fourth field. These points are then guaranteed to be listed in the leading edge or trailing edge objects. For an example of the format of a blade file containing leading and trailing edge designations, try the following:

1. Load a blade file into ANSYS TurboGrid.

2. Optionally change the type of leading or trailing edge between single and double (using the **LE Type** and **TE Type** settings which are described below).

3. Save the blade file and examine it in a text editor, noting the **le1**, **te1** (and, for double edges **le2** and **te2**) entries in the fourth field.

10.3.3.1.4.1. Cut-off or square

This is an option that may be used to create a flat leading/trailing edge between two edge curves. See *Figure 10.5* (p. 83) for an example of a "cut-off or square" trailing edge.

Figure 10.5 Trailing Edge with Pair of Edge Curves
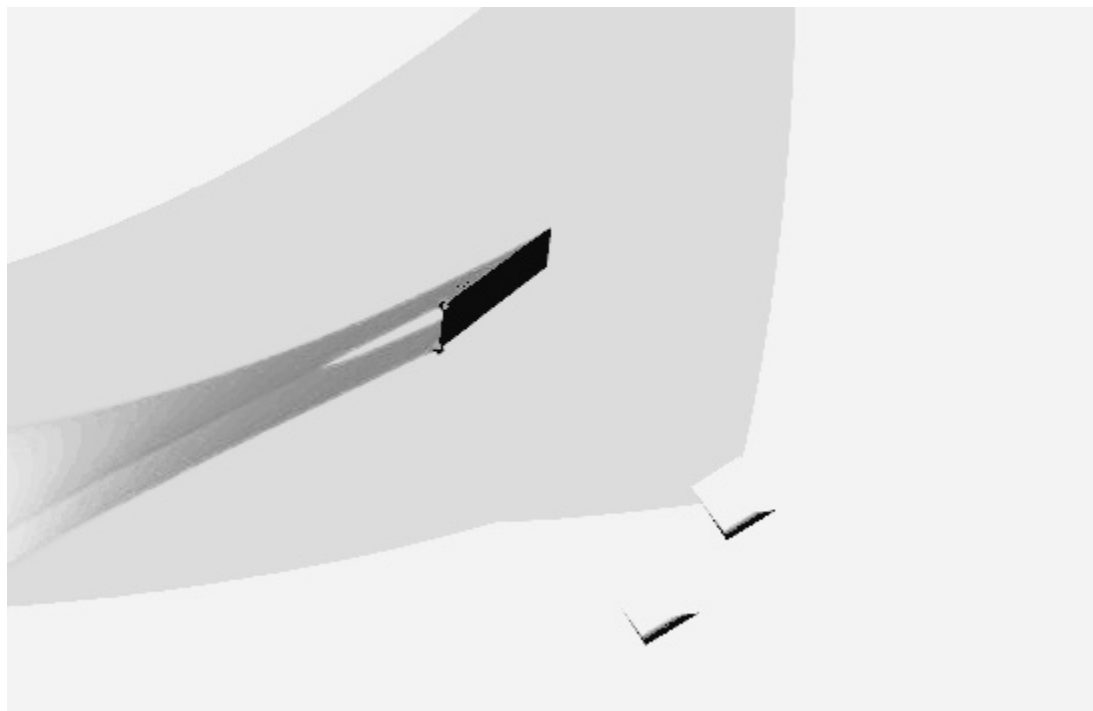
The **Cut-off or square** option is intended for use on blade profiles that have discontinuities in slope at the edges. Use of this option on rounded or blunt leading/trailing edges may result in edge curves that are not placed in optimal locations (i.e., further adjustments may be necessary).

10.3.3.1.4.2. Line of rotation on hub and shroud

This option, available in the `Blade Set` object, generates a line of rotation from each cut-off point of a "cut-off or square" blade edge. A "cut-off or square" edge having the same axial and radial coordinates at both cut-off points will have lines of rotation that coincide. In this case, the **Line of rotation on hub and shroud** option will divide the passage mesh, hub, and shroud at the cut-off edge by a surface of revolution about the machine axis. Additional 2D and 3D regions will be created as a result of this division of the passage. This allows, for example, the mesh to be used in a simulation where the portion of the passage mesh surrounding the blade rotates, while the rest of the passage mesh is stationary.

This option may still be used if the cut-off points do not have the same axial and radial coordinates, although the interface between the divisions will not be a surface of revolution.

As a result of the division of the passage, the topology (see *Topology* (p. 90)) will also be affected. For details, see *Topology with Cut-off or Square Leading or Trailing Edges* (p. 112).

10.3.3.1.5. Bias of Blade towards High Periodic

The **Bias of Blade towards High Periodic** > **Factor** setting, available in the `Blade Set` object, controls the location of the periodic surface in the passage, with respect to the first blade of the blade set. Set **Factor** to a number somewhere between `0` and `1`. A higher value brings the high periodic surface closer to the first blade (causing the low periodic surface to move further from the blade). A lower value brings the low periodic surface closer to the blade. The main purpose of this feature is to avoid having a periodic surface too close to (esp. touching), the blade surface. Such a situation can occur for some blade geometries.

10.3.3.1.6. Curve or Surface Visibility

The visibility of curves and surfaces for a blade object works in the same way as for the hub. For details, see *Curve or Surface Visibility* (p. 77).

10.3.3.1.7. Reread Button

If the file that defines the blade profile has been modified since it was last loaded, clicking the **Reread** button will cause the blade profile data to be reloaded. The hub and shroud objects also have this feature.

10.3.3.1.8. Save & Load Button

If a leading or trailing edge point has been moved, the **Save & Load** button appears in place of the **Reread** button. Selecting this button causes the modified blade profile data to be saved to a profile file (you will be prompted for the name), then reloaded (from the same file) in order for the new geometry to take effect. The leading edge and trailing edge objects also have this feature.

10.3.3.2. Transform Tab

The **Transform** tab contains specifications for rotational and translational transforms that adjust the geometry. The settings on this tab adjust the blade geometry only. The hub and shroud can also have their coordinates transformed using the corresponding tab of those objects.

10.3.3.2.1. General Rotation

Specify an axis via two points and a rotation angle about the axis.

10.3.3.2.2. Translation

Specify the direction and distance to translate the geometry. The direction may be any one of: `Machine Axis`, `X Axis`, `Y Axis`, `Z Axis`, or `Vector` (any specified direction).

10.3.3.2.3. Axial Rotation

Specify the angle to revolve the geometry about the machine axis.

10.3.3.3. Leading Edge

The leading edge curve is the most upstream part of the blade. To modify the blade's leading edge, select the leading edge object from the object selector. The leading edge can only be defined after the blade has been defined.

10.3.3.3.1. Leading Edge Tab

10.3.3.3.1.1. Curve

The leading edge curve appears green in the viewer window and each point on the curve (one for each blade profile, or rib, curve) is marked with an octahedron symbol. The points are listed from hub to shroud. Edit the individual leading edge point locations to create the correct leading edge curve for the blade.

The quality of the mesh is dependent on the accuracy of the leading edge curve. In most cases, the initial placement of leading edge points is adequate. However there are some cases when some points may need to be moved.

Any change to the leading edge changes the blade surfaces, which changes the periodic surfaces as well as the hub and shroud surfaces. In order to avoid the delays associated with recreating the entire geometry after any modification to the leading edge, another step is required. The blade must be saved after the modifications to the leading edge are complete. For details, see *Save Blade As Command* (p. 16). The saved blade file contains data in the fourth field to designate the leading edge points. To complete the leading edge modification, the blade object must be updated with this new blade file. For details, see *The Blade Set Object and Blade Objects* (p. 77).

There are several steps required to edit the location of a point on the leading edge curve.

1. Select the point to edit by clicking on it in the list of points. Alternatively, you can pick the point to edit from the viewer after clicking *Select* ![icon] (on the viewer toolbar).

2. Click in any coordinate widget and then click the new location for the point in the viewer. The coordinates of the picked point are displayed in the object editor. Alternatively, type the coordinates of the new location for the point or use the embedded sliders. The units of the coordinates are the base units or solution units. For details, see *Units* (p. 75).

3. Click **Apply** to save the new location of the current point.

Picking Mode can also be used to select and translate leading edge points in the viewer. For details, see *Selecting and Dragging Objects while in Viewing Mode* (p. 55). After moving the leading edge point, it will snap to the nearest point on the current profile curve.

If the leading edge points were designated in the blade file, it may not be necessary to adjust them since, in that case, the leading edge curve may already be acceptable. For details, see *Coordinate System and Blade File Definition* (p. 78).

10.3.3.3.1.2. Save & Load Button

If a leading or trailing edge point has been moved, the **Save & Load** button appears. Clicking this button causes the modified blade profile data to be saved to a profile file (you will be prompted for the name), then reloaded (from the same file) in order for the new geometry to take effect. The blade and trailing edge objects also have this feature.

10.3.3.4. Trailing Edge

The trailing edge curve is the most downstream part of the blade. The trailing edge curve appears red in the viewer window. Because of the similarity of the trailing edge with the leading edge, see *Leading Edge* (p. 84) for details.

10.3.4. The Hub Tip and Shroud Tip Objects

The shroud tip is the portion of the blade surface that exists as a result of the blade not extending all the way to the shroud. By default, the blade extends all the way to the shroud (i.e., no shroud tip exists). To create a shroud tip, edit the `Shroud Tip` object after the blade has been defined.

The hub tip is similar to the shroud tip, except that it is at the hub end of the blade. The settings for the hub tip are stored in the `Hub Tip` object.

Note

It is possible to specify a hub tip and a shroud tip for the same blade.

Note

Using a tip clearance on one blade but not another is possible by changing the CCL for individual blades, but could produce an invalid mesh near the tip of any blade that does not have tip clearance.

10.3.4.1. Hub Tip and Shroud Tip Tabs

The following sections describe the object editor settings for the `Hub Tip` and `Shroud Tip` objects:
10.3.4.1.1. Override Upstream Geometry Options
10.3.4.1.2. Clearance Type

10.3.4.1.1. Override Upstream Geometry Options

Selecting the **Override Upstream Geometry Options** check box enables you to change the hub/shroud tip settings for the case of TurboGrid running in Workbench with an upstream geometry provided by BladeEditor. If you do not select the **Override Upstream Geometry Options** check box, the hub/shroud tip settings are displayed but are not editable, being controlled by the upstream BladeEditor geometry.

Note

In Release 14.0, only the shroud tip geometry can be defined in BladeEditor.

10.3.4.1.2. Clearance Type

The **Tip Option** setting has the following options:

- `None`

  This option causes the blade to extend from hub to shroud.

- `Constant Span`

  This option creates a tip at the span location that you specify. A span of 0.0 represents the hub and a span of 1.0 represents the shroud. The span value for the hub tip must be between 0 and 0.5; the span value for the shroud tip must be between 0.5 and 1.0.

- `Normal Distance`

  This option creates a tip so that the gap distance is whatever you specify.

- Variable Normal Distance

  This option creates a tip so that the gap distance varies from leading edge to trailing edge using values that you specify. The gap distance varies linearly with m-coordinate as calculated on the hub (for a hub tip) or shroud (for a shroud tip).

- Profile Number

  This option creates a tip at the blade profile that you specify. The available profiles are defined in the blade file. For details, see *The Blade Set Object and Blade Objects* (p. 77). The profile closest to the hub is profile 1.

  Note

  The blade surface and resulting mesh can be adversely affected when a blade shroud tip is defined by the second-last profile, and the last profile has a shape that is inconsistent with where the blade would exist if the blade were extended in the spanwise direction without the last profile. To avoid this problem, you can either ensure that the last profile is consistent with the second last profile, or manually remove the last profile from the curve file. A similar statement applies for the hub end of a blade having a hub tip.

10.3.5. The Low Periodic and High Periodic Objects

The low periodic surface extends from hub to shroud and inlet to outlet along the side of the mesh that has the lowest Theta values (The direction of Theta is determined by the machine rotation axis and the right-hand rule.). The high periodic surface is on the opposite side of the passage from the low periodic surface (i.e., on the high-Theta side).

The periodic surfaces can be modified only after the blade has been defined.

10.3.5.1. Data Tab

Set **Method** to one of the following:

- Automatic

  This option creates the periodic surface automatically. The shape of the periodic surface is based on the geometry of the adjacent blades. The position of the automatically-generated periodic surface is governed by the **Bias of Blade towards High Periodic** setting found in the `Blade Set` object, as well as the **# of Bladesets** setting of the `Machine Data` object.

- From File

  This option allows you to load a periodic surface that was saved to a file. The **Length Units** setting controls the units that will be used to interpret the coordinate data in the file. The **Rotation Angle** setting transforms the loaded periodic surface by rotating by the specified angle about the machine rotation axis. Note that the **# of Bladesets** setting of the `Machine Data` object will not change automatically due to changes to the **Rotation Angle** setting.

10.3.5.2. Rendering Properties

To modify the color or rendering of the low or high periodic surface, right-click the `Low Periodic` or `High Periodic` object (respectively), then select **Render (Properties)** > **Edit Options**.

10.3.6. The Inlet and Outlet Objects

The inlet region of the passage is controlled by the `Inlet` object. The outlet region is controlled by the `Outlet` object.

You can modify the `Inlet` and `Outlet` objects only after the blade has been defined.

Note

If a 3D inlet (or outlet) region is added, the inlet (or outlet) object becomes the interface between this 3D region and the topology. For details, see *Inlet Domain and Outlet Domain Settings* (p. 120).

10.3.6.1. Inlet Tab or Outlet Tab

10.3.6.1.1. Control Angle

By default, ANSYS TurboGrid automatically adjusts the geometry upstream and downstream of the blade. You can override the angle of the periodic surface relative to the rotation axis. This can be useful for improving the quality of the mesh and/or guiding the mesh in the general flow direction.

10.3.6.1.2. Initialize points to full extents of Hub or Shroud curves

The **Initialize points to full extents of Hub/Shroud curves** option is only visible before the hub, shroud, and blade are loaded. When selected, all inlet points will move to the very beginning of the passage (as far away from the blade as possible). Similarly, all outlet points will move to the very end of the passage. Such a feature can be useful when the shape of the blade is likely to intersect the default placement of the inlet points.

10.3.6.1.3. Automatically Generate Intermediate Points

This option is only available before the hub, shroud, and blade are loaded. It is on by default. When selected, a set of intermediate points will be generated between the low hub point and low shroud point upon loading the geometry (hub, shroud and blade). The purpose of these points is to control the meridional shape of the inlet surface. The automatic generation process aims to add an appropriate number of points, placed at locations that follow the shape of the leading edge of the blade.

10.3.6.1.4. Curve

The inlet/outlet curve consists of a set of points that guide the meridional shape of the inlet end of the passage mesh as a function of the local spanwise direction. By default, a set of points is automatically generated (see *Automatically Generate Intermediate Points* (p. 88)).

The inlet/outlet curve can be defined by creating, modifying and deleting points in the list of points. There must be a point at both the hub and shroud while additional points are optional. The points are connected linearly and must be listed in order from hub to shroud, or they will be ignored.

A point can be selected by clicking on it in the list of points or by picking it in the viewer window. For details on picking points in the viewer, see *Selecting and Dragging Objects while in Viewing Mode* (p. 55). The available icons become active when a point is selected. Alternatively, you can right-click an existing point in the list of points to see the same options. Each icon and its function is described in the following table:

Icon	Description
	New adds a point in the list of points below the selected point. The initial location of the new point on the Inlet is between the locations of the selected point and the following point in the list of points. You cannot create a new point after the low shroud point.
	Generate intermediate points removes all points except for the low hub point and the low shroud point, then automatically generates a set of zero or more points between them.
	Delete removes the selected point from the list of points. You cannot delete the low hub point or the low shroud point.
	Read from File opens a load file window. Select the file with the inlet points and click **Open**.

You can modify the location of the selected point by:

1. Typing in the coordinates (axial, radial, or both, depending on the selected **Method** option), or

2. Selecting and dragging the point in the viewer using the left mouse button. This can be done as for a control point. For details, see *How to Select and Move a Control Point* (p. 129).

Set **Method** to one of the available options:

- `From A and R`

- `Set A`

- `Set R`

Regardless of the method used to modify the location of the selected point, the location is restricted. The low hub point snaps to the line of intersection between the hub and low periodic surfaces. The low shroud point snaps to the line of intersection between the shroud and low periodic surfaces. Use the OUTLINE object as a guide when dragging the points. Any other points between the hub and shroud snap to the low periodic surface and are not constrained in the spanwise or streamwise directions. Inlet points cannot be moved further downstream on the low periodic surface than a position level with the leading edge of the blade. After you have clicked **Apply**, or dragged a point, the **Selected Point** setting will show the actual position to which the point has moved.

> **Note**
>
> The points are defined using the A-R coordinate system as opposed to the X-Y coordinate system.

The visibility of the inlet (or outlet) points is controlled by the **Point Visibility** check box.

10.3.6.2. Using an Adjacent Stage to Define the Inlet/Outlet Stage Interface

You can automatically adjust a stage interface (i.e., the inlet/outlet points that shape the end of the passage mesh) by loading a blade from the adjacent stage. To do this:

1. Click the **Interface** button in `Inlet` or `Outlet` editor, depending on the location of the next stage.

   The **Open Blade File** dialog box appears.

2. Select and open the file that defines the blade in the adjacent stage.

Note that the hub and shroud curves are required to extend past this blade.

ANSYS TurboGrid will then do the following:

- Adjust the stage interface location according to the shape of the new blade.
- Add a graphical representation of the new blade to the `Inlet` or `Outlet` object, as applicable.
- Turn on the visibility of the `Inlet` or `Outlet` object, as applicable.
- Make the **Points generated using adjacent stage** check box available, and initially selected.

The **Points generated using adjacent stage** option overrides the **Curve** settings, forcing them to be computed based on the blade from the adjacent stage.

10.3.6.3. Using Stage Interfaces

In order to produce a proper stage interface, you should follow these guidelines:

- Use the same hub curve for both stages, and the same **Curve Type** setting: `Bspline` or `Piece-Wise Linear`. The hub curve must extend through both stages to do this.

  The same applies to the shroud curve.

  As a less preferable alternative, you can use hub curves for each stage. The curves should meet at a point, and not overlap. When using this method, there is a risk of a discontinuity in slope where the curves meet.

- Use the same interface points for stage interfaces. Do this by saving the interface (inlet or outlet geometry object, as applicable), then loading the interface for the adjacent stage.

10.3.7. The Outline Object

The outline is a group of curves on the outer extents of the geometry. The outline includes the full extent of the hub and shroud read from the files, independent of the inlet and outlet locations. To modify the color or rendering of the geometry outline, right-click the `OUTLINE` object, then select **Render (Properties)** > **Edit Options**. You can only modify the outline after the blade has been defined.

10.4. Topology

The topology is a structure of blocks that acts as a framework for positioning mesh elements. Topology blocks represent sections of the mesh that contain a regular pattern of hexahedral (hex) elements. They are laid out adjacent to each other without overlap or gaps, with shared edges and corners between adjacent blocks, such that the entire domain is filled. By using topology blocks to control the placement of hex elements, a valid hex mesh can be generated to fill a domain of arbitrary shape. The topology is invariant from hub to shroud and is viewed/edited on 2-D layers which are located at various spanwise stations (see *Layers* (p. 122)).

The topology blocks can be arranged in a regular (structured) pattern, an irregular (unstructured) pattern, or in a pattern consisting of structured patches and unstructured patches. The choice of which approach should be followed should be based on whichever method minimizes the maximum skew of the topology blocks, since the skew in the hex elements of the mesh is directly related (differs only because of mesh smoothing). The topology should then be investigated at various layers (especially the hub and shroud layers) to check its quality since the mesh quality is directly dependent.

Note

> To visualize the topology on a layer, turn on the visibility using the visibility check box for that layer in the object selector and ensure that at least one topology visibility setting for the layer is turned on. The **Topology Visibility** setting controls the visibility of the yellow line segments that outline the topology block edges. The **Master Topology Visibility** setting controls the visibility of the violet line segments that outline the master topology edges. For details, see *Topology Visibility* (p. 126).

A key feature of ANSYS TurboGrid is the visibility of the surface mesh on the topology. As you adjust the topology, ANSYS TurboGrid adjusts the surface mesh in real time so that the true effect of topology changes is visible. To help identify problem areas in the surface mesh before you generate the full 3D mesh, you can visualize mesh statistics on the layers.

Topology blocks generally contain the same number of mesh elements along each side. The mesh elements vary in size across topology blocks in a way that produces a smooth transition[1] within and between blocks. This is accomplished by shifting the nodes ("node biasing"[2]) toward, or away from, certain block edges. The topology blocks are positioned by default so that the mesh element sizes vary as smoothly as possible, given the constraints. The following changes to the topology object settings can influence the variation in mesh element edge ratios:

- Changing the number of topology blocks along certain paths (e.g., across the blade passage or from a blade's leading edge to the inlet). For details, see *Blade Blocks Calculation* (p. 108), and *Blade to Periodic/Inlet/Outlet Surface Block Distribution* (p. 109).

- Moving control points which control the position of block edges on a given layer. For details, see *Master Control Points* (p. 128) and *Added Control Points* (p. 131).

ANSYS TurboGrid features two different topology sets: ATM Optimized and Traditional Topologies. ATM Optimized is the default setting and allows high-quality meshes to be easily generated. Traditional Topologies give you more control over mesh generation. Both sets are discussed in the following topics:

- *ATM Optimized Topology* (p. 91)
- *Traditional Topologies* (p. 99)

10.4.1. ATM Optimized Topology

This chapter provides a description of the Automatic Topology and Meshing (ATM Optimized) feature. ATM Optimized topology is the default setting in ANSYS TurboGrid and an alternative to the topologies mentioned in *The Topology Set Object and Topology Objects* (p. 99). This topology type enables you to create high-quality meshes with minimal effort; there is no need for control point adjustment.

Important

> ATM is not applicable for blades that have square leading or trailing edges.

The basic steps to using the ATM Optimized feature are as follows:

---

[1]The smooth transition is not preserved if ordinary control points are moved. However, the smooth transition is preserved if master control points are moved.

[2]Node biasing is an important aspect of mesh generation because it helps to reduce the number of mesh elements.

1. Load the blade geometry and set the geometry parameters (for example, inlet/output points) as necessary.

2. Open `Topology Set` and set **Topology Definition** > **Placement** to **ATM Optimized**.

3. Click **Apply**.

4. Open `Mesh Data`.

5. On the **Mesh Size** tab, set **Method** to `Target Passage Mesh Size` or `Global Size Factor` as appropriate.

6. Click **Apply**.

7. Right-click **Topology Set** and clear **Suspend Object Updates**.

   ANSYS TurboGrid computes the topology and mesh for the hub and shroud layers based on the current default mesh size.

8. Click *Create Mesh* 🖻 to generate the volume mesh.

Afterwards you can view the 2-D mesh on a layer and view the layer statistics, just as you would with one of the traditional control point topologies. The Master Topology (magenta) shows the master block topology curves. The Topology (gold) shows the background mesh, which is typically useful only for developer debugging. The Refined Mesh (brown) shows the final mesh for the layer; this is the most useful of the three.

The following topics are discussed:

10.4.1.1. Changing the Overall Mesh Size

The ATM Optimized method automatically computes a default mesh and sets the base mesh dimensions. Each unique mesh dimension has an edge refinement factor that is multiplied by the base mesh dimension and a global size factor to determine the final size of the mesh dimension. Local mesh size adjustments can be made by adjusting the local edge refinement factors. The overall mesh size can be changed by using one of the following methods on the **Mesh Size** tab in the **Mesh Data** object editor:

• Set **Method** to `Global Size Factor` and set **Size Factor**.

   This option directly sets the Global Size Factor that is used to multiply all the individual edge element counts for the base (default) mesh. Increasing this factor will increase the overall mesh size, and decreasing this factor will decrease the overall mesh size, but the change is not linear. In this mode, if you change the spanwise mesh size or the boundary layer refinement, or make any local edge refinements to the mesh, the Global Size Factor will stay fixed and the overall mesh size may change. This factor, when used with proportional refinement, can be used to scale the mesh size for a mesh refinement study.

• Set **Method** to `Target Passage Mesh Size` and set **Node Count**.

This option enables you to choose the approximate mesh size (in nodes), and lets ANSYS TurboGrid compute the mesh dimensions automatically, including the size factor. In this mode, if you change the spanwise mesh size or the boundary layer refinement, or make any local refinements to the mesh, ANSYS TurboGrid will adjust the mesh size factor (**Edge Refinement Size Factor**) automatically to achieve the desired target mesh size.

10.4.1.2. Changing the Boundary Layer Resolution near the Blade

Within the O-grid block around the blade, ANSYS TurboGrid uses a constant expansion rate to distribute the elements normal to the blade surface, and to assure a smooth transition to the passage mesh. The expansion rate is determined by the number of elements normal to the blade, the 'base mesh distribution' and the size of the elements in the adjacent passage mesh. While you cannot directly specify the expansion rate, you can control the expansion rate by adjusting the number of elements normal to the blade.

When the topology is created, ANSYS TurboGrid creates a `Boundary Layer Control` object under the `Mesh Data` object in the object selector that provides direct control of the mesh resolution in the O-grid and also displays the base and actual Block Edge Split Counts for the O-grid. The O-grid element count normal to the blade is defined by the local Edge Refinement Factor times Global Size Factor times the base mesh dimension for the O-grid. ANSYS TurboGrid automatically computes the base mesh dimension, and this remains fixed for a given geometry/topology. Increasing or decreasing the Edge Refinement Factor will increase or decrease the mesh resolution in the O-grid, respectively.

The Edge Refinement Factor can be specified directly, or it can be computed by ANSYS TurboGrid. These options are described as follows:

• Use `Proportional to Mesh Size` — This option lets ANSYS TurboGrid compute the Edge Refinement Factor as the Proportional Offset Factor Ratio times the Global Size Factor. This method maintains a similar O-grid expansion rate when the Global Size Factor is changed. On the **Mesh Size** tab of the `Mesh Data` object editor, set **Boundary Layer Refinement Control** > **Method** to `Proportional to Mesh Size`, specify the **Factor Base** or **Factor Ratio**, and then click **Apply**. The Edge Refinement Factor is computed as the sum of **Factor Base**, **Factor Ratio** and **Global Size Factor**. For both **Factor Base** and **Factor Ratio**, increasing or decreasing the value will effectively increase or decrease the expansion rate, respectively.

• Use `Edge Refinement Factor` — This option lets you directly specify the Edge Refinement Factor. This method does not maintain a similar O-grid expansion rate when the Global Size Factor is changed, and therefore the Edge Refinement Factor may need to be adjusted when the mesh size is changed. On the **Mesh Size** tab of the `Mesh Data` object editor, set **Boundary Layer Refinement Control** > **Method** to `Edge Refinement Factor`. Specify **Parameters** > **Factor**, and then click **Apply**.

• Use `First Element Offset` via y+ height and Reynolds number — This option lets you specify the approximate maximum y+ height for the blade. ANSYS TurboGrid uses this information to compute the Edge Refinement Factor. The specified offset and the actual offset may differ. The elements near the surface of the blade will not all have a constant height, but they will generally be at or less than the specified maximum. Furthermore, the y+ calculation is only approximate. On the **Mesh Size** tab of the `Mesh Data` object editor, set **Boundary Layer Refinement Control** > **Method** to `First Element Offset`, set **Near Wall Element Size Specification** > **Method** to `y+`, enter a suitable **Reynolds No.** (Reynolds number), specify **Parameters** > **Offset Y+** and then click **Apply**.

• Use `First Element Offset` via Absolute offset height — This option lets you specify the approximate maximum near wall element height. ANSYS TurboGrid uses this information to compute the Edge Refinement Factor. The elements near the surface of the blade will not all have a constant height, but they will generally be at or less than the specified maximum. On the **Mesh Size** tab of the `Mesh Data`

object editor, set **Boundary Layer Refinement Control** > **Method** to `First Element Offset`, set **Near Wall Element Size Specification** > **Method** to `Absolute`, specify **Parameters** > **Offset** and then click **Apply**.

For the **Edge Refinement Factor** and **First Element Offset** methods, the parameters controlling the boundary layer distribution can also be edited on the **Boundary Layer Control** object editor. If there is more than one boundary layer in the topology, all the boundary layers can be changed at the same time by selecting the **Apply to All** check box on the **Mesh Data** object editor; or they can be edited individually on the **Boundary Layer Control** object editor.

10.4.1.3. Changing the Mesh Size near Five-Edge Vertices

For details, see Five-Edge Vertex Mesh Size Reduction.

10.4.1.4. Changing the Number of Elements on a Selected Topology Edge

You can control the number of elements along a given edge, but because of the smoothing process you cannot control the distribution of the elements along the edge. To increase or decrease the number of elements on a given topology edge, right-click the edge in the viewer and select one of the following:

- Insert Edge Split Control - This option inserts an `Edge Split` object under the `Mesh Data` object. You can then edit this object to adjust the Edge Split Factor. The `Edge Split` object also displays the base and actual Block Edge Split Counts for the selected edge.

- Increase Edge Refinement - This option lets you choose a preset amount by which to increase the Edge Split Factor. An `Edge Split` object is automatically added for the selected edge if one has not been created already.

- Decrease Edge Refinement - This option lets you choose a preset amount by which to decrease the Edge Split Factor. An `Edge Split` object is automatically added for the selected edge if one has not been created already.

The `Edge Split` object also displays the base and actual Block Edge Split Counts. You can remove the control by right-clicking the `Edge Split` object in the object selector and selecting **Delete**.

10.4.1.5. Changing the Spanwise Mesh Distribution

ANSYS TurboGrid has several methods for controlling the spanwise mesh distributions. The default method is `Proportional`. These methods are selected on the **Passage** tab of the `Mesh Data` object, and they are described below:

- Proportional - With this option ANSYS TurboGrid automatically computes the number and distribution of elements in the spanwise direction so that the near wall element heights on the hub and shroud match the maximum near wall element height on the blade. ANSYS TurboGrid adjusts the spanwise mesh count so that the average element aspect ratio (spanwise height divided by streamwise length) near the midspan is approximately equal to the blade aspect ratio divided by the Proportional **Factor**. Increasing or decreasing the Proportional **Factor** increases or decreases, respectively, the mesh resolution in the spanwise direction. This option is only available when using the ATM Optimized method.

- Element Count and Size - This option lets you choose the spanwise element count and the element height near the hub and shroud, and it operates the same as for the traditional control point topologies.

- Boundary Layer - This option gives you more control over the spanwise mesh distribution by providing additional biasing adjustments near the hub and shroud compared with the `Element Count and Size` option. It operates the same as for the traditional control point topologies.

- Uniform - This option lets you specify the number of spanwise elements that will be distributed uniformly from hub to shroud.

The **Passage** tab of the `Mesh Data` object also shows the approximate size of the elements next to the wall. The values are reported in y+ or absolute distance depending on the **Near Wall Element Size Specification** > **Method** setting on the **Mesh Size** tab.

10.4.1.6. Locking the Mesh Dimensions

In some situations, for example when you are comparing minor geometry variations, you may want to keep the mesh dimensions the same from one case to the next. This will reduce the influence of the mesh on the CFD results. To lock the mesh dimensions, select the **Lock mesh size** check box on the **Mesh Size** tab of the **Mesh Data** object. This setting will maintain the current mesh dimensions and prevent any subsequent changes to the global or local mesh controls. The lock status will be preserved from one session to the next if the project or state file is saved. To unlock the mesh, clear the **Lock mesh size** check box.

10.4.1.7. Using Splitter Blades with ATM

The ATM method now supports splitter blades which meet the following criteria:

- The main and splitter blades have rounded leading edges and cut-off trailing edges.

- The leading edge for the main blade is positioned ahead (meridionally) of the splitter leading edge.

- The trailing edges for both blades are located at the same meridional position.

10.4.1.8. Advanced Topology Control

By default, ANSYS TurboGrid automatically selects the appropriate topology based on the blade style (cut-off or rounded edges) and blade angles. If you want to override the selection, then you can do it by editing the CCL. The topology pieces are set using the following CCL lines:

- The first line specifies which topology pieces to use:

```
ATM Topology List = "leading edge topology choice" , "mid-passage topology choice" ,
" trailing edge topology choice"
```

The available topology pieces are listed in the table below and can be specified either using the template name or the topology file name. If multiple topology pieces are set, each must be delimited by a comma and should be ordered in the same way as the CCL above. User defined topology pieces can also be specified using their file name. For more information on setting up user-defined topologies in ANSYS TurboGrid, please contact your customer support representative.

If you choose to let ANSYS TurboGrid specify the topology pieces, ANSYS TurboGrid will also automatically add the CCL above to show the topology pieces chosen. The available templates and corresponding file names are shown in the table below:

Template Name	Topology File Name
LECutoffBlock1by4	LE1by4.tgt
LECutoffHerringboneHigh	LEHerringHigh.tgt
LECutoffHerringboneLow	LEHerringLow.tgt
LERoundedSymmetricStar	LEs2by4.tgt

Template Name	Topology File Name
PassageBlock1by4	mid1by4.tgt
TECutoffBlock1by4	TE1by4.tgt
TECutoffHerringboneHigh	TEHerringHigh.tgt
TECutoffHerringboneLow	TEHerringLow.tgt
TERoundedSymmetricStar	TEs2by4.tgt

Note

– Pictures of the various topology pieces can be found in ATM Topology in the *TurboGrid Reference Guide*.

– Leading edge topology pieces are prefixed with "LE". Similarly, mid-passage and trailing edge pieces are prefixed with "mid" and "TE", respectively.

• The second line determines whether you or ANSYS TurboGrid selects the topology pieces:

```
Override ATM Topology List = "True/False"
```

If the value is set to true, then you select the topology pieces using the first line. If the value is set to false, then ANSYS TurboGrid automatically selects and lists the topology pieces in the first expression. The topology pieces are selected based on the average blade angle: if the angle is between 0 and 10 degrees — ANSYS TurboGrid chooses the H style; if the angle is greater than 10 degrees — ANSYS TurboGrid chooses the herringbone.

10.4.1.9. Advanced Local Refinement Control

When ANSYS TurboGrid initializes the ATM topology, it builds the topology in pieces and then conglomerates the block structure to minimize the number of blocks in the final topology. Because you can control only the number of elements along a given topology edge and not the distribution, having more blocks in specific areas, for example near the trailing edge, gives you better control of the mesh resolution in these areas. Preserving specific vertices from being conglomerated in these situations will give you better control of the local mesh resolution — the specific vertices will be dependent on the topology pieces used. For a list of the available topology pieces, see *Advanced Topology Control* (p. 95).

You can retain additional vertices with CCL of the following format: "template name" , "vertex number". This must be repeated for each new vertex number, leading to the form:

```
TOPOLOGY SET:
  Override Conglomeration Retained Vertices List = true
  Conglomeration Retained Vertices List = "template name", "vertex number", "template name", "vertex number", ...
END
```

The following is an example of CCL that provides additional control near the leading and trailing edges for rounded blades:

```
TOPOLOGY SET:
  Override Conglomeration Retained Vertices List = true
  Conglomeration Retained Vertices List = LERoundedSymmetricStar, 0, TERoundedSymmetricStar, 0, LERoundedSymmetricStar,
END
```

Topology piece diagrams and node numbers can be found in ATM Topology in the *TurboGrid Reference Guide*.

10.4.1.10. Advanced Mesh Element Size Control

You can scale the thickness of the outer row of mesh elements in the blade tip gap by adjusting the CCL parameter `Blade Tip First Element Size Factor`. The default value of this parameter is `1.0`. The figure below shows which mesh elements are scaled by this parameter (colored blue), and the direction in which they are scaled.
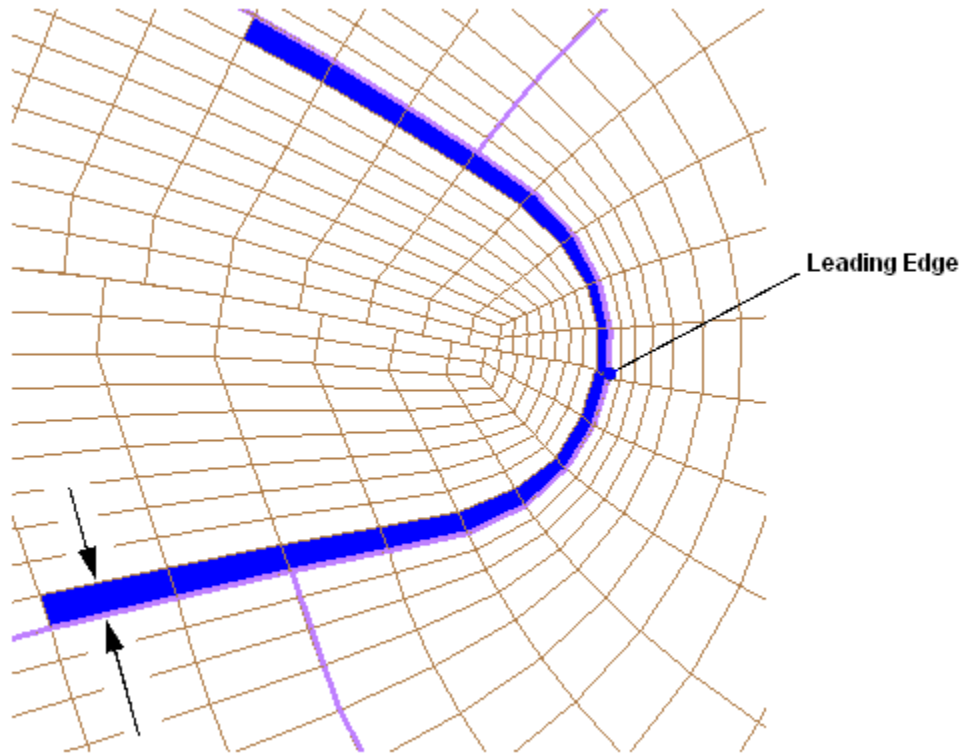
Blade Tip First Element Size Factor scales this dimension.

10.4.1.11. Example Mesh

The following figures show the resulting mesh for an example axial turbine blade.

Figure 10.6 Mesh on the Hub for an Axial Turbine Blade
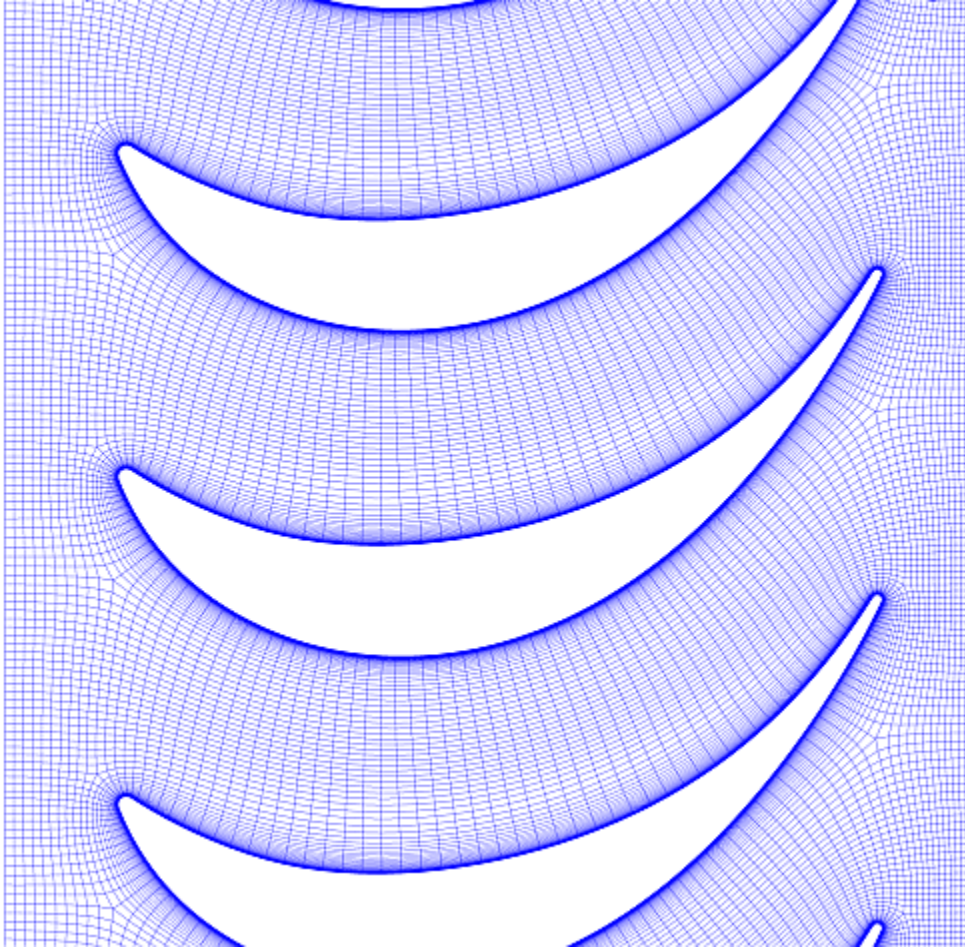
Figure 10.7 Mesh at the Tip for an Axial Turbine Blade
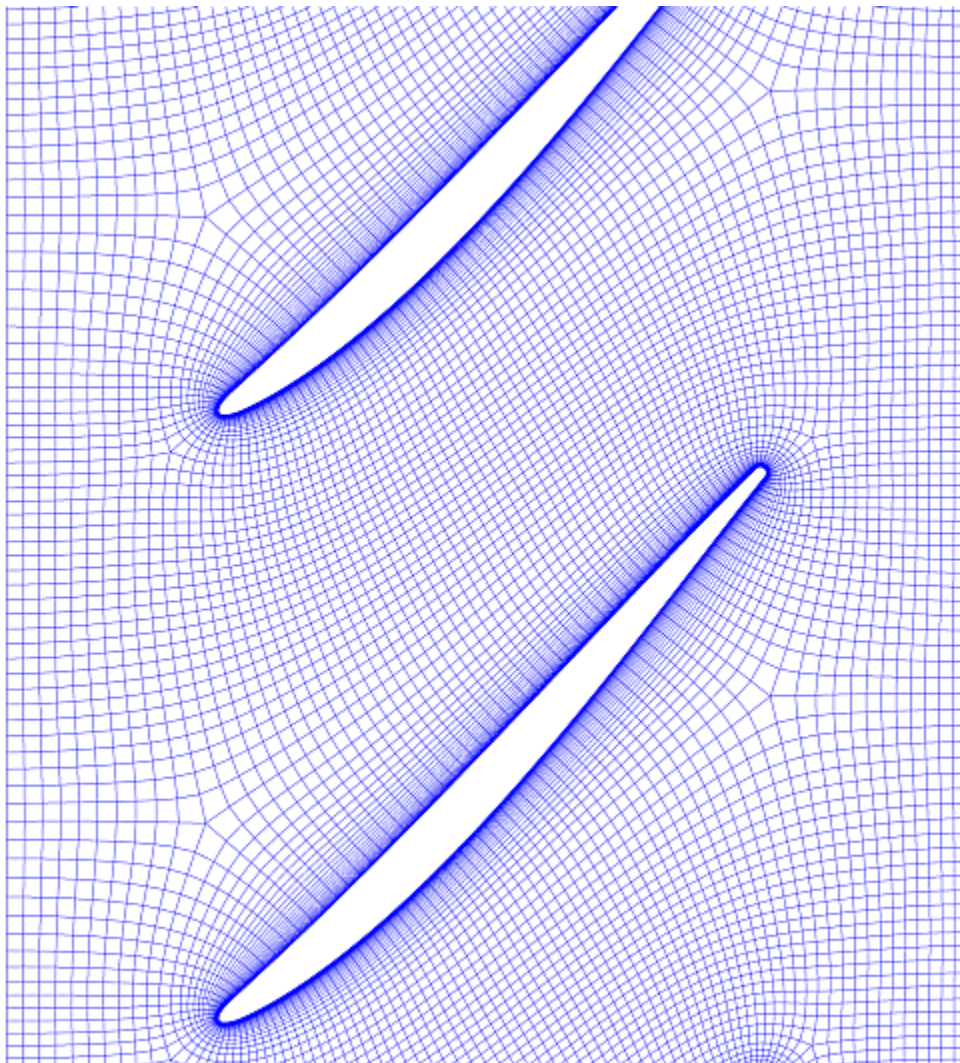
10.4.2. Traditional Topologies

The following topics are discussed:

10.4.2.1. The Topology Set Object and Topology Objects

To define the topology settings that affect the topology of every blade in the blade set, edit the `Topo-logy Set` object from the object selector or click *Edit Topology Set* 🖫 . To define or modify the settings that affect the topology for individual blades in the blade set, edit the corresponding blade-specific topology objects stored under the `Topology Set` object. You can only define the topology set or topology for an individual blade after the geometry has been defined. For details, see *Steps to Create a Mesh* (p. 71).

10.4.2.1.1. Definition Tab

10.4.2.1.1.1. Apply Settings To All Topologies Check Box

When the **Apply Settings To All Topologies** check box is selected, some `Topology Set` settings overwrite the corresponding blade-specific topology settings. When the check box is not selected, those same settings are disabled, enabling you to set the corresponding blade-specific settings independently.

If you change a setting for a blade-specific topology object, and the `Topology Set` object has a corresponding setting, then the **Apply Settings To All Topologies** check box will be cleared to help prevent accidentally losing the blade-specific settings in case you reapply the `Topology Set` object.

10.4.2.1.1.2. Topology Definition

The **Topology Definition** settings govern the general way in which the mesh is generated.

These settings are described in the following sections:

- *Placement* (p. 100)
- *Method* (p. 100)

10.4.2.1.1.2.1. Placement

The **Placement** setting makes it possible to choose between:

- ATM Optimized

  For details, see *ATM Optimized Topology* (p. 91).

- Traditional with Control Points

  This enables you to select from the topology types described in the next section.

10.4.2.1.1.2.2. Method

The **Method** options are described next:

- `None` prevents the creation of topology.

  This is useful when making changes to the geometry since the topology and mesh preview are not recreated after every minor geometry change.

- `H/J/C/L-Grid` allows a separate choice of topology type for the upstream and downstream ends of the passage mesh. The decisions are automatic by default but an override is available. For more information, see *H/J/C/L Topology Definition* (p. 106).

- `H-Grid` applies a topology of type H-Grid. An example of a topology of type H-Grid is shown in *Figure 10.8* (p. 101).

Figure 10.8 Topology of Type H-Grid
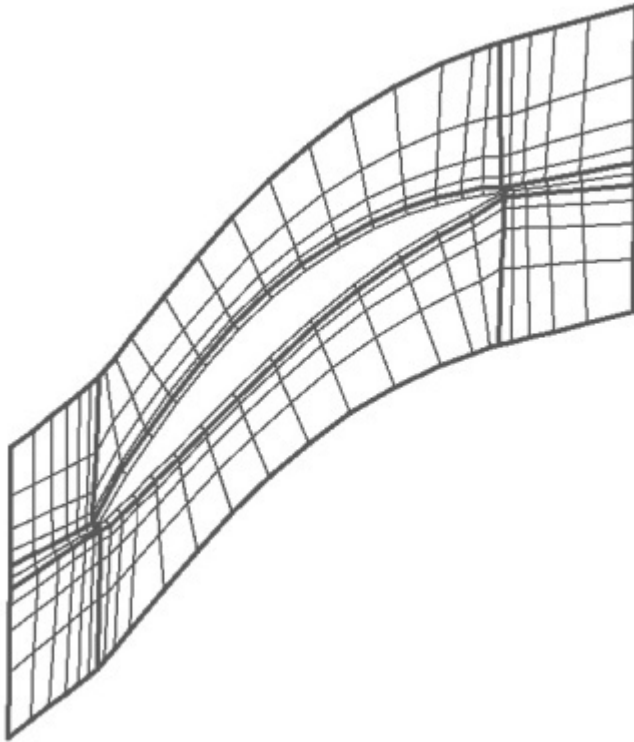
- `J-Grid` applies a topology of type J-Grid with an optional embedded O-Grid that surrounds the blade. An example of a topology of type J-Grid is shown in *Figure 10.9* (p. 102). A topology of type J-Grid always wraps in opposite directions around the leading and trailing edges (unlike choosing the J-J case of a topology of type H/J/C/L-Grid. For more information, see *H/J/C/L Topology Definition* (p. 106).).

Figure 10.9 Topology of Type J-Grid
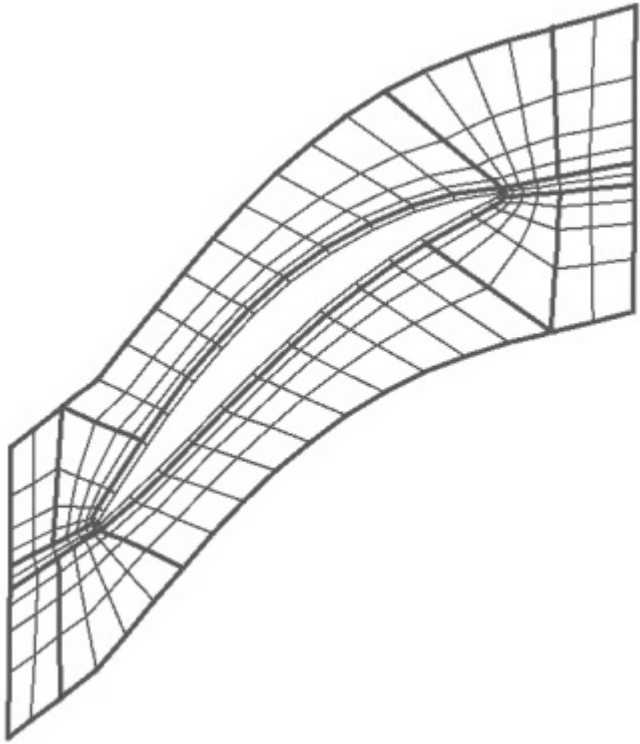
- `H-Grid Dominant` adds some topology blocks in a structured manner around an optional embedded O-Grid that surrounds the blade (as for a topology of type H-Grid), then completes the mesh by adding topology blocks in an unstructured manner. The structured blocks extend upstream of the leading edge, downstream of the trailing edge, and from the blade to the periodic surfaces. An example of a topology of type H-Grid Dominant is shown in *Figure 10.10* (p. 103).

Figure 10.10 Topology of type H-Grid Dominant
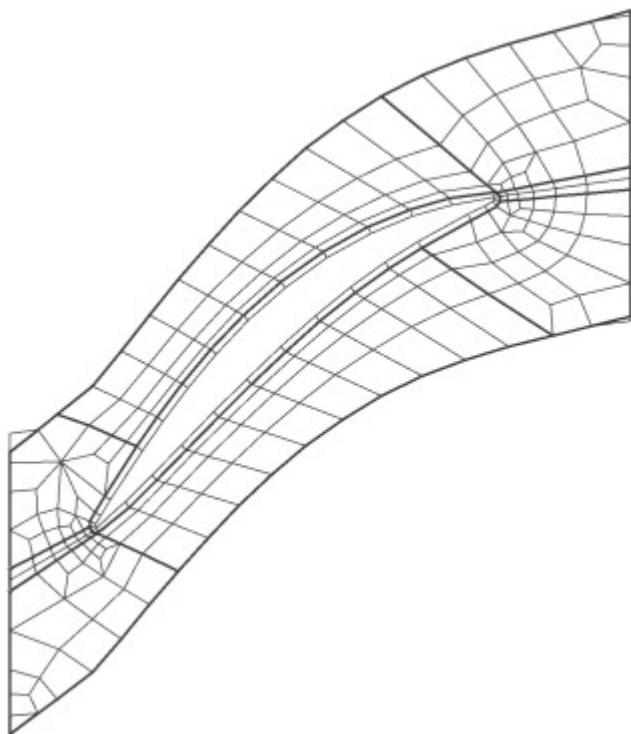
- `From File` uses a previously saved topology file as a starting point for a new topology.

  Topology files can be saved in the GUI (see *Save Topology Command* (p. 17)), or created manually using a text editor. If you have a topology file that you want to share with anyone using your installation of ANSYS TurboGrid, it should be saved in the directory `<CFXROOT>/etc/templates`. A typical template (`.tgt` file extension) for a topology of type H-Grid is provided in this directory.

  Note

  For a case with multiple blades, if you perform any of the following actions:

  – Freeze the topology

  – Manually lock one or more of the advanced parameters for a splitter blade

  – Load a topology from a file

  and subsequently, when attempting to change the topology, you encounter an error message about topology settings, then there is likely a problem due to the splitter blade advanced parameters being locked. In this case, for each splitter blade, go to the **Advanced Parameters** tab and clear all the check boxes that have "Override" in the name, then click **Apply**. You should then be able to successfully change the topology.

10.4.2.1.1.3. Include O-Grid

An O-Grid is a topology that forms a continuous loop around the blade profile (as would be seen in a blade-to-blade view). Using an O-Grid around the blade yields excellent boundary layer resolution and near-orthogonal elements on the blade.

To use an O-Grid, select **Include O-Grid**.

The **Width Factor** setting is used to adjust the thickness of the O-Grid. A width factor of *n* means that the O-Grid thickness is roughly *n* times the average blade width at a particular span location (The span location is set on the **Advanced Parameters** tab.).

10.4.2.1.1.4. One-To-One Interface Ranges

10.4.2.1.1.4.1. Periodic

The **Periodic** setting controls which parts of one side of the periodic surface are connected to the other side via one-to-one node matching. The possible options are:

- `Full`

  This is the default option. All of the nodes on one periodic surface correspond in a 1-to-1 fashion with nodes on the other periodic surface.

- `Between Blades & Upstream`

  The periodic surfaces have a 1-to-1 correspondence along the section between blades, and along the inlet block, but not along the outlet block. The latter is connected by a GGI interface.

- `Between Blades & Downstream`

  The periodic surfaces have a 1-to-1 correspondence along the section between blades, and along the outlet block, but not along the inlet block. The latter is connected by a GGI interface.

- `Between Blades`

  The periodic surfaces have a 1-to-1 correspondence along the section between blades, but not along the portions in the inlet and outlet blocks. The latter are connected by GGI interfaces.

- `None`

  A GGI interface is used along the entire periodic surface. In general, the nodes along one side of a GGI interface do not correspond with nodes on the other side.

10.4.2.1.1.4.2. Passage

The **Passage** setting controls the type of interface for all passage interfaces (i.e., all interfaces between adjacent blades in the blade set). Setting **Passage** to `Full` produces 1-to-1 interfaces. Setting **Passage** to `None` produces GGI interfaces. In general, the nodes along one side of a GGI interface do not correspond with nodes on the other side.

10.4.2.1.1.5. Tip Topology

Tip Topology is available when a hub or shroud tip (*The Hub Tip and Shroud Tip Objects* (p. 86)) exists. The **Hub** and **Shroud** settings have the following options:

- `H-Grid`

  Most of the blade is filled with an H-Grid block. The leading edge and trailing edge ends are filled with butterfly mesh blocks. If the H-Grid tip mesh is reasonably orthogonal, this is by far the best choice.

  This option is not available when using a topology of type `H/J/C/L-Grid` for the passage since, in general, this topology type yields different numbers of elements along each side of the blade.

- `H-Grid Not Matching`

  Similar to `H-Grid` except that the grid is split along the center of the blade thickness and there is no one-to-one connection of nodes between the halves. This option is useful when the distribution or number of nodes along each side of the blade is mismatched. When mesh alignment across the blade is difficult or impossible, `H-Grid Not Matching` is the best choice for such situations.

  The GGI interface is created near the mean camber line of the blade, rather than around the perimeter. This has the advantage of reducing the GGI surface area and the curvature of the GGI surfaces. It is also coincident in the tip-to-shroud direction to reduce errors, and is located away from regions of large flow gradients (i.e., away from the large gradients that occur at the passage/tip junction.

Important

If importing a grid with a non-matching interface into CFX-Pre, you will need to set up the interface manually when you define the pre-processing for your simulation. Refer to the Domain Interfaces section of the CFX-Pre documentation for details.

10.4.2.1.1.6. Preserve Control Points on Layers

The **Preserve Control Points on Layers** option, when selected, will cause the customized relative displacements of added local control points to be preserved if and when the topology block distribution changes.

10.4.2.1.1.7. Periodicity

Control the behavior of periodic topological vertices and mesh nodes using one of two methods for **Projection**:

- `Float on Curves` limits the periodic vertices and nodes to lie on the curve formed by intersecting the span surface (at the specified location), and the periodic surface.

- `Float on Surface` is the default method, and will allow periodic vertices and nodes to float on the span surface. The result is a topology and mesh which has less stretching between blades and improved skewness properties along the periodic boundary.

10.4.2.1.2. Advanced Parameters Tab for Topology Objects

The **Advanced Parameters** tab exists for every blade topology object.

Note

For a case with multiple blades, if you perform any of the following actions:

- Freeze the topology

- Manually lock one or more of the advanced parameters for a splitter blade

- Load a topology from a file

and subsequently, when attempting to change the topology, you encounter an error message about topology settings, then there is likely a problem due to the splitter blade advanced parameters being locked. In this case, for each splitter blade, go to the **Advanced Parameters** tab and clear all the check boxes that have "Override" in the name, then click **Apply**. You should then be able to successfully change the topology.

10.4.2.1.2.1. H/J/C/L Topology Definition

(applies only for topologies of type `H/J/C/L-Grid`)

The topology type settings used for the upstream and downstream ends of the passage are shown in the drop-down boxes. When the override is turned off, the settings are determined automatically. When the override is turned on, the settings are manually adjustable.

When the **One-to-one Interfaces: Periodic Range** option on the **Definition** tab of the `Topology Set` object is not set to `Full`, the upstream and/or downstream ends that do not have 1-to-1 periodicity can have a topology of type L-Grid, provided that there is only one blade. L-Grid topologies are not supported for multiple-blade systems. An example of a topology of type L-Grid is shown in *Figure 10.11* (p. 107).

Figure 10.11 Topology of Type L-Grid
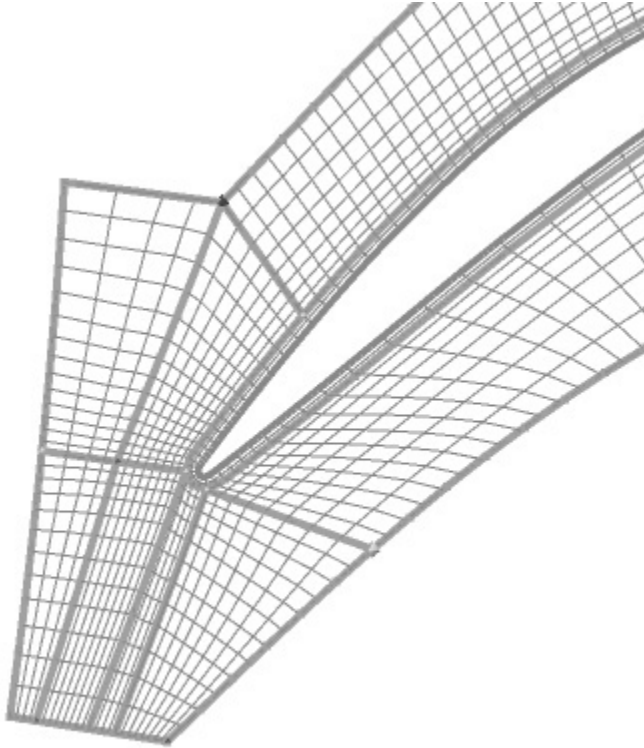
For topology types J-Grid and L-Grid, the orientation of the bend is automatically chosen as that which produces the smallest bending angle.

C-Grid is another available topology type for the leading and/or trailing edges when `H/J/C/L Topology` is selected. Before you can set the topology type for the leading or trailing edge to `C-Grid`, you must select the **H/J/C/L Topology Definition** > **Override default parameters** check box. An example of a topology of type C-Grid is shown in *Figure 10.12* .

Figure 10.12 Topology of Type C-Grid
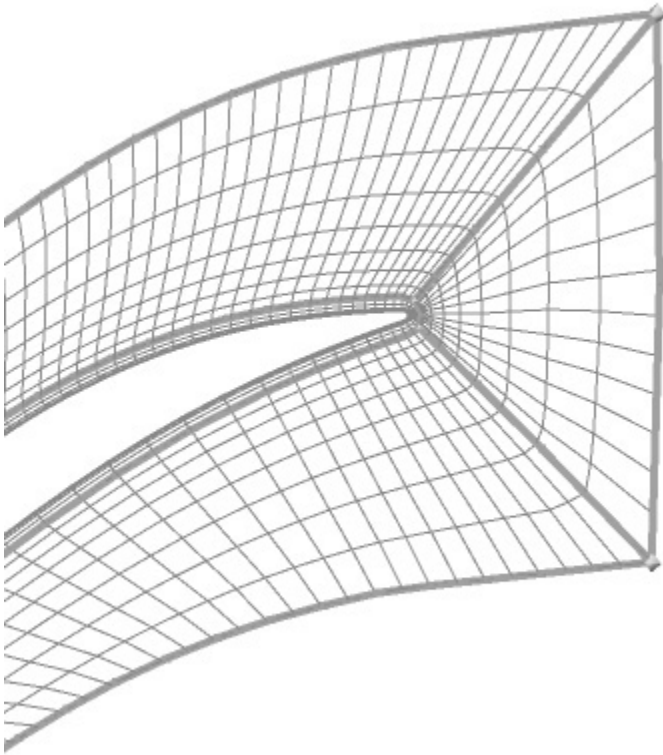
H-Grid Dominant is another available topology type for the leading and/or trailing edges when `H/J/C/L Topology` is selected.

If you specify a topology of type `H/J/C/L-Grid`, ANSYS TurboGrid assigns default topology types for the upstream and downstream ends as follows:

1. The blade metal angle is calculated at the span location of the topology (set on the **Advanced Parameters** tab) by linearly interpolating from the corresponding angles at the hub and shroud.

2. If this angle is greater than 60° and 1-to-1 periodicity is not set for that upstream/downstream end, a topology of type L-Grid is used; otherwise if the angle is greater than 45°, a topology of type J-Grid is used; otherwise a topology of type H-Grid is used.

10.4.2.1.2.2. Blade Blocks Calculation

The **Blade Blocks Calculation** settings define the number of topology blocks along each side of the blade from the leading edge to the trailing edge. In the case of multiple blades per blade set, these settings are available only for the first blade in the set.

The number of blocks is the starting point which ANSYS TurboGrid uses to create a topology. The driving force for topology creation is to approach isotropy and orthogonality, meaning that each block is as close to having equal lengths and right angles as possible. The entered value for **# of Blocks** is treated as a target, but may not exactly match the resulting topology. In most cases, the number of blocks in the topology is quite close to the number of blocks defined.

Specify the blade block distribution using a uniform or non-uniform method. If you select non-uniform, enter a target value for the number of constant blocks (**Const Blocks**), which determines a target for the number of the blocks which have a uniform distribution (the actual number may not be the same

as the target value). If set to zero (default), the maximum number of constant blocks is automatically calculated.

10.4.2.1.2.3. Blade to Periodic/Inlet/Outlet Surface Block Distribution

These options are for specifying the block count from the blade to the following surfaces:

*   `Periodic Surface`
*   `Inlet`
*   `Outlet`

In the case of multiple blades per blade set, the blade to inlet and blade to outlet settings are available only for the first blade in the set.

Block Count

The **Block Count** parameter is the most basic parameter for controlling the number of topology blocks between the blade and the applicable surface (periodic, inlet, or outlet). The availability of this parameter is dependent on other topology settings.

Blade to Inlet Surface Block Distribution: Extra at LE

(applies for `H/J/C/L-Grid` topology when topology type J-Grid is applied at the leading edge)

The **Extra at LE** option causes the specified number of rows of topology blocks to be added. The added rows extend from the inlet to the periodic surface that makes an acute angle with the inlet (as measured in the local plane of the layer on the inside of the layer outline). The rows pass by, and touch, the leading edge O-Grid (or the leading edge itself, if no O-Grid exists), increasing the number of topology rows between the blade and the inlet.

Blade to Outlet Surface Block Distribution: Extra at TE

(applies for `H/J/C/L-Grid` topology when topology of type J-Grid is applied at the trailing edge)

The **Extra at TE** option for **Blade to Outlet Surface Block Distribution** is analogous to that for **Blade to Inlet Surface Block Distribution**.

10.4.2.1.2.4. Sharp Leading/Trailing Edges

To improve mesh orthogonality, ANSYS TurboGrid automatically adapts the topology for sharp leading/trailing edges.

Figure 10.13 (p. 110) shows the effect of the sharp edge treatment on the topology near a blade edge. Note that the sharp edge treatment improves element face angles in the refined mesh.

Figure 10.13 No Sharp Edge Treatment versus Sharp Edge Treatment
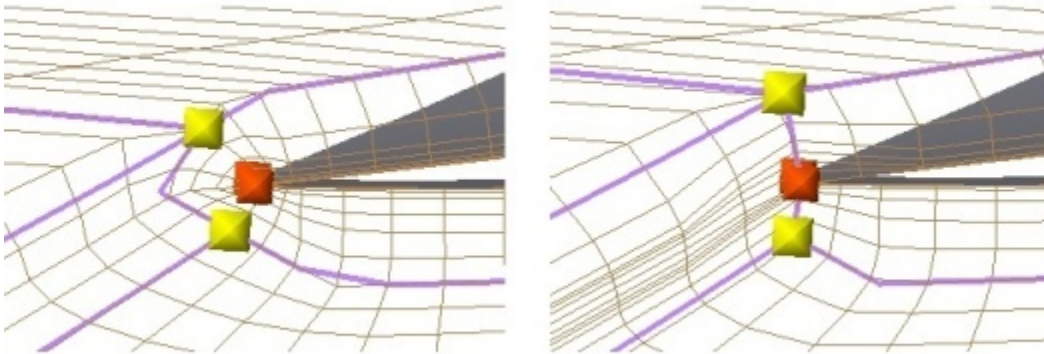
For blade edges that are slightly blunt, you may want to manually control whether a sharp edge treatment is applied or not applied. For the leading edge, do the following:

1. Select the **Override Sharp LE Determination** check box.

2. Select or clear the **Sharp Leading Edge** check box to force the sharp edge treatment to be applied or not applied.

The trailing edge settings work similarly.

Note that the sharp edge treatment is not supported for cut-off edges.

10.4.2.1.2.5. O-Grid Corner Vertices

For a cut-off or square blade edge, the two vertices at the end of an O-grid can be positioned using two different methods which are explained next, and illustrated in *Figure 10.14* (p. 111):

- At Same AR

    This option causes the O-Grid end vertices to be positioned at the same axial and radial coordinates as the blade end.

- Project to OGrid

    This option causes the O-Grid end vertices to be positioned at the projected locations of the blade end corners, with projections being outward from the blade in a direction normal to the O-Grid.

Figure 10.14 O-Grid Corner Vertex Placement: At Same AR versus Project to OGrid
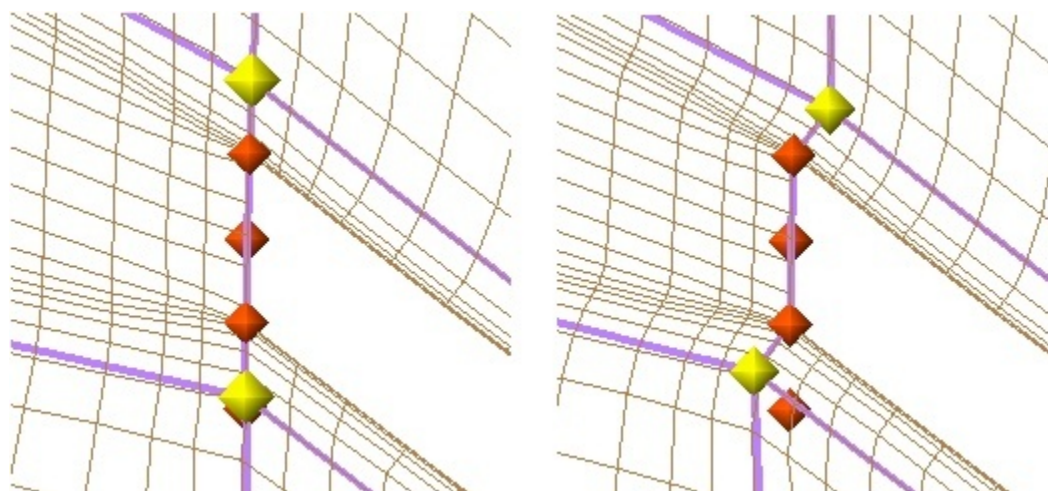
10.4.2.1.3. Span Location for Controlling Topology

The span is the distance between the hub and shroud, expressed as a fraction between 0 and 1. The span is calculated when m'-Theta coordinates are created. For details, see *Transformation Commands and Coordinate Systems* (p. 143).

ANSYS TurboGrid uses a span parameter for several reasons related to topology, including:

- Obtaining blade metal angles so that ANSYS TurboGrid can automatically set topology types after you have specified the topology method: `H/J/C/L-Grid`.

- Calculating an appropriate number of topology blocks between the blade and the periodic interface or interface between blades.

By default, the span location is set to `0.5` (that is, 50% of the span). To adjust the span parameter, called `Span Location`, edit the `Topology Set` object using the **Command Editor** dialog box.

10.4.2.1.4. Freeze Button

In the course of creating a mesh, after you have decided on a topology type, you can freeze (make static) the topology advanced parameters by doing either of the following:

- Clicking the **Freeze** button, which is in the object editor for the `Topology Set` object and any blade-specific topology object. This freezes all topology advanced parameters for all blade-specific topology objects.

- Turning on all of the overrides manually on the **Advanced Parameters** tabs of all blade-specific topology objects.

If any of the topology advanced parameters are not static (that is, they do not have overrides turned on), they will be recalculated if any geometry object is loaded or changed. This could result in an un-wanted change to the topology.

Geometry changes include:

- Loading a different hub, shroud, or profile curve

- Moving an inlet or outlet point (for example, inlet low shroud point)

- Saving a state file on one computer and then restoring the state on another (the effect of round-off differences)

Since changes to topology can destroy local control points, ensure that topology advanced parameters are frozen.

Note

When the `H-Grid Dominant` topology type option is selected, the "freeze" in topology is accomplished by saving the topology to a file and then loading it. This is performed semi-automatically; you only need to provide the file name for the topology file. In this case, upon loading the topology, the method will change to `From File` and, as usual for that method, the overrides on the **Advanced Parameters** tab will be permanently selected.

10.4.2.2. Topology with Cut-off or Square Leading or Trailing Edges

10.4.2.2.1. Using the Cut-off or Square Option

The blade geometry object allows leading and trailing edges to be specified as "cut-off or square" (For details, see *Cut-off or square* (p. 83).). This option can affect the topology.

Figure 10.15 (p. 112) shows an example of how the `Cut-off or Square` option causes an O-Grid to be truncated from the end of the blade, starting at the pair of edge curves.

Figure 10.15 Cut-off Trailing Edge using Topology of Type H-Grid with O-Grid
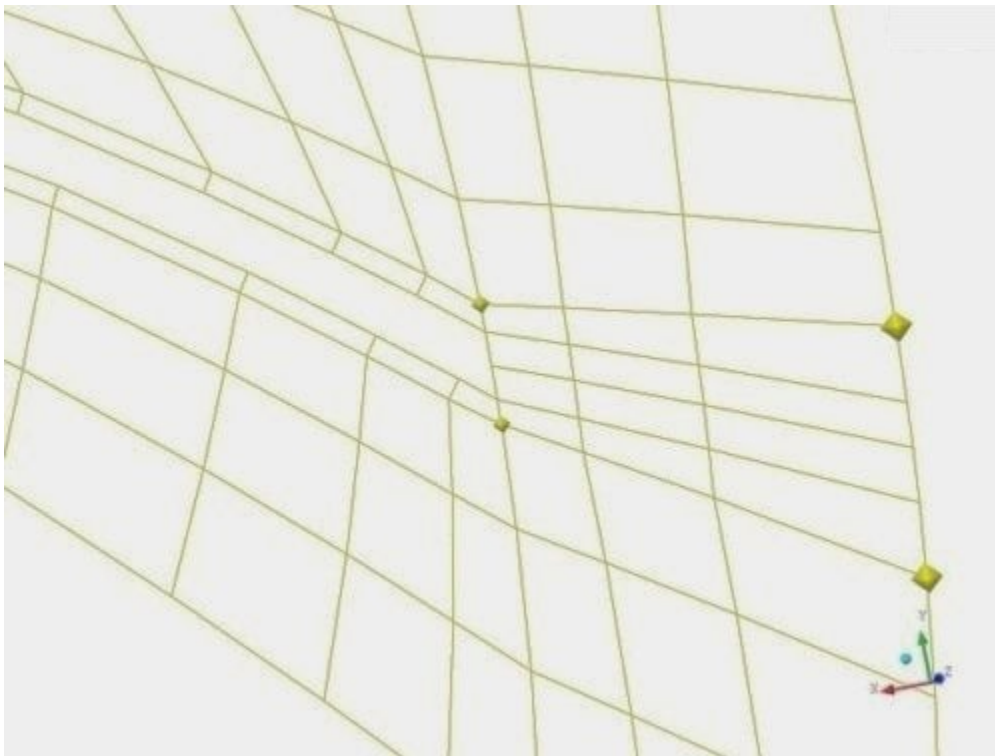
The **Cut-off or Square** option can also be used for a topology with no O-Grid present. In this case, the topology points surrounding the blade edge will be constrained to lie on the edge curves.

10.4.2.2.1.1. Automatic Generation of Edge Split Controls

When a "cut-off or square" edge is used (or when a topology of type C-Grid is used), ANSYS TurboGrid automatically adds edge split controls just off the end of the blade in order to improve element size matching in that region. The edge split controls are found under the `Mesh Data` object. If you open one of the automatically-created edge split controls in the object editor, you will see that the **Automatically computed split factor** check box is selected by default. When selected, this check box causes the edge split value to be calculated automatically. To override the **Split Factor** setting, clear the check box. The **Split Factor** setting is multiplied by the global block edge split count in order to determine the local block edge split count. For details about edge split controls, see *Edge Split Controls* (p. 121).

10.4.2.2.2. Using the Line of Rotation on Hub and Shroud Option

When the **Line of Rotation on Hub and Shroud** option is selected in the `Blade Set` object, certain topology points on the hub and shroud layers are constrained to lie on lines of rotation (about the machine axis) starting from the cut-off edges.

The topology points that are affected are those on the topological paths between the cut-off edges of the blade and the periodic surfaces. When this option is applied, these points are constrained to lie on the line of rotation, about the machine axis, of the corresponding cut-off edge point. As a result of this added constraint, the affected topology points lose 1 degree of freedom; those on the periodic surfaces and O-Grid (if applicable) become fixed, while the remaining affected topology points become constrained to the line of rotation.

Only topologies of type `H-Grid` are affected by this option, including the H-Grid portion of a topology of type `H/J/C/L-Grid` or `H-Grid Dominant`.

10.5. Mesh Data

To define the mesh properties for the entire mesh, edit the `Mesh Data` object from the object selector or click *Edit Mesh Data* . To define or modify the mesh properties of a blade-specific portion of the mesh, edit the corresponding object stored under the `Mesh Data` object.

Defining the `Mesh Data` object does not create the mesh. The mesh data can only be set after the topology has been defined, but it is recommended that you set the required mesh size before you create the layers (discussed later), because the true mesh nodes are displayed on the topology layers as the topology is adjusted. For details on creating a mesh, see *Steps to Create a Mesh* (p. 71) and *Mesh Command* (p. 31).

10.5.1. The Mesh Data Objects

The `Mesh Data` object contains settings that affect the mesh globally. The individual blade mesh data objects (stored under the `Mesh Data` object in the object selector) contain a subset of the settings of the `Mesh Data` object, and affect the mesh for individual blades in the blade set.

10.5.1.1. Mesh Size Tab

The **Mesh Size** tab is applicable to the `Mesh Data` object.

10.5.1.1.1. Method

The **Method** setting controls the mesh density, and can be set to one of the following:

- `Target Passage Mesh Size`

  The `Target Passage Mesh Size` method sets a target for the number of nodes in the mesh passage. The pre-set options for the **Node Count** setting are `Coarse (20000)`, `Medium (100000)`, and `Fine (250000)`. There is also an option, `Specify`, to specify the target number of nodes.

  ANSYS TurboGrid attempts to reach the target number of nodes by:
  - Adjusting the number of elements placed along each topology block edge (This can be viewed after pressing **Apply** by changing the method to `Topology Block Edge Split` or `Global Size Factor` as applicable.)
  - Adjusting the number of elements from hub (or hub tip) to shroud (or shroud tip) if these have not been explicitly set.

- `Topology Block Edge Split` (Traditional topology only)

  The `Topology Block Edge Split` method defines how fine the mesh is on each layer. Enter the value for the number of elements placed along each topology block edge. For example, if you set **# of Elements** to 2, there are two elements placed along each topology block edge.

  The topology block edge split can also be controlled locally. For details, see *Edge Split Controls* (p. 121).

- `Global Size Factor` (ATM topology only)

  The `Global Size Factor` method defines the overall mesh size. To increase the resolution of the mesh, increase the size factor using the **Size Factor** setting.

10.5.1.1.2. Boundary Layer Refinement Control

In the context of the **Boundary Layer Refinement Control** settings, the boundary layer is represented by the set of topology blocks along the pressure and suction sides of the blade.

The available **Method** options for boundary layer refinement control are:

- Proportional to Mesh Size
- Edge Refinement Factor
- First Element Offset

10.5.1.1.2.1. Proportional to Mesh Size

The `Proportional to Mesh Size` option controls the number of elements across the boundary layer in proportion to the specified values for **Factor Base** and **Factor Ratio**. If the blade has a cut-off edge, **Factor Ratio** also controls the number of elements along the cut-off edge. Increasing the **Factor Base** or **Factor Ratio** increases the number of elements across the boundary layer and along the cut-off edge (if applicable). Conversely, decreasing the value of **Factor Base** or **Factor Ratio** decreases the number of elements.

10.5.1.1.2.2. Edge Refinement Factor

The `Edge Refinement Factor` option does not maintain an O-grid expansion rate similar to `Proportional to Mesh Size` when the global size factor is changed, and therefore the edge refinement factor may need to be adjusted when the mesh size is changed. Increasing or decreasing the edge refinement factor will increase or decrease the mesh resolution in the O-grid, respectively.

10.5.1.1.2.3. First Element Offset

The `First Element Offset` option controls the near-wall height for elements. ANSYS TurboGrid uses the near-wall height to compute the edge refinement factor. The specified **Offset** value and the actual offset may differ. The elements near the surface of the blade will not all have a constant height, but they will generally be at or less than the specified **Offset** value, which is treated as a maximum. The **Method** can be set to either `y+` or `Absolute`.

10.5.1.1.2.4. Cutoff Edge to Boundary Layer

The **Cutoff Edge To Boundary Layer** setting controls the number of elements along the cut-off edge in proportion to the specified **Factor**. The number of elements along the cut-off edge is also proportional to the number of elements across the boundary layer. Increasing the **Factor** increases the number of elements along the cut-off edge. Conversely, decreasing the **Factor** decreases the number of elements along the cut-off edge.

10.5.1.1.2.5. Five-Edge Vertex Mesh Size Reduction

An ATM-based mesh may contain one or more vertices that each join exactly five edges. At these locations, the mesh generation process will tend to pull the mesh away from the five-edge vertex to make the surrounding element edges more orthogonal, resulting in relatively larger elements surrounding the vertex than the rest of the mesh in their vicinity. The **Five-Edge Vertex Mesh Size Reduction** > **Factor** setting, when set to a value less than 1.0, will reduce this tendency, trading off between mesh orthogonality for sizing.
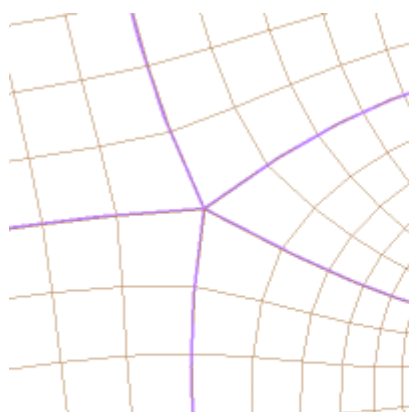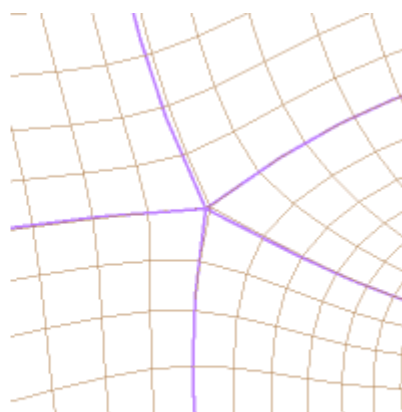
Factor = 1.0   Factor = 0.5

10.5.1.1.2.6. Near Wall Element Size Specification

The **Near Wall Element Size Specification** setting controls the method by which the near-wall node spacing is specified on the **Passage** and **Hub/Shroud Tip** tabs. The near-wall node spacing is the distance between a wall (e.g., hub, shroud, or blade) and the first layer of nodes from the wall.

The available **Method** options for calculating the near wall spacing are:

- `y+`

    For details, see *Y Plus* (p. 116).

- `Normalized`

For details, see *Normalized* (p. 116).

- `Absolute`

  For details, see *Absolute* (p. 116).

10.5.1.1.2.6.1. Y Plus

The `y+` method allows you to set the near wall spacing, Δy, in accordance with a target value of y^+.

The target value of y^+ may then be specified on the **Passage** and **Hub/Shroud Tip** tabs, as applicable (i.e., when a near wall size is required by the specified distribution method).

The following formula relates the near wall spacing to y^+:

$$\Delta y = L \; \Delta y^+ \; \sqrt{80} \; R \, e_x^{1/14} \; \frac{1}{R \, e_L} \qquad\qquad \textbf{(10–1)}$$

where L is the blade chord, Δy^+ is the specified target y+ value, $R \, e_x$ is the Reynolds number based on the distance along the chord (measured from the leading edge), and $R \, e_L$ is the Reynolds number based on chord length. ANSYS TurboGrid approximates L as the algebraic average of the chord lengths of each blade profile in the blade file. You must specify $R \, e_L$. ANSYS TurboGrid approximates $R \, e_x$ as being equal to the specified value of $R \, e_L$.

> **Note**
>
> If you specify a near wall size (on the **Passage** or **Hub/Shroud Tip** tab) when the near wall method is not y^+, you can switch the method to `y+` (at least temporarily) to see the estimated value of y^+ as a setting on the **Passage** or **Hub/Shroud Tip** tab.

10.5.1.1.2.6.2. Normalized

The `Normalized` method allows you to set the near wall spacing as a normalized value. Normalization is interpreted as the absolute distance divided by the maximum possible distance. The latter is one of the following, as appropriate:

- boundary layer thickness
- distance from hub (or hub tip, if it exists) to the shroud (or shroud tip, if it exists)
- thickness of the O-Grid
- distance from the hub tip to the hub, or shroud tip to the shroud.

10.5.1.1.2.6.3. Absolute

The `Absolute` method allows you to set the near wall spacing directly. Such a specification (i.e., on the **Passage** or **Hub/Shroud Tip** tab) requires a dimensional value with units of distance.

10.5.1.1.3. Inlet Domain and Outlet Domain Check Boxes

The **Inlet Domain** and **Outlet domain** check boxes determine whether or not the inlet and outlet domains are to be generated as part of the mesh. Settings that affect these grid regions are found on the **Inlet/Outlet** tab.

10.5.1.2. Passage Tab

The **Passage** tab is applicable to the `Mesh Data` object, and is used to specify distribution settings. Distribution settings are described generically in *Distribution Settings in General* (p. 119).

10.5.1.2.1. Spanwise Blade Distribution Parameters

Use the **Spanwise Blade Distribution Parameters** section to control the distribution of mesh elements in the spanwise direction along the blade.

The **Method** setting and its associated settings are described in *Distribution Settings in General* (p. 119).

10.5.1.2.2. Apply O-Grid Parameters To All Blades / O-Grid

The **O-Grid** section (for single-blade cases) and **Apply O-Grid Parameters To All Blades** section (for multiple-blade cases) describe distribution of the nodes in the O-Grid, outward from the blade surface. The section is available if an O-Grid is defined for any blade. To include an O-Grid in the mesh, see *Include O-Grid* (p. 103).

The **Method** setting and its associated settings are described in *Distribution Settings in General* (p. 119).

The **Distance** setting controls how the O-Grid distance varies from hub to shroud, and has the following options:

- `Proportional to Radius`

  Using the `Proportional to Radius` method (which is default), the O-Grid distance from hub to shroud is proportional to the average span radius. This may result in the O-Grid distance increasing in size from hub to shroud.

- `Proportional to Width`

  Using the `Proportional to Width` method, the O-Grid distance from hub to shroud is proportional to the average span blade thickness.

- `Constant`

  Using the `Constant` method, the O-Grid distance is constant from hub to shroud.

10.5.1.3. Hub Tip and Shroud Tip Tabs

The **Shroud Tip** tab is available when a shroud tip exists; the **Hub Tip** tab is available when a hub tip exists. See *The Hub Tip and Shroud Tip Objects* (p. 86) for details on defining the shroud (or hub) tip. These tabs are applicable to the `Mesh Data` object, and are used to specify distribution settings.

10.5.1.3.1. Hub Tip Distribution Parameters and Shroud Tip Distribution Parameters

Use these settings to control the distribution of mesh elements in the spanwise direction across the tip gap.

Set Method to one of the following:

- `Match Expansion at Blade Tip`

- `Element Count and Size`

- `Uniform`

These methods are described in *Distribution Settings in General* (p. 119).

10.5.1.3.2. Apply Blade Tip Parameters to All Blades Check Box

The type of topology for the blade tip is selected on the **Definition** tab of the topology object (see *Tip Topology* (p. 104)). When using the `H-Grid` or `H-Grid Not Matching` topology types, you may edit the number of elements across the blade thickness by selecting **Override default # of Elements**, then setting the appropriate value(s) for the number of mesh elements. The settings that are available depend on the type of tip topology (set in the applicable topology object). The following list contains topology-specific information about the settings:

- `H-Grid`

  Set **# of Elements** to indicate the number of elements across the blade thickness away from the leading edge and trailing edge. This should be an even number.

  Set **# of LE Elements/# of TE Elements** to indicate the number of rows in the butterfly mesh that fills the leading/trailing edge. This number must be at least 2 less than the **# of Elements** since the difference divided by 2 becomes the number of columns in the butterfly mesh.

- `H-Grid Not Matching`

  Set **# of Elements** to indicate the number of elements across the blade thickness away from the leading edge and trailing edge.

10.5.1.3.2.1. Tip Centerline Location

The **Tip Centerline Location** setting is available when the following conditions are met:

- The corresponding blade tip, hub or shroud, exists.

- The tip topology is set to `H-Grid Not Matching`.

- The corresponding edge of the blade, leading or trailing, is cut off.

The setting controls how the blade centerline behaves at the cut-off end. The options are:

- `Automatic`

  Effectively selects one of the other two methods automatically, based on the geometry. If one of the corners of the cut-off edge is at a sufficiently acute angle (default is 45°), then the `Corner` option is selected, otherwise the `Middle` option is selected. The threshold angle can be set by defining the parameter `GGI Tip Angle To Switch Mean Line Into Cut Off Corner` (to an angular value; default unit is `[degree]`) in the CCL for the blade-specific mesh data object.

- `Middle`

  The centerline meets the cut-off blade edge in the middle (between the corners).

- `Corner`

The centerline meets the corner of the cut-off blade edge.

10.5.1.4. Distribution Settings in General

The distribution of elements along the blade (in the spanwise direction) is controlled by one of these methods:

*   End Ratio

    The `End Ratio` method is default. Specify an end ratio, which is the ratio of the largest element size (measured in the pertinent direction) to the element size at the pertinent wall. All other parameters that affect the distribution (i.e., those for the `Element Count and Size` method) will be set automatically. The advantage of using the **End Ratio** method is that a change in mesh density will result in a self-similar mesh.

*   Element Count and Size

    To use the `Element Count and Size` method, specify the number of elements that span the pertinent extent and whether the distribution is uniform or non-uniform. If the distribution is non-uniform, you must also specify the number of uniformly-distributed elements (**Const Elements**) and the element size next to the pertinent wall(s).

*   Boundary Layer

    To use the `Boundary Layer` method (not applicable for O-Grid settings), specify distribution parameters for three sections: boundary layer at the hub, boundary layer at the tip/shroud, and the section in between. **Layer Offset** means the thickness of the boundary layer and **Wall Offset** means the thickness of the first element next to the wall.

*   Expansion Rate

    To use the `Expansion Rate` method, specify an expansion rate and the element size next to the blade wall.

*   Uniform

    To use the `Uniform` method, specify the number of elements. Each element is the same size.

Note

The **Const Elements** value cannot be negative and must be at least 2 less than the **# of Elements** value for that region.

The distribution of elements across a tip gap (in the spanwise direction) is controlled by one of these methods:

1. Match Expansion at Blade Tip

    The `Match Expansion at Blade Tip` method is default. This method will apply the same expansion factor as that used along the blade span, and will match the size of the elements on each side of the tip clearance gap with the size of the first element along the blade. An appropriate number of elements is then derived.

2. Element Count and Size

    (described above)

3. Uniform

    (described above)

When using the End Ratio method, the element count and wall size(s) are updated accordingly in the **Element Count and Size** settings (When two wall sizes are adjusted, they are adjusted to the same value.). When using the Element Count and Size method, the **End Ratio** setting is updated accordingly.

In the case of an O-Grid distribution, the **Blade End Ratio** is the size ratio of the passage mesh element nearest the O-Grid to the O-Grid element nearest the blade surface. A special feature of the distribution settings for O-Grids is that, if **# of Elements** is set to 0, the number of elements will be calculated automatically so that the best possible matching of element sizes at the interface between the O-Grid and the passage is ensured.

Size of Elements Next to Wall (Normalized) is normalized based on the total distance along the blade (not counting any tip clearance gaps). For example, on the **Hub Tip** tab, a value of 0.05 for **Size of Elements Next to Wall (Normalized)** > **Hub** represents an element size of five percent of the distance along the blade, applied on the elements next to the hub, and a value of 0.05 for the **Size of Elements Next to Wall (Normalized)** > **Tip** represents an element size of five percent of the distance along the blade, applied on the elements next to the hub end of the blade (both in the tip clearance gap and on the blade).

Note

The **Size of Elements Next to Wall (Normalized)** value must be less than 1 (unity) divided by the number of elements for that region. If it is not, ANSYS TurboGrid decreases it to meet this criterion.

To increase the quality of the mesh, try to minimize drastic changes in element size. Wherever possible, attempt to have gradual increases and decreases in element size in all directions.

10.5.1.5. Inlet/Outlet Tab

The geometries of the inlet and outlet domains of the mesh are controlled by the hub and shroud curves, and the Inlet and Outlet geometry objects. For details, see *The Hub and Shroud Objects* (p. 76) and *The Inlet and Outlet Objects* (p. 88).

The **Inlet/Outlet** tab contains settings that affect the mesh in the inlet and outlet domains. The **Inlet Domain** and **Outlet domain** check boxes on the **Mesh Size** tab of the Mesh Data object determine whether or not the inlet and outlet domains are to be generated as part of the mesh.

The nodes of the inlet and outlet domains match one-to-one with the nodes of the passage domain where they meet at the interfaces. The rest of the mesh in the inlet domain is then as near to being isotropic and orthogonal as possible.

10.5.1.5.1. Inlet Domain and Outlet Domain Settings

Setting **Mesh Type** to H-Grid (default) or H-Grid in Parametric Space specifies an H-Grid type mesh in the inlet/outlet domain that preserves the boundary layer mesh resolution at the hub and shroud, and automatically matches the element size at the interface between the inlet/outlet domain and passage domain. In this case, the streamwise distribution of elements is non-uniform. The **# of Elements** setting controls the number of elements in the inlet/outlet domain in the streamwise direction.

If the number of elements is 0, the number of elements in the inlet/outlet domain is determined by matching the element size at the passage interface and expanding the size at a prescribed rate for each successive element. The default element size ratio is 1.15.

When **Mesh Type** is set to `H-Grid in Parametric Space`, the resulting mesh will try to follow a parametric space created by an elliptic smoothing method. This type of mesh is particularly suitable for return channels.

Important

Clicking the **Apply** button does not create the mesh, it only saves the `Mesh Data` object and updates the 2D mesh previews. To create the mesh, select **Insert** > **Mesh** from the main menu or click *Create Mesh* .

Note

The number of elements is a starting point and the actual mesh may vary slightly if it results in a better mesh quality.

10.5.1.6. Mesh Around Blade Tab

The **Mesh Around Blade** tab is applicable to the blade-specific mesh data objects.

The O-Grid settings on this tab are the same as the O-Grid settings on the **Passage** tab for the `Mesh Data` object, except they are blade-specific. For details, see *Apply O-Grid Parameters To All Blades / O-Grid* (p. 117).

The **Size of Elements Next to Wall (Normalized)** setting is described in *Distribution Settings in General* (p. 119).

The **Hub Tip**, and **Shroud Tip** settings are for controlling the number of elements across the hub or shroud tip gap. They are similar to the settings on the **Hub Tip** and **Shroud Tip** tabs for the `Mesh Data` object. For details, see *Apply Blade Tip Parameters to All Blades Check Box* (p. 118).

10.5.1.6.1. Tip Centerline Location

For details, see *Tip Centerline Location* (p. 118).

10.5.2. Edge Split Controls

The number of elements placed along each topology block edge is globally controlled by the **Mesh Size** tab for the `Mesh Data` object. This value can be scaled locally by right-clicking a topology block edge and then selecting `Insert Edge Split Control`. This causes an editor to appear for edge split controls. It also causes all affected topology block edges to become highlighted with red lines. After entering a scale factor (the **Split Factor** setting) to multiply the default block edge split and then clicking **Apply**, the affected topology block edges are updated. To see the result, turn on the visibility of the refined mesh.

After applying a local edge split control, a new object is created and stored under `Mesh Data` in the object tree. This object may be edited or deleted in order to change or remove the local edge split control.

10.6. Layers

A layer shows the topology projected onto a given span. The addition of layers improves the 3D mesh by adapting the topology to the local geometry before mesh generation. Creating (and adjusting if necessary) additional layers enhances the quality of the mesh by creating a curve for the mesh to follow between the hub and the shroud. The more complex the blade shape-change from hub to shroud, the more layers will be required. The main user task in ANSYS TurboGrid is the adjustment of control points (discussed later in this section) to alter the topology on various layers (mainly the hub and shroud layers).

By default, and as a minimum, there are two layers present: one at the hub and one at the shroud. In many cases, additional layers are required to improve mesh quality.

10.6.1. Adding Layers

Layers can be added from the editor for the `Layers` object (see *The Layers Object* (p. 123)), or by right-clicking a layer object in the selector, and selecting the appropriate command from the **Insert** submenu of the shortcut menu. Layers can also be added automatically at the time of mesh creation (default), depending on a setting found in the editor for the `Layers` object.

10.6.2. Deleting Layers

The object selector and the editor for the `Layers` object (*The Layers Object* (p. 123)) each have an icon to delete layers (✕) as well as a shortcut menu option for deleting layers.

10.6.3. Editing the Settings of Layers

The editor for the `Layers` object (*The Layers Object* (p. 123)) and the editor for layer objects (*Layer Objects* (p. 126)) are used to control various properties of layers.

10.6.4. Layer Visibility

There are visibility check boxes next to each layer listed in the object selector. By right-clicking a layer object, a shortcut menu will appear, allowing the following visibility options as applicable:

- `Show`

  Makes the layer visible.

- `Show + Hide All Siblings`

  Makes the layer visible and turns off the visibility of all other layers.

- `Hide`

  Turns off the visibility of the layer.

The visibility of a layer can also be controlled by the visibility settings in the individual layer objects.

To change which parts of a layer are visible, select the layer(s) from the object selector, then right-click on the selection and select an appropriate render option from the shortcut menu. For details, see *Master Topology Visibility* (p. 126), *Topology Visibility* (p. 126), and *Refined Mesh Visibility* (p. 126).

10.6.5. The Layers Object

The `Layers` object is used to control the individual layer objects. You can access the Layers objects in the object selector.

10.6.5.1. Layers Tab

A number of possible operations are available for a layer by clicking it in the list box and then selecting one of the icons on the right of the list box, or by right-clicking the object and then selecting an operation from the shortcut menu.

10.6.5.1.1. New Layer

Clicking *New Layer* (or right-clicking a layer and selecting **New Layer**) creates a new layer at the most suitable span location as determined by ANSYS TurboGrid.

10.6.5.1.2. Delete Selected Layers

Clicking *Delete Selected Layer(s)* removes the selected layer(s). Hold **Shift** or **Ctrl** to select multiple layers to delete. You cannot delete the hub or the shroud layers.

10.6.5.1.3. Inserting Layers Automatically

Clicking *Auto Add Layers* in the editor for the `Layers` object, or selecting **Insert** > **Layers Automatically** from the shortcut menu for a layer object, causes ANSYS TurboGrid to insert layers automatically, using built-in heuristics to determine an appropriate number of layers to insert, and the location of each layer. The number of layers that ANSYS TurboGrid proposes to add is shown in the editor for the `Layers` object.

Selecting **Insert** > **Layer Automatically** from the shortcut menu for any layer object will cause ANSYS TurboGrid to insert one layer automatically using built-in heuristics to determine an appropriate location. The position of the inserted layer does not depend on which layer object you right-click to access the shortcut menu.

10.6.5.1.4. Insert Layer After Selected Layer

The **Insert Layer After** and **Insert** > **Layer After Selected Layer** shortcut menu items will cause a new layer to be created at the span halfway between the selected layer and the next layer. For example, if the only existing layers are on the hub and the shroud, and **Insert** > **Layer After Selected Layer** is selected from the hub, ANSYS TurboGrid inserts the new layer at a span of 0.5.

10.6.5.1.5. Span Location

The span location of a layer can be modified in the **Span Location** box.

10.6.5.2. Advanced Parameters Tab

10.6.5.2.1. Automatically generate required layers at mesh creation

If you leave **Automatically generate required layers at mesh creation** turned on, then, upon creating a mesh, ANSYS TurboGrid will check to see if the number of layers is adequate (according to its recom-

mended number of layers). If the current number of layers is deemed to be inadequate, new layers will be added automatically (before the mesh is generated) and a notification that layers are being added will appear in the status bar in the lower-left corner. The number of layers that ANSYS TurboGrid proposes to add is shown in the editor for the `Layers` object.

Note

The recommended number of layers is a recommendation based on blade shape, size of the topology around the blade, proximity of inlet/outlet, and so on. As such the recommendation is not always reliable.

10.6.5.2.2. Leading And Trailing Edge O-Grid Control Points

This option controls the constraint on the control point found at the leading or trailing edge of a layer on the outside edge of the O-Grid (provided that an O-Grid exists, and that the leading/trailing edge is not **Cut-off or square** or **Line of rotation on hub and shroud**, as specified in the editor for the blade).

The following methods are available:

- `Point`

  The point will be entirely constrained.

- `Curve`

  The point will be allowed to move along the outer edge of the O-Grid.

- `Surface`

  The point will be allowed to move freely within the layer.

10.6.5.2.3. Default Control Point Type

This setting controls the type (`Local` or `Master`) of control points that are added. After a control point has been added, its type can be changed. For details, see *Change control point type to Master* (p. 128) and *Change control point type to Local* (p. 128).

10.6.5.2.4. Orthogonality Factors

The **Orthogonality Factors** settings control how the elements are distributed along the blade surface for the hub or shroud layer. The values range from 0, meaning that the topology is aligned across the blade, to 1, meaning that the orthogonality of the passage mesh with respect to the blade is maximized (possibly at the expense of orthogonality across the blade). The orthogonality factors are interpolated to the intermediate layers using the values at the hub and shroud layers.

Another way to control the distribution of elements along the blade surface is to manually add and move control points along the blade surface. For details, see *Added Control Points* (p. 131). Due to the influence of orthogonality factors, though, such manual control of the element distribution can be restricted, especially near the blade leading and trailing edges, and when the displacements of the control points are large.

10.6.5.2.5. Floating the Leading and Trailing Edges on the Blade

The **Float leading and trailing edges on blade** option lets the O-Grid slide around the leading and trailing edges for better mesh quality on highly-twisted blades. Although it is possible to obtain similar results by adjusting the leading/trailing edge curves, it is more convenient to use this topology feature.

To cause the O-Grid to slide around the leading or trailing edge:

1. Select the **Float leading and trailing edges on blade** option on the **Advanced Parameters** tab of the `Layers` object and click **Apply**.

2. Introduce a master control point on the O-Grid, directly in front of the leading edge or behind the trailing edge, as applicable.

    Note that "cut-off or square" edges and sharp edges, have no such part of the O-Grid. The **Float leading and trailing edges on blade** option is only applicable for blades with single leading/trailing edge curves on edges that are not sharp.

3. Move the master control point along the O-Grid.

    The refined mesh will show that the mesh slides around the leading/trailing edge.

    Note

    The mesh may shift slightly on the blade when you select the **Float leading and trailing edges on blade** option, even if you have not moved any control points.

The figure below shows the effect of selecting the **Float leading and trailing edges on blade** option on the leading edge of a highly-twisted blade. The left side shows the option turned off, and the right side shows the option turned on. Note the distribution of elements on the blade, as shown by the refined mesh.
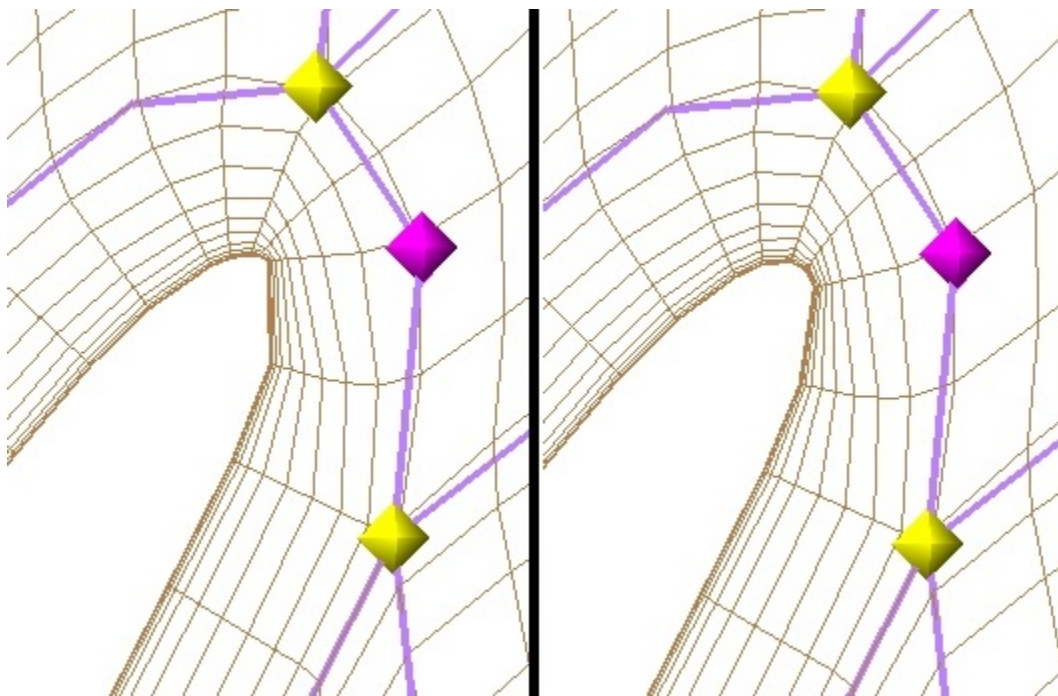
10.6.6. Layer Objects

You can access the layer objects under the `Layers` object in the object selector.

10.6.6.1. Data Tab

10.6.6.1.1. Master Topology Visibility

Clicking in the box will toggle the visibility of the master topology. The master topology is shown as violet line segments that connect the default master control points (not any added ones).

You can also change the **Master Topology Visibility** setting from the shortcut menu after selecting and right-clicking the layer(s) in the object selector, or right-clicking a layer in the viewer.

10.6.6.1.2. Topology Visibility

Clicking in the box will toggle the visibility of the topology. The topology is shown as yellow line segments.

You can also change the **Topology Visibility** setting from the shortcut menu after selecting and right-clicking the layer(s) in the object selector, or right-clicking a layer in the viewer.

10.6.6.1.3. Refined Mesh Visibility

Selecting **Refined Mesh Visibility** causes the refined mesh to appear in the viewer.

You can also change the **Refined Mesh Visibility** setting from the shortcut menu after selecting and right-clicking the layer(s) in the object selector, or right-clicking a layer in the viewer.

When **Refined Mesh Visibility** is selected, the following refined-mesh quality statistics—Mesh Measure—are available for viewing:

- `Minimum Face Angle`

- `Maximum Face Angle`

- `Maximum Aspect Ratio`

To display problem areas in the refined mesh, double-click one of the **Mesh Measure** statistics in the list. *Figure 10.16* (p. 127) shows an example of problem areas, shown with thick red lines, for the `Minimum Face Angle` statistic.

Figure 10.16 Refined Mesh Showing Areas of Unacceptable Minimum Face Angle
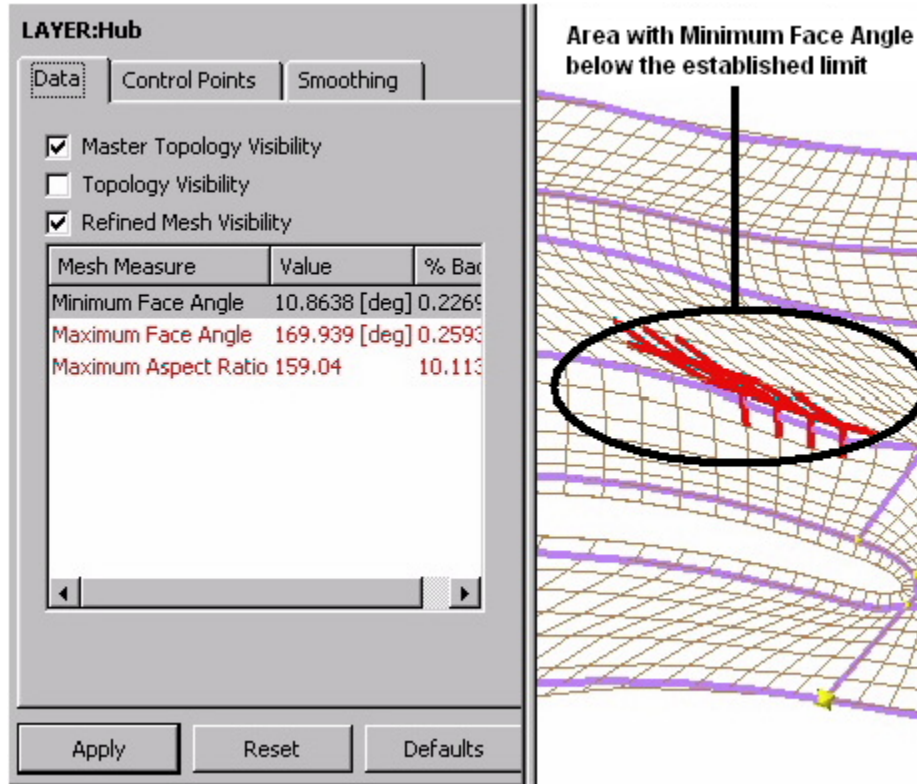
The `Mesh Limits` object holds the criteria that determine if the **Mesh Measure** statistics are acceptable.

If, for a given layer, **Refined Mesh Visibility** is turned on and either the `Minimum Face Angle` or `Maximum Face Angle` is outside the applicable limit, the layer will be listed in red text in the selector tree.

10.6.6.2. Control Points Tab

10.6.6.2.1. User Control Points

Toggles the application of all modifications to control points.

10.6.6.2.2. Override default control point size factor

This option allows you to override the default size of control points as they appear in the viewer.

10.6.6.2.3. New Control Point

After selecting *New Control Point* ⬚ , click the topology in the viewer where you want to add a control point. A new control point (of type Local) will be created at the nearest intersection of topology line segments (even if the topology line segments are not visible). At least one of the master topology, topology, or refined mesh must be visible.

10.6.6.2.4. New Control Points Mode

The *New Control Points Mode* ⊞ tool is similar to the *New Control Point* ⧉ tool, except that multiple control points can be added with successive mouse clicks. To stop adding control points, select this tool a second time.

10.6.6.2.5. Delete

The *Delete* ✗ tool deletes the selected control point(s).

10.6.6.2.6. Change control point type to Master

The *Change control point type to Master* ◇ tool changes the selected control point(s) to type Master.

10.6.6.2.7. Change control point type to Local

The *Change control point type to Local* ○ tool changes the selected control point(s) to type Local.

10.6.6.3. Smoothing Tab

On the **Smoothing** tab, the smoothing level of the refined mesh may be set to `None`, `Low`, `Medium`, `High`, or `Specify`. The first 4 items correspond with 0, 1, 3, and 5 smoothing iterations respectively. The `Specify` option allows you to enter the number of iterations.

10.6.7. Master Control Points

10.6.7.1. Definition and Purpose

Master control points are points that specify the location of key topology block corners on a given layer. They can be moved to improve the quality of a mesh. Unacceptable element face angles, edge length ratios, etc. in a particular region of a mesh can usually be eliminated by appropriately moving one or more master control points. For example, a common adjustment is to modify the position of a master control point on the leading edge of a blade to improve the orthogonality of the surrounding topology blocks.

The orientation of the wake region can be adjusted by modifying the master control points on the inlet and outlet (although it may be more beneficial to optimize the inlet and outlet control angles first. For details, see *Control Angle* (p. 88).).

Another use for master control points is to concentrate mesh elements for the purpose of resolving a flow feature. For example, certain master control points can be moved closer together in order to help resolve a wake or shock wave.

10.6.7.2. Availability

Most topologies include master control points when they are first created. You can move, but not delete, these master control points. Master control points can be added. For details, see *Added Control Points* (p. 131).

10.6.7.3. How to Select and Move a Control Point

You can select and move control points in the following ways:

- Select a control point by holding **Ctrl**+**Shift**, and clicking the mouse while pointing at a control point.

- Select and move a control point by holding **Ctrl**+**Shift** and dragging using the left mouse button. When the mouse button is released, the topology and mesh will update interactively.

- If you want to reset a control point to its original position, right-click the point and select **Reset Offset** from the shortcut menu.

- To select and drag control points without holding down **Ctrl**+**Shift**, you can click the *Select* icon, then select and drag control points with the left mouse button.

- You may also right-click a point and select **Edit** to select that point for editing.

Also see *Control Point Selection and Highlighting* (p. 132) and *Selecting and Dragging Objects while in Viewing Mode* (p. 55).

> **Note**
>
> When multiple instances of the layer are displayed, any instance can be used for selecting or adjusting control points, but only the first instance can be used for inserting control points.

> **Note**
>
> Moving a control point causes the mesh, if it exists, to be out-of-date. As a reminder that the mesh must be re-created, a yellow lightning bolt appears on the *Create Mesh* icon (i.e., the icon switches from ⊞ to ⚡). Moving a control point does not cause the deletion of the surface objects found under the `3D Mesh` object in the object selector.

10.6.7.4. Range of Influence
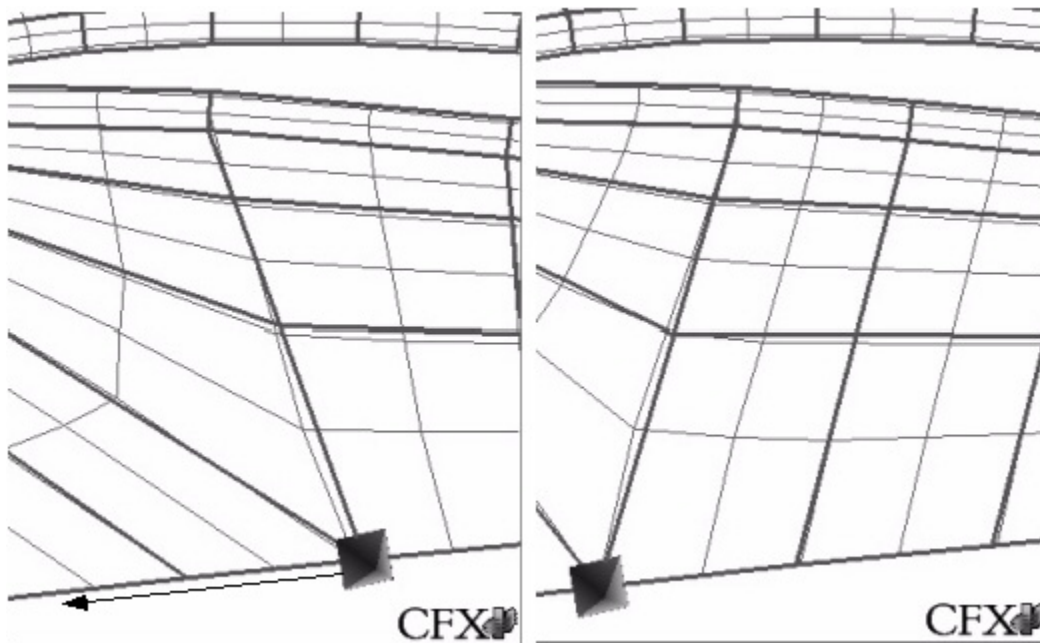
Master control points can be manually adjusted in order to adjust the layout of relatively large portions of the topology layout at a time. By contrast, local control points affect only the topology blocks that immediately surround them. For details, see *Local Control Points* (p. 131).

Moving a master control point causes some of the surrounding topology blocks to be adjusted in a way that maintains smooth trends in topology block size. This, in turn, promotes smooth trends in mesh element size.

The topology block edges that are influenced by a given master control point depend on the location of the latter, and are highlighted when the master control point is selected. An exception to this rule can occur when moving master control points along the surface of the blade. For details, see *Orthogonality Factors* (p. 124).

There are ways of restricting the range of influence of a master control point:

- The range of influence of a master control point can be reduced to extend only as far as the first topology node encountered on a master topology line. For details, see *Mixed Influence Command* (p. 140).

- You can limit the range of influence of an added master control point so that it stops at another master control point by making the latter "sticky". To make a control point "sticky", right-click it in the viewer and select the **Sticky** menu command. For details, see *Sticky Command* (p. 142).

10.6.7.5. Advice on Moving Master Control Points

Before moving master control points, you may want to adjust the inlet and outlet control angles if they are not suitable for your geometry. This can apply particularly for high angle blades. The control angle adjustment procedure is given in *Control Angle* (p. 88).

It is recommended that you adjust the master control points on the hub and shroud before adjusting those on the intermediate layers because the latter are initialized by the former. If you move control points on either the hub or the shroud layer after creating intermediate layers then, when you attempt to create a mesh, a message will be displayed asking if you want to re-initialize the master control points on the intermediate layers.

If you have introduced layers and then subsequently moved master control points on the hub and/or shroud, you will be prompted on whether or not to re-interpolate the control point positions on intermediate layers. It is generally recommended that you update intermediate layers by selecting **Update & Continue**.
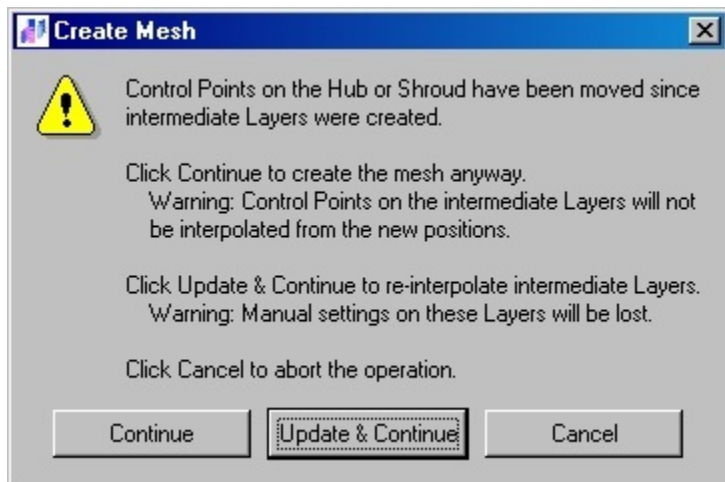
10.6.8. Local Control Points

Local control points can be manually adjusted in order to adjust the layout of the immediately adjacent topology blocks. By contrast, master control points affect larger portions of the topology. For details, see *Master Control Points* (p. 128).

10.6.9. Master versus Local Control Points

The difference between a control point of type **Master** and a control point of type **Local** is described in the following table.

Control Points of Type "Master"	Control Points of Type "Local"
represented by an octahedron	represented by a sphere
movement of the control point can influence topology line segments that are beyond those in contact with it	movement of the control point influences only those topology line segments in contact with it
preserves element size matching across the affected topology blocks	does not preserve element size matching across the affected topology blocks (uses uniform element sizes)

10.6.10. Added Control Points

Sometimes it is beneficial to move topology vertices that have no associated control point. In this case, you may be able to add a control point to such a vertex. This might be necessary, for example, near the leading edge to get fine control of the topology distribution.

How to Add a Control Point

A control point can be added using the mouse by holding **Ctrl**+**Shift** and right-clicking a topology vertex. Alternatively, you can add control points using the object editor as follows:

1. Open the object editor for a particular layer.

2. Click *New Control Point* ⬚ in the editor, then left click over the topology block corner where you want the control point added. A number of control points can be created simultaneously on a layer by clicking *New Control Points Mode* ⬚ in the editor for layer objects. Each left click will then add a new control point. Once you have completed creating new control points, click *New Control Points Mode* ⬚ a second time to exit from this mode.

> **Note**
>
> When multiple instances of the layer are displayed, only the first instance can be used for adding/selecting/adjusting control points.

You might find it helpful to turn off the visibility of the hub or shroud geometry object before attempting to add control points to the corresponding hub or shroud layer. This prevents the picking mechanism

from picking the geometry object instead of the intended topology vertex. An easy way to turn off all geometry objects is to select **Display** > **Hide Geometry Objects**. For details, see *Hide/Unhide Geometry Objects Commands* (p. 48).

A colored symbol will be drawn where a control point is added. Move it as you would move a master control point. For details, see *How to Select and Move a Control Point* (p. 129).

The color of the control point indicates its constraint:

- You can move a magenta control point in any direction on the layer.

- A yellow control point can be moved along a curve (e.g., the O-Grid boundary curve or periodic curves).

- It is not possible to move a red control point.

To delete a control point, select it in the object editor then click *Delete* ✖.

The size of the control points can be edited in the object editor.

By default, an added control point is of type Local (This default can be changed by the **Default Control Point Type** setting in the editor for the `Layers` object.). The type of a control point can be toggled to Master (and back to Local) by double-clicking it in the list in the editor for layer objects. Changes made in this way are specific to the current layer only. An alternative way to toggle the control point type is by right-clicking the point in the list and using the shortcut menu. This method allows changes to affect the current layer or all layers, depending on which command is selected. For details, see *Shortcut Menu Commands* (p. 136).

10.6.11. Control Point Selection and Highlighting

To select a control point, click *Select* ⬚ in the viewer (if necessary), then click the control point in the viewer. An alternative way to select a control point is to hold down **Ctrl**+**Shift** and click the point in the viewer. For added control points, a third method is to click the appropriate entry in the list of control points in the editor for layer objects on the **Data** tab. If selecting an added control point from the viewer, the appropriate editor for layer objects will open with the entry for the control point highlighted in the list of control points.

After *Select* ⬚ has been clicked, holding the mouse pointer over a control point (or any other object) will result in the type and name of the control point (or object) appearing in the lower-left corner of the ANSYS TurboGrid window.

After selecting a control point, the latter is shown highlighted with red lines. If any master topology edges would be affected by a movement of the selected control point, they are highlighted with red lines.

Also see *How to Select and Move a Control Point* (p. 129).

10.6.12. Saving Layers to State Files

When saving a state file that contains layers, make sure that the topology is frozen (see *Freeze Button* (p. 111)) and included in the state file. On loading the state file, the layer information will then be applied to the topology that existed at the time the state file was saved.

10.6.13. Loading Layers from State Files

Before loading a state file that contains layers, reset the topology to `None` so that any existing topology, layers, and control points are deleted.

10.7. 3D Mesh

The 3D mesh generated by ANSYS TurboGrid has objects associated with it. These objects are useful for generating and examining the mesh.

10.7.1. The 3D Mesh Object

The `3D Mesh` object contains a **Generate** button, which is enabled when it is possible to generate (or refresh) a mesh. This button creates the mesh, and is the same as selecting **Insert** > **Mesh** from the main menu or clicking *Create Mesh* 🖺.

You can open the `3D Mesh` object to see node and element counts for the passage, inlet, outlet, and the whole mesh.

10.7.2. Surface Group and Turbo Surface Objects

The `3D Mesh` branch contains objects which display the mesh on geometry surfaces and turbo surfaces. Of these display objects, only the turbo surface object `Show Mesh` is visible by default. To view another surface, select the visibility check box for that object.

The mesh can be displayed only after it has been created. Changes made to the `Geometry`, `Topology`, or `Mesh Data` objects will cause the mesh, and the surface objects found under the `3D Mesh` object, to be deleted. Control point movement causes the mesh to be out-of-date if it exists, but does not delete or affect the surface objects found under the `3D Mesh` object. Control point movement also causes a yellow lightning bolt to appear on the *Create Mesh* icon (i.e., the icon switches from 🖺 to 🖺) to serve as a reminder that the mesh must be re-created, and that the `3D Mesh` surface objects are no longer up-to-date.

10.7.2.1. 3D Mesh Turbo Surfaces

The turbo surface is a flexible object that allows you to closely examine every area of the mesh. One turbo surface exists in the `3D Mesh` branch of the object selector after you create a mesh. You can create other turbo surfaces. For details, see *Turbo Surface Command* (p. 36).

10.7.2.2. 3D Mesh Surface Groups

A collection of surface groups exists in the 3D Mesh branch of the object selector after you create a mesh. For details, see *Surface Groups* (p. 41).

10.8. Mesh Analysis

The `Mesh Analysis` branch offers a variety of tools to analyze the quality of the mesh. The mesh can be analyzed only after it has been created. If changes are made to the `Geometry`, `Topology`, `Layers`, or `Mesh Data` objects, the mesh cannot be analyzed again until it has been recreated. For details, see *Mesh Command* (p. 31).

10.8.1. Mesh Statistics
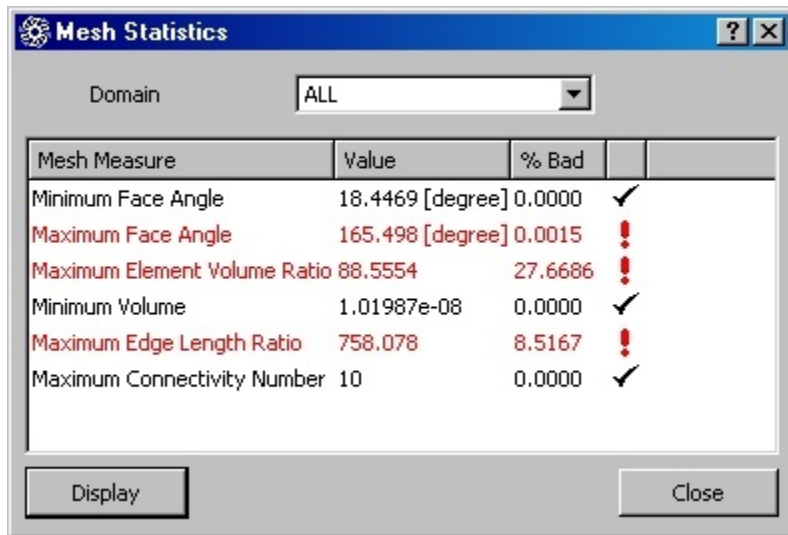
The `Mesh Statistics` object provides details about the quality of the current mesh. The `Mesh Statistics` object cannot be viewed until the mesh is created. For details, see *Steps to Create a Mesh* (p. 71). To view the mesh statistics, double-click the `Mesh Analysis` object or `Mesh Statistics` object in the object selector.

The `Mesh Statistics` object does not open in the object editor like the other objects in the object selector. The `Mesh Statistics` object is in a window of its own so it can stay open while other objects are edited in the object editor. This allows for constant monitoring of the statistics.

The available mesh statistics include `Minimum Face Angle`, `Maximum Face Angle`, `Element Volume Ratio`, `Edge Length Ratio` and `Connectivity Number`. Statistics displayed in red are outside the limits defined in the `Mesh Analysis` > `Mesh Limits` object. For details, see *Mesh Limits* (p. 134). To display the regions where the mesh statistics are outside the limits, select the mesh measure, then click **Display**. Alternatively, double-click the mesh analysis variable in the list.

Each calculated mesh measure is added to the list of available variables for creating new plots (e.g., a contour plot).

10.8.2. Mesh Limits

The `Mesh Limits` object defines the acceptable values for the mesh analysis variables. These variables are described here. The default limits are generally good and should highlight any problem areas of the mesh.

10.8.2.1. Maximum Face Angle

This is the greatest face angle for all faces that touch the node. For each face, the angle between the two edges of the face that touch the node is calculated. The largest angle from all faces is returned. The maximum face angle can be considered to be a measure of skew.

10.8.2.2. Minimum Face Angle

This is the smallest face angle for all faces that touch the node.

10.8.2.3. Connectivity Number

`Connectivity Number` is the number of elements that touch a node. This variable is the maximum connectivity number on any element. For the unstructured ANSYS CFX solver, this value is not important. However, high connectivity numbers in much of the mesh can have an adverse effect on the speed of the structured CFX-TASCflow solver.

10.8.2.4. Element Volume Ratio

`Element Volume Ratio` is defined as the ratio of the maximum volume of an element that touches a node, to the minimum volume of an element that touches a node. The value returned can be used as a measure of the local expansion factor.

10.8.2.5. Minimum Volume

`Minimum Volume` is used to ensure that no negative volumes are created within the passage. The value given is that of the minimum volume of an element touching any of the nodes.

10.8.2.6. Edge Length Ratio

This is a ratio of the longest edge of a face divided by the shortest edge of the face. For each face,

$$\frac{max\ (l_1, l_2)}{min\ (l_1, l_2)}$$

is calculated for the two edges of the face that touch the node. The largest ratio is returned. The edge length ratio can be considered to be a measure of aspect ratio.

10.8.3. Mesh Statistics Parameters - Order Of Importance

Generally, the mesh statistics can be ranked as follows (most important to least important)

1. `Minimum Volume` - these MUST be fixed before the mesh would be usable.

2. `Maximum Face Angle`/`Minimum Face Angle` - these should be improved until they fall within the constraints (minimum of 15° and maximum of 165°), if possible. Values close to, but just outside the constraints may still be acceptable for your simulation.

3. `Edge Length Ratio` - this can often be fixed by increasing the number of elements from hub to shroud. The default limit is 100, so values close to this will normally be acceptable.

4. `Element Volume Ratio` - depending on the mesh, it may not be possible to satisfy this constraint.

5. `Connectivity Number` - may or may not be pertinent depending on the type of solver used. For details, see *Connectivity Number* (p. 135).

10.8.4. Volume

The `Mesh Analysis > Show Limits` volume object provides a convenient way to examine the mesh. It functions exactly the same as the volume object in the create menu, except that the only variables available are the mesh statistics variables. For details, see *Volume Command* (p. 37).

> **Note**
>
> The visibility of the volume object is off by default.

10.9. User Defined Objects

The `User Defined` branch in the object selector is initially empty. Any new objects created using the **Insert** menu will be displayed in this branch. Examples include points and legends. For details, see *Insert Menu* (p. 31).

10.10. Default Instance Transform

The default instance transform is initially applied to all objects for which instance transforms are possible. As a result, editing the definition of the default instance transform causes all such plots and objects to be transformed. Additional instance transform objects can be created. For details, see *Instance Transform Command* (p. 44). A different instance transform can be applied to an object using its **Render** tab. For details, see *Applying Instance Transforms to Objects* (p. 8).

10.11. Shortcut Menu Commands

You can perform many actions from shortcut menus; simply right-click one of the following:

- The viewer background (i.e., a blank area in the viewer)

- An object in the viewer

- An object listed in the object selector

- An object listed in the object editor

The following sections describe all of the shortcut menu commands that can appear:

- *Auto Add Layers and Insert Layers Automatically Commands* (p. 137)

- *Color Command* (p. 137)

- *Copy Control Points to Hub and Shroud Command* (p. 138)

- *Copy Smoothing Levels to All Layers Command* (p. 138)

- *Copy to Hub, Copy All to Hub, and Copy Control Points to Hub Commands* (p. 138)

- *Copy to Shroud, Copy All to Shroud, and Copy Control Points to Shroud Commands* (p. 138)

- *Create Mesh Command* (p. 138)

- *Create New View Command* (p. 138)

- *Delete Command* (p. 139)

- *Delete New View Command* (p. 139)

- *Edit Command* (p. 139)

- *Edit in Command Editor Command* (p. 139)

- *Fit View Command* (p. 139)

- *Hide Command* (p. 139)

- *Insert Blade Command* (p. 139)

- *Insert Layer After and Insert Layer After Selected Layer Commands* (p. 139)
- *Insert Layer Automatically Command* (p. 139)
- *Insert Local and Insert Master Commands* (p. 140)
- *Insert USER DEFINED Object Command* (p. 140)
- *Insert Edge Split Control Command* (p. 140)
- *Interpolating Control Point Offsets for Inner Layers* (p. 140)
- *Make Local Command* (p. 140)
- *Make Master Command* (p. 140)
- *Master Influence Command* (p. 140)
- *Mixed Influence Command* (p. 140)
- *Predefined Camera Commands* (p. 141)
- *Save Picture Command* (p. 141)
- *Projection Commands* (p. 141)
- *Render Properties Edit Options Command* (p. 141)
- *Render Properties Show Curves Command* (p. 141)
- *Render Properties Show Surfaces Command* (p. 141)
- *Render Properties Topology and Refined Mesh Visibility Commands* (p. 142)
- *Reset Offset Command* (p. 142)
- *Show Object and Show Commands* (p. 142)
- *Show and Hide All Siblings Command* (p. 142)
- *Sticky Command* (p. 142)
- *Suspend Object Updates Command* (p. 142)
- *Toggle Axis Visibility Command* (p. 143)
- *Toggle Ruler Visibility Command* (p. 143)
- *Transformation Commands and Coordinate Systems* (p. 143)
- *Update Now Command* (p. 144)
- *Viewer Options Command* (p. 144)

10.11.1. Auto Add Layers and Insert Layers Automatically Commands

For details, see *Inserting Layers Automatically* (p. 123).

10.11.2. Color Command

The **Color** command provides a subset of the functionality offered by the **Color** tab. The latter is available by selecting an eligible object from the object selector, then clicking *Rendering* . For details, see *Color Tab* (p. 5).

10.11.3. Copy Control Points to Hub and Shroud Command

The **Copy Control Points to Hub & Shroud** command is available for any layer other than the hub or shroud layer. This command has the combined effect of the **Copy Control Points to Hub** and **Copy Control Points to Shroud** commands.

10.11.4. Copy Smoothing Levels to All Layers Command

The **Copy Smoothing Levels to All Layers** command is available by right-clicking on a layer at a location other than a control point (i.e., on a master topology, topology, or refined mesh line), or by right-clicking a layer listed in the object selector. This command takes the smoothing setting found on the **Smoothing** tab for the layer (in the object editor) and applies it to all layers.

10.11.5. Copy to Hub, Copy All to Hub, and Copy Control Points to Hub Commands

The **Copy to Hub** command is available by right-clicking a control point on the shroud layer. It copies the offset of the control point to the corresponding control point on the hub layer, and creates the corresponding control point if it does not already exist.

The **Control Point** > **Copy All to Hub** command is available by right-clicking on a layer at a location other than a control point (i.e., on a master topology, topology, or refined mesh line). This command carries out the **Copy to Hub** command for all control points on the layer.

The **Copy Control Points to Hub** command is available for the shroud layer when you select the latter from the object selector. It is the same as the **Control Point** > **Copy All to Hub** command.

10.11.6. Copy to Shroud, Copy All to Shroud, and Copy Control Points to Shroud Commands

The **Copy to Shroud** command is available by right-clicking a control point on the hub layer. It copies the offset of the control point to the corresponding control point on the shroud layer, and creates the corresponding control point if it does not already exist.

The **Control Point** > **Copy All to Shroud** command is available by right-clicking on a layer at a location other than a control point (i.e., on a master topology, topology, or refined mesh line). This command carries out the **Copy to Shroud** command for all control points on the layer.

The **Copy Control Points to Shroud** command is available for the hub layer when you select the latter from the object selector. It is the same as the **Control Point** > **Copy All to Shroud** command.

10.11.7. Create Mesh Command

The **Create Mesh** command is available when the topology has been created. It is the same as selecting **Insert** > **Mesh**. For details, see *Mesh Command* (p. 31).

10.11.8. Create New View Command

Adds a new view based on the current state of the viewer.

10.11.9. Delete Command

The **Delete** command is analogous to **Make Master** command, and operates on added control points.

10.11.10. Delete New View Command

Deletes an existing user-defined view.

10.11.11. Edit Command

The **Edit** command opens the object editor for the selected object.

10.11.12. Edit in Command Editor Command

The **Edit in Command Editor** command opens the command editor dialog box for the selected object. For details, see *Command Editor Command* (p. 66).

10.11.13. Fit View Command

The **Fit View** command centers the view on the displayed objects and sets the zoom to an appropriate level for viewing all of the visible objects. Only the active viewport is affected by this command.

Note

Objects which have their visibility check box cleared do not necessarily fit into the viewer. Only visible objects are considered.

10.11.14. Hide Command

The **Hide** command turns off the visibility of the selected object(s). To turn the visibility back on, find the object in the object selector and select its check box or use the **Show Object** command available in the shortcut menu that appears when right-clicking the viewer background.

10.11.15. Insert Blade Command

The **Insert Blade** command adds a blade to the blade set. After selecting this command, ANSYS TurboGrid prompts you for a name for the blade, then generates the new blade object. If you already have a blade loaded, most settings for the new blade will default to those from the first blade. Simply change the **File Name** setting to indicate the file for the added blade and click **Apply**.

10.11.16. Insert Layer After and Insert Layer After Selected Layer Commands

For details, see *Insert Layer After Selected Layer* (p. 123).

10.11.17. Insert Layer Automatically Command

For details, see *Inserting Layers Automatically* (p. 123).

10.11.18. Insert Local and Insert Master Commands

The **Insert Local** and **Insert Master** commands are available by right-clicking on a layer at a location other than a control point (i.e., on a master topology, topology, or refined mesh line). These commands insert a control point of the corresponding type on the layer at the nearest suitable location to where you right-click.

10.11.19. Insert USER DEFINED Object Command

The **Insert** > **USER DEFINED Object** command creates a new object. It functions like the **Insert** menu. For details, see *Insert Menu* (p. 31).

10.11.20. Insert Edge Split Control Command

The **Insert Edge Split Control** command is available by right-clicking a topology line between topology nodes. This command creates/edits an edge split control object. Edge split control objects appear in the object selector under `Mesh Data`. For details, see *Edge Split Controls* (p. 121).

10.11.21. Interpolating Control Point Offsets for Inner Layers

If you have more than two layers, and you move a control point on the hub or shroud layer, then the next time you try to generate a mesh, the control points on the inner layers will move. If you manually adjust any control point on any inner layer, then move a control point on the hub or shroud layer, ANSYS TurboGrid will prompt you the next time you try to generate a mesh. The prompt will offer the choice of keeping your custom movements for the inner layers, or recalculating all control points for all inner layers. In order to recalculate all control points for a single layer, or subset of layers, select the layer(s), then use the shortcut menu option: **Interpolate CP Offsets**, or **Control Point** > **Interpolate Offsets**. You may select an inner layer by right clicking it in the viewer. You may select one or more inner layers using the object selector, or the list of layers that appears when editing the `Layers` object in the object editor.

10.11.22. Make Local Command

The **Make Local** command is analogous to **Make Master** command, and operates on added master control points.

10.11.23. Make Master Command

The **Make Master** command is available by right-clicking a local control point. It can be used to make the local control point type Master by selecting **Make Master** > **on Current Layer**, It can be used to make the local control point, and all of the control points in the corresponding location on the other layers, type Master by selecting **Make Master** > **on All Layers**.

10.11.24. Master Influence Command

The **Master Influence** command removes the effect of the **Mixed Influence** command. For details, see *Mixed Influence Command* (p. 140).

10.11.25. Mixed Influence Command

When a control point is selected, a red line indicates the range of influence on the topology nodes. For details, see *Range of Influence* (p. 129).

The **Mixed Influence** command reduces the range of influence of a master control point along master topology lines. After this command has been applied to a master control point other than a default master control point, the range of influence of the control point stops at the first topology node encountered on any master topology line.

10.11.26. Predefined Camera Commands

The **Predefined Camera** commands allow you to set the viewing angle according to one of the built-in presets. These commands also set the zoom level automatically after adjusting the viewing angle.

10.11.27. Save Picture Command

The **Save Picture** command is available by right-clicking the background. It is the same as selecting **File** > **Save Picture**.

10.11.28. Projection Commands

10.11.28.1. Perspective Command

The **Perspective** command sets a view with a fixed amount of perspective.

10.11.28.2. Orthographic Command

The **Orthographic** command sets an orthographic view.

10.11.29. Render Properties Edit Options Command

The **Render (Properties)** > **Edit Options** command enables you to adjust color and render settings for the selected object. For details, see *Color Tab* (p. 5) and *Render Tab* (p. 6).

10.11.30. Render Properties Show Curves Command

The **Render (Properties)** > **Show Curves** command is available by right-clicking an object (e.g., the hub, shroud or blade surface). Use this command to view the raw curve file data instead of the surface.

10.11.31. Render Properties Show Surfaces Command

The **Render (Properties)** > **Show Surfaces** command is available by right-clicking the hub or shroud curve, or one of the blade curves. Use this command to view the surface instead of the raw curve file data.

Note

The blade surface and resulting mesh can be adversely affected when a blade shroud tip is defined by the second-last profile and the last profile has a shape that is inconsistent with where the blade would exist if the blade were extended in the spanwise direction while ignoring the last profile. To avoid this problem, you can either ensure that the last profile is consistent with the second last profile, or manually remove the last profile from the curve file. A similar statement applies for the hub end of a blade having a hub tip.

10.11.32. Render Properties Topology and Refined Mesh Visibility Commands

The **Render (Properties)** > **Master Topology Visibility**, **Topology Visibility**, and **Refined Mesh Visibility** commands toggle the visibility of the corresponding items for the selected layer(s).

10.11.33. Reset Offset Command

The **Reset Offset** command is available by right-clicking a control point that has been moved since it was created. It resets the position of the control point to its original location.

10.11.34. Show Object and Show Commands

The **Show Object** command is available by right-clicking the background. It turns on the visibility of any geometry, volume, or mesh data object.

The **Show** command turns on the visibility of the object(s) selected in the object selector.

10.11.35. Show and Hide All Siblings Command

The **Show + Hide All Siblings** command turns on the visibility of the object selected in the object selector, and turns off the visibility of every sibling object in the tree.

10.11.36. Sticky Command

The **Sticky** command, when selected, makes a master control point "sticky". A "sticky" master control point usually limits the range of influence of an added master control point that lies on the same master topology line; the range of influence stops at the "sticky" master control point. In some cases, a "sticky" master control point does not restrict the range of influence of another master control point on the same topology line.

Note

A sticky control point will not remain stationary if you move a pre-defined master control point on the same master topology line.

10.11.37. Suspend Object Updates Command

You can save time by appropriately suspending objects. The **Suspend Object Updates** command toggles the state of suspension of an object. By suspending an object, you prevent it from being updated in response to changes to other objects.

For example, the `Topology Set` object depends on the `Geometry` object, and, when not suspended, will be reprocessed each time the geometry is changed. If you want to make several changes to the geometry, you can skip the intermediate processing of the topology by suspending the `Topology Set` object. Note that the `Topology Set` object is initially suspended when you start a new case.

To process a suspended object, you can either deselect the **Suspend Object Updates** command or select the **Update Now** command. If you select the **Update Now** command, the object will be processed once, but will remain suspended.

Objects that depend on a suspended object are also suspended. An icon overlay indicates whether or not an object is suspended. Icon overlays are illustrated in *Table 1.1: Icon Overlays and Text Styles* (p. 2).

10.11.38. Toggle Axis Visibility Command

The **Toggle Axis Visibility** command is available by right-clicking the background. It controls the visibility of the axis orientation indicator that appears in the lower-right corner of the viewer.

10.11.39. Toggle Ruler Visibility Command

The **Toggle Ruler Visibility** command is available by right-clicking the background. It controls the visibility of the ruler that appears near the lower edge of the viewer.

10.11.40. Transformation Commands and Coordinate Systems

The **Transformation** commands are available by right-clicking the background.

You can use different coordinate systems to display various views of the geometry and mesh objects in the viewer window. This is useful when closely examining every aspect of the mesh before saving it for use in a CFD solver.

10.11.40.1. Cartesian

The Cartesian coordinate system is used by default in the viewer window when there is only one viewport. The 3 axis coordinates for the Cartesian coordinate system are X, Y and Z.

10.11.40.2. Blade-to-Blade

The blade-to-blade coordinate system is used to display the geometry in the $m' - \theta$ conformal space which is familiar to blade designers. The $m' - \theta$ coordinate space is angle-preserving and minimizes the effect of changing radius on viewing and manipulation. By utilizing the blade-to-blade coordinate system, a wide variety of machine types, from axial to radial, can be treated similarly.

m' is defined in *Meridional* .

Note

Due to the fact that m' is ill-conditioned at $r = 0$, you can expect to see different behavior in cases which extend to the machine axis.

10.11.40.3. Meridional

The meridional coordinate system is one transformation used by blade designers. It is useful for viewing the flow path as well as the upstream and downstream extents of the mesh. The three axis coordinates for the meridional coordinate system are A (axial), R (radial) and θ (Theta). The viewer shows only the A and R coordinates.

Variable	Description
a	Axial location
r	Radius or radial location
s	Normalized distance along meridional curve (e.g., from 0 to 1)

Variable	Description
S	The particular value of s that corresponds to the point location for which m and m' are to be computed
m	Meridional coordinate (distance along meridional curve)
m'	Normalized meridional coordinate (radius normalized distance along meridional curve)

$$m = \int_0^S \sqrt{\left(\frac{dr}{ds}\right)^2 + \left(\frac{da}{ds}\right)^2} \, ds$$

$$m' = \int_0^S \left[\frac{\sqrt{\left(\frac{dr}{ds}\right)^2 + \left(\frac{da}{ds}\right)^2}}{r} \right] ds$$

10.11.40.4. 3D Turbo

The three axis coordinates for the 3D Turbo Coordinate System are M (m'), T (Theta) and S (span).

10.11.41. Update Now Command

The **Update Now** command causes a suspended object to be processed. For more details, see *Suspend Object Updates Command* .

10.11.42. Viewer Options Command

The **Viewer Options** command is available by right-clicking the background. It is the same as selecting **Edit** > **Options** and then navigating to **TurboGrid** > **Viewer**. For details, see *Viewer* .

Index

Symbols
3D Turbo coordinate system, 144

A
ANSYS CFX, 18
ANSYS TurboGrid
    workflow, 1, 71
ANSYS Workbench parameter, 63
appearance, 24
    font, 25
    GUI (Graphical User Interface), 25
append
    session, 29
    state, 14
append to current simulation, 14
apply button, 5
area function, 58
areaAve function, 58
areaInt function, 58
auto save, 24
automatic y+, 115
ave function, 59
axis visibility, 24

B
bitmap (bmp), 21
blade, 77
    hub tip, 86
    leading edge, 84
    save, 16
    shroud tip, 86
    trailing edge, 85
    transformations, 84
blade set, 13, 77
blade-to-blade coordinate system, 143
bmp (bitmap), 21
boundary condition
    file, 10

C
calculator, 57
    function, 57
Cartesian coordinate system, 143–144
CFD Viewer State (3D), 21
CFX-TASCflow, 18
CGNS, 18
color
    range, 6

tab in object editor, 5
    undefined, 6
command editor, 140
command editor dialog box, 66
    shortcut menu access, 139
command timeout, 26
commands
    expression, 62
connectivity number, 135
conservative variable values, 65
contour
    insert, 42
    range, 43
control angle
    inlet, 88
control points
    added, 131
    interpolation on inner layers, 140
    local, 128
    master, 128
    master and local types, 124, 132
    selection and highlighting, 132
    sticky, 130, 142
    user, 127
coordinate systems, 143
count function, 59
culling, 7
curve
    file, 10
    load, 12
cut line, 33
cut-off or square leading or trailing edge, 83

D
default instance transform, 136
defaults button, 5
direction, 61
display
    menu, 47
domain
    inlet, 120
    outlet, 120
double buffering, 25
draw
    faces, 7
    lines, 8

E
edge length ratio, 135
edge split control, 121
    automatic generation of, 113
    insertion of, 140

N

near wall element size specification, 115
new
    session, 29
number of elements, 113

O

O-Grid topology, 103
object
    culling, 7
    instance transform, 8
object editor, 4
    color tab, 5
    common options, 5
    geometry, 5
    render tab, 6
    shortcut menu, 136
object selector, 2
    shortcut menu, 136
options
    appearance, 24
    units, 26
    viewer, 24
orthographic, 141
outlet, 88
    domain, 120
outline, 90
overwrite
    session, 29
    state, 14

P

path variables, 40–41
pattern
    topology, 100
periodic surface
    bias towards high periodic, 84
periodic surfaces
    high, 87
    low, 87
perspective, 141
picking mode, 52
picture
    format, 20
    quality, 22
    scale, 22
    use screen capture, 21
pictures, 20
pivot point, 26
plane
    bounds, 35

    insert, 34
    sample, 36
    selection, 36
    slice, 36
    translation, 36
play
    session, 28
png (Portable Network Graphics), 21
point
    insert, 32
    selection, 33
    translation, 33
PostScript (ps), 21
ppm (Portable Pixel Map), 21
preserve control points on layers, 105
probe function, 60
ps (PostScript), 21

Q

qualitative function, 57
quit, 22

R

range
    color, 6
    contour, 43
recent
    files, 22
record session, 29
redo, 23
render
    faces, 7
    instance transforms, 8
    lines, 8
    tab in object editor, 6
reread button, 84
reset button, 5
rotate, 25, 52

S

sample
    line, 34
    plane, 36
save
    all objects check box, 15
    auto, 24
    blade, 16
    mesh, 17
    pictures, 20
    state, 15
    topology, 17