CFD EXPERTS

Simulate the Future

WWW.CFDEXPERTS.NET

Ansys 2021 R2
Engineering What's Ahead.

Ansys TurboGrid Tutorials

Ansys
ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2021 R2
July 2021

Published in the U.S.A.

Table of Contents

List of Figures

Chapter 1: Introduction to the Ansys TurboGrid Tutorials

The Ansys TurboGrid tutorials are designed to introduce general mesh-generation techniques used in Ansys TurboGrid.

Note:

Unless otherwise stated, each tutorial assumes that you are using Ansys TurboGrid in stand-alone mode.

You should review the following topics before attempting to start a tutorial for the first time:

1.1. Setting the Working Directory and Starting Ansys TurboGrid

Before you start Ansys TurboGrid, set the working directory.

1. Start the Ansys TurboGrid Launcher.
2. Select a working directory.
3. Click the **TurboGrid 2021 R2** button.

1.2. Changing the Display Colors

If viewing objects in Ansys TurboGrid becomes difficult due to contrast with the background, the colors can be altered for improved viewing. You can access the color options by going through the following steps:

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

   The **Options** dialog box appears.

2. Adjust the color settings under `TurboGrid` > `Viewer`.
3. Click **OK**.

1.3. Editor Buttons

The Ansys TurboGrid interface uses editors to enter the data required to create a mesh. The editors have standard buttons, which are described next:

- **Apply** applies the information contained within all the tabs of an editor.
- **OK** is the same as **Apply**, except that the editor automatically closes.
- **Cancel** and **Close** both close the editor without applying or saving any changes.
- **Reset** 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 the **Apply** button is clicked.
- **Defaults** restores the system default settings for all the tabs of the edited object.

1.4. Using Help

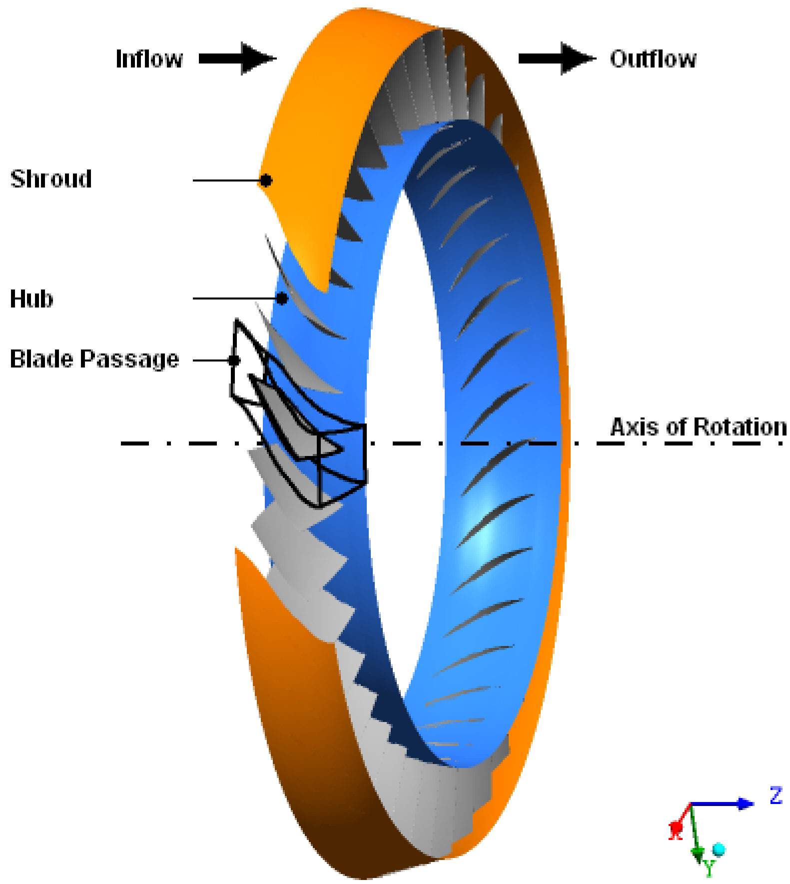
To open the Ansys Help, select **Help** > **Contents**.

Context-sensitive help is provided for many of the object editors and other parts of the interface. To invoke the context-sensitive help for a particular editor or other feature, ensure that the window is active, place the mouse pointer over the feature, and press **F1**. Not every area of the interface supports context-sensitive help.

Chapter 2: Rotor 37

This tutorial includes:

This tutorial demonstrates the basic workflow for generating a CFD mesh using Ansys TurboGrid. As you work through this tutorial, you will create a mesh for a blade passage of an axial compressor blade row. A typical blade passage is shown by the black outline in the figure below.



The blade row contains 36 blades that revolve about the negative Z axis. A clearance gap exists between the blades and the shroud, with a width of 2.5% of the total span. Within the blade passage, the maximum diameter of the shroud is approximately 51 cm.

You will save the mesh in a format that can be used by Ansys CFX in a CFD simulation.

If this is the first tutorial you are working with, it is important to review Introduction to the Ansys TurboGrid Tutorials (p. 7) before beginning.

2.1. Overview of the Mesh Creation Process

Before Ansys TurboGrid can create a mesh, you must provide it with several pieces of information. Such information includes the location of the geometry files (hub, shroud, and blades), the mesh topology type, and the distribution of mesh nodes. All of the data that you provide is stored in a set of data objects known as *CCL objects*.

The Ansys TurboGrid user interface organizes the CCL objects in a tree view known as the *object selector*. You can use the object selector to select and edit the CCL objects; the objects are listed from top to bottom in the standard order for creating a mesh. The user interface also has a toolbar for selecting and editing the CCL objects; the icons are arranged from left to right in the standard order for creating a mesh.

Regardless of whether you use the object selector or the toolbar, you should generally follow this sequence when creating a mesh:

1. Define the geometry by loading files and changing settings as required.

2. Optionally choose a topology method, then unsuspend the `Topology Set` object if necessary.

3. Optionally modify the `Mesh Data` settings that govern the number and the distribution of nodes in various parts of the mesh.

   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 other aspects of the mesh. Note that changing the mesh density can affect the mesh quality.

4. Optionally add intermediate 2D layers that guide the 3D mesh in the spanwise direction. By default, layers are added as required when you generate the topology or the 3D mesh.

5. Check the mesh quality. As required, make adjustments to the `Mesh Data` settings and the number and distribution of layers.

6. Save the mesh and state as required.

2.2. Preparing the Working Directory

1. Create a working directory.

   Ansys TurboGrid uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `rotor37.zip` file here.

3. Unzip `rotor37.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `BladeGen.inf`
   - `profile.curve`
   - `hub.curve`
   - `shroud.curve`

4. Set the working directory and start Ansys TurboGrid.

   For details, see Setting the Working Directory and Starting Ansys TurboGrid (p. 7).

2.3. Defining the Geometry

The provided geometry files, which consist of a `BladeGen.inf` file plus three curve files, were created using BladeGen. To load the information contained in those files, you will load the `BladeGen.inf` file. Ansys TurboGrid uses this file to set the axis of rotation, the number of blades, and a length unit that characterizes the scale of the machine. It also uses this file to identify the curve files, which it then loads to define the curvature of the hub, shroud, and a single blade. The geometric data from the input files is processed to generate a geometric representation, an outline of which appears in the viewer.

After the geometry has been generated, you are invited to browse through the objects created under the `Geometry` object in the object selector.

Initially, the blades extend from the hub to the shroud. After inspecting the geometry, you will create the required gap between the blade and the shroud.

Load the `BladeGen.inf` file:

1. Click **File** > **Load TurboGrid Init File**.
2. Open `BladeGen.inf` from the working directory.

   The progress bar at the bottom right of the screen shows the geometry generation progress. After the geometry has been generated, the viewer in the **Mesh** workspace shows the hub, shroud, and blade for one passage. Along the blade, you can see the leading and trailing edge curves (green and red lines, respectively). An outline drawing (the `Outline` object) traces the 3D space that is available for meshing; the latter consists of an inlet domain, passage, and outlet domain. In this tutorial, you will generate a mesh for the passage only.

   Note:

   It is possible to adjust the upstream and downstream extents of the hub and shroud surfaces (by changing the `Inlet` and `Outlet` geometry objects). It is also possible to create an extended mesh that includes the inlet and outlet domains (by editing the `Mesh Data` settings).

Examine the geometry in the **3D Viewer**:

1. In the **Mesh** workspace, toggle the visibility check box next to each object in the object selector and observe the change in the viewer.

   Note the correlation between the geometry objects listed in the object selector and the locations in the geometry.
2. In order to avoid cluttering the view, ensure that the visibility is turned on only for these objects: `Hub`, `Shroud`, `Blade 1`, `Outline`.

Examine and set the machine type:

1. Open `Geometry` > `Machine Data` from the object selector by double-clicking `Machine Data` in the object selector, or by right-clicking `Machine Data` and selecting **Edit** from the shortcut menu that appears.

   Here you can see basic information about the geometry. Note that the units specified for `Base Units` represent the scale of the geometry being meshed; these units are not used for importing geometric data nor do they govern the units written to a mesh file; they are used for the internal representation of the geometry to minimize computer round-off errors.

2. Set **Machine Type** to `Axial Compressor`.

   Setting the machine type helps TurboGrid choose appropriate topology templates later in the mesh creation process.

3. Click **Apply**.

Examine the hub:

1. Open `Geometry` > `Hub`.

   Here you can see information about which file was used for hub data and how the file was interpreted. Similar information can be seen by opening the `Shroud` and `Blade 1` objects. Note that, for the `Hub` and `Shroud` objects, the **Curve Type** parameter is set to `Piece-wise linear`; this is a result of loading a `BladeGen.inf` file.

2. Click *Display all blade instances* to obtain a view of the entire geometry.

3. Click *Display single blade instance* to show a single blade instance once again.

To complete the geometry, create a small gap between the blade and the shroud. The blade should be shortened to 97.5% of its original span because the gap width, as specified in the problem description, is 2.5% of the total span.

1. Open `Geometry` > `Blade Set` > `Shroud Tip`.

2. Set **Tip Option** to `Constant Span`.

3. Set **Span** to `0.975`.

4. Click **Apply**.

The names of the objects in the `Geometry` branch of the object selector are shown in black non-italic text, indicating that the `Geometry` objects are all defined. This completes the geometry definition.

2.4. Creating the Topology and Mesh

The `Topology Set` object is initially suspended in order to save computational effort while defining the geometry. When you unsuspend the `Topology Set` object, the remaining computations are performed, resulting in a 3D mesh.

- Right-click `Topology Set` and turn off **Suspend Object Updates**.

  The `Topology Set` object name in the object selector changes to black non-italic text, indicating that this object is generated. The topology appears on the hub and shroud as a structure of thick lines. Thinner lines show where individual mesh elements are located.

  The 3D mesh is also generated. The number of nodes and elements are displayed at the bottom left.

Change the view to clearly show the topology on the hub:

1. Click *Hide all geometry objects* .

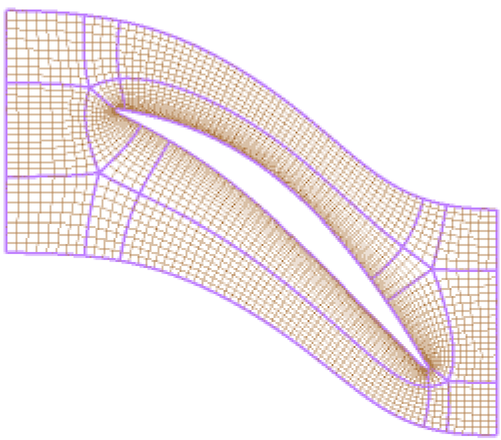
2. Turn on the visibility of `Layers` > `Hub` to show the topology on the hub.

3. Click *Hide all mesh objects* .

4. Right-click a blank area in the viewer, and click **Predefined Camera** > **View From +X** from the shortcut menu.

The heavy lines in Figure 2.1: ATM Topology and 2D Mesh on the Hub (p. 14) indicate the topology lines; the thinner lines show the 2D mesh for the hub.

Figure 2.1: ATM Topology and 2D Mesh on the Hub



2.5. Reviewing the Mesh Data Settings

The `Mesh Data` settings control the number and distribution of mesh elements.

1. Open `Mesh Data`.

2. Note that **Method** is set to `Global Size Factor` and **Size Factor** is set to `1`.

   In the status bar in the bottom-left corner of Ansys TurboGrid, you can see that the number of mesh nodes is on the order of 200000.

2.6. Reviewing the Mesh Quality on the Hub and Shroud Tip Layers

Layers are constant-span surfaces. You can display the topology on a layer. You have already seen the hub layer in Figure 2.1: ATM Topology and 2D Mesh on the Hub (p. 14). At this point, there are two layers: `Layers` > `Hub`, and `Layers` > `Shroud Tip`.

If the topology were grossly skewed or distorted on the hub or shroud tip layer, the `Layers` object would be shown with red text in the object selector.

Now the topology is defined and the mesh quality is acceptable on all layers.

The mesh is generated automatically after the `Layers` object is processed. Later in this tutorial, you will check 3D mesh measures and inspect the mesh visually.

2.7. Looking at Mesh Data Values

The `Mesh Data` editor tabs display information about the mesh. In the following steps, you will examine the number and distribution of elements from hub to shroud tip and from shroud tip to shroud.

1. Open `Mesh Data`.

2. Click the **Passage** tab.

   Look in the **Spanwise Blade Distribution Parameters** frame. **Method** is set to `Proportional` with a factor of `1.0`. The other boxes in the frame are disabled, but show the current value for each option that Ansys TurboGrid has calculated.

   You can see that **# of Elements** is `25`.

3. Click the **Shroud Tip** tab.

   Look in the **Shroud Tip Distribution Parameters** frame. **Method** is set to `Match Expansion at Blade Tip`. You can see that the number of elements from shroud tip to shroud is 16.

2.8. Analyzing the Mesh Quality

3D mesh measures are available. These are analogous to the 2D mesh measures that are calculated on layers. As for the 2D mesh measures, the 3D mesh measures have quality criteria set in the `Mesh Analysis` > `Mesh Limits` object.

Mesh measures of some mesh elements may fall outside the criteria. When any mesh measure fails to meet the criteria, `Mesh Analysis (Error)` > `Mesh Statistics (Error)` will appear in red text in the object selector. You can open `Mesh Analysis (Error)` to display the **Mesh Statistics** dialog box. In the **Mesh Statistics** dialog box, you can select one of the items in red and click **Display** to see the locations in the mesh where the statistics fail to meet the corresponding criterion.

Not all of the mesh measures carry the same importance. For example, it is necessary to have a mesh with no negative volumes. Generally, poor angles should also be fixed, but the `Maximum Element Volume Ratio` and `Maximum Edge Length Ratio` values should be judged based on your requirements.

Check the 3D mesh statistics:

1. For a visual frame of reference, ensure that `Layers` > `Hub` and `Layers` > `Shroud Tip` are visible.

2. Open `Mesh Analysis` or `Mesh Analysis` > `Mesh Statistics`.

   The **Mesh Statistics** dialog box appears, and shows that the mesh statistics are acceptable based on the current quality criteria.

3. Close the **Mesh Statistics** dialog box.

2.9. Visualizing the Hub-to-Shroud Element Distribution

To demonstrate the use of the `3D Mesh` visualization objects, look at the mesh distribution from hub to shroud as follows:

1. Click *Unhide geometry objects* .

2. Turn off the visibility of the following objects:

   - `Geometry` > `Blade Set` > `Blade 1`
   - `Layers` > `Hub`
   - `Layers` > `Shroud Tip`

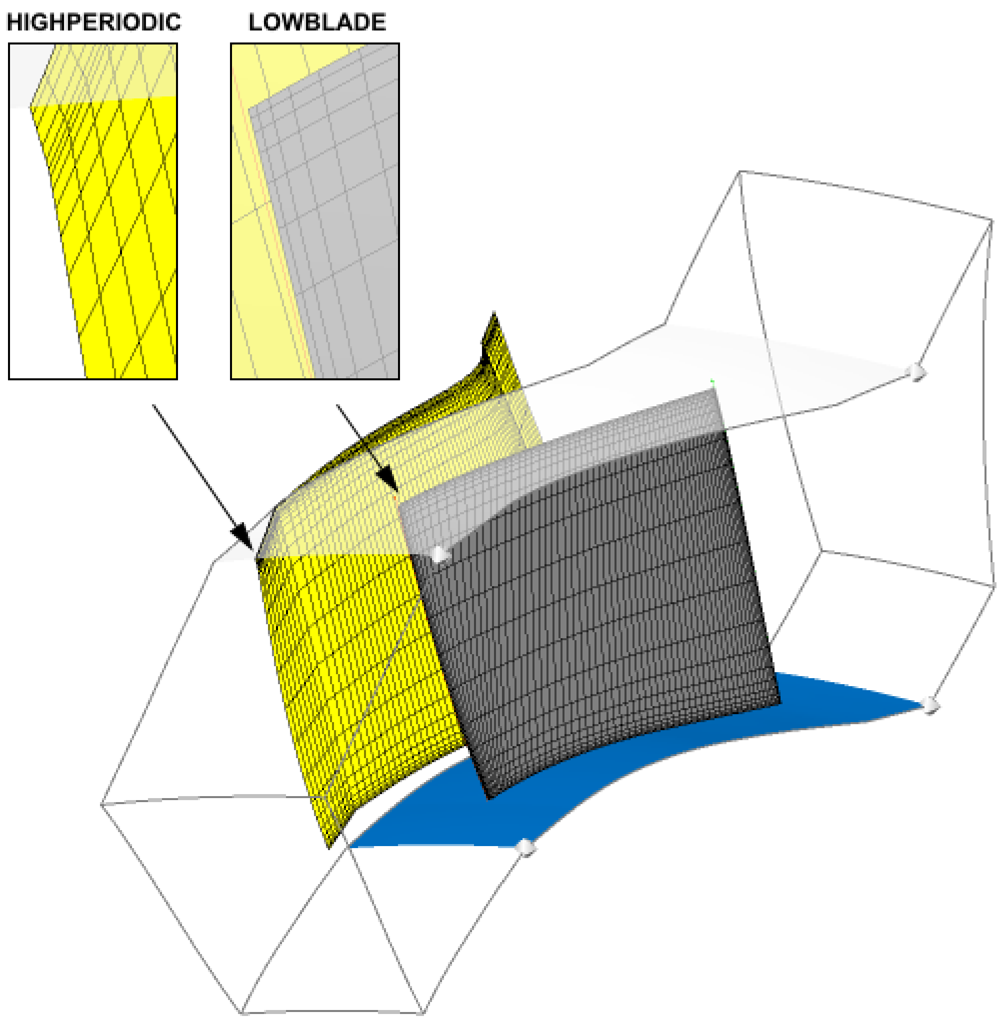
3. Turn on the visibility of the following objects under `3D Mesh`:

   - `HIGHPERIODIC`.
   - `LOWBLADE GEO HIGH`
   - `LOWBLADE GEO LOW`

4. Observe the element distribution from hub to shroud tip and from shroud tip to shroud.

   See Figure 2.2: Hub-to-Shroud Element Distribution (p. 17).

Figure 2.2: Hub-to-Shroud Element Distribution



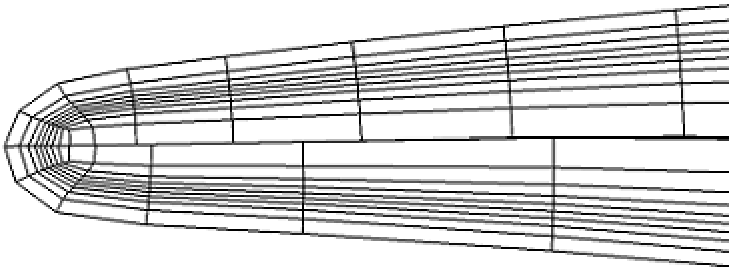
2.10. Observing the Shroud Tip Mesh

A mesh interface exists in the shroud tip gap. In order to see this interface:

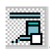
1. Turn on the visibility of `3D Mesh` > `SHROUD TIP`.
2. Click *Hide all geometry objects* .
3. Zoom in to view the mesh on the shroud tip.

   Figure 2.3: Surface Group: Tip Near Trailing Edge (p. 18) shows this mesh at the trailing edge of the blade. Note how the nodes do not line up along the middle of the blade, due to the default use of a general grid (GGI) interface along the shroud tip of the blade.

Figure 2.3: Surface Group: Tip Near Trailing Edge



4. Turn off the visibility of `3D Mesh` > `SHROUD TIP`.

5. Click *Unhide geometry objects* .

2.11. Examining the Mesh Qualitatively

You will now examine the mesh qualitatively using a turbo surface. Change the `Show Mesh` turbo surface so that it appears on the hub, and color it to show the variation in `Edge Length Ratio` (a variable that was computed at the time the mesh was generated):

1. Turn on the visibility of `3D Mesh` > `Show Mesh`.

2. Open `3D Mesh` > `Show Mesh`.

3. Leave **Variable** set to K.

   K is equal to the node number in the spanwise direction, ranging from 1 at the hub to a positive integer value at the shroud.

4. Set **Value** to `1`.

   This will cause the turbo surface to appear on the hub.

5. Click the **Color** tab.

6. Set **Mode** to `Variable`.

7. Set **Variable** to `Edge Length Ratio`.

8. Set **Range** to `Local`.

   This will cause the range of colors in the color map to be distributed over the range of values found on the turbo surface, rather that over the global range or a user-defined range.

9. Click the **Render** tab.

10. Ensure that **Draw Faces** is selected.

11. Click **Apply**.

12. To avoid visual conflicts between the turbo surface and the hub, which are coincident, turn off the visibility of `Geometry` > `Hub`.

Note that you can edit the rendering properties of the hub to achieve a similar result. The advantage of using a turbo surface is that you can redefine its location. For example, you could change the value of K in the current turbo surface to see `Edge Length Ratio` on a different nodal plane.

Note:

You can create new turbo surfaces. To begin the process of creating a new turbo surface, click **Insert** > **User Defined** > **Turbo Surface**.

Note:

To show distinct color bands, you could make a contour plot object that applies to an existing locator (geometric surface, turbo surface, or other graphic objects that involve surfaces). To begin the process of creating a contour plot, ensure that you have a suitable locator already defined, then click **Insert** > **User Defined** > **Contour**.

Tip:

For objects that are colored by a variable, it is best to view them with lighting turned off so that the colors are not altered according to the angle of view. The lighting is controlled by a setting on the **Render** tab.

2.12. Creating a Legend

In the previous section, you modified a turbo surface by coloring it according to `Edge Length Ratio`. To reveal the color map used to match values of `Edge Length Ratio` with particular colors, create a legend for the turbo surface:

1. Click **Insert** > **User Defined** > **Legend**.
2. Click **OK** to accept the default name.
3. Set **Plot** to `TURBO SURFACE:Show Mesh`.
4. Set **Title Mode** to `Variable and Location`.
5. Click **Apply**.

   A legend appears in the viewer, showing the correspondence between values of `Edge Length Ratio` and colors for the `Show Mesh` object.

You may want to modify `3D Mesh` > `Show Mesh` to plot it on different locations, or to color it by different variables. The legend will be updated automatically whenever you make changes to the turbo surface.

2.13. Saving the Mesh

Save the mesh:

1. Click **File** > **Save Mesh As**.
2. Ensure that **Files of type** is set to `Ansys CFX Mesh Files`.
3. Set **Export Units** to `cm`.
4. Set **File name** to `rotor37.gtm`.
5. Ensure that your working directory is set correctly.
6. Click **Save**.

2.14. Saving the State (Optional)

If you want to revisit this mesh at a later date, save the state:

1. Click **File** > **Save State As**.
2. Enter an appropriate state filename.
3. Click **Save**.

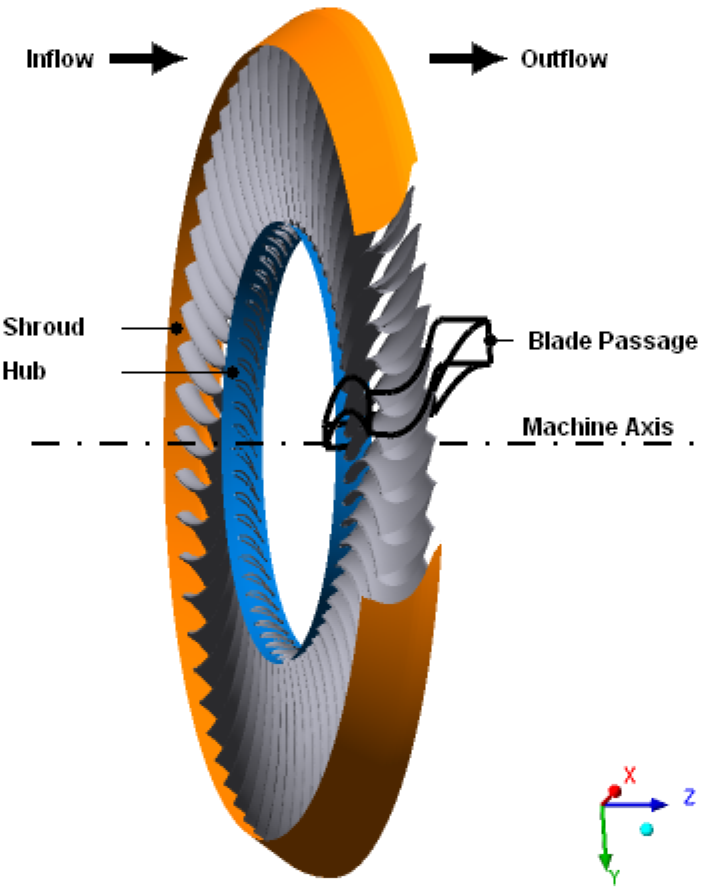
Chapter 3: Steam Stator

This tutorial includes:

This tutorial teaches you how to:

- Import hub, shroud, and blade geometry from individual curve files.
- Change the method of constructing the hub and shroud curve types.
- Make colored surfaces to show variations in mesh measures (such as `Minimum Face Angle`).

As you work through this tutorial, you will create a mesh for a blade passage of a steam stator. A typical blade passage is shown by the black outline in the figure below.



The stator contains 60 blades distributed about the Z axis. A clearance gap exists between the blades and the shroud, with a width of 2% of the total span. Within the blade passage, the maximum diameter of the shroud is approximately 97.5 cm.

If this is the first tutorial you are working with, it is important to review Introduction to the Ansys TurboGrid Tutorials (p. 7) before beginning.

3.1. Preparing the Working Directory

1. Create a working directory.

   TurboGrid uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `stator.zip` file here.

3. Unzip `stator.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `BladeGen.inf`
   - `shroud.curve`

- `hub.curve`
- `profile.curve`

4. Set the working directory and start TurboGrid.

   For details, see Setting the Working Directory and Starting Ansys TurboGrid (p. 7).

3.2. Defining the Geometry

In the first tutorial, you loaded a `BladeGen.inf` file in order to specify the machine data (# of blade sets, rotation axis, and units) and curve files. In this tutorial, you will enter such data manually using the **Load Profile Points** command.

3.2.1. Loading the Curves

Load the curve files for the steam stator as follows:

1. Click **File** > **Load Profile Points**.

   The geometry browser (the main object editor of the **Geometry** workspace) appears. TurboGrid fills in the names of the curve files based on the files that are present in the working directory; The first `.crv` or `.curve` file found that has a name containing "hub", "shroud", or "blade"/"profile" is selected as the hub, shroud, or blade file, respectively.

2. Ensure that, under **Point Data Definition** > **TurboGrid Curve Files**, **Hub** is set to `./hub.curve`, **Shroud** is set to `./shroud.curve`, and **Blade** is set to `./profile.curve`.

3. Set **Point Data Definition** > **Coordinates and Units** > **Coordinates** to `Cartesian` and **Length Units** to `cm`.

   These units are used to interpret the data in the curve files.

4. Set **Geometry Setup** > **Rotation** > **Method** to `Principal Axis` and **Axis** to `Z`.

5. Set **Geometry Setup** > **# of Bladesets** to `60`.

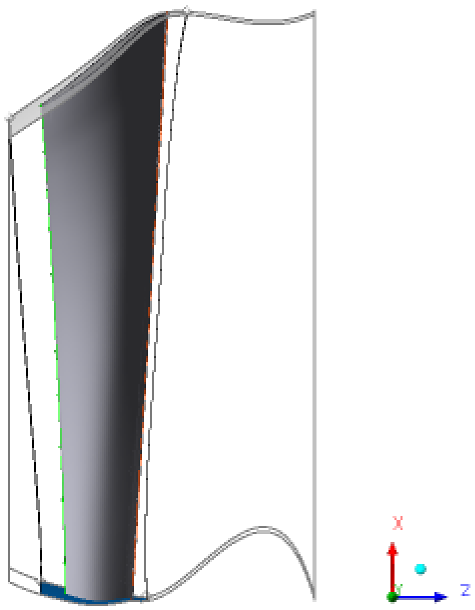
6. Click **Apply** to save the settings.

7. Return to the **Mesh** tab to view the geometry.

The progress bar at the bottom right of the screen shows the geometry generation progress. After the geometry has been generated, you can see the hub, shroud, and blade for one passage. Along the blade, you can see the leading and trailing edge curves (green and red lines, respectively). Near the blade, you can see the inlet and outlet markers (white octahedrons).

- Rotate the geometry into the position shown in Figure 3.1: Incorrect Hub and Shroud Representations (p. 24).

Figure 3.1: Incorrect Hub and Shroud Representations



As shown in Figure 3.1: Incorrect Hub and Shroud Representations (p. 24), the hub and shroud are greatly distorted. This is the result of using spline curves to construct the hub and shroud based on relatively few data points. This problem will be corrected in the next section.

3.2.2. Setting the Curve Type

Set the method of constructing the hub and shroud as follows:

1. Open `Geometry` > `Hub`.
2. Set **Geometric Representation** > **Curve Type** to `Piece-wise linear`.
3. Click **Apply**.
4. Set `Shroud` in the same way.

3.2.3. Defining the Shroud Tip

To complete the geometry, create a small gap between the blade and the shroud. The blade should be shortened to 98% of its original span because the gap width is 2% of the total span, as specified in the problem description.

1. Open `Geometry` > `Blade Set` > `Shroud Tip`.
2. Set **Tip Option** to `Constant Span`.
3. Set **Span** to `0.98`.
4. Click **Apply**.

This completes the geometry definition.

3.3. Creating the Topology and Mesh

1. Click *Hide all geometry objects*  to turn off the visibility of the geometry.
2. Right-click `Topology Set` and turn off **Suspend Object Updates**.
3. Turn on the visibility of `Layers` > `Hub` to show the topology on the hub.
4. Turn on the visibility of `Layers` > `Shroud Tip` to show the topology on the shroud tip.

The topology and 3D mesh are generated.

Note:

It may be useful to keep the same topology when studying a range of blade geometries, or the same blade on different computers. To keep the same topology, use the **Manual (Advanced)** setting for the topology. This setting is available after clicking **Edit** > **Options** and selecting **Enable Advanced Features**.

3.4. Reviewing the Mesh Data Settings

TurboGrid 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 global size factor to determine the final mesh size. The overall mesh size is controlled using the **Method** setting in the **Mesh Data** object editor on the **Mesh Size** tab. Setting the **Method** to `Target Passage Mesh Size` enables you to specify a **Node Count**. Using this method specifies an approximate mesh size (in nodes) and lets TurboGrid compute the mesh dimensions automatically. Setting the **Method** to `Global Size Factor` enables you to specify a **Size Factor**. Increasing this factor will increase the overall mesh size, and decreasing it will decrease the overall mesh size. The change is nonlinear.

The **Boundary Layer Refinement Control** settings affect the mesh in the O-Grid region around the blade:

- Note that, when **Boundary Layer Refinement Control** > **Method** is set to `Proportional to Mesh Size`, the number of elements across the boundary layer is calculated as Base Count * Global Size Factor * (Factor Base + Factor Ratio * Global Size Factor). The default values of Factor Base and Factor Ratio are 3 and 0 respectively.
- The **Target Maximum Expansion Rate** setting affects the expansion rates that are used just outside the blade profile.
- The **Near Wall Element Size Specification** settings control the method by which the near-wall node spacing is specified on the **Passage**, **Hub Tip**, and **Shroud Tip** tabs. The near-wall node spacing is the distance between a wall (for example, hub, shroud, or blade) and the first layer of nodes from the wall. The **Method** setting has these options:
  - Y Plus — The y+ method sets the near-wall spacing to a target value, y+, in relation to a set Reynolds number.

- Absolute — The `Absolute` method enables you to set the near-wall spacing directly on the **Passage**, **Hub Tip**, and **Shroud Tip** tabs.

- The **Inlet Domain** and **Outlet Domain** check boxes enable you to generate the inlet and outlet domains as part of the mesh. Settings that affect these grid regions are found on the **Inlet/Outlet** tab.

Selecting the **Lock mesh size** check box forces the total number of nodes and elements to remain constant.

On the **Shroud Tip** tab, you can use the **Blade Tip** > **Override Target Maximum Expansion Rate** setting to override the **Target Maximum Expansion Rate** value set on the **Mesh Size** tab, to govern the expansion rate of elements that cross the tip mesh.

3.5. Reviewing the Mesh Quality on the Hub and Shroud Layers

If the topology were grossly skewed or distorted on the hub or shroud layer, the `Layers` object would be shown with red text in the object selector. Since the `Layers` object is shown in black text, the mesh contains no regions with high skew on the hub or shroud.

By default, TurboGrid automatically generates the recommended number of layers before the mesh is generated. This default behavior can be disabled by editing the `Layers` object so that **Insertion Mode** is not set to `Automatic - Adaptive`.

A turbo surface of constant "K" (a nodal coordinate) appears. This surface is listed in the object selector as `3D Mesh` > `Show Mesh`. You can change the location and coloring of this surface to explore the mesh.

3.6. Analyzing the Mesh

Check the 3D mesh statistics:

1. Open `Mesh Analysis`.

   The mesh statistics are acceptable based on the current quality criteria.

2. Close the **Mesh Statistics** dialog box.

3.6.1. Examining the Mesh Qualitatively

The predefined surfaces found under the `3D Mesh` object in the object selector are useful for showing variations in the mesh statistics.

In the following section, you will color `3D Mesh` > `Show Mesh` by `Minimum Face Angle`. You will then create a legend for that object.

3.6.1.1. Editing a Turbo Surface

Turbo Surfaces can be created by selecting **Insert** > **User Defined** > **Turbo Surface**. In this case, you will simply edit the predefined turbo surface.

1. Open `3D Mesh` > `Show Mesh`.

2. Leave **Variable**, and **Value** unchanged.

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

4. Set **Variable** to `Minimum Face Angle`.

5. Set **Range** to `Local`.

   This will cause the range of colors in the color map to be distributed over the range of values found on the turbo surface, rather that over the global range or a user-defined range.

6. Click the **Render** tab.

7. Ensure that **Draw Faces** is selected.

8. Click **Apply** to apply the changes to the turbo surface.

3.6.1.2. Creating a Legend

To illustrate the scale of the `Minimum Face Angle` variable, create a legend for the turbo surface:

1. Click **Insert** > **User Defined** > **Legend**.

2. Click **OK** to accept the default name.

3. Set **Plot** to `TURBO SURFACE:Show Mesh`.

4. Set **Title Mode** to `Variable and Location`.

5. Click **Apply** to create the legend.

3.7. Saving the Mesh

Save the mesh:

1. Click **File** > **Save Mesh As**.

2. Ensure that **Files of type** is set to `Ansys CFX Mesh Files`.

3. Set **Export Units** to `cm`.

4. Set **File name** to `steam_stator.gtm`.

5. Ensure that your working directory is set correctly.

6. Click **Save**.

3.8. Saving the State (Optional)

If you want to revisit this mesh at a later date, save the state:

1. Click **File** > **Save State As**.

2. Enter an appropriate state filename.

3. Click **Save**.

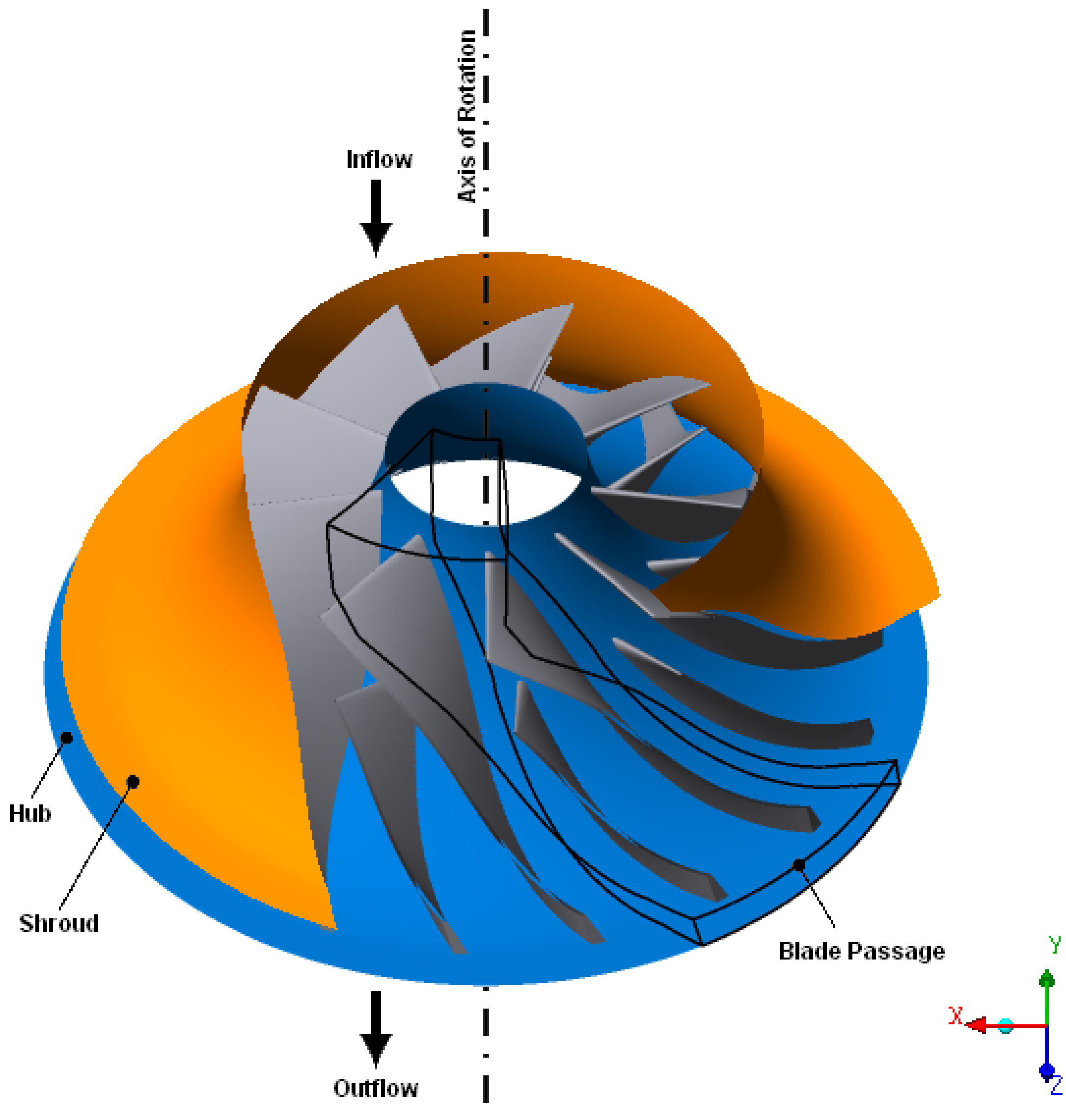
Chapter 4: Radial Compressor

This tutorial includes:

This tutorial teaches you how to:

- Set machine data and load curve files independently.
- Specify a "cut-off or square" edge on a blade.
- Choose an appropriate topology family under **ATM Topology**.
- Change the distribution of mesh elements along a cut-off edge.

As you work through this tutorial, you will create a mesh for a blade passage of a radial compressor blade row. A typical blade passage is shown by the black outline in the figure below.



The blade row contains 9 main blades and 9 splitter blades that revolve about the negative Z axis. The blades have cut-off trailing edges. A clearance gap exists between the blades and the shroud, with a width of 5% of the total span. Within the blade passage, the maximum diameter of the shroud is approximately 125 mm.

If this is the first tutorial you are working with, it is important to review Introduction to the Ansys TurboGrid Tutorials (p. 7) before beginning.

4.1. Preparing the Working Directory

1. Create a working directory.

   Ansys TurboGrid uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `radcomp.zip` file here.
3. Unzip `radcomp.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `BladeGen.inf`
   - `profile.crv`
   - `hub.crv`
   - `shroud.crv`

4. Set the working directory and start Ansys TurboGrid.

   For details, see Setting the Working Directory and Starting Ansys TurboGrid (p. 7).

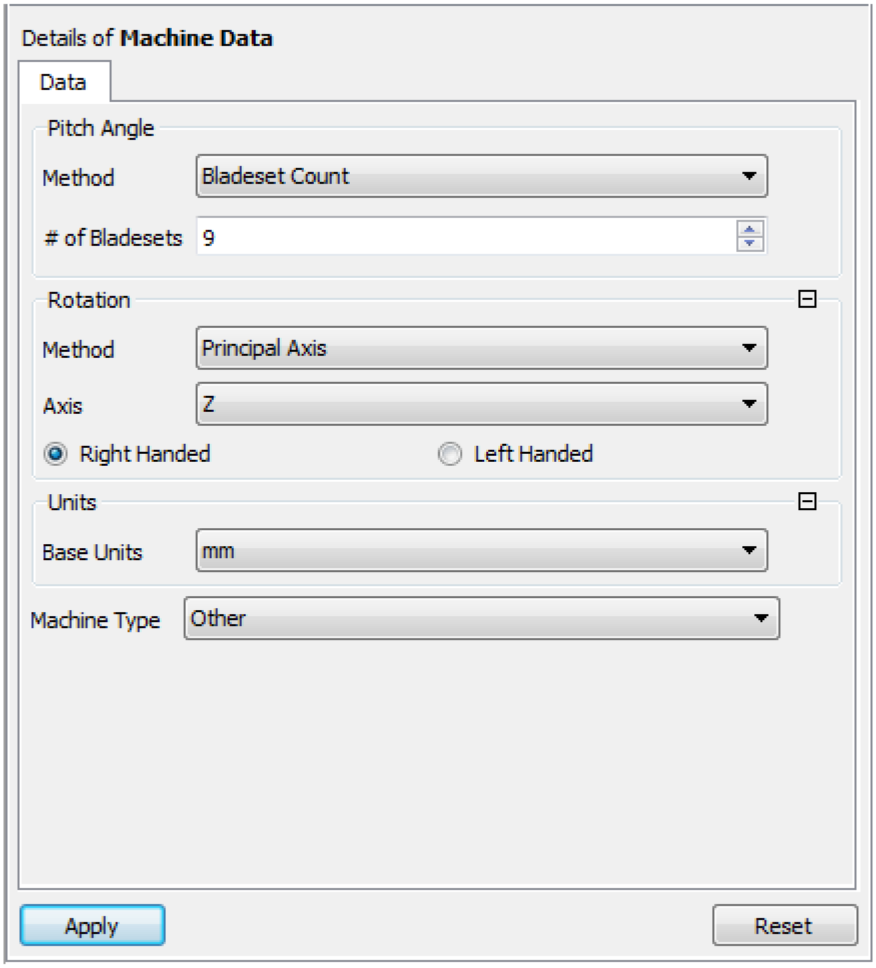
4.2. Defining the Geometry

In the Rotor 37 tutorial, you loaded a `BladeGen.inf` file in order to specify the machine data (# of blade sets, rotation axis, and units) and the hub, shroud, and blade curve files. In the Steam Stator tutorial, you entered the same data using the **Load Profile Points** command. In this tutorial, you will define the machine data and curve files individually, by editing the corresponding geometry objects.

4.2.1. Defining the Machine Data

Set up the `Machine Data` object, which contains basic information about the geometry:

1. In the **Mesh** workspace, open `Geometry` > `Machine Data`.



2. Set **# of Bladesets** to 9.

3. Set **Base Units** to `mm`.

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

4.2.2. Defining the Hub

1. Open `Geometry` > `Hub`.

2. Set **Length Units** to `mm`.

3. Ensure that **File Name** is set to `./hub.crv` from your working directory.

4. Click **Apply**.

4.2.3. Defining the Shroud

1. Open `Geometry` > `Shroud`.

2. Set **Length Units** to `mm`.

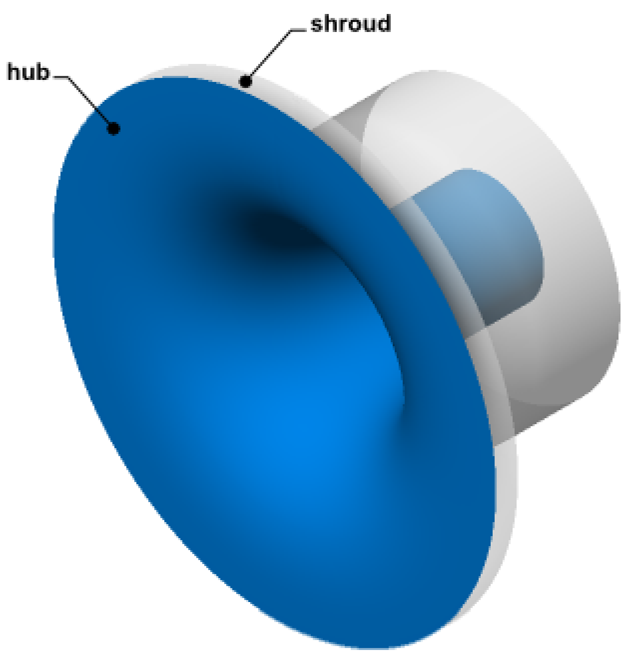
3. Ensure that **File Name** is set to `./shroud.crv` from your working directory.

4. Click **Apply**.

> **Note:**
>
> If you had loaded the `BladeGen.inf` file, the **Curve Type** settings for the `Hub` and `Shroud` objects would have been set to `Piece-wise linear` instead of the default: `Bspline`. Either setting will work for this geometry.

At this point, the entire hub and shroud surfaces are shown. After a blade is defined (in the next step), the hub and shroud will be trimmed to show only one passage.

Figure 4.1: Hub and Shroud of Radial Compressor



4.2.4. Defining the Blade

1. In the **Mesh** workspace, open `Geometry` > `Blade Set` > `Blade 1`.

2. Ensure that **File Name** is set to `./profile.crv`.

3. Set **Length Units** to `mm`.

4. Set **Geometric Representation** > **Method** to `Specify`.

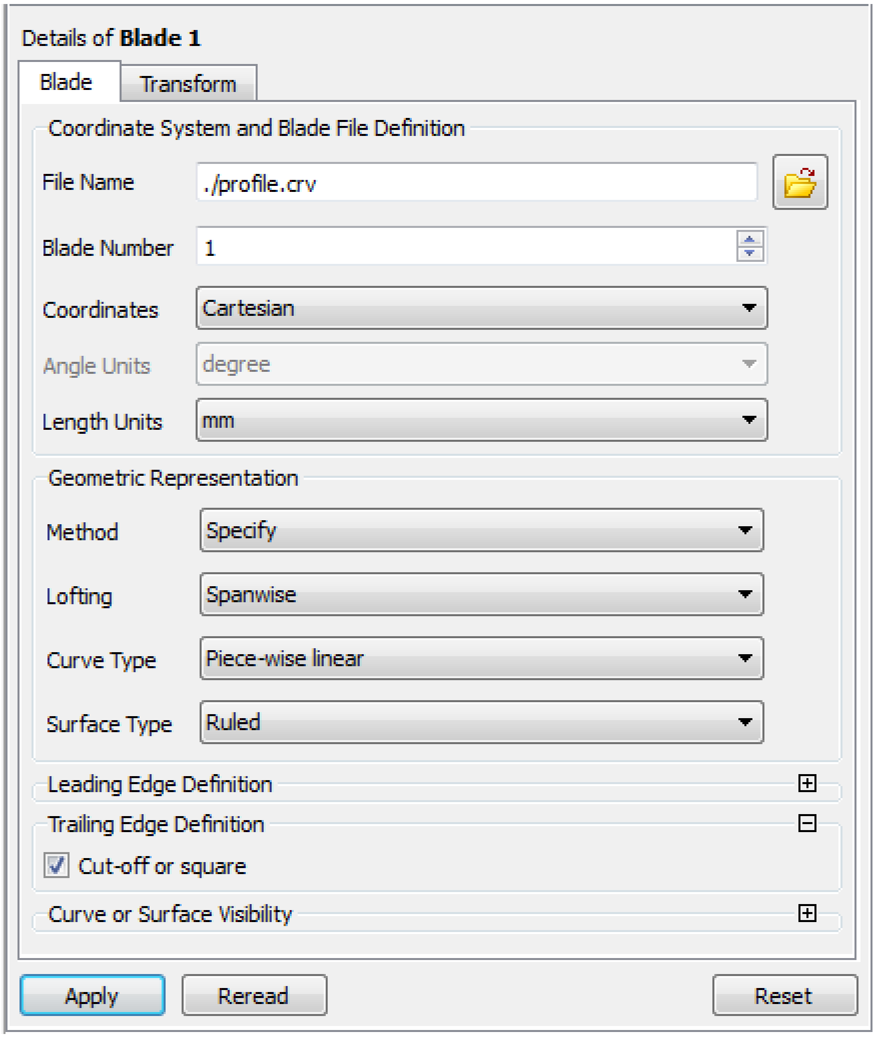
5. Set **Geometric Representation** > **Lofting** to `Spanwise`.

6. Set **Geometric Representation** > **Curve Type** to `Piece-wise linear`.

7. Set **Geometric Representation** > **Surface Type** to `Ruled`.

8. Under **Trailing Edge Definition**, select **Cut-off or square**.

9. Click **Apply**.



The progress bar at the bottom right of the screen shows the geometry generation progress. After the geometry has been generated, you can see the hub, shroud, and blade for one passage. Along the blade, you can see the leading and trailing edge curves (green and red lines, respectively).

4.2.5. Defining the Splitter Blade

1. Right-click `Geometry` > `Blade Set` and select **Insert** > **Blade**.

2. Click **OK** to accept the default name.

3. Set **Blade Number** to 2.

4. Click **Apply**.

4.2.6. Defining the Shroud Tip

To complete the geometry, create a small gap between the blade and the shroud. The blade should be shortened to 95% of its original span because the gap width is 5% of the total span, as specified in the problem description.

1. Open `Geometry` > `Blade Set` > `Shroud Tip`.
2. Set **Tip Option** to `Constant Span`.
3. Set **Span** to `0.95`.
4. Click **Apply**.

4.3. Creating the Topology and Mesh

In this tutorial, you will manually choose a set of ATM topology templates, called a *topology family*.

1. Click the `Topology Viewer` tab.
2. Open `Topology Set`.
3. Browse through **ATM Topology** > **Method**.

   ATM Topology > **Method** provides a list of topology families from which you can manually choose. When you mouse over or cursor through the list, the Topology Viewer shows a picture of the highlighted topology family and a description of the type of blade that the family best fits.
4. Set **ATM Topology** > **Method** to `Single Splitter`.
5. Click **Apply** to set the topology.
6. Right-click `Topology Set` and turn off **Suspend Object Updates**.

   The topology and 3D mesh are generated.

4.4. Mesh Data Settings

The `Mesh Data` object indicates that there is a problem with mesh quality.

1. Click the **3D Viewer** tab.
2. Expand the `Mesh Data` object in the object selector.
3. Open one of the `Boundary Layer Control` objects under `Mesh Data`.

Note that the near wall expansion rate is outside the established limit within the boundary layer region. The affected mesh regions are colored red in the viewer.

Add more elements to the boundary layer region as follows:

1. Open the `Mesh Data` object.

2. On the **Mesh Size** tab, ensure that **Boundary Layer Refinement Control** > **Method** is set to `Proportional to Mesh Size`.

3. Set **Boundary Layer Refinement Control** > **Parameters** > **Factor Base** to `4.0`.

4. Click **Apply**.

The `Mesh Data` object no longer indicates that there is a problem with mesh quality.

As an exercise, change the distribution of elements across the cut-off edge as follows:

1. Set **Cutoff Edge Split Factor** > **Trailing** to `0.9`.

2. Click **Apply**.

Note that, for a blade that has one rounded edge and one cut-off edge, the distribution of elements across the blade tip mesh is governed by the distribution across the cut-off edge.

4.5. Analyzing the Mesh

Check the 3D mesh statistics:

1. Open `Mesh Analysis`.

   The mesh statistics are acceptable based on the current quality criteria.

2. Close the **Mesh Statistics** dialog box.

4.6. Saving the Mesh

Save the mesh:

1. Click **File** > **Save Mesh As**.

2. Ensure that **Files of type** is set to `Ansys CFX Mesh Files`.

3. Set **Export Units** to `cm`.

4. Set **File name** to `radial_compressor.gtm`.

5. Ensure that your working directory is set correctly.

6. Click **Save**.

4.7. Saving the State (Optional)

If you want to revisit this mesh at a later date, save the state:

1. Click **File** > **Save State As**.

2. Enter an appropriate state filename.

3. Click **Save**.

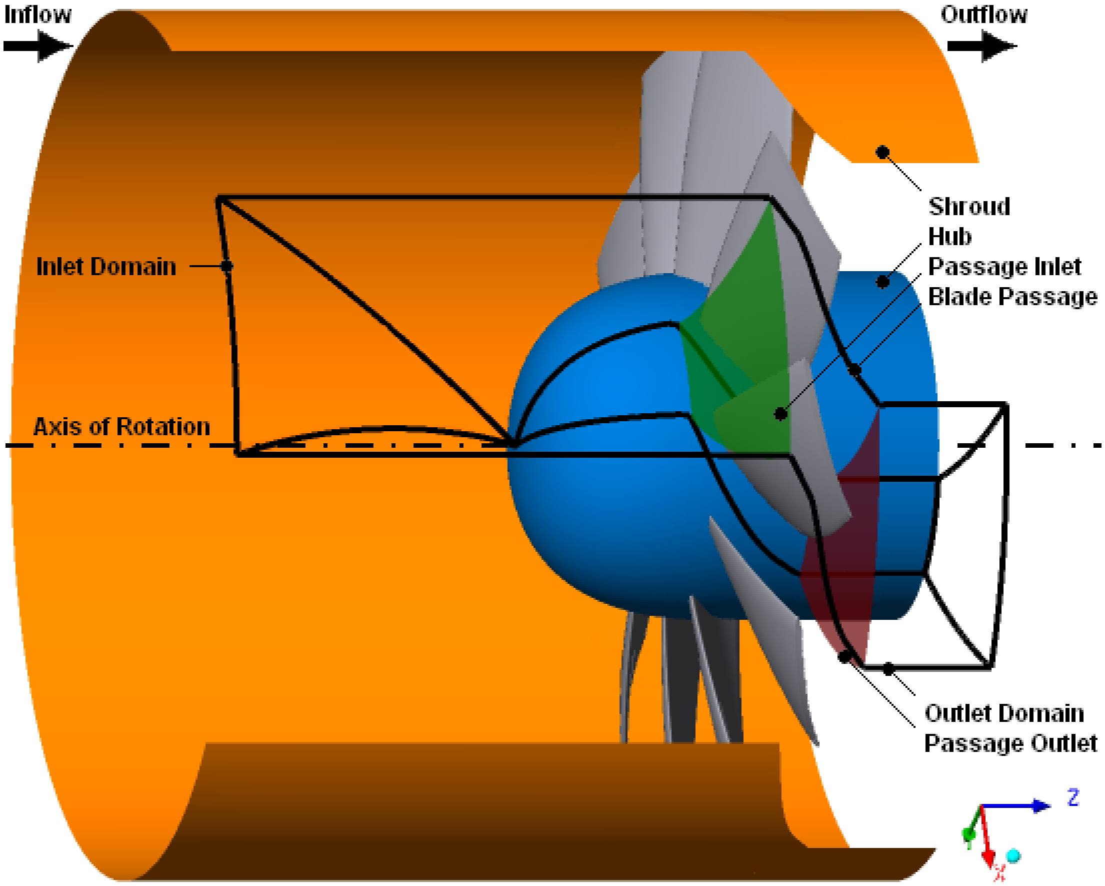
Chapter 5: Axial Fan

This tutorial includes:

This tutorial teaches you how to:

- Switch to a Meridional (A-R) projection in the viewer.
- Change the shape and position of the `Inlet` and `Outlet` geometry objects that bound the blade passage in the streamwise direction.
- Use Local Mesh Refinement
- Extend the mesh by adding inlet and outlet domains.

As you work through this tutorial, you will create a mesh for a blade passage of a fan. A typical blade passage, inlet domain, and outlet domain, are shown by the black outline in the figure below.



The fan contains 10 blades that revolve about the negative Z axis. A clearance gap exists between the blades and the shroud, with a width of 2% of the total span. The shroud diameter is approximately 26.4 cm.

Let the mesh contain an inlet domain and an outlet domain.

If this is the first tutorial you are working with, it is important to review Introduction to the Ansys TurboGrid Tutorials (p. 7) before beginning.

5.1. Preparing the Working Directory

1. Create a working directory.

   Ansys TurboGrid uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `fan.zip` file here.

3. Unzip `fan.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `BladeGen.inf`
   - `shroud.curve`

- `hub.curve`
- `profile.curve`

4. Set the working directory and start Ansys TurboGrid.

   For details, see Setting the Working Directory and Starting Ansys TurboGrid (p. 7).

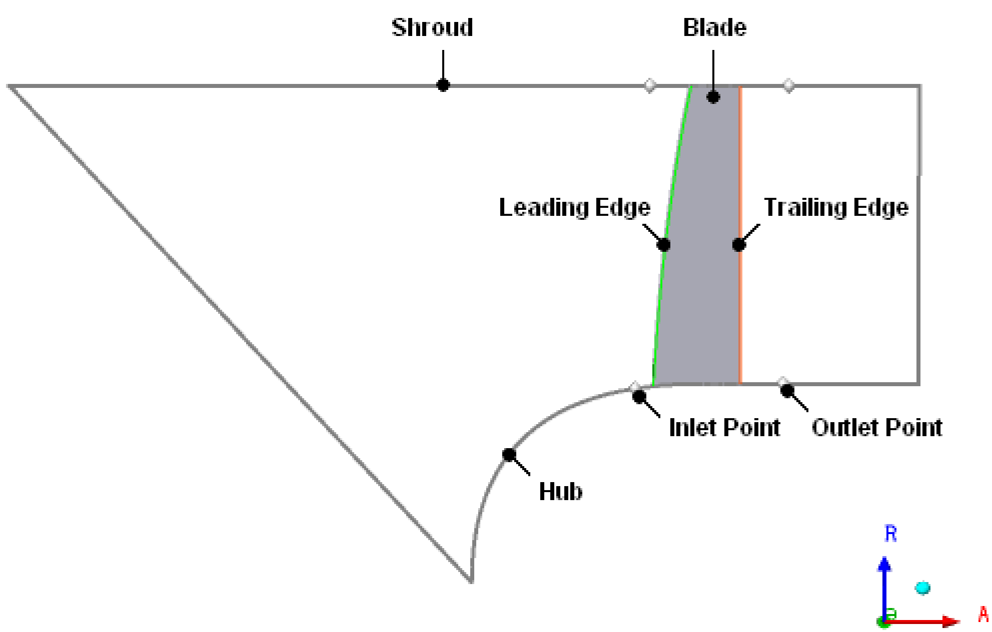
5.2. Defining the Geometry

To obtain the basic geometry, you will load a `BladeGen.inf` file. After inspecting the geometry and improving the shape of the inlet and outlet, you will finish defining the geometry by creating the required gap between the blade and the shroud.

Load the `BladeGen.inf` file, then inspect the geometry by viewing it in axial-radial coordinates:

1. Click **File** > **Load TurboGrid Init File**.
2. Open `BladeGen.inf` from the working directory.
3. In the **Mesh** workspace, right-click a blank area in the viewer, and click **Transformation** > **Meridional (A-R)** from the shortcut menu.

The passage inlet, which appears in the object selector as `Geometry` > `Inlet`, is the upstream end of the blade passage (but not necessarily the upstream end of the mesh, since, as you will see in this tutorial, you can add an inlet domain upstream of the passage inlet). The passage inlet is generated by revolving a curve, which is defined in an axial-radial plane, about the machine axis. That curve, in turn, is generated according to a set of points, known here as *inlet points*. These points appear as white octahedrons in the viewer. The passage outlet is analogous to the passage inlet, and is downstream of the blade passage.



Notice that, in this case, there are two inlet points and they are located at different distances from the blade. In order to obtain a high-quality mesh topology for the blade passage, the inlet points should be repositioned.

Reposition the inlet and outlet points as follows, and observe the movement of the inlet and outlet points in the viewer:

1. Open `Geometry` > `Inlet`.
2. Select **Interface Specification Method** > **Points**.
3. Select `Low Hub Point`, then set **Method** to `Set A` and **Location** to `-0.008`.
4. Click **Apply**.
5. Select `Low Shroud Point`, then set **Method** to `Set A` and **Location** to `0.002`.
6. Click **Apply**.
7. Open `Geometry` > `Outlet`.
8. Select **Interface Specification Method** > **Points**.
9. Select `Low Hub Point`, then set **Method** to `Set A` and **Location** to `0.03`.
10. Click **Apply**.
11. Select `Low Shroud Point`, then set **Method** to `Set A` and **Location** to `0.03`.
12. Click **Apply**.

To complete the geometry, create a small gap between the blade and the shroud. The blade should be shortened to 98% of its original span because the gap width is 2% of the total span, as specified in the problem description.

1. Open `Geometry` > `Blade Set` > `Shroud Tip`.
2. Set **Tip Option** to `Constant Span`.
3. Set **Span** to `0.98`.
4. Click **Apply**.

5.3. Creating the Topology and Mesh

1. Right-click a blank area in the viewer and select **Transformation** > **Cartesian (X-Y-Z)** from the shortcut menu.
2. Click *Hide all geometry objects* .

   This gives you an unobstructed view of the topology, and later the mesh.
3. Right-click `Topology Set` and turn off **Suspend Object Updates**.

4. Turn on the visibility of `Layers` > `Hub` to show the topology on the hub.

5. Turn on the visibility of `Layers` > `Shroud Tip` to show the topology on the shroud tip.

The topology and 3D mesh are generated.

Note:

Mesh quality issues are discussed later in this tutorial.

5.4. Decreasing the Mesh Density

There are several ways to control the mesh size:

- Changing the global size factor.
- Using proportional refinement in the boundary layer.
- Changing the number of mesh elements in the spanwise direction in the passage.
- Changing the edge refinement on a specific edge, including within the boundary layer.

Begin by changing the global size factor and the amount of refinement in the boundary layer:

1. Open `Mesh Data`.

2. On the **Mesh Size** tab, set **Method** to `Global Size Factor`.

3. Set **Size Factor** to `0.9`.

   An overall decrease in mesh size can be useful in reducing the computational resources required for simulation.

4. Ensure that **Boundary Layer Refinement Control** > **Method** is set to `Proportional to Mesh Size`.

5. Set **Factor Base** to `2.6`.

6. Click **Apply**.

   Observe that the number of nodes has been reduced and the element size has increased in the boundary layer mesh. With proportional refinement enabled, the relationship between the height of the first element in the boundary layer and the global size factor should be approximately inversely proportional (that is, an increase in the global size factor will cause a decrease in the element height).

7. Right-click a blank area in the viewer, and select **Predefined Camera** > **Isometric View (X Up)**.

5.5. Observing the Mesh

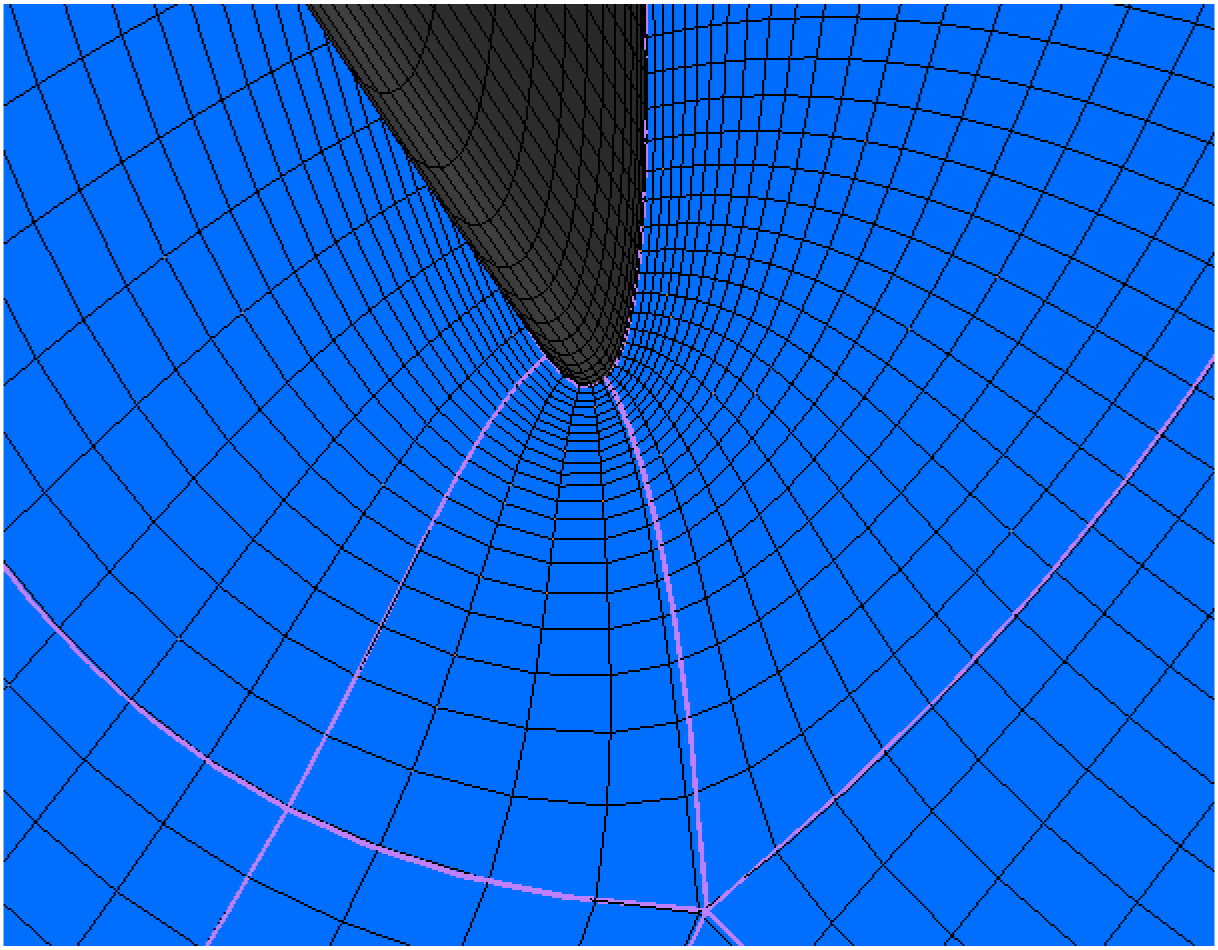
A K-Plane is displayed by default. This shows the 2D mesh on a layer. The plane can be moved in the spanwise direction by holding **Ctrl** + **Shift** and dragging using the left mouse button.

1. Turn on the visibility of the following objects under `3D Mesh`:

   - `HIGHBLADE GEO HIGH`
   - `HIGHBLADE GEO LOW`
   - `HUB`
   - `LOWBLADE GEO HIGH`
   - `LOWBLADE GEO LOW`
   - `SHROUD`
   - `Show Mesh`

2. Note that the mesh element density is higher near the blade and hub, as can be seen in Figure 5.1: Mesh at Blade-Hub Intersection (p. 45).

Figure 5.1: Mesh at Blade-Hub Intersection



The number of mesh elements in the spanwise direction is automatically changed depending on the global size factor and the mesh element size at the boundary layer.

Next, increase the mesh size in the spanwise direction by a factor of 1.5:

3. Open `Mesh Data`.

4. On the **Passage** tab, set **Spanwise Blade Distribution Parameters** > **Method** to `Proportional` and **Factor** to `1.5`.

   Note that the disabled **# of Elements** field indicates the total number of elements in the spanwise direction. This will now increase.

5. Click **Apply**.

   The number of elements has increased.

5.6. Using the Locking Feature

Note:

This section is for information only. Do not use the locking feature in this tutorial.

When you are using Ansys Workbench, Ansys TurboGrid enables you to use the **Lock mesh size** feature. Once activated, the total number of nodes and elements will remain constant. This holds true even if the geometry of the blade is changed. The size of the mesh elements will be readjusted, but the total number will not be changed. The **Lock mesh size** check box is in the `Mesh Data` object editor on the **Mesh Size** tab.

5.7. The Y+ Functionality

Another method of controlling the mesh size at the boundary layer is specifying the y+ height and Reynolds number. This option lets you specify the maximum y+ height for the blade, which is then used to calculate the edge refinement factor. The actual first element offset will not be consistent across the boundary layer, although it should be equal or less than the maximum specified. The edge refinement calculation is only an approximation.

You will enable the option for y+, then set the offset to 15. You will also set the Reynolds number to 500,000.

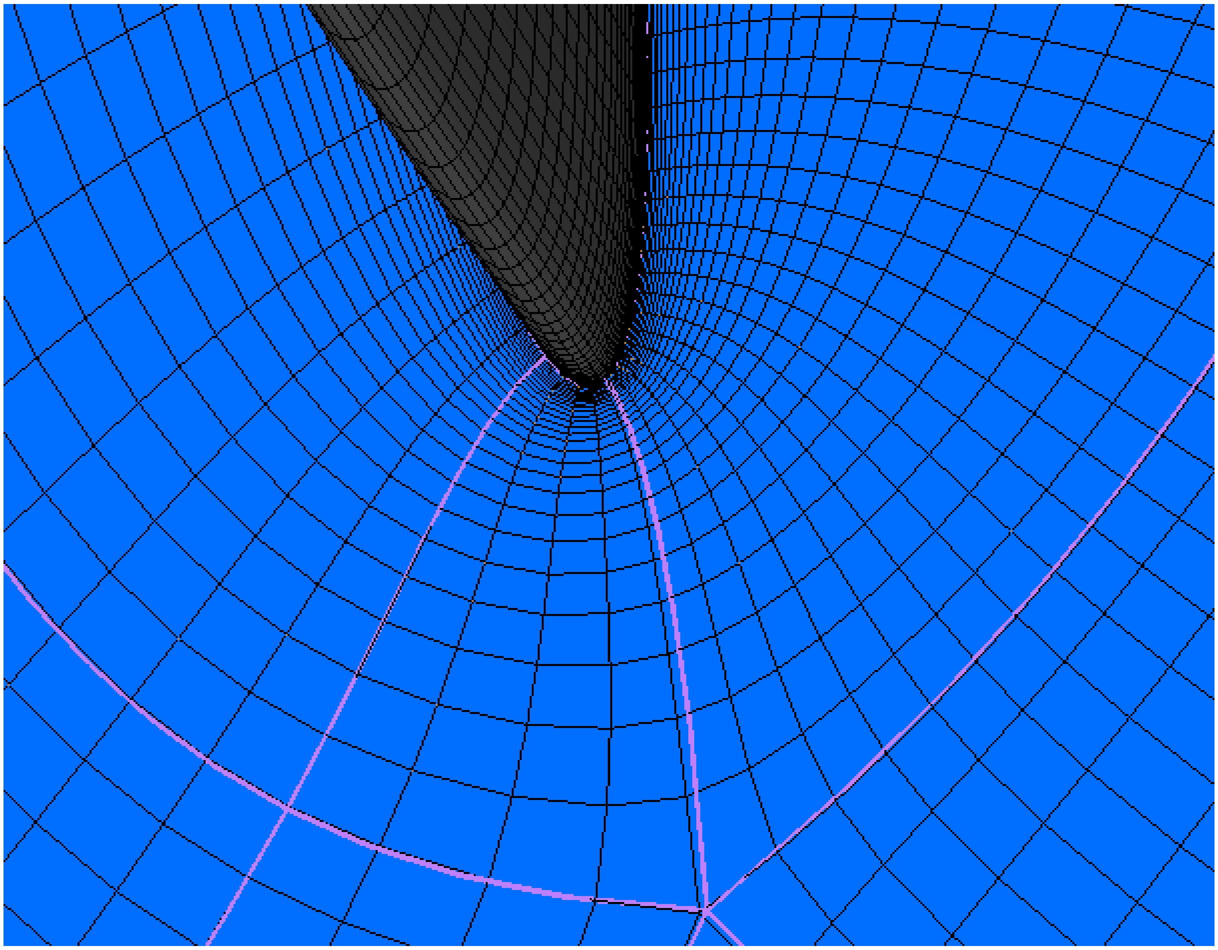
1. Open `Mesh Data`.
2. On the **Mesh Size** tab, set **Boundary Layer Refinement Control** > **Near Wall Element Size Specification** > **Method** to `y+`.
3. Set **Reynolds No.** to `5e5`.
4. Change **Boundary Layer Refinement Control** > **Method** to `First Element Offset`.

   The field for specifying **Offset Y+** is enabled.
5. Set **Parameters** > **Offset Y+** to 5.
6. Click **Apply**.

   You should see an increase in the mesh density at the boundary layer.

   The mesh now has smaller elements near the boundary layer, as shown in in Figure 5.2: Mesh at Blade-Hub Intersection After Y+ Specification (p. 47).

Figure 5.2: Mesh at Blade-Hub Intersection After Y+ Specification



5.8. Using Local Mesh Refinement

Local mesh refinement is especially useful when attempting to manipulate the mesh near a specific boundary without tampering with the surroundings. Once local mesh refinement has been implemented, changing the global size factor will affect the localized area as well. However, the locally refined mesh will remain discernible from its surroundings. You will implement this feature at the shroud boundary, upstream of the blade as indicated in the figure below. The local mesh size will be increased by 100%.

1. Click *Hide all mesh objects* .

   To be able to see which boundary to modify, it is best to hide the currently generated mesh. Ultimately, only the topology will be visible when refinements are made.

2. Right-click a blank area in the viewer and select **Predefined Camera** > **View from +X** from the shortcut menu.

3. Turn off the visibility of `Layers` > Hub.

4. Right-click the edge of the shroud tip layer, marked A in Figure 5.3: Edge to be Refined in Shroud Tip Layer (p. 48), and select **Increase Edge Refinement** > **100%**.

After a few seconds of processing, you should observe the mesh size increasing by a factor of 2 at the edge you selected. Only topologically parallel edges are affected by this change.

Figure 5.3: Edge to be Refined in Shroud Tip Layer

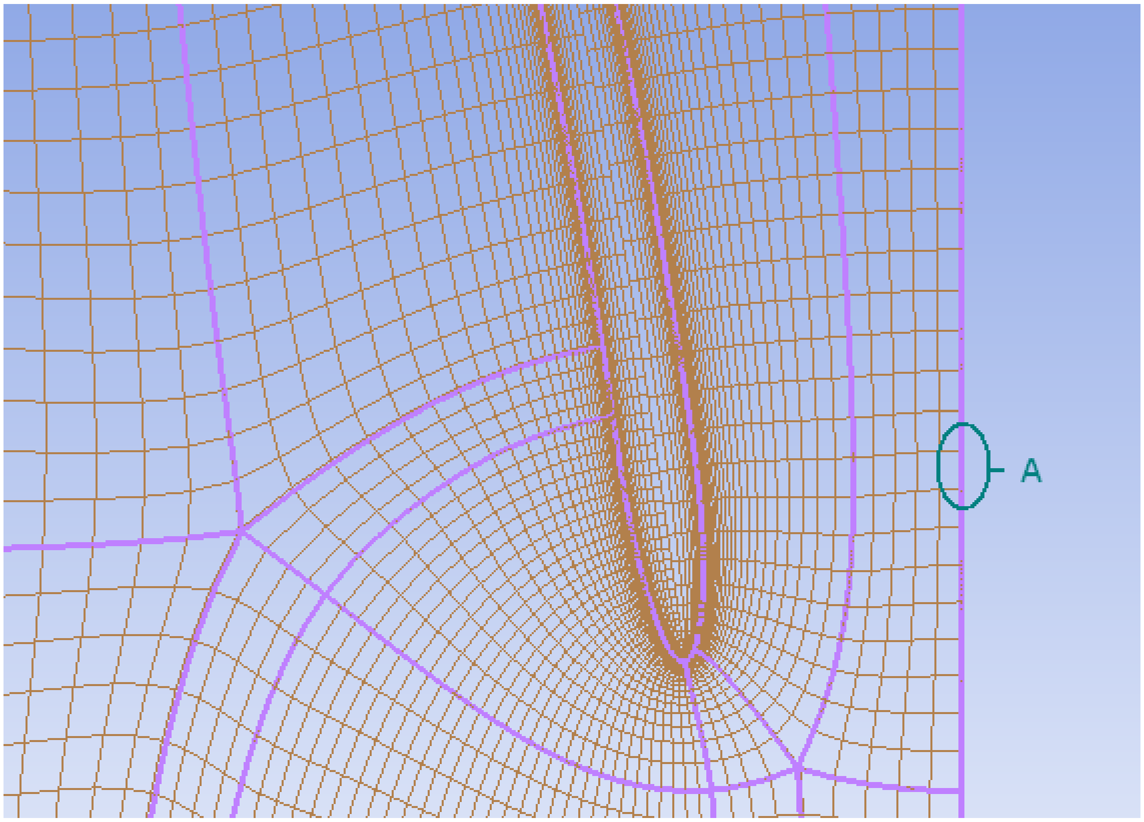
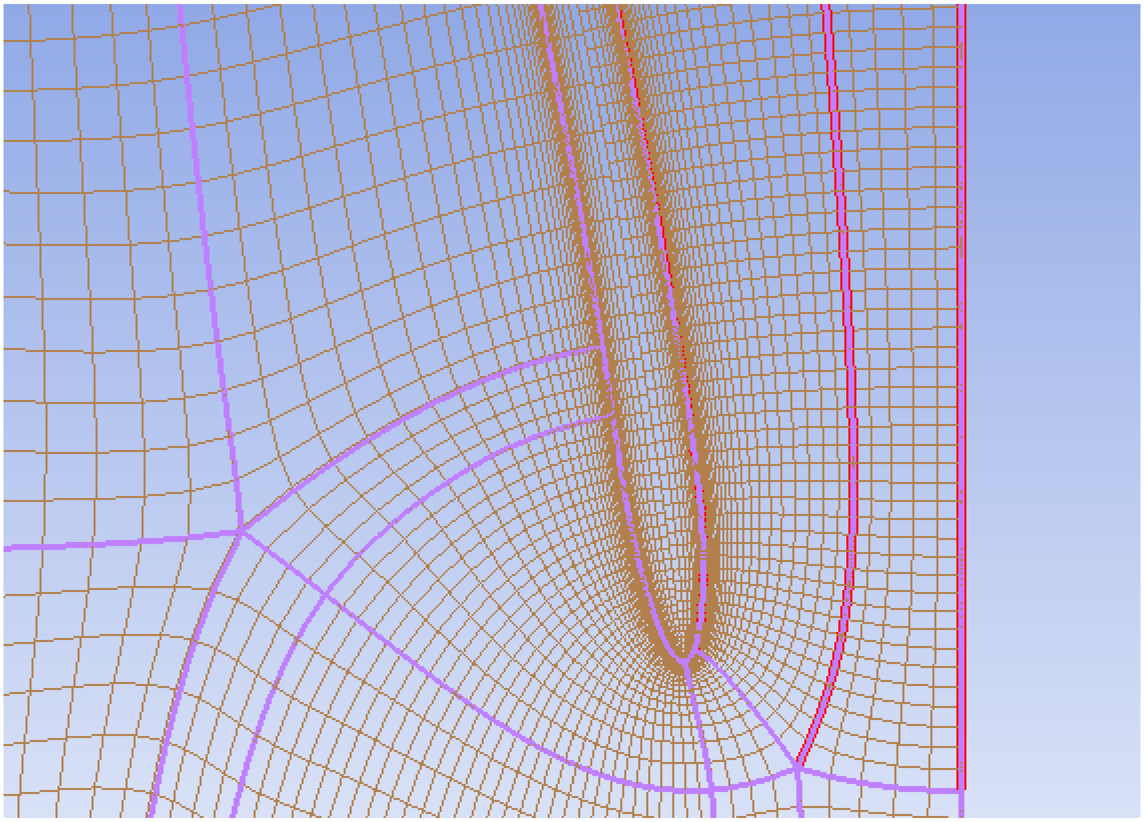


Figure 5.4: After Refinement



5.9. Analyzing the Mesh

Inspect the mesh quality of the 3D mesh:

1. Open `Mesh Analysis`.

   The mesh statistics are acceptable based on the current quality criteria.

2. Close the **Mesh Statistics** dialog box.

5.10. Adding Inlet and Outlet Domains

As specified in the problem description, the mesh should contain an inlet domain and an outlet domain.

1. Open `Mesh Data`.
2. On the **Mesh Size** tab, ensure that **Inlet Domain** and **Outlet Domain** are selected.
3. Click **Apply**.

5.11. Analyzing the New Mesh

1. Open `Mesh Analysis`.

Note that the `Maximum Edge Length Ratio` mesh measure is extremely large. By displaying this mesh measure, you will see that some of the mesh elements that exceed the criterion are at the inlet where the mesh meets the rotation axis. This is to be expected wherever the hub reaches the axis of rotation because at these locations the element edges have zero length.

2. View the mesh on the inlet and outlet (not the passage inlet and outlet, but the inlet and outlet of the entire mesh) by turning on the visibility of the corresponding `3D Mesh` objects.

5.12. Saving the Mesh

Save the mesh:

1. Click **File** > **Save Mesh As**.
2. Ensure that **Files of type** is set to `Ansys CFX Mesh Files`.
3. Set **Export Units** to `cm`.
4. Set **File name** to `fan.gtm`.
5. Ensure that your working directory is set correctly.
6. Click **Save**.

5.13. Saving the State (Optional)

If you want to revisit this mesh at a later date, save the state:

1. Click **File** > **Save State As**.
2. Enter an appropriate state filename.
3. Click **Save**.

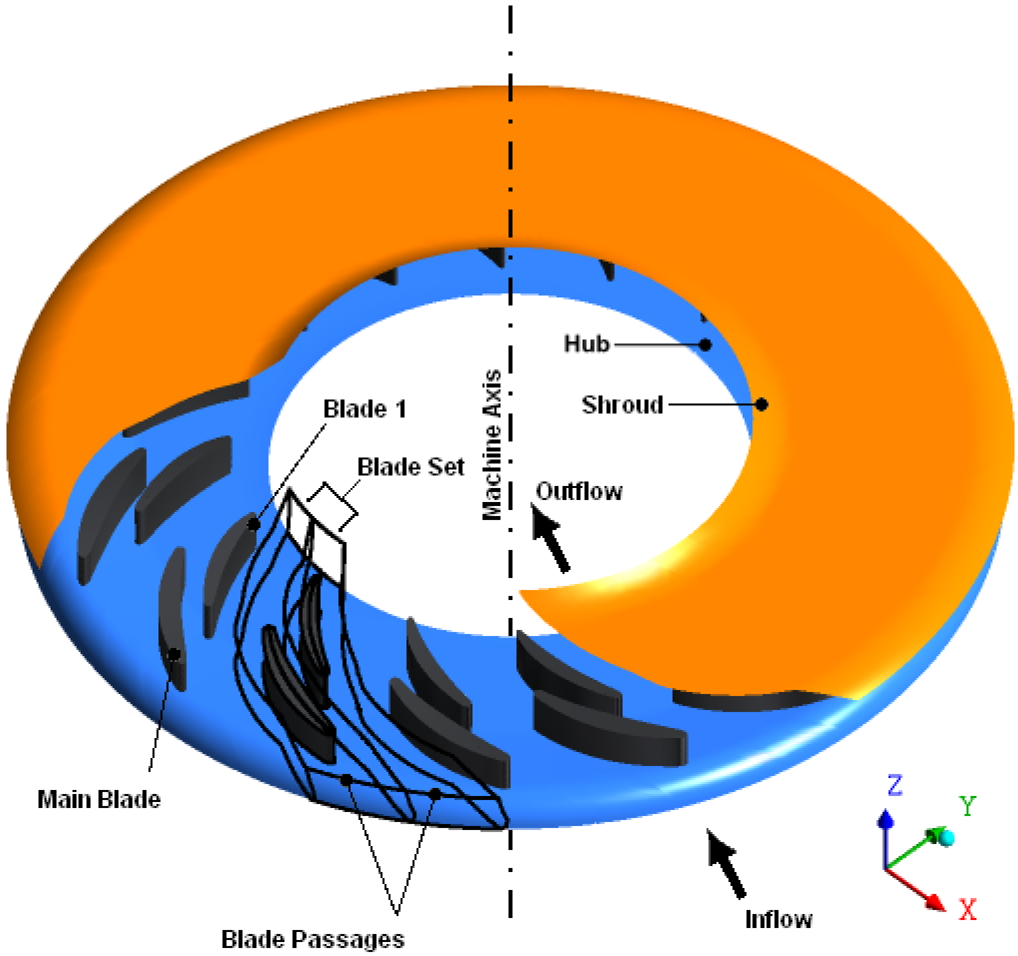
Chapter 6: Tandem Vane

This tutorial includes:

This tutorial teaches you how to:

- Create a mesh involving tandem vanes using a topology template.

As you work through this tutorial, you will create a mesh for a blade set of a radial machine component that has tandem vanes. A typical blade set is shown by the black outline in the figure below.



The component has 16 blade sets, each containing one main blade and one tandem vane. Within the blade passages, the maximum diameter of the shroud is approximately 52.2 cm.

You will begin by loading the geometry from a `BladeGen.inf` file. You will then select a topology template and set the mesh density.

If this is the first tutorial you are working with, it is important to review Introduction to the Ansys TurboGrid Tutorials (p. 7) before beginning.

6.1. Preparing the Working Directory

1. Create a working directory.

   Ansys TurboGrid uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `tandem.zip` file here.

3. Unzip `tandem.zip` to your working directory.

   Ensure that the following tutorial input files are in your working directory:

   - `BladeGen.inf`

- `shroud.curve`
- `hub.curve`
- `profile.curve`

4. Set the working directory and start Ansys TurboGrid.

   For details, see Setting the Working Directory and Starting Ansys TurboGrid (p. 7).

6.2. Defining the Geometry

1. Click **File** > **Load TurboGrid Init File**.
2. Open `BladeGen.inf` from the working directory.

6.3. Creating the Topology and Mesh

While TurboGrid can automatically generate acceptable meshes for basic turbomachinery, you may need to specify topology templates for complex blade configurations, such as tandem vanes.

Select the appropriate template as follows:

1. In the **Mesh** workspace, open `Topology Set`.
2. Set **ATM Topology** > **Method** to `Tandem Vane Aligned High`.
3. Click **Apply**.
4. Right-click **Topology Set** and turn off **Suspend Object Updates**.

The topology and 3D mesh are generated.

The error indicated for `Mesh Data` > `Main Blade Boundary Layer Control` is caused by the near-wall expansion rates. This will be resolved in the next section.

6.4. Setting the Mesh Density

1. Open `Mesh Data`.
2. On the **Mesh Size** tab, set **Method** to `Global Size Factor`.
3. Set **Size Factor** to `1.35`.
4. Click **Apply**.

6.5. Saving the Mesh

Save the mesh:

1. Click **File** > **Save Mesh As**.
2. Ensure that **Files of type** is set to `Ansys CFX Mesh Files`.
3. Set **Export Units** to `cm`.
4. Set **File name** to `tandemvane.gtm`.
5. Ensure that your working directory is set correctly.
6. Click **Save**.

6.6. Saving the State (Optional)

If you want to revisit this mesh at a later date, save the state:

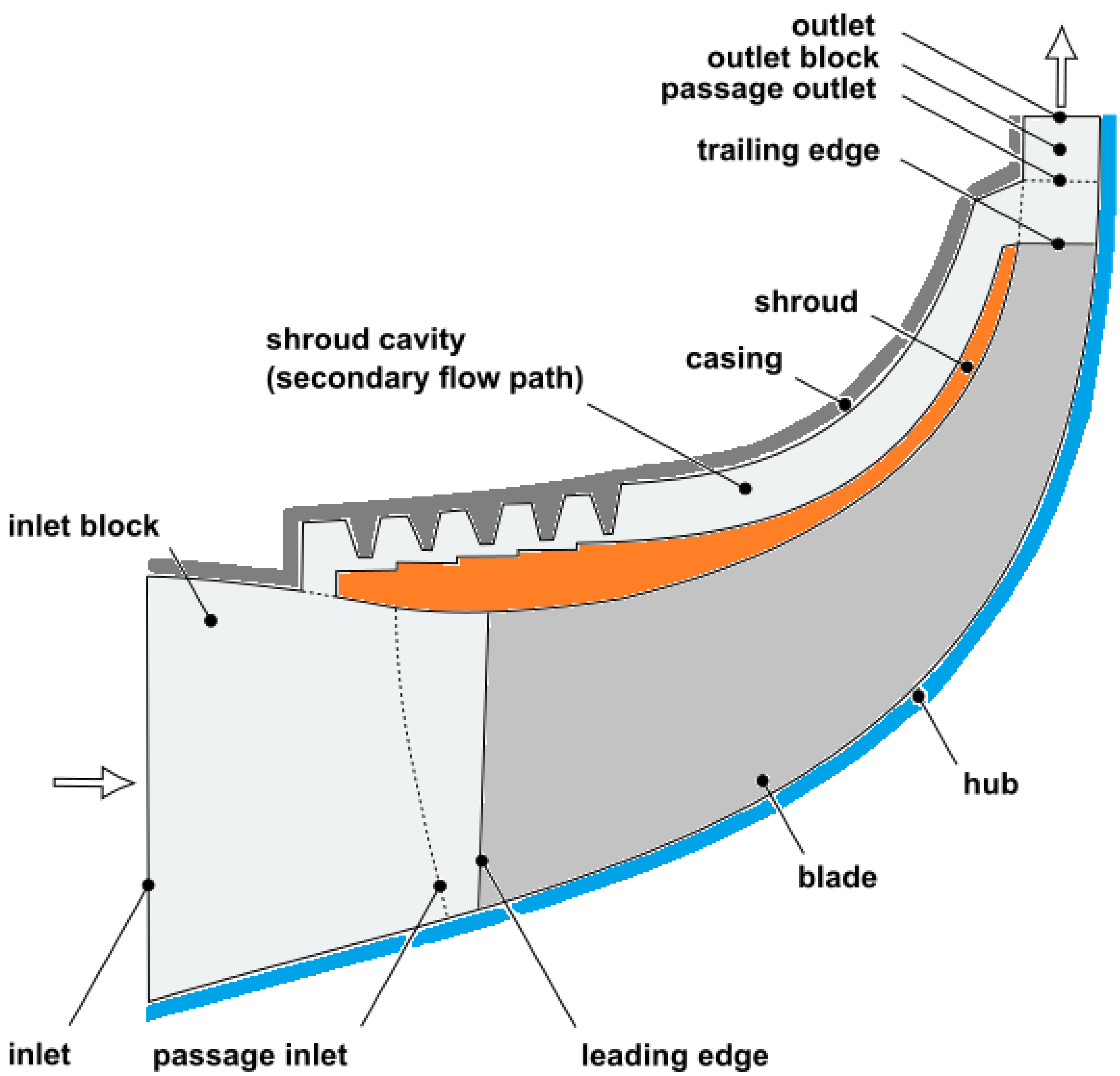
1. Click **File** > **Save State As**.
2. Enter an appropriate state filename.
3. Click **Save**.

Chapter 7: Secondary Flow Path Meshing

This tutorial includes:

This tutorial demonstrates the basic workflow for generating a CFD mesh for a shrouded centrifugal compressor using Ansys BladeEditor and Ansys TurboGrid. Part of the shroud is attached to the blades. An axisymmetric shroud cavity separates the stationary parts of the shroud from the rotating part. As you work through this tutorial, you will use BladeEditor to outline a cross section of the shroud cavity on an axial-radial plane. You will then use TurboGrid to produce a mesh that includes a secondary flow path. The resulting mesh is usable by Ansys CFX in a CFD simulation.

A schematic diagram of the case is shown below.



If this is the first tutorial you are working with, it is important to review Introduction to the Ansys TurboGrid Tutorials (p. 7) before beginning.

7.1. Preparing the Working Directory

1. Create a working directory.

   Ansys TurboGrid uses a working directory as the default location for loading and saving files for a particular session or project.

2. Download the `secondary_path.zip` file here.

3. Unzip `secondary_path.zip` to your working directory.

   Ensure that the following tutorial input file is in your working directory:

   - `shrouded impeller with cavity.wbpz`

Alternatively, if you want to run TurboGrid in stand-alone mode (without having to run DesignModeler/BladeEditor), the following files are needed:

- `secondarypath_turbogrid.tginit`
- `secondarypath_turbogrid.x_b`

If running in stand-alone mode:

1. Launch TurboGrid.
2. Load the `.tginit` file.

   You can expect a message stating "Geometry filepath was not specified". Click **OK** to dismiss it.
3. Continue this tutorial from Associating CAD Objects with Topology in TurboGrid's Geometry Workspace (p. 61).

7.2. Preparing the Geometry in BladeEditor

In the following sections, you will load an Ansys Workbench project file that contains the Ansys BladeEditor geometry, then in Ansys BladeEditor, you will define line bodies and curve groups that will be used by TurboGrid in creating a mesh.

7.2.1. Loading the Provided Project File

1. Start Ansys Workbench.
   - To launch Ansys Workbench on Windows, click the **Start** menu, then select **All Programs > ANSYS 2021 R2 > Workbench 2021 R2**.
   - To launch Ansys Workbench on Linux, open a command line interface, type the path to `runwb2` (for example, `~/ansys_inc/v212/Framework/bin/Linux64/runwb2`), then press **Enter**.
2. From the main menu, select **File** > **Open**.

   The **Open** dialog box appears.
3. Browse to the working directory, set **File name** to `shrouded impeller with cavity.wbpz`, and click **Open**.

   The **Save As** dialog box appears.
4. Accept the default name and click **Save**.

7.2.2. Creating Line Bodies and Curve Groups

The true secondary flow path has an axisymmetric geometry. In this section, you will use BladeEditor to create a 2D closed loop, consisting of line bodies, that outlines the secondary flow path on an axial-radial plane. A 2D sketch of the secondary flow path is provided with the project just opened.

A secondary flow path can have one or more interfaces to the shroud or hub. In this tutorial, there are two interfaces to the shroud. In order to define any given interface to the shroud, the outline must not simply run closely along the shroud curve for the extent of the interface. Instead, to avoid an ill definition (sensitive to numerical round-off) of the interface location, the outline is required to cross the shroud curve into the main passage at one end of the interface, then cross back over the shroud curve at the other end of the interface. For meshing purposes, the outline will be, in effect, trimmed by the shroud curve. For an interface of a secondary flow path to the hub, a similar requirement applies.

A **Geometry** system is visible in the **Project Schematic** view.

1. Right-click the Geometry cell, and select **Edit Geometry in DesignModeler...** to launch DesignModeler.

   You are now ready to edit the geometry using BladeEditor (which is accessed via DesignModeler).

2. In DesignModeler, select **Concept** > **Lines From Sketches**.

   The details view shows the properties for a new line object. The Base Objects property is ready to be defined.

3. In the tree view, select `A:Shrouded Impeller` > `MerPlane1` > `S1_ShroudCavity`.

4. In the details view, beside Base Objects, click **Apply**.

   Property Base Objects is now set to 1 Sketch. The selected sketch, S1_ShroudCavity, will appear in the tree view under the line object after you generate the latter.

5. Click **Generate**.

6. Select **Tools** > **Attribute**.

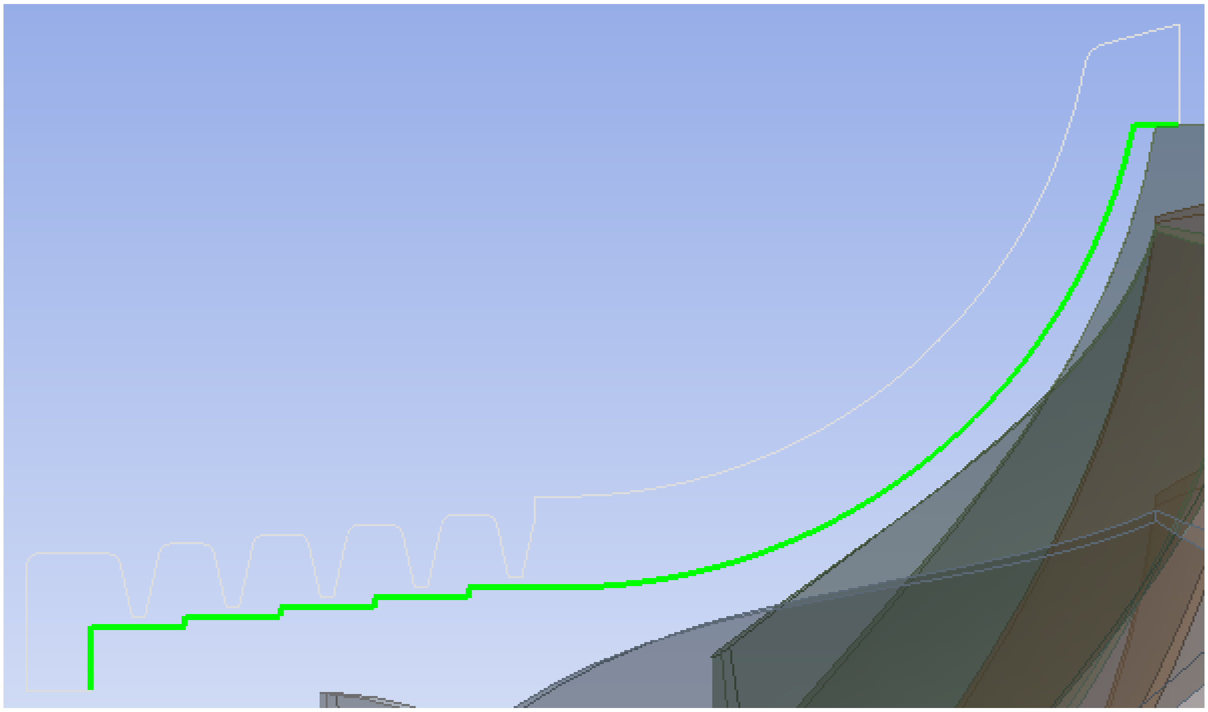
   The details view shows the properties for a new attribute object.

7. Set Attribute Feature Name to `ImpellerShroudGroup`.

8. Set Attribute Name to `NamedSelection:ImpellerShroud2D`.

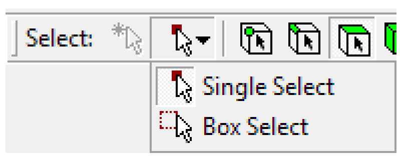
   Note that the prefix "NamedSelection:" is required.

9. For property Geometry, select all 12 of the curves (edges) of the sketch for the secondary passage (S1_ShroudCavity) that are between the shroud interfaces and that are on the main passage side of the cavity, as shown below.



a. Start by clicking the field, beside property Geometry, that currently states "None (Document Level Attribute)".

   The field will change into two buttons: **Apply** and **Cancel**.

b. Click *Single Select* 

to start preparing for the selection of curves from the viewer.

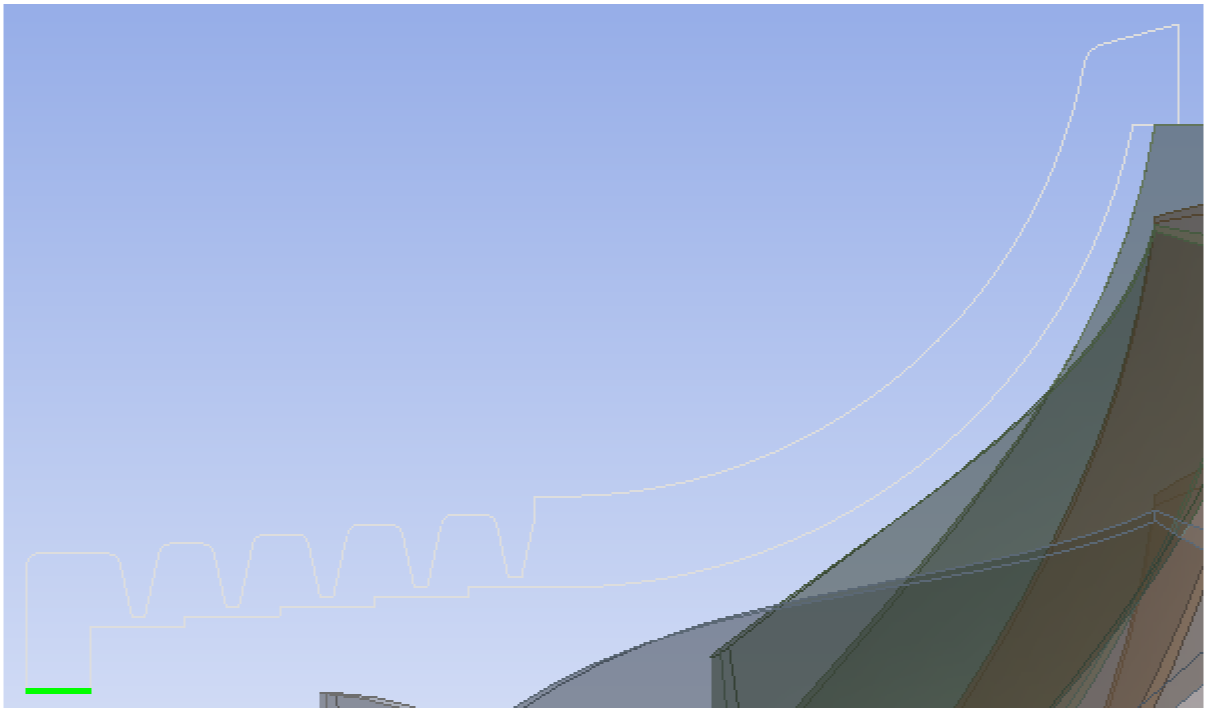
c. Click the selection filter for edges  to finish preparing for the selection of curves from the viewer.

d. Click a curve in the viewer, then, while holding **Ctrl**, click each of the other 11 curves to add them to the selection.

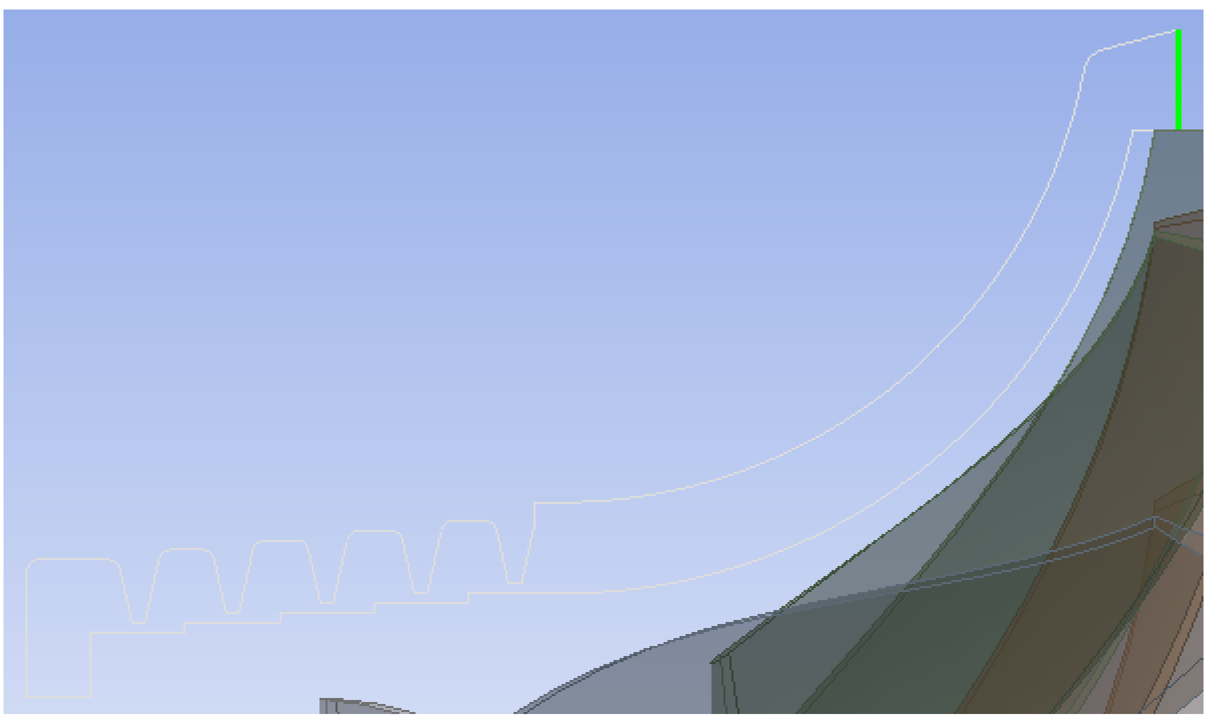
With the appropriate curves selected for property Geometry, click **Apply**, then click **Generate**.

10. Create three other attribute objects as follows:

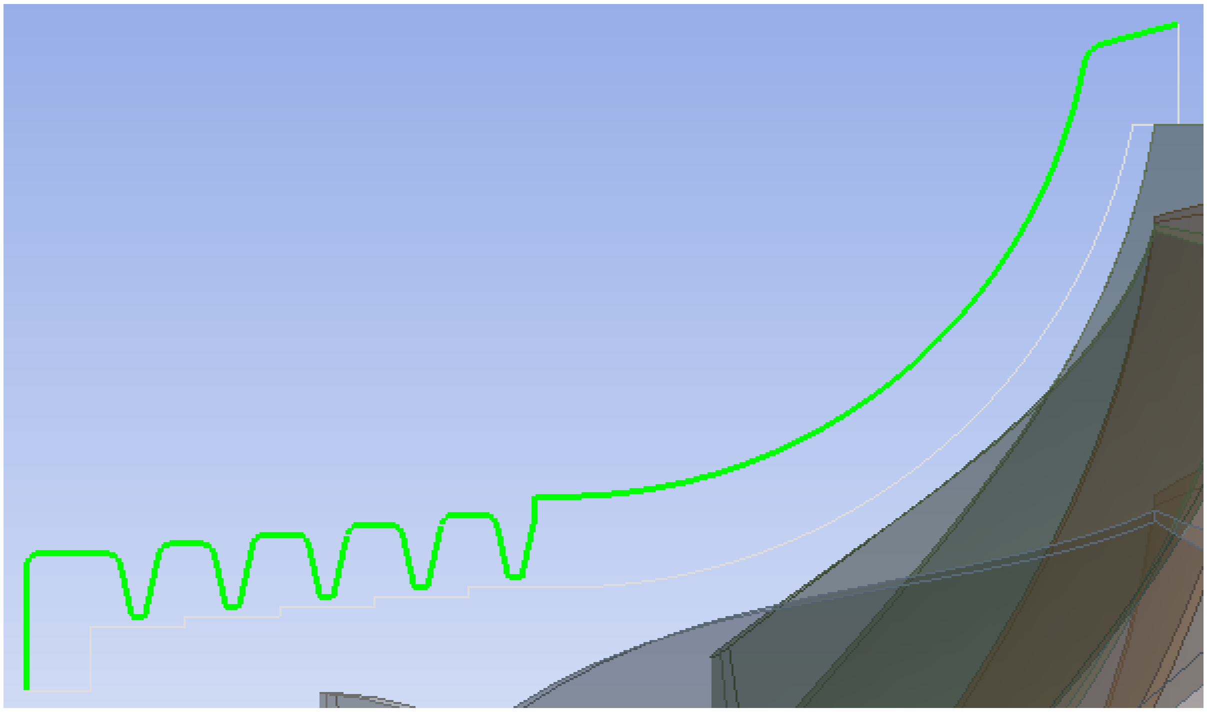
   a. Attribute Feature Name=InterfaceLEGroup; Attribute Name=NamedSelection:InterfaceLE2D; Geometry=curve shown



b. Attribute Feature Name=InterfaceTEGroup; Attribute Name=NamedSelection:InterfaceTE2D; Geometry=curve shown



c. Attribute Feature Name=CasingGroup; Attribute Name=NamedSelection:Casing2D; Geometry=36 curves shown



7.3. Defining the Geometry in TurboGrid

1. Right-click the Geometry cell, and select **Transfer Data To New** > **TurboGrid**.

   A new TurboGrid system appears with a link from the upstream Geometry system.

2. Right-click the Turbo Mesh cell, and select **Edit** to launch TurboGrid.

Before a mesh can be generated in TurboGrid, the geometry must be defined as per the following sections:

7.3.1. Associating CAD Objects with Topology in TurboGrid's Geometry Workspace

1. In TurboGrid, click the **Geometry** tab to switch to the **Geometry** workspace.

2. In the object selector, right-click **Topological Entity Instances** > **Secondary Flow Paths** and select **Insert Secondary Passage**.

   The **Create New Secondary Passage Topology** dialog box appears.

3. Set **Name** to `Shroud Cavity` and click **OK**.

   Incomplete secondary flow path object `Shroud Cavity` appears in the tree under `Secondary Flow Paths`. This object requires boundary entity definitions. In this case, you will add four boundary entity definitions: one for each of the two interfaces with the shroud, and one for each side of the cavity (casing side and shroud side).

4. Right-click `CAD Families` > `CASING2D` and select **Create Entity and Assign Family** > **Shroud Cavity**.

   The **Create New Secondary Passage Boundary** dialog box appears.

5. Accept the default name (`CASING2D1`) and click **OK**.

   Boundary entity `CASING2D1` is listed in the tree under `Shroud Cavity`. Incomplete secondary flow path object `Shroud Cavity` indicates an error (via red text) because it does not currently represent a closed loop.

6. Using the same technique as for the previous two steps, create three other boundary entities under `Shroud Cavity`: `IMPELLERSHROUD2D1`, `INTERFACELE2D1`, `INTERFACETE2D1`.

   For example, create boundary entity `IMPELLERSHROUD2D1` using CAD Family `IMPELLER SHROUD2D`.

   When you are finished this step, secondary flow path object `Shroud Cavity` will represent a closed loop and will no longer indicate an error.

7.3.2. Completing the Geometry Definition in TurboGrid's Mesh Workspace

Set the type for each of the four boundaries of the shroud cavity:

1. In TurboGrid, click the **Mesh** tab to switch to the **Mesh** workspace.
2. In the object selector, under `Geometry` > `Secondary Flow Paths` > `Shroud Cavity`:
   a. Open `CASING2D1` and ensure that its **Boundary Type** is set to `Wall`.
   b. Open `IMPELLERSHROUD2D1` and ensure that its **Boundary Type** is set to `Wall`.
   c. Open `INTERFACELE2D1`, set its **Boundary Type** to `Shroud Interface` and click **Apply**.
   d. Open `INTERFACETE2D1`, set its **Boundary Type** to `Shroud Interface` and click **Apply**.

Switch to the axial-radial view so that you can better see the location of the cavity with respect to the main passage:

- In the upper-left corner of the **3D Viewer**, select **View 3**, then click *Fit View* .

The inlet and outlet blocks should preferably contain the interfaces to the cavity:

- This is the case for the inlet block, which contains `INTERFACELE2D1`. As a result, the shroud can be (and will be, later in this tutorial) automatically divided into regions that align with the edges of `INTERFACELE2D1`. Such alignment helps to improve mesh quality near the interface.
- This is not the case for the outlet block, which fails to contain `INTERFACETE2D1`. In this case, the upstream edge of `INTERFACETE2D1` is already aligned with the topology in the main passage

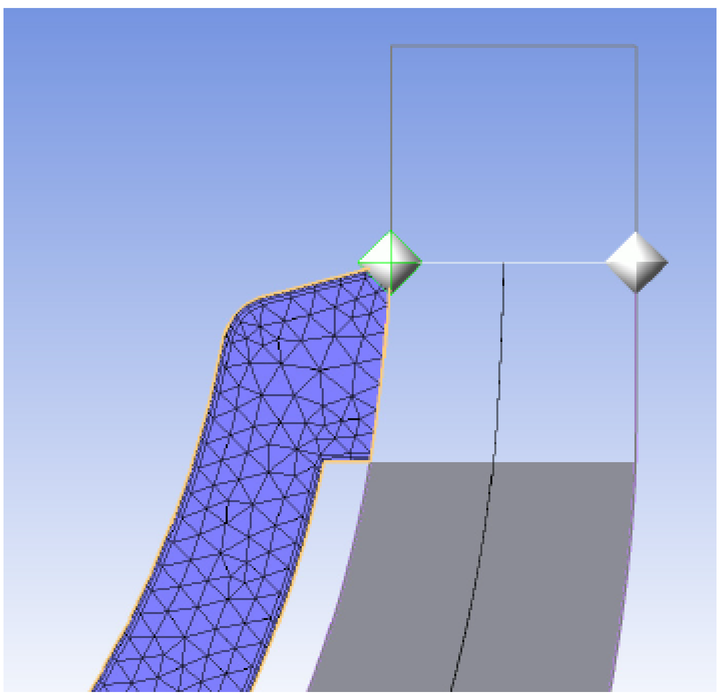
(the cut-off trailing edge). In order to create topological alignment between the other edge of `INTERFACETE2D1` and the topology in the main passage, you will next move the passage outlet to that edge.

Move the passage outlet as follows:

1. Open `Geometry` > `Outlet`.
2. Select **Interface Specification Method** > **Points**.
3. Select `Low Hub Point`, then set **Method** to `Set R` and **Location** to `0.054`.
4. Click **Apply**.
5. Select `Low Shroud Point`, then set **Method** to `Set R` and **Location** to `0.054`.
6. Click **Apply**.

The passage outlet should now be located as shown by Figure 7.1: Passage Outlet at Downstream Edge of Shroud Cavity (p. 63).

Figure 7.1: Passage Outlet at Downstream Edge of Shroud Cavity



This completes the geometry definition.

7.4. Creating the Topology and Initial Mesh

The `Topology Set` object is initially suspended in order to save computational effort while defining the geometry. When you unsuspend the `Topology Set` object, the remaining computations are performed, resulting in a 3D mesh.

Make changes to the `Topology Set`, `Layers`, and `Mesh Data` objects as follows:

1. Open `Topology Set`.
2. Select **Split Mesh Regions At Trailing Edge** and click **Apply**.

   This causes the shroud region in the main passage to be subdivided so that the cavity can be joined downstream of the cut-off trailing edge to a mesh region called "SHROUD DOWNSTREAM".
3. Open `Layers`.
4. Set **Insertion Mode** to `Manual - Uniform`.
5. Set **Count** to `1`.
6. Click **Apply**.
7. Open `Mesh Data`.
8. On the **Mesh Size** tab, set **Method** > **Size Factor** to `1.3` and **Parameters** > **Factor Base** to `4.0`.
9. Ensure that **Inlet Domain** and **Outlet Domain** are selected.
10. Click **Apply**.

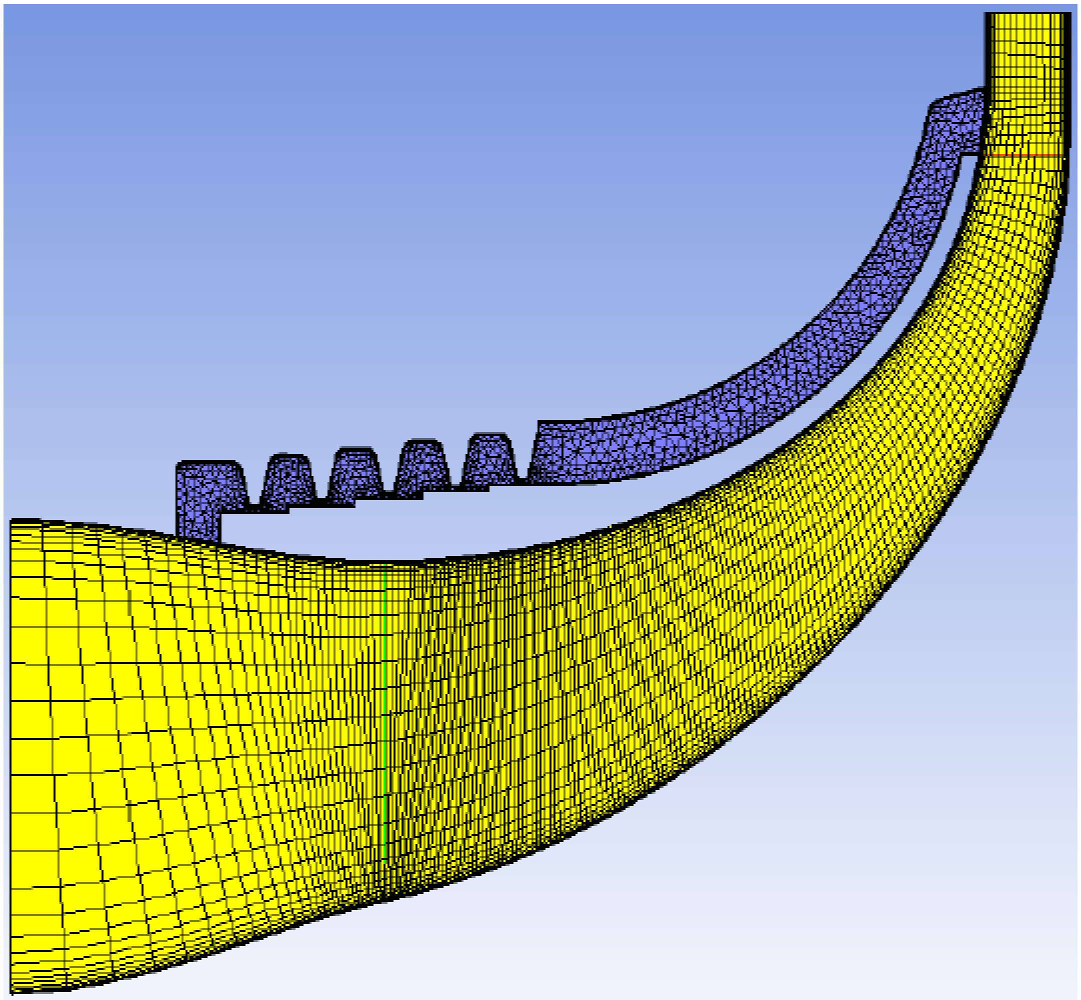
Next, generate the mesh:

- Right-click `Topology Set` and turn off **Suspend Object Updates**.

  The 3D mesh is generated. The number of nodes and elements are displayed at the bottom left.

Figure 7.2: Meridional View of Mesh (with Blade LE and TE) (p. 65) shows the current mesh.

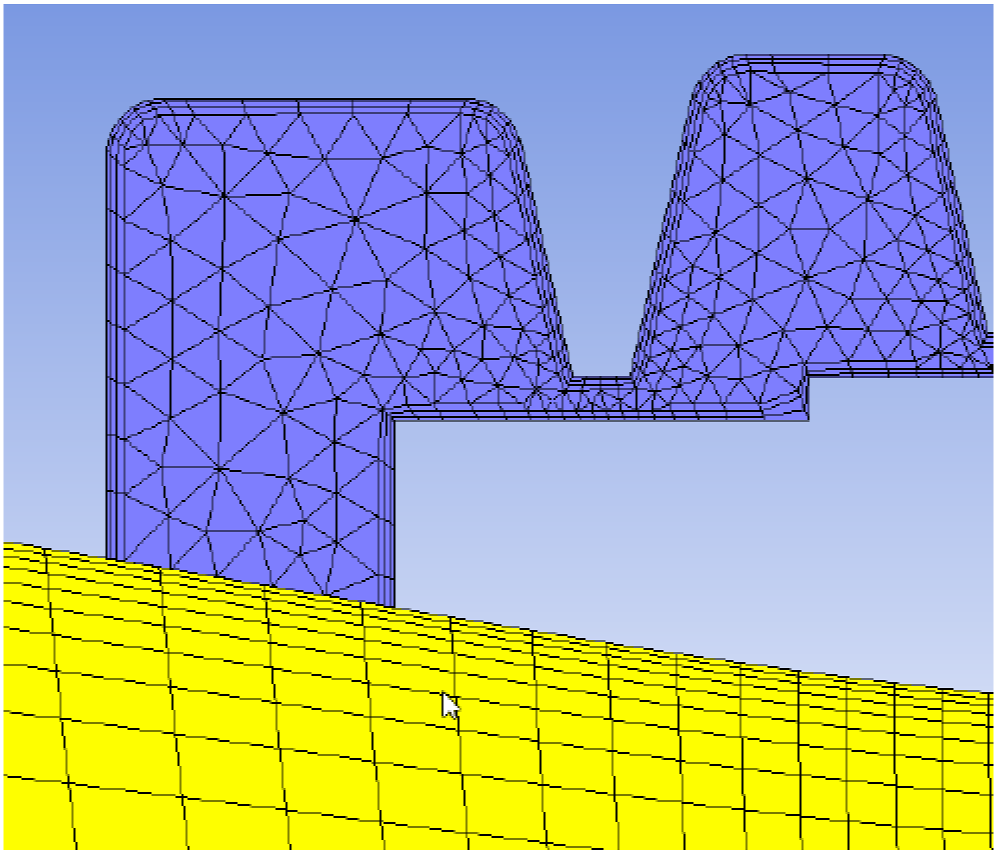
Figure 7.2: Meridional View of Mesh (with Blade LE and TE)



7.5. Aligning Topology at the Upstream Shroud Interface

As shown in Figure 7.3: Upstream Shroud Cavity Interface with Inlet Block - Misaligned Topology (p. 66), the cavity and main passage meshes are not topologically aligned at the edges of the interface.

Figure 7.3: Upstream Shroud Cavity Interface with Inlet Block - Misaligned Topology



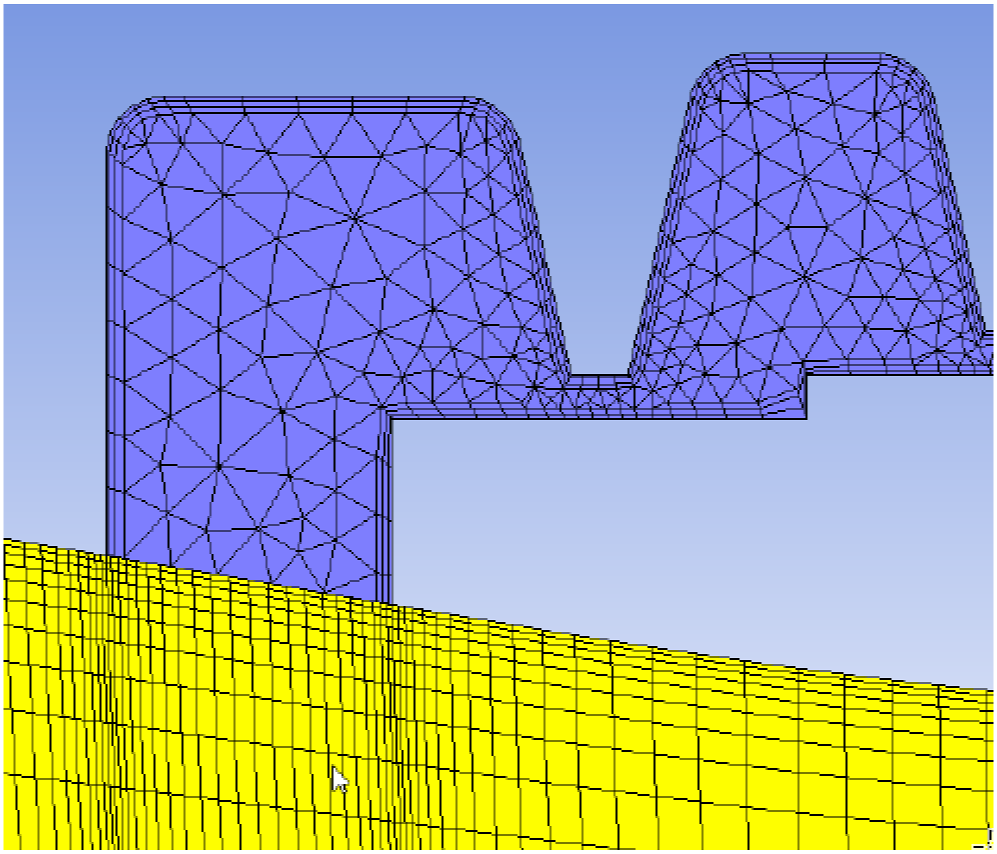
1. Open `Geometry` > `Secondary Flow Paths` > `Shroud Cavity` > `INTERFACELE2D1`.

2. Select **Automatically Manage Opening Region** and click **Apply**.

   A new shroud region, `AUTO INTERFACELE2D1`, is generated and can be reviewed in the `Geometry` > `Shroud` object editor on the **Shroud Regions** tab.

As shown in Figure 7.4: Upstream Shroud Cavity Interface with Inlet Block - Aligned Topology (p. 67), the cavity and main passage meshes are now topologically aligned at the edges of the interface.

Figure 7.4: Upstream Shroud Cavity Interface with Inlet Block - Aligned Topology



7.6. Reviewing the Shroud Interfaces

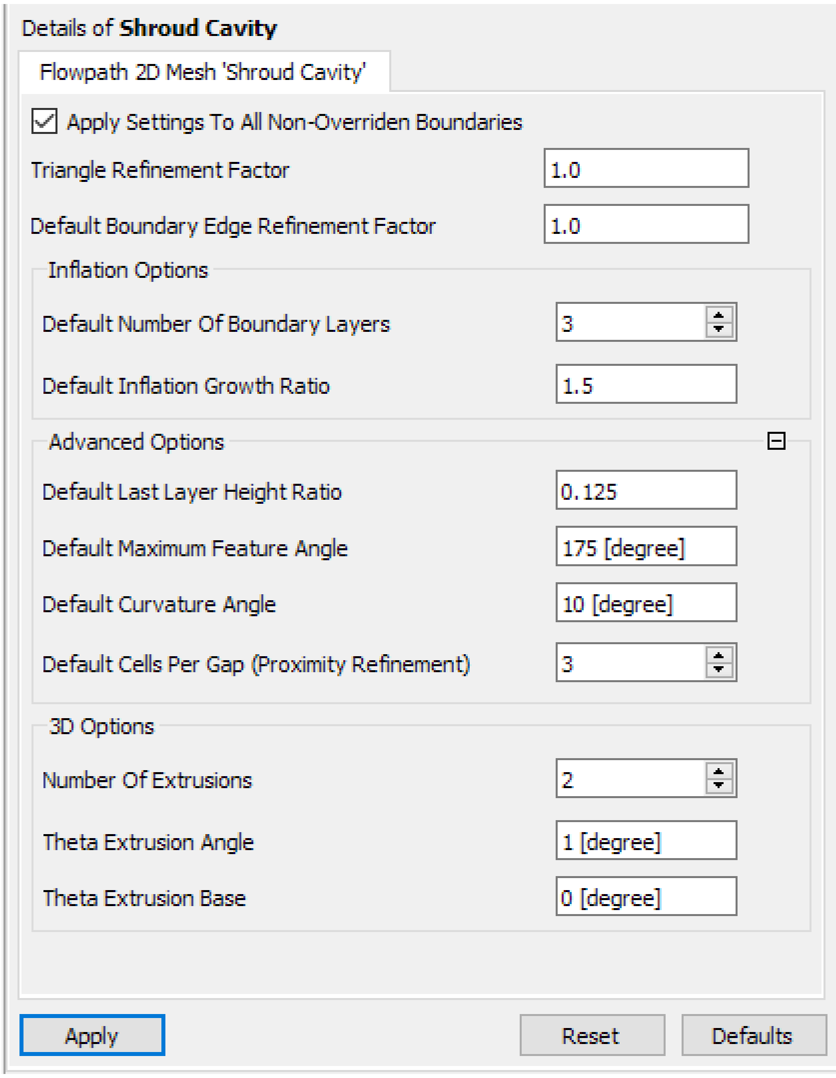
The interface connecting regions are listed under `3D Mesh`:

- Leading edge interface regions:
  - AUTO INTERFACELE2D1
  - Shroud Cavity > Shroud Cavity INTERFACELE2D1
- Trailing edge interface regions:
  - SHROUD DOWNSTREAM
  - Shroud Cavity > Shroud Cavity INTERFACETE2D1

7.7. Secondary Flow Path Mesh Parameters

Adjust the mesh for the secondary flow path as follows:

1. Open **Mesh Data** > **Shroud Cavity**.



2. Increase the overall mesh refinement by increasing `Default Boundary Edge Refinement Factor` to 2.

3. Increase boundary layer refinement and decrease the height of the first layer of elements along all shroud cavity boundaries by increasing **Inflation Options** > **Default Number Of Boundary Layers** to 6.

   You can review the height of the first layer of elements along a shroud cavity boundary by looking in the object editor for `Mesh Data` > `Shroud Cavity` > `[Boundary name]`.

4. Decrease the boundary layer element growth rate by decreasing **Inflation Options** > **Default Inflation Growth Ratio** to `1.3`.

5. Increased refinement in the labyrinth seal tooth gaps by increasing **Advanced Options** > **Default Cells Per Gap (Proximity Refinement)** to 6.

6. Click **Apply**.

The current design only requires a thin mesh. For cases that require a mesh that is thicker in the circumferential direction, 3D options are available.