8. BASIC TURBO MODEL WITH UNSTRUCTURED MESH

This tutorial employs a simple turbine blade configuration to illustrate the basic turbo modeling functionality available in GAMBIT. It illustrates the steps and procedures required for importing data that describes the turbo blade, creating a geometric model that describes the flow region surrounding the blade, meshing the model, and exporting the mesh. The example presented here uses 3-D boundary layers to control the shape of the mesh in the regions immediately adjacent to the blade and employs an unstructured hexahedral mesh.

In this tutorial, you will learn how to:

- Import a turbo data file
- Create a turbo profile
- Modify a turbo profile to affect the shape of a turbo volume
- Create a turbo volume
- Define turbo zones
- Apply 3-D boundary layers to a turbo volume
- Mesh a turbo volume
- View a turbo volume mesh using both 3-D and 2-D perspectives
- Export a turbo volume mesh

8.1 Prerequisites

Prior to reading and performing the steps outlined in this tutorial, you should familiarize yourself with the steps, principles, and procedures described in Tutorials 1, 2, 3, and 4.

8.2 Problem Description

Figure 8-1 shows the turbomachinery configuration to be modeled and meshed in this tutorial. The configuration consists of a turbine rotor on which are affixed 60 identical blades, each of which is spaced equidistant from the others on the rotor hub. Each blade includes a concave (*pressure*) side and a convex (*suction*) side, and the rotor rotates counterclockwise about the *x*-axis, extracting work from the fluid (air) as it flows between the blades (see Figure 8-2).

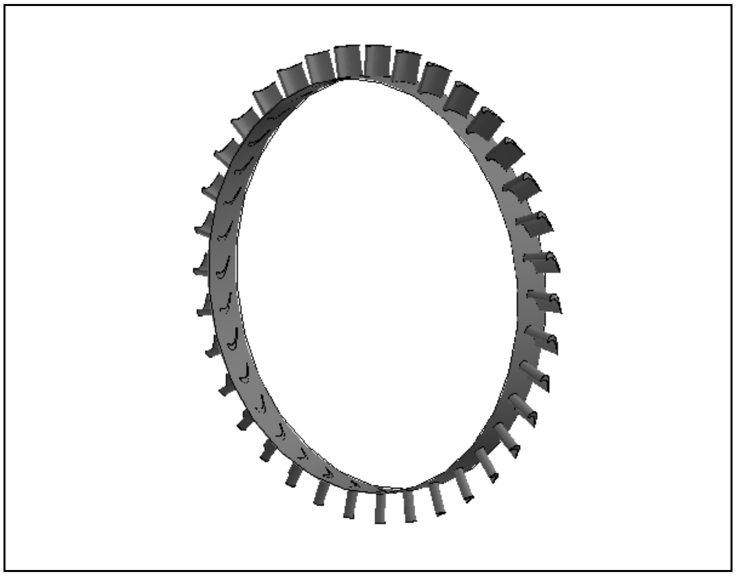


Figure 8-1: 60-blade turbine rotor

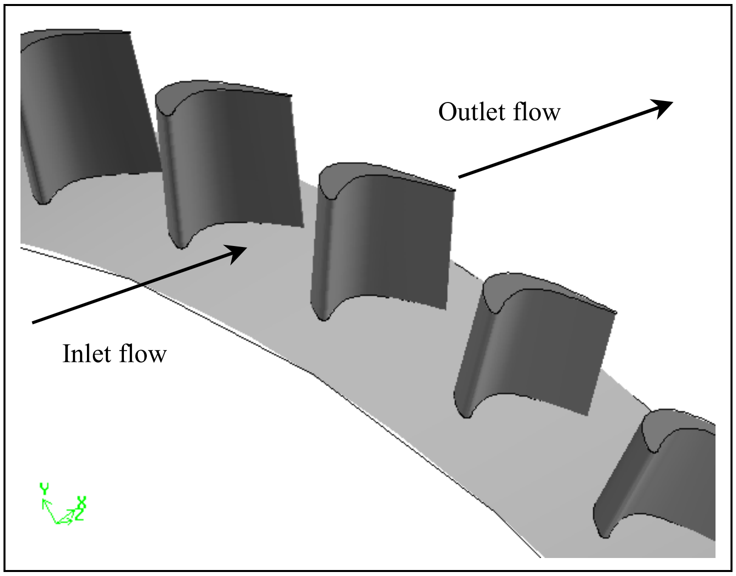


Figure 8-2: Turbine rotor blade configurations

The overall goal of this tutorial is to create a geometric model of the flow region immediately surrounding one of the turbo blades and to mesh the model using an unstructured hexahedral mesh.

8.3 Strategy

In general, the GAMBIT turbo modeling procedure includes seven basic steps:

1) Creating or importing edge data that describes the turbo profile
2) Creating the turbo profile
3) Creating the turbo volume
4) Assigning zone types to regions of the turbo volume
5) Decomposing the turbo volume
6) Meshing the turbo volume
7) Viewing the turbo volume

This tutorial illustrates six of the seven steps listed above. The tutorial excludes the turbo decomposition step, because the turbo volume is to be meshed using unstructured hexahedral mesh elements. Turbo volume decomposition is primarily used to facilitate the creation of structured meshes (see Tutorial 9 in this guide).

NOTE: In this tutorial, the turbo-volume viewing operation (Step 7, above) is illustrated in conjunction with the mesh examination step (see "Step 11:Examine the Mesh," below).

8.4 Procedure

1. Copy the file

   `path/Fluent.Inc/gambit2.x/help/tutfiles/turbo_basic.tur`

   (where `2.x` is the GAMBIT version number) from the GAMBIT installation area in the directory `path` to your working directory.

2. Start GAMBIT using the session identifier "`Basic_Turbo`".

Step 1: Select a Solver

1. Choose the solver from the main menu bar:

   Solver → **FLUENT 5/6**

   The choice of solver affects the types of options available in the **Specify Boundary Types** *form (see "Step 12:Specify Zone Types," below). For some systems,* **FLUENT 5/6** *is the default solver. The currently selected solver is shown at the top of the* GAMBIT *GUI.*

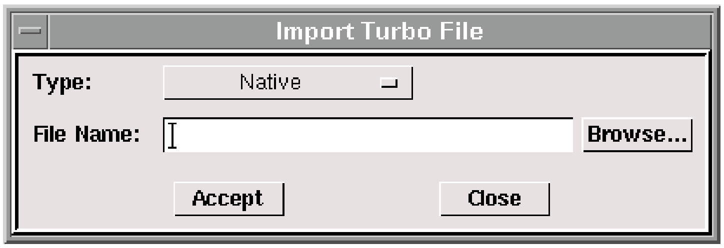
Step 2: Import a Turbo Data File

Turbo data files contain information that GAMBIT *uses to define the turbo profile (see "Step 3:Create the Turbo Profile," below). Such information includes: point data that describes the shapes of the profile edges, edge-continuity data, and specification of the rotational axis for the turbo volume.*

1. Select the **Import Turbo File** option from the main menu bar.

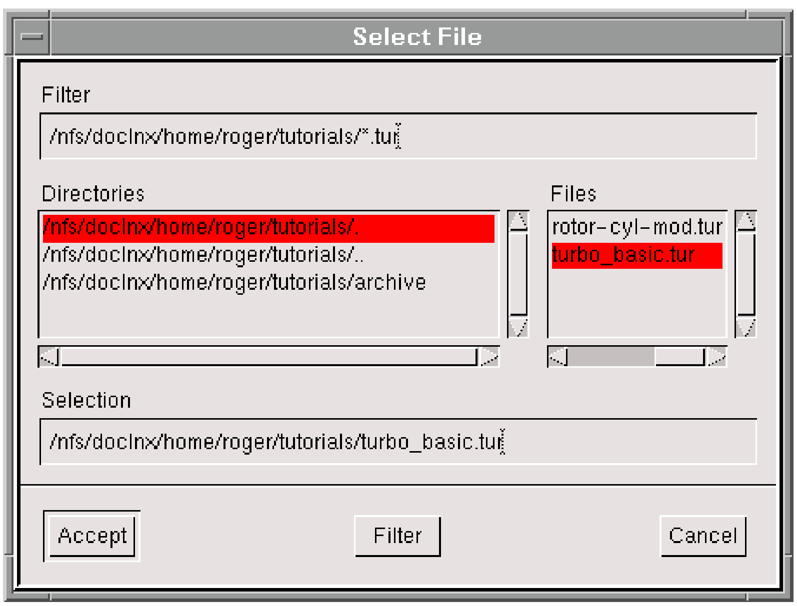
   File → **Import** → **Turbo...**

   This command sequence opens the **Import Turbo File** *form.*



2. Click the **Browse...** button.

   This action opens the **Select File** *form.*



a) In the Files list, select `turbo_basic.tur`.

b) On the **Select File** form, click Accept.

3. On the **Import Turbo File** form, click **Accept**.

   GAMBIT *reads the information contained in the data file and constructs the set of edges shown in Figure 8-3. The two straight edges shown in the figure describe the hub and casing for the turbo volume. The two sets of curved edges constitute cross sections of a single turbo blade.*

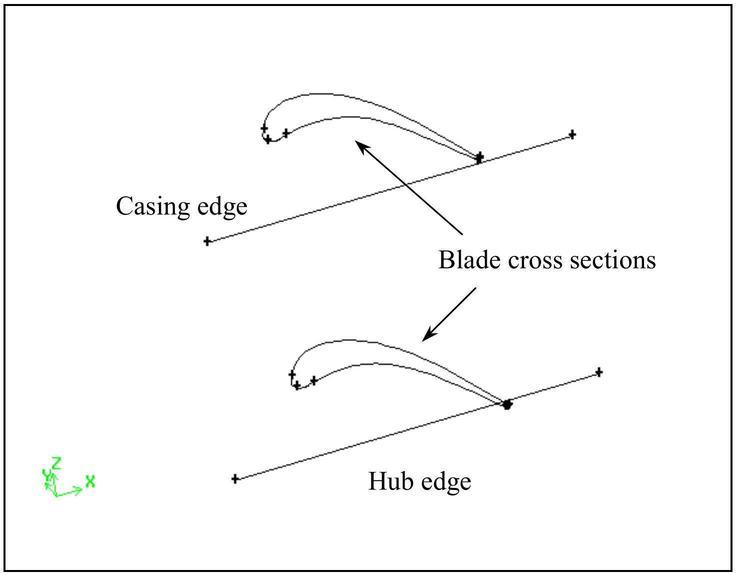


Figure 8-3: Imported turbo geometry

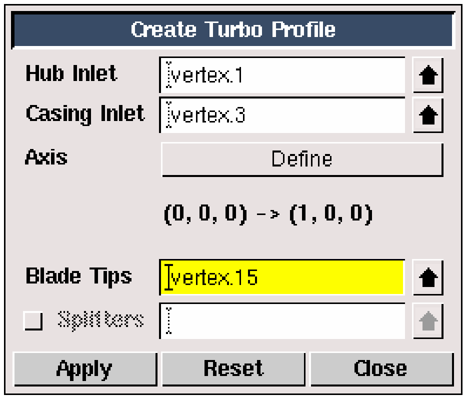
Step 3: Create the Turbo Profile

The turbo profile defines the basic characteristics of the turbo volume, including the shapes of the hub, casing, and periodic (side) surfaces. In GAMBIT*, the edges that describe the hub, casing, and blade cross sections are defined by means of their inlet endpoint vertices.*

1. Specify the hub, casing, and blade-cross-section edges of the turbo profile.

   TOOLS → **TURBO** → **CREATE PROFILE**

   This command sequence opens the **Create Turbo Profile** *form.*



   In this step, you will specify vertices that define the hub, casing, and blade cross-sections. In addition, you will specify the axis of revolution for the turbo configuration. All instructions listed in this step refer to the vertex labels shown in Figure 8-4.

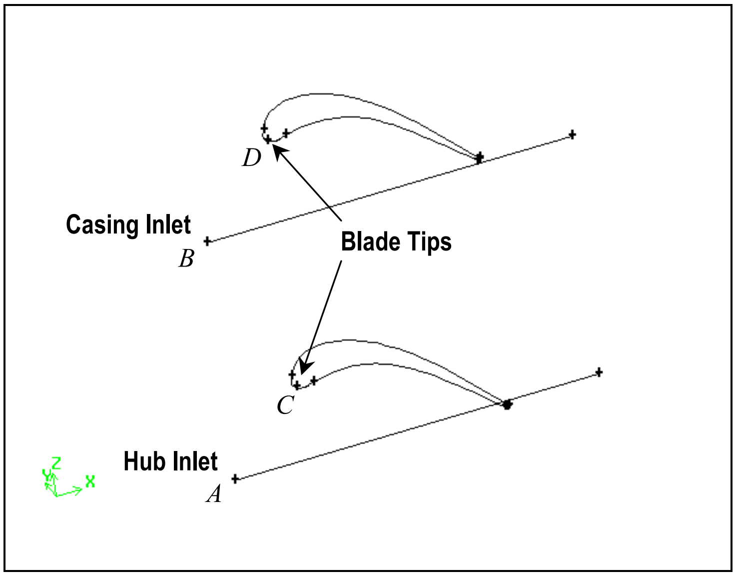


Figure 8-4: Vertices used to specify the turbo profile

a) Activate the **Hub Inlet** list box on the **Create Turbo Profile** form.

To activate an input field, such as a list box, on any GAMBIT *specification form,* left-*click in the input box located adjacent to the field label—in this case, "***Hub Inlet***". (By default,* GAMBIT *activates the* **Hub Inlet** *field when you open the* **Create Turbo Profile** *form.)*

b) Select vertex *A*.

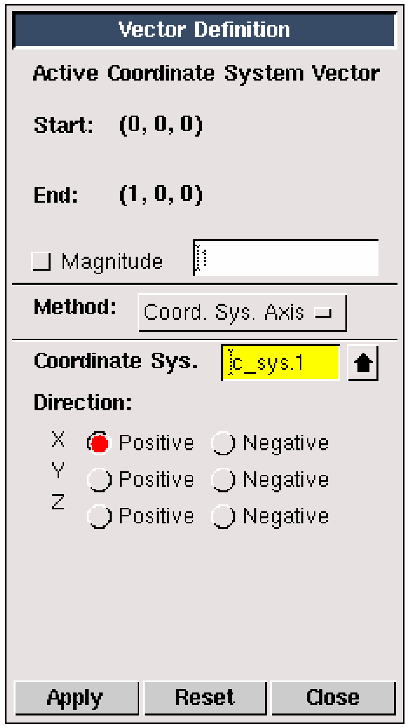
c) Activate the **Casing Inlet** list box.

d) Select vertex *B*.

e) Specify the *x* axis as the axis of revolution for the turbo configuration.

   i. Click the **Axis:**Define pushbutton.

      This action opens the **Vector Definition** *form.*



ii. Select the **Direction**:X-Positive option.

iii. On the **Vector Definition** form, click **Apply**.

f) Activate the **Blade Tips** list box.

g) Select vertex *C*.

h) Select vertex *D*.

! *The order in which the* **Blade Tips** *vertices are selected is important to the definition of a turbo profile. Specifically, the* **Blade Tips** *vertices must be selected in order from the hub cross section to the casing cross section.*

i) Click **Apply** to accept the vertex selections and create the turbo profile.

GAMBIT *creates the turbo profile shown in Figure 8-5.*

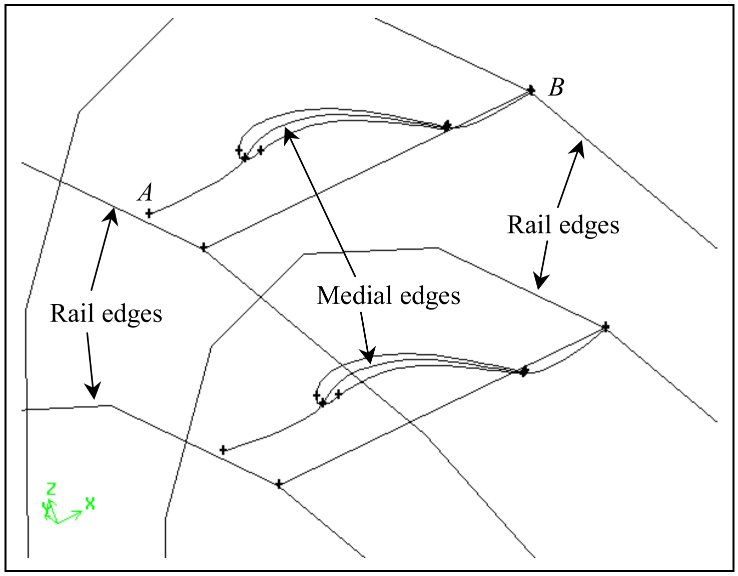


Figure 8-5: Turbo profile

The profile includes six new edges, four of which are real edges and two of which are virtual edges. The four real edges are circular arc ("rail") edges that are formed by revolving the hub and casing endpoint vertices about the axis of revolution for the profile. The two virtual edges are "medial" edges, the centermost shapes of which represent the mean shapes of the blade cross sections. The endpoint vertices of the medial edges are hosted by the rail edges, and the medial edges are defined such that they pass through the leading and trailing vertices of the blade cross sections. The medial edges define the shapes of the periodic surfaces on the turbo volume (see "Step 5:Create the Turbo Volume," below).

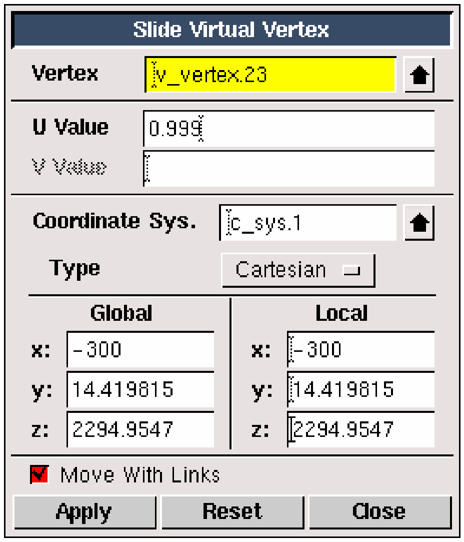
Step 4: Modify the Inlet and Outlet Vertex Locations

It is often useful to control the shape of the turbo volume such that its inlet and outlet surfaces represent smooth flow transitions to and from the inlet and outlet ends, respectively, of the turbo blade. In GAMBIT*, you can control the shape of the turbo volume by adjusting the positions of the medial-edge endpoint vertices prior to constructing the volume.*

1. Open the **Slide Virtual Vertex** form.

   TOOLS → **TURBO** → **SLIDE VIRTUAL VERTEX**

   This command sequence opens the **Slide Virtual Vertex** *form.*



   a) Select the inlet endpoint vertex of the medial edge for the casing blade cross section (vertex *A* in Figure 8-5, above).

   b) In the **U Value** field, enter the value `0.999`.

      As an alternative to entering a value in the **U Value** *field, you can select the vertex in the graphics window and drag it along its host rail edge until the* **U Value** *field value is 0.999.*

   c) Retain the Move With Links (*default*) option.

The Move With Links *option specifies that* GAMBIT *is to apply the current* **Slide Virtual Vertex** *specifications to all medial-edge inlet endpoint vertices in addition to the selected vertex.*

d) Click **Apply** to accept the new position of the medial-edge inlet endpoint vertices.

e) Select the outlet endpoint vertex of the medial edge for the casing blade cross section (vertex *B*).

f) In the **U Value** field, enter the value `0.019`.

g) Retain the Move With Links (*default*) option.

h) Click **Apply** to accept the new position of the medial-edge outlet endpoint vertices.

The modified turbo profile appears as shown in Figure 8-6.

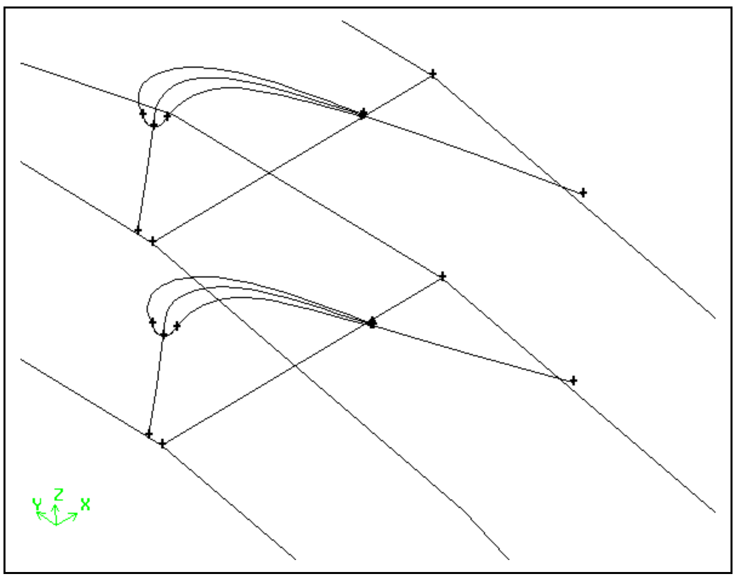


Figure 8-6: Turbo profile with modified inlet and outlet vertex locations

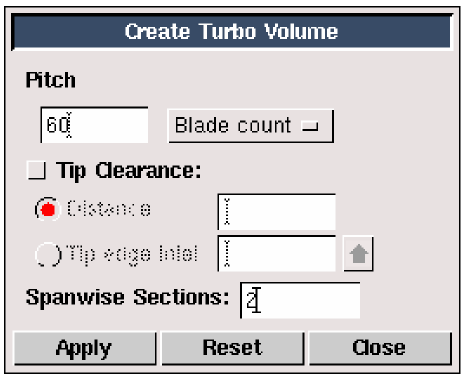
Step 5: Create the Turbo Volume

A "turbo volume" is a 3-D region—which is defined by a set of one or more geometric volumes—that represents the flow environment surrounding the turbo blade. The turbo volume characteristics are determined by the turbo profile and by specification of the number of blades on the rotor (or angle between blades), the tip clearance, and the number of spanwise sections. This example does not *include a tip clearance but does include spanwise sectioning.*

1. Specify the pitch and number of spanwise sections for the turbo volume.

   TOOLS  → **TURBO**  → **CREATE TURBO VOLUME** 

   This command sequence opens the **Create Turbo Volume** *form.*



   a) In the **Pitch** text box, enter `60`.

   b) On the **Pitch** option button (located to the right of the **Pitch** text box), select the Blade count option.

   c) In the **Spanwise Sections** text box, enter `2`.

   d) Click **Apply**.

      GAMBIT *creates the turbo volume shown in Figure 8-7.*

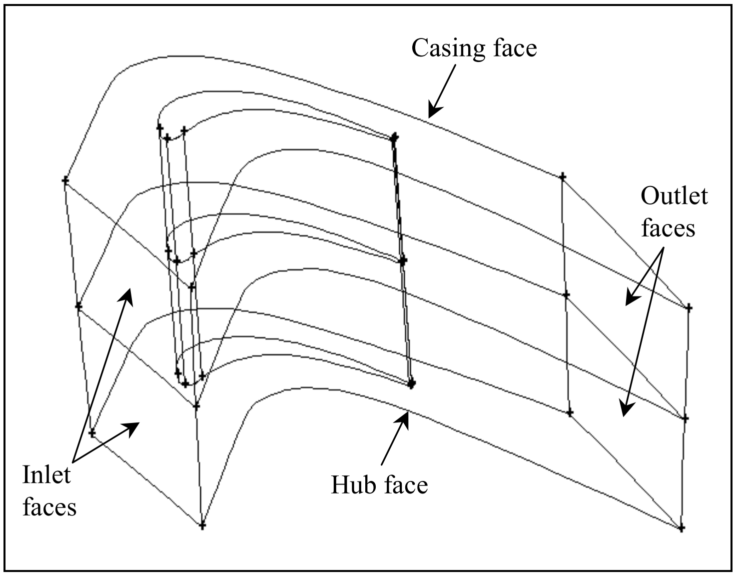


Figure 8-7: Turbo volume—consisting of two geometric volumes

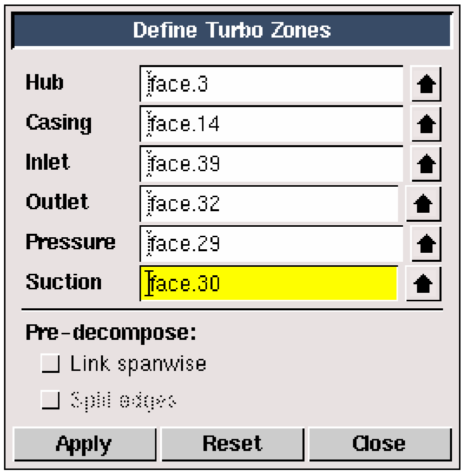
Step 6: Define the Turbo Zones

This step standard zone types to surfaces of the turbo volume. The zone-type specifications determine which faces are linked for meshing. In addition, GAMBIT *automatically associates turbo zone types to boundary zone definitions for some solvers.*

1. Specify the faces that constitute the hub, casing, inlet, outlet of the turbo volume, as well as the pressure and suction sides of the turbo blade.

   TOOLS  → **TURBO**  → **DEFINE TURBO ZONES** 

   This command sequence opens the **Define Turbo Zones** *form.*



   a) Activate the **Hub** list box.

   b) Select the bottom (*hub*) face of the turbo volume (see Figure 8-7, above).

   c) Activate the **Casing** list box.

   d) Select the top (*casing*) face of the turbo volume.

   e) Activate the **Inlet** list box.

   f) Select the two *inlet* faces.

   g) Activate the **Outlet** list box.

   h) Select the two *outlet* faces.

i) Activate the **Pressure** list box.

j) Select the six faces on the inner-curve (*pressure* side) of the turbo blade.

k) Activate the **Suction** list box.

l) Select the six faces on the outer-curve (*suction* side) of the turbo blade.

m) Click **Apply** to assign the turbo zone types.

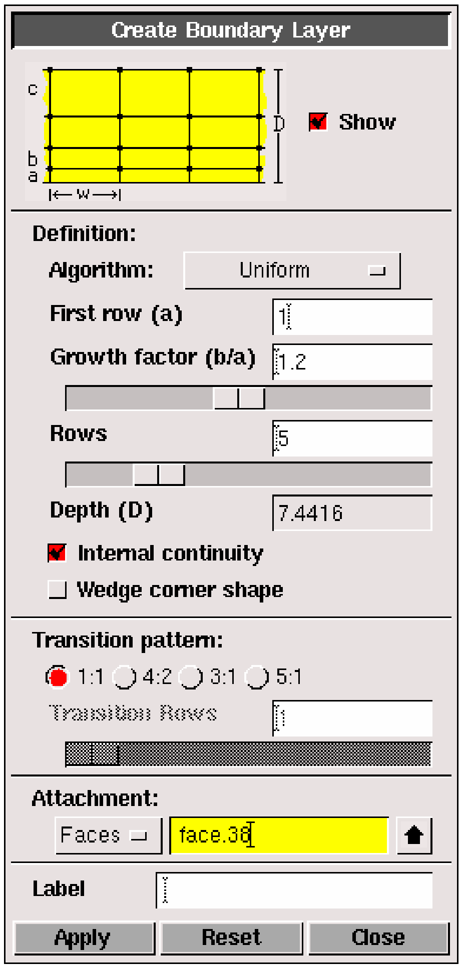
Step 7: Apply 3-D Boundary Layers

For turbo models, 3-D boundary layers allow you to ensure the creation of high-quality mesh elements in regions adjacent to the turbo blade surfaces. Such boundary layers are particularly useful when the turbo volume is to be meshed using an unstructured meshing scheme.

1. Specify the hub, casing, and blade-cross-section edges of the turbo profile.

   TOOLS → **TURBO** → **CREATE/MODIFY BOUNDARY LAYERS**

   This command sequence opens the **Create Boundary Layer** *form.*



a) In the **First row** text box, enter a value of `1`.

b) In the **Growth factor** text box, enter a value of `1.2`.

c) In the **Rows** text box, specify a value of `5`, either by direct input of the value or by sliding the **Rows** slider bar.

GAMBIT *automatically calculates a* **Depth** *value of 7.4416, based on the* **First row***,* **Growth factor***, and* **Rows** *specifications.*

d) Select the **Internal continuity** option.

e) In the **Attachment** input field, select the Faces option.

f) Activate the Faces list box, and select the 12 faces that comprise the pressure and suction sides of the turbo blade.

g) Click **Apply**.

Figure 8-8 shows the 3-D boundary layers projected onto the three spanwise surfaces of the turbo volume.

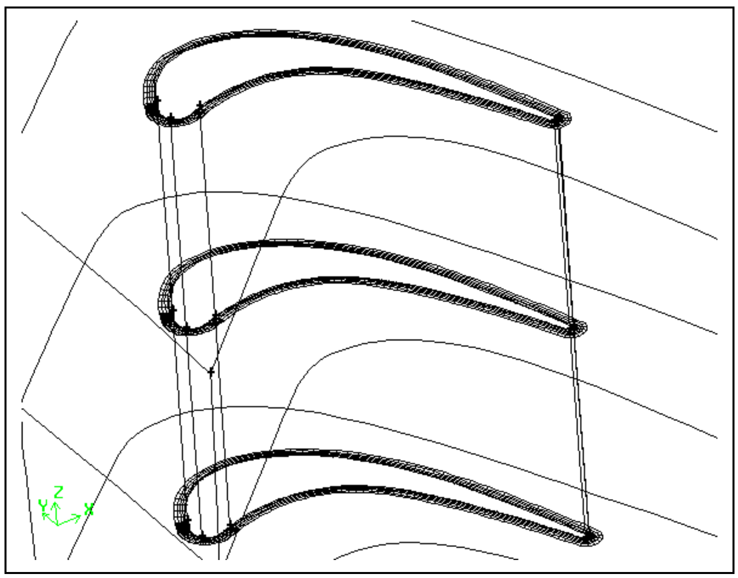
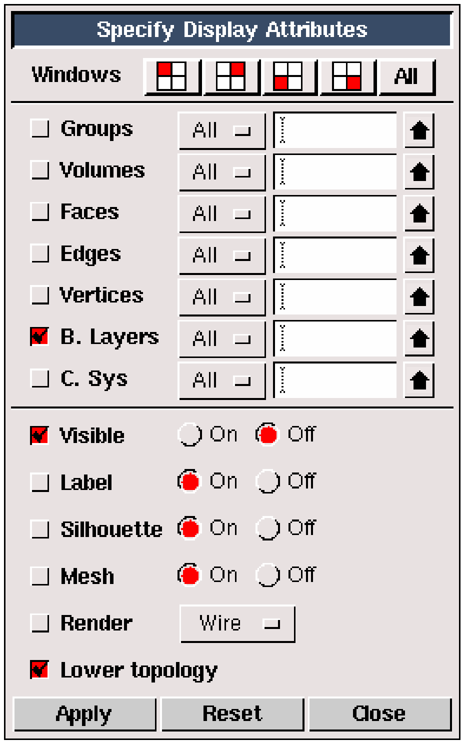


Figure 8-8: Turbo volume with 3-D boundary layers

By default, GAMBIT *displays the boundary layers in the graphics window unless they are made invisible by direct user action. The boundary layer display can make it difficult to view the model during subsequent steps in the modeling process; therefore, it is advisable to render the boundary layers invisible before continuing the tutorial.*

2. Select the **SPECIFY DISPLAY ATTRIBUTES**  command button on the **Global Control** toolpad.

This action opens the **Specify Display Attributes** *form.*



a) Select the **B. Layers** check box.

b) Select the **Visible:**Off option.

c) Click **Apply**.

GAMBIT *turns off the display of the boundary layers.*

d) **Close** the **Specify Display Attributes** form.

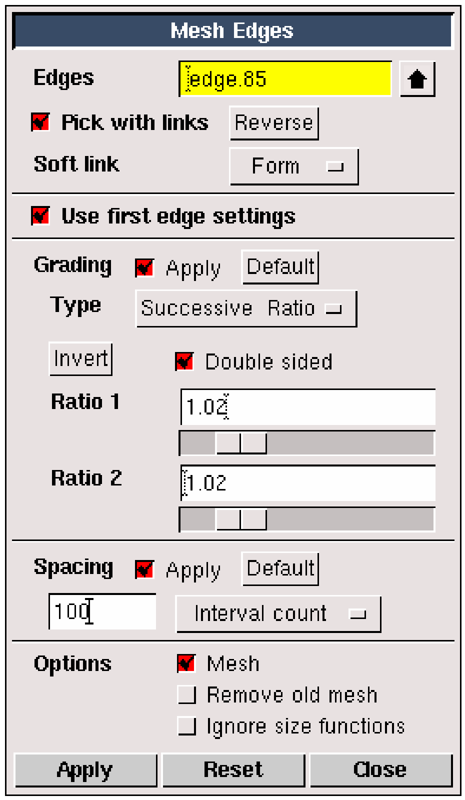
Step 8: Mesh the Blade Cross-Section Edges

In this step, you will pre-mesh the edges that represent the blade cross sections, thereby ensuring a finer mesh in proximity to the turbo blade surfaces than is created in the bulk of the turbo volume.

1. Mesh the centermost pressure-side edges of the turbo blade.

   TOOLS → **TURBO** → **MESH EDGES/FACES/VOLUMES**

   This command sequence opens the **Mesh Edges** *form.*



   a) Activate the **Edges** list box, and select the three centermost edges on the *pressure* side of the blade cross sections.

   b) On the **Grading:Type** option button, retain Successive Ratio.

c) In the **Ratio** input field, enter a value of `1.02`.

d) Select the Double sided option.

When you select the Double sided *option,* GAMBIT *changes the* **Ratio** *input field to* **Ratio 1** *and displays a field named* **Ratio 2** *that contains a ratio specification identical to that of* **Ratio 1** *(that is, 1.02).*

e) On the **Spacing** option button, select Interval count.

f) In the **Spacing** text box, enter a value of `100`.

g) Click **Apply**.

GAMBIT *meshes the selected edges as shown in Figure 8-9. The* Double sided *option with a ratio of 1.02 grades the edges such that mesh nodes are bunched near the endpoint vertices of the edges.*

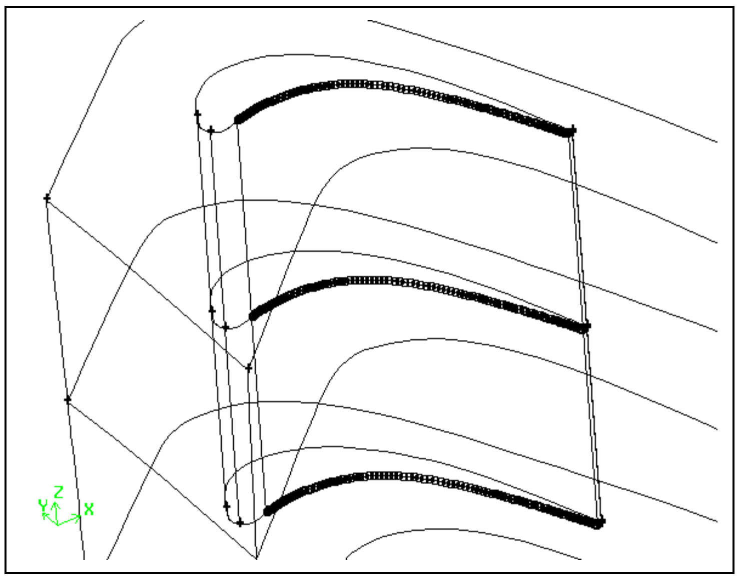


Figure 8-9: Meshed centermost pressure-side edges of the turbo blade

2. Mesh the suction-side edges of the turbo blade.

a) Activate the **Edges** list box, and select the three centermost edges on the *suction* side of the blade cross sections.

b) On the **Grading:Type** option button, retain Successive Ratio.

c) In the **Ratio** input field, enter a value of `1.02`.

d) Select the Double sided option.

e) On the **Spacing** option button, retain Interval count.

f) In the **Spacing** text box, enter a value of `110`.

g) Click **Apply**.

3. Mesh the leading edges of the turbo blade.

a) Activate the **Edges** list box.

b) Select the six edges (two edges on each cross section) on either side of the *leading* vertices for the top, middle, and bottom blade cross sections.

! *When selecting the edges, modify the edge senses, as necessary, such that they point away from the leading vertices of the cross sections. When you select an edge in the graphics window,* GAMBIT *automatically displays an arrowhead in the middle of the edge to indicate the sense of the edge. To change the sense of any selected edge, Shift-middle-click the edge. (NOTE: If the sense-direction arrowhead is obscured by mesh nodes displayed on the edge, set the* **Interval count** *to* `1` *while selecting edges for meshing.)*

c) On the **Grading:Type** option button, retain Successive Ratio.

d) In the **Ratio** input field, enter a value of `1.05`.

The single-sided meshing option with a ratio of 1.05 grades the edges such that mesh nodes are bunched near the leading vertices of the edges—that is, in the regions of highest curvature for the edges.

e) On the **Spacing** option button, retain Interval count.

f) In the **Spacing** text box, enter a value of `15`.

g) Click **Apply**.

4. Mesh the trailing edges of the turbo blade.

a) Activate the **Edges** list box.

b) Select the six edges (two edges on each cross section) on either side of the *trailing* vertices for the three blade cross sections.

c) On the **Grading:Type** option button, retain Successive Ratio.

d) In the **Ratio** input field, enter a value of `1`.

e) On the **Spacing** option button, retain Interval count.

f) In the **Spacing** text box, enter a value of `3`.

g) Click **Apply**.

Figure 8-10 shows the final edge-mesh configuration for the turbo blade cross sections.

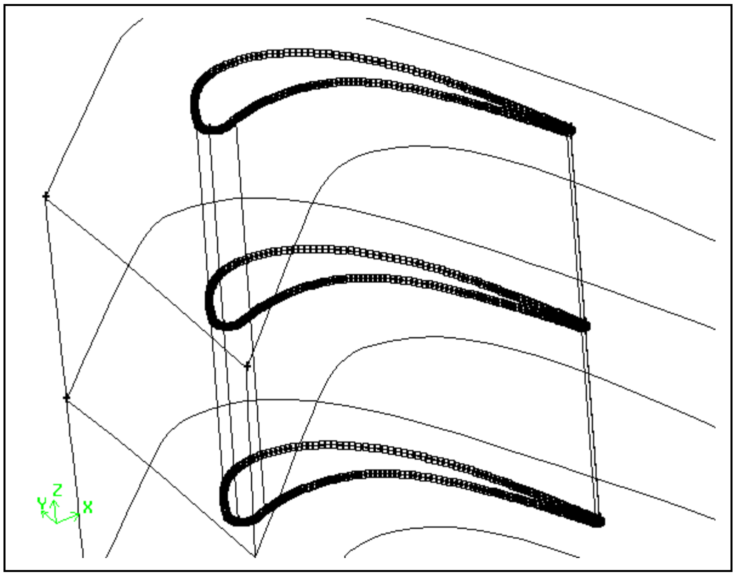


Figure 8-10: Meshed edges of turbo blade cross sections

Step 9: Mesh the Center Spanwise Face

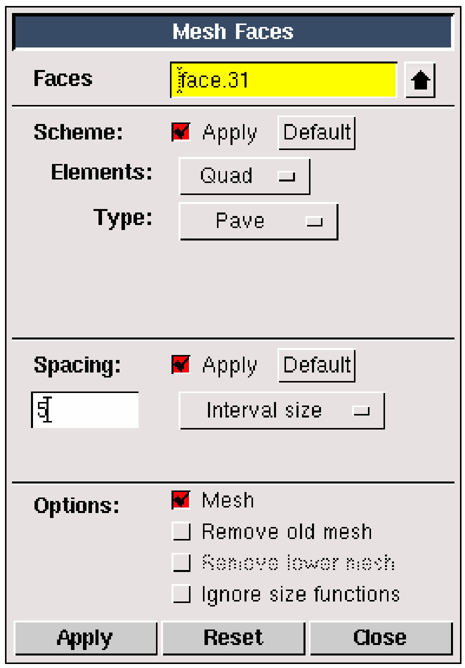
To create an unstructured mesh for this example, it is best to pre-mesh the middle spanwise face and to employ the middle face as a source face for a Cooper *meshing scheme applied to the two geometric volumes. The use of the middle face as a source face ensures that the* Cooper *scheme produces a mesh with minimal distortion throughout the turbo volume.*

1. Mesh the center spanwise face of the turbo volume.

   TOOLS → **TURBO** → **MESH EDGES/FACES/VOLUMES**

   R Mesh Faces

   This command sequence opens the **Mesh Faces** *form.*



   a) Activate the **Faces** list box, and select the middle spanwise face.

      GAMBIT *automatically selects the* Quad *and* Pave **Scheme** *options based on the face characteristics.*

b) On the **Scheme:Elements** option button, retain the Quad option.

c) On the **Scheme:Type** option button, retain the Pave option.

d) On the **Spacing** option button, select the Interval size option.

e) In the **Spacing** text box, enter a value of `5`.

f) Click **Apply**.

GAMBIT *meshes the middle spanwise face as shown in Figure 8-11.*

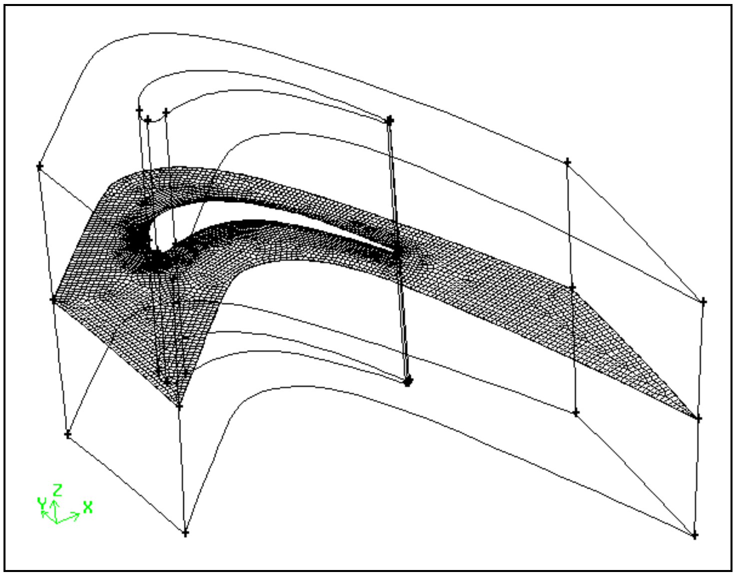


Figure 8-11: Meshed center spanwise face

Step 10: Mesh the Volumes

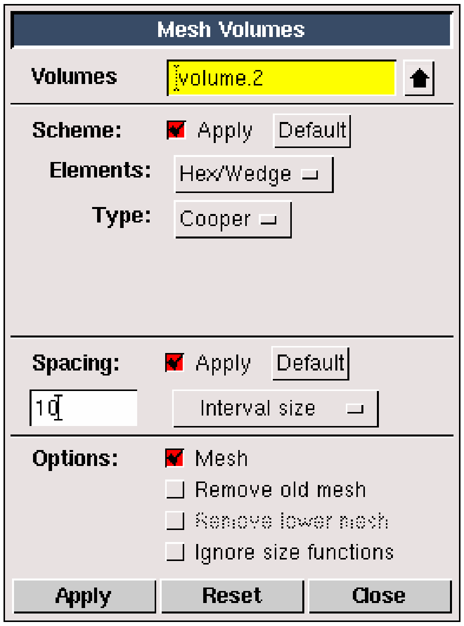
In this step, you will apply a Cooper *meshing scheme to the two geometric volumes that comprise the turbo volume.*

1. Mesh the turbo volume.

TOOLS  → **TURBO**  → **MESH EDGES/FACES/VOLUMES** 

R 

This command sequence opens the **Mesh Volumes** *form.*



a) Activate the **Volumes** list box, and select the both of the geometric volumes that comprise the turbo volume.

   GAMBIT *automatically selects the* **Scheme:Elements**:Hex/Wedge *and* **Scheme: Type:**Cooper *options for the selected volumes.*

b) Retain the automatically selected **Scheme** options.

c) On the **Spacing** option button, select Interval size.

d) In the **Spacing** text box, enter a value of `10`.

e) Click **Apply**.

GAMBIT *meshes the volumes as shown in Figure 8-12.*

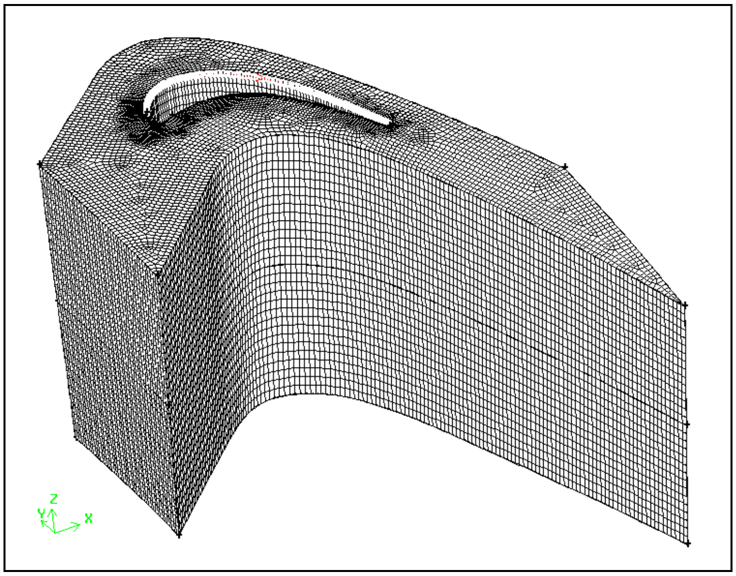
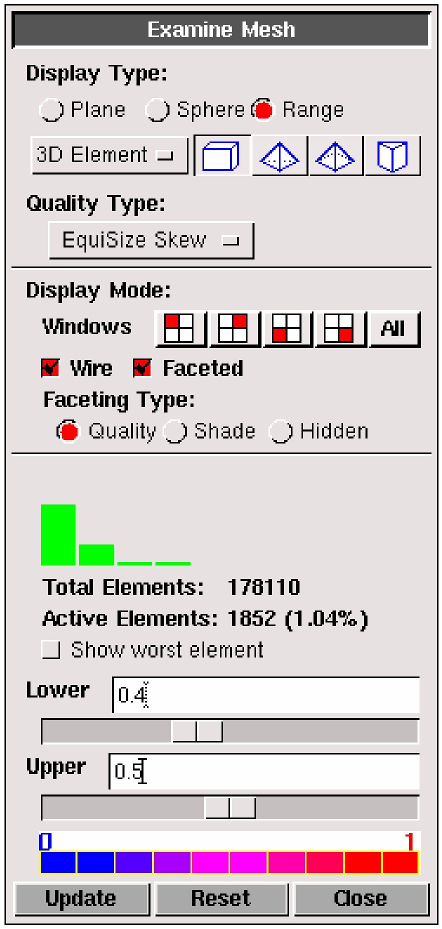


Figure 8-12: Meshed volumes

Step 11: Examine the Mesh

1. Select the **EXAMINE MESH**  command button at the bottom right of the **Global Control** toolpad.

   This action opens the **Examine Mesh** *form.*



   The **Examine Mesh** *form allows you to view various mesh characteristics for the 3-D mesh. For example, Figure 8-13 displays volume mesh elements for which the* EquiSize Skew *parameter is between 0.4 and 0.5 for this example.*

   a) Click **Update** at the bottom of the **Examine Mesh** form.

GAMBIT *does* not *automatically update the graphics display when you open the* **Examine Mesh** *form or modify its specifications, such as* **Display Type** *or* **Quality Type***. To update the graphics display, you must click the* **Update** *push-button located at the bottom of the form.* GAMBIT *displays the* **Update** *pushbutton label in red lettering whenever the display needs to be updated to reflect the current* **Examine Mesh** *specifications.*

Some **Examine Mesh** *operations automatically update the graphics display. For example, if you select the* **Display Type:**Range *option and click one of the histogram bars,* GAMBIT *automatically updates the display.*

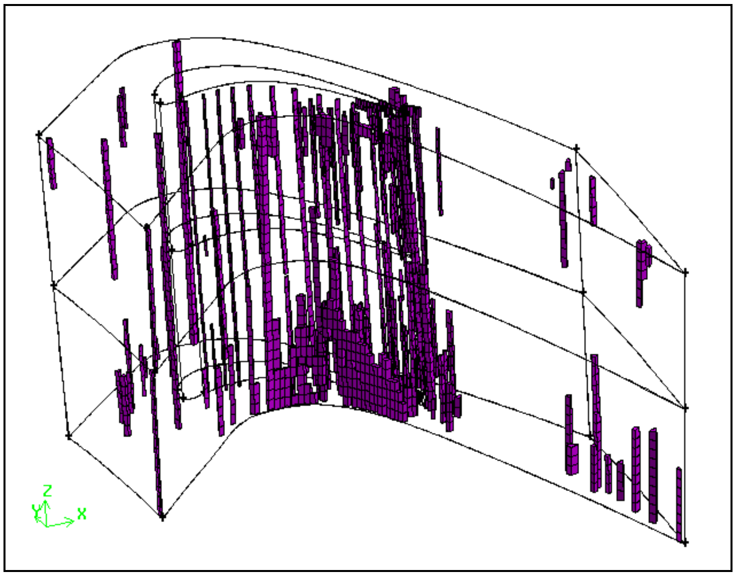
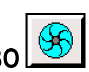
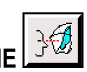


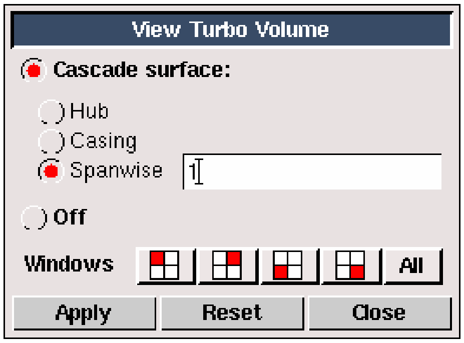
Figure 8-13: Hexahedral mesh elements—EquiSize Skew = 0.4–0.5

The **Examine Mesh** *command and options can be used in conjunction with the* **View Turbo Volume** *command to view 2-D characteristics of the mesh on the hub, casing, and spanwise surfaces. Such views are particularly useful when examining the mesh on highly twisted blades.*

2. Display the middle spanwise surface in a cascade turbo view.

TOOLS  → **TURBO**  → **VIEW TURBO VOLUME** 

This command sequence opens the **View Turbo Volume** *form.*

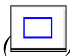


a) Select the **Cascade surface:**Spanwise option.

b) In the Spanwise text box, enter a value of `1`.

The **Cascade surface** *specifications described above specify a flattened, 2-D display of the middle spanwise surface of the turbo volume.*

c) Click **Apply**.

Figure 8-14 displays face mesh elements for which the EquiSize Skew *parameter is between 0.1 and 0.3 for this example. (NOTE: To view the 2-D face elements shown in Figure 8-14, select the* **Display Type:** 2D Element *option on the* **Examine Mesh** *form, and specify the display of quadrilateral (**) elements.)*

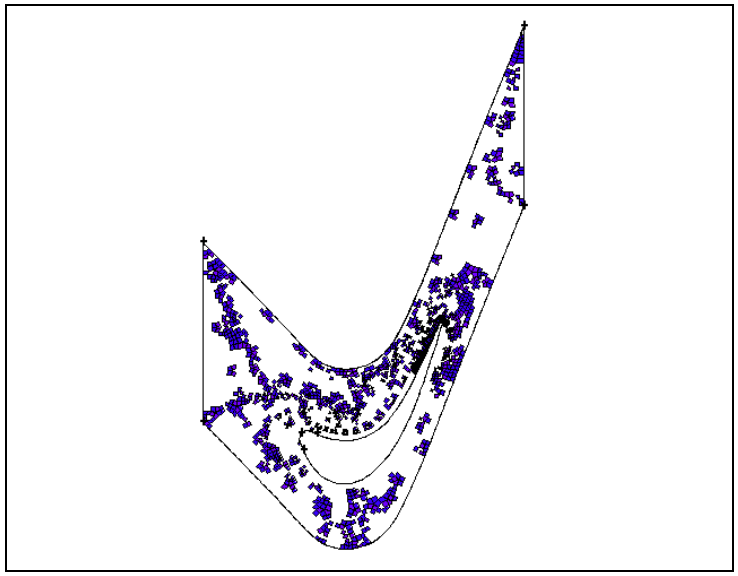


Figure 8-14: Quadrilateral mesh elements—EquiSize Skew = 0.1–0.3

Figure 8-15 displays a zoomed view of the mesh in the region surrounding the blade tip.

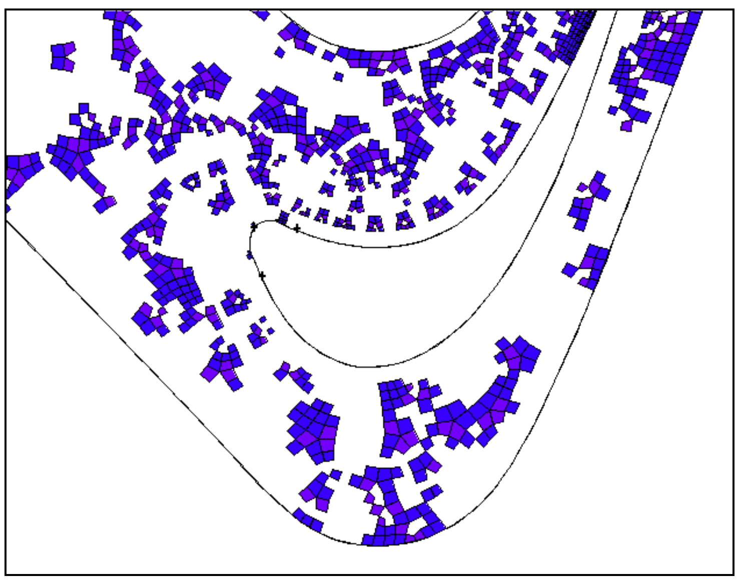


Figure 8-15: Quadrilateral mesh elements—zoomed view near blade tip

d) Select the **Off** option and click **Apply** to turn off the cascade turbo view before specifying zone types.

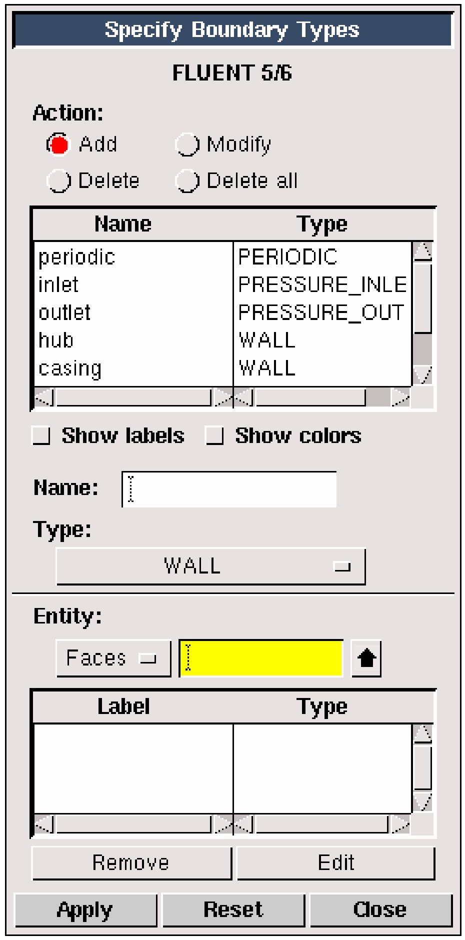
Step 12: Specify Zone Types

You can use the **Specify Boundary Types** *command to apply solver-specific boundary zone specifications to surfaces of the turbo volume. For some solver options, including* **Fluent 5/6**, GAMBIT *automatically assigns such boundary zone specifications.*

1. Check the automatically applied boundary zone types.

ZONES  → **SPECIFY BOUNDARY TYPES** 

This command sequence opens the **Specify Boundary Types** *form.*



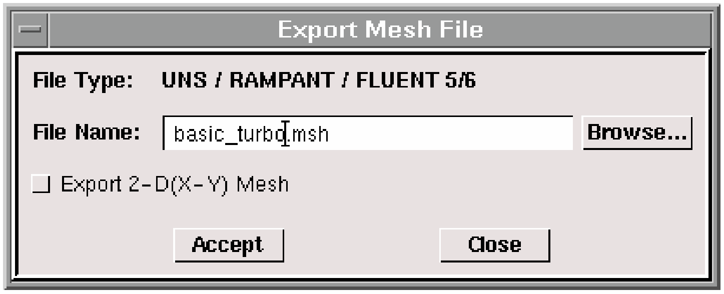
Step 13: Export the Mesh and Exit GAMBIT

1. Export a mesh file.

   a) Open the **Export Mesh File** form.

   File → Export → Mesh...

   This command sequence opens the **Export Mesh File** *form.*



   i. Enter the **File Name** for the file to be exported—for example, the file name "`basic_turbo.msh`".

   ii. Click **Accept**.

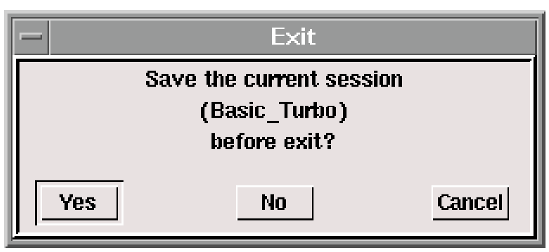
   GAMBIT *writes the mesh file to your working directory.*

2. Save the GAMBIT session and exit GAMBIT.

   a) Select **Exit** from the **File** menu.

   File → Exit

   This action opens the **Exit** *form.*



   b) Click **Yes** to save the current session and exit GAMBIT.

8.5 Summary

This tutorial demonstrates the use of the basic turbo modeling operations available in GAMBIT. The edge data that describes the geometry of the turbo profile was imported from a turbo data file, and the completed turbo profile was adjusted to affect the shape of the turbo volume. The turbo volume was divided into two spanwise sections, each of which was meshed by means of a Cooper scheme that employed the common face between them as a source face. Three-dimensional boundary layers were applied to the surfaces of the turbo blade to ensure a high-quality mesh in proximity to the turbo blade. Finally, the mesh examining capabilities in GAMBIT were used in conjunction with the turbo viewing capability to examine the 2-D mesh on the middle spanwise face.