GAMBIT 2.2

Tutorial Guide

September 2004

For GAMBIT Technical Support contact information, visit the Fluent, Inc. Web site at www.fluent.com.

Fluent, Incorporated
Centerra Resource Park
10 Cavendish Court
Lebanon, NH 03766

TABLE OF CONTENTS

0. USING THIS TUTORIAL GUIDE

0.1 What's in This Guide

This guide contains step-by-step examples that teach you how to use GAMBIT to create and mesh various geometries. Each example illustrates at least one new concept with respect to GAMBIT geometry creation and mesh generation.

Tutorial 1 includes explicit instructions for all steps in the geometry creation, mesh generation, and examination of a completed mesh. Its purpose is to introduce the beginning user to several basic features and operations that are available in GAMBIT. The remaining tutorials are designed for the user who has read or worked through Tutorial 1 or who is already familiar with GAMBIT. Consequently, they are not as explicit in their instructions as is Tutorial 1.

Some of the tutorials involve the importation of geometry data from existing files. Specifically, the tutorials that involve data import, and the type of data imported, are as follows:

- Tutorial 5—IGES data
- Tutorial 6—IGES data
- Tutorial 7—Turbo data
- Tutorial 8—ACIS data
- Tutorial 9—Turbo data
- Tutorial 11—STEP data
- Tutorial 12—Direct CAD Import
- Tutorial 13—IGES data
- Tutorial 14—STEP data

The files that contain the data to be imported are located in the directory where GAMBIT is installed. (NOTE: This file is included on your installation tape or CD.)

0.2 How to Use This Guide

If you are new to GAMBIT, you should first work through Tutorial 1 in order to familiarize yourself with the GAMBIT graphical user interface (GUI) and with basic geometry creation and meshing procedures. You may then want to try a tutorial that demonstrates features that you are going to use in your application. For example, if you are planning to start from an existing geometry that requires some cleanup, you should look at Tutorial 5. Each tutorial demonstrates different GAMBIT features, so it is recommended that you do each tutorial in order to get the full benefit from this Tutorial Guide.

Note that Step 1 in Tutorials 2 through 6 requires you to select the solver to be used for the CFD calculation. In many cases, you could select a different solver than the one used in the tutorial. The solver selection is included in the tutorials to demonstrate the process of selecting a solver. It also illustrates that the choice of solver dictates the options available in various forms (for example, the boundary types available in the **Specify Boundary Types** form).

0.3 Font Conventions

The following font conventions are used throughout this manual to represent user input data, the titles of forms and command buttons, options, and the names of modeling objects.

Font	*Description*	*Example(s)*
`Courier`	Command line arguments, file names, and other user input from the keyboard	`volume create sphere` `GAMBIT.ini`
Arial Narrow, Bold	Titles of buttons, selectors, fields, and forms as they appear in the graphical user interface	**Model** **Volume** **Vertex**
Arial Narrow	Titles of options and commands	Interval size Lower topology
Arial Narrow, Italic	Names of GAMBIT topological entities and coordinate systems	*edge.1* *vertex.3*

0.4 Using the Mouse

The GAMBIT GUI is designed for use with a three-button mouse. The function associated with each mouse button varies according to whether the mouse is operating on menus and forms, or in the graphics window. Some graphics-window mouse operations involve keyboard keys in conjunction with the mouse.

0.4.1 Menus and Forms

Mouse operations for GAMBIT menus and forms require only the left and right mouse buttons and do not involve any keyboard key operations. Most of the mouse operations performed on GAMBIT GUI menus and forms require only the left mouse button. The right mouse button is used to open menus related to command buttons on the toolpads. On some forms that include a text window, the right mouse button opens a hidden menu of options such as that described in "Using a Pick List Form" in Section 3.2.8 of the GAMBIT User's Guide.

0.4.2 Graphics Window

There are three general types of GAMBIT GUI graphics-window mouse operations:

- Display
- Task
- Vertex creation

Display operations allow you to directly manipulate the appearance of the model in any of the enabled graphics-window quadrants. *Task* operations allow you to specify topological entities and to execute geometry and meshing operations. The *vertex creation* operation allows you to create vertices on any displayed coordinate-system grid. (For further information on these operations, see Section 3.3.2 of the GAMBIT User's Guide.)

Display Operations

GAMBIT graphics-window display operations employ all three mouse buttons as well as the *Ctrl* keyboard key.

Keyboard Key/ Mouse Button	*Mouse Motion*	*Description*
Left-click	Left-drag the cursor in any direction.	Rotates the model
Middle-click	Middle-drag the cursor in any direction.	Translates the model
Right-click	Right-drag the cursor vertically.	Zooms the model in or out
Right-click	Right-drag the cursor horizontally.	Rotates the view of the model about the center of the graphics window
Ctrl-left-click	Left-drag the cursor diagonally.	Enlarges the model, retaining the model proportions. When you release the mouse button, GAMBIT enlarges the display
Double-middle-click		Displays the model as shown immediately before the current view

Task Operations

GAMBIT graphics window task operations employ all three mouse buttons in conjunction with the *Shift* key to allow you to pick entities and to execute actions related to GAMBIT forms. There are two types of task operations:

- Picking entities
- Executing actions

Picking Entities

Many GAMBIT modeling and meshing operations require you to specify one or more entities to which the operation applies. There are two ways to specify an entity for a GAMBIT operation:

- Input the entity name in the appropriate list box on the specification form or select it from the appropriate pick list.
- Use the mouse to “pick” the entity from the model as displayed in the graphics window.

When you use the mouse to pick an entity from the model that is displayed in the graphics window, GAMBIT inserts the entity name in the currently active pick list as if you had specified its name on the currently open specification form.

GAMBIT provides two types of entity-picking operations, each of which involves the *Shift* key. Throughout this guide, you will see expressions such as “*Shift*-left-click,” which indicates that you should press and hold the *Shift* key while clicking the left mouse button. The two entity picking operations are as follows:

Operation	*Description*
Shift-left-click	Highlights the entity in the graphics window and includes it in the currently active pick list.
Shift-middle-click	Toggles between adjoining multiple entities of a given type.

To select a *group* of objects, *Shift*-left-drag the mouse to create a box around the objects to be selected. The specific objects chosen depend on the direction of mouse-pointer movement when the box is created. Specifically:

- If you *Shift*-left-drag the mouse pointer toward the lower part of the graphics window when creating the box, GAMBIT selects all objects touched by the box.
- If you *Shift*-left-drag the mouse pointer toward the upper part of the graphics window when creating the box, GAMBIT selects only those objects that are completely enclosed in the box.

Executing Actions

When you *Shift*-right-click the mouse in the graphics window, GAMBIT accepts the selection of an entity and moves the focus to the next pick list in the form. If the current pick list is the last one in the form, *Shift*-right-click executes the operation associated with the currently open form. In this case, the *Shift*-right-click operation is equivalent to the act of clicking **Apply** on the bottom of the form.

0.5 GUI Components

GAMBIT allows you to construct and mesh models by means of its graphical user interface (GUI), which is designed to be mouse-driven. The GAMBIT GUI (Figure 0-1) consists of eight components, each of which serves a separate purpose with respect to the creating and meshing of a model. The following sections briefly describe the GUI components.

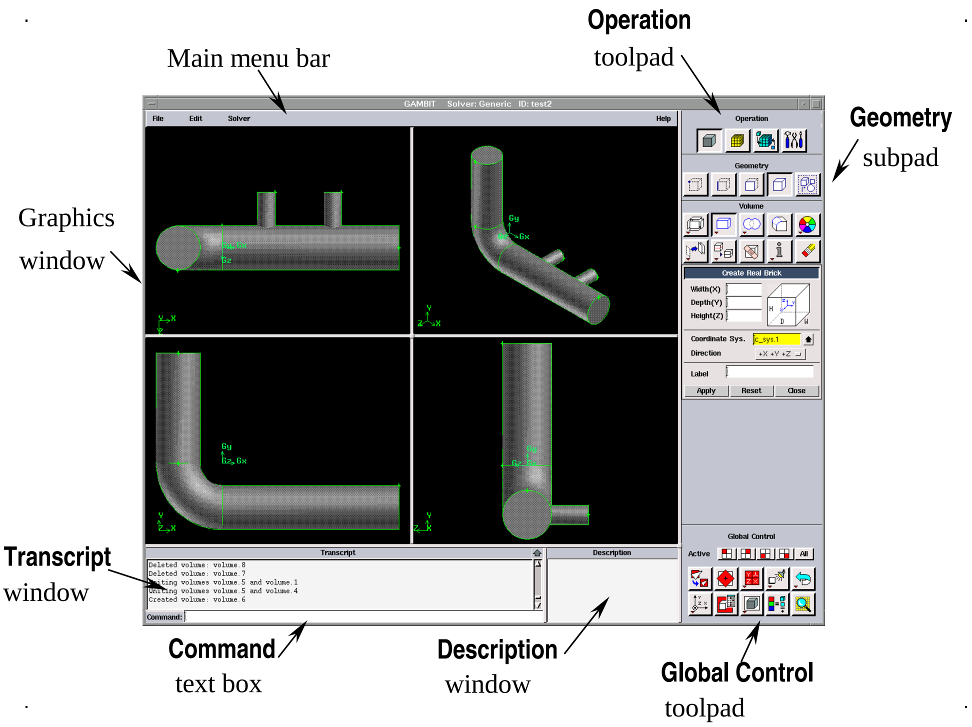


Figure 0-1: The GAMBIT GUI

0.5.1 Graphics Window

The *graphics window* is the region of the GUI in which the model is displayed. It is located in the upper left portion of the GUI and occupies most of the screen in the default layout. Chapter 3 of the GAMBIT User's Guide presents a more detailed description of the graphics window.

0.5.2 Main Menu Bar

The *main menu bar* is located at the top of the GUI, directly above the graphics window. It contains four menu items. Each of the items is associated with its own menu of commands that allow you to perform various GAMBIT operations. To open the menu associated with any item, left-click the item name (for example, **File**).

Chapter 4 of the GAMBIT User's Guide presents detailed descriptions of the menu items, as well as the commands available on each associated menu.

0.5.3 Operation Toolpad

The ***Operation*** *toolpad* is located in the upper right portion of the GUI. It consists of a field of command buttons, each of which performs a specific function associated with the process of creating and meshing a model.

Within the **Operation** toolpad, command buttons are grouped according to their hierarchy and purpose in the overall scheme of creating and meshing the model. The topmost group constitutes the *main pad*. All other command button groups constitute *subpads*.

Subpads

When you click a main-pad command button, GAMBIT opens an associated *subpad*. For example, if you click the **GEOMETRY** command button on the main pad, GAMBIT opens the **Geometry** subpad.

Each subpad contains command buttons that perform operations related to the overall purpose of the subpad. For example, the **Geometry** subpad contains command buttons that allow you to perform operations related to the creation and refinement of model geometry.

Some of the command buttons located on subpads open related subpads of their own. For example, when you click the **VOLUME** command button on the **Geometry** subpad, GAMBIT opens the **Geometry**/**Volume** subpad.

Each command button on the **Geometry**/**Volume** subpad is associated with a *specification form* that allows you to specify parameters related to the function indicated on the button.

Toolpad Command Buttons

Toolpad command buttons allow you to execute program commands related to building, meshing, or viewing the model and working with the GUI. Some toolpad command buttons cause a direct action to occur; others open specification forms.

All toolpad command buttons contain symbols representing their functions. Buttons that perform more than one function (multifunction command buttons) contain small, downward-pointing arrowheads in their lower left corners.

For complete descriptions of the GAMBIT GUI toolpad and command buttons, see Chapter 3 of the GAMBIT User’s Guide.

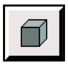
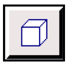
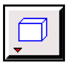
Tutorial Convention—Toolpad Command Buttons

GAMBIT geometry and meshing procedures operate by means of specification forms. Each specification form is associated with a unique combination of GAMBIT toolpad command buttons.

This tutorial guide employs the following convention to indicate the command button combination associated with any specification form:

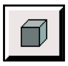
L1 → ***L2*** → ***L3***

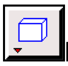
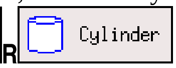
where ***L1*** represents the main-pad command button, and ***L2*** and ***L3*** represent the second- and third-level subpad command buttons, respectively. For example, the command button combination associated with the **Create Real Brick** form is as follows:

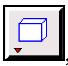
GEOMETRY  → **VOLUME**  → **CREATE VOLUME** 

Note that the toolpad choices are indicated in two ways:

- The name of the command button that appears in the **Description** window of the GAMBIT GUI
- A picture of the command button

When you see this kind of flow chart in a tutorial, you should left-click the command buttons in the order shown so that they appear depressed. A command button has a black border on its top and left-hand side when it is depressed. The **GEOMETRY** command button  at the top of the **Operation** toolpad in Figure 0-1 is an example of a depressed button. The black border is on the bottom and right-hand side when the button is not depressed; see the **MESH** command button  in Figure 0-1. Note that if a button is already depressed, you need not click that button again. In fact, clicking a selected button will unselect it.

Toolpad choices that require pressing the right mouse button are indicated by an **R** to the right of the corresponding command button icon, followed by the icon to select from the list of available functions. For example, **R**

indicates that you should right-click the **CREATE VOLUME** command button , then choose the **CREATE REAL CYLINDER** option 

from the resulting list. **CREATE REAL CYLINDER** is the text that is written in the **Description** window when you hold the mouse cursor over the 

menu item.

0.5.4 Form Field

When you click any subpad command button (except **UNDO**), GAMBIT opens an associated *specification form*. Specification forms, such as that shown in Figure 0-2, allow you to specify parameters related to modeling and meshing operations, the assignment of boundary attributes, and the creation and manipulation of GAMBIT coordinate systems and grids.

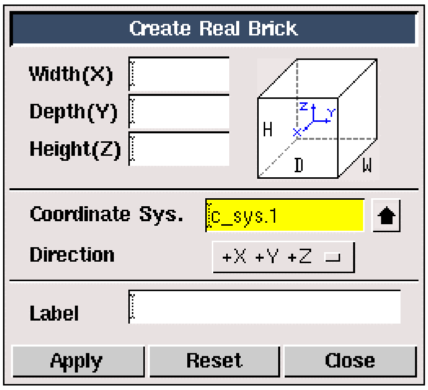


Figure 0-2: Example GAMBIT specification form

When you open a specification form, it appears in the *form field*. The form field is located at the right side of the GUI, immediately below the **Operation** toolpad.

Text boxes allow you to input alphanumeric data. They are located on forms and appear as white, indented rectangles (for example, the **Width** text box in Figure 0-2). The title of any text box appears immediately to its left. To enter data by means of a text box, left-click in the box to enable it for user input, and then input the data from the keyboard.

0.5.5 Global Control Toolpad

The ***Global Control*** *toolpad* is located at the lower right corner of the GUI. Its purpose is to allow you to control the layout and operation of the graphics window as well as the appearance of the model as displayed in any particular quadrant. Section 3.4 of the GAMBIT User's Guide describes the function and use of each button on the **Global Control** toolpad.

0.5.6 Description Window

The ***Description*** *window* is located at the bottom of the GUI, immediately to the left of the **Global Control** toolpad. The purpose of the **Description** window is to display messages describing the various GUI components, including sashes, fields, windows, and command buttons.

Messages displayed in the **Description** window describe the component of the GUI corresponding to the current location of the mouse pointer. As you move the mouse pointer across the screen, GAMBIT updates the **Description** window message to reflect the change in the location of the pointer.

0.5.7 Transcript Window and Command Text Box

The ***Transcript*** *window* is located in the lower left portion of the GUI. The ***Command*** *text box* is located immediately below the **Transcript** window.

The purpose of the **Transcript** window is to display a log of commands executed and messages displayed by GAMBIT during the current modeling session. The **Command** text box allows you to perform GAMBIT modeling and meshing operations by means of direct keyboard input, rather than by means of mouse operations on the GUI. See the GAMBIT Command Reference Guide for more details.

1. CREATING AND MESHING BASIC GEOMETRY

This tutorial illustrates geometry creation and mesh generation for a simple geometry using GAMBIT.

In this tutorial you will learn how to:

- Start GAMBIT
- Use the **Operation** toolpad
- Create a brick and an elliptical cylinder
- Unite two volumes
- Manipulate the display of your model
- Mesh a volume
- Examine the quality of the mesh
- Save the session and exit GAMBIT

1.1 Prerequisites

This tutorial assumes you have no prior experience of working with GAMBIT. You should, however, read Chapter 0, “Using This Tutorial Guide,” to familiarize yourself with the GAMBIT interface and with conventions used in the tutorial instructions.

1.2 Problem Description

The model consists of an intersecting brick and elliptical cylinder. The basic geometry is shown schematically in Figure 1-1.

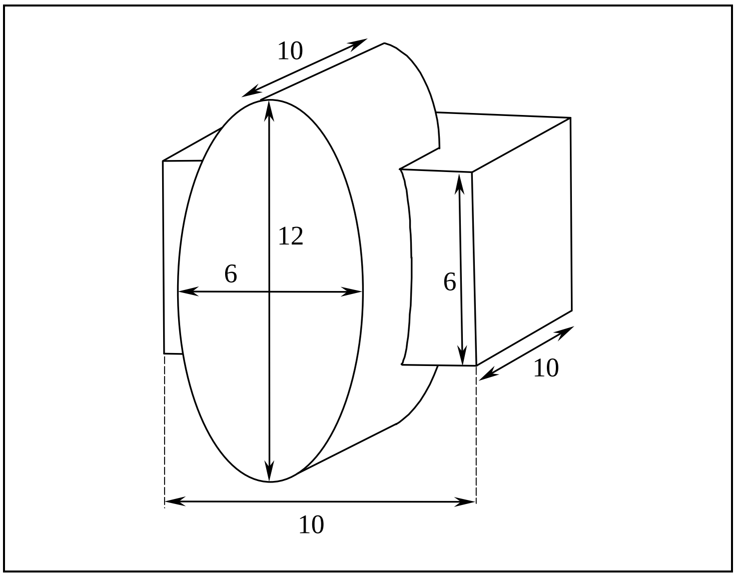


Figure 1-1: Problem specification

1.3 Strategy

This first tutorial illustrates some of the basic operations for generating a mesh using GAMBIT. In particular, it demonstrates:

- How to build the geometry easily using the “top-down” solid modeling approach
- How to create a hexahedral mesh automatically

The “top-down” approach means that you will construct the geometry by creating volumes (bricks, cylinders, etc.) and then manipulating them through Boolean operations (unite, subtract, etc.). In this way, you can quickly build complicated shapes without first creating the underlying vertices, edges, and faces.

Once you have built a valid geometry model, you can directly and (in many cases) automatically create the mesh. In this example, the Cooper meshing algorithm is used to automatically create an unstructured, hexahedral mesh. More complicated geometries may require some manual decomposition before you can create the mesh; this is demonstrated in subsequent tutorials.

The steps you will follow in this tutorial are listed below:

- Create two volumes (a brick and an elliptical cylinder).
- Unite the two volumes.
- Automatically generate the mesh.
- Examine the quality of the resulting mesh.

To keep this introductory tutorial short and simple, certain steps that you would normally follow have been omitted:

- Adjusting the distribution of nodes on individual edges of the geometry
- Setting continuum types (for example, identifying which mesh zones are fluid and which are solid) and boundary types

These details, as well as others, are covered in subsequent tutorials.

1.4 Procedure

Type

```
gambit -id basgeom
```

to start GAMBIT.

> *This command opens the* GAMBIT *graphical user interface (GUI). (See Figure 1-2.)* GAMBIT *uses the name you specify (in this example,* `basgeom`*) as a prefix to all files it creates: for example,* `basgeom.jou`.

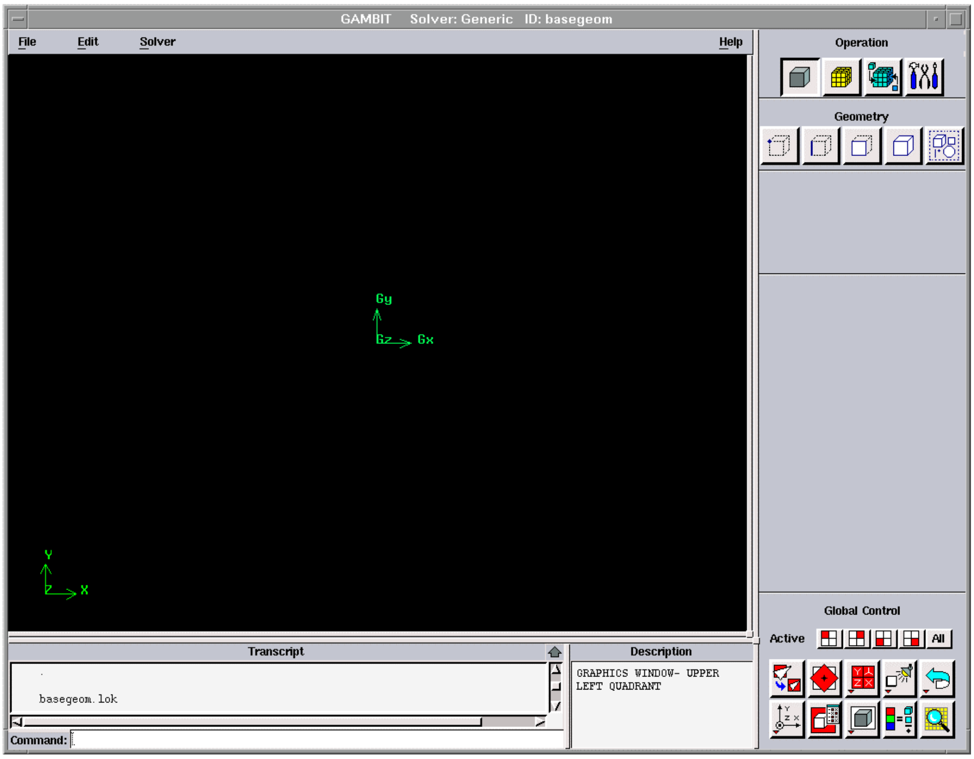
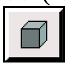


Figure 1-2: The GAMBIT graphical user interface (GUI)

Step 1: Create a Brick

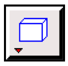
1. Create a brick by doing the following:

   a) In the **Operation** toolpad (located in the top right corner of the GAMBIT GUI), select the **GEOMETRY**  command button by clicking on it with the left mouse button. If the **Geometry** subpad does not appear when you select the **GEOMETRY** command button, click it again.

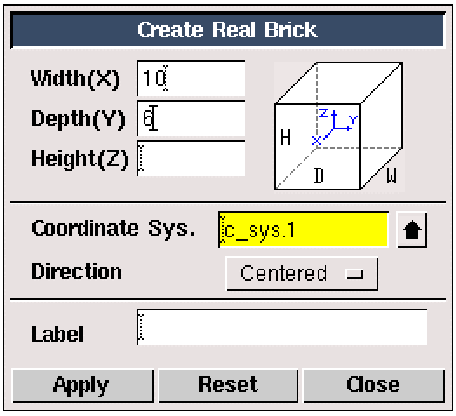
      The name of a command button is displayed in the **Description** *window at the bottom of the* GAMBIT *GUI when you hold the mouse cursor over the command button. The* **GEOMETRY** *command button will appear depressed when it is selected. Selecting the* **GEOMETRY** *command button opens the* **Geometry** *subpad. Note that when you first start* GAMBIT*, the* **GEOMETRY** *command button is selected by default.*

   b) Use the left mouse button to select the **VOLUME**  command button in the **Geometry** subpad.

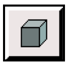
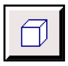
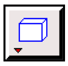
      Again, this command button will be depressed when selected. Selecting this command button opens the **Geometry/Volume** *subpad.*

   c) Use the left mouse button to select the **CREATE VOLUME**  command button in the **Geometry/Volume** subpad.

      This command sequence opens the **Create Real Brick** *form.*



The above description of selecting command buttons can be shortened to the following:

GEOMETRY  → **VOLUME**  → **CREATE VOLUME** 

The selection of the command buttons will be represented using this method for the remainder of this tutorial, and in all subsequent tutorials.

d) Left-click in the text entry box to the right of **Width** in the **Create Real Brick** form, and enter a value of `10` for the **Width** of the brick.

e) Use the *Tab* key on the keyboard to move to the **Depth** text entry box, and enter `6` for the **Depth** of the brick.

The text entry box for **Height** *can be left blank;* GAMBIT *will set this value to be the same value as the* **Width** *by default.*

f) Select Centered from the option menu to the right of **Direction**.

NOTE: When you first open the **Create Real Brick** *form, the* Centered *option is selected by default.*

i) Hold down the left mouse button on the option button to the right of **Direction** until the option menu appears.

ii) Select Centered from the list.

g) Click **Apply**.

A message appears in the **Transcript** *window at the bottom left of the* GAMBIT *GUI to indicate that a volume, called* volume.1*, was created. The volume will be visible in the graphics window, as shown in Figure 1-3.*

If you make a mistake at any point in the geometry creation process, you can use the **UNDO** *command button*  *to undo multiple levels of geometry creation. At this point, you have only performed one operation, so you can only undo one operation.*

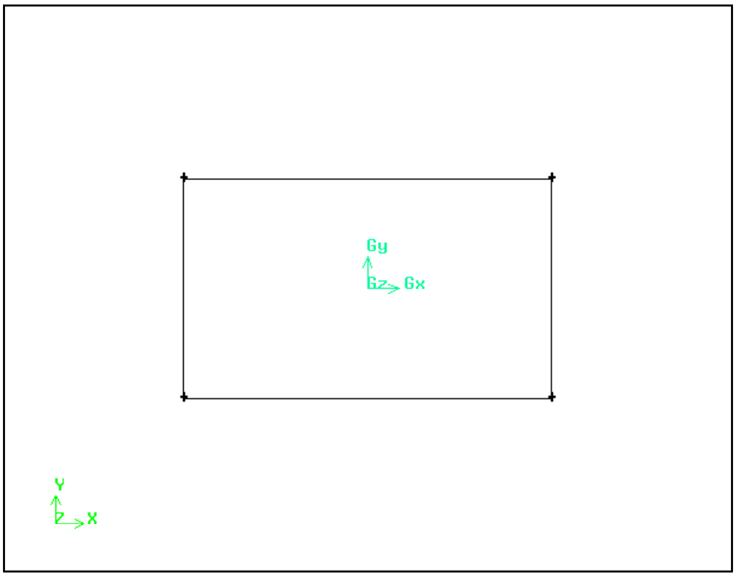


Figure 1-3: Rectangular brick volume (side view)

Step 2: Create an Elliptical Cylinder

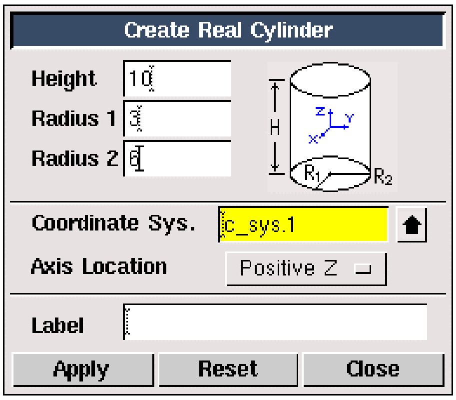
1. Create an elliptical cylinder.

   a) Hold down the *right* mouse button while the cursor is on the **CREATE VOLUME** command button.

   b) Select the **CREATE REAL CYLINDER** option Cylinder from the resulting menu.

   ! **CREATE REAL CYLINDER** *is the text that is written in the* **Description** *window when you hold the mouse cursor over the* Cylinder *menu item.*

   This command sequence opens the **Create Real Cylinder** *form.*



The above method of selecting command buttons can be shortened to the following:

GEOMETRY → **VOLUME** → **CREATE VOLUME** **R** Cylinder

where **R** indicates a toolpad choice using the right mouse button.

   c) Enter a **Height** of 10.

   d) Enter a value of 3 for **Radius 1**.

   e) Enter a value of 6 for **Radius 2**.

f) Retain the default **Axis Location** of Positive Z.

g) Click **Apply**.

The brick and elliptical cylinder are shown in Figure 1-4.

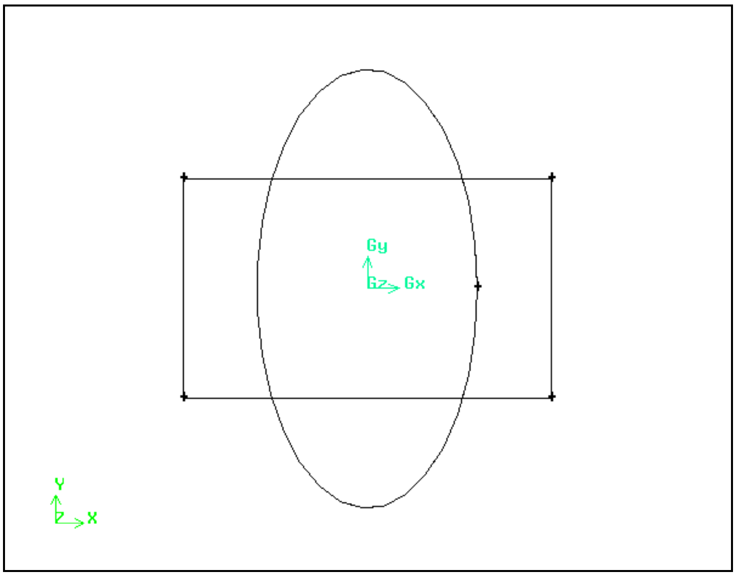


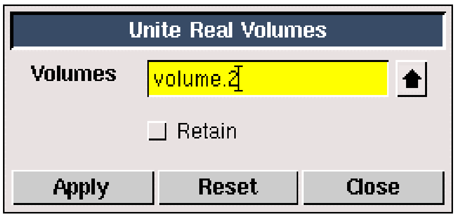
Figure 1-4: Brick and elliptical cylinder

Step 3: Unite the Two Volumes

1. Unite the brick and elliptical cylinder into one volume.

GEOMETRY → **VOLUME** → **BOOLEAN OPERATIONS**

This command sequence opens the **Unite Real Volumes** *form.*



Notice that the **Volumes** *list box is yellow in the* **Unite Real Volumes** *form at this point. The yellow color indicates that this is the active field in the form, and any volume selected will be entered into this box on the form.*

a) Hold down the *Shift* key on the keyboard and select the brick by clicking on one of its edges in the graphics window using the left mouse button.

! *The Shift key must always be held down when selecting entities in the graphics window using the left mouse button. This operation will be referred to as Shift*-left-click *in all further steps.*

*The brick will appear red in the graphics window and its name (*volume.1*) will appear in the* **Volumes** *list box in the* **Unite Real Volumes** *form.*

b) *Shift*-left-click the elliptical cylinder in the graphics window.

c) Click **Apply** to accept the selection and unite the elliptical cylinder and brick.

! *Alternatively, you could continue to hold down the Shift key and click the right mouse button in the graphics window to accept the selection of the volumes. This method allows you to rapidly accept selections and apply operations with minimal movement of the mouse.*

! *The Shift key must always be held down when clicking the right-mouse button to accept the selection of entities in the graphics window. This operation is referred to as Shift*-right-click*.*

The volume is shown in Figure 1-5. You can rotate the display (as shown in Figure 1-5) by holding down the left mouse button in the graphics window and moving the mouse to the left. More information on manipulating the graphics display is given in the next step.

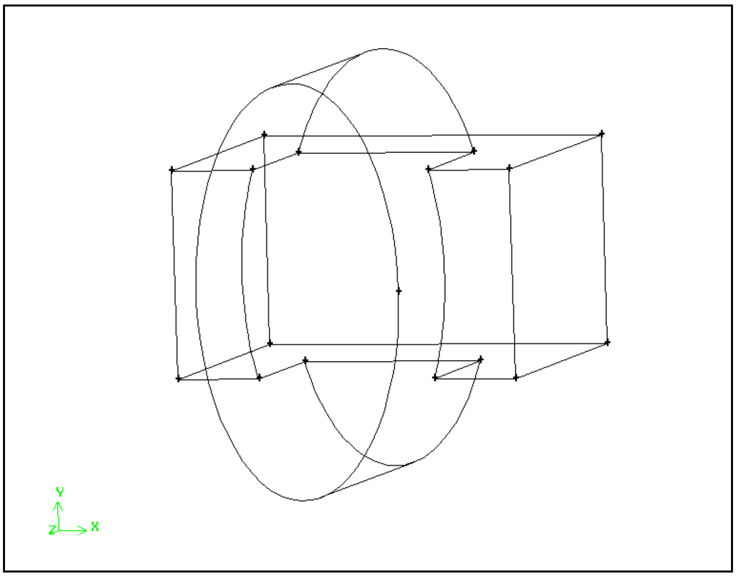


Figure 1-5: Brick and elliptical cylinder united into one volume

Step 4: Manipulate the Display

1. Zoom out from the current view by holding down the right mouse button in the graphics window and pushing the mouse away from you.

2. Rotate the view around the screen center by holding down the right mouse button and moving the mouse from side to side.

3. Rotate the view in free-form mode by holding down the left mouse button and moving the mouse.

4. Translate the display by holding down the middle mouse button and moving the mouse.

5. Divide the graphics window into four quadrants by clicking the **SELECT PRESET CONFIGURATION**  command button in the **Global Control** toolpad.

   GAMBIT *divides the graphics window into four quadrants and applies a different orientation to the model in each of the four quadrants. Each view of the graphics window can be manipulated independently. All changes to the model appear in all portions of the graphics window, unless you disable one or more quadrants.*

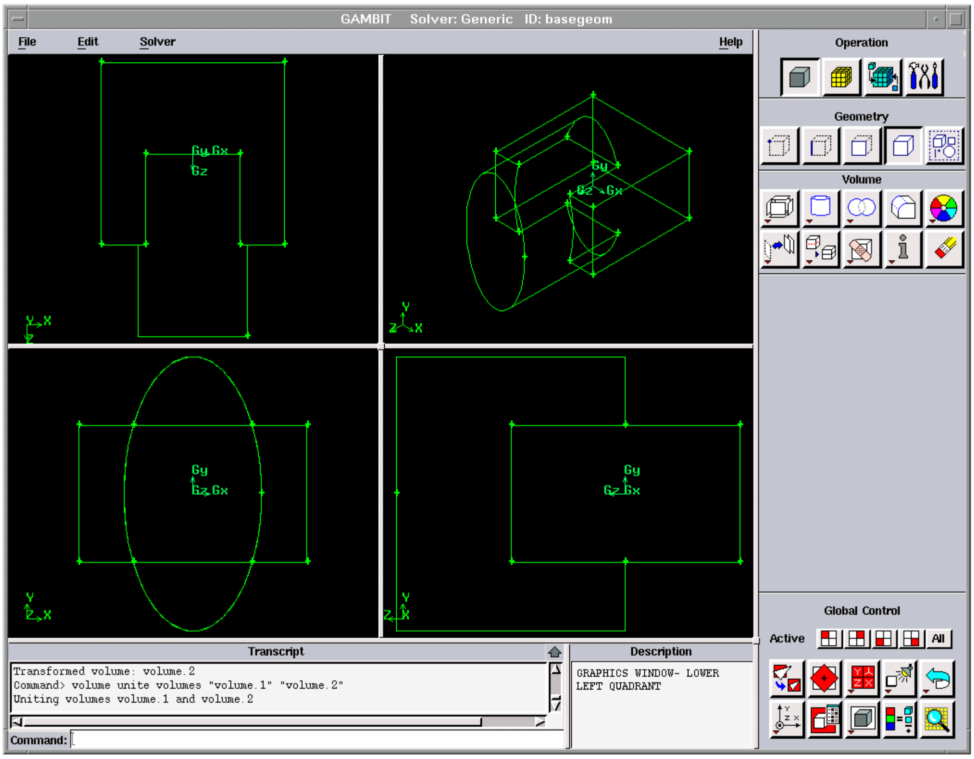
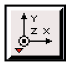


Figure 1-6: GAMBIT GUI—four graphics-window quadrants

6. Restore a single display of the model.

   a) Use the left mouse button to select the graphics-window "sash anchor"—the small gray box in the center of the graphics window.

   b) Use the mouse to drag the sash anchor to the bottom right corner of the graphics window.



7. Restore the front view of the model by left-clicking the **ORIENT MODEL** command button in the **Global Control** toolpad.

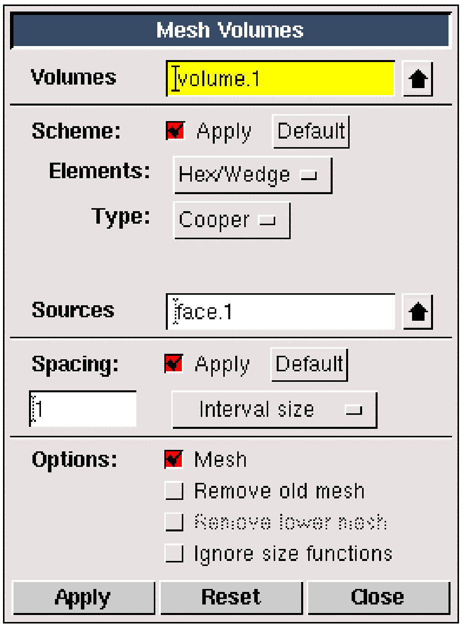
8. Scale the model to fit the graphics window by clicking the **FIT TO WINDOW** command button in the **Global Control** toolpad.

Step 5: Mesh the Volume

1. Create a mesh for the volume.

   MESH → **VOLUME** → **MESH VOLUMES**

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



   a) *Shift*-left-click the volume in the graphics window.

      GAMBIT *will automatically choose the* Cooper **Scheme Type** *as the meshing tool to be used, and will use an* **Interval size** *of* 1 *(the default) under* **Spacing**. *See the* GAMBIT *Modeling Guide, Chapter 3 for details about the* Cooper *meshing tool.*

   b) Click **Apply** at the bottom of the **Mesh Volumes** form.

      This accepts the volume you selected as the one to be meshed. It also accepts the source faces (the faces whose surface meshes are to be swept through the volume to form volume elements) that GAMBIT *has chosen for the* Cooper *meshing scheme and starts the meshing. A status bar appears at the top of the* GAMBIT *GUI to indicate how much of the meshing is complete.*

The volume will be meshed as shown in Figure 1-7.

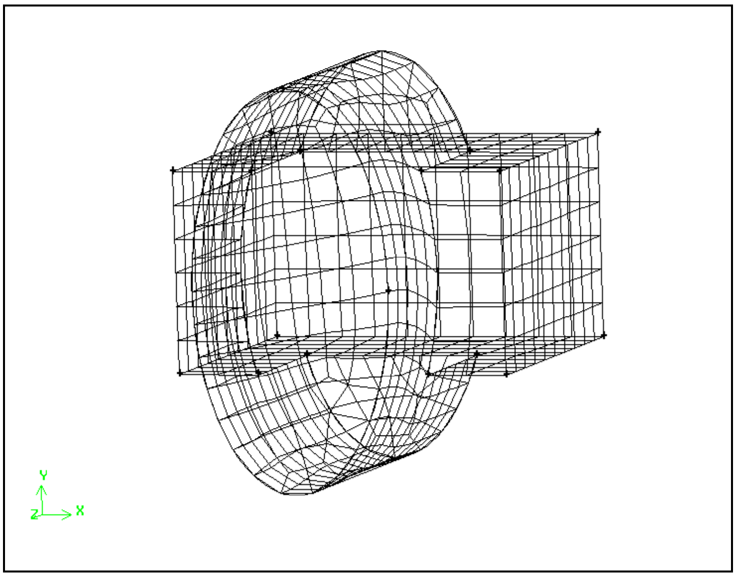


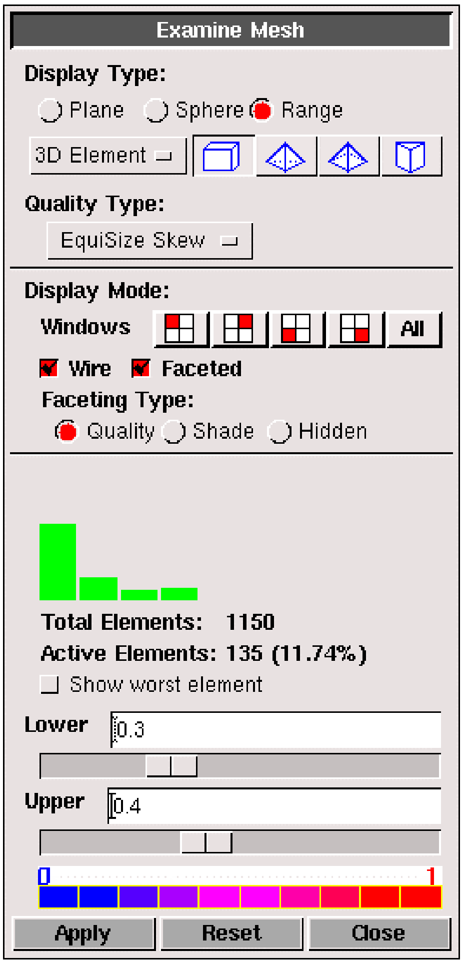
Figure 1-7: Meshed volume

Step 6: Examine the Mesh

It is important that you check the quality of the resulting mesh, because properties such as skewness can greatly affect the accuracy and robustness of the CFD solution. GAMBIT *provides several quality measures (sometimes called "metrics") with which you can assess the quality of your mesh. In the case of skewness measures such as* EquiAngle Skew *and* EquiSize Skew*, for example, smaller values are more desirable. It is also important to verify that all of the elements in your mesh have positive area/volume. You should consult the documentation for the target CFD solver for additional mesh quality guidelines.*

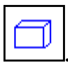
1. Select the **EXAMINE MESH**  command button at the bottom right of the **Global Control** toolpad.

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



a) Select Range under **Display Type** at the top of the **Examine Mesh** form.

A histogram appears at the bottom of the form. The histogram consists of a bar chart representing the statistical distribution of mesh elements with respect to the specified **Quality Type**. *Each vertical bar on the histogram corresponds to a unique set of upper and lower quality limits.*

The 3D Element *type selected by default at the top of the form is a brick* .

b) Select or retain EquiSize Skew from the **Quality Type** option menu.

c) Click on one of the green vertical bars in the histogram to view elements within a certain quality range.

Each element has a value of skewness between 0 and 1, where 0 represents an ideal element. The histogram is divided into 10 bars; each bar represents a 0.1 increment in the skewness value. For a good mesh, the bars on the left of the histogram will be large and those on the right will be small.

Figure 1-8 shows the view in the graphics window if you click on the fourth bar from the left on the histogram (representing cells with a skewness value between 0.3 and 0.4).

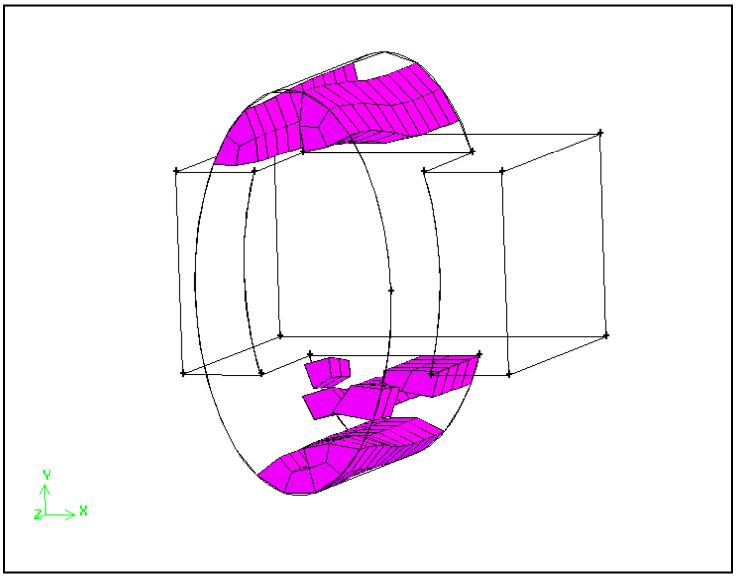


Figure 1-8: Elements of the mesh within a specified quality range

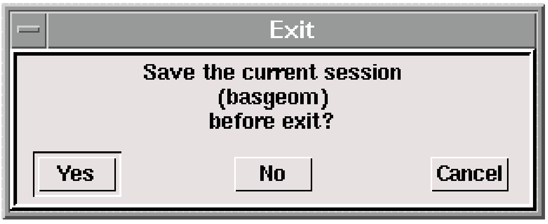
d) Move the **Upper** and **Lower** slider boxes beneath the histogram to redefine the quality range to be displayed.

Step 7: Save the Session and Exit GAMBIT

1. Save the GAMBIT session and exit GAMBIT.

   File $\longrightarrow$ **Exit**

   GAMBIT *will ask you whether you wish to save the current session before you exit.*



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

1.5 Summary

This tutorial provided a quick introduction to GAMBIT by demonstrating how to create a simple 3-D geometry using the "top-down" modeling approach. The Cooper scheme was used to automatically generate an unstructured, hexahedral mesh. For more information on the Cooper scheme, consult the GAMBIT Modeling Guide.

2. MODELING A MIXING ELBOW (2-D)

In this tutorial, you will use GAMBIT to create the geometry for a mixing elbow and then generate a mesh. The mixing elbow configuration is encountered in piping systems in power plants and process industries. It is often important to predict the flow field and temperature field in the neighborhood of the mixing region in order to properly design the location of inlet pipes.

In this tutorial you will learn how to:

- Create vertices using a grid system
- Create arcs by selecting the center of curvature and the endpoints of the arc
- Create straight edges between vertices
- Split an arc using a vertex point
- Create faces from edges
- Specify the distribution of nodes on an edge
- Create structured meshes on faces
- Set boundary types
- Prepare the mesh to be read into FLUENT 4
- Export a mesh

2.1 Prerequisites

This tutorial assumes that you have worked through Tutorial 1 and you are consequently familiar with the GAMBIT interface.

2.2 Problem Description

The problem to be considered is shown schematically in Figure 2-1. A cold fluid enters through the large pipe and a warmer fluid enters through the small pipe. The two fluids mix in the elbow.

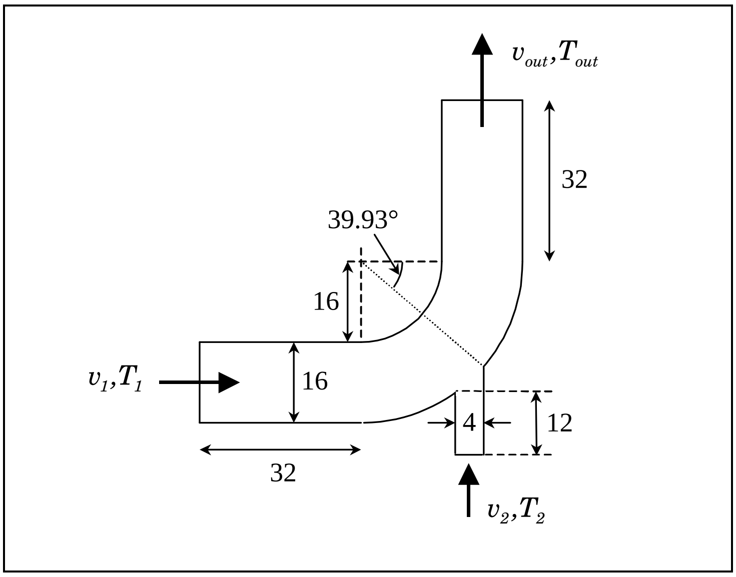


Figure 2-1: Problem specification

2.3 Strategy

In this tutorial, you will build a 2-D mesh using a "bottom-up" approach (in contrast to the "top-down" approach used in Tutorial 1). The "bottom-up" approach means that you will first create some vertices, connect the vertices to create edges, and connect the edges to make faces (in 3-D, you would stitch the faces together to create volumes). While this process by its very nature requires more steps, the result is, just as in Tutorial 1, a valid geometry that can be used to generate the mesh.

The mesh created in this tutorial is intended for use in FLUENT 4, so it must be a single block, structured mesh. However, this mesh can also be used in any of the other Fluent solvers. This type of mesh is sometimes called a mapped mesh, because each grid point has a unique I, J, K index. In order to meet this criterion, certain additional steps must be performed in GAMBIT and are illustrated in this tutorial. After creating the straight edges and arcs that comprise the geometry, you will create two faces: one for the main flow passage (the elbow) and one for the smaller inlet duct. The mesh is generated for the larger face using the Map scheme; this requires that the number of grid nodes be equal on opposite edges of the face. You will force GAMBIT to use the Map scheme to mesh the smaller face as well.

Several other features are also demonstrated in this tutorial:

- Using a background grid and "snap-to-grid" to quickly create a set of vertices.
- Using "pick lists" as an alternative to mouse clicks for picking entities.
- Specifying a non-uniform distribution of nodes on an edge.
- Setting boundary types.
- Exporting a mesh for a particular Fluent solver (FLUENT 4 in this case).

2.4 Procedure

Start GAMBIT.

Step 1: Select a Solver

1. Choose the solver you will use to run your CFD calculation by selecting the following from the main menu bar:

   Solver → **FLUENT 4**

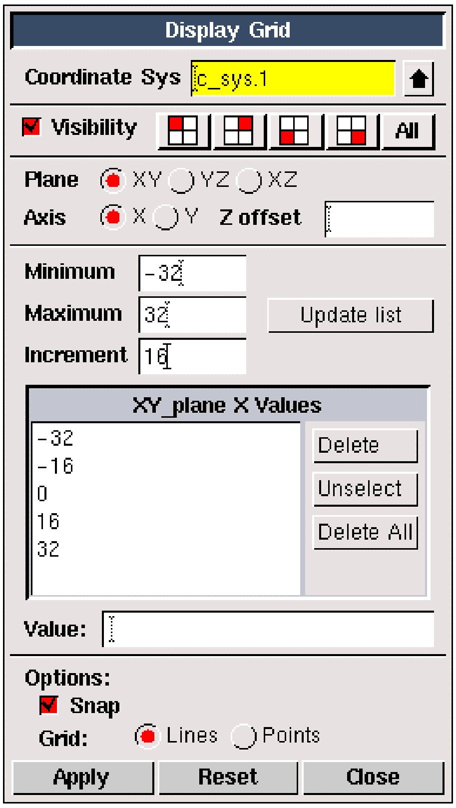
   This selects the **FLUENT 4** *solver as the one to be used for the CFD calculation. The choice of a solver dictates the options available in various forms (for example, the boundary types available in the* **Specify Boundary Types** *form). The solver currently selected is indicated at the top of the* GAMBIT *GUI.*

Step 2: Create the Initial Vertices

1. Create vertices to define the outline of the large pipe of the mixing elbow.

   TOOLS → **COORDINATE SYSTEM** → **DISPLAY GRID**

   This command sequence opens the **Display Grid** *form.*



   a) Check that **Visibility** is selected.

      This ensures that the background grid will be visible when it is created.

   b) Select X (the default) to the right of **Axis**.

   c) Enter a **Minimum** value of `-32`, a **Maximum** value of `32`, and an **Increment** of `16`.

   d) Click the Update list button.

This creates a background grid with four cells in the x direction and enters the x coordinates in the **XY_plane X Values** *list.*

e) Select Y to the right of **Axis**.

f) Enter a **Minimum** value of `–32`, a **Maximum** value of `32`, and an **Increment** of `16`.

g) Click the Update list button.

This creates a background grid with four cells in the y direction and enters the y coordinates in the **XY_plane Y Values** *list.*

h) Check that **Snap** is selected under **Options**.

The vertices you create later in this step will be "snapped" to points on the grid where the grid lines intersect.

i) Select Lines (the default) to the right of **Grid**.

The grid will be displayed using lines rather than points.

j) Click **Apply**.

GAMBIT *creates a four-by-four grid in the graphics window. To see the whole grid, you must zoom out the display (see Figure 2-2). You can zoom out the display by pressing and holding down the right mouse button while moving the cursor vertically upward in the graphics window.*

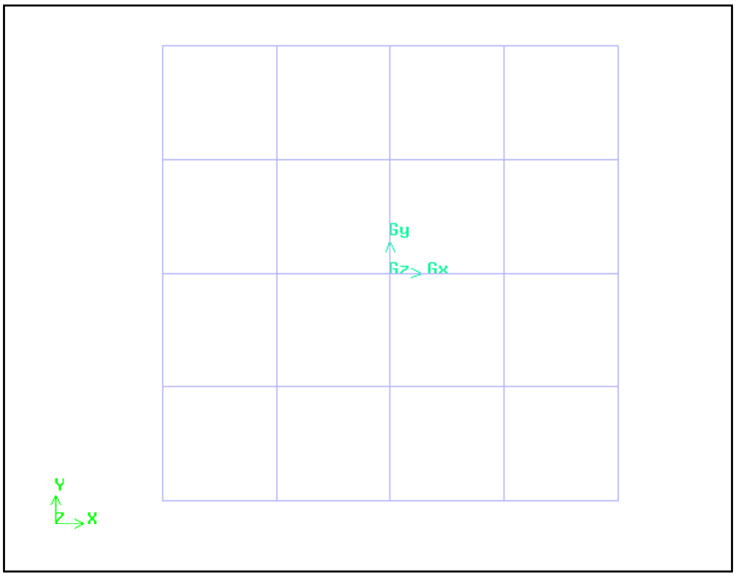


Figure 2-2: Four-by-four grid to be used for creating vertices

NOTE: You cannot use the **FIT TO WINDOW**  *command button (located on the* **Global Control** *toolpad) to zoom out the display because* GAMBIT *does not treat the grid as a model component to be fit within the graphics window.*

k) *Ctrl*-right-click the nine grid points shown in Figure 2-3.

"*Ctrl*-right-click" *indicates that you should hold down the Ctrl key on the keyboard and click on the point at which the vertex is to be created using the right mouse button.*

You can use the **UNDO** *command button*  *if you create any of the vertices incorrectly.*

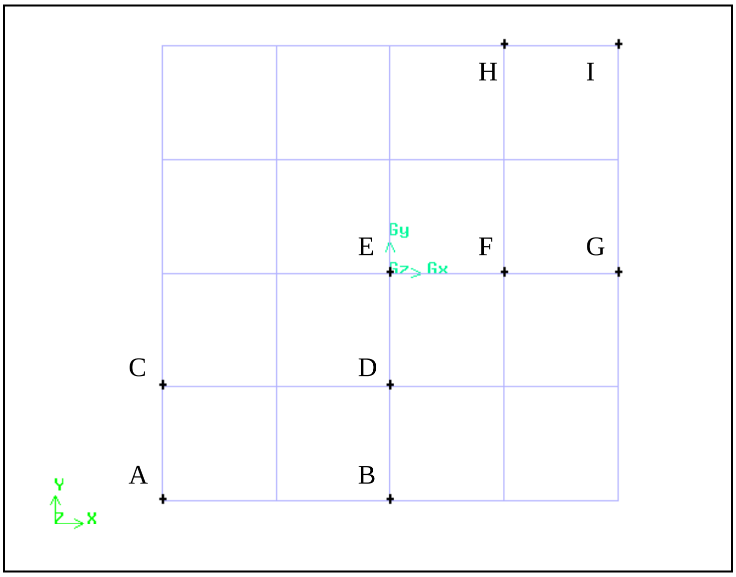


Figure 2-3: Create vertices at grid points

l) Unselect the **Visibility** check box in the **Display Grid** form and click **Apply**.

The grid will be removed from the graphics window and you will be able to clearly see the nine vertices created, as shown in Figure 2-4.

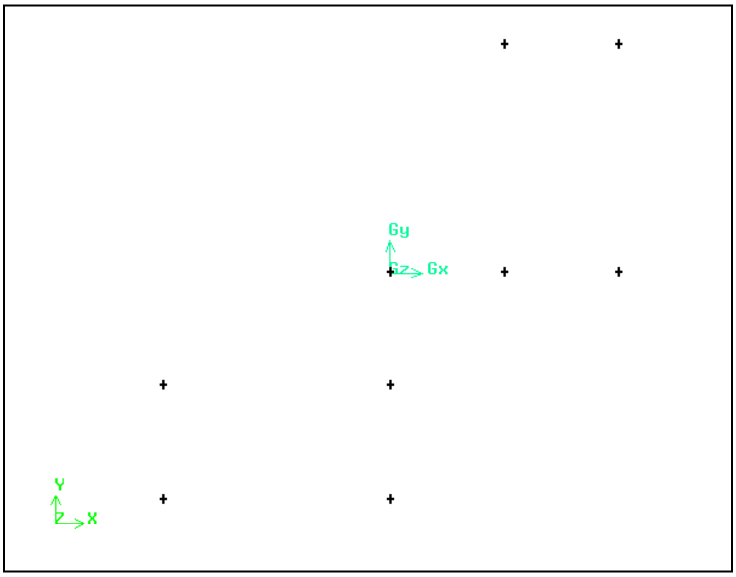


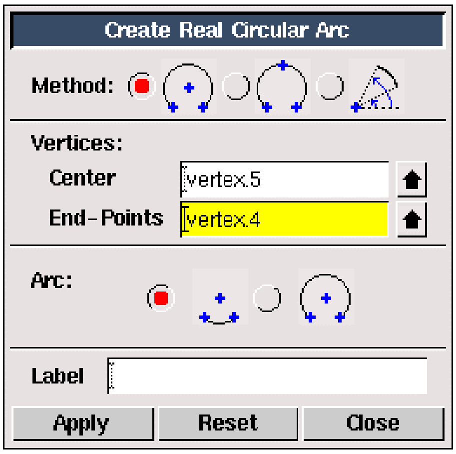
Figure 2-4: Vertices for the main pipe

Step 3: Create Arcs for the Bend of the Mixing Elbow

1. Create an arc by selecting the following command buttons in order:

   GEOMETRY → **EDGE** → **CREATE EDGE** R Arc

   This command sequence opens the **Create Real Circular Arc** *form.*



   a) Retain the default **Method**.

      Notice that the **Center** *list box is yellow in the* **Create Real Circular Arc** *form at this point. The yellow color indicates that this is the active field in the form, and any vertex selected will be entered into this box on the form.*

   b) *Shift*-left-click the vertex in the center of the graphics window (vertex E in Figure 2-5).

      The selected vertex will appear red in the graphics window and its name will appear in the **Center** *list box under* **Vertices** *in the form.*

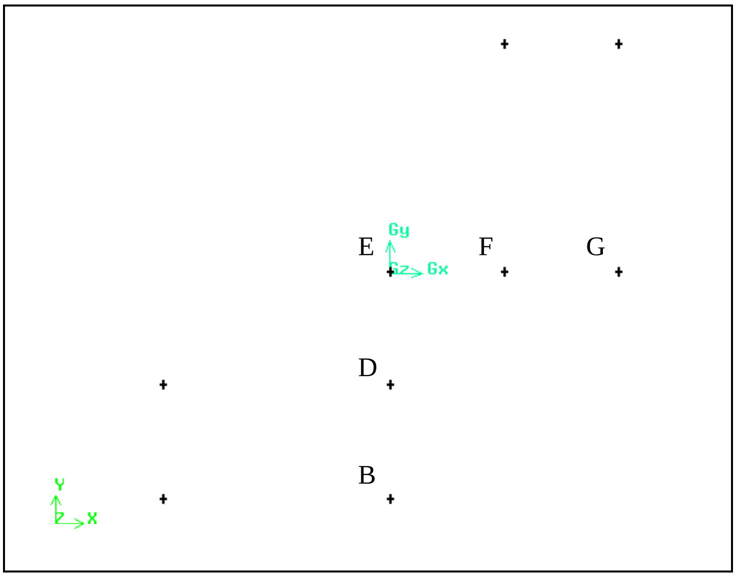


Figure 2-5: Vertices used to create arcs

c) Left-click in the list box to the right of **End-Points** to accept the selection of vertex E and make the **End-Points** list box active.

! *Alternatively, you could continue to hold down the Shift key and click the right mouse button in the graphics window to accept the selection of the vertex and move the focus to the* **End-Points** *list box.*

Note that the **End-Points** *list box is now yellow—that is, this is now the active list box, and any vertex selected will be entered in this box.*

d) *Shift*-left-click the vertex to the right of the center vertex in the graphics window (vertex F in Figure 2-5).

The vertex will turn red.

e) Select the vertex directly below the one in the center of the graphics window (vertex D in Figure 2-5).

f) Click **Apply** to accept the selected vertices and create the arc.

2. Repeat the above steps to create a second arc. The center of the arc is the vertex in the center of the graphics window (vertex E in Figure 2-5). The endpoints of the arc are the vertices to the right and below the center vertex that have not yet been selected (vertices G and B, respectively, in Figure 2-5). The arcs are shown in Figure 2-6.

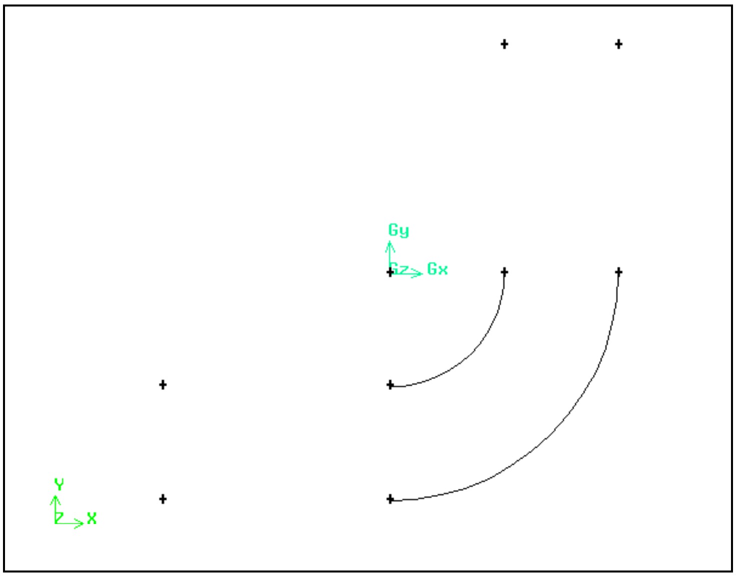


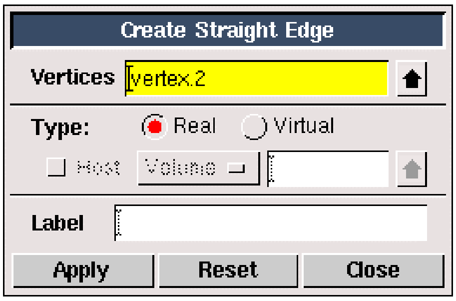
Figure 2-6: Vertices and arcs

Step 4: Create Straight Edges

1. Create straight edges for the large pipe.

   GEOMETRY → **EDGE** → **CREATE EDGE** R Straight

   This command sequence opens the **Create Straight Edge** *form.*

   a) *Shift*-left-click the left endpoint of the smaller arc (vertex D in Figure 2-7).

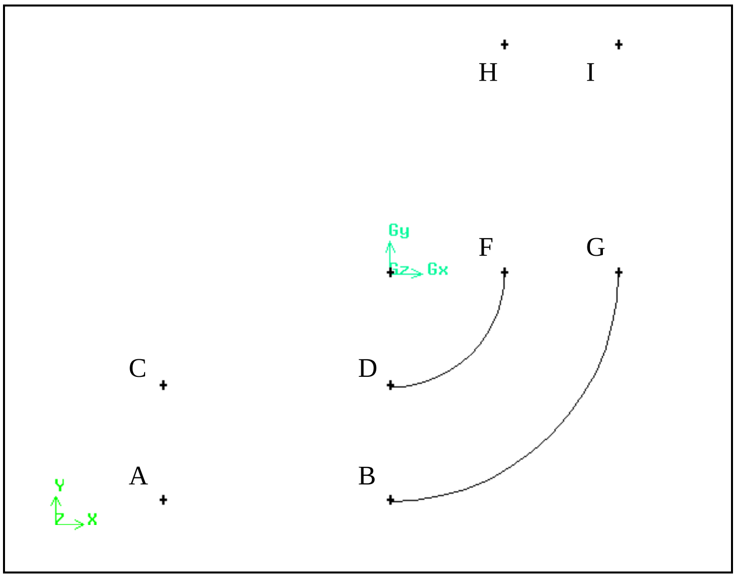
   

   Figure 2-7: Vertices used to create straight edges

   b) *Shift*-left-click the vertices marked C, A, and B in Figure 2-7, in order.

c) Click **Apply** to accept the selection of the vertices.

Three straight edges are drawn between the vertices.

d) *Shift*-left-click the vertices marked F, H, I, and G in Figure 2-7, in order.

e) Click **Apply** to accept the selection of the vertices.

The graphics window with the arcs and straight edges is shown in Figure 2-8.

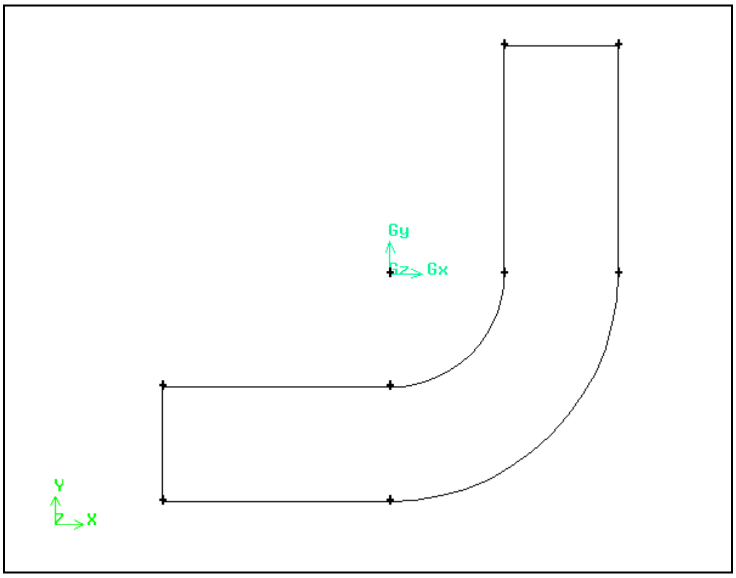


Figure 2-8: Arcs and edges

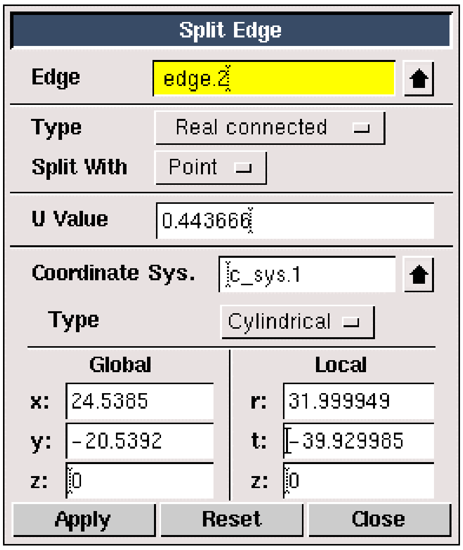
Step 5: Create the Small Pipe for the Mixing Elbow

In this step, you will create vertices on the outer radius of the bend of the mixing elbow and split the large arc into three smaller arcs. Next, you will create vertices for the inlet of the small pipe. Finally, you will create the straight edges for the small pipe.

1. Create vertices on the outer radius of the bend, and split the large arc into three sections.

   GEOMETRY → **EDGE** → **SPLIT/MERGE EDGES**

   This command sequence opens the **Split Edge** *form.*

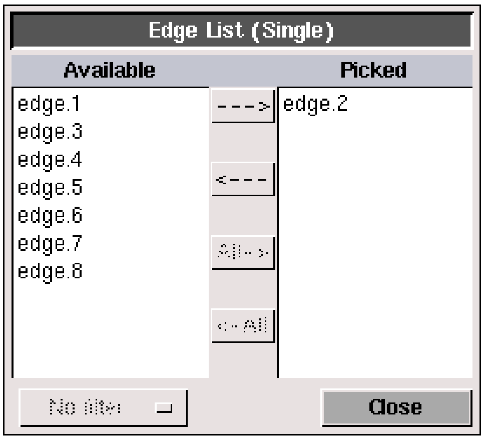


   a) Select the large arc as the edge to split by using the **Edge** pick list.

      Note that you could select the edge in the graphics window; a pick list provides an alternate way of picking an element.

      i. Left-click the black arrow to the right of the **Edge** list box in the **Split Edge** form.

This action opens the **Edge List** *form. There are two types of pick-list forms:* **Single** *and* **Multiple***. In a* **Single** *pick-list form, only one entity can be selected at a time. In a* **Multiple** *pick-list form, you can select multiple entities.*



ii. Select *edge.2* under **Available** in the **Edge List** form.

> ! *Note that the* **Available** *names may be different in your geometry, depending on the order in which you created the edges.*

iii. Click the – – –> button to pick *edge.2*.

edge.2 will be moved from the **Available** *list to the* **Picked** *list. The large arc is the edge that should be selected and shown in red in the graphics window.*

iv. Close the **Edge List** form.

*This method of selecting an entity can be used as an alternative to Shift-*left-click *in the graphics window. See the* GAMBIT *User's Guide for more information on pick lists.*

b) Select Real connected (the default) under **Type** in the **Split Edge** form.

You should select this option because the edge you selected is real geometry, not virtual geometry, and because you want the two edges created by the split to share the vertex created when GAMBIT *does the split. See the* GAMBIT *Modeling Guide for more information on real and virtual geometry.*

c) Select Point (the default) to the right of **Split With**.

You will split the edge by creating a point on the edge and then using this point to split the edge.

d) Select Cylindrical from the **Type** option menu.

You can now use cylindrical coordinates to specify where GAMBIT *should split the edge.*

e) Input a value of `–39.93` degrees next to **t** under **Local**.

This is the angle between the horizontal direction and the position of the right-hand side of the opening of the small pipe on the bend of the mixing elbow, as shown in Figure 2-1.

f) Click **Apply**.

The large arc is split into two smaller arcs and a vertex is created.

g) Use the **Edge List** form (or *Shift*-left-click in the graphics window) to select the larger of the two arcs just created (*edge.9*).

h) Input a value of `–50.07` degrees next to **t** under **Local**.

This is the angle between the horizontal direction and the position of the left-hand side of the opening of the small pipe on the bend of the mixing elbow (-90° + 39.93°), as shown in Figure 2-1.

i) Click **Apply**.

The arc is split into two parts and a second vertex is created on the bend of the mixing elbow, as shown in Figure 2-9.

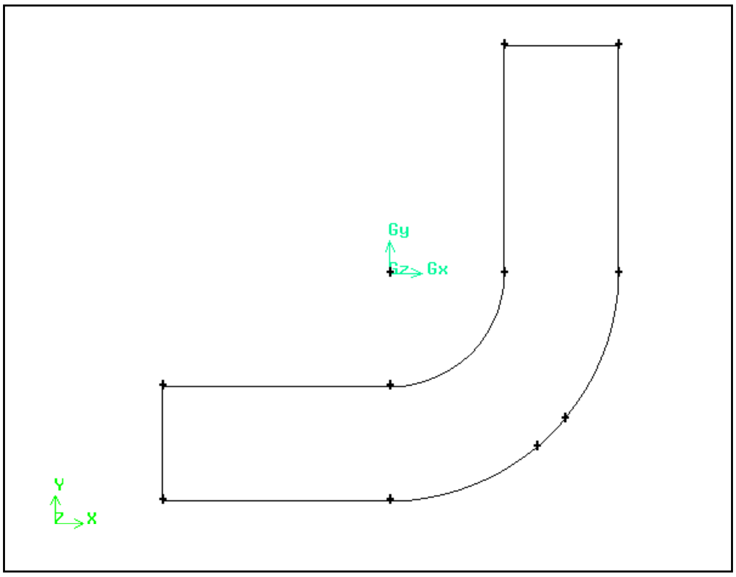
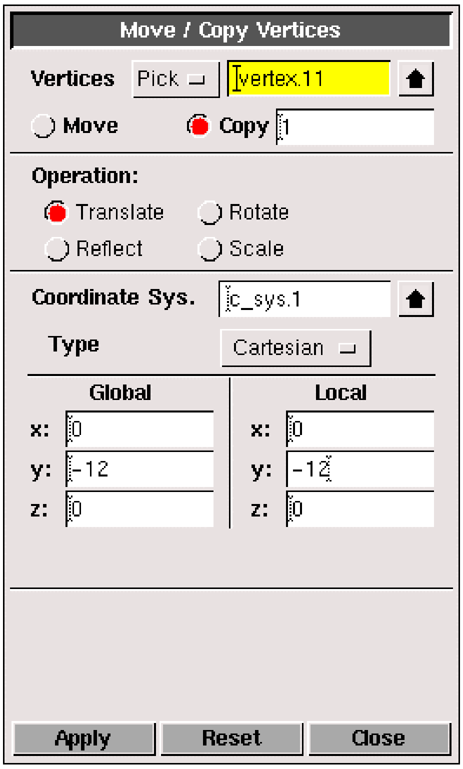


Figure 2-9: Vertices created on outer radius of mixing elbow bend

2. Create points at the small inlet.

GEOMETRY → **VERTEX** → **MOVE/COPY VERTICES**

This command sequence opens the **Move / Copy Vertices** *form.*



a) Select the second vertex created on the bend of the mixing elbow.

b) Select **Copy** under **Vertices** in the **Move / Copy Vertices** form.

c) Select Translate (the default) under **Operation**.

d) Enter the translation vector (`0, -12, 0`) under **Global** to create the new vertex at a position 12 units below the vertex you selected.

The inlet is 12 units below the second point created on the outer radius of the bend.

Note that GAMBIT *automatically fills in the values under* **Local** *as you enter values under* **Global**.

e) Click **Apply**.

f) Click the **FIT TO WINDOW**  command button at the top left of the **Global Control** toolpad to scale the model to fit into the graphics window.

g) Select the vertex just created in the graphics window.

h) Enter the translation vector (`4`, `0`, `0`) under **Global** in the **Move / Copy Vertices** form to create the new vertex at a position 4 units to the right of the vertex you selected.

i) Click **Apply**.

The vertices are shown in Figure 2-10.

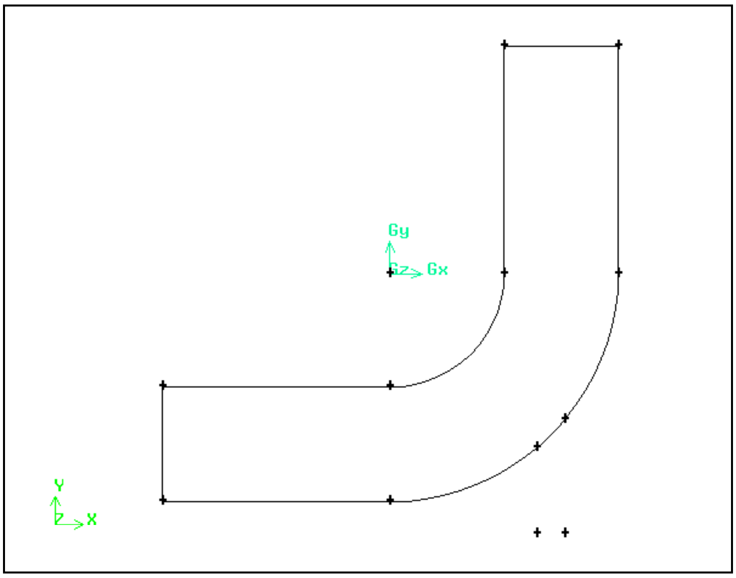
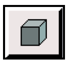
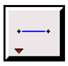
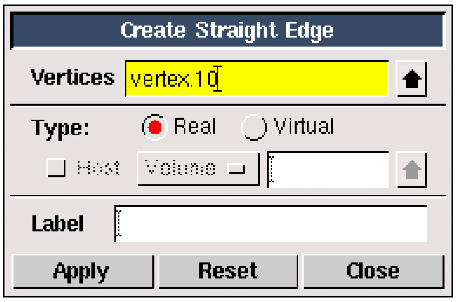


Figure 2-10: Vertices to define the small pipe

3. Create straight edges for the small pipe.

GEOMETRY  → **EDGE**  → **CREATE EDGE** 

This command sequence opens the **Create Straight Edge** *form.*



a) Create straight edges for the small pipe by selecting the vertices marked K, L, M, and J in Figure 2-11, in order, and accepting the selection.

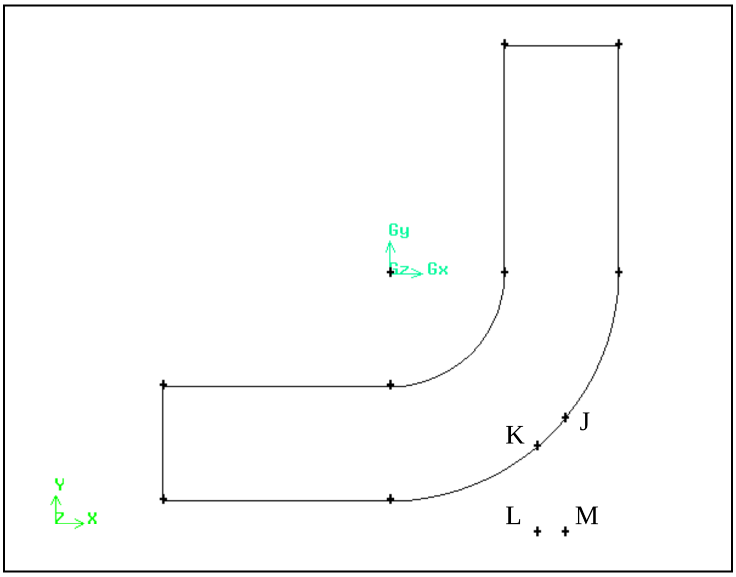


Figure 2-11: Vertices to be used to create small pipe

The small pipe is shown (with the large pipe) in Figure 2-12.

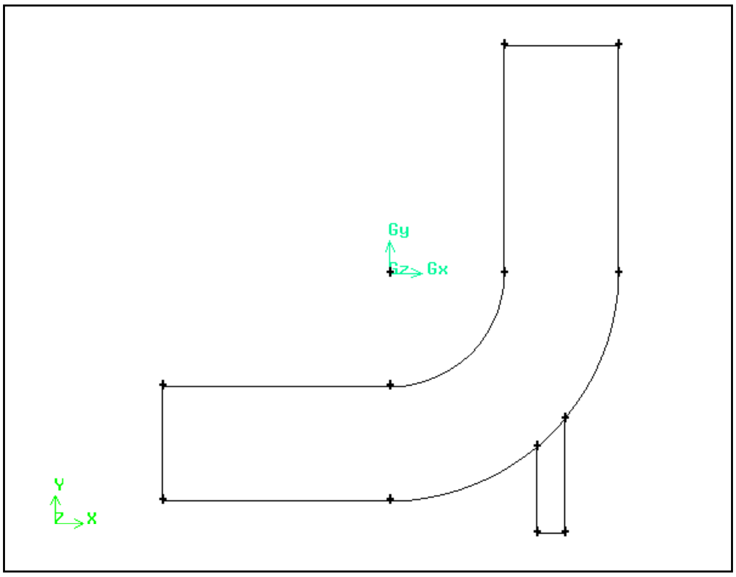


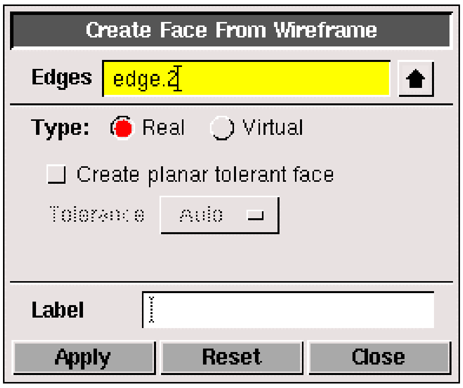
Figure 2-12: Completed small pipe

Step 6: Create Faces From Edges

1. Create a face for the large pipe.

   GEOMETRY → **FACE** → **FORM FACE**

   This command sequence opens the **Create Face From Wireframe** *form.*

   a) *Shift*-left-click each edge of the large pipe, in turn, to form a continuous loop.

   ! *The large pipe is created from the 10 edges shown in Figure 2-13. If you select an incorrect edge, click* **Reset** *in the* **Create Face From Wireframe** *form to unselect all edges, and then reselect the correct edges.*

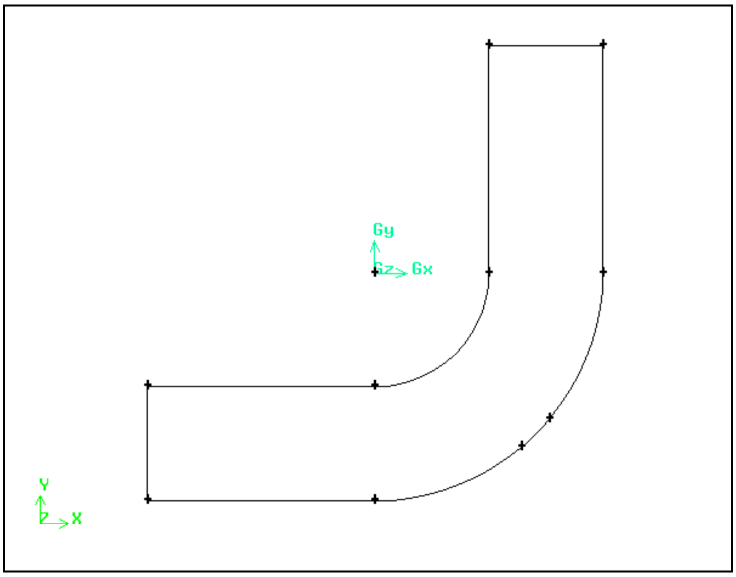


Figure 2-13: Edges used to create face for large pipe

Note that the edges must form a continuous loop, but they can be selected in any order. An alternative method to select several edges is to Shift-left-drag *a box around the edges. The box does not have to completely enclose the edges; it only needs to enclose a portion of an edge to select it. The edges will be selected when you release the mouse button.*

b) Click **Apply** to accept the selected edges and create a face.

The edges of the face will turn blue.

2. Create a face for the small pipe by selecting the four edges shown in Figure 2-14 and then accepting the selected edges.

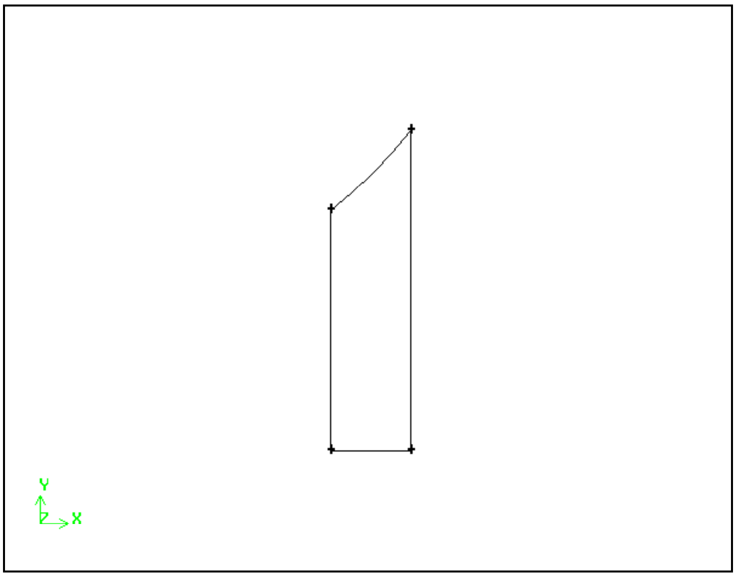


Figure 2-14: Edges used to create face for small pipe

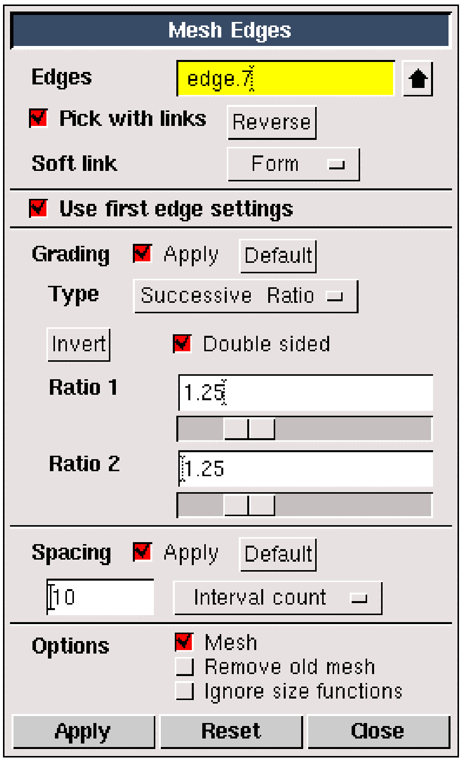
Step 7: Specify the Node Distribution

The next step is to define the grid density on the edges of the geometry. You will accomplish this graphically by selecting an edge, assigning the number of nodes, and specifying the distribution of nodes along the edge.

1. Specify the node density on the inlet and outlet of the large pipe.

   MESH → **EDGE** → **MESH EDGES**

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



   a) *Shift*-left-click the edge marked EA in Figure 2-15.

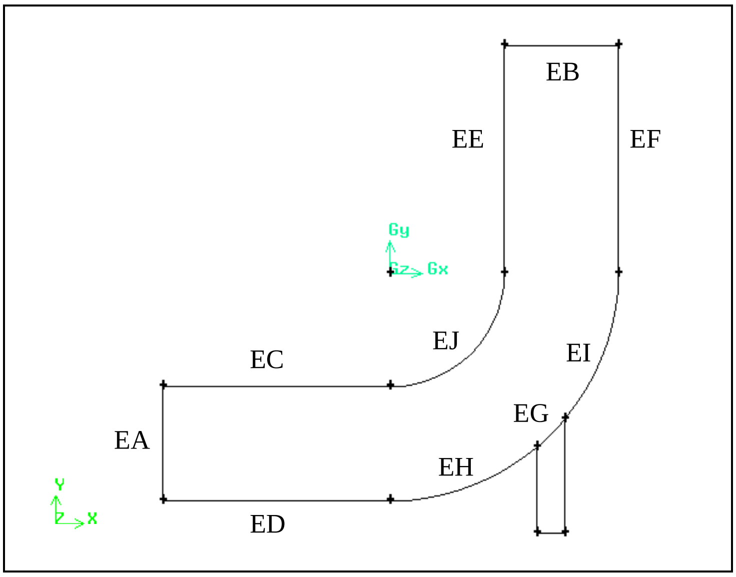


Figure 2-15: Edges to be meshed

The edge will change color and an arrow and several circles will appear on the edge.

b) *Shift*-left-click the edge marked EB in Figure 2-15.

c) Check that Apply is selected to the right of **Grading** in the **Mesh Edges** form and that Successive Ratio is selected in the **Type** option menu.

The Successive Ratio *option sets the ratio of distances between consecutive points on the edge equal to the specified* **Ratio**.

d) Enter `1.25` in the text entry box to the right of **Ratio**.

Alternatively, you can slide the **Ratio** *slider box (the small, gray rectangle with a vertical line in its center that is located on the slider bar) until* `1.25` *is displayed in the* **Ratio** *text box.*

e) Select the Double sided check box under **Grading**.

If you specify a Double sided *grading on an edge, the element intervals are graded in two directions from a starting point on the edge.* GAMBIT *determines the starting point such that the intervals on either side of the point are approximately the same length.*

Note that **Ratio** *changes to* **Ratio 1** *and* **Ratio 2** *when you select the* Double sided *check box. In addition, the value you entered for* **Ratio** *is automatically entered into both the* **Ratio 1** *and the* **Ratio 2** *text entry boxes.*

f) Select Interval count from the option menu under **Spacing** and enter a value of `10` in the text entry box. Check that Apply is selected to the right of **Spacing**.

GAMBIT *will create 10 intervals on the edge.*

g) Click the **Apply** button at the bottom of the form.

Figure 2-16 shows the mesh on the inlet and outlet edges of the large pipe.

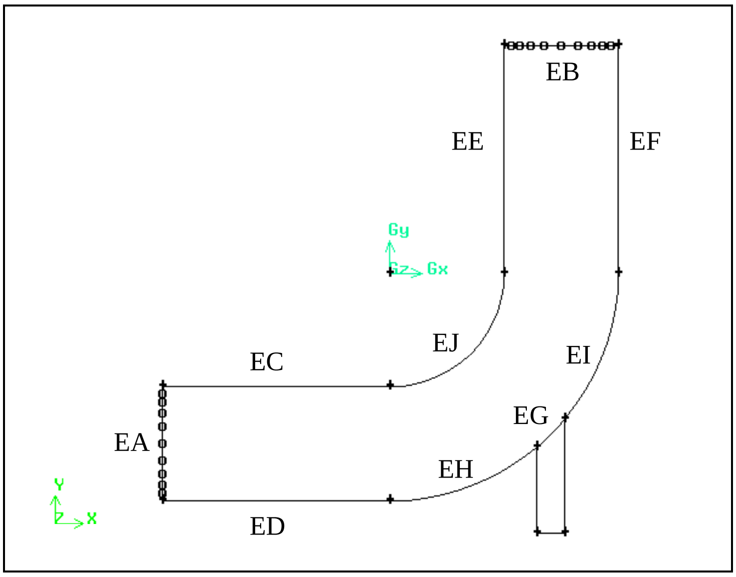


Figure 2-16: Edge meshing on inlet and outlet of large pipe

2. Mesh the four straight edges of the large pipe.

   a) Select the edges marked EC, ED, EE, and EF in Figure 2-16.

   b) Check that Apply is selected to the right of **Grading** in the **Mesh Edges** form and click the Default button to the right of **Grading**.

      GAMBIT *will unselect the* Double sided *check box and set the* **Ratio** *to* `1`.

   c) Check that Apply is selected to the right of **Spacing** and select Interval count from the option menu.

   d) Enter a value of `15` in the text entry box below **Spacing** and click the **Apply** button at the bottom of the form.

      Figure 2-17 shows the mesh on the straight edges of the large pipe.

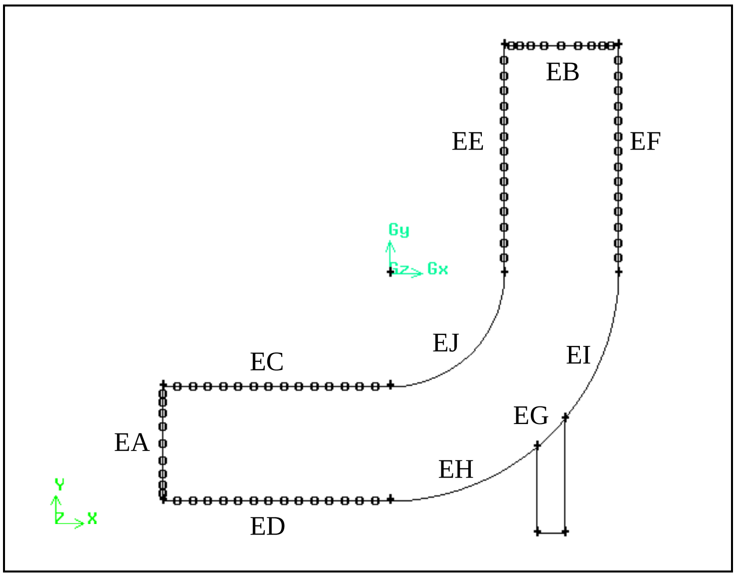


Figure 2-17: Mesh on the straight edges of the large pipe

3. Mesh the edge connecting the two pipes.

   a) Select the edge marked EG in Figure 2-17.

   b) Check that Apply is selected to the right of **Grading** in the **Mesh Edges** form and enter a value of `1` for the **Ratio**.

   c) Check that Apply is selected to the right of **Spacing**, select Interval count from the option menu, and enter a value of `6` in the text entry box below **Spacing**.

   d) Click the **Apply** button at the bottom of the form.

4. Mesh the two edges on the outer radius of the bend of the mixing elbow.

   a) Select the edge marked EH in Figure 2-17. The arrow should point towards the small pipe. *Shift*-middle-click the edge to reverse the direction of the arrow if necessary.

      ! *The arrow is small and you may have to zoom into the edge to see it. It is located near the center of the edge.*

   b) Select the edge marked EI in Figure 2-17. The arrow should point towards the small pipe. Shift-middle-click the edge to reverse the direction of the arrow if necessary.

   c) Check that Apply is selected to the right of **Grading** in the **Mesh Edges** form and enter a value of `0.9` for the **Ratio**.

   d) Check that Apply is selected to the right of **Spacing**, select Interval count from the option menu, and enter a value of `12` in the text entry box below **Spacing**.

   e) Click the **Apply** button at the bottom of the form.

      The mesh on the two edges on the outer radius of the bend is shown in Figure 2-18.

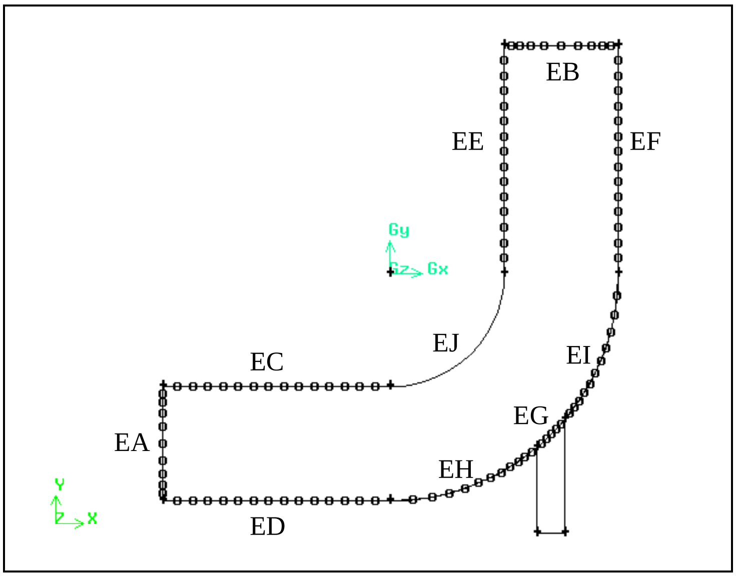


Figure 2-18: Mesh on outer bend of pipe

5. Set the grading for the inner bend of the mixing elbow.

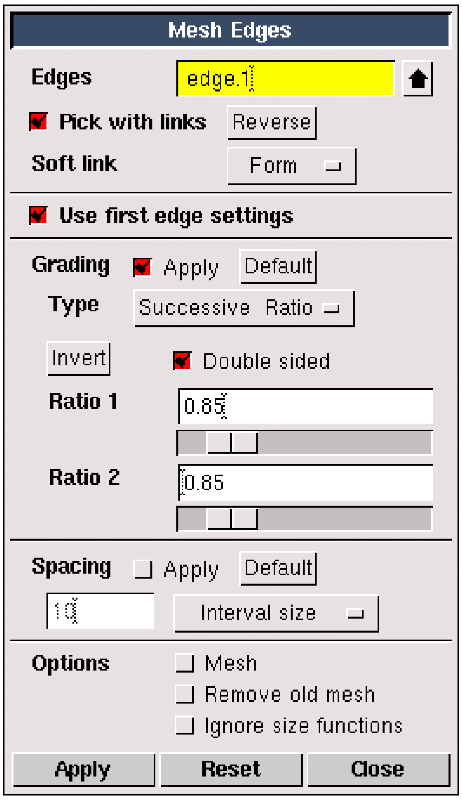
   a) Select the edge marked EJ in Figure 2-18.

   b) Check that Apply is selected to the right of **Grading** in the **Mesh Edges** form and enter a value of `0.85` for the **Ratio**.

   c) Select the Double sided check box.

   d) Unselect the Apply check box to the right of **Spacing**.

      You will not set a spacing on this edge, instead you will let GAMBIT *calculate the spacing for you when it meshes the face. You will mesh the face using a mapped mesh, so the number of nodes on the inner bend of the mixing elbow must equal the number of nodes on the outer bend, and* GAMBIT *will determine the correct number of nodes for you automatically.*



e) Unselect the **Mesh** check box under **Options** and click the **Apply** button at the bottom of the form.

You unselected the **Mesh** *check box because at this point you do not want to mesh the edge; you only want to apply the* **Grading** *to the edge.* GAMBIT *will mesh the edge using the specified* **Grading** *when it meshes the large pipe of the mixing elbow in the next step.*

Figure 2-19 shows the edge meshing for the mixing elbow geometry.

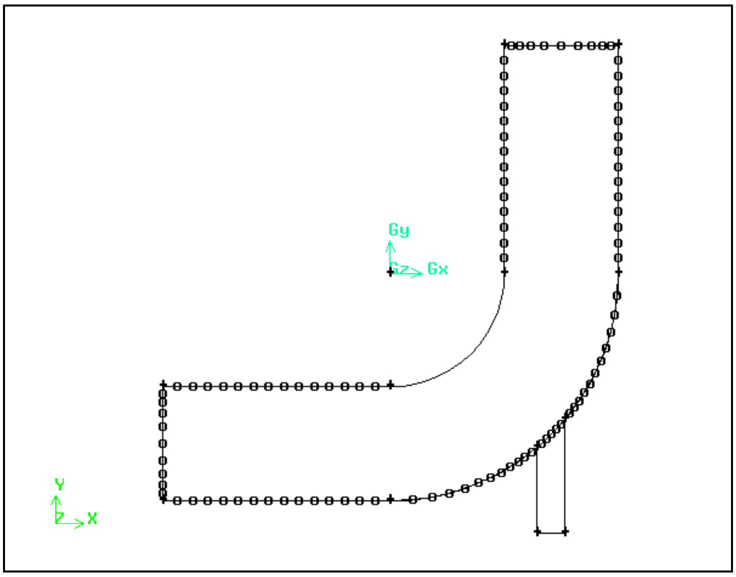


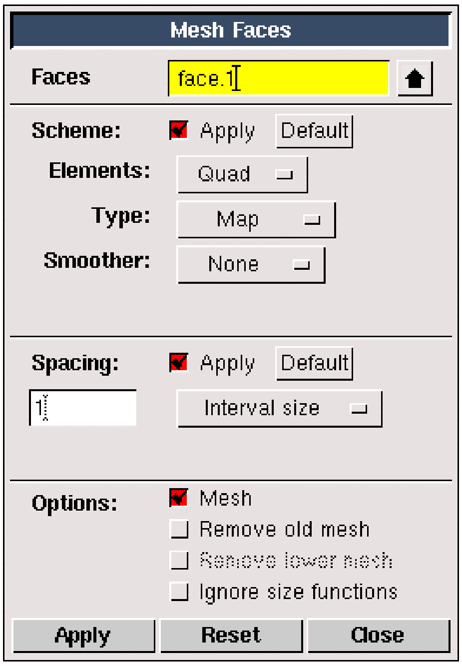
Figure 2-19: Edge meshing for the mixing elbow

Step 8: Create Structured Meshes on Faces

1. Create a structured mesh for the large pipe.

   MESH → **FACE** → **MESH FACES**

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



   a) *Shift*-left-click the large pipe in the graphics window.

      *Note that four of the vertices on this face are marked with an "*E*" in the graphics window; they are* End *vertices. Therefore,* GAMBIT *will select the* Map **Type** *of* **Scheme** *in the* **Mesh Faces** *form. See the* GAMBIT *Modeling Guide for more information on* Map *meshing.*

   b) Click the **Apply** button at the bottom of the form.

      GAMBIT *will ignore the* Interval size *of* 1 *under* **Spacing**, *because the mapped meshing scheme is being used and the existing edge meshing fully determines the mesh on all edges.*

Notice that GAMBIT *calculates the number of nodes on the inner bend of the mixing elbow and displays these nodes before creating the mesh on the face. The face will be meshed as shown in Figure 2-20.*

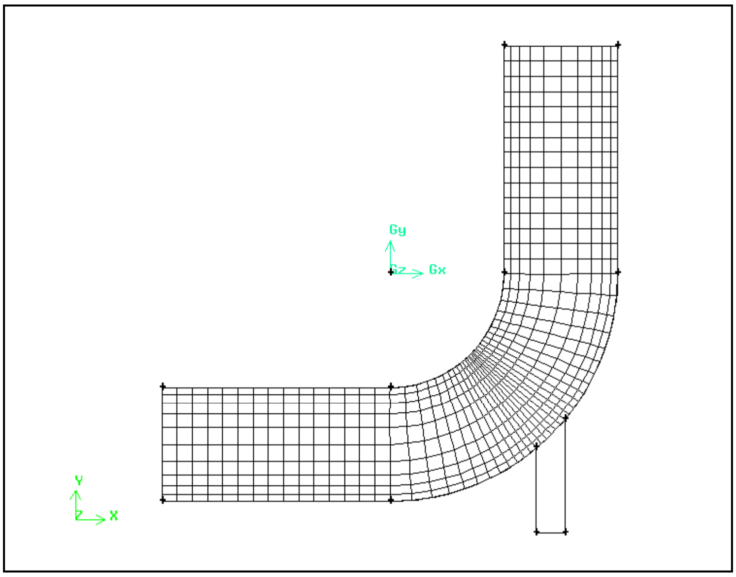


Figure 2-20: Structured mesh on the large pipe of the mixing elbow

2. Mesh the small pipe of the mixing elbow.

   a) Select the small pipe in the graphics window.

      You will force GAMBIT *to use the* Map *scheme to mesh the smaller face.*

   b) In the **Mesh Faces** form, select Quad from the **Elements** option menu under **Scheme** and Map from the option menu to the right of **Type**.

      This is an example of "enforced mapping", where GAMBIT *automatically modifies the face vertex type on the face to satisfy the chosen meshing scheme. See the* GAMBIT *Modeling Guide for more information on face vertex types.*

   c) Retain the default Interval size of `1` under **Spacing** and click the **Apply** button at the bottom of the form.

      The structured mesh for the entire elbow is shown in Figure 2-21.

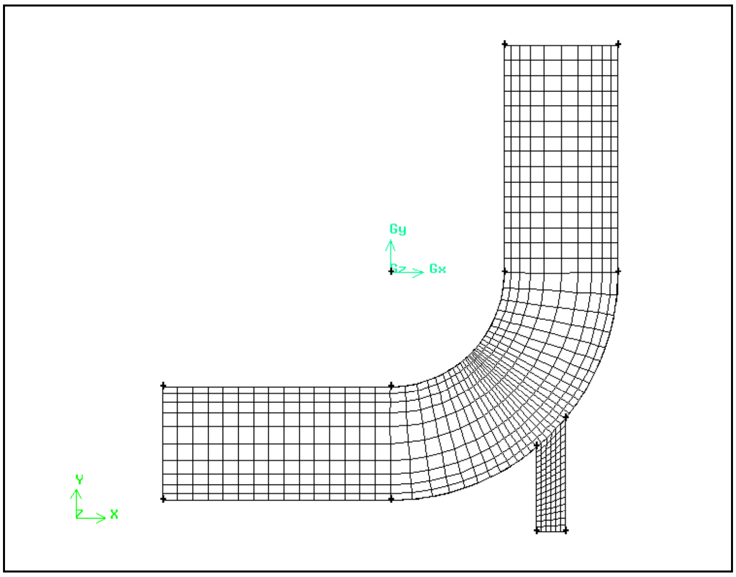


Figure 2-21: Structured mesh for the mixing elbow

Step 9: Set Boundary Types

1. Remove the mesh from the display before you set the boundary types.

   This makes it easier to see the edges and faces of the geometry. The mesh is not deleted, just removed from the graphics window.

   a) Click the **SPECIFY DISPLAY ATTRIBUTES** command button  at the bottom of the **Global Control** toolpad.

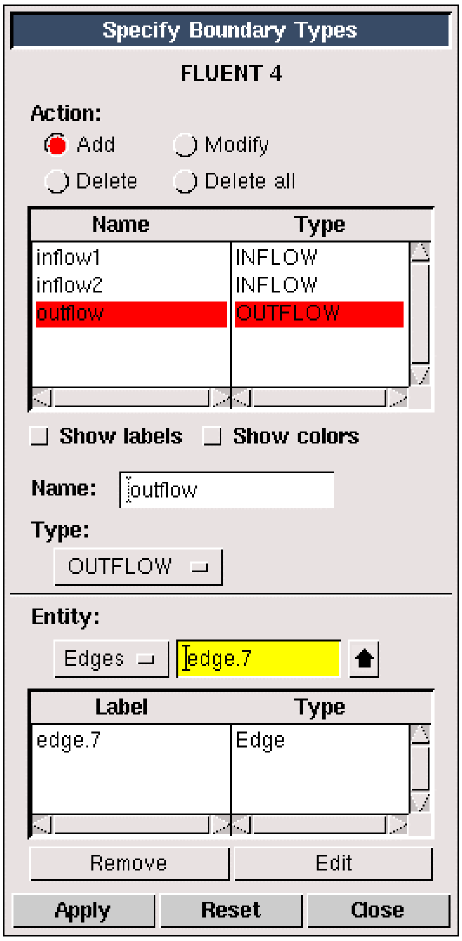
   b) Select the Off radio button to the right of **Mesh** near the bottom of the form.

   c) Click **Apply** and close the form.

2. Set boundary types for the mixing elbow.

   ZONES  → **SPECIFY BOUNDARY TYPES** 

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



Note that **FLUENT 4** *is shown as the chosen solver at the top of the form. The* **Specify Boundary Types** *form displays different* **Type***s depending on the solver selected.*

a) Define two inflow boundaries.

i. Enter the name `inflow1` in the **Name** text entry box.

If you do not specify a name, GAMBIT *will give the boundary a default name based on what you select in the* **Type** *and* **Entity** *lists.*

ii. Select INFLOW in the **Type** option menu.

iii. Change the **Entity** to Edges by selecting Edges in the option menu below **Entity**.

iv. *Shift*-left-click the main inflow for the mixing elbow in the graphics window (marked EA in Figure 2-22) and click **Apply** to accept the selection.

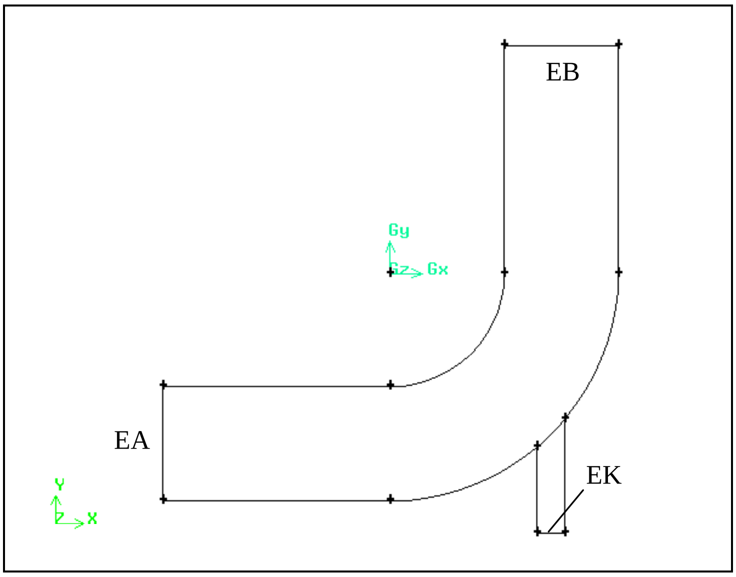


Figure 2-22: Boundary types for edges of mixing elbow

This edge will be set as an inflow boundary.

v. Enter `inflow2` in the **Name** text entry box.

vi. Check that INFLOW is still selected in the **Type** option menu and select the edge marked EK in Figure 2-22 (the inlet for the small pipe). Click **Apply** to accept the selection of the edge.

b) Define an outflow boundary.

i. Enter `outflow` in the **Name** text entry box.

ii. Change the **Type** to OUTFLOW by selecting OUTFLOW in the option menu below **Type**.

iii. Select the main outflow for the mixing elbow (the edge marked EB in Figure 2-22) and click **Apply** to accept the selection.

The inflow and outflow boundaries for the mixing elbow are shown in Figure 2-23. (NOTE: To display the boundary types in the graphics window, select the **Show labels** *options on the* **Specify Boundary Types** *form.)*

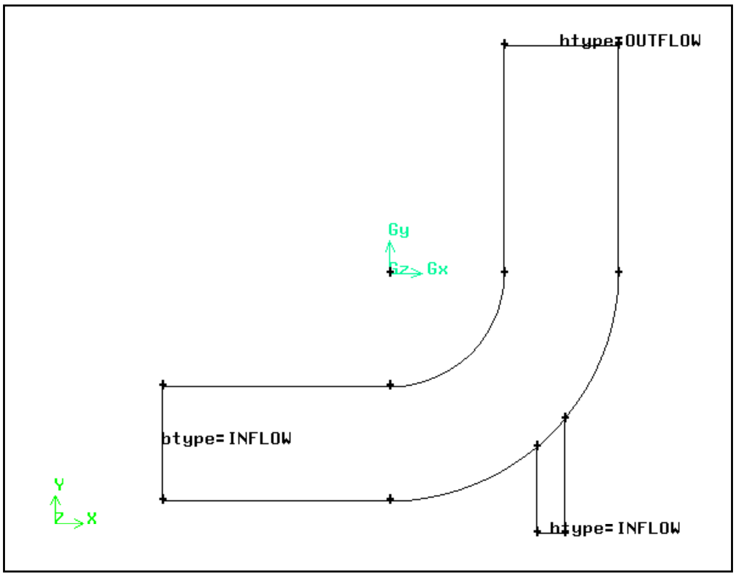


Figure 2-23: Inflow and outflow boundaries for the mixing elbow

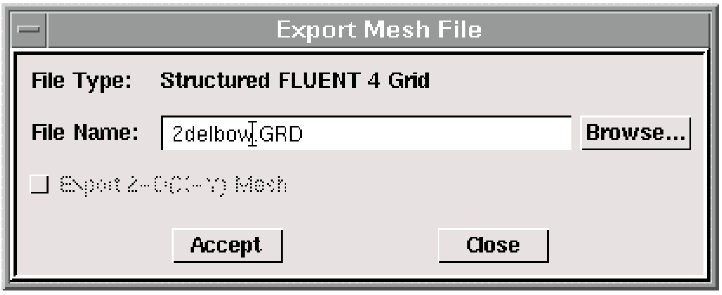
Note that you could also specify the remaining outer edges of the mixing elbow as wall boundaries. This is not necessary, however, because when GAMBIT *saves a mesh, any edges (in 2-D) on which you have not specified a boundary type will be written out as wall boundaries by default. In addition, when* GAMBIT *writes a mesh, any faces (in 2-D) on which you have not specified a continuum type will be written as* FLUID *by default. This means that you do not need to specify a continuum type in the* **Specify Continuum Types** *form for this tutorial.*

Step 10: Export the Mesh and Save the Session

1. Export a mesh file for the mixing elbow.

    File ⟶ **Export** ⟶ **Mesh...**

    This command sequence opens the **Export Mesh File** *form. Note that the* **File Type** *is* **Structured FLUENT 4 Grid**.



   a) Enter the **File Name** for the file to be exported (`2-DELBOW.GRD`).

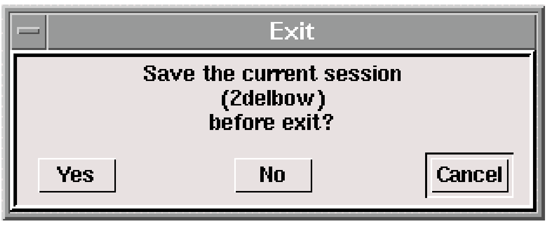
   b) Click **Accept**.

      The file will be written to your working directory.

2. Save the GAMBIT session and exit GAMBIT.

    File ⟶ **Exit**

    GAMBIT *will ask you whether you wish to save the current session before you exit.*



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

2.5 Summary

This tutorial shows you how to generate a 2-D mesh using the “bottom-up” approach. Since the mesh is to be used in FLUENT 4, it was generated in a single block, structured fashion. Several other features that are commonly used for 2-D mesh generation were also demonstrated, including entering vertices using a background grid, creating straight edges and arcs, and specifying node distributions on individual edges. As compared to Tutorial 1, which omitted some details, all steps required to create a mesh ready to read into the solver were covered, including how to set boundary types, choose a specific Fluent solver, and finally write out the mesh file.

3. MODELING A THREE-PIPE INTERSECTION (3-D)

This tutorial employs "primitives"—that is, predefined GAMBIT modeling components and procedures. There are two types of GAMBIT primitives:

- Geometry
- Mesh

Geometry primitives are volumes possessing standard shapes—such as bricks, cylinders, and spheres. *Mesh* primitives are standard mesh configurations.

In this tutorial, you will use geometry primitives to create a three-pipe intersection. You will decompose this geometry into four parts and add boundary layers. Finally, you will mesh the three-pipe intersection and will employ a mesh primitive to mesh one part of the decomposed geometry.

In this tutorial you will learn how to:

- Create volumes by defining their dimensions
- Split a volume
- Use GAMBIT journal files
- Add boundary layers to your geometry
- Prepare the mesh to be read into POLYFLOW

3.1 Prerequisites

This tutorial assumes you have worked through Tutorial 1 and you are consequently familiar with the GAMBIT interface.

3.2 Problem Description

The problem to be considered is shown schematically in Figure 3-1. The geometry consists of three intersecting pipes, each with a diameter of 6 units and a length of 4 units. The three pipes are orthogonal to each other. The geometry can be represented as three intersecting cylinders and a sphere octant at the corner of the intersection.

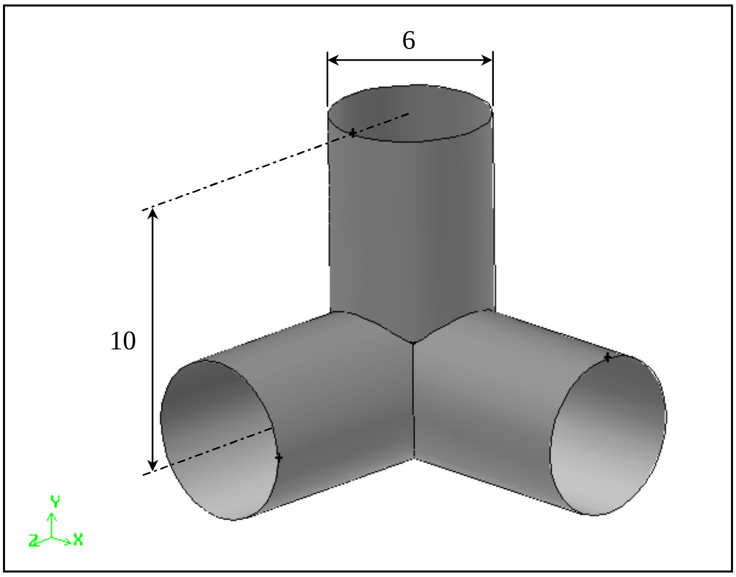


Figure 3-1: Problem specification

3.3 Strategy

In this tutorial, you will quickly create the basic geometry for a three-pipe intersection. The basic geometry can be automatically meshed with tetrahedra, but your goal in this tutorial is to create a conformal, hexahedral mesh for POLYFLOW, which requires some decomposition of the geometry before meshing. Thus, the tutorial shows some of the typical procedures for decomposing a complicated geometry into “meshable” volumes.

The first decomposition involves using a brick to split off a portion of the three-pipe intersection. The resulting volume is described as a sphere “octant” (one-eighth of a sphere) residing in the corner of the intersection, as shown in Figure 3-2. This volume, which is very similar in shape to a tetrahedron, will therefore be meshed using GAMBIT’s Tet Primitive scheme. Note that this creates a hexahedral mesh in a tetrahedral topology; it does *not* create tetrahedral cells.

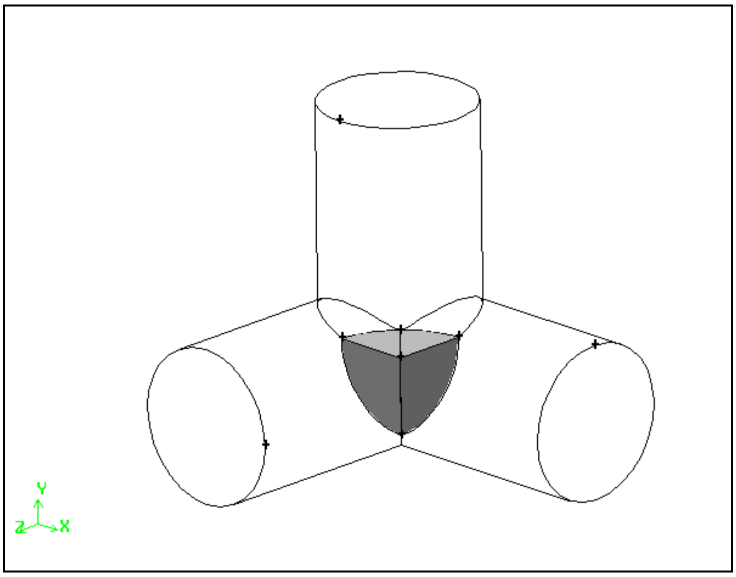


Figure 3-2: Decomposition of the three-pipe intersection geometry

The remaining geometry is then split into three parts, one for each pipe, as shown in Figure 3-1. To do this, you will create an edge and three faces that are used to split the volume into the required three parts. These volumes are meshed using GAMBIT’s Cooper scheme (described in detail in the GAMBIT Modeling Guide). This tutorial illustrates three different ways to specify the source faces required by the Cooper scheme.

Two other helpful topics are covered in this tutorial: the use of journal files and the meshing of boundary layers. The journal file contains a record of all your command inputs to GAMBIT. This file can be edited and your inputs can be converted into variable parameters that allow subsequent geometries (with changes in key dimensions, for example) to be quickly created and meshed. The boundary layer meshing tools in GAMBIT allow you to control how the mesh is refined near walls and other boundaries.

3.4 Procedure

Start GAMBIT.

Step 1: Select a Solver

1. Choose the solver you will use to run your CFD calculation by selecting the following from the main menu bar:

   Solver ⟶ **POLYFLOW**

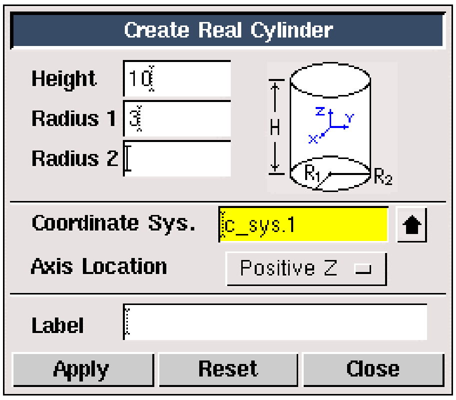
   The choice of a solver dictates the options available in various forms (for example, the boundary types available in the **Specify Boundary Types** *form). The solver currently selected is indicated at the top of the* GAMBIT *GUI.*

Step 2: Create the Geometry

1. Create the three pipes for the intersection.

   GEOMETRY ⟶ **VOLUME** ⟶ **CREATE VOLUME** **R** Cylinder

   This command sequence opens the **Create Real Cylinder** *form.*



   a) Create the first pipe.

      i. Enter a **Height** of `10` in the **Create Real Cylinder** form.

      ii. Enter 3 for **Radius 1**.

The text entry box for **Radius 2** *can be left blank;* GAMBIT *will set this value by default to be the same value as* **Radius 1**.

iii. Select Positive Z (the default) in the list to the right of **Axis Location**.

iv. Click **Apply**.

b) Create the second pipe. Use the same **Height** and **Radius 1** as above, and select Positive X in the list to the right of **Axis Location**.

c) Create the third pipe. Use the same **Height** and **Radius 1** as above, and select Positive Y in the list to the right of **Axis Location**.

2. Click the **FIT TO WINDOW** command button , at the top left of the **Global Control** toolpad, to view all three cylinders.

You can rotate the view by holding down the left mouse button and moving the mouse. The cylinders are shown in Figure 3-3.

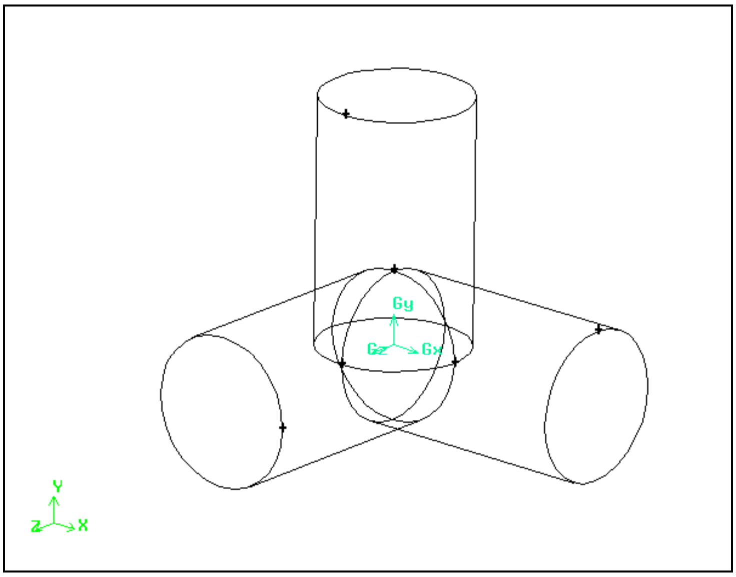
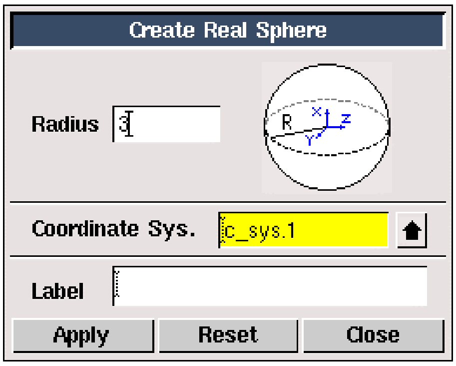


Figure 3-3: Three cylinders for the three-pipe intersection

3. Create a sphere to complete the basic geometry.

GEOMETRY → **VOLUME** → **CREATE VOLUME** R Sphere

This command sequence opens the **Create Real Sphere** *form.*



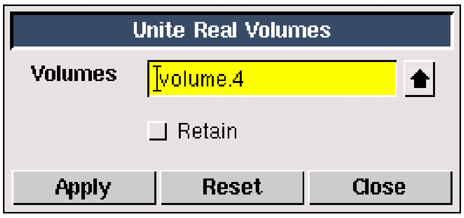
a) Enter 3 for the **Radius**.

b) Click **Apply**.

4. Unite the four volumes into one volume.

GEOMETRY → **VOLUME** → **BOOLEAN OPERATIONS**

This command sequence opens the **Unite Real Volumes** *form.*



a) Shift-left-click all of the volumes in the graphics window, and click **Apply**.

These volumes will be united into one volume. The completed geometry is shown in Figure 3-4.

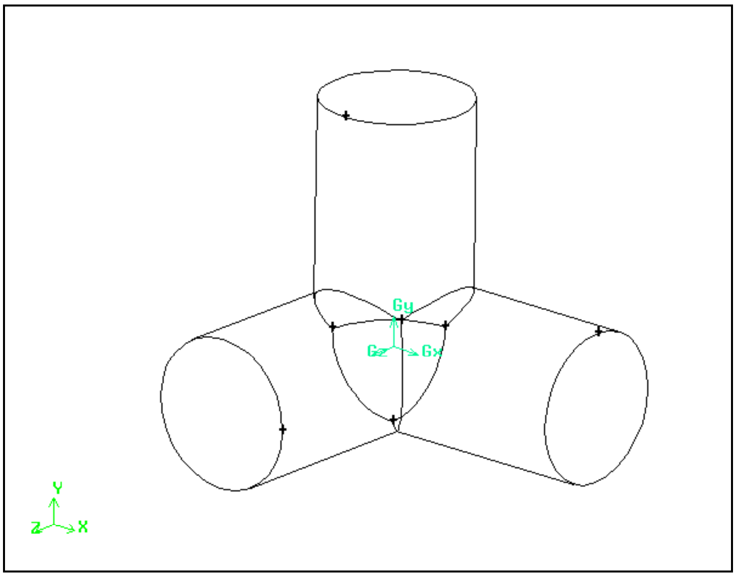


Figure 3-4: The completed geometry

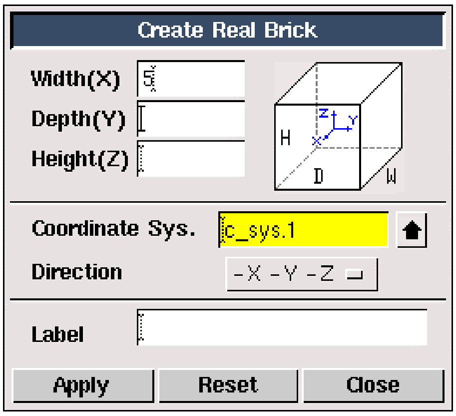
Step 3: Decompose the Geometry

It is possible to automatically mesh this full geometry using the TGrid *scheme. However, it is not possible to automatically mesh this geometry with conformal hexahedra. In order to generate a conformal hexahedral mesh, you must now decompose the geometry into portions, each fulfilling the criteria of available hexahedral meshing schemes. In this example, you will create a brick that will be used to split the three-pipe volume, forming a sphere octant (one-eighth of a sphere) where the three pipes intersect. You will then create an edge, and use it to form three faces inside the geometry. These faces will be used to split the three-pipe intersection volume into three pipe sections.*

1. Create a brick.

   GEOMETRY → **VOLUME** → **CREATE VOLUME** R Brick

   This command sequence opens the **Create Real Brick** *form.*



   a) Enter a value of 5 for the **Width** of the brick.

      GAMBIT *will set the* **Depth** *and the* **Height** *of the brick to be the same as the* **Width** *if no values are entered in these fields in the form.*

   b) Select -X -Y -Z in the list next to **Direction**.

   c) Click **Apply**.

      The view in the graphics window is shown in Figure 3-5.

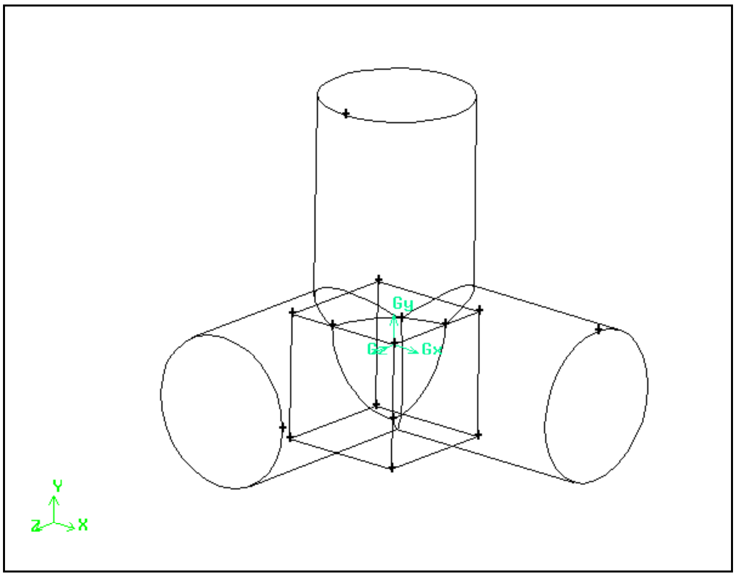


Figure 3-5: Three-pipe geometry and brick

2. Split the volume and create a sphere octant volume where the three pipes intersect.

   If you split one volume with another volume, the following volumes will result:

   - *Volumes corresponding to the common region(s) from intersection.*
   - *Volumes corresponding to the region(s) defined by subtracting the second volume from the first.*

   In other words, splitting a volume results in a combination of the intersection and subtraction Boolean operations. The order of selecting the volumes is important. For example, Figure 3-6 shows the difference between splitting volume A using volume B, and vice versa.

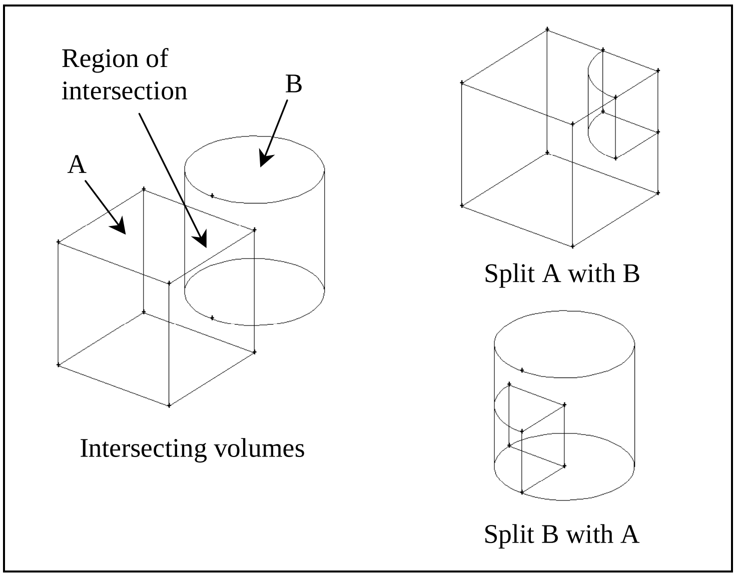
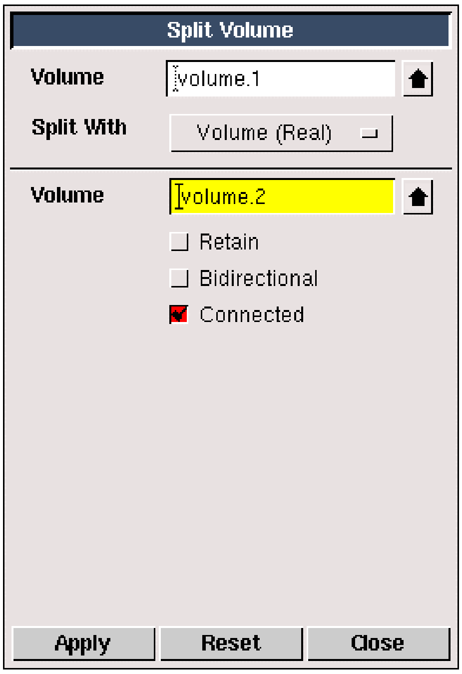


Figure 3-6: Splitting volumes

GEOMETRY → **VOLUME** → **SPLIT/MERGE VOLUMES**

This command sequence opens the **Split Volume** *form.*



a) Select the three-pipe volume in the graphics window.

b) Select Volume (Real) as the **Split With** option.

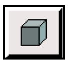
c) Left-click in the **Volume** list box located below the **Split With** section to make the **Volume** list box active.

d) Unselect the Bidirectional option.

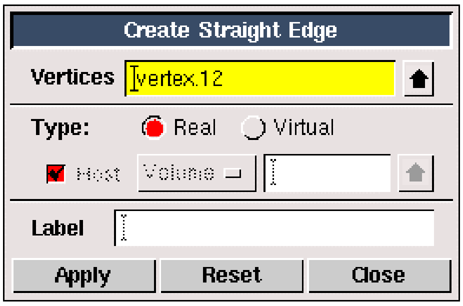
e) Select the brick and click **Apply** to accept the selection.

GAMBIT *will split the three-pipe volume using the brick, leaving two volumes: the three pipes (*volume.1*) and the sphere octant (*volume.3*).*

3. Create a straight edge inside the three-pipe volume.

GEOMETRY  → **EDGE**  → **CREATE EDGE** 

This command sequence opens the **Create Straight Edge** *form.*



a) *Shift*-left-click the vertex at the origin (*Gx*, *Gy*, *Gz*).

b) Select the vertex that is shared by all three cylinders ($x = y = z$).

c) Click **Apply** to accept the selected vertices and create an edge between them.

The edge is shown in Figure 3-7 and will appear yellow in the graphics window.

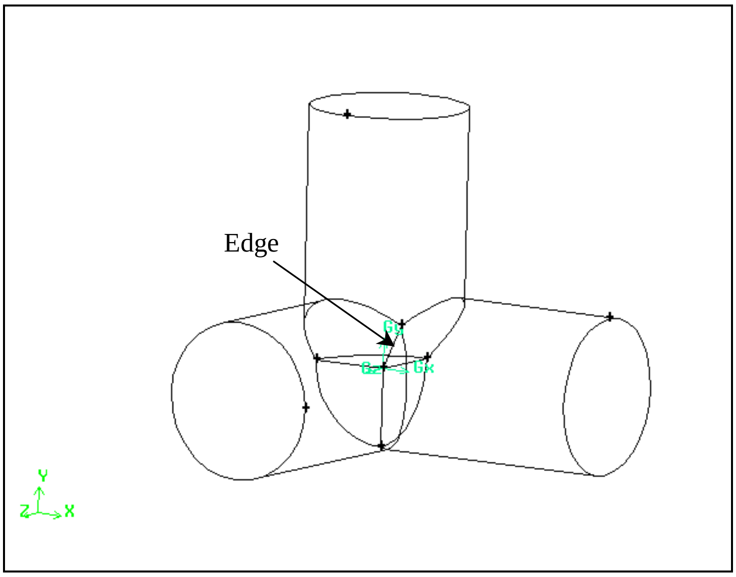
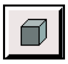
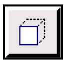
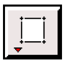
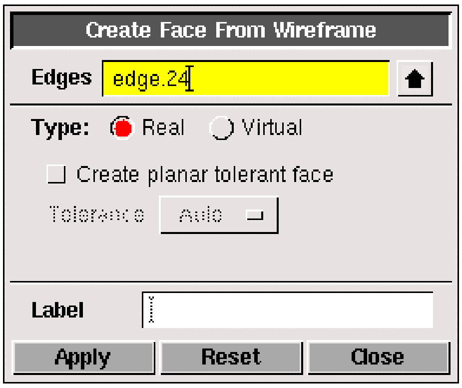


Figure 3-7: Straight edge created inside the volume

4. Create faces inside the three-pipe volume.

GEOMETRY  → **FACE**  → **FORM FACE** 

This command sequence opens the **Create Face From Wireframe** *form.*



a) Create a face inside the geometry using the edge created in the previous step.

i. Select the edge created in the previous step.

ii. Select a curved edge on one of the cylindrical surfaces that is connected to the edge just selected.

iii. Select the edge that closes the loop.

The three edges to be selected are shown in Figure 3-8.

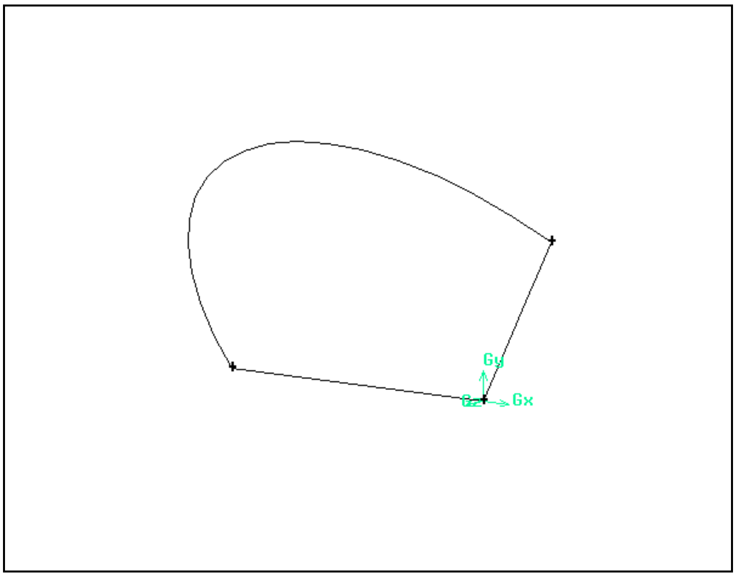


Figure 3-8: Three edges used to create a face

iv. Click **Apply** to accept the selection and create a face.

The edge created in the previous step will turn blue.

b) Create a second face by selecting the blue edge, a different curved edge connected to the blue edge, and the edge that closes the loop.

c) Create a third face by selecting the blue edge, the third curved edge connected to the blue edge, and the edge that closes the loop.

The three faces are shown in Figure 3-9. It may be useful to remove the volumes from the display; it is then easier to see the faces you created. The volumes are not deleted, just removed from the graphics window. To remove the volumes from the display, click the **SPECIFY DISPLAY ATTRIBUTES** *command button*  *at the bottom of the* **Global Control** *toolpad. Select the check box to the left of* **Volumes**. *Select the* Off *radio button to the right of* **Visible** *near the bottom of the form, and click* **Apply**. *Turn the visibility of the volumes back on after you have examined the faces.*

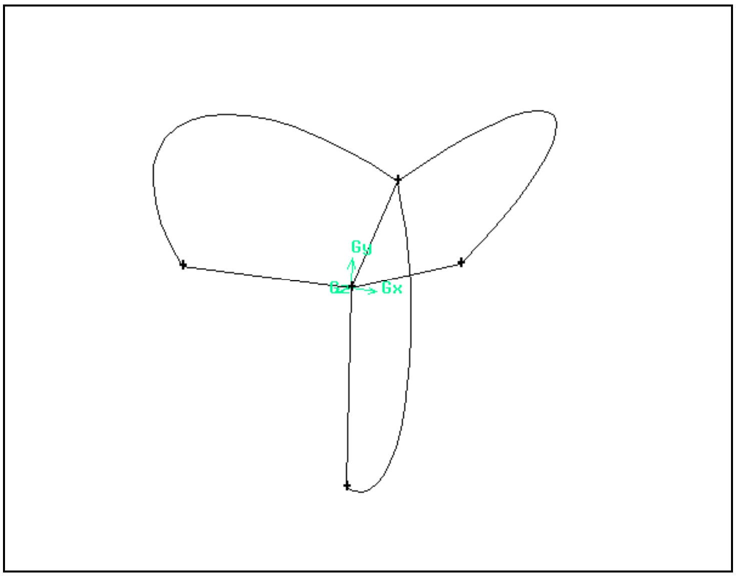
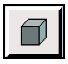
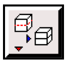
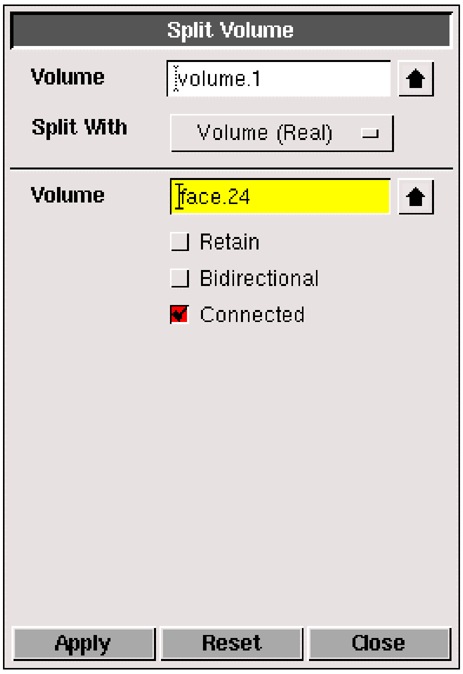


Figure 3-9: Three faces created inside the pipe intersection

5. Split the three-pipe volume using the faces created in the previous step.

   GEOMETRY  → **VOLUME**  → **SPLIT/MERGE VOLUMES** 

   This command sequence opens the **Split Volume** *form.*



a) Select the three-pipe volume in the graphics window.

b) Select Face (Real) as the **Split With** option.

c) Left-click in the **Face** list box located *below* the **Split With** section to make the **Face** list box active.

d) Pick one of the internal faces created in the steps above.

Shift-middle-click *on a face if you need to unselect it and select the face next to it.*

e) Click **Apply** to split the volume.

f) Repeat Steps (a) through (e), using one of the other two internal faces to split the three-pipe volume.

g) Repeat Steps (a) through (e) again, using the remaining internal face to split the three-pipe volume.

GAMBIT *will create three volumes that are connected with common geometry. The decomposed geometry is shown in Figure 3-10 and is now ready to be meshed.*

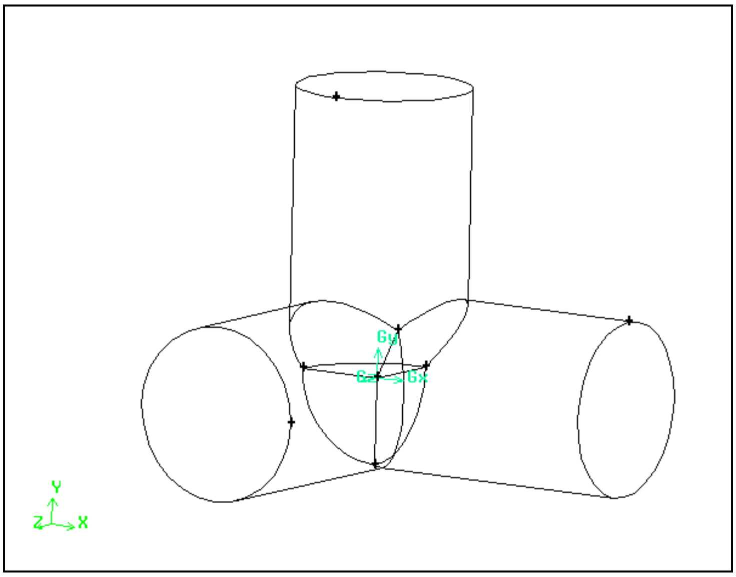


Figure 3-10: Decomposed geometry

Step 4: Journal Files

! *Note that this step is not an essential part of the tutorial and is designed to provide information on using journal files in* GAMBIT.

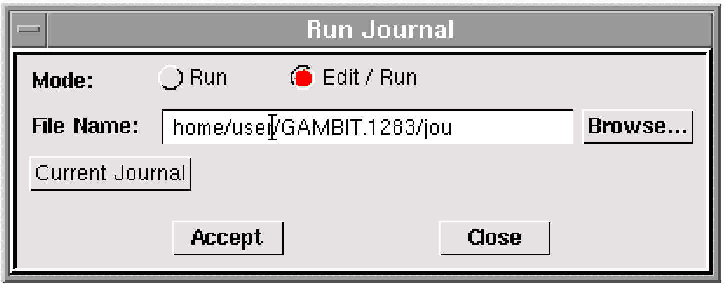
Every time a GUI operation is performed in GAMBIT, *the corresponding commands are automatically written to a journal file. This journal file, therefore, provides a backup copy of all the commands for the current session.*

Journal files can be used to recreate a geometry or mesh that was created in a previous session. You can view, run, and edit journal files in GAMBIT. *See the* GAMBIT *User's Guide for more information on journal files.*

1. View the journal file for the current GAMBIT session.

   File → **Run Journal...**

   This command sequence opens the **Run Journal** *form.*



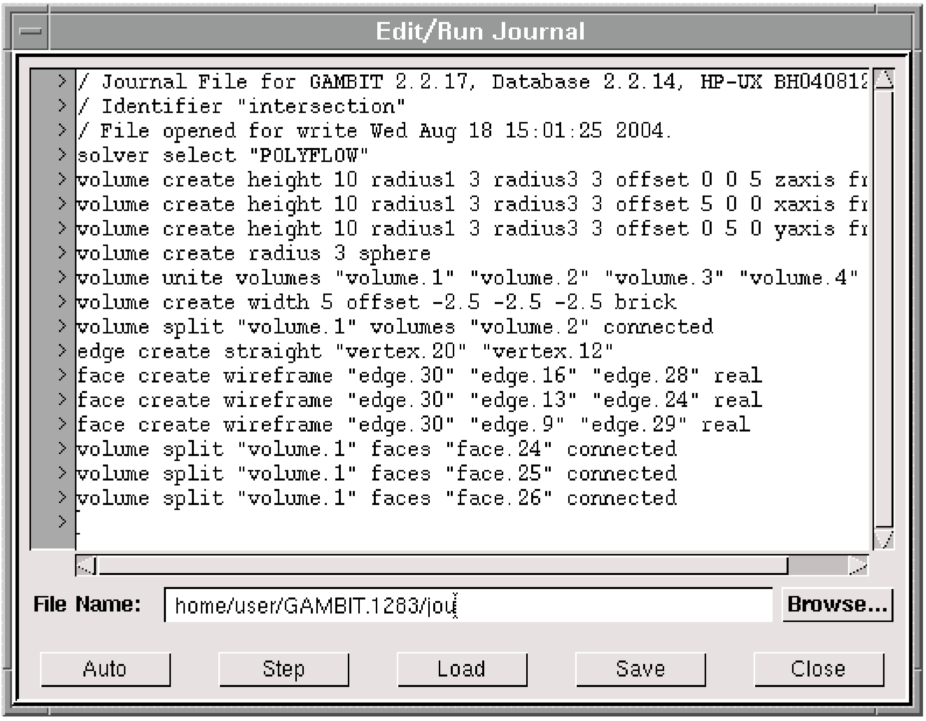
   a) Select the Edit / Run **Mode** option at the top of the form.

   b) Click the Current Journal button.

      The **File Name** *for the current journal file will appear in the form.*

   c) Click **Accept**.

      This action opens the **Edit/Run Journal** *form. You can see the journal file for the current session, showing every step completed.*



2. Edit the current journal file.

   a) Left-click at the end of the first line and press the *Enter* key.

      GAMBIT *will open a new line where you can type a command.*

   b) Type `reset` in the new line.

      ! *If you run the journal file without executing the* `reset` *command,* GAMBIT *creates new geometry on top of the existing geometry.*

3. Save the journal file with a new name.

   a) In the **File Name** text entry box at the bottom of the form, delete the text “`GAMBIT.#####/jou`”.

      ##### is the process identifier for the current GAMBIT *session. In the above form, ##### is* `1283`*.*

   b) Rename the journal file by typing `3pipe.geo` in the **File Name** text entry box.

c) Click the **Save** button at the bottom of the form.

The file will be saved to your working directory. By saving the journal file to another name, you ensure that it will not be overwritten or appended.

4. Replay the steps you have taken in the current session.

a) Hold down the right mouse button in the **TEXT EDIT FIELD** (this name will be displayed in the **Description** window when the mouse cursor is over this field) of the **Edit/Run Journal** form until a menu appears. Choose the **Select All** option in the menu.

A black box in the **LINE EXECUTION COLUMN** *of the* **Edit/Run Journal** *form indicates that a line is selected. Note that all lines are now marked with a black box. You can select/unselect individual lines by clicking the left mouse button on the arrow on the left side of the required line.*

b) Repeatedly click the **Step** button at the bottom of the **Edit/Run Journal** form until a cylinder appears in the graphics window.

Note that GAMBIT*'s current position in the journal file is marked by an asterisk in the* **LINE EXECUTION COLUMN** *of the* **Edit/Run Journal** *form. The* **Step** *button allows you to step through a journal file one line at a time. Each time the* **Step** *button is clicked,* GAMBIT *will execute the next highlighted line; it will skip any lines that are not highlighted.*

GAMBIT *has used the information in the journal file to recreate the first cylinder you created in Step 2.*

c) Click the **Step** button again.

A second cylinder appears in the graphics window.

d) Click the **Auto** button in the **Edit/Run Journal** form.

The **Auto** *button allows you to automatically rerun a journal file. If the* **Auto** *button is used,* GAMBIT *will automatically execute all lines that are highlighted, and skip any lines that are not highlighted.* GAMBIT *just used your journal file to redo the geometry creation and decomposition for the three-pipe intersection. Each line of the journal file was displayed in the* **Transcript** *window as it was executed.*

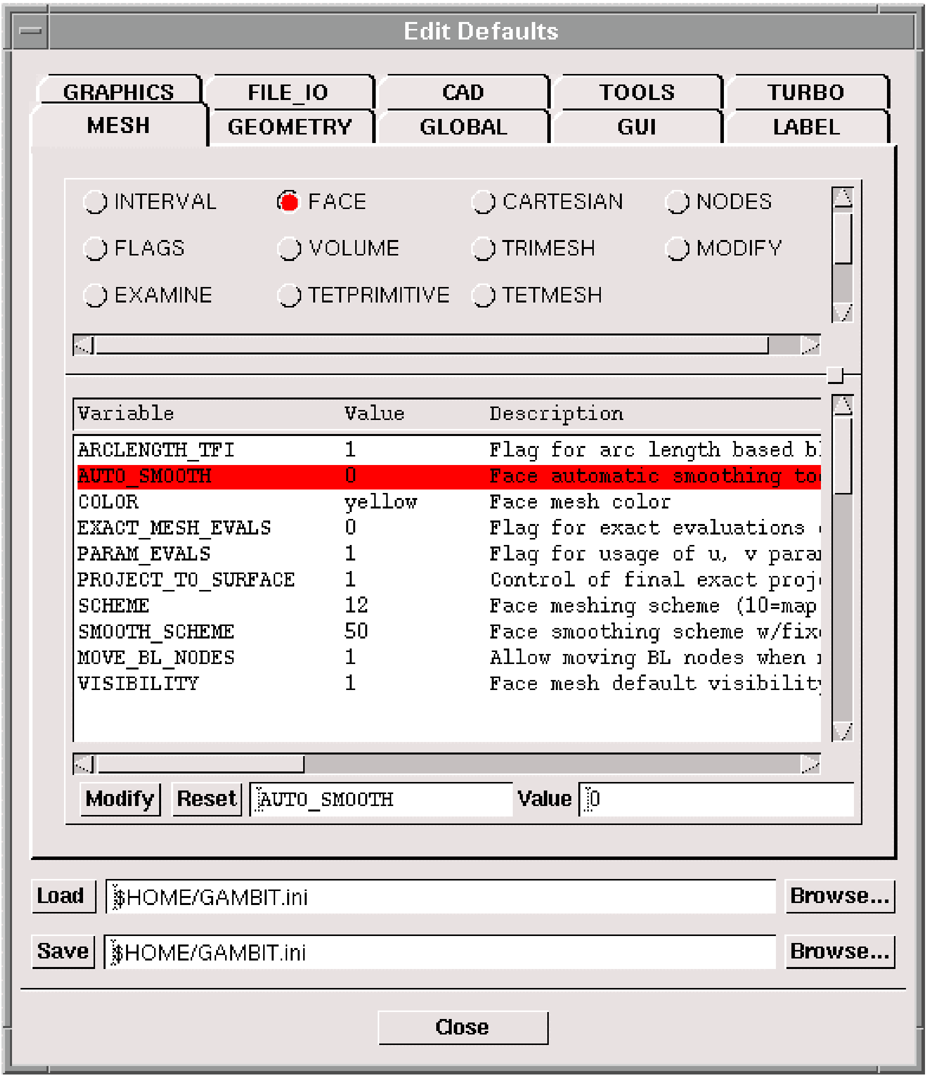
e) Close the **Edit/Run Journal** form.

Step 5: Turn Off Automatic Smoothing of the Mesh

It is necessary to turn off smoothing of the mesh in this example to prevent the boundary layers from being smoothed out during the volume meshing.

Edit → **Defaults...**

This command sequence opens the **Edit Defaults** *form.*



1. Select the **MESH** tab at the top of the form.

    This displays the types of meshing for which you can set defaults.

2. Select the FACE radio button.

    GAMBIT *displays the* `Variables` *for which defaults are set in a list in the* **Edit Defaults** *form.*

3. Select `AUTO_SMOOTH` in the `Variable` list.

    `AUTO_SMOOTH` *will appear in the text entry box at the bottom of the list and its default value will appear in the* **Value** *text entry box.*

4. Enter a value of `0` in the **Value** text entry box.

5. Click the **Modify** button to the left of `AUTO_SMOOTH`.

    The **Value** *of the variable* `AUTO_SMOOTH` *will be updated in the list.*

6. Close the **Edit Defaults** form.

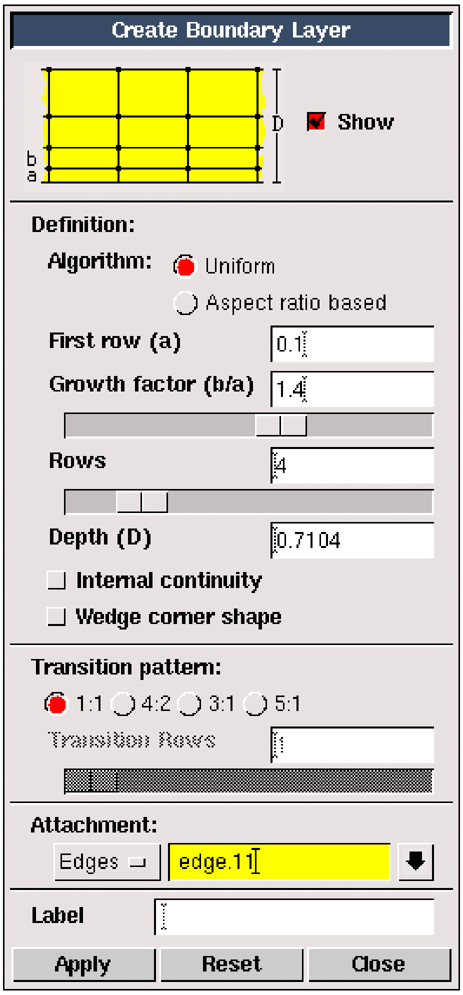
Step 6: Apply Boundary Layers at Walls

Boundary layers are layers of elements growing out from a boundary into the domain. They are used to locally refine the mesh in the direction normal to a face or an edge. A single boundary layer can be attached to several face/edge pairs or volume/face pairs. The direction of the boundary layer is indicated during picking with an arrow that points towards the middle of the active face or volume.

1. Create boundary layers on the edges where the sphere octant intersects the pipes.

   MESH → **BOUNDARY LAYER** → **CREATE BOUNDARY LAYER**

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



a) Enter `0.1` next to **First row** under **Definition**.

This defines the height of the first row of elements normal to the edge.

b) Enter `1.4` next to **Growth factor**.

This sets the ratio of distances between consecutive rows of elements.

c) Move the slider box below **Rows** until the number of rows = `4`.

This defines the total number of element rows. Notice that GAMBIT *updates the* **Depth** *automatically. The depth is the total height of the boundary layer.*

d) Retain the default **Algorithm** (Uniform).

e) Retain the default **Transition pattern** (1:1).

f) Select one of the three curved edges where the sphere octant intersects the pipes (Figure 3-11).

The boundary layer will be displayed on the edge.

g) Check that the arrow indicating the direction of the boundary layer is pointing towards the origin (*Gx*, *Gy*, *Gz*). If it is not, *Shift*-middle-click the edge until the arrow is pointing in the correct direction.

h) Select a second curved edge where the sphere octant intersects the pipes and ensure that the arrow on the edge is pointing towards the origin.

i) Repeat for the third curved edge.

The boundary layers will be displayed on the edges as shown in Figure 3-11.

j) Click **Apply** in the **Create Boundary Layer** form to apply the boundary layers to the edges.

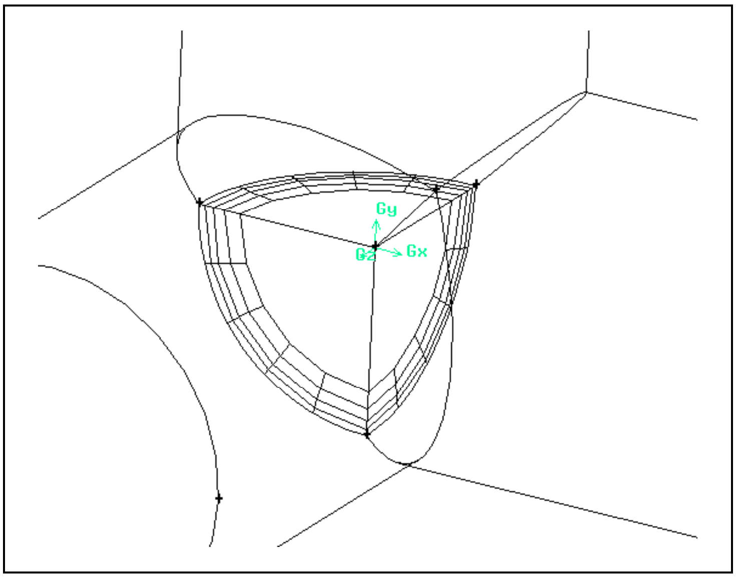


Figure 3-11: Boundary layer on three edges of the sphere octant

2. Repeat the above steps to create the same boundary layer on the three curved edges where the three pipes intersect, as shown in Figure 3-12. Again, the arrows on the edges must point towards the origin.

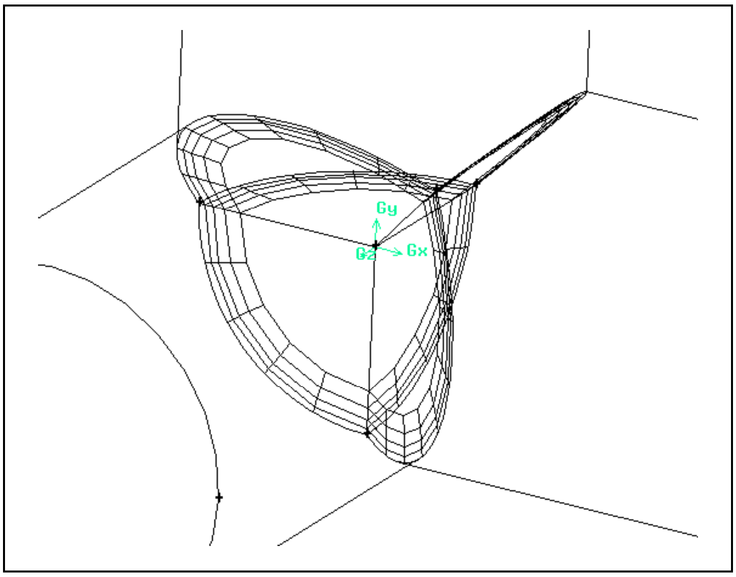
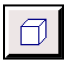


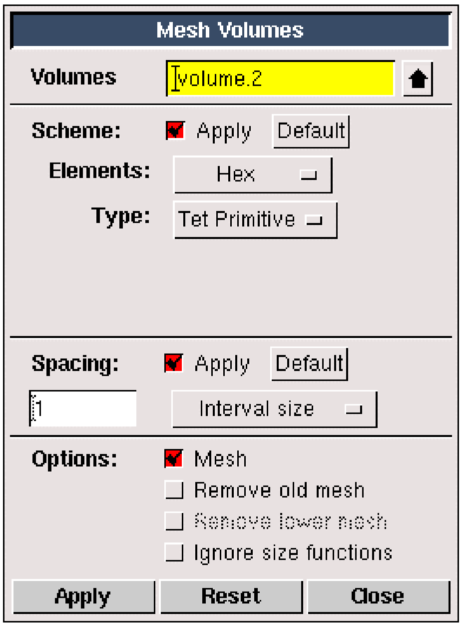
Figure 3-12: Boundary layers on the three edges where the pipes intersect

Step 7: Mesh the Sphere Octant Volume

1. Mesh the sphere octant.

   MESH  → **VOLUME**  → **MESH VOLUMES** 

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



   a) Select the sphere octant in the graphics window.

      GAMBIT *automatically selects* Hex **Elements** *and the* Tet Primitive **Type** *under* **Scheme** *in the* **Mesh Volumes** *form, because the volume represents a logical tetrahedron. (NOTE: The* Tet Primitive *scheme divides a logical tetrahedron into four logical-hexahedral blocks and creates hexahedral mesh elements in each block. The* Tet Primitive *scheme does* not *create tetrahedral mesh elements. (See the* GAMBIT *Modeling Guide.))*

   b) Accept the default Interval size under **Spacing** in the **Mesh Volumes** form and click the **Apply** button at the bottom of the form.

      The mesh for the sphere octant is shown in Figure 3-13 Note the boundary layers you applied on three faces of the sphere octant.

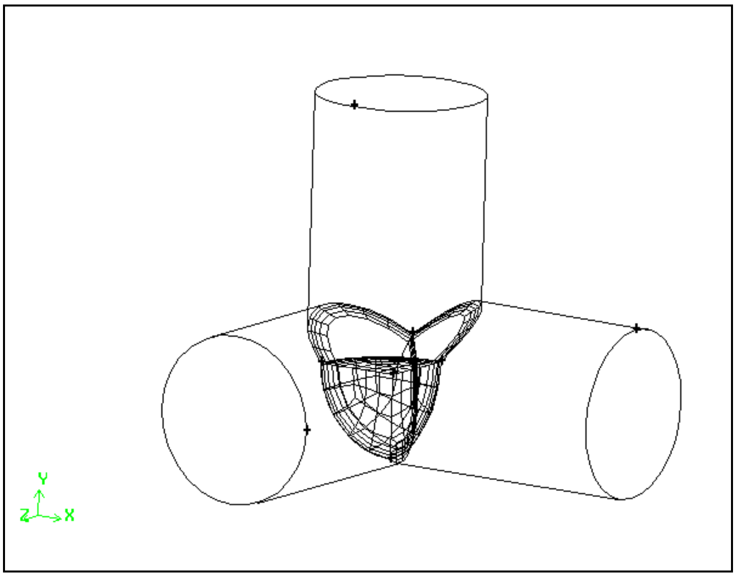


Figure 3-13: Mesh on sphere octant

2. Remove the mesh from the display before you mesh the three pipes.

   This makes it easier to see the edges and faces of the geometry. The mesh is not deleted, just removed from the graphics window.

   a) Click the **SPECIFY DISPLAY ATTRIBUTES** command button  at the bottom of the **Global Control** toolpad.

   b) Select the **Mesh:**Off option near the bottom of the form.

   c) Click **Apply** and close the form.

      The boundary layers will still be visible in the graphics window.

Step 8: Mesh the Pipe Volumes

You will now mesh the three pipes. These volumes will be meshed using GAMBIT*'s* Cooper *scheme (described in detail in the* GAMBIT *Modeling Guide). This tutorial illustrates three different ways to specify the source faces (the faces whose surface meshes are to be swept through the volume to form volume elements) required by the* Cooper *scheme. In the first example, you will modify the face vertex types for the side face of one pipe. This is the safest way to ensure correct meshing. In the second example, you will enforce the* Submap *scheme on the side face of the pipe. In the third example, you will enforce the* Cooper *meshing scheme for the volume and hand-pick all the source faces.*

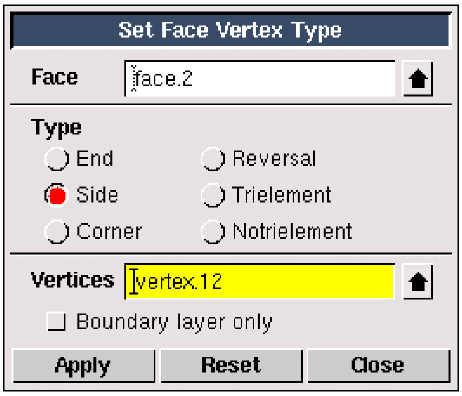
1. Mesh one of the pipes by changing the vertex type on the wall face to Side and then using the Cooper meshing scheme to mesh the volume.

   By changing the vertex type to Side *on the wall face of the pipe, you will enable* GAMBIT *to use the* Submap *scheme on this face. The criteria for the* Cooper *meshing scheme will then be fulfilled for the pipe, and the pipe can be meshed using the* Cooper *scheme.*

   a) Change the vertex type on the wall face to Side.

   MESH → **FACE** → **SET FACE VERTEX TYPES**

   This command sequence opens the **Set Face Vertex Type** *form.*



   i. Select the **Type:**Side (default) option.

ii. Select the wall face of the pipe (shown in Figure 3-14) in the graphics window. Note the vertex on the wall face marked with an “E” for End (where the three pipes intersect).

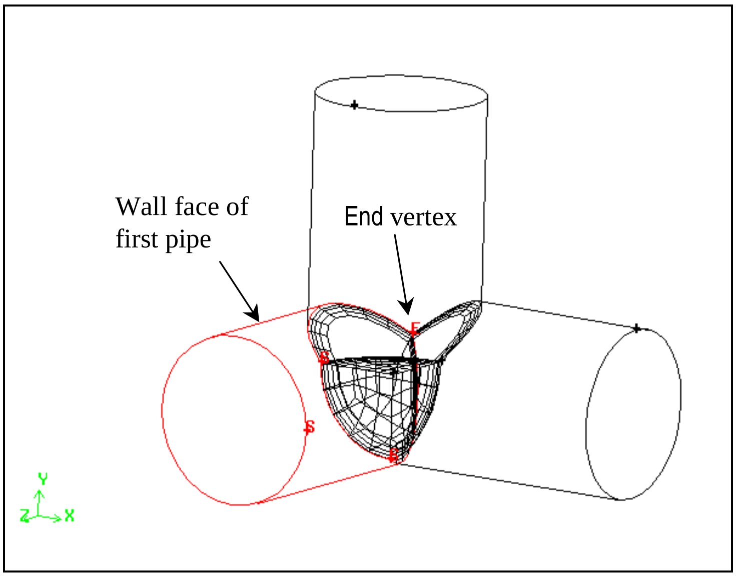


Figure 3-14: Wall face of the first pipe volume showing the End vertex

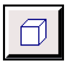
iii. Left-click in the **Vertices** list box.

iv. Select the vertex that was marked with an “E” in the graphics window (where the three pipes intersect, as shown in Figure 3-14).

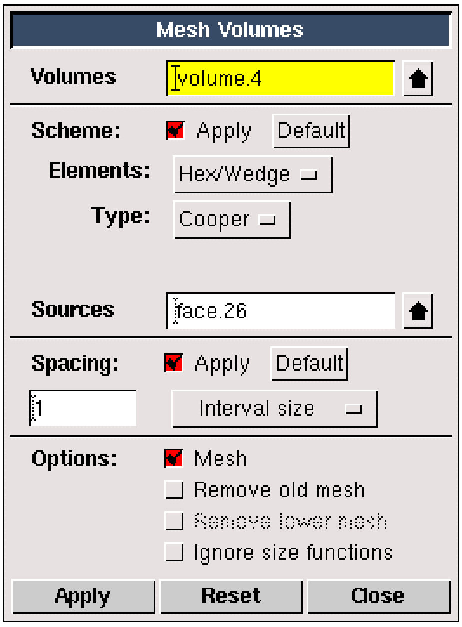
v. Click **Apply** in the **Set Face Vertex Type** form.

The vertex will be changed to **Type** *“S” for* Side. *A message will appear in the* **Transcript** *window stating that the vertex was set to type* Side.

b) Mesh the pipe volume using the Cooper meshing scheme.

MESH  → **VOLUME**  → **MESH VOLUMES** 

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



i. Select the first pipe volume in the graphics window.

Note that Hex/Wedge **Elements** *and the* Cooper **Type** *are automatically selected under* **Scheme** *in the* **Mesh Volumes** *form because you changed the vertex type on the wall face to* Side.

GAMBIT *automatically selects the source faces because you changed the vertex type on the wall face to* Side.

ii. Retain the default Interval size of `1` and click the **Apply** button at the bottom of the form.

The pipe will be meshed as shown in Figure 3-15.

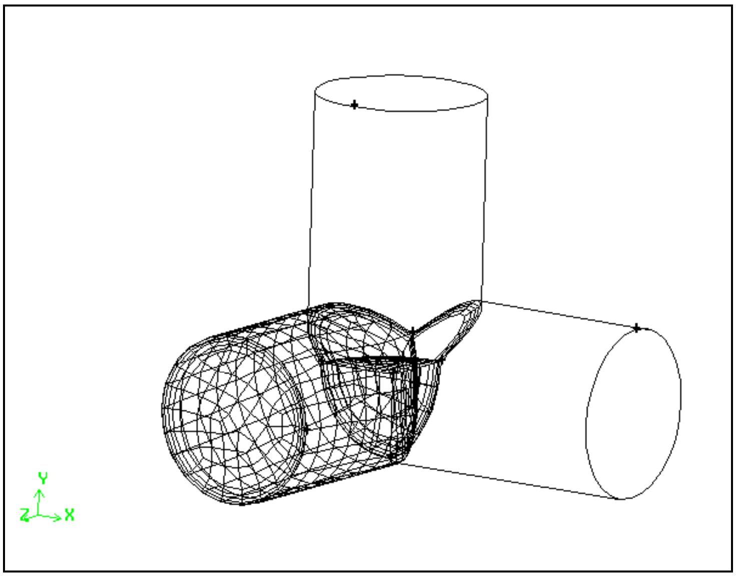


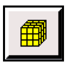
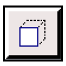
Figure 3-15: Pipe meshed by changing the vertex type on the wall face to **Side** and using the **Cooper** meshing scheme

! *It may be useful to remove the mesh from the display at this point; it is then easier to see the faces of the geometry for the other two pipes. The mesh is not deleted, just removed from the graphics window. To remove the mesh from the display, click the* **SPECIFY DISPLAY ATTRIBUTES** *command button*  *at the bottom of the* **Global Control** *toolpad. Select the* Off *radio button to the right of* **Mesh** *near the bottom of the form and click* **Apply**.

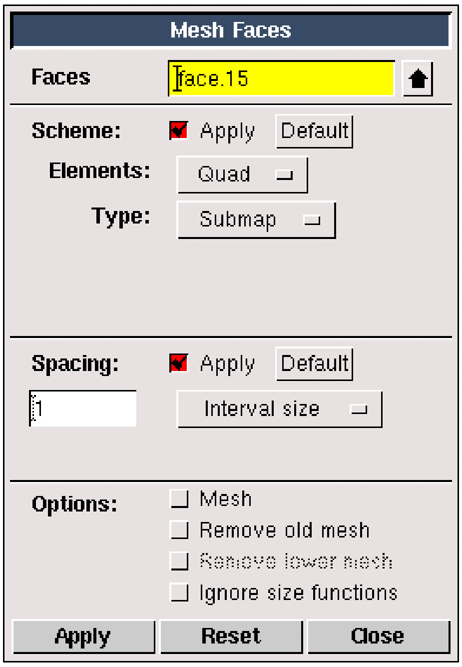
2. Mesh the second pipe using the Submap scheme on the wall face of the pipe and using the Cooper meshing scheme to mesh the volume.

   By enforcing the Submap *scheme on the wall face of the pipe,* GAMBIT *will automatically modify the vertex types on this face to honor the* Submap *scheme. The criteria for the* Cooper *meshing scheme will then be fulfilled for the pipe, and the pipe can be meshed using the* Cooper *scheme.*

   a) Set the meshing scheme for the wall face to Submap.

      MESH  → **FACE**  → **MESH FACES** 

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



i. Select the wall face of the second pipe (shown in Figure 3-16) in the graphics window.

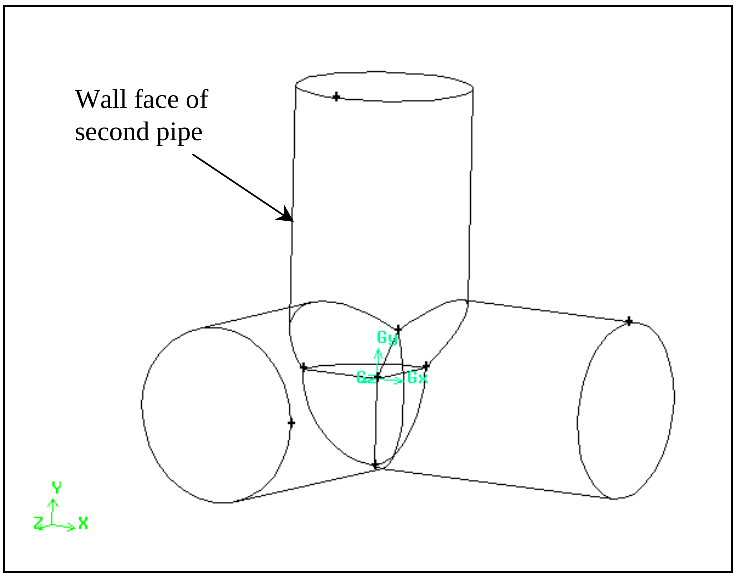


Figure 3-16: Wall face of the second pipe volume

ii. Select Quad in the **Elements** option menu under **Scheme**, and select Submap under **Type**.

See the GAMBIT *Modeling Guide for more information on the* Submap *scheme.*

iii. Retain the default Interval size of `1`.

iv. Unselect the Mesh check box under **Options**.

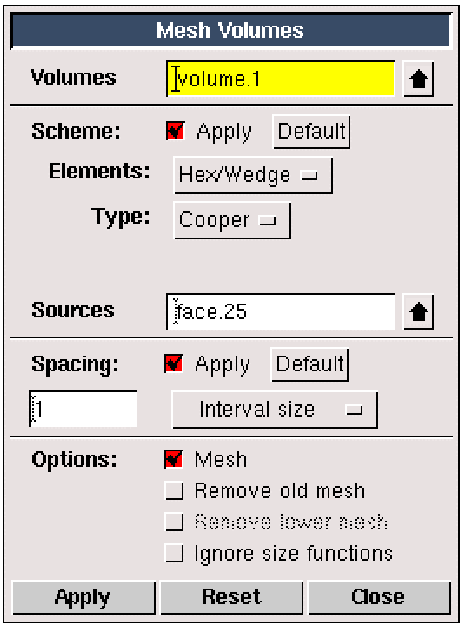
You unselected the Mesh *check box because at this point you do not want to mesh the face; you only want to apply the meshing* **Scheme** *to the face.* GAMBIT *will mesh the face using the specified* **Scheme** *when it meshes the pipe volume.*

v. Click the **Apply** button at the bottom of the form.

b) Mesh the pipe volume using the Cooper meshing scheme.

MESH $\longrightarrow$ **VOLUME** $\longrightarrow$ **MESH VOLUMES**

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



i. Select the pipe volume in the graphics window.

Note that Hex/Wedge **Elements** *and the* Cooper **Type** *are automatically selected under* **Scheme** *in the* **Mesh Volumes** *form because you enforced the* Submap *scheme on the side face of the pipe.*

GAMBIT *automatically selects the source faces because you enforced the* Submap *scheme on the side face of the pipe.*

ii. Retain the default Interval size of `1` under **Spacing** and click the **Apply** button at the bottom of the form.

The pipe will be meshed as shown in Figure 3-17.

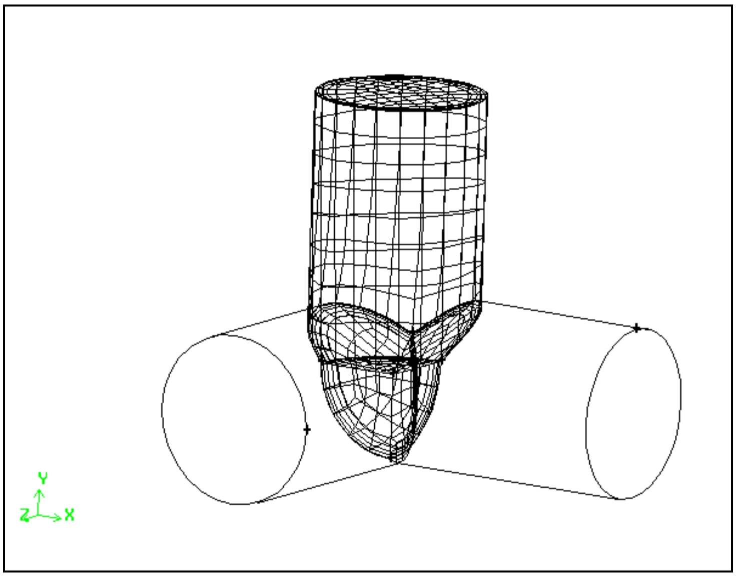
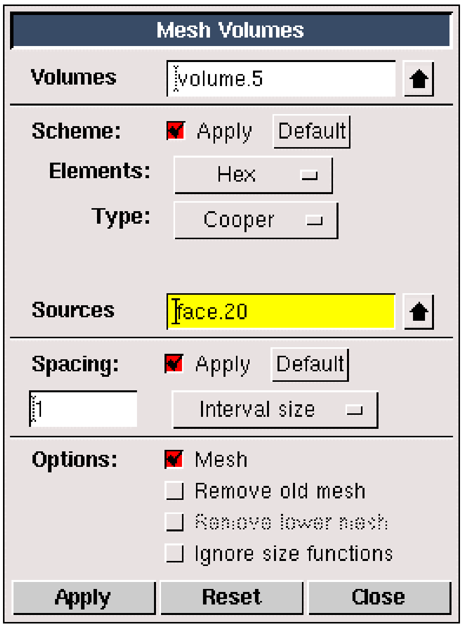


Figure 3-17: Pipe meshed using the Submap scheme for the wall face of the pipe and the project scheme for the volume

! *It may be useful to remove the mesh from the display at this point; it is then easier to see the faces of the geometry for the last pipe.*

3. Mesh one of the pipes by hand-picking the source faces and then using the Cooper meshing tool.



a) Select the third pipe in the graphics window.

The criteria for the Cooper *scheme are not fulfilled for this pipe, because* GAMBIT *cannot mesh the side face of the volume using the* Map *or* Submap *meshing schemes. However, you can force* GAMBIT *to use the* Cooper *scheme on this volume by selecting the* Cooper *scheme and then manually picking the source faces (the faces whose surface meshes are to be swept through the volume to form volume elements). When you click* **Apply**, GAMBIT *will automatically enforce the* Submap *scheme on the side face and modify the vertex types to honor the scheme selected. See the* GAMBIT *Modeling Guide for more information on using the* Cooper *meshing scheme.*

b) Select Hex in the **Elements** option menu under **Scheme**, and select Cooper under **Type**.

c) Left-click in the **Sources** list box in the form, and then pick the source faces for the mesh by selecting all the faces of the pipe except the pipe wall. The faces are marked A through D in Figure 3-18.

! *Shift*-middle-click *on a face to unselect it and select the face next to it. You can also click* **Reset** *in the* **Mesh Volumes** *form to unselect all faces and volumes, and reset all parameters entered in the form.*

The four faces to be selected are at opposite ends of the pipe, as shown in Figure 3-18. You can select the faces in the graphics window, or you can use the **Sources** *pick list.*

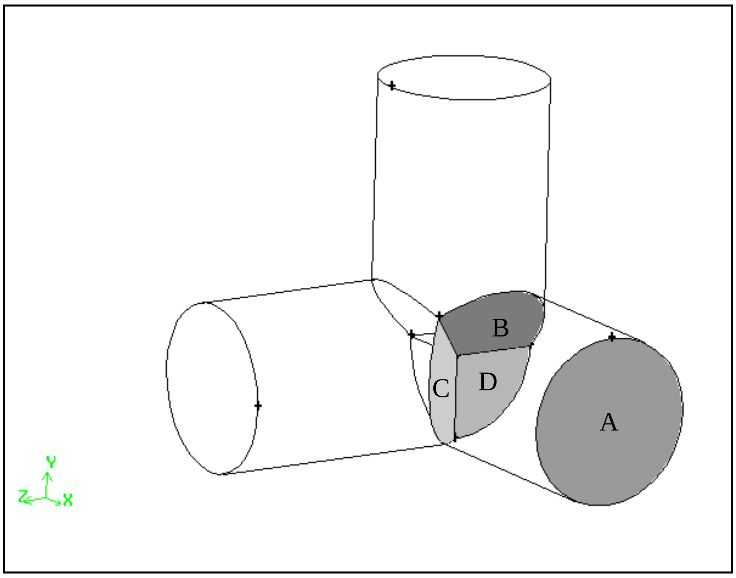
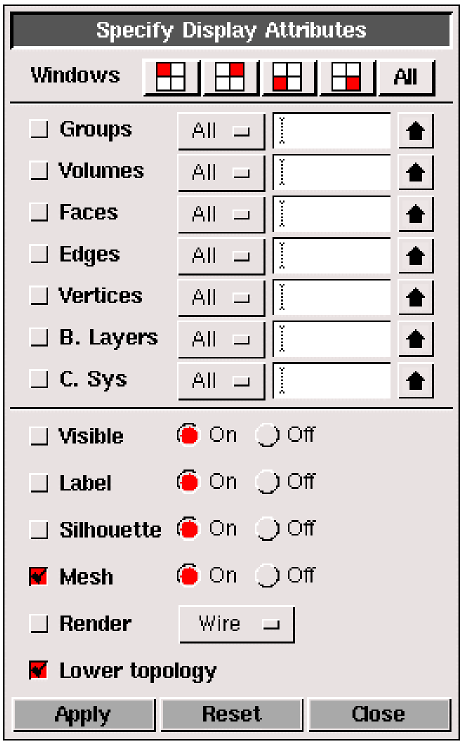


Figure 3-18: Source faces used to mesh one of the pipe volumes using the Cooper meshing scheme

d) Retain the default Interval size of `1` under **Spacing** and click the **Apply** button at the bottom of the form.

4. Display the full mesh for the three-pipe intersection.

a) Click the **SPECIFY DISPLAY ATTRIBUTES** command button  at the bottom of the **Global Control** toolpad to open the **Specify Display Attributes** form.



b) Select the **Mesh:On** option near the bottom of the form.

c) Click **Apply** to display the mesh, then **Close** the form.

The mesh for the three-pipe intersection is shown in Figure 3-19.

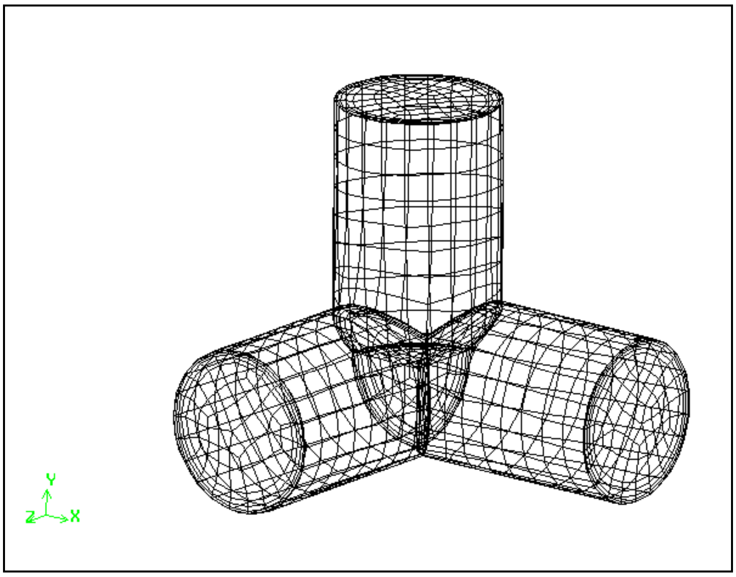
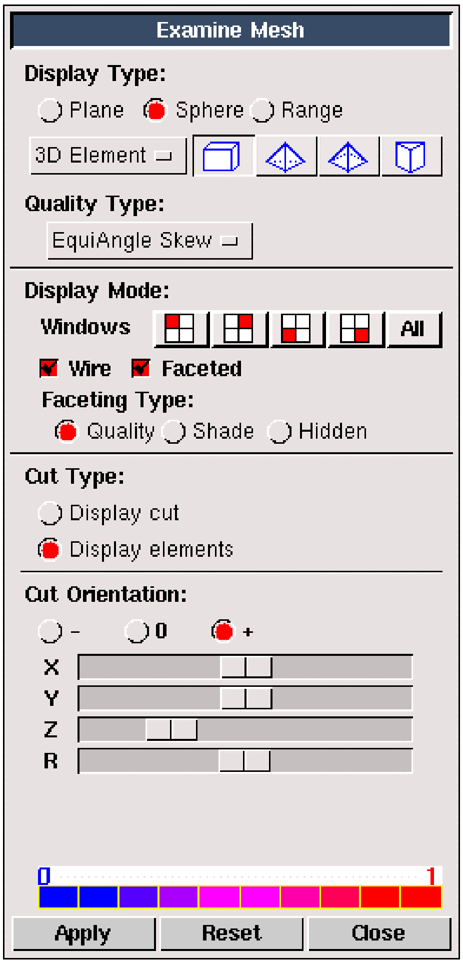


Figure 3-19: Face meshes for the three-pipe intersection

Step 9: Examine the Quality of the Mesh

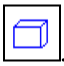
1. Select the **EXAMINE MESH**  command button at the bottom right of the **Global Control** toolpad.

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



   a) Select the Sphere option under **Display Type**.

This creates a section through the grid that is spherical in shape. For the three-pipe geometry, a spherical section displays more useful information than a planar section.

The 3D Element *type selected by default at the top of the form is a brick* .

b) Select or retain EquiAngle Skew from the **Quality Type** option menu.

c) Select the **+** option under **Cut Orientation** near the bottom of the form.

*The "**+**" option indicates that only elements on the positive side of the cut are displayed. For a sphere, this means that only elements on the inside of the sphere will be visible. The "**0**" option displays elements on the cut, and the "**-**" option displays elements on the negative side of the cut (the outside of the sphere in this case).*

d) Click the **SELECT PRESET CONFIGURATION**  command button in the **Global Control** toolpad.

This divides the graphics window into four quadrants and displays a different view of the spherical section of the grid in each quadrant.

e) Hold down the left mouse button on the **X** slider box in the **Examine Mesh** form and move it until the spherical cut is centered in the x direction in the graphics window.

f) Move the **Y** and **Z** slider boxes to center the spherical cut in the y and z directions in the graphics window. The graphics window display is shown in Figure 3-20.

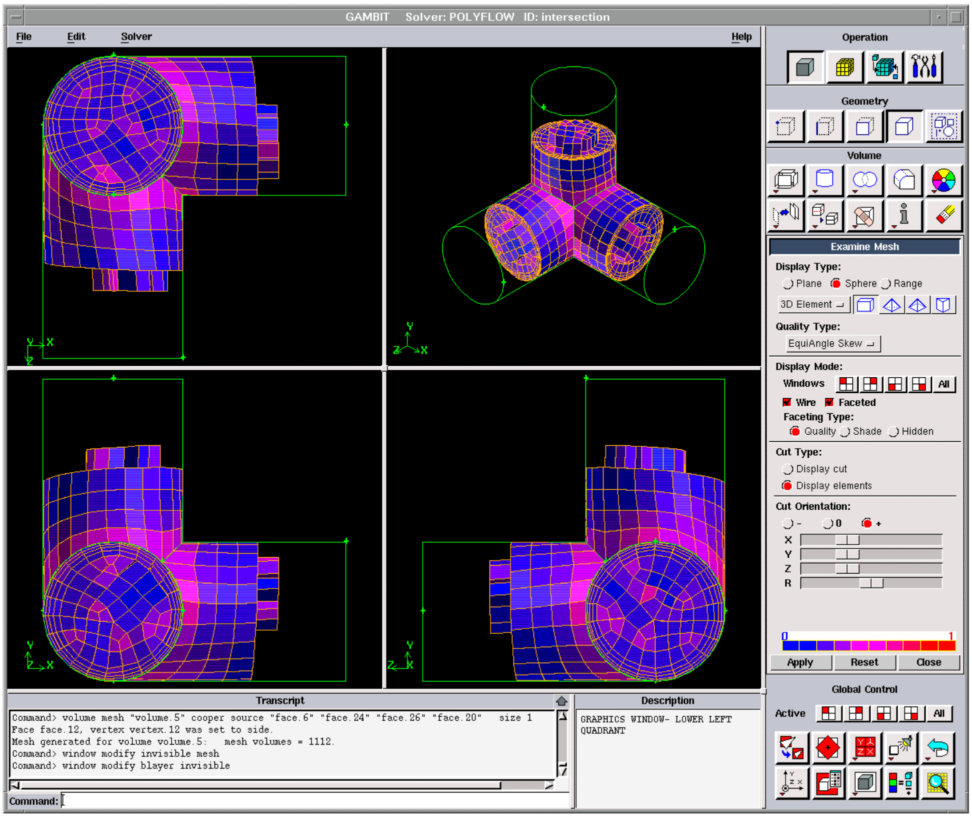


Figure 3-20: Spherical cut centered in the x, y, and z directions

g) Move the **R** slider box in the **Examine Mesh** form to view the mesh on different spherical cuts in the graphics window.

h) Hold down the left mouse button on the **GRAPHICS-WINDOW SASH** anchor, the small gray box in the center of the four quadrants of the graphics window, and drag it diagonally across the graphics window to the bottom right corner.

This restores a single window.

i) Close the **Examine Mesh** form.

The spherical cut of the mesh will be removed and the face meshes will be restored.

Step 10: Set Boundary Types

1. Remove the mesh and boundary layers from the display before you set the boundary types.

   This makes it easier to see the edges and faces of the geometry. The mesh and boundary layers are not deleted, just removed from the graphics window.

   a) Click the **SPECIFY DISPLAY ATTRIBUTES** command button  at the bottom of the **Global Control** toolpad.

   b) Select the Off radio button to the right of **Mesh** near the bottom of the form.

   c) Click **Apply**.

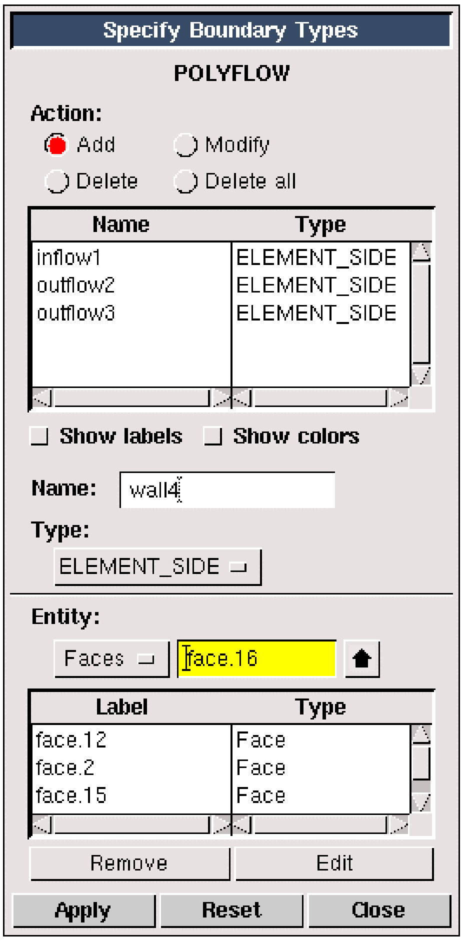
   d) Select the check box to the left of **B. Layers** and select Off from the option menu to the right of **Visible** near the bottom of the form.

   e) Click **Apply** and close the form.

2. Set boundary types for the three-pipe intersection.

   ZONES  → **SPECIFY BOUNDARY TYPES** 

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



a) Define the flow inlet.

i. Enter the name `inflow1` in the **Name** text entry box.

> **POLYFLOW** *boundary and continuum names require number suffixes (such as "*`1`*" in the name "*`inflow1`*"). The numbers should be assigned sequentially, as illustrated in this tutorial. (See the POLYFLOW Users' Guide and online FAQs for more information.)*

ii. Select ELEMENT_SIDE in the **Type** option menu.

> *The specific boundary types will be defined inside the* **POLYFLOW** *solver.*

iii. Check that **Faces** is selected as the **Entity**.

iv. *Shift*-left-click the end of one of the pipes (the face marked A in Figure 3-21) and click **Apply** to accept the selection.

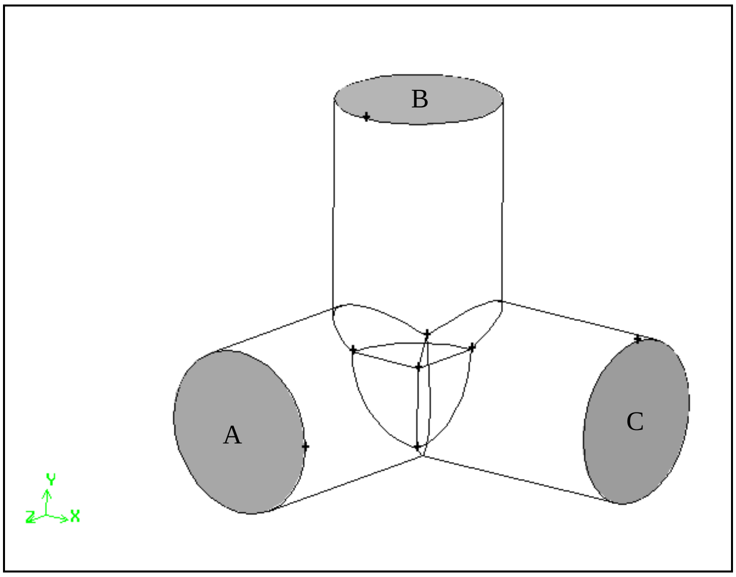


Figure 3-21: Boundary types for faces of the three-pipe intersection

b) Define the two flow outlets.

i. Enter the name `outflow2` in the **Name** text entry box.

ii. Check that ELEMENT_SIDE is still selected in the **Type** option menu and *Shift*-left-click the end of one of the other pipes in the graphics window (the face marked B in Figure 3-21).

iii. Click **Apply** to accept the selection of the face.

iv. Set the **Type** for the end of the third pipe (the face marked C in Figure 3-21) to be ELEMENT_SIDE, using the **Name** `outflow3`.

c) Define wall boundary types for the walls of the three-pipe intersection.

i. Enter the name `wall4` in the **Name** text entry box.

ii. Check that ELEMENT_SIDE is still selected in the **Type** option menu and pick all of the wall faces (the outer walls of the three pipes and the outer face of the sphere octant) in the graphics window.

 ! *You will select four faces in total.*

iii. Click **Apply** to accept the selection of the faces.

 The boundaries for the three-pipe intersection are shown in Figure 3-22. (NOTE: To display the boundary types in the graphics window, select the **Show labels** *options on the* **Specify Boundary Types** *form.)*

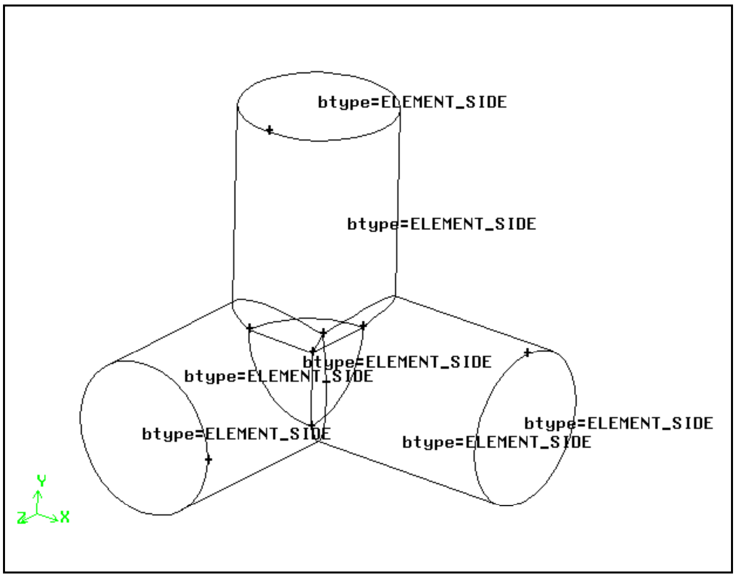


Figure 3-22: Boundary types for the three-pipe intersection

Note that when GAMBIT *writes a mesh, any volumes (in 3-D) on which you have not specified a continuum type will be written as* FLUID *by default. This means that you do not need to specify a continuum type in the* **Specify Continuum Types** *form for this tutorial.*

Step 11: Export the Mesh and Save the Session

1. Export a mesh file for the three-pipe intersection.

   File → **Export** → **Mesh...**

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

   a) Enter the **File Name** for the file to be exported (`intersection.neu`).

   b) Click **Accept**.

      The file will be written to your working directory.

2. Save the GAMBIT session and exit GAMBIT.

   File → **Exit**

   GAMBIT *will ask you whether you wish to save the current session before you exit.*

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

3.5 Summary

In this tutorial, you created geometry consisting of three intersecting pipes. Before creating the mesh, you decomposed the three-pipe geometry into four volumes: the three individual pipes and the wedge-shaped corner of the intersection (the sphere “octant”). These constituent volumes were readily meshed using GAMBIT’s Cooper and Tet Primitive meshing schemes.

4. MODELING A COMBUSTION CHAMBER (3-D)

In this tutorial, you will create the geometry for a burner using a top-down geometry construction method in GAMBIT (creating a volume using solids). You will then mesh the burner geometry with an unstructured hexahedral mesh.

In this tutorial you will learn how to:

- Move a volume
- Subtract one volume from another
- Shade a volume
- Intersect two volumes
- Blend the edges of a volume
- Create a volume using the sweep face option
- Prepare the mesh to be read into FLUENT 5/6

4.1 Prerequisites

This tutorial assumes that you have worked through Tutorial 1 and you are consequently familiar with the GAMBIT interface.

4.2 Problem Description

The problem to be considered is shown schematically in Figure 4-1. The geometry consists of a simplified fuel injection nozzle that feeds into a combustion chamber. You will only model one quarter of the burner geometry in this tutorial, because of the symmetry of the geometry. The nozzle consists of two concentric pipes with radii of 4 units and 10 units respectively. The edges of the combustion chamber are blended on the wall next to the nozzle.

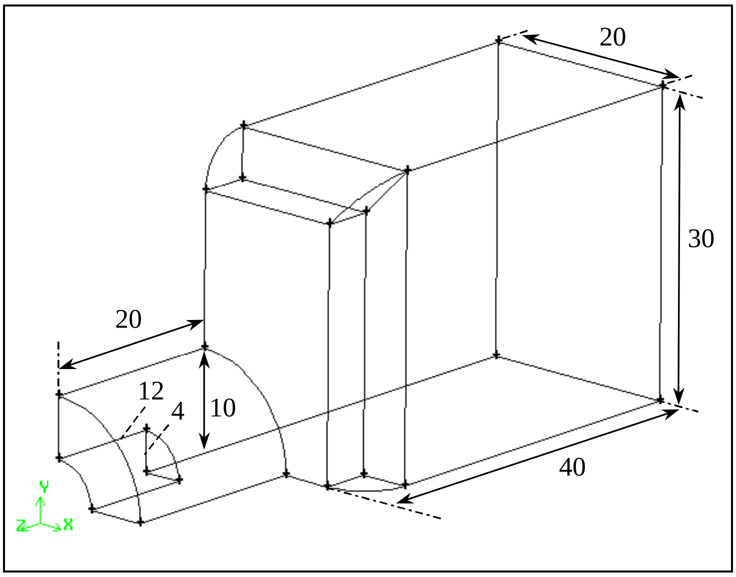


Figure 4-1: Problem specification

4.3 Strategy

In this tutorial, you will create a combustion chamber geometry using the “top-down” construction method. You will create volumes (in this case, bricks and cylinders) and use Boolean operations to unite, intersect, and subtract these volumes to obtain the basic geometry. Finally, using the “blend” command, you will round off some edges to complete the geometry creation.

For this model, it is not possible to simply pick the geometry and mesh the entire domain with hexahedral elements, because the Cooper tool (which you will be using in this tutorial) requires two groups of faces, one group topologically parallel to a sweep path, and the other group topologically perpendicular. However, the rounded (blended) edges fit in neither group. See the GAMBIT Modeling Guide for a more detailed description of the Cooper tool. You need to decompose the geometry into portions that can be meshed using the Cooper tool. There are several ways to decompose geometry in GAMBIT. In this example, you will use a method whereby portions of the volume around the blend are split off from the main volume. A detailed description of the decomposition strategy for the combustion chamber is given below.

Note that there are several faces in the geometry for which the default meshing scheme is the Pave scheme; most of these faces are perpendicular to the z direction. There are also geometrical protrusions in the z direction, so this should be chosen as the main direction for the Cooper meshing scheme. To make this possible, the paved faces in the x and y directions (the two symmetry planes in the geometry shown in Figure 4-2) must be changed to use the Submap or Map meshing scheme.

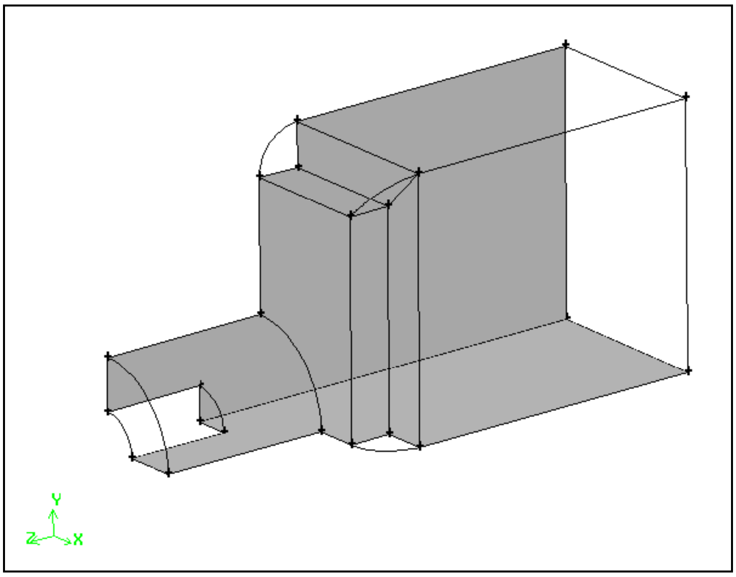


Figure 4-2: The two symmetry planes in the combustion chamber geometry

By default, GAMBIT selects the Pave meshing scheme for these two faces because each has a rounded edge where the blend occurs. If you split off the rounded corners of both faces and connect them through a volume, you can use the Submap meshing scheme on the remaining faces, and hence the Cooper meshing scheme for the volume.

Instead of creating two faces, one on each symmetry plane, you will create a face at the junction of the two blended edges (face A in Figure 4-3). This face will then be swept in two directions onto the symmetry planes (creating faces B and C in Figure 4-3), to split the volume into three parts. The three volumes can then be meshed individually using the Cooper tool.

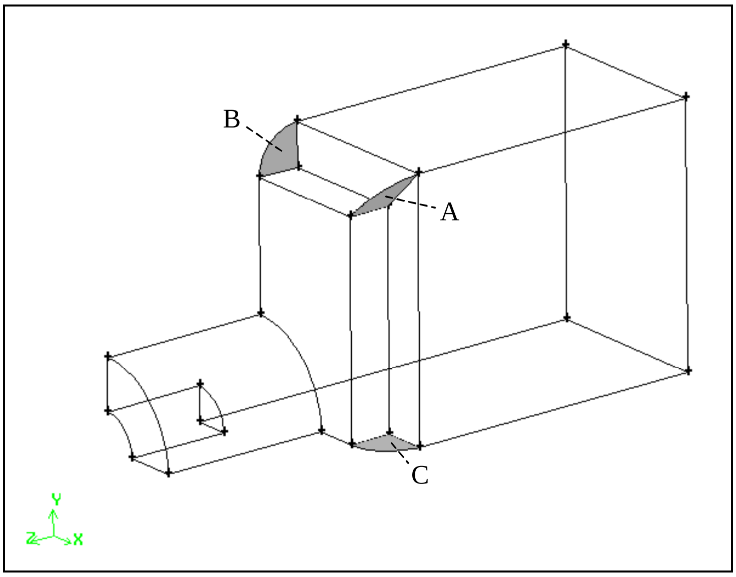


Figure 4-3: Faces created at the blended edges and on the symmetry planes

This tutorial also demonstrates a few ways of controlling the mesh density and the meshing schemes used on individual faces. You will mesh the small quarter-circle face that forms the second inlet with a Tri Primitive scheme and a finer mesh size. Similarly, you will mesh the annular face of the primary inlet with a fine mapped mesh. To meet the requirements of the Cooper tool, you will also need to create a mapped mesh on the face between these two faces. Finally, you will use the automatic Cooper tool to mesh the remaining faces and the volume.

4.4 Procedure

Start GAMBIT.

Step 1: Select a Solver

1. Choose the solver you will use to run your CFD calculation by selecting the following from the main menu bar:

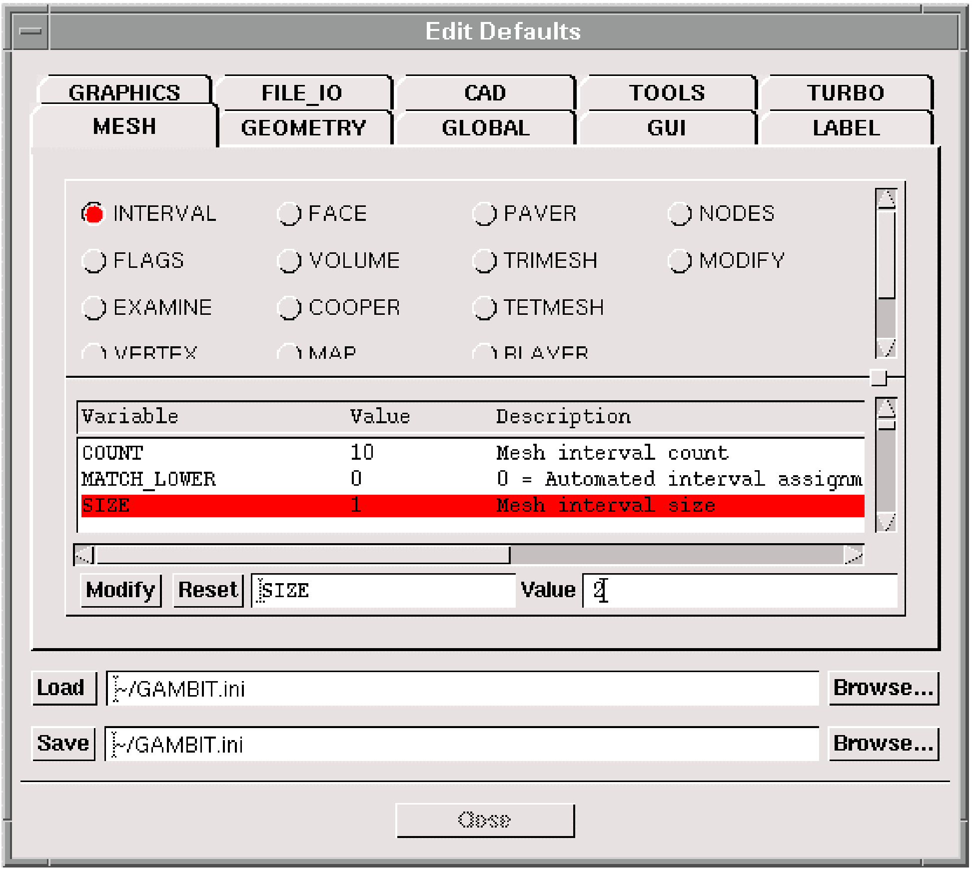
   Solver → **FLUENT 5/6**

   The choice of a solver dictates the options available in various forms (for example, the boundary types available in the **Specify Boundary Types** *form). For some systems,* **FLUENT 5/6** *is the default solver. The solver currently selected is indicated at the top of the* GAMBIT *GUI.*

Step 2: Set the Default Interval Size for Meshing

In this tutorial, you will change the default interval size used for meshing. The mesh spacing is, by default, based on the interval size parameter, which you will modify in the **Edit Defaults** *form. The value you enter should be the estimated average size of a mesh element in the model. This value will appear as the default* Interval size *on all meshing forms. You will be able to change it on the meshing forms, if required.*

Edit → **Defaults...**



1. Select the **MESH** tab at the top of the form.

2. Select the INTERVAL radio button near the top of the form.

3. Select `SIZE` in the `Variable` list.

   `SIZE` *will appear in the space at the bottom of the list and its default value will appear in the* **Value** *text entry box.*

4. Enter a value of 2 in the **Value** text entry box.

5. Click the **Modify** button to the left of `SIZE`.

   The **Value** *of the variable* `SIZE` *will be updated in the list.*

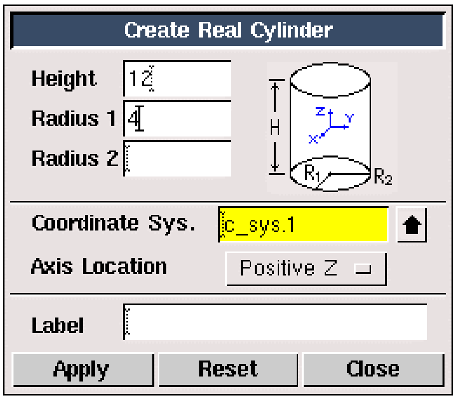
6. Close the **Edit Defaults** form.

Step 3: Create Two Cylinders

1. Create a cylinder to form the opening of the burner.

   GEOMETRY → **VOLUME** → **CREATE VOLUME** R Cylinder

   This command sequence opens the **Create Real Cylinder** *form.*



   a) Enter a value of `12` for the **Height** of the cylinder.

   b) Enter a value of `4` for **Radius 1** of the cylinder.

      The text entry box for **Radius 2** *can be left blank;* GAMBIT *will set this by default to be the same value as* **Radius 1**.

   c) Select Positive Z (the default) as the **Axis Location**.

   d) Click **Apply**.

2. Repeat the steps above to create a cylinder of **Height** = `20` and **Radius 1** = `10` along the Positive Z axis.

3. Click the **FIT TO WINDOW** command button, at the top left of the **Global Control** toolpad, to see the cylinders created.

   The two cylinders are shown in Figure 4-4. Hold down the left mouse button and move the mouse to rotate the view in the graphics window. You can zoom out from the current view by holding down the right mouse button and pushing the mouse away from you.

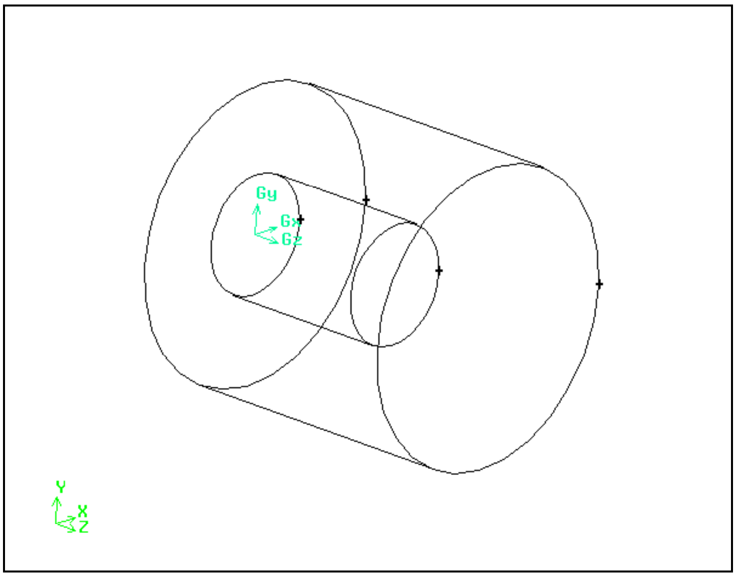
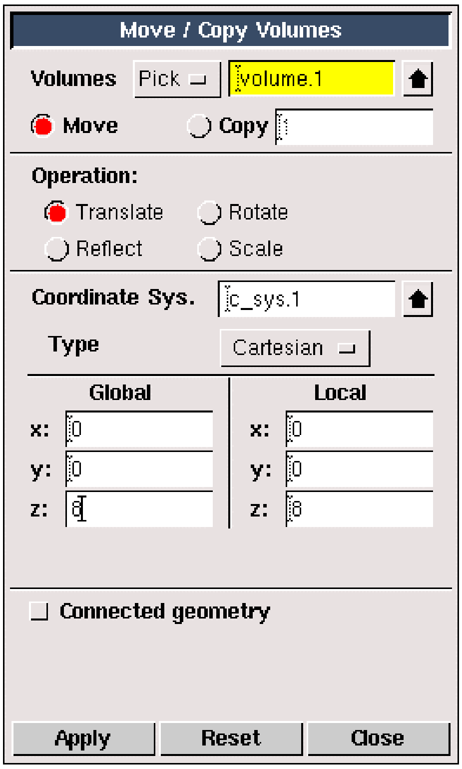


Figure 4-4: Two cylinders

4. Move the first cylinder you created so that it is at the front of the large cylinder.

GEOMETRY → **VOLUME** → **MOVE/COPY/ALIGN VOLUMES**

This command sequence opens the **Move / Copy Volumes** *form.*



a) *Shift*-left-click the small cylinder in the graphics window.

"volume.1" will be entered next to **Volumes** *in the* **Move / Copy Volumes** *form.*

b) Select **Move** (the default) under **Volumes** in the **Move / Copy Volumes** form.

c) Select Translate (the default) under **Operation**.

d) Enter a **Global** translation vector of (`0`, `0`, `8`) to move the cylinder 8 units in the z direction.

Note that GAMBIT *automatically fills in the values under* **Local** *as you enter values under* **Global**.

e) Click **Apply**.

The two cylinders are shown in Figure 4-5. Notice that the small cylinder has been moved from the back of the large cylinder to the front.

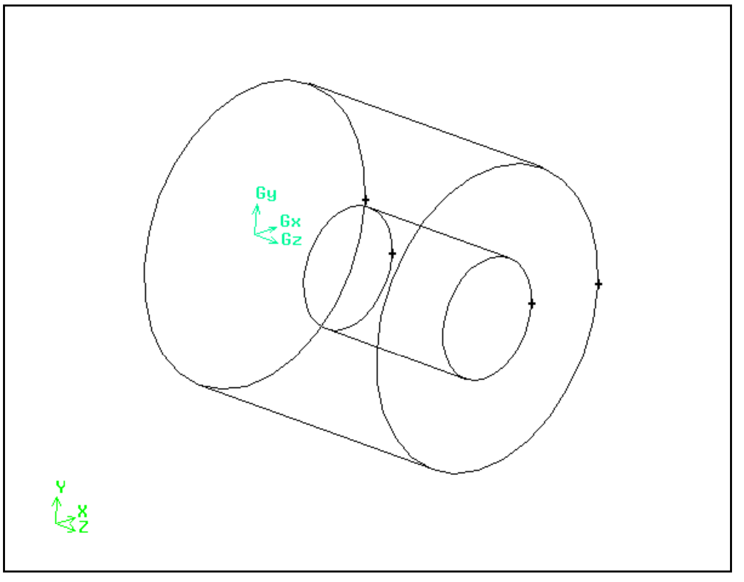


Figure 4-5: Two cylinders after moving the small cylinder

Step 4: Subtract the Small Cylinder From the Large Cylinder

1. Create one volume from the two cylinders by subtracting one cylinder from the other.

    The order of selecting the volumes is important. For example, Figure 4-6 shows the difference between subtracting volume B from volume A, and vice versa.

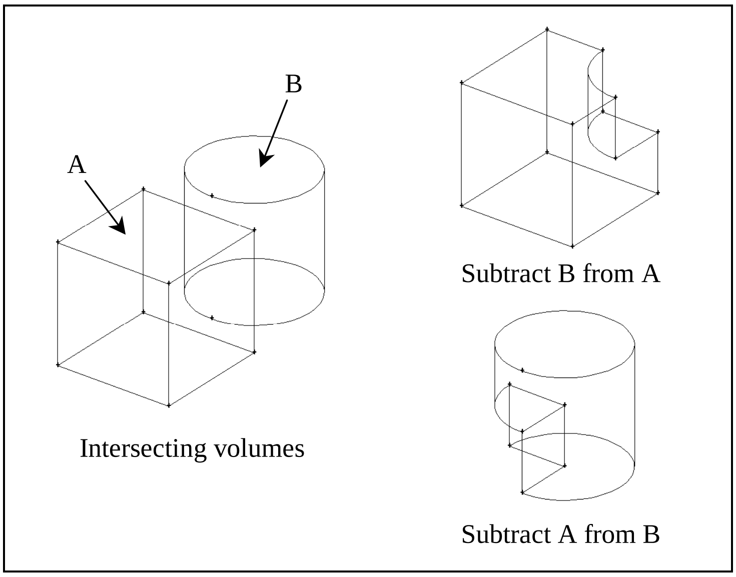
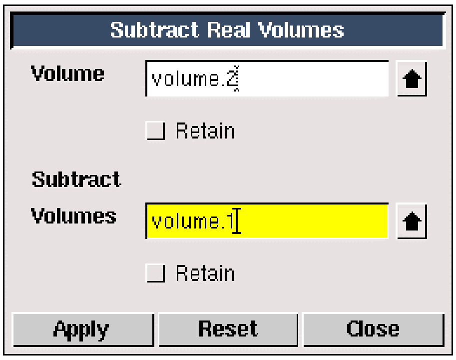


Figure 4-6: Subtracting volumes

GEOMETRY → **VOLUME** → **BOOLEAN OPERATIONS** R Subtract

This command sequence opens the **Subtract Real Volumes** *form.*



a) *Shift*-left-click the large cylinder in the graphics window.

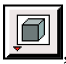
b) Left-click in the list box to the right of **Subtract Volumes** to accept the selection of *volume.2* and make the **Subtract Volumes** list box active.

! *Alternatively, you could continue to hold down the Shift key and click the right mouse button in the graphics window to accept the selection of the large cylinder and move the focus to the* **Subtract Volumes** *list box.*

c) Select the small cylinder and accept the selection.

Selecting the cylinders in this order ensures that the small cylinder is subtracted from the large cylinder and not vice versa.

Step 5: Shade and Rotate the Display

1. Click the **RENDER MODEL** command button , in the middle of the bottom row of the **Global Control** toolpad, to create a shaded view of the volume.

2. Hold down the left mouse button and drag the mouse to rotate the graphics display and see the cylindrical hole created in the large cylinder (see Figure 4-7).

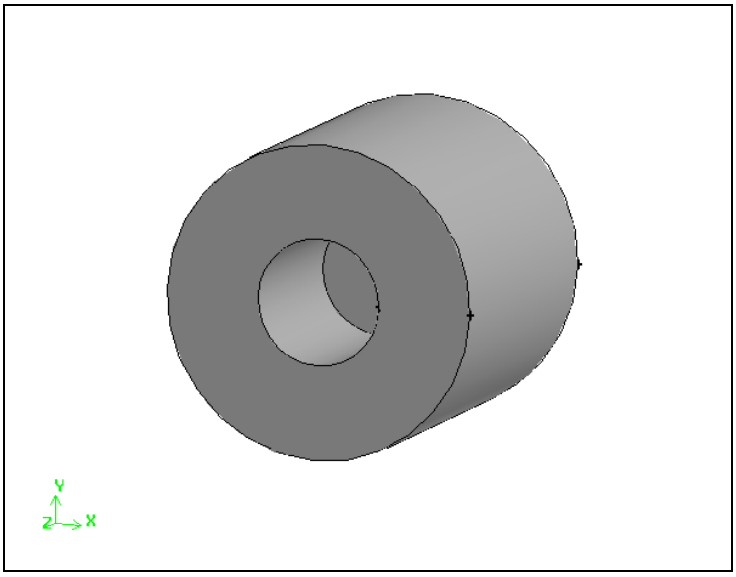


Figure 4-7: Shaded geometry showing hole in large cylinder

3. To return to the unshaded view, right-click on the **RENDER MODEL** command button  in the **Global Control** toolpad and select 

from the resulting list.

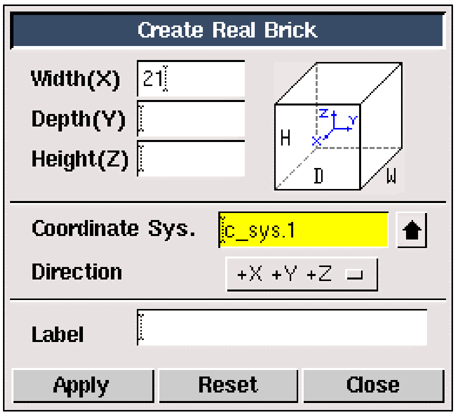
Step 6: Remove Three Quarters of the Cylindrical Volume

In this step, you will create a brick that will be intersected with the cylindrical volume. Three quarters of the cylindrical volume will be removed, leaving the volume for the entrance of the burner.

1. Create a brick that will be intersected with the cylindrical volume already created.

   GEOMETRY → **VOLUME** → **CREATE VOLUME** R Brick

   This command sequence opens the **Create Real Brick** *form.*



   a) Enter a value of `21` for the **Width** of the brick.

      The text entry boxes for **Depth** *and* **Height** *can be left blank;* GAMBIT *will set these values by default to be the same value as the* **Width**, *to create a cube.*

   b) Use a **Direction** of +X +Y +Z (the default).

   c) Click **Apply**.

      Figure 4-8 shows the cylindrical volume and the brick.

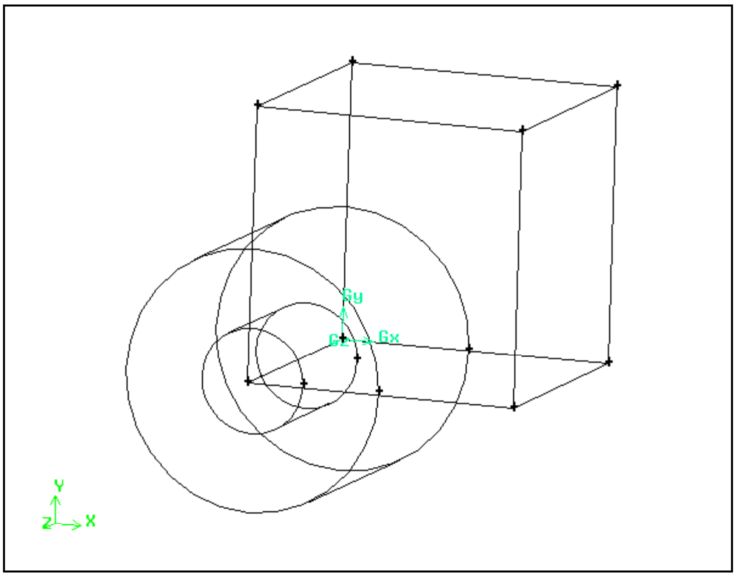
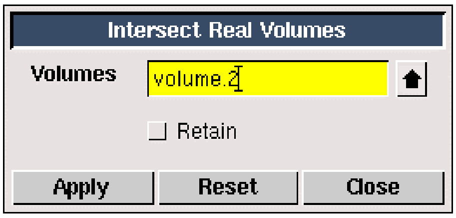


Figure 4-8: Cylindrical volume and brick

2. Intersect the brick and the cylindrical volume.

GEOMETRY → **VOLUME** → **BOOLEAN OPERATIONS** R Intersect

This command sequence opens the **Intersect Real Volumes** *form.*



a) *Shift*-left-click the brick in the graphics window.

b) Select the cylindrical volume in the graphics window.

c) Click **Apply** to accept the selection of the volumes.

The order in which the two volumes are selected is not important to the outcome of the operation but does *affect the numbering of subsequent geometric entities. The cylindrical volume will be trimmed so that only the part inside the brick remains, as shown in Figure 4-9.*

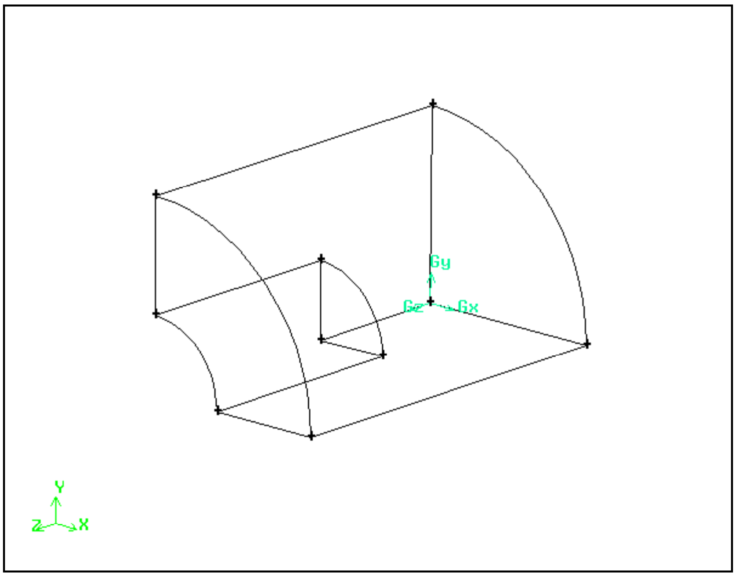
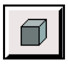
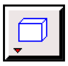


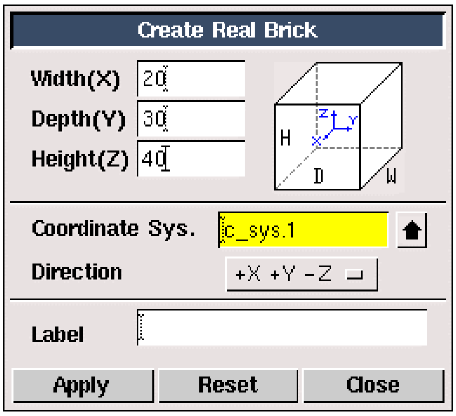
Figure 4-9: One quarter of the cylindrical volume remains

Step 7: Create the Chamber of the Burner

1. Create a brick for the chamber.

   GEOMETRY  → **VOLUME**  → **CREATE VOLUME** 

   This command sequence opens the **Create Real Brick** *form.*

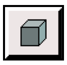
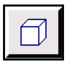
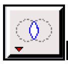
   a) Enter a value of `20` for the **Width** of the brick, `30` for the **Depth**, and `40` for the **Height**.

   b) Change the **Direction** to +X +Y -Z by selecting this option in the option menu to the right of **Direction**.

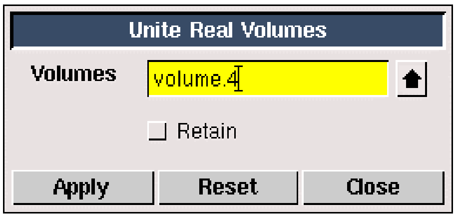
   c) Click **Apply**.

2. Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the brick created.

3. Unite the brick and cylindrical volume into a single volume.

GEOMETRY  → **VOLUME**  → **BOOLEAN OPERATIONS** R 

   This command sequence opens the **Unite Real Volumes** *form.*



a) *Shift*-left-click the cylindrical volume in the graphics window.

b) Select the brick and click **Apply** to accept the selection.

The order in which you select the two volumes is not important when you are uniting them. The brick and the cylindrical volume will be united as shown in Figure 4-10.

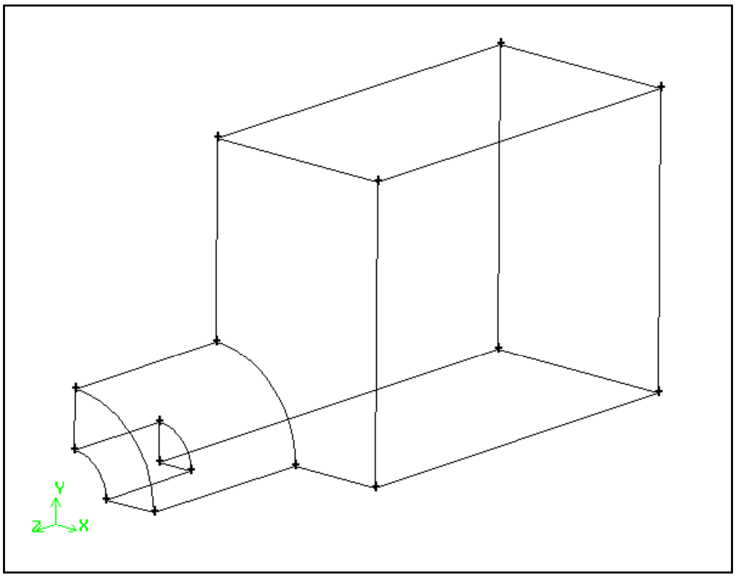


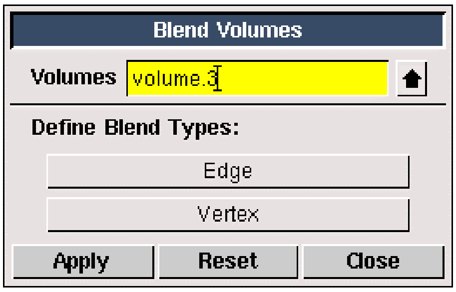
Figure 4-10: Brick and cylindrical volume are united

Step 8: Blend the Edges of the Chamber

1. Blend (round off) two edges of the chamber geometry to give it a more rounded shape.

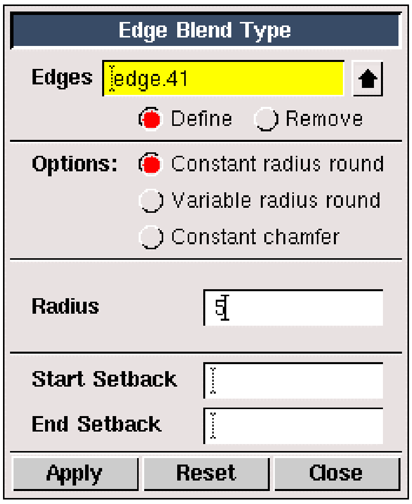
   GEOMETRY → **VOLUME** → **BLEND VOLUMES**

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



   a) Click the **Edge** button under **Define Blend Types**.

      This action opens the **Edge Blend Type** *form.*



      i. *Shift*-left-click the two edges to be blended, as shown in Figure 4-11.

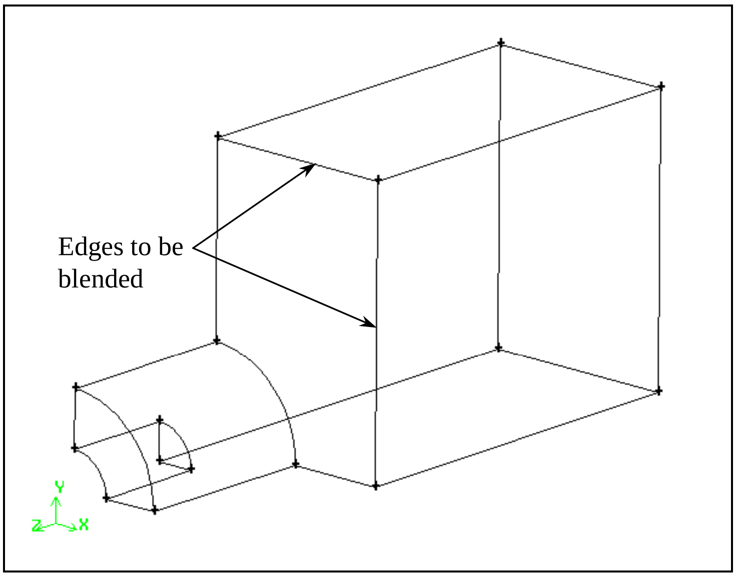


Figure 4-11: Edges to be blended

ii. Select Constant radius round (the default) under **Options** in the **Edge Blend Type** form.

iii. Enter 5 for the **Radius**.

iv. Click **Apply** in the **Edge Blend Type** form and **Close** the form.

b) *Shift*-left-click the volume in the graphics window.

c) Click **Apply** in the **Blend Volumes** form.

The burner geometry with the blended edges is shown in Figure 4-12.

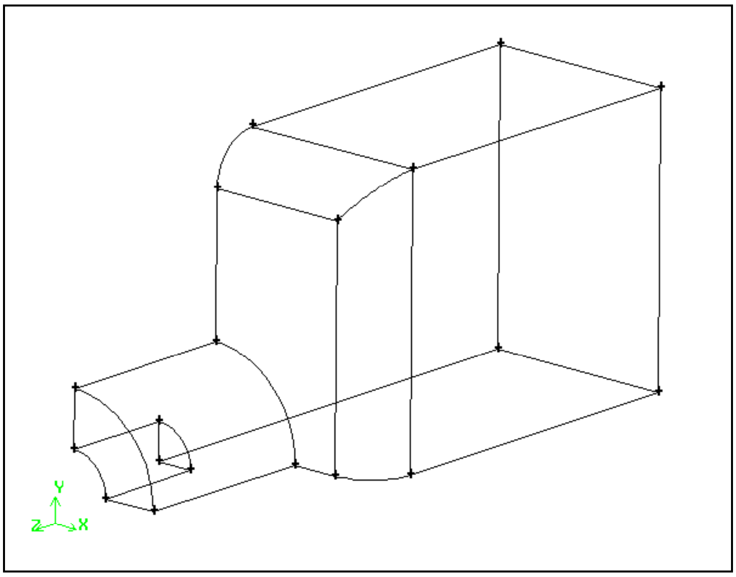
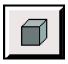
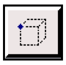
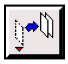


Figure 4-12: Burner with blended edges

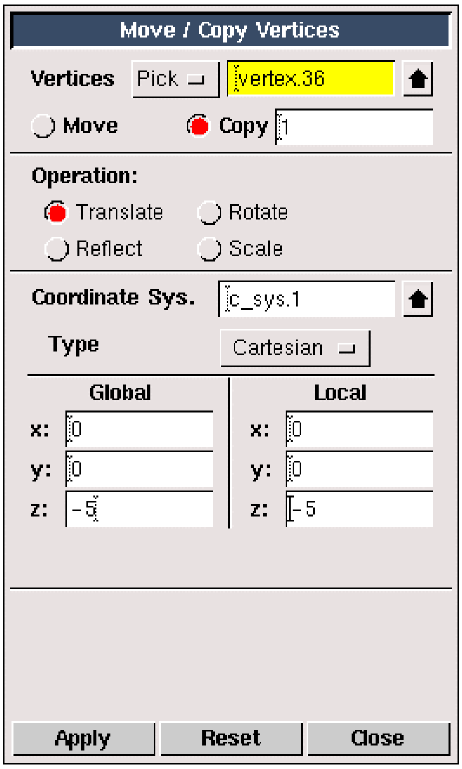
Step 9: Decompose the Geometry

For this model, it is not possible to simply pick the geometry and mesh it with a hexahedral mesh. The Cooper *meshing scheme requires that all "source" faces are topologically parallel, and that all other faces can be meshed using the* Map *or* Submap *meshing scheme. However, the curved faces resulting from the blend operation do not satisfy the* Cooper *criteria. Therefore, you will need to decompose the geometry into portions that are each suitable for the* Cooper *tool. There are several ways to decompose geometry in* GAMBIT. *In this example, you will use a method whereby portions of the volume around the blend are split off from the main volume. To do this, you will create a vertex near the junction of the two blended edges. You will then use this vertex to create straight edges, and use these edges to create a face. This face will then be swept in two directions to create two volumes. These two volumes will be used to split the burner volume into three parts. It will then be possible to mesh each of these parts individually using the* Cooper *tool.*

1. Create a vertex inside the volume.

   GEOMETRY  → **VERTEX**  → **MOVE/COPY/ALIGN VERTICES** 

   This command sequence opens the **Move / Copy Vertices** *form.*



a) Select the vertex marked A in Figure 4-13.

To zoom in to an area of the graphics window, hold down the Ctrl *key and use the left mouse button to draw a box around the area you want to view.*

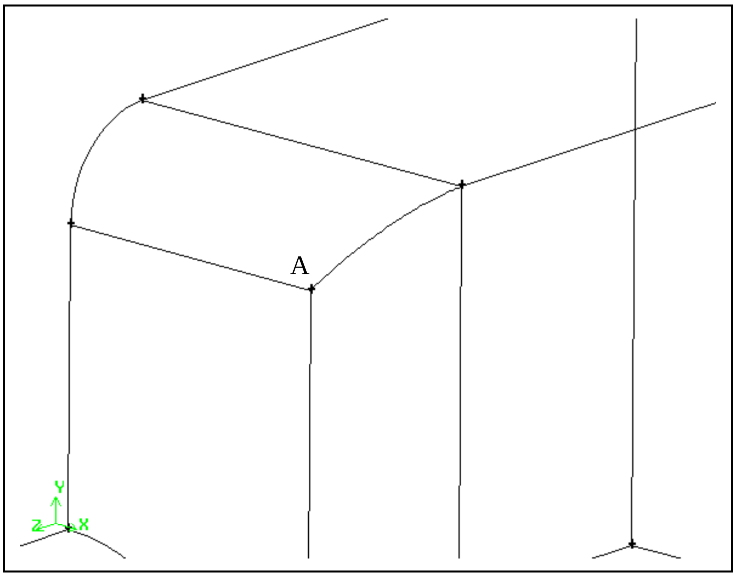


Figure 4-13: Vertex to copy

b) Select **Copy** under **Vertices** in the **Move / Copy Vertices** form.

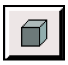
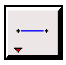
c) Select Translate (the default) under **Operation**.

d) Enter the translation vector (`0`, `0`, `-5`) under **Global**.

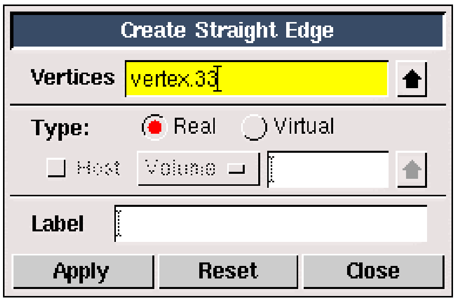
e) Click **Apply**.

The vertex will be visible in the graphics window as a white cross near where the two blended edges meet. See vertex B in Figure 4-14.

2. Create two straight edges using the new vertex.

GEOMETRY  → **EDGE**  → **CREATE EDGE** 

This command sequence opens the **Create Straight Edge** *form.*



a) *Shift*-left-click the vertex marked A in Figure 4-14.

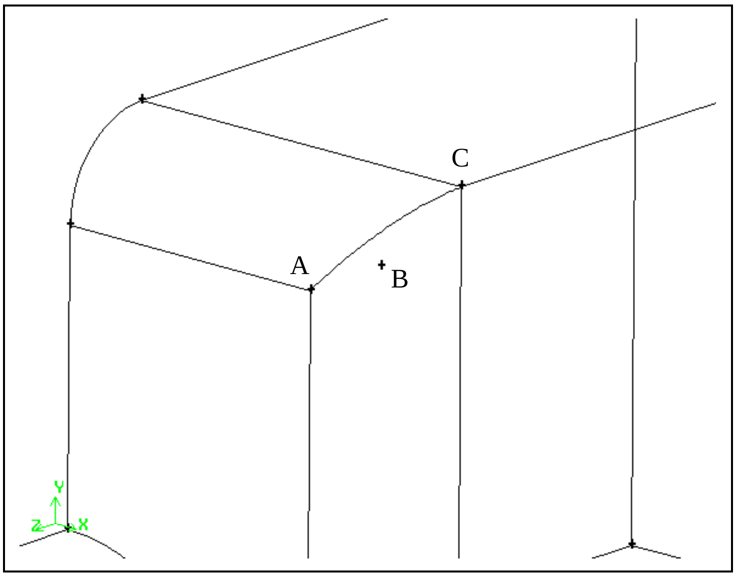


Figure 4-14: Vertices to be selected to create edges

b) *Shift*-left-click the vertices marked B and C in Figure 4-14, in order.

c) Click **Apply** to accept the selected vertices and create two edges.

The edges are shown in Figure 4-15.

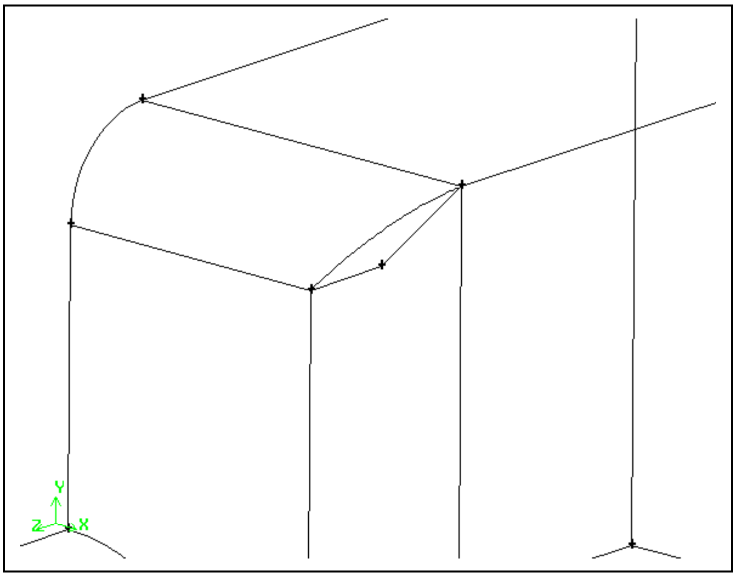
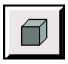
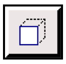
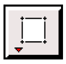
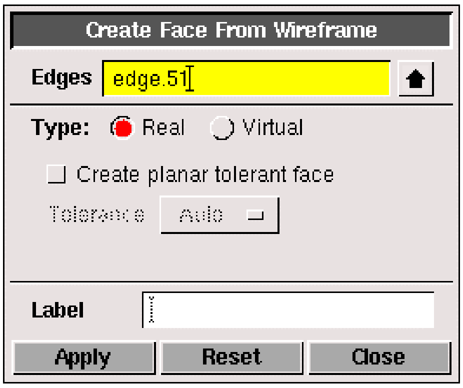


Figure 4-15: Two new straight edges

3. Create a face using the two new edges.

GEOMETRY  → **FACE**  → **FORM FACE** 

This command sequence opens the **Create Face From Wireframe** *form.*



a) *Shift*-left-click the edge marked D in Figure 4-16.

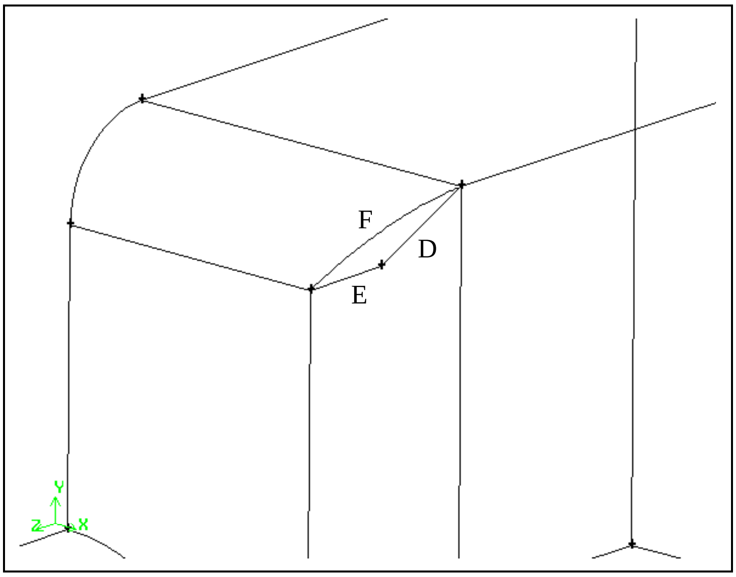
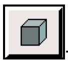
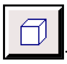
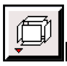


Figure 4-16: Edges used to create the face

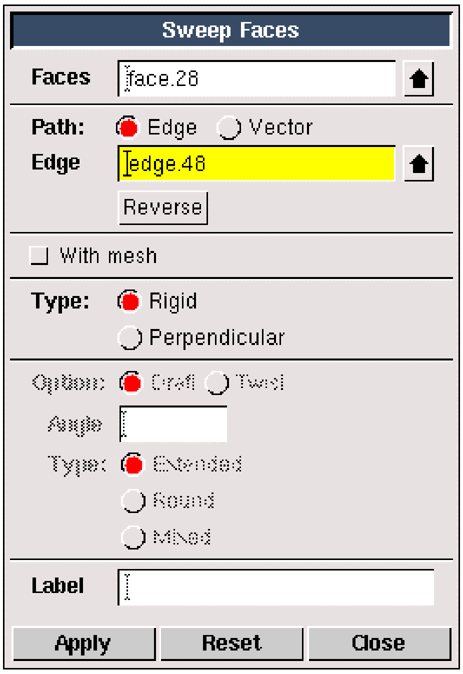
b) *Shift*-left-click the edges marked E and F in Figure 4-16.

c) Click **Apply** to accept the selected edges and create a face.

4. Create a volume by selecting the new face and sweeping it along the direction defined by an edge.

GEOMETRY  → **VOLUME**  → **FORM VOLUME**  **R** 

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



a) Select the new face in the graphics window.

Shift-middle-click *on a face to unselect it and select the face next to it.*

b) Left-click in the list box to the right of **Edge** to make the **Edge** list box active.

c) Select the edge marked G in Figure 4-17.

! *A red arrow will appear on the edge, indicating the direction in which the face will be swept. This arrow should be pointing away from the face you selected. If it is not, click the* Reverse *button in the* **Sweep Faces** *form to reverse the direction of the arrow and the sweep.*

d) Click **Apply** to sweep the face.

The volume created by the sweep is shown in Figure 4-18.

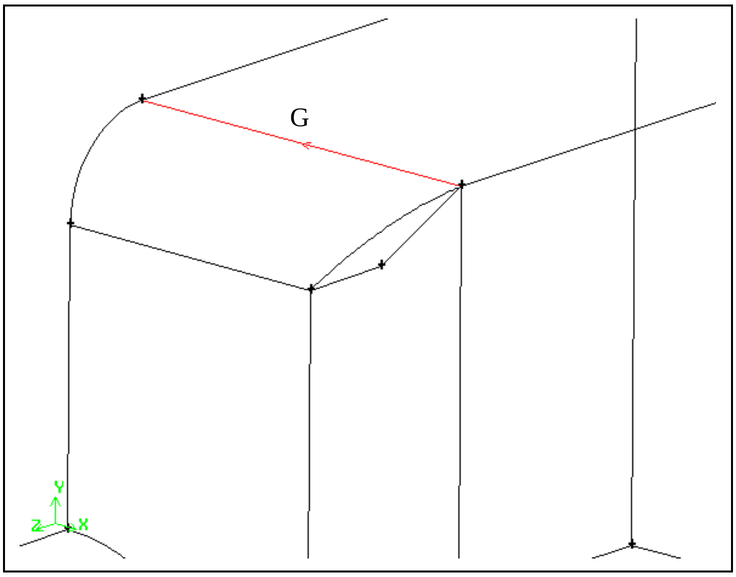


Figure 4-17: Edge to be used for sweep face

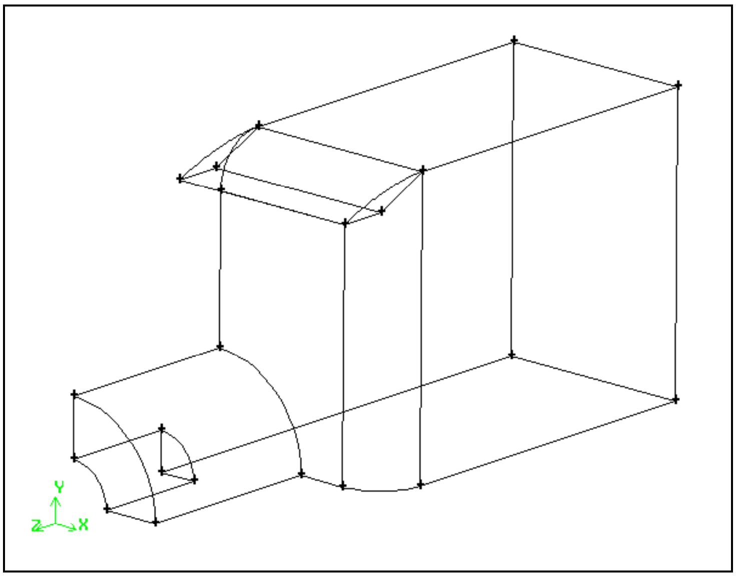


Figure 4-18: Face swept parallel to an edge to form a volume

Note that the volume created by the **Sweep Faces** *operation extends outside the boundaries of the burner box.*

5. Sweep the same face in a different direction.

   a) Select the face marked H in Figure 4-19.

   b) Left-click in the list box to the right of **Edge** to make the **Edge** list box active.

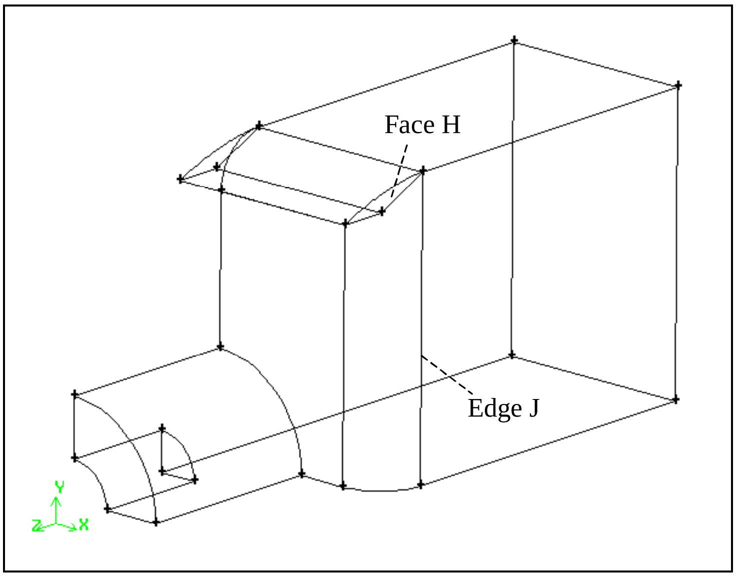


Figure 4-19: Face and edge to be used for sweep face

   c) Select the edge marked J in Figure 4-19.

   d) Click Reverse to reverse the direction of the edge.

      ! *Again, the arrow on this edge should be pointing away from the selected face.*

   e) Click **Apply**.

      The volume created by the sweep is shown in Figure 4-20.

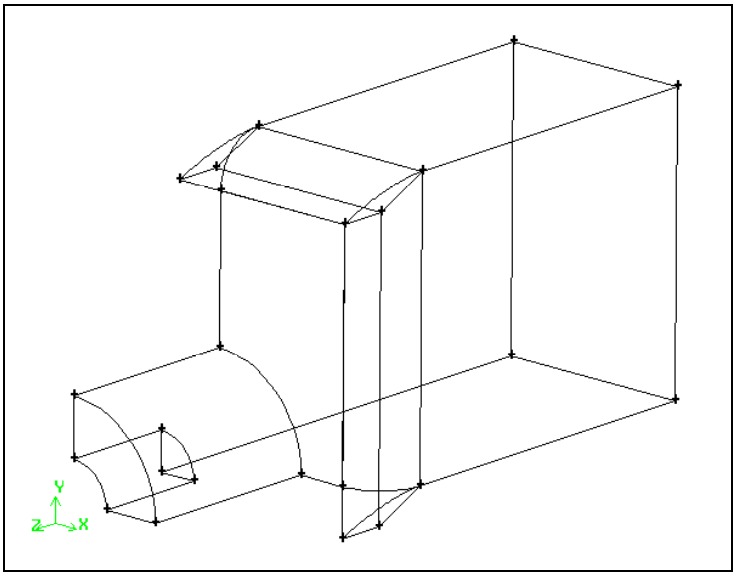


Figure 4-20: Face swept parallel to an edge to form a second volume

6. Split the large burner volume using the two smaller volumes.

   If you split one volume with another volume, the following volumes will result:

   - *Volumes corresponding to the common region(s) from intersection.*
   - *Volumes corresponding to the region(s) defined by subtracting the second volume from the first.*

   In other words, splitting a volume results in a combination of the intersection and subtraction Boolean operations. The order of selecting the volumes is important. For example, Figure 4-21 shows the difference between splitting volume A using volume B, and vice versa.

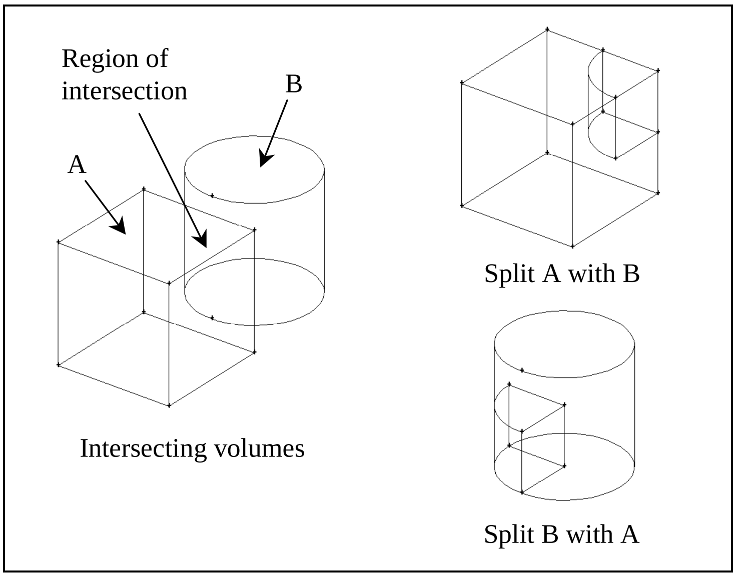
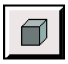
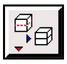
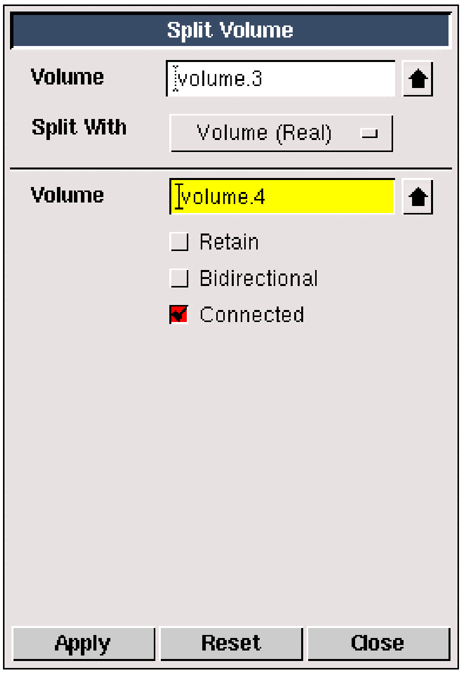


Figure 4-21: Splitting volumes

GEOMETRY  $\longrightarrow$ **VOLUME**  $\longrightarrow$ **SPLIT/MERGE VOLUMES** 

This command sequence opens the **Split Volume** *form.*



a) Select the large burner geometry in the graphics window.

b) Select Volume (Real) as the **Split With** option.

c) Left-click in the **Volume** list box located *below* the **Split With** section to make it active.

d) Select the first volume created using the sweep face method.

e) Unselect the Bidirectional option.

f) Click **Apply**.

g) Left-click in the **Volume** list box located *above* the **Split With** section to make it active.

h) Select the large burner geometry in the graphics window.

i) Retain Volume (Real) as the **Split With** option.

j) Left-click in the **Volume** list box located *below* the **Split With** section to make it active.

k) Select the second volume created using the sweep face method.

l) Click **Apply**.

The complete decomposed burner geometry (see Figure 4-22) is now ready to be meshed.

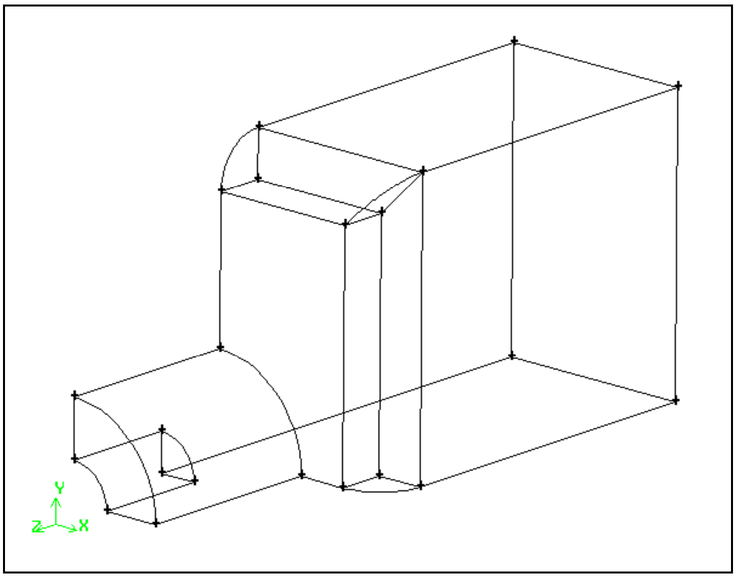


Figure 4-22: Decomposed burner geometry

Step 10: Generate an Unstructured Hexahedral Mesh

In the meshing section of this tutorial you will use:

- Cooper *tool*
- *Face meshing schemes*
- *Variable global mesh densities*

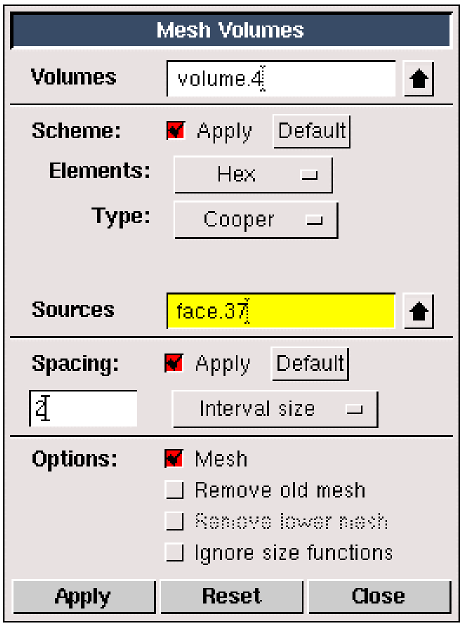
It is possible to use the Cooper *tool to automatically mesh the entire model with a uniform mesh size, but this tutorial will instead demonstrate a few ways of controlling the mesh density and the meshing schemes used. Typically, the* Cooper *tool will use the* Pave *meshing scheme on all source faces, if certain criteria are not met. See the* GAMBIT *Modeling Guide for more information on* GAMBIT*'s meshing tools.*

The two small volumes will be meshed first using the Cooper *meshing scheme. For the remaining volume, you will mesh some faces first. In this case, you will mesh the small quarter-circle face with a* Tri Primitive *scheme and a finer mesh size. Similarly, you will mesh the annular face of the inlet with a fine mapped mesh. However, to ensure that the face in-between the quarter-circle and the annular faces has a mapped (or submapped) mesh, which is required for the* Cooper *tool, you will mesh this face before meshing the annular face. Finally, you will use the automatic* Cooper *tool to mesh the remaining faces and volume.*

1. Generate a mesh for one of the small volumes.

   MESH → **VOLUME** → **MESH VOLUMES**

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



a) Select the volume at the top front of the burner geometry.

The volume to be selected is shown in Figure 4-23.

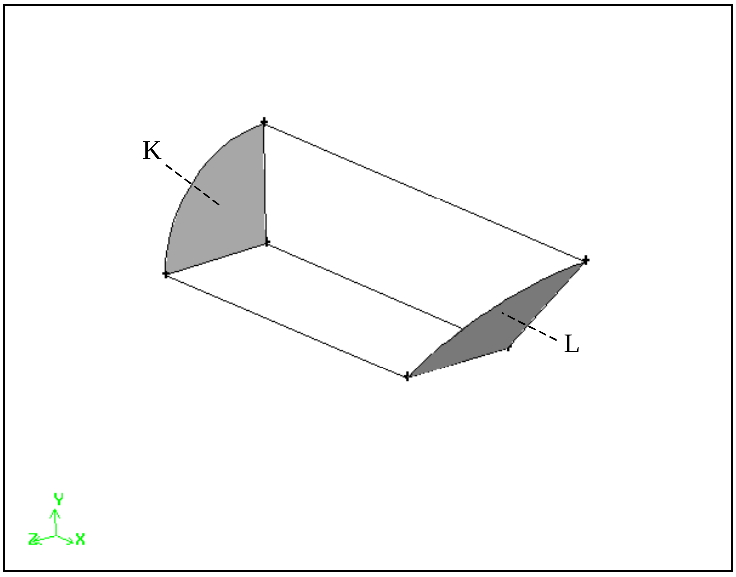


Figure 4-23: Faces to select for first Cooper meshing operation

In this case the criteria for the Cooper *scheme are not fulfilled. This is because* GAMBIT *will not automatically mesh the back face of the volume using the* Map *or* Submap *meshing scheme, because the angle at one of the face corners is not close enough to 90° for it to be automatically classified with the* End *vertex type, which is a requirement for automatic* Map *meshing on a four-sided face. However, you can force* GAMBIT *to use the* Cooper *scheme on this volume by selecting it and then manually picking the source faces (the faces whose surface meshes are to be swept through the volume to form volume elements). When you click* **Apply**, GAMBIT *will automatically enforce the* Submap *scheme on all side faces not already set to use the* Map *or* Submap *schemes, and will modify the vertex types to honor the scheme selected. See the* GAMBIT *Modeling Guide for more information on using the meshing schemes.*

b) Select Hex from the **Elements** option menu under **Scheme** in the **Mesh Volumes** form and select Cooper from the **Type** option menu.

c) Left-click in the **Source** list box (which will turn yellow), and then select the faces marked K and L in Figure 4-23 as the **Source** faces.

The faces are at opposite ends of the volume.

! *If you select the wrong face, and the face you want is the one next to the face selected, Shift*-middle-click *on the face to unselect it and select the face next to it. You can also click* **Reset** *in the* **Mesh Volumes** *form to reset everything you set in the form.*

d) Retain the default Interval size of 2 under **Spacing** in the **Mesh Volumes** form.

Note that this is the interval size for meshing that you set as the default in Step 2 of this tutorial.

e) Click the **Apply** button at the bottom of the form.

Notice that all faces are meshed before GAMBIT *meshes the volume. The mesh is shown in Figure 4-24.*

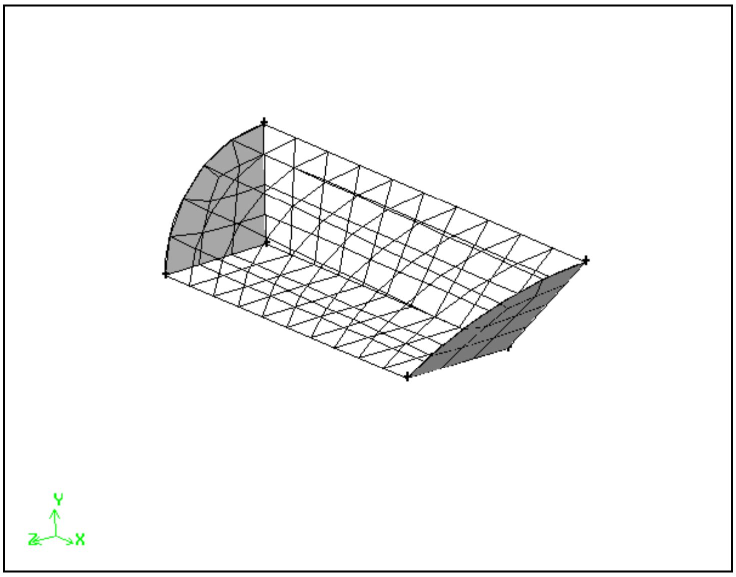


Figure 4-24: Mesh generated for the first small volume in the burner geometry

2. Generate a mesh for the other small volume in the burner geometry.

   a) Select the volume at the side of the burner geometry.

      The volume to be selected is shown in Figure 4-25.

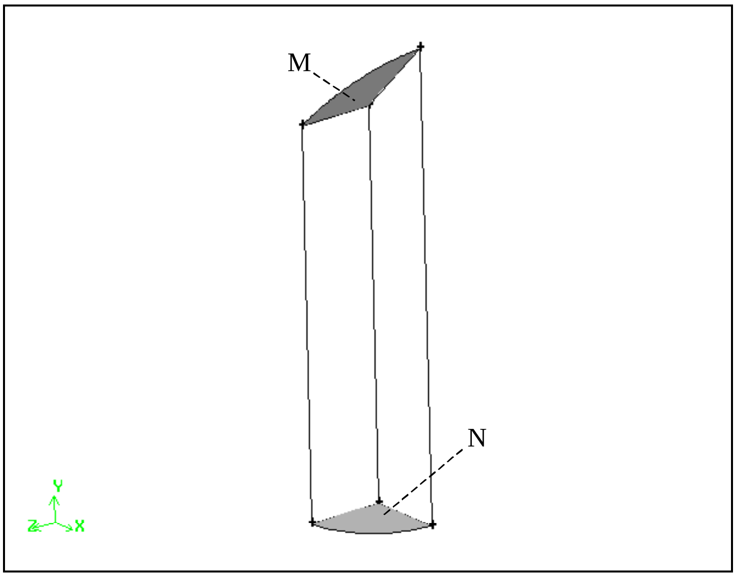


Figure 4-25: Faces to select for cooper meshing on the volume at the side of the burner geometry

b) Select Hex from the **Elements** option menu under **Scheme** in the **Mesh Volumes** form and select Cooper from the **Type** option menu.

c) Left-click in the **Source** list box (which will turn yellow), and select the faces marked M and N in Figure 4-25 as the **Source** faces.

The faces are at opposite ends of the volume.

! *If you select the wrong face, and the face you want is the one next to the face selected, Shift*-middle-click *on the face to unselect it and select the face next to it. You can also click* **Reset** *in the* **Mesh Volumes** *form to unselect all faces, and then select the correct faces.*

d) Retain the default Interval size of 2 under **Spacing** in the **Mesh Volumes** form and click the **Apply** button at the bottom of the form.

The mesh is shown in Figure 4-26.

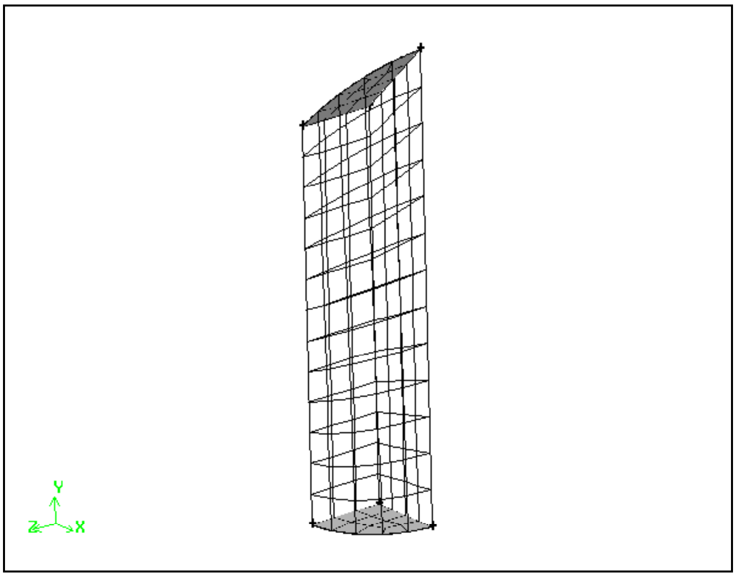


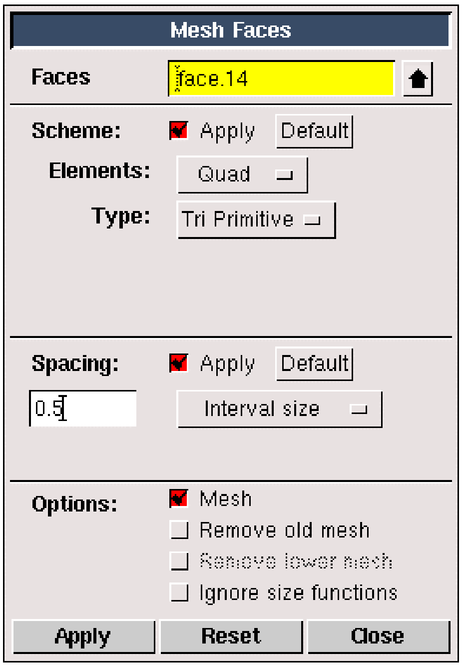
Figure 4-26: Mesh generated for the second small volume in the burner geometry

Next, you will mesh the small face where the burner entrance and the burner chamber meet, and the face at the entrance to the burner. In GAMBIT, *you can "pre-mesh" any source faces on a volume by selecting a meshing scheme and size, to improve the quality of the final mesh.*

3. Mesh the small face where the burner entrance and the burner chamber meet.

MESH → **FACE** → **MESH FACES**

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



a) Select the face marked P in Figure 4-27.

Notice that GAMBIT *automatically selects the* Tri Primitive **Scheme** *in the* **Mesh Faces** *form. See the* GAMBIT *Modeling Guide for more information on the* Tri Primitive *scheme.*

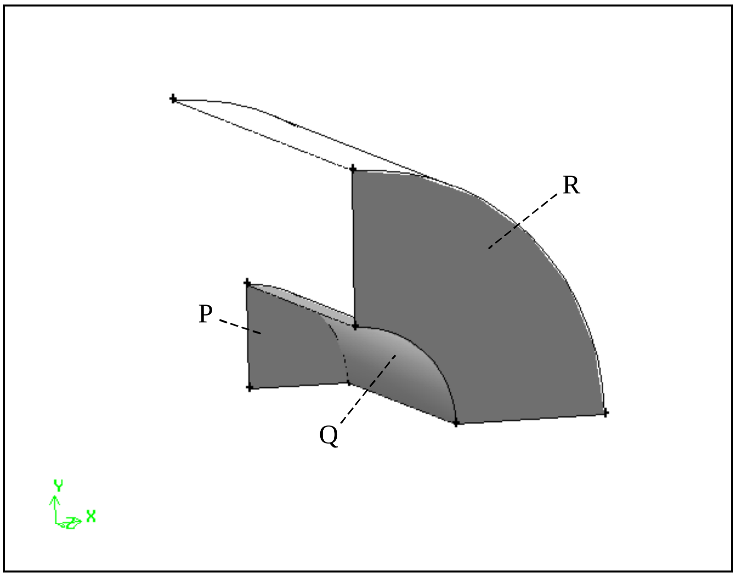


Figure 4-27: Faces to be meshed in the burner geometry

b) Enter `0.5` for the Interval size under **Spacing** and click the **Apply** button at the bottom of the form.

The face will be meshed as shown in Figure 4-28.

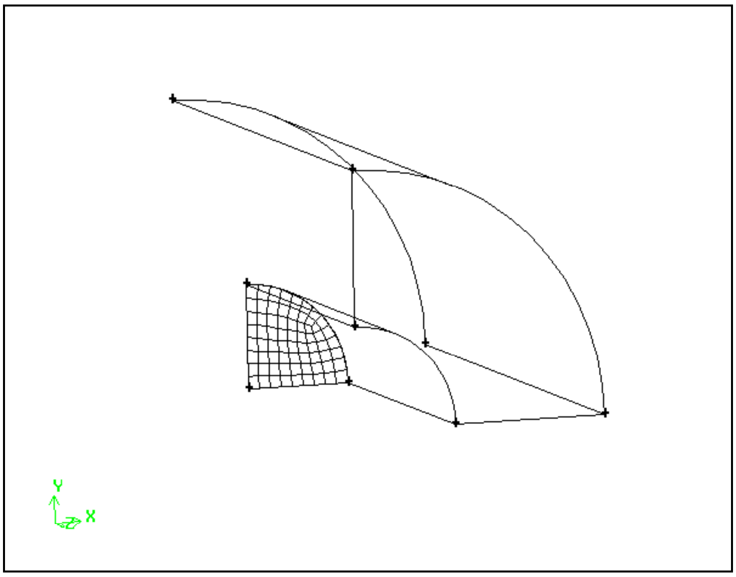


Figure 4-28: Mesh on small face

4. Mesh the curved face along the entrance to the burner.

   a) Select the face marked Q in Figure 4-27.

      GAMBIT *will automatically select the* Map **Scheme** *in the* **Mesh Faces** *form. See the* GAMBIT *Modeling Guide for more information on the* Map *meshing scheme.*

   b) Retain the default **Interval size** of 2 under **Spacing** and click the **Apply** button at the bottom of the form.

      The face will be meshed as shown in Figure 4-29.

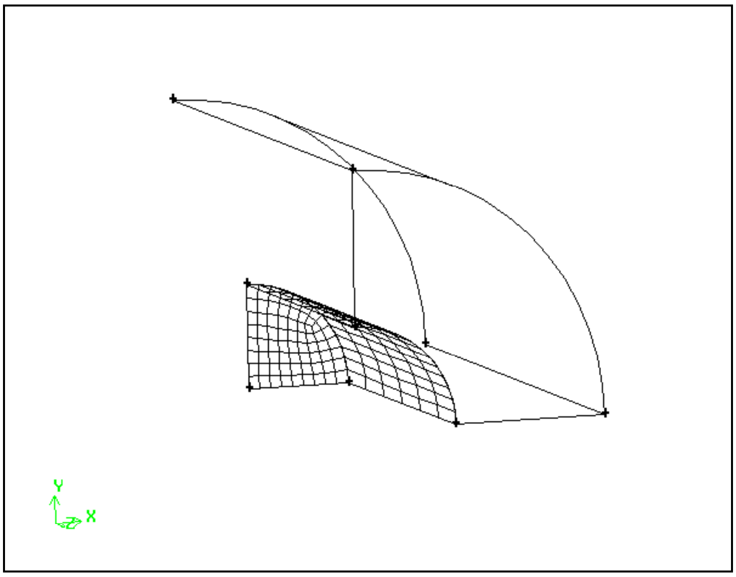


Figure 4-29: Mesh on curved face along the entrance to the burner

5. Mesh the face at the entrance to the burner.

   a) Select the face marked R in Figure 4-27.

      GAMBIT *will automatically select the* Map **Scheme** *in the* **Mesh Faces** *form.*

   b) Enter `1` for the Interval size under **Spacing** and click the **Apply** button at the bottom of the form.

      The face will be meshed as shown in Figure 4-30.

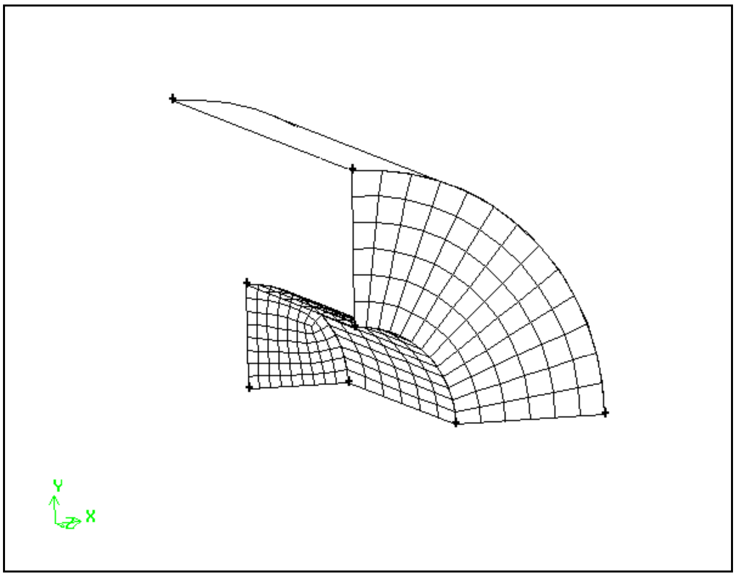


Figure 4-30: Mesh on face at entrance of burner

6. Create a mesh for the rest of the volume of the burner.

   MESH → **VOLUME** → **MESH VOLUMES**

   a) Select the remaining burner geometry in the graphics window.

      GAMBIT *will automatically choose the* Cooper **Scheme** *as the meshing tool to be used, and will use an* Interval size *of* 2 *(the default) under* **Spacing**. *It will also select the source faces it requires to generate the* Cooper *mesh. These faces are marked* S *through* X *in Figure 4-31.*

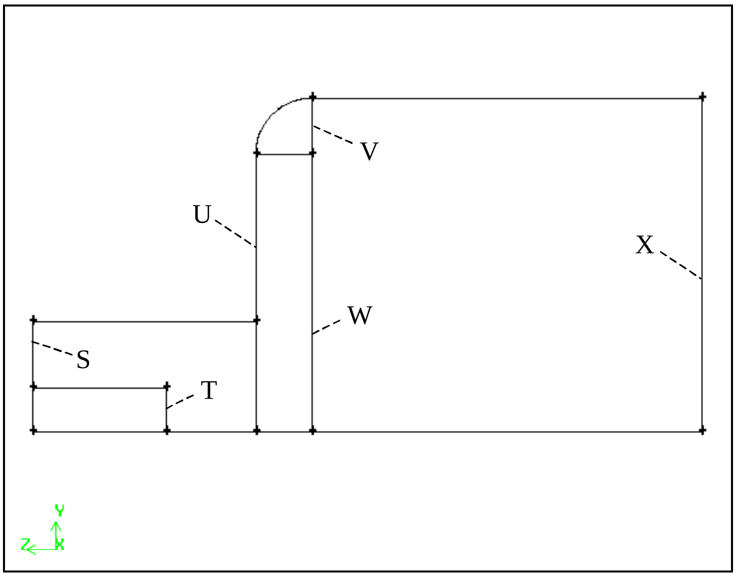
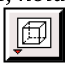
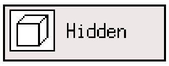
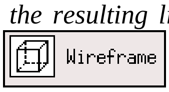


Figure 4-31: Source faces to be used for cooper mesh

b) Click **Apply** at the bottom of the **Mesh Volumes** form.

This accepts the volume you selected as the one to be meshed and the source faces GAMBIT *has chosen for the* Cooper *meshing scheme, and starts the meshing. The complete mesh is shown in Figure 4-32.*

Notice that hidden line removal has been turned on in Figure 4-32 to make the mesh easier to see. To turn on hidden line removal, hold down the right mouse button on the **RENDER MODEL** *command button*  *in the* **Global Control** *toolpad and select* 

from the resulting list. To view the mesh without hidden line removal, reselect the 

option.

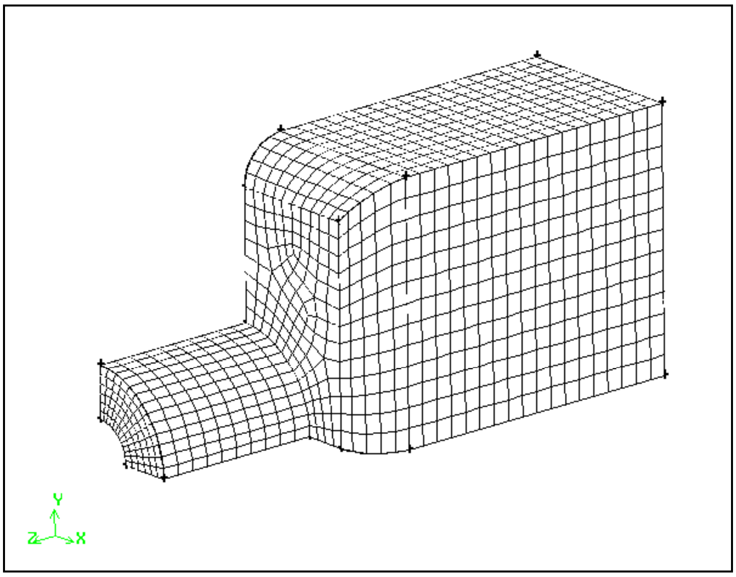
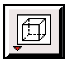
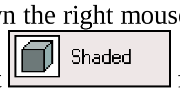


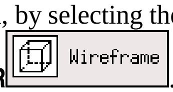
Figure 4-32: Volume mesh for the burner geometry

7. You can view a shaded display of the mesh using the **RENDER MODEL** command button in the **Global Control** toolpad

   a) Hold down the right mouse button on the **RENDER MODEL** command button  and select 

 from the resulting list.

   b) Rotate and translate the volume to view the mesh.

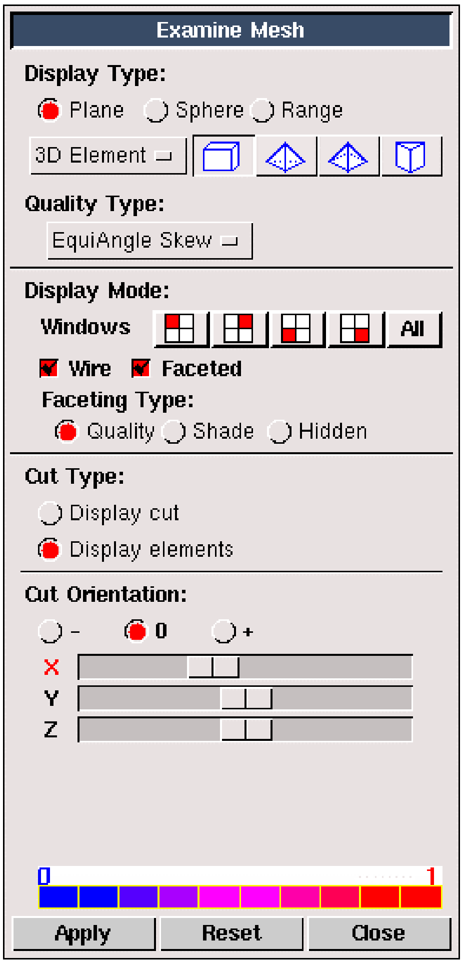
   c) When you are finished, return to the wireframe view of the model, by selecting the following command buttons in the **Global Control** toolpad:  **R**

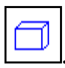
.

Step 11: Examine the Quality of 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 **3D Element** *type selected by default at the top of the form is a brick* .

   a) Select the Plane option under **Display Type**.

b) Select or retain EquiAngle Skew from the **Quality Type** option menu.

c) Hold down the left mouse button on the **X** slider box and move it to view slices of the mesh with different x values.

An example is shown in Figure 4-33. The mesh is drawn as a wireframe (by default) as you drag the slider box, and it is colored by EquiAngle Skew *quality when you release the slider box. As you sweep a plane through the x values, you will see the way in which the* Cooper *tool has automatically decomposed the volume internally to mesh it with hexahedral elements.*

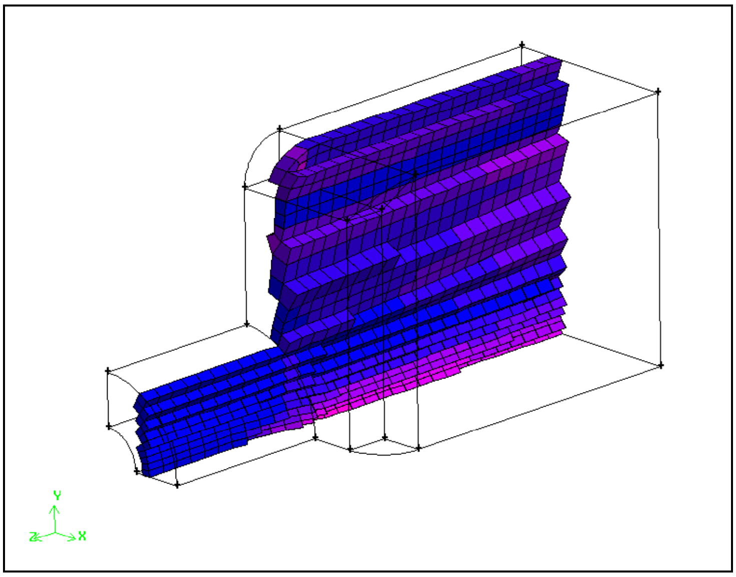


Figure 4-33: Slice of the mesh in the x direction

d) Use the **Y** and **Z** sliders to view slices in the y and z directions.

e) Select Range under **Display Type**, and then click with the left mouse button on the histogram bars that appear at the bottom of the **Examine Mesh** form to highlight elements in a particular quality range.

Figure 4-34 shows the view in the graphics window if you click on the fifth bar from the left on the histogram (representing cells with a skewness value between 0.4 and 0.5). These low values for the maximum skewness indicate that the mesh is acceptable. The histogram consists of a bar chart representing the statistical distribution of mesh elements with respect to the *specified* **Quality Type**. *Each vertical bar on the histogram corresponds to a unique set of upper and lower quality limits.*

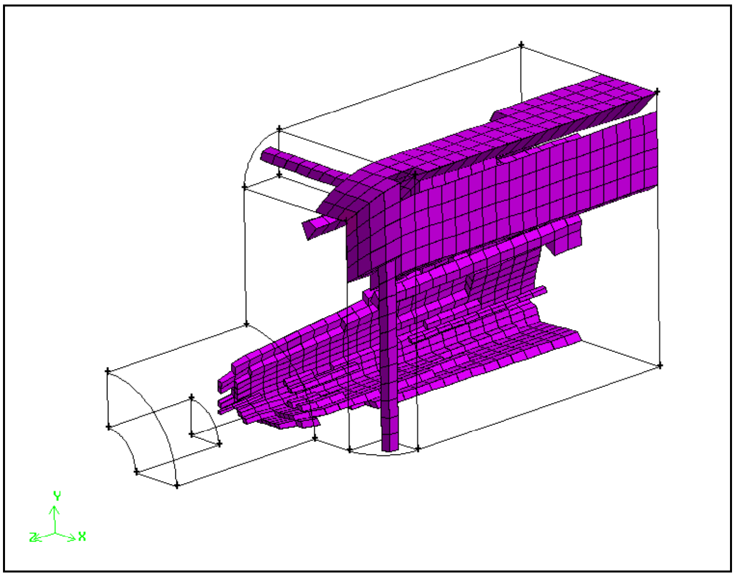
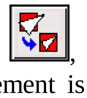


Figure 4-34: Elements within a specified quality range

When you select the **Display Type:**Range *option on the* **Examine Mesh** *form,* GAMBIT *displays the* Show worst element *option immediately below the statistics displayed under the histogram. If you select the* Show worst element *option,* GAMBIT *displays only the "worst" element as determined by the current* **Quality Type** *quality metric.*

f) Select the Show worst element option.

g) Click the **FIT TO WINDOW** command button , at the top left of the **Global Control** toolpad, to see where the worst element is located with respect to the entire geometry.

h) Close the **Examine Mesh** form by clicking the **Close** button at the bottom of the form.

Step 12: Set Boundary Types

1. Remove the mesh from the display before you set the boundary types.

   This makes it easier to see the edges and faces of the geometry. The mesh is not deleted, just removed from the graphics window.

   a) Click the **SPECIFY DISPLAY ATTRIBUTES** command button  at the bottom of the **Global Control** toolpad.

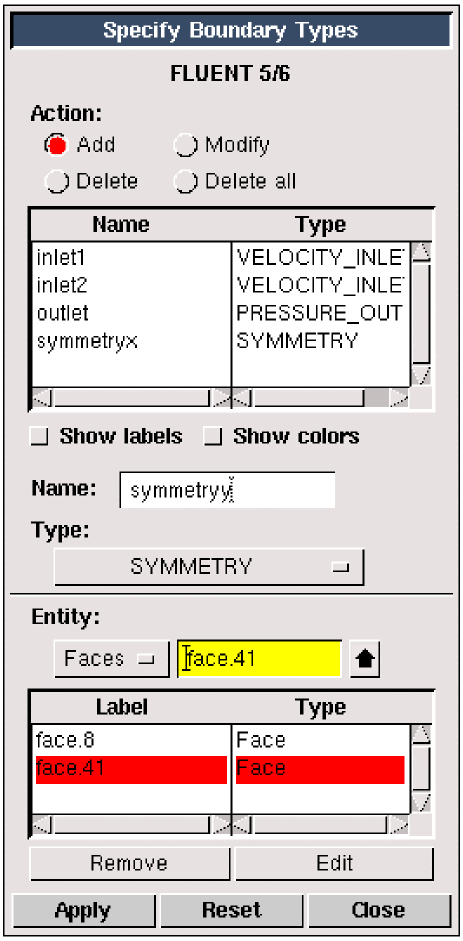
   b) Select the Off radio button to the right of **Mesh** near the bottom of the form.

   c) Click **Apply** and close the form.

2. Set boundary types for the burner.

   ZONES  → **SPECIFY BOUNDARY TYPES** 

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



a) Define two velocity inlets.

i. Enter the name "inlet1" in the **Name** text-entry field.

ii. Select VELOCITY_INLET in the **Type** option menu.

iii. Check that Faces is selected as the **Entity**.

iv. *Shift*-left-click the face marked A in Figure 4-35 and click **Apply** to accept the selection.

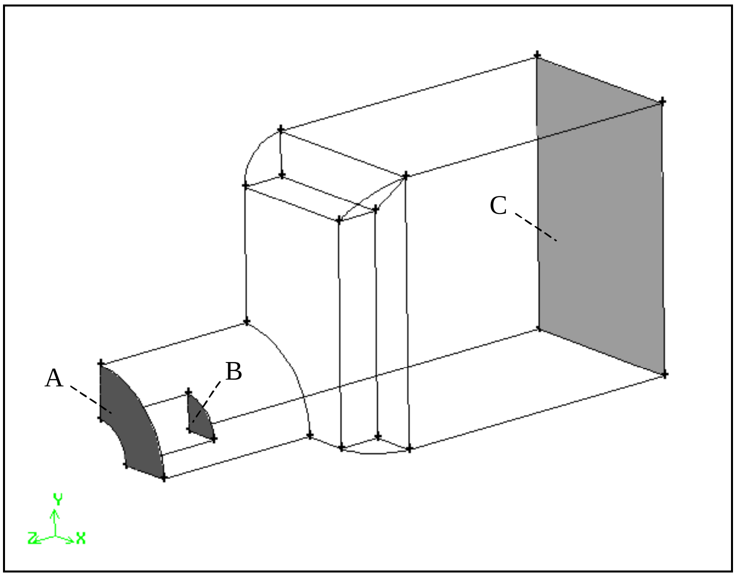


Figure 4-35: Boundary-type faces for the burner—inlet and outlet

This face will be set as a velocity inlet.

v. Enter the name “`inlet2`” in the **Name** text-entry field.

vi. Check that VELOCITY_INLET is still selected in the **Type** option menu, select the face marked B in Figure 4-35, and click **Apply**.

b) Define a pressure outlet.

i. Enter the name “`outlet`” in the **Name** text-entry field.

ii. Change the **Type** to PRESSURE_OUTLET by selecting it in the option menu below **Type**.

iii. Select the face marked C in Figure 4-35 and click **Apply** to accept the selection.

c) Define symmetry boundary types for the two faces normal to the x axis.

i. Enter the name “`symmetryx`” in the **Name** text entry box.

ii. Change the **Type** to SYMMETRY.

iii. Select the two faces on the left side of the geometry as you look at it from the front (the faces marked D and E in Figure 4-36). Accept the selection of the faces.

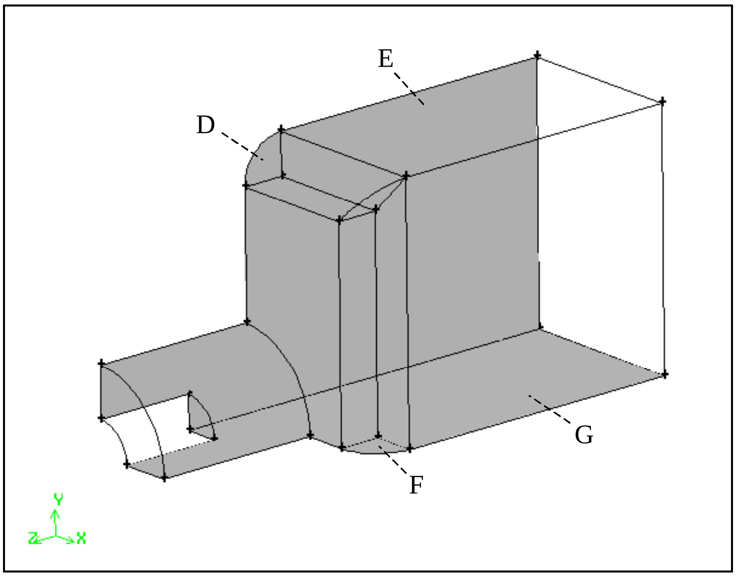


Figure 4-36: Boundary-type faces for the burner—symmetry

d) Define symmetry boundary types for the two faces normal to the *y* axis.

i. Enter the name `symmetryy` in the **Name** text entry box.

ii. Check that SYMMETRY is still selected in the **Type** option menu and select the two faces on the bottom of the geometry (the faces marked F and G in Figure 4-36). Accept the selection of the faces.

The velocity inlet, pressure outlet, and symmetry boundaries for the 3-D combustion chamber are shown in Figure 4-37. To display the boundary-type labels in the graphics window, select the **Show labels** *option on the* **Specify Boundary Types** *form. To display colors associated with each boundary-type assignment, select the* **Show colors** *option. (NOTE:* GAMBIT *automatically shades the faces for the* **Show colors** *option.)*

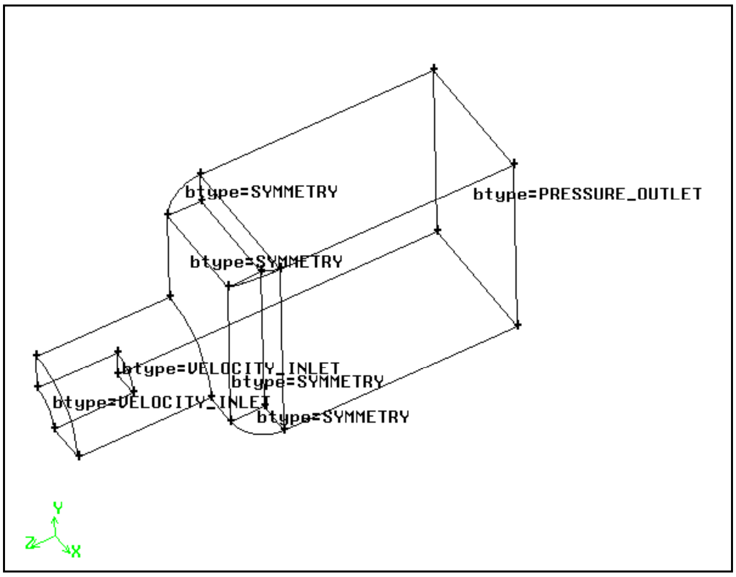


Figure 4-37: Boundary types for the combustion chamber

Note that you could also specify the remaining outer faces of the model as WALL *boundaries. This is not necessary, however, because when* GAMBIT *saves a mesh, any external faces (in 3-D) for which you have not specified a boundary type will be written out as* WALL *boundaries by default.*

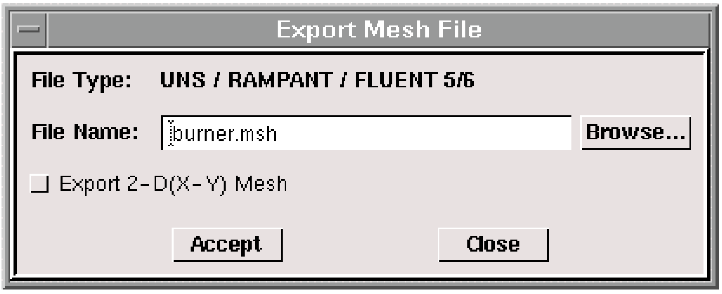
In addition, when GAMBIT *writes a mesh, any volumes (in 3-D) for which you have not specified a continuum type will be written as* FLUID *by default. This means that you do not need to specify a continuum type in the* **Specify Continuum Types** *form for this tutorial.*

Step 13: Export the Mesh and Save the Session

1. Export a mesh file.

   File → **Export** → **Mesh...**

   This command sequence opens the **Export Mesh File** *form. Notice that the* **File Type** *at the top of the form is* **UNS / RAMPANT / FLUENT 5/6**.



   a) Enter the **File Name** for the file to be exported (`burner.msh`).

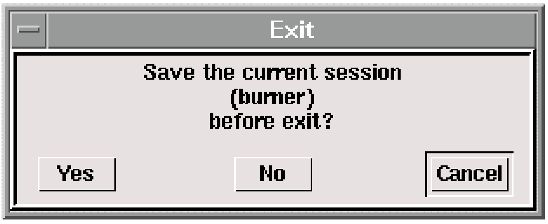
   b) Click **Accept**.

      The file will be written to your working directory.

2. Save the GAMBIT session and exit GAMBIT.

   File → **Exit**

   GAMBIT *will ask you whether you wish to save the current session before you exit.*



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

4.5 Summary

In this tutorial, you created the geometry and hexahedral mesh for a 3-D combustion chamber using a top-down construction approach. The use of Boolean operations for uniting, subtracting, and intersecting volumes was demonstrated. The blend volumes command was used to create a rounded shape on the edges of the combustion chamber. Next, the geometry was decomposed into smaller volumes for which the Cooper meshing scheme could be used. Several different ways of meshing the source faces needed by the Cooper scheme were shown.

5. SEDAN GEOMETRY—VIRTUAL CLEANUP

In this tutorial you will import an IGES file containing the geometry for a sedan automobile, clean up the geometry, and mesh it with triangles and tetrahedra.

In this tutorial you will learn how to:

- Import an IGES file
- Specify the way in which the geometry will be colored
- Connect edges, using a manual and an automatic method
- Merge faces
- Create a triangular surface mesh
- Mesh a volume with a tetrahedral mesh
- Prepare the mesh to be read into FLUENT 5/6

5.1 Prerequisites

This tutorial assumes you have worked through Tutorial 1 and, therefore, that you are familiar with the GAMBIT GUI.

5.2 Problem Description

The problem to be considered is shown schematically in Figure 5-1; it is the external body of a luxury sedan. You will generate a mesh on the outside of the car body; therefore, you will create a brick around the sedan to represent the flow domain.

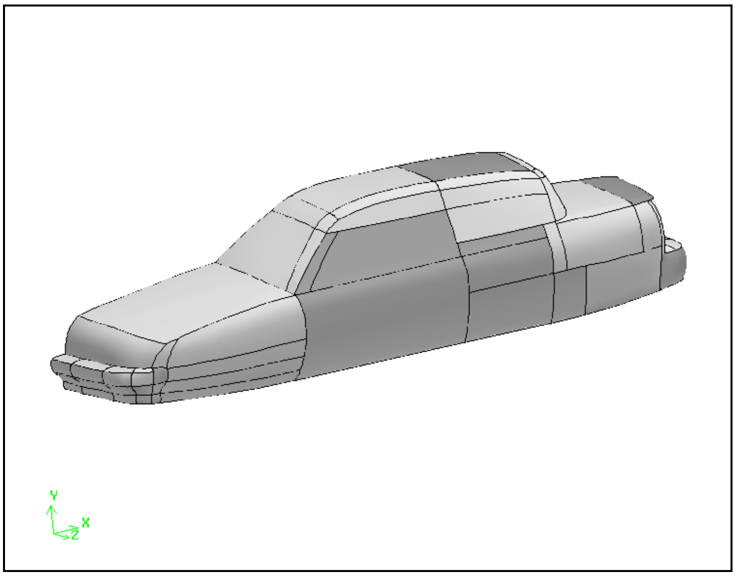


Figure 5-1: Sedan geometry

5.3 Strategy

In this tutorial, you will create a fully unstructured tetrahedral mesh around a car-body geometry imported as an IGES file. This tutorial illustrates the steps you would typically follow to prepare an imported CAD geometry for meshing. The imported geometry is "dirty"—that is, there are gaps between some of the surfaces that make it unsuitable for creating a CFD mesh. After examining the raw imported geometry to identify its problems, such as unconnected edges, you will clean up the geometry using the tools available in GAMBIT.

Most of the gaps can be fixed automatically either during mesh import or subsequently by means of the "connect edge" command. The original CAD geometry is not modified during the fixing process; the modifications required to eliminate the gaps are made using "virtual" geometry, which lies on top of the "real" geometry. Some edges in the original geometry are very short and will be eliminated using the "vertex connect" command. Other edges are not automatically connected, because they are farther apart than the specified tolerance. You will connect such edges manually.

The imported geometry includes a number of small surfaces, the edges of which may unnecessarily constrain the mesh generation process. Using the "merge faces" command, GAMBIT allows you to easily combine these surfaces prior to meshing. You can then have GAMBIT automatically create a triangular mesh on the car body.

Since the imported geometry consists only of the car body, you need to create a suitable domain around the car in order to conduct a CFD analysis (this is loosely equivalent to placing the car in a wind tunnel). The remainder of the tutorial shows how to add a real box around the car body, use virtual geometry to create some missing faces, and finally stitch all faces together into a single volume. This volume can then be meshed (without any decomposition) using a tetrahedral meshing scheme.

5.4 Procedure

1. Copy the file

   `path/Fluent.Inc/gambit2.x/help/tutfiles/sedan.igs`

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

2. Start GAMBIT.

Step 1: Select a Solver

1. Choose the solver from the main menu bar:

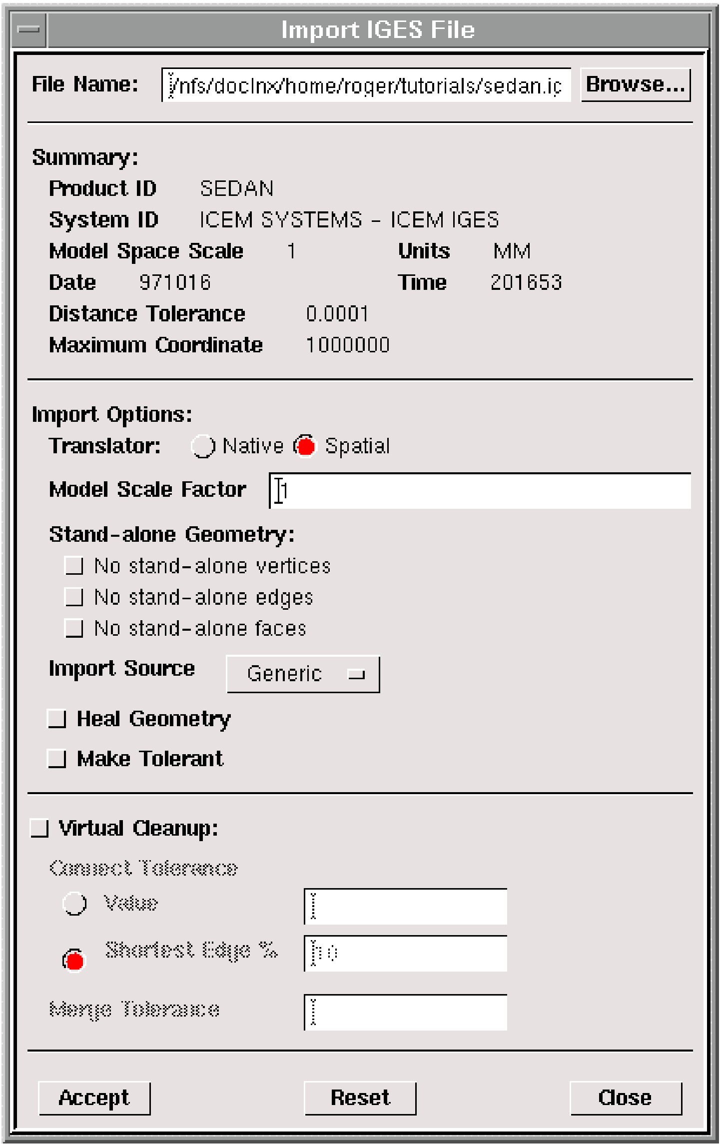
   Solver → **FLUENT 5/6**

   The choice of a solver dictates the options available in various forms (for example, the boundary types available in the **Specify Boundary Types** *form). For some systems,* **FLUENT 5/6** *is the default solver. The solver currently selected is shown at the top of the* GAMBIT *GUI.*

Step 2: Import the IGES File As-Is

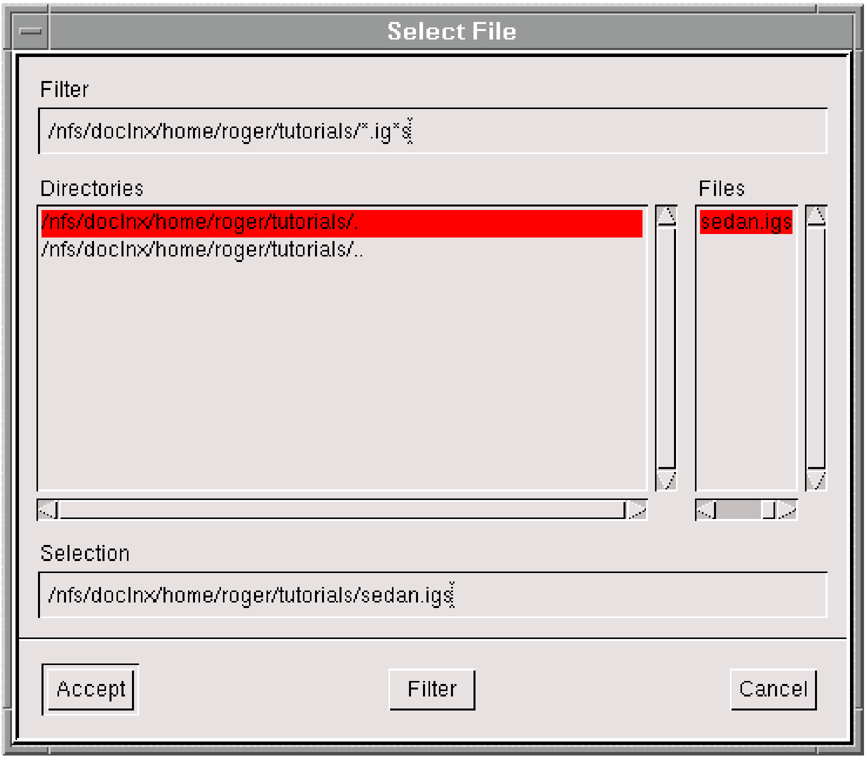
File → Import → IGES ...

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



1. Click on the **Browse...** button.

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



   a) Select `sedan.igs` in the **Files** list.

   b) Click Accept in the **Select File** form.

2. On the **Import IGES File** form, unselect the **Make Tolerant** option, and click **Accept**.

   The IGES file for the sedan body will be read into GAMBIT *(see Figure 5-2).*

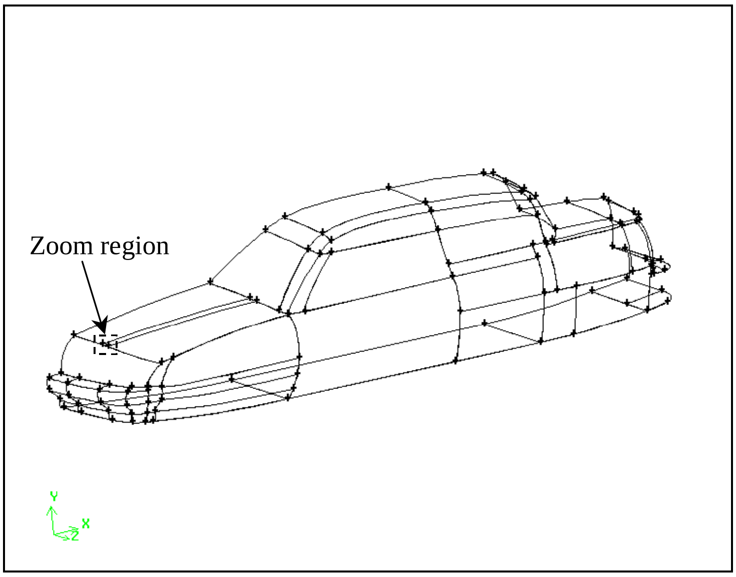
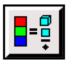
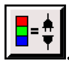


Figure 5-2: Imported sedan body

3. Click the **SPECIFY COLOR MODE** command button  in the **Global Control** toolpad to change to the graphics display to connectivity-based coloring.

   The **SPECIFY COLOR MODE** *command button will change to* . *When* GAMBIT *is in the connectivity display mode, the model is displayed with colors based on connectivity between entities rather than based on entity types.*

   In this case, the colors of all edges in the graphics window will change to orange, indicating that the faces are not connected to each other—that is, that there are gaps between the edges that bound the faces.

4. *Ctrl*-drag the mouse in the graphics window to create the zoom-region box shown in Figure 5-2, above (at the front of the sedan hood). When you release the mouse button, GAMBIT zooms in on the region.

   Figure 5-3 shows the zoomed region, illustrating an example of the gaps that exist between the faces that comprise the surface of the sedan.

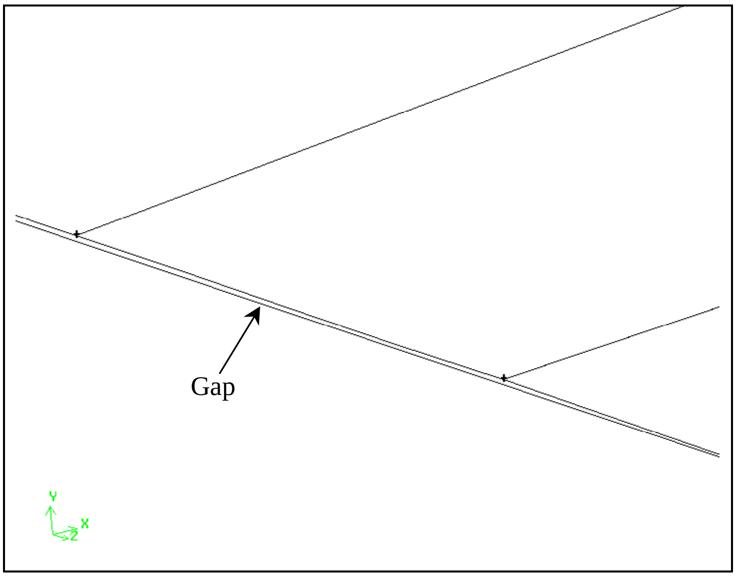


Figure 5-3: Zoomed region showing unconnected faces on sedan hood

In addition to gaps such as those shown in Figure 5-3, the geometry imported as-is from the IGES file contains a number of small edges that can inhibit meshing of the model. GAMBIT provides options on the **Import IGES File** *form that allow you to eliminate many of the gaps and short edges during import. The next step in this tutorial illustrates the use of one such option—***Virtual Cleanup**.

Step 3: Reset and Import the IGES File Using Virtual Cleanup

The **Virtual Cleanup** *option on the* **Import IGES File** *form allows you to automatically eliminate many features, such as short edges and gaps between unconnected faces, during geometry import. By eliminating many such features during geometry import, you can greatly simplify the process of creating a meshable model.*

1. Reset GAMBIT.

   a) On the **Command** line, type `reset`, and press *Enter*.

      GAMBIT *deletes the current geometry from the graphics window and resets the GUI.*

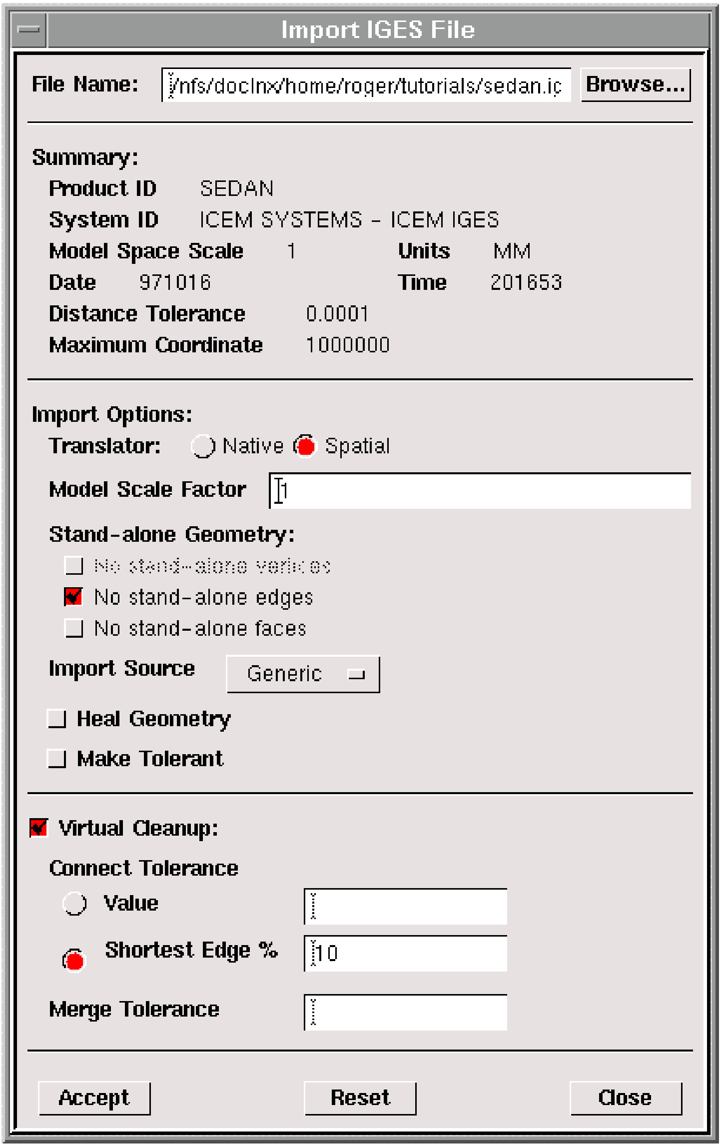
2. Reselect the FLUENT 5/6 solver from the main menu bar.

   Solver $\rightarrow$ **FLUENT 5/6**

3. Open the **Import IGES File** form.

   File $\rightarrow$ **Import** $\rightarrow$ **IGES ...**

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



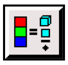
GAMBIT *retains the* **File Name** *selected in Step 2—that is,* `sedan.igs`.

4. Under **Import Options: Stand-alone Geometry**, select the No stand-alone edges check box.

This option instructs GAMBIT *not to read in any edges that do not belong to faces or volumes. Such edges can be deleted after the geometry has been read into* GAMBIT*, but this option eliminates the extra step.*

5. Set the connect tolerance to 10% of the shortest edge by selecting the **Virtual Cleanup** toggle and specifying the **Shortest Edge %** at `10`.

   This substep invokes an automated sequence of connect operations that attempt to clean up the imported geometry after it is read into GAMBIT.

6. Click the **SPECIFY COLOR MODE** command button  in the **Global Control** toolpad to change to the graphics display to connectivity-based coloring.

7. On the **Import IGES File** form, click **Accept**.

   The IGES file for the sedan body will again be read into GAMBIT*. This time, however,* GAMBIT *will perform virtual clean-up operations to eliminate gaps and many small edges. As the clean-up operation progresses, edges that are initially displayed as orange (with the connectivity color mode on) will turn blue. Figure 5-4 shows the sedan geometry after import and clean-up operations are complete.*

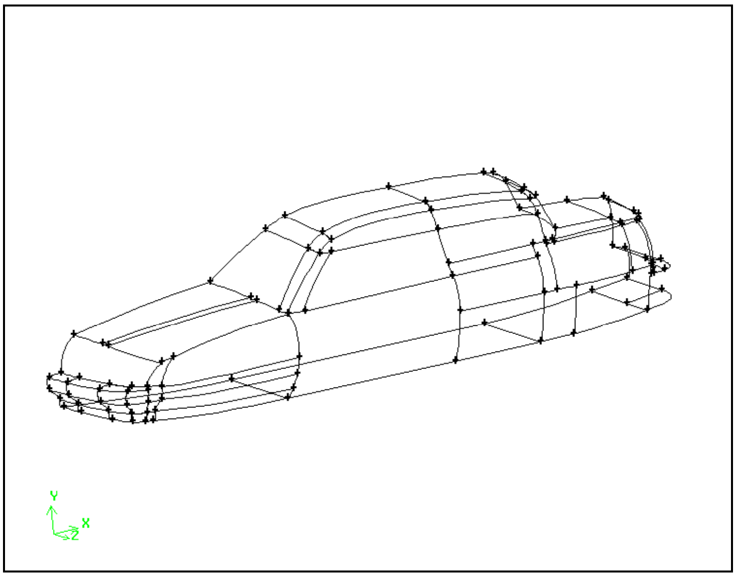
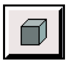
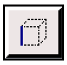
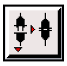


Figure 5-4: Imported sedan body—with **Virtual Cleanup** option

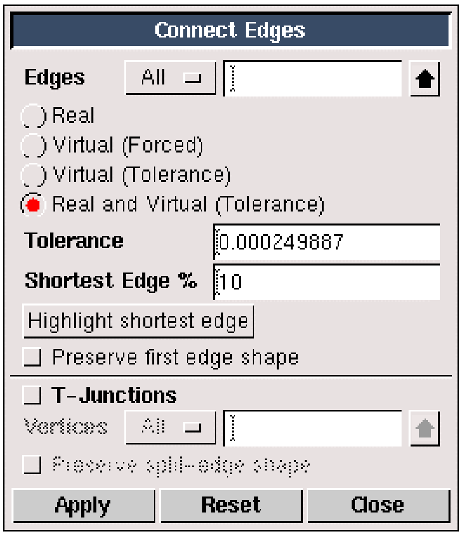
Step 4: Eliminate Very Short Edges

Even after the virtual clean-up operations, the imported IGES geometry is still somewhat "dirty"—that is, there are a few short edges and gaps between the faces that need to be repaired. In this step, you will eliminate the short edges.

1. Find the shortest edge.

   GEOMETRY  → **EDGE**  → **CONNECT/DISCONNECT EDGES** 

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

   a) Select All from the option menu to the right of **Edges**.

   b) Select the Real and Virtual (Tolerance) option.

   c) Press the Highlight shortest edge button.

      GAMBIT *will highlight (in white) the shortest edge—along with its label—in the graphics window.*

   d) Zoom in near the highlighted edge by pressing the *Ctrl* key while using the mouse to drag a box around the edge.

      Figure 5-5 shows the general area on the sedan that contains the shortest edge. Figure 5-6 shows a zoomed view of the edge.

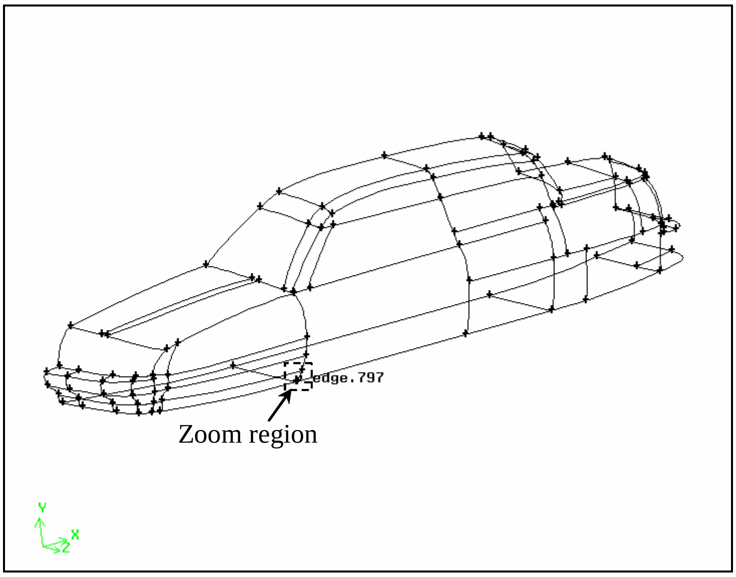


Figure 5-5: Sedan—showing general area of shortest edge location

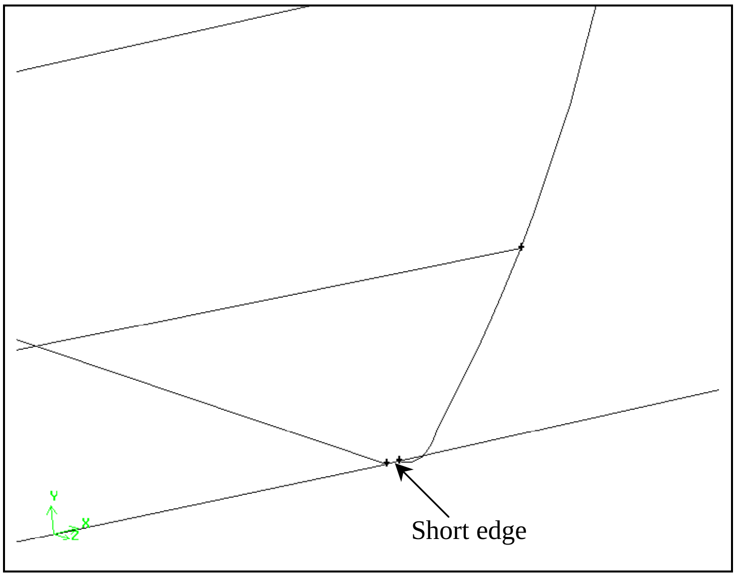
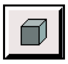
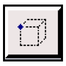
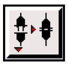
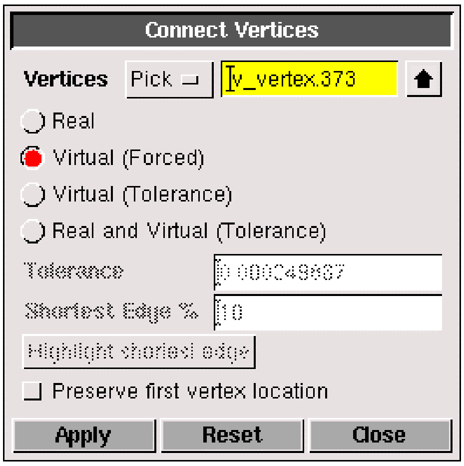


Figure 5-6: Sedan—showing zoomed area near shortest edge

2. Remove the shortest edge.

GEOMETRY  → **VERTEX**  → **CONNECT/DISCONNECT VERTICES** 

This command sequence opens the **Connect Vertices** *form.*



a) Select the Virtual (Forced) option.

b) Pick the two endpoint vertices on the shortest edge.

c) Click **Apply**.

When GAMBIT *attempts to connect these two vertices, an error message is generated stating that connecting these two vertices can cause invalid geometry. The geometry is protected from such operations by means of a default setting.*

d) Open the **Edit Defaults** form (select **Defaults...** from the **Edit** menu on the main menu bar), and change the value of the `GEOMETRY.VERTEX.CONNECT_REMOVE_SHORT_EDGE` variable to 1.

e) Repeat step (c).

This time, GAMBIT *connects the vertices.*

f) Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full sedan in the graphics window.

g) Select the Virtual (Tolerance) option to activate the Highlight shortest edge button on the **Connect Vertices** form.

h) Click the Highlight shortest edge button and repeat steps (a), (b), and (c) to eliminate the next shortest edge (see Figure 5-7).

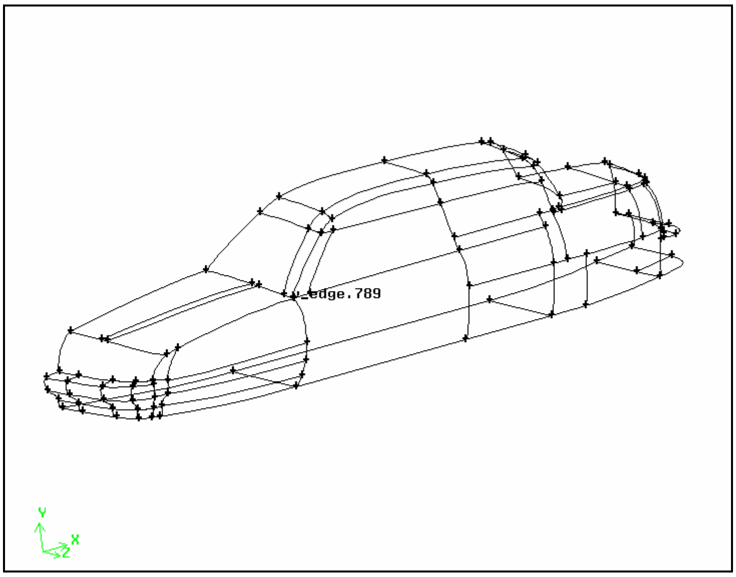


Figure 5-7: Sedan—showing location of next shortest edge

By eliminating the two shortest edges in the model, you ensure that all edge meshing intervals are of reasonable size, thereby reducing the possibility of creating highly distorted elements during meshing.

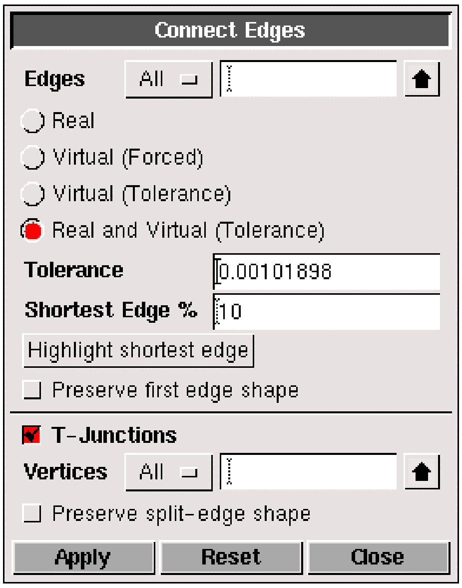
Step 5: Automatically Connect All Remaining “Duplicate” Edges

The imported IGES geometry is still “dirty”—that is, there are a few gaps remaining between the faces that make it unsuitable for creating a mesh. In this step, you will “clean up” the geometry automatically using the **Connect Edges** *operation.*

1. Connect all edges in the geometry that are less than a specified tolerance apart using an automatic method.

   GEOMETRY → **EDGE** → **CONNECT/DISCONNECT EDGES**

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



   a) Select All from the option menu to the right of **Edges**.

   b) Select the Real and Virtual (Tolerance) option.

      You want GAMBIT *to connect all real and virtual edges that are within a tolerance distance of each other.*

   c) Retain the default value of `10` for the **Shortest Edge %**.

   d) Select the **T-Junctions** option.

This option ensures that edges that do not match up correctly will be connected. GAMBIT *will perform edge splits and then reconnect the geometry; an example is shown in Figure 5-8.*

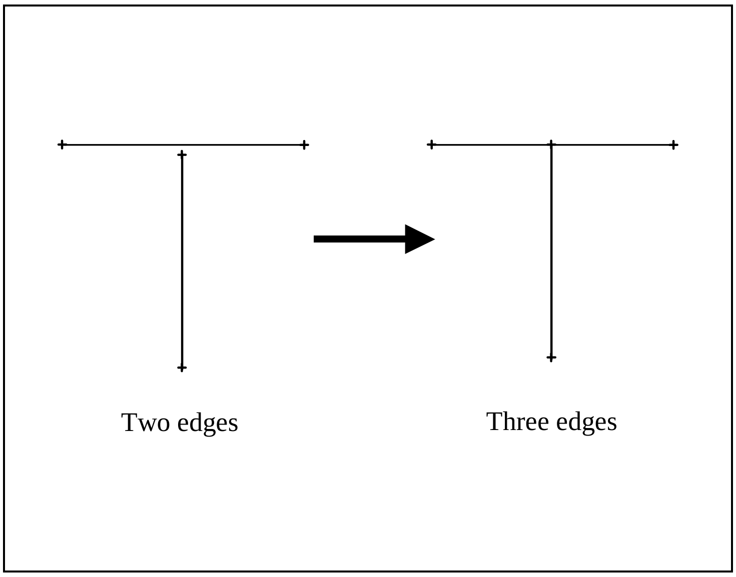


Figure 5-8: Connecting edges

e) Click **Apply**.

A few more edges turn blue in the graphics window as they are connected.

! *The edges on the symmetry plane will remain orange because they do not have any other edges with which they can be connected.*

2. Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full sedan in the graphics window.

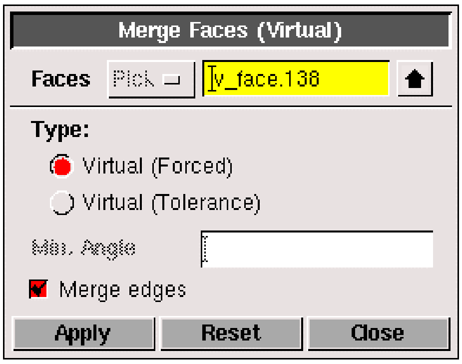
Step 6: Merge Faces

In many cases, the IGES model contains more detail than you need for meshing. The imported geometry for the sedan includes a number of small faces, the edges of which may constrain the mesh generation process unnecessarily. In GAMBIT, *you can merge faces together prior to meshing.*

1. Merge some of the faces on the sedan hood.

GEOMETRY → **FACE** → **SPLIT/MERGE/COLLAPSE/SIMPLIFY FACES**
R Merge Faces (Virtual)

This command sequence opens the **Merge Faces (Virtual)** *form.*



a) Under **Type**, select Virtual (Forced).

b) Zoom in to the hood of the sedan by holding down the *Ctrl* key on the keyboard while dragging a box around the hood of the car with the left mouse button.

c) Select the three faces on the top of the hood as shown in Figure 5-9.

d) Retain the Merge Edges option to facilitate geometry cleanup during merging.

e) Click **Apply** to accept the selected faces and merge them into one face, as shown in Figure 5-10.

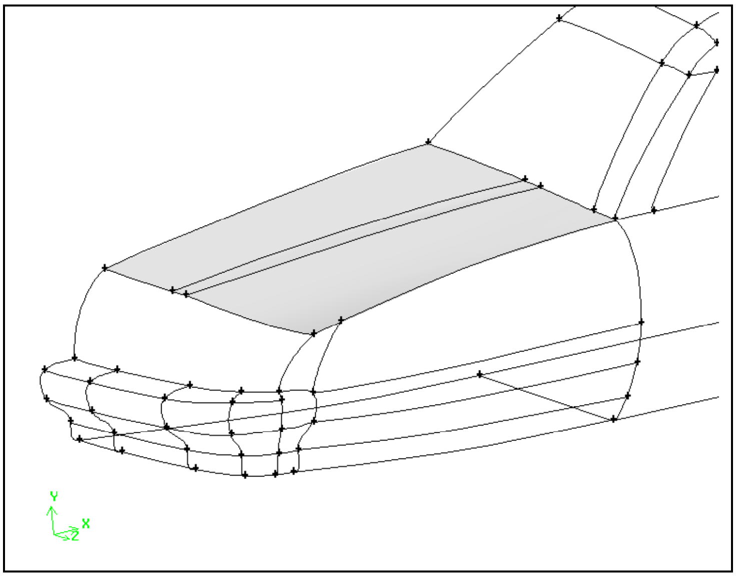


Figure 5-9: Three faces on hood of sedan

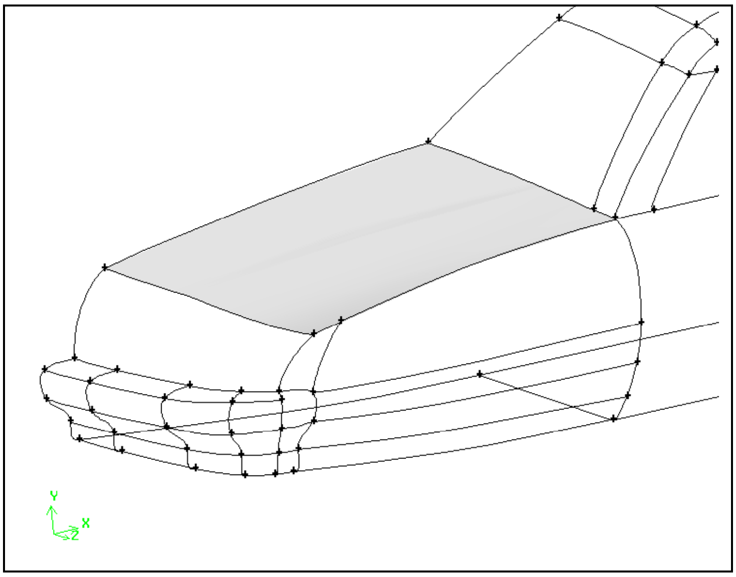


Figure 5-10: Three faces merged on hood of sedan

2. Merge four faces on the trunk of the car (see Figure 5-11) using the method described above. The merged faces are shown in Figure 5-12.

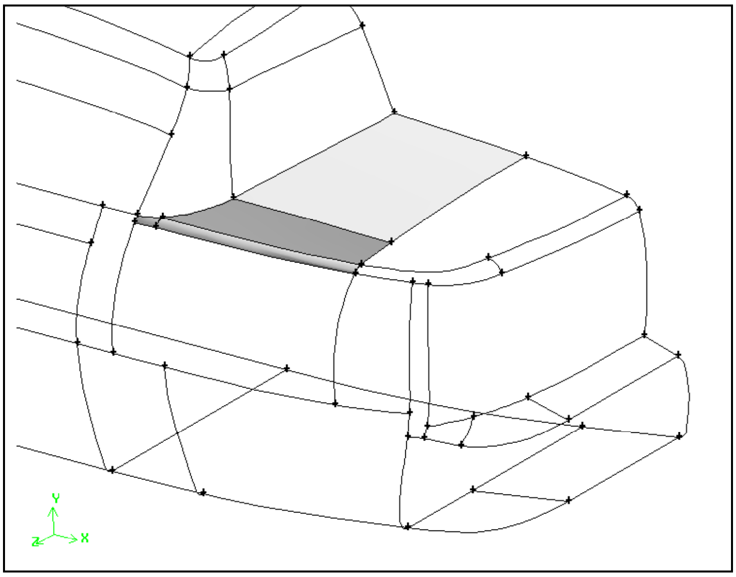


Figure 5-11: Four faces on trunk of sedan

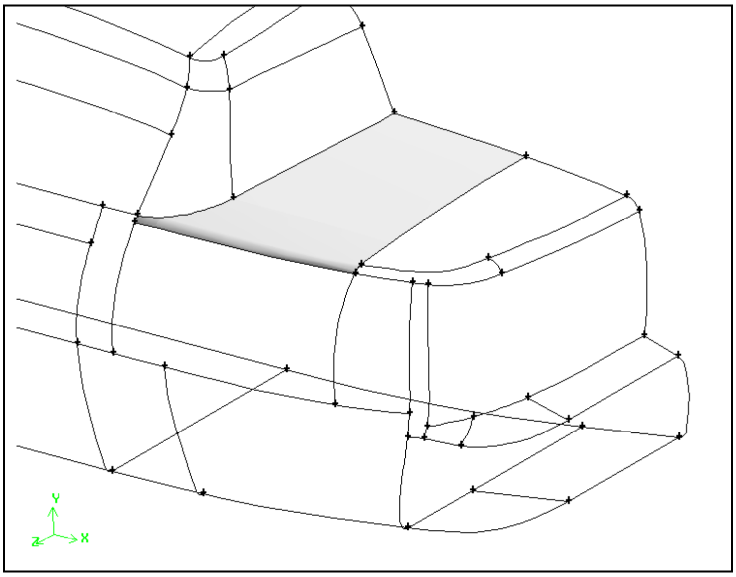


Figure 5-12: Four faces merged on trunk of sedan

3. Merge three faces near the rear end of the trunk of the car (see Figure 5-13) using the method described above. The merged faces are shown in Figure 5-14.

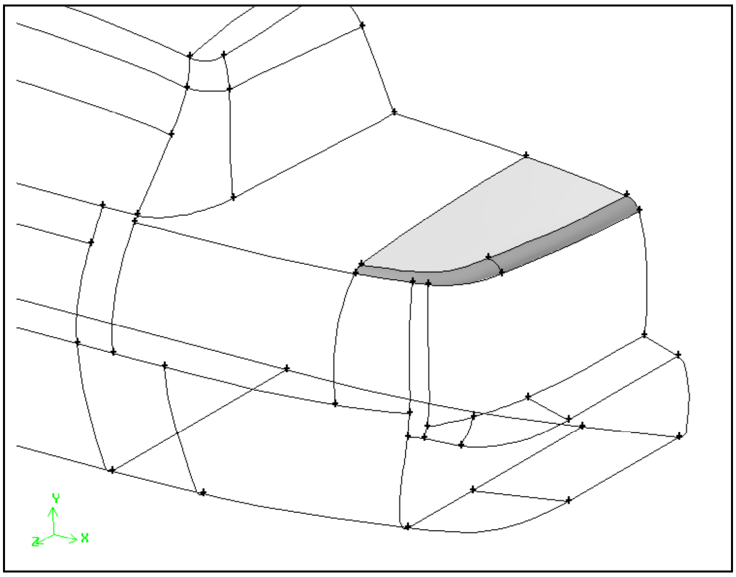


Figure 5-13: Three faces near rear end of trunk

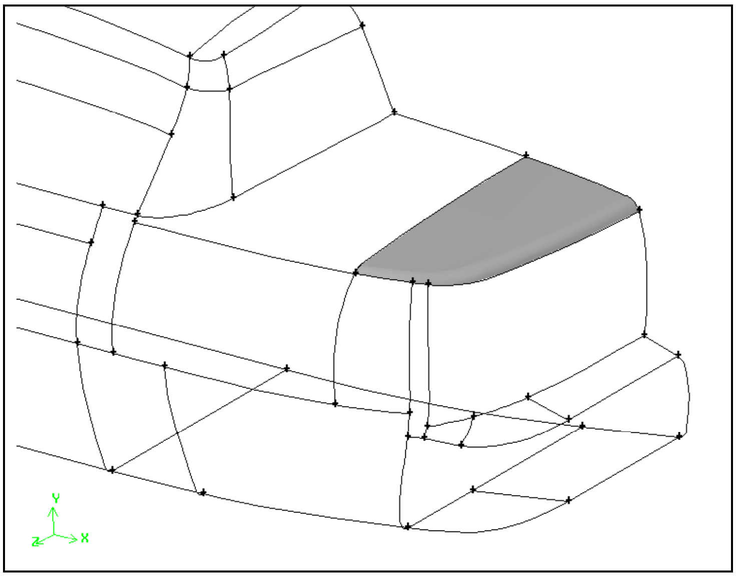


Figure 5-14: Three faces merged near rear end of trunk

4. Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full sedan in the graphics window.

   The top portion of the trunk should now consist of two large faces, as shown in Figure 5-15.

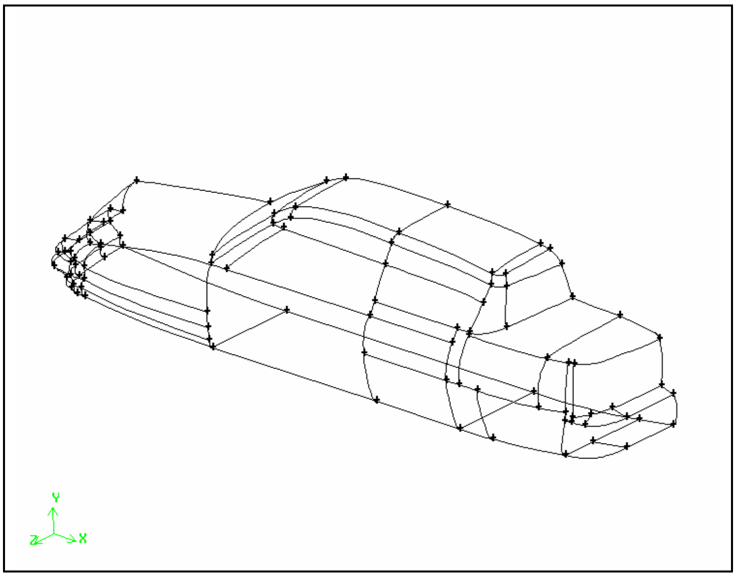
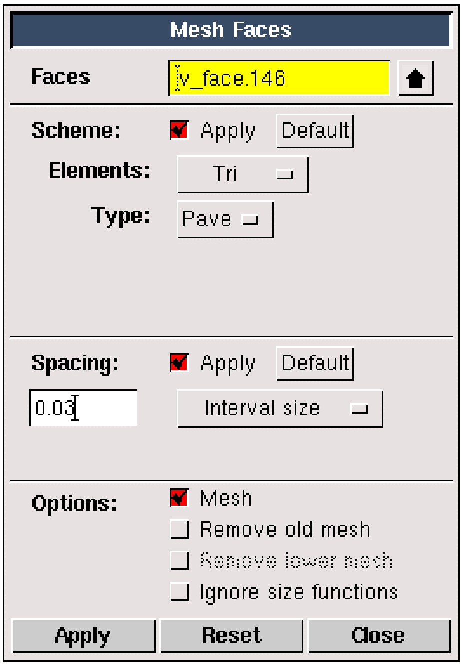


Figure 5-15: Merged faces on sedan

Step 7: Mesh Faces on Car Body

1. Create a surface mesh on the faces of the car body.

   MESH → **FACE** → **MESH FACES**



   a) Select all the faces on the car body by holding down the *Shift* key and using the left mouse button to drag a box around the whole geometry in the graphics window.

      ! *It may take a while for* GAMBIT *to select all the faces.* GAMBIT *analyzes each face to determine suitable meshing schemes. You should wait until all the edges turn red before going on to the next step.*

   b) Select Tri from the **Elements** option button under **Scheme**.

      GAMBIT *automatically selects the* **Type:**Pave *option. For more information on face meshing schemes, see the* GAMBIT *Modeling Guide.*

   c) Enter an Interval size of `0.03` under **Spacing** and click the **Apply** button at the bottom of the form.

GAMBIT *will mesh the car body surfaces. A portion of the mesh is shown in Figure 5-16.*

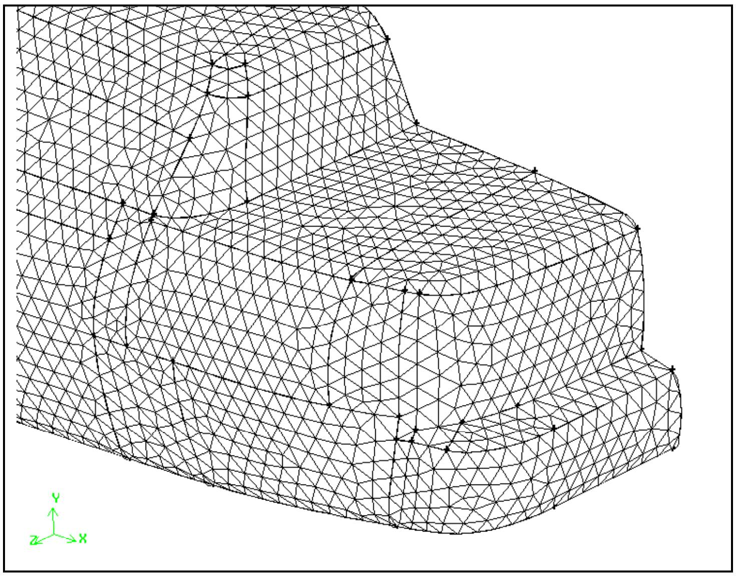


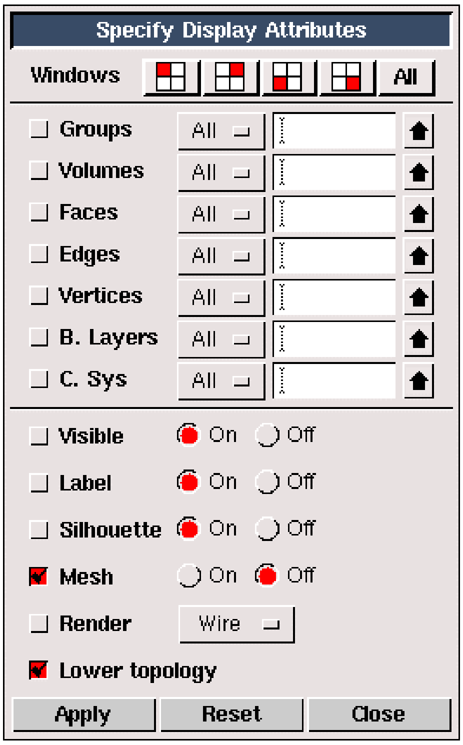
Figure 5-16: Surface mesh on rear of car body

2. Remove the mesh from the display.

! *This will make it easier to see what to do in the next steps. The mesh is not deleted, just removed from the graphics window.*

a) Click the **SPECIFY DISPLAY ATTRIBUTES** command button  at the bottom of the **Global Control** toolpad.

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



b) Select the Off radio button to the right of **Mesh** near the bottom of the form.

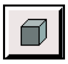
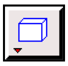
GAMBIT *will automatically select the* **Mesh** *check box.*

c) Click **Apply** and close the form.

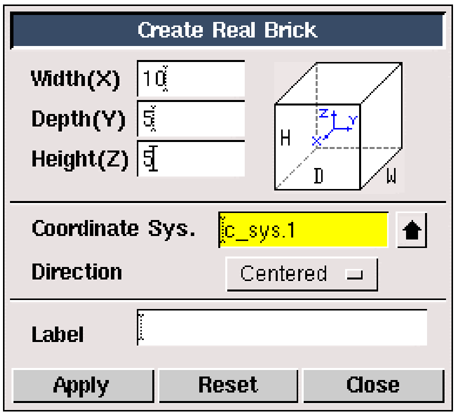
The mesh will be removed from the graphics window.

Step 8: Create a Brick Around the Car Body

1. Create a brick.

   GEOMETRY  → **VOLUME**  → **CREATE VOLUME** 

   This command sequence opens the **Create Real Brick** *form.*



   a) Enter a value of `10` for the **Width** of the brick.

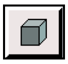
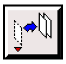
   b) Enter `5` for the **Depth** and `5` for the **Height**.

   c) Select Centered from the option menu to the right of **Direction**.

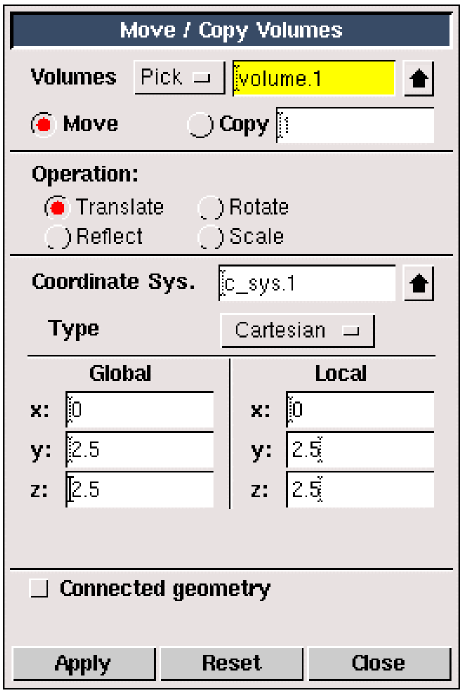
   d) Click **Apply**.

2. Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full sedan and the brick just created in the graphics window.

3. Move the brick to the desired location relative to the sedan.

   GEOMETRY  → **VOLUME**  → **MOVE/COPY/ALIGN VOLUMES** 

   This command sequence opens the **Move / Copy Volumes** *form.*



a) *Shift*-left-click the brick in the graphics window.

b) Select **Move** (the default) under **Volumes** in the **Move / Copy Volumes** form.

c) Select Translate (the default) under **Operation**.

d) Enter (`0`, `2.5`, `2.5`) under **Global** to move the brick 2.5 units in the *y* direction and 2.5 units in the *z* direction.

Note that GAMBIT *automatically fills in the values under* **Local** *as you enter values under* **Global**.

e) Click **Apply**.

4. Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full sedan and the brick in the graphics window.

   The brick and sedan are shown in Figure 5-17.

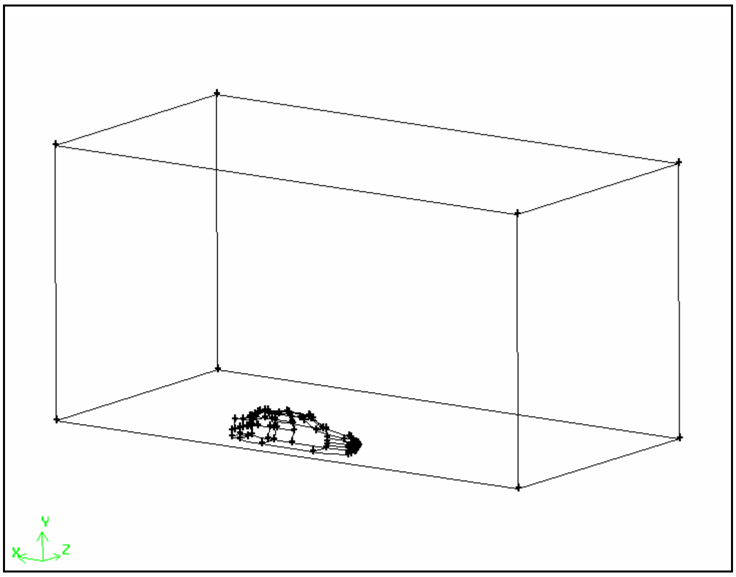
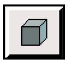
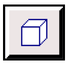


Figure 5-17: Brick and sedan

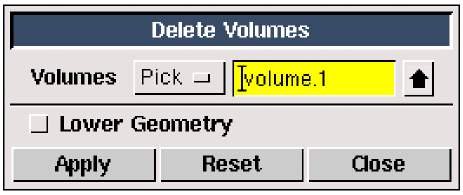
Step 9: Remove Unwanted Geometry

You cannot simply subtract the car from the brick to produce the flow domain around the car, because you used "virtual geometry" to clean up the car body and GAMBIT *cannot perform Boolean operations on virtual geometry. Instead, you must "stitch together" a virtual volume from the virtual faces of the car and the real faces of the brick. To do this you will delete the volume of the brick, leaving the lower geometry (the faces) behind. In the next steps, you will create virtual edges and faces.*

1. Delete the volume of the brick, leaving the faces behind.

   GEOMETRY  → **VOLUME**  → **DELETE VOLUMES** 

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



   a) *Shift*-left-click the brick in the graphics window.

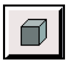
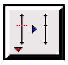
   b) Unselect the **Lower Geometry** option and click **Apply**.

      The brick volume will be deleted, but all its components (faces, edges, and vertices) will remain in the geometry, because you deselected the **Lower Geometry** *option.*

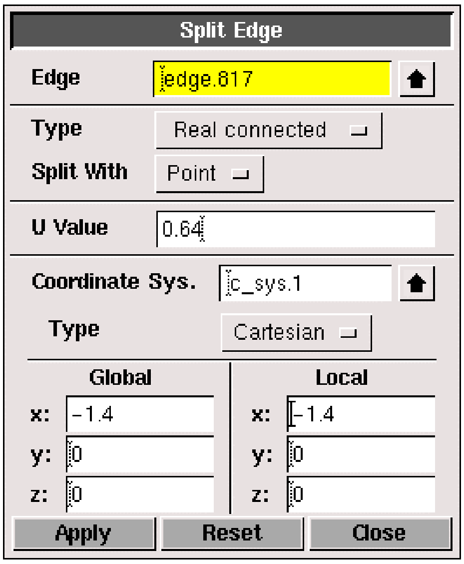
Step 10: Create Straight Edges on the Symmetry Plane

In this step, you will create two straight edges that will be used in the next step to create faces on the symmetry plane.

1. Split the bottom edge of the symmetry plane into three sections.

   GEOMETRY  → **EDGE**  → **SPLIT/MERGE EDGES** 

   This command sequence opens the **Split Edge** *form.*



   a) Retain the **Type:**Real connected option.

   b) Select **Split With** Point (the default).

      You will split the edge by creating a point on the edge and then using this point to split the edge.

   c) Use the *Ctrl* key and the left mouse button to zoom in to the sedan and the line at the bottom of the symmetry plane, similar to the view shown in Figure 5-18.

   d) Select the edge at the bottom of the symmetry plane in the graphics window.

   e) Enter a **U Value** of `0.64` in the **Split Edge** form and click **Apply**.

The vertex needs to be close to the front of the sedan. A **U Value** *of* `0.64` *will place the vertex in the correct position, but it is the position relative to the sedan that is important, not the exact* **U Value***.*

The edge is split into two parts and a vertex is created near the front bumper of the sedan, as shown in Figure 5-18.

f) Select the longer edge of the two edges just created in the graphics window.

g) Enter a **U Value** of `0.57` in the **Split Edge** form and click **Apply**.

Again, the position of the vertex relative to the sedan is more important than the exact **U Value***.*

The edge will be split and a second vertex created near the rear bumper of the sedan, as shown in Figure 5-18.

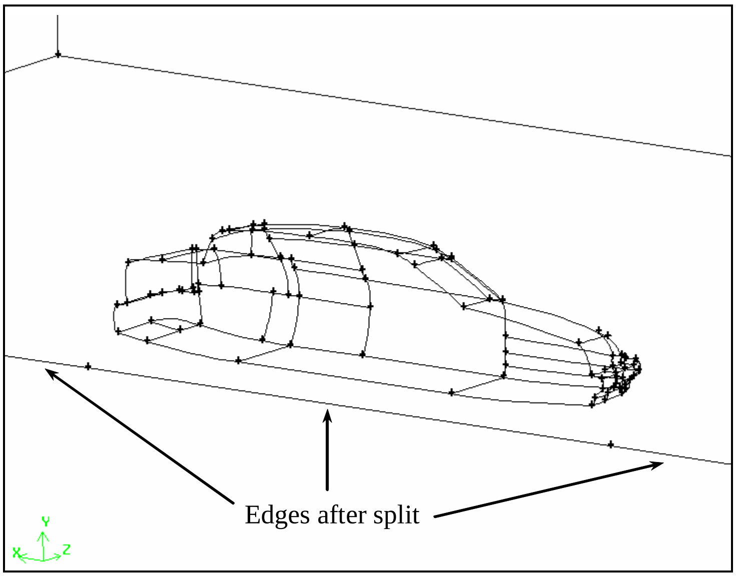
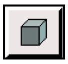
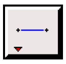
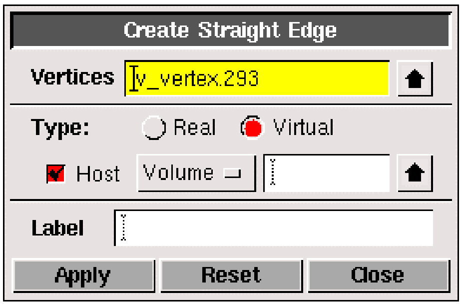


Figure 5-18: Bottom edge of symmetry plane is split into three edges

2. Create straight edges between the two points just created and two points on the sedan.

   GEOMETRY  → **EDGE**  → **CREATE EDGE** 

   This command sequence opens the **Create Straight Edge** *form.*



   a) Select the Virtual option to the right of **Type**.

      You must use Virtual *because the vertex to be used on the car body is a virtual vertex.*

   b) Zoom in to the front of the sedan, so that you can see the front bumper and the first vertex created on the edge at the bottom of the symmetry plane, as shown in Figure 5-19.

   c) *Shift*-left-click the first vertex created on the bottom edge of the symmetry plane.

   d) *Shift*-left-click the vertex on the sedan that is also on the symmetry plane, as shown in Figure 5-19.

      ! *Make sure that you select the vertex that is on the symmetry plane as well as the sedan. The vertex will be on an orange edge if it is on both the symmetry plane and the sedan geometry.*

   e) Click **Apply** to accept the selected vertices and create an edge, as shown in Figure 5-19.

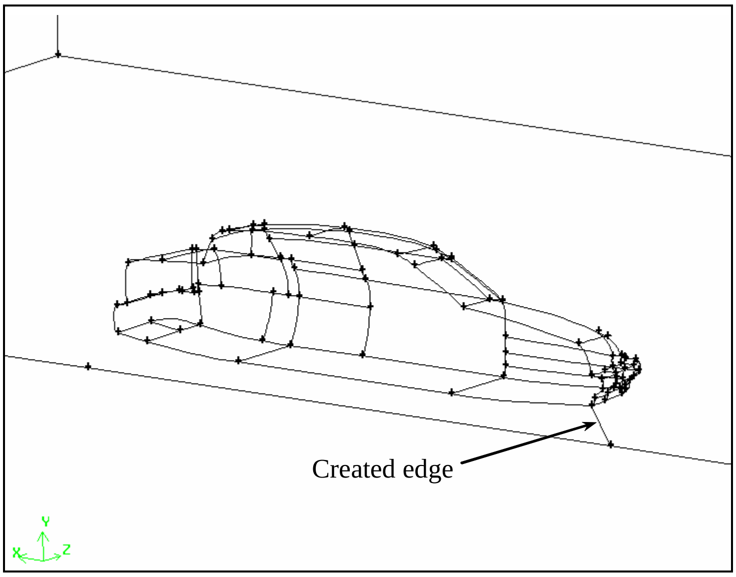


Figure 5-19: Edge from bottom of symmetry plane to front of sedan

3. Create a straight edge from the second vertex created on the bottom edge of the symmetry plane to the rear bumper of the sedan, as shown in Figure 5-20.

   ! *Again, make sure you select the vertex that is on both the sedan geometry and the symmetry plane.*

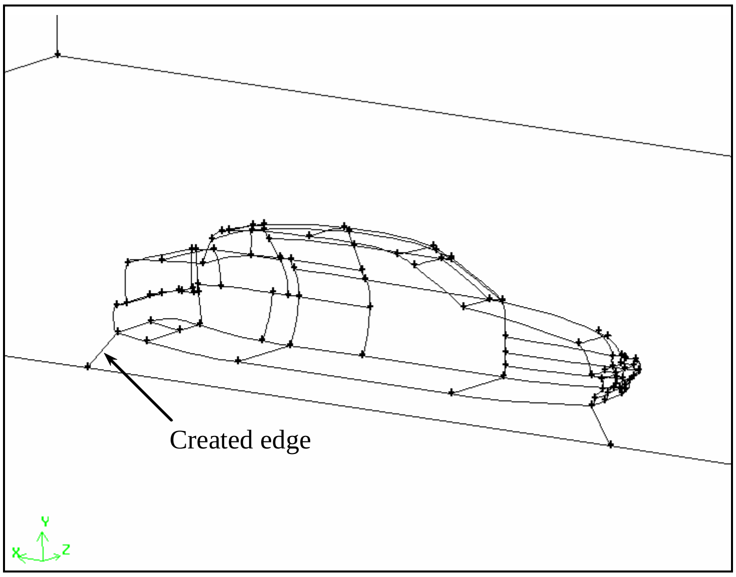


Figure 5-20: Edge from bottom of symmetry plane to rear of sedan

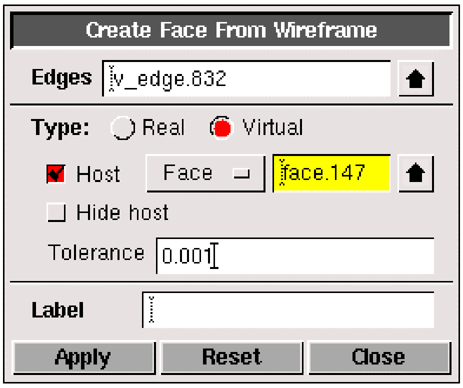
Step 11: Create Faces on the Symmetry Plane

In this step, you will create two new faces on the symmetry plane by stitching edges together. You will use the existing symmetry plane on the brick as a host. The two faces you create in this step will be used to create a volume in the next step.

1. Create a new face on the symmetry plane by stitching edges together.

   GEOMETRY → **FACE** → **FORM FACE**

   This command sequence opens the **Create Face From Wireframe** *form.*



   a) Select the **Type:**Virtual option.

      You must use Virtual *because the edges to be selected on the car body are virtual edges.*

   b) *Shift*-left-click the four edges underneath the sedan, the two small diagonal edges on the symmetry plane, and the middle edge at the bottom of the symmetry plane.

      ! *The area under the sedan where the edges to be selected are located is shown in Figure 5-21, and the edges to be selected are shown in Figure 5-22.*

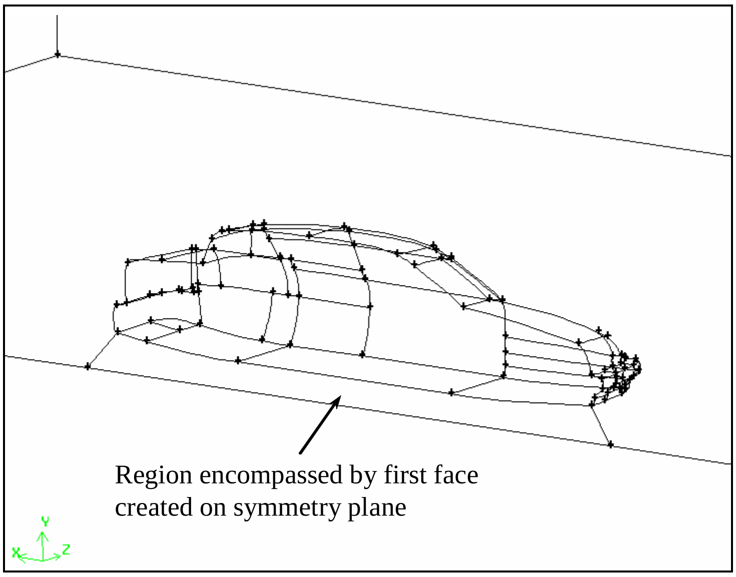


Figure 5-21: Area under sedan where edges to be selected are located

! *You should select seven edges in total. Pay particular attention to any very small edges. If you select an incorrect edge, Shift-middle-click on the edge to deselect it and select the edge next to it.*

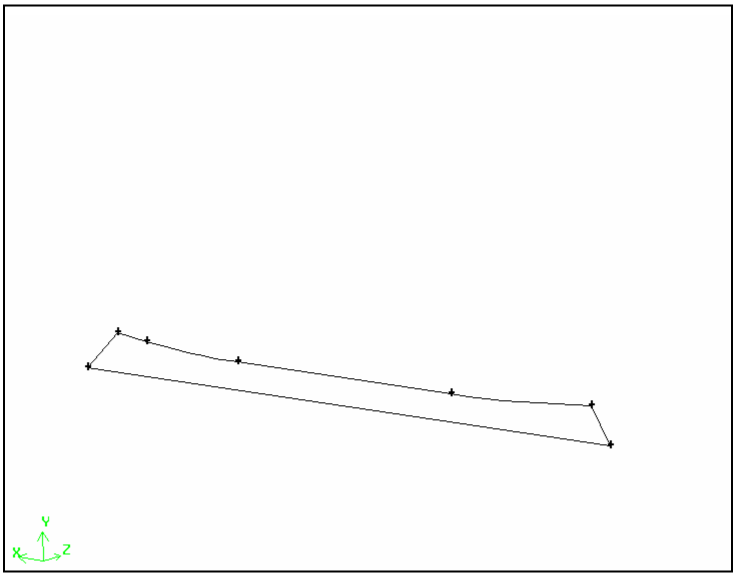


Figure 5-22: Edges used to create face at bottom of sedan

c) Select the Host check box in the **Create Face From Wireframe** form.

d) Select Face from the Host option menu.

e) *Shift*-left-click the back face of the brick (the symmetry plane) in the graphics window, as shown in Figure 5-23.

If you select the wrong face, Shift-middle-click on the face to deselect it and select the face next to it.

f) Enter `0.001` in the Tolerance text entry box.

g) Click **Apply** to accept the selection and create the face.

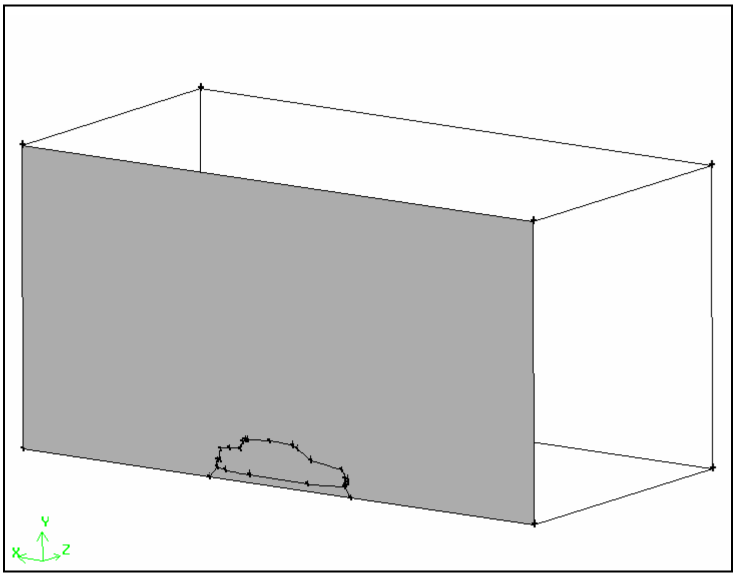


Figure 5-23: Symmetry plane of the brick

2. Create a second face on the symmetry plane.

   a) Check that the Virtual option is selected next to **Type**.

   b) Left-click in the **Edges** list box in the **Create Face From Wireframe** form.

   c) Select all the edges shown in Figure 5-24.

      ! *You should select 25 edges in total.*

   d) Left-click in the list box to the right of Host in the form.

   e) *Shift*-left-click the back face of the brick (the symmetry plane) in the graphics window, as shown in Figure 5-23.

   f) Click **Apply** to accept the selection and create the face.

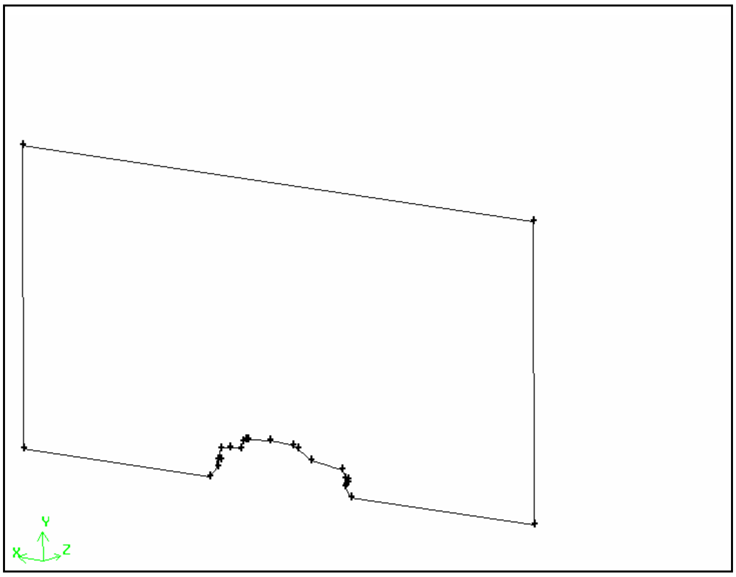
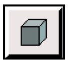
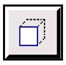
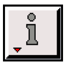
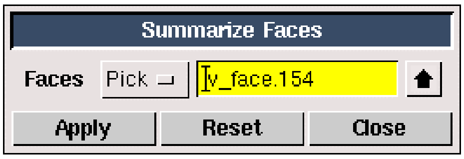


Figure 5-24: Edges used to create face at top of sedan

3. Verify the creation of the faces.

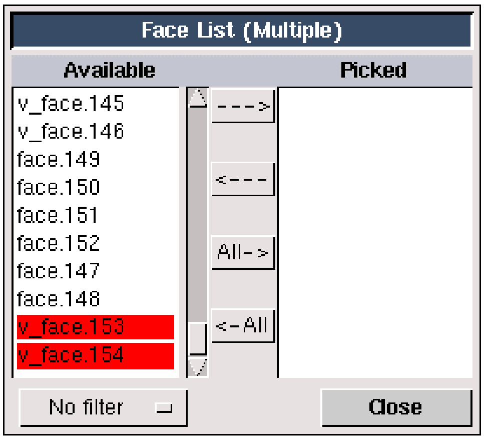
GEOMETRY  → **FACE**  → **SUMMARIZE/QUERY FACES/TOTAL ENTITIES** 

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



a) Left-click the black arrow to the right of the **Faces** list box.

This action opens the **Face List** *form. There are two types of pick-list forms:* **Single** *and* **Multiple***. In a* **Single** *pick-list form, only one entity can be selected at a time. In a* **Multiple** *pick-list form, you can select multiple entities.*



i. Select the two faces at the bottom of the **Available** list in the **Face List** form.

! *Note that the names of entities in the* **Available** *list may be different in your geometry. In the form shown above, the last two faces in the* **Available** *list are v_face.153 and v_face.154, but you might see faces with different numbers.*

ii. Click the – –> button to pick the two faces.

The two faces will be moved from the **Available** *list to the* **Picked** *list, and they will be highlighted in the graphics window.*

iii. Check that the two faces highlighted in the graphics window are the correct faces that you should have created in the previous steps.

Figure 5-22 and Figure 5-24 show the faces that you should have created.

iv. Close the **Face List** form.

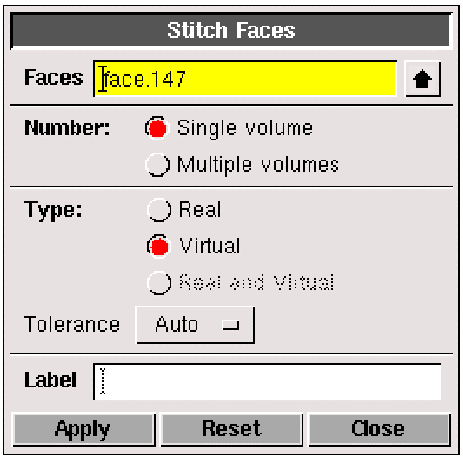
b) Click **Reset** in the **Summarize Faces** form to unselect the two faces in the graphics window.

Step 12: Create a Volume

1. Use the faces to create a volume.

   GEOMETRY → **VOLUME** → **FORM VOLUME**

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



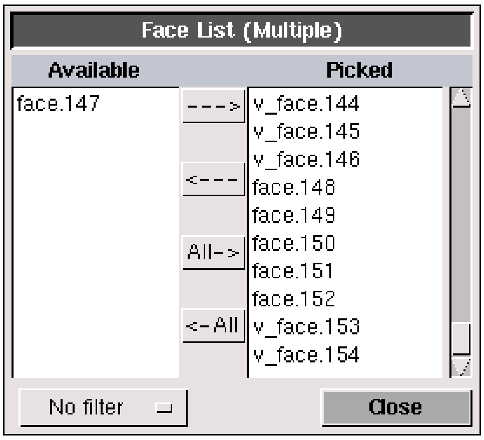
   a) Select the **Number:**Single volume option.

   b) Select the **Type:**Virtual option.

   c) Select the symmetry-plane face in the graphics window (as shown in Figure 5-23) and remember the label name (for example, *face.147*).

   d) Left-click the black arrow to the right of the **Faces** list box.

      This action opens the **Face List** *form.*



i. Click on the All –> button to move all the faces from the **Available** list to the **Picked** list.

ii. Select the name of the symmetry-plane face in the **Picked** list.

The symmetry-plane face will be highlighted in the graphics window.

iii. Click the <—— button to move the symmetry-plane face back into the **Available** list.

iv. Close the **Face List** form.

e) Click **Apply** in the **Stitch Faces** form to accept the selection of the faces and create the volume.

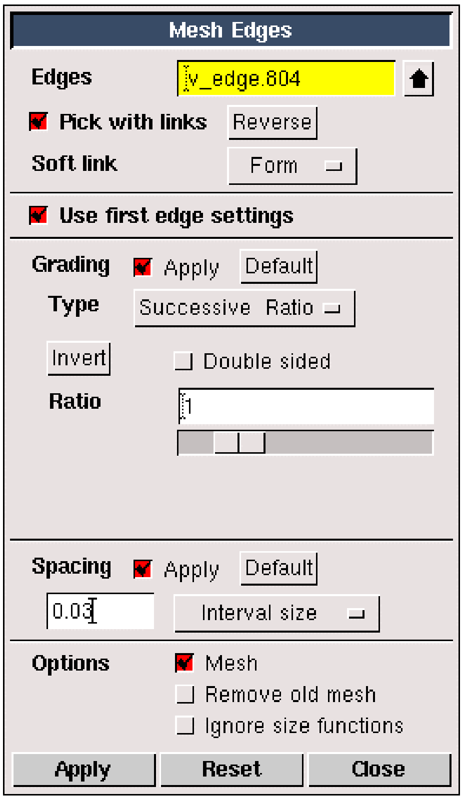
Step 13: Mesh the Edges

When you created the mesh on the faces of the sedan, you used a fine mesh. For the volume, you will create a more coarse mesh, so you will need to instruct GAMBIT *to gradually change the mesh density between the coarse and fine meshes. To do this, you will specify the distribution of nodes along some edges in the geometry.*

1. Define the grid density on three edges of the geometry underneath the sedan.

   MESH  → **EDGE**  → **MESH EDGES** 

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



   a) Select the edges marked A, B, and C in Figure 5-25 (the two small edges you created underneath the sedan and the middle section of the edge underneath the sedan that you split into three sections).

The edges will change color and an arrow and several circles will appear on each edge.

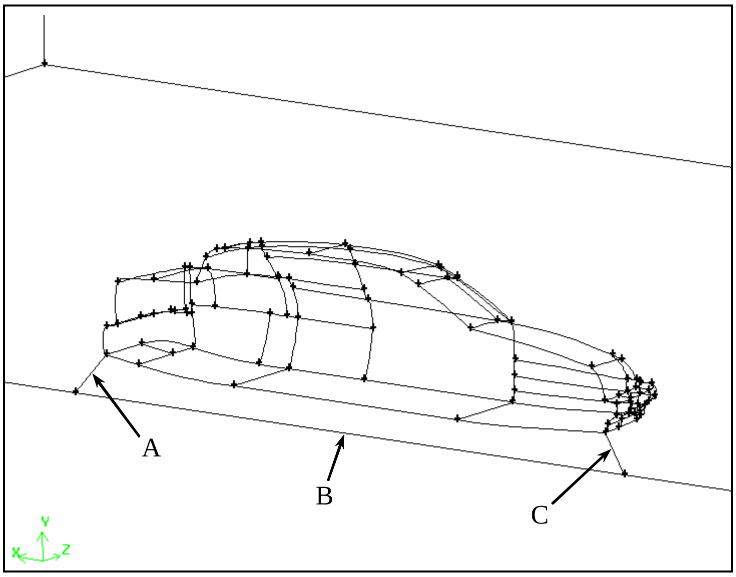


Figure 5-25: Edges underneath the sedan to be selected for edge meshing

b) Check that **Apply** is selected to the right of **Grading** in the **Mesh Edges** form and that Successive Ratio is selected from the **Type** option menu.

The Successive Ratio *option sets the ratio of distances between consecutive points on the edge equal to the* **Ratio** *specified in the* **Mesh Edges** *form.*

c) Retain the default **Ratio** of `1`.

d) Check that **Apply** is selected to the right of **Spacing**. Select Interval size from the option menu under **Spacing** and enter a value of `0.03` in the text entry box.

e) Click the **Apply** button at the bottom of the form.

Figure 5-26 shows the mesh on two of the edges underneath the sedan.

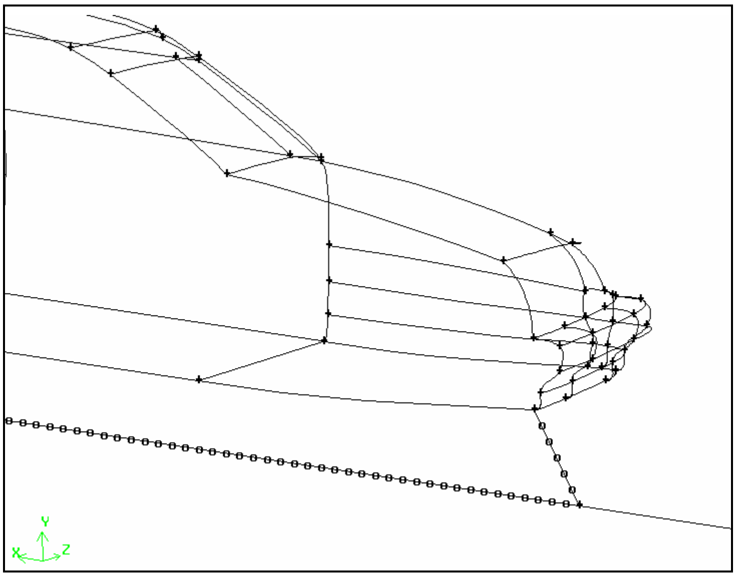


Figure 5-26: Edge meshing near the front of the sedan

2. Set the default variable that invokes flexible grading of the edges.

   a) Select **Defaults** from the **Edit** menu on the main menu bar.

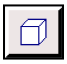
   b) On the **Edit Defaults** form, **Modify** the `MESH.EDGE.FLEXIBLE_GRADING` default variable to set it to `1`.

      The flexible-grading default variable controls the manner in which GAMBIT *meshes any ungraded edges that are connected to meshed or graded edges. If you set the* `MESH.EDGE.FLEXIBLE_GRADING` *default variable to 1,* GAMBIT *grades the edge such that its interval lengths adjacent to the connecting vertex are similar to those on the already meshed or graded edge(s) to which it is connected. If an ungraded edge is connected to more than one graded or meshed edges at a single vertex, GAMBIT averages the lengths on the graded and/or meshed edges to determine the appropriate interval length on the ungraded edge.*

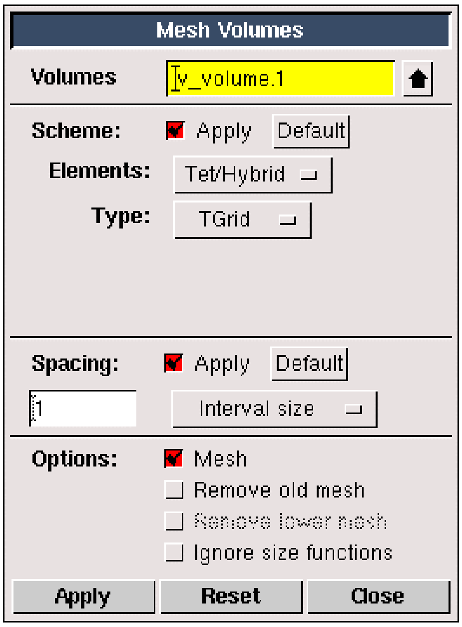
   c) **Close** the **Edit Defaults** form.

Step 14: Mesh the Volume

1. Mesh the volume with a coarser mesh than the mesh on the car faces.

   MESH  → **VOLUME**  → **MESH VOLUMES** 

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



   a) Select the volume in the graphics window.

   b) Select Tet/Hybrid from the **Elements** option menu under **Scheme** in the **Mesh Volumes** form, and select TGrid from the **Type** option menu.

      See the GAMBIT *Modeling Guide for more information on meshing schemes.*

   c) Retain the default **Interval size** of `1` under **Spacing** and click the **Apply** button at the bottom of the form.

      The surface of the volume mesh is shown in Figure 5-27.

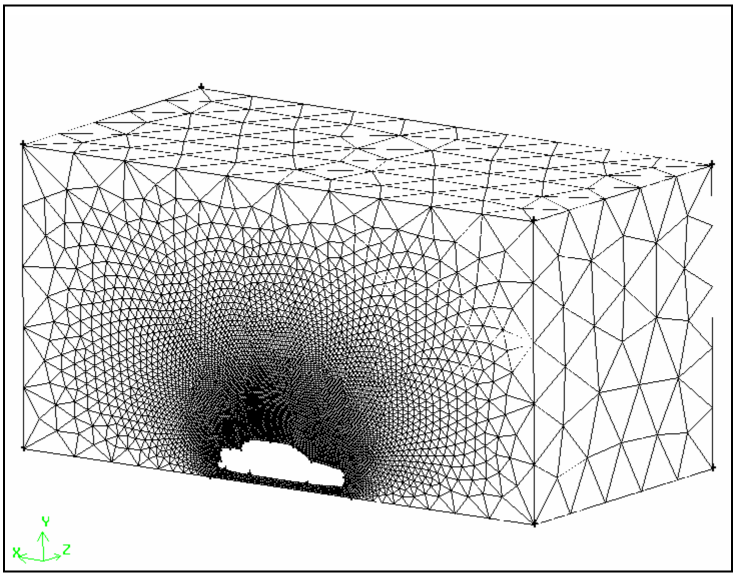
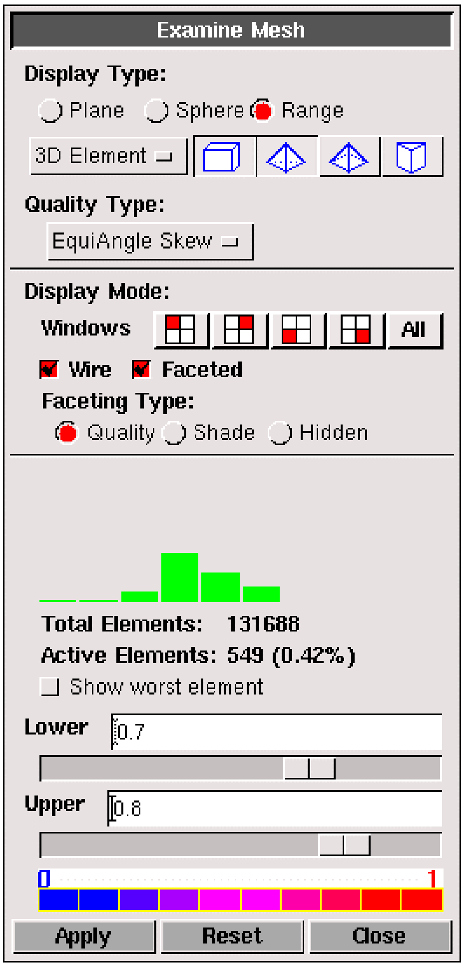


Figure 5-27: A portion of the volume mesh

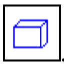
Step 15: Examine the Volume Mesh

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

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



   a) Select **Range** under **Display Type** at the top of the form.

The **3D Element** *type selected by default at the top of the form is a brick* *. You will not see any mesh elements in the graphics window when you first open the* **Examine Mesh** *form, because there are no hexahedral elements in the mesh.*

b) Left-click on the tetrahedron icon  next to **3D Element** near the top of the form.

The mesh elements will now be visible in the graphics window.

c) Select EquiSize Skew from the **Quality Type** option menu.

This is the default skewness measure for tetrahedra in TGrid.

d) Left-click the histogram bars that appear at the bottom of the **Examine Mesh** form to highlight elements in a particular quality range.

Figure 5-28 shows the view in the graphics window if you click on the fifth bar from the right on the histogram (representing cells with a skewness value between 0.7 and 0.8). These low values for the maximum skewness indicate that the mesh is acceptable.

The histogram consists of a bar chart representing the statistical distribution of mesh elements with respect to the specified **Quality Type***. Each vertical bar on the histogram corresponds to a unique set of upper and lower quality limits.*

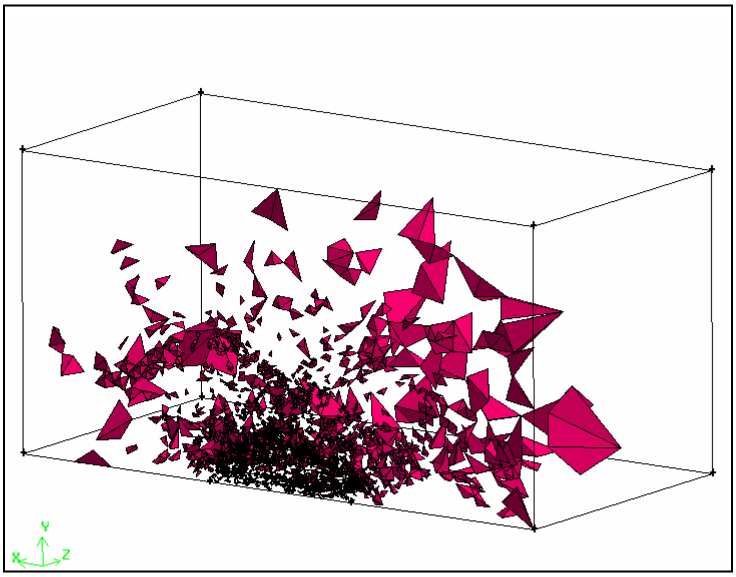


Figure 5-28: Elements within a specified quality range

e) Close the **Examine Mesh** form by clicking the **Close** button at the bottom of the form.

Step 16: Set Boundary Types

1. Remove the mesh from the display before you set the boundary types.

   This makes it easier to see the edges and faces of the geometry. The mesh is not deleted, just removed from the graphics window.

   a) Click the **SPECIFY DISPLAY ATTRIBUTES** command button  at the bottom of the **Global Control** toolpad.

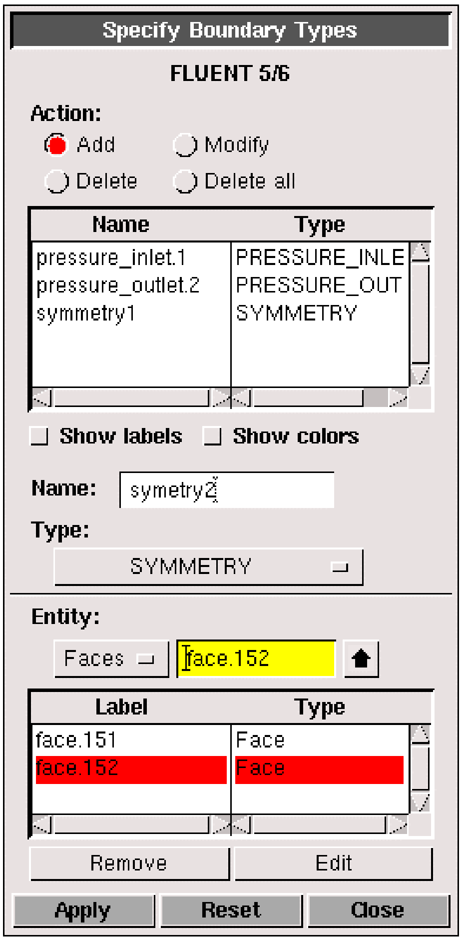
   b) Select the Off radio button to the right of **Mesh** near the bottom of the form.

   c) Click **Apply** and close the form.

2. Set boundary types for the sedan.

   ZONES  → **SPECIFY BOUNDARY TYPES** 

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



a) Define the pressure inlet boundary.

i. Select PRESSURE_INLET in the **Type** option menu.

ii. Check that Faces is selected as the **Entity**.

iii. *Shift*-left-click the face on the brick in front of the car in the graphics window (marked A in Figure 5-29) and click **Apply** to accept the selection.

This face will be set as a pressure inlet.

GAMBIT *will give the boundary a default name based on what you select in the* **Type** *and* **Entity** *lists (pressure_inlet.1 in this example). You can also specify a name for a boundary by entering a name in the* **Name** *text entry box.*

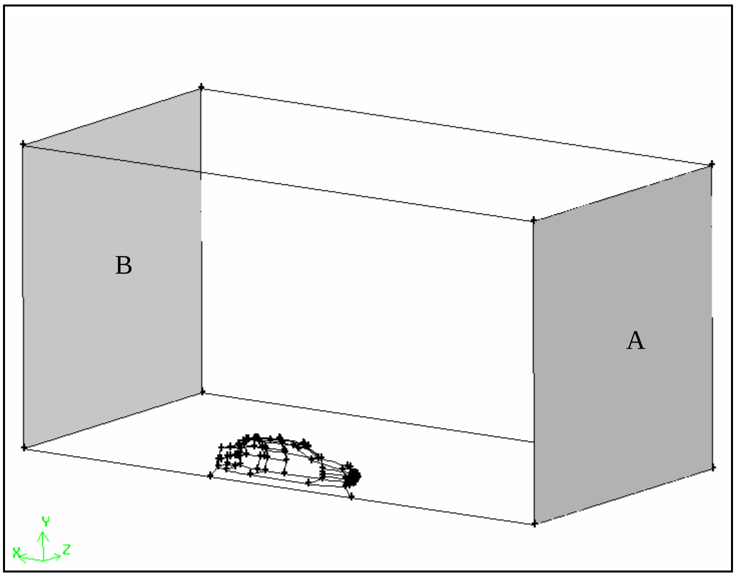


Figure 5-29: Pressure inlet (A) and pressure outlet (B) for the sedan geometry

b) Define the pressure outlet boundary.

i. Change the **Type** to PRESSURE_OUTLET by selecting it from the option menu below **Type**.

ii. Select the face on the brick behind the car in the graphics window (marked B in Figure 5-29) and click **Apply** to accept the selection.

c) Define symmetry boundary types for the two faces on the symmetry plane of the brick.

i. Enter "`symmetry1`" in the **Name** text entry box.

ii. Select SYMMETRY from the **Type** option menu.

iii. Select the two faces you created on the symmetry plane of the brick (the faces marked C and D in Figure 5-30) and click **Apply** to accept the selection.

GAMBIT *will merge the two faces into a single symmetry zone.*

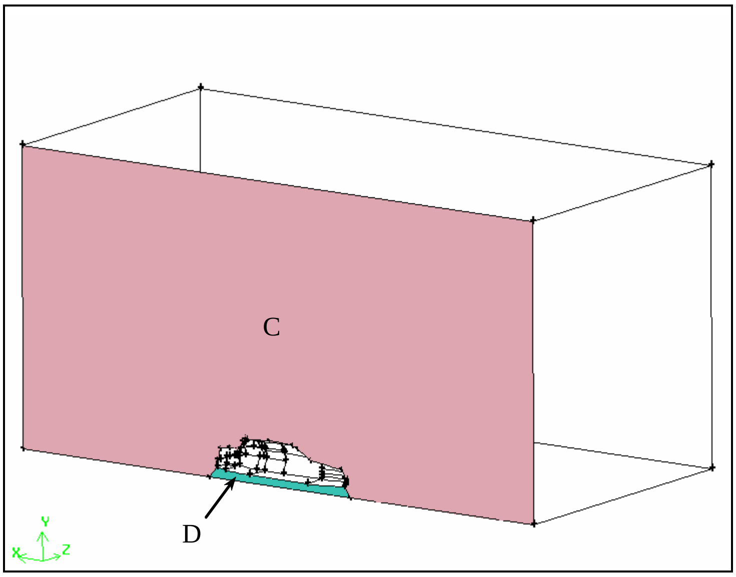


Figure 5-30: Two faces created on the symmetry plane of the brick

d) Define symmetry boundary types for the top face of the brick and the side face opposite the symmetry plane.

i. Enter "`symmetry2`" in the **Name** text entry box.

ii. Check that SYMMETRY is selected in the **Type** option menu.

iii. Select the faces on the brick that are above and to the side of the sedan (the faces marked E and F in Figure 5-31) and accept the selection.

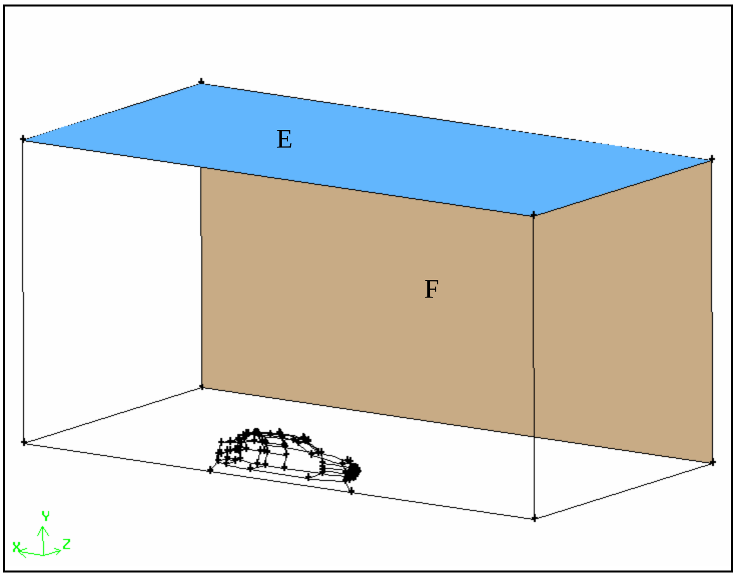


Figure 5-31: Two symmetry boundaries for the sedan geometry

The pressure inlet, pressure outlet, and symmetry boundaries for the sedan geometry are shown in Figure 5-32. (NOTE: To display the boundary types in the graphics window, select the **Show labels** *options on the* **Specify Boundary Types** *form.)*

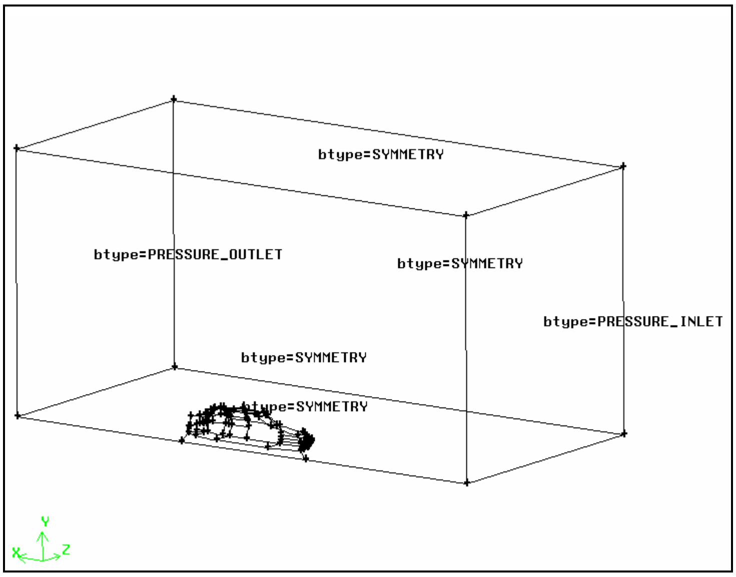


Figure 5-32: Boundary types for the sedan geometry

Note that you could also specify the remaining outer edges of the sedan geometry as wall boundaries. This is not necessary, however, because when GAMBIT *saves a mesh, any faces (in 3D) on which you have not specified a boundary type will be written out as wall boundaries by default.*

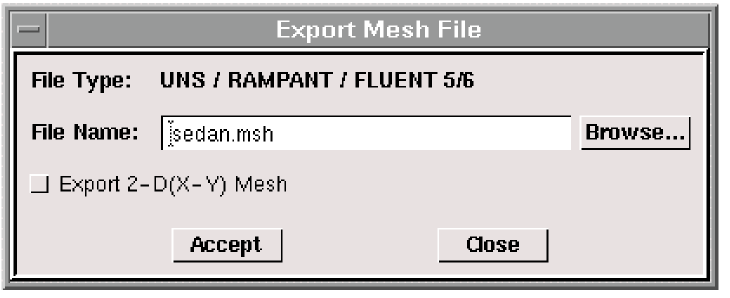
In addition, when GAMBIT *writes a mesh, any volumes (in 3D) on which you have not specified a continuum type will be written as fluid by default. This means that you do not need to specify a continuum type in the* **Specify Continuum Types** *form for this tutorial.*

Step 17: Export the Mesh and Save the Session

1. Export a mesh file for the sedan.

   File → **Export** → **Mesh...**

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



   a) Enter the **File Name** for the file to be exported (`sedan.msh`).

   b) Click **Accept** in the **Export Mesh File** form.

      The file will be written to your working directory.

2. Save the GAMBIT session and exit GAMBIT.

   File → **Exit**

   GAMBIT *will ask you whether you wish to save the current session before you exit.*

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

5.5 Summary

This tutorial illustrated how to import geometry from an external CAD package as an IGES file, and mesh it. Several geometry “cleanup” operations were demonstrated. Additional geometry was created to construct a box around the car-body geometry, and an unstructured tetrahedral volume mesh was generated.

6. SEDAN GEOMETRY—TOLERANT IMPORT

In this tutorial you will import an IGES file containing the geometry for a sedan automobile, clean up the geometry, and mesh it with triangles and tetrahedra.

In this tutorial you will learn how to:

- Import an IGES file using "tolerant modeling"
- Specify the way in which the geometry will be colored
- Merge faces to facilitate meshing
- Apply size functions to control mesh quality
- Mesh a volume with a tetrahedral mesh
- Prepare the mesh to be read into FLUENT 5/6

6.1 Prerequisites

This tutorial assumes that you are familiar with the GAMBIT GUI. You should also familiarize yourself with the previous tutorial, which employs GAMBIT virtual clean-up operations for importing the sedan geometry.

6.2 Problem Description

Figure 6-1 shows the sedan geometry used in this tutorial. The external body of the sedan is represented by a set of connected faces. To model the flow around the sedan body, you will create a brick volume that represents the flow domain.

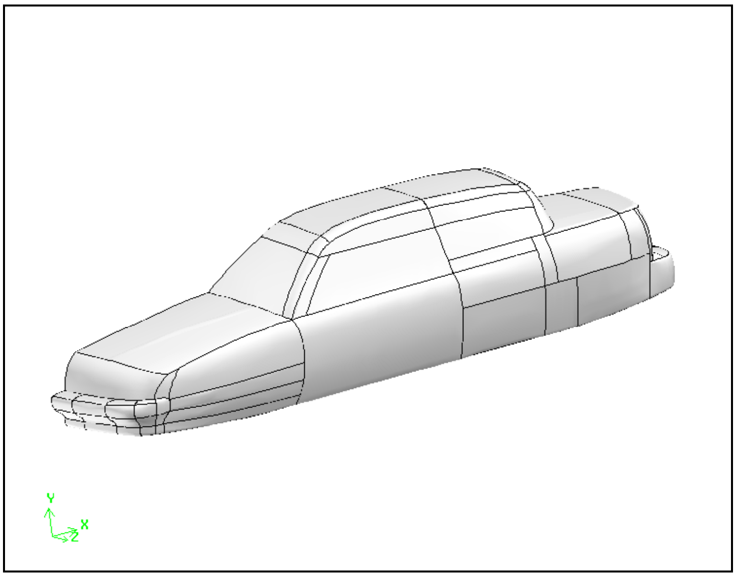


Figure 6-1: Sedan geometry

6.3 Strategy

In this tutorial, you will create a fully unstructured tetrahedral mesh around a car-body geometry imported as an IGES file. This tutorial illustrates the steps you would typically follow to prepare an imported CAD geometry for meshing. The IGES-file contains “dirty” geometry—that is, gaps exist between some of the surfaces that make it unsuitable for creating a CFD mesh. You will clean up the geometry using the GAMBIT “tolerant modeling” capability. The tolerant modeling option automatically assigns a tolerance value to each imported vertex and edge to maintain topological integrity for the imported model. The original CAD geometry is not modified during the import process.

The imported geometry includes a number of small surfaces, the edges of which may unnecessarily constrain the mesh generation process. Using the “merge faces” command, GAMBIT allows you to easily combine these surfaces prior to meshing. You can then have GAMBIT automatically create a triangular mesh on the car body.

Since the imported geometry consists only of the car body, you need to create a suitable domain around the car in order to conduct a CFD analysis (this is loosely equivalent to placing the car in a wind tunnel). The remainder of the tutorial shows how to add a real box around the car body, use virtual geometry to create some missing faces, and finally stitch all faces together into a single volume. This volume can then be meshed (without any decomposition) using a tetrahedral meshing scheme.

6.4 Procedure

1. Copy the file

    path/Fluent.Inc/gambit2.*x*/help/tutfiles/sedan.igs

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

2. Start GAMBIT.

Step 1: Select a Solver

1. Choose the solver from the main menu bar:

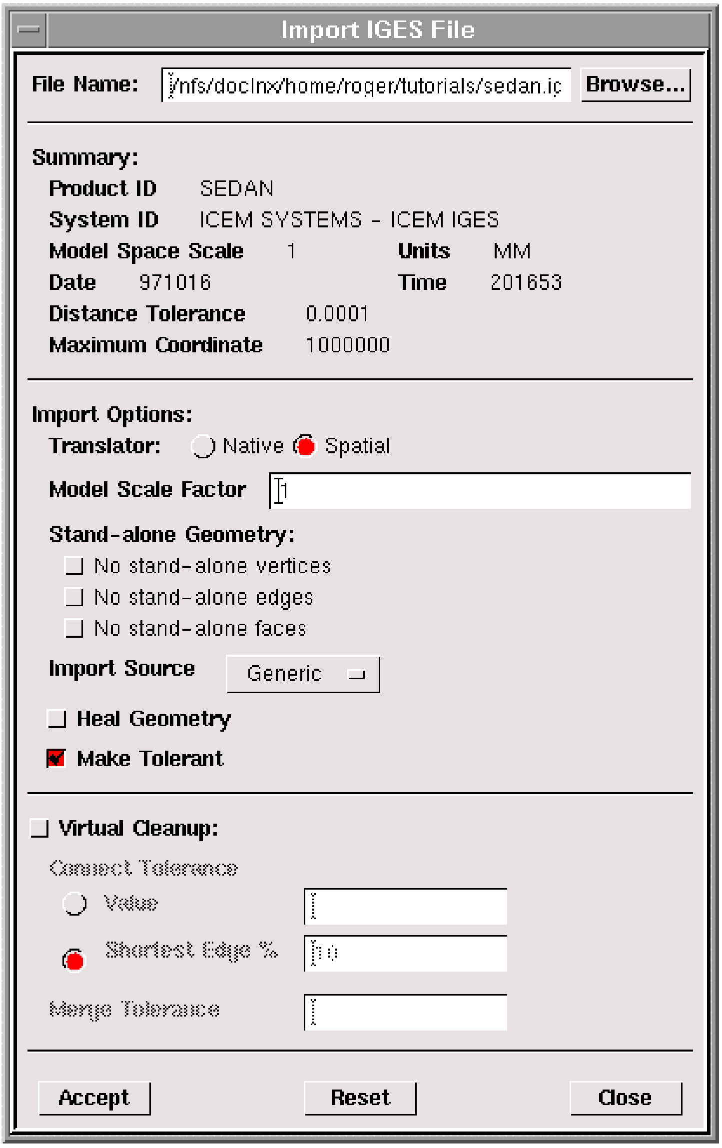
    Solver → **FLUENT 5/6**

    The choice of a solver dictates the options available in various forms (for example, the boundary types available in the **Specify Boundary Types** *form). For some systems,* **Fluent 5/6** *is the default solver. The solver currently selected is shown at the top of the* GAMBIT *GUI.*

Step 2: Import the IGES File

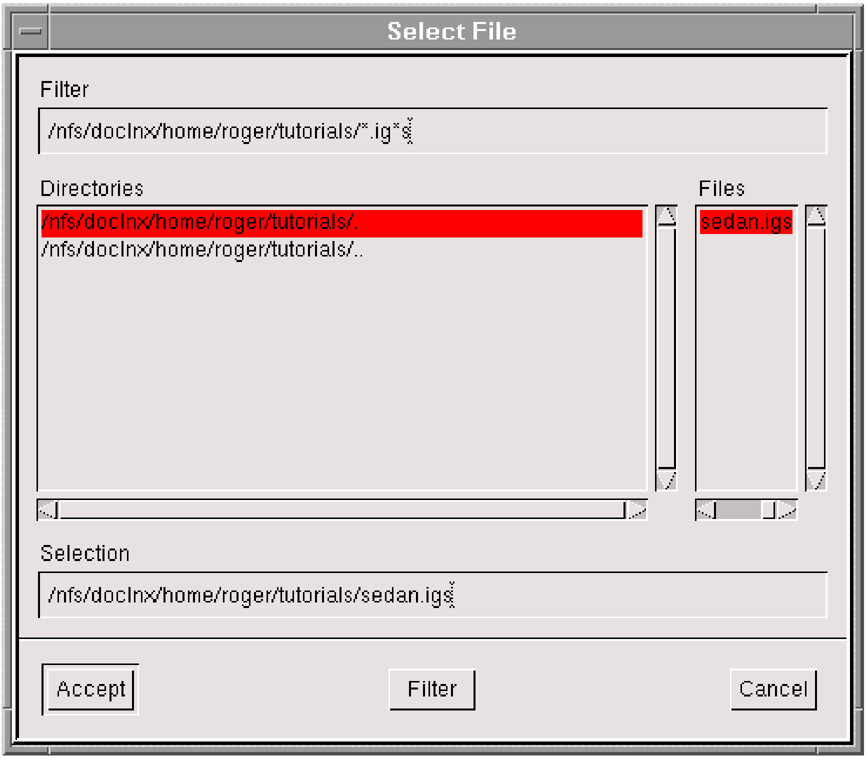
File −→ **Import** −→ **IGES ...**

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



1. Click on the **Browse...** button.

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



   a) Select `sedan.igs` in the **Files** list.

   b) Click Accept in the **Select File** form.

2. On the **Import IGES File** form, retain the **Make Tolerant** option.

   The **Make Tolerant** *option sets individual tolerances for edges and vertices so that entities that are not connected to within normal* GAMBIT *default tolerances are treated as connected entities.*

3. Click **Accept**.

   The IGES file for the sedan body will be read into GAMBIT *as real geometry (see Figure 6-2).*

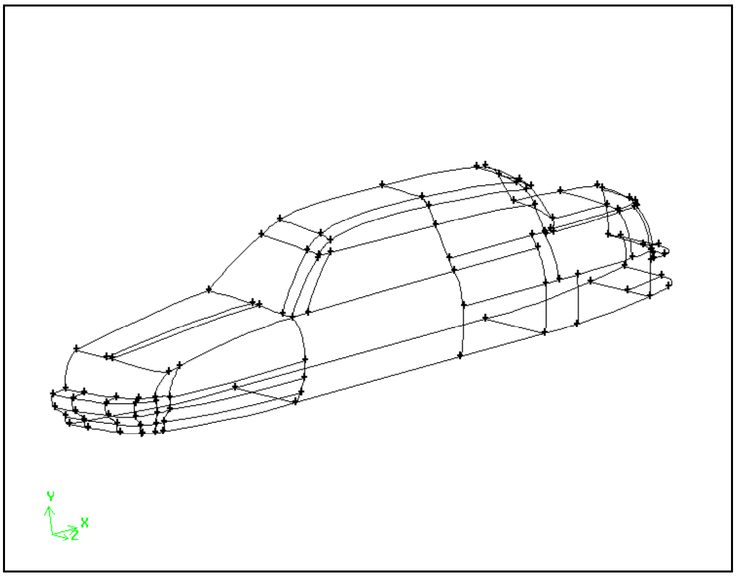
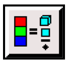
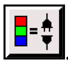


Figure 6-2: Imported sedan body

4. Click the **SPECIFY COLOR MODE** command button  in the **Global Control** toolpad to change the graphics display to connectivity-based coloring.

   The **SPECIFY COLOR MODE** *command button will change to* *. When* GAMBIT *is in the connectivity display mode, the model is displayed with colors based on connectivity between entities rather than based on entity types. In this case, the colors of all edges in the graphics window are blue, indicating that the faces are connected to each other.*

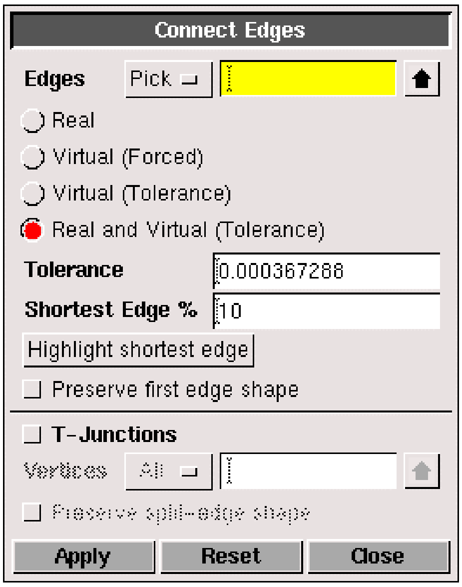
Step 3: Check for Very Short Edges

To ensure that the model does not contain very short edges that can inhibit meshing, you will highlight and examine the shortest edge in the model.

1. Find the shortest edge.

GEOMETRY → **EDGE** → **CONNECT/DISCONNECT EDGES**

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



a) Select the Real and Virtual (Tolerance) option.

b) Press the Highlight shortest edge button.

GAMBIT *will highlight the shortest edge—along with its label—in the graphics window.*

c) Zoom in near the highlighted edge by pressing the *Ctrl* key while using the mouse to drag a box around the edge.

Figure 6-3 shows the general area on the sedan that contains the shortest edge. Figure 6-4 shows a zoomed view of the edge.

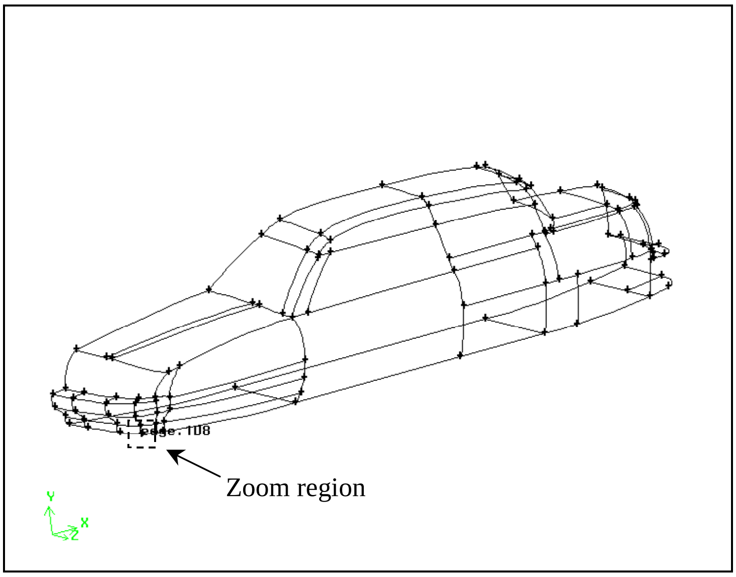


Figure 6-3: Sedan—showing general area of shortest edge location

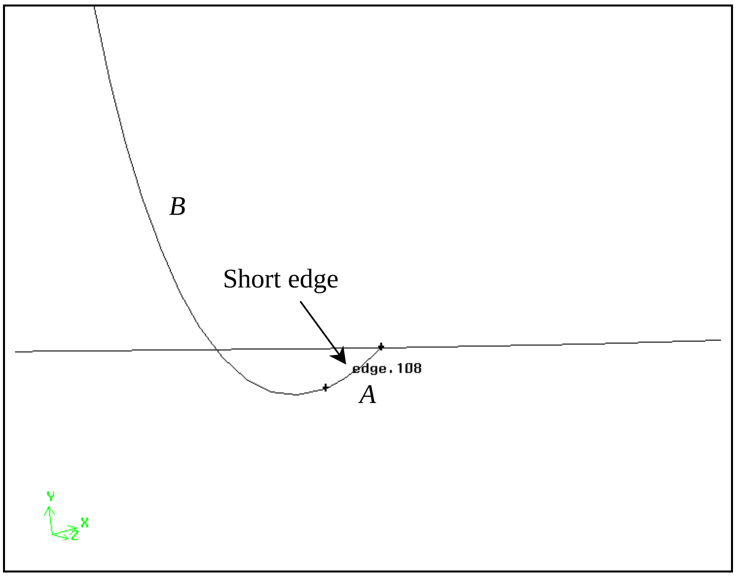


Figure 6-4: Sedan—showing zoomed area near shortest edge

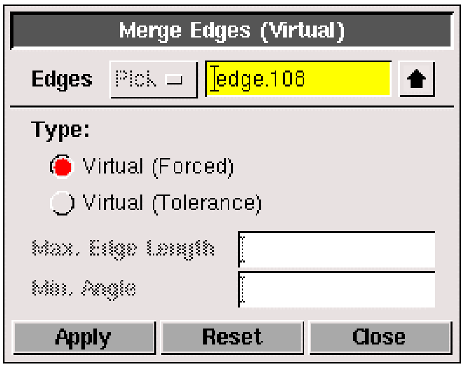
In this case, the shortest edge in the model is short enough to possibly cause meshing problems and should be removed.

Step 4: Merge Edges to Remove the Shortest Edge

1. Merge the shortest edge with its adjacent edge.

GEOMETRY → **EDGE** → **SPLIT/MERGE EDGES** R Merge Edges (Virtual)

This command sequence opens the **Merge Edges (Virtual)** *form.*



a) In the graphics window, select (pick) the shortest edge and the curved edge to which it is connected (edges *A* and *B* in Figure 6-4, above).

b) Retain the Virtual (Forced) option.

c) Click **Apply**.

GAMBIT *merges the edges, thereby eliminating the shortest edge.*

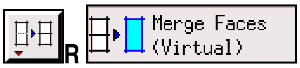
d) Click the **FIT TO WINDOW** command button at the top left of the **Global Control** toolpad to see the full sedan in the graphics window.

Step 5: Merge Faces

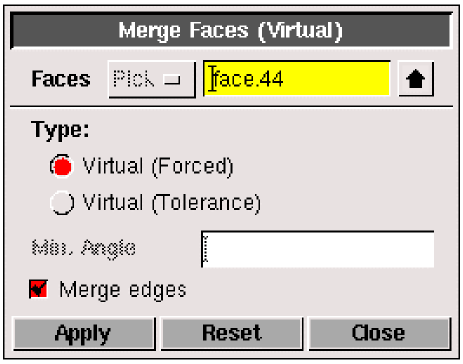
In many cases, the IGES model contains more detail than you need for meshing. The imported geometry for the sedan includes a number of small faces, the edges of which may constrain the mesh generation process unnecessarily. In GAMBIT, *you can merge faces together prior to meshing.*

1. Merge some of the faces on the sedan hood.

GEOMETRY → **FACE** → **SPLIT/MERGE/COLLAPSE/SIMPLIFY FACES**



This command sequence opens the **Merge Faces (Virtual)** *form.*



a) Retain the **Type:**Virtual (Forced) option.

b) Select (pick) the three faces on the top of the hood as shown in Figure 6-5.

c) Retain the Merge Edges option to facilitate geometry cleanup during merging.

d) Click **Apply** to merge the faces as shown in Figure 6-6.

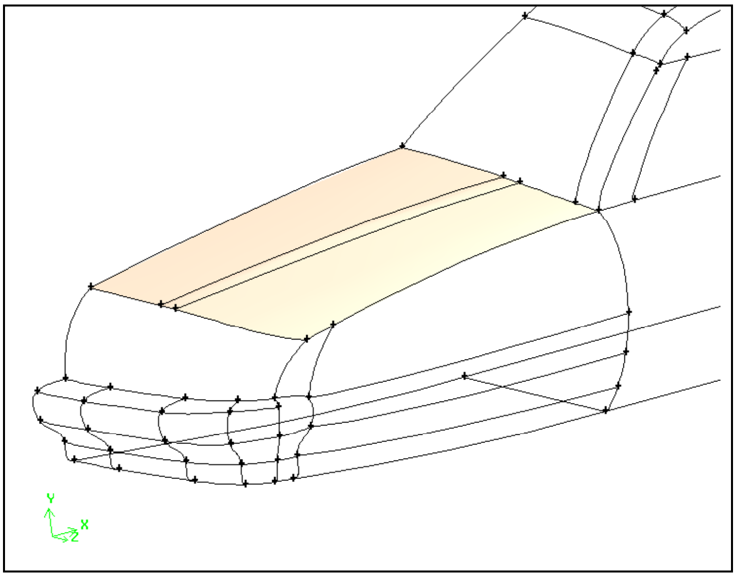


Figure 6-5: Three faces on hood of sedan

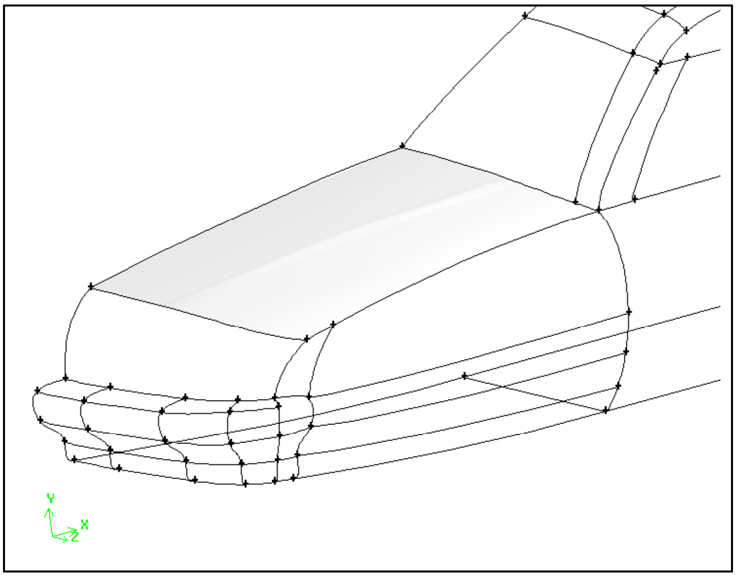


Figure 6-6: Three faces merged on hood of sedan

2. Merge four faces on the trunk of the car (see Figure 6-7) using the method outlined above. The merged faces are shown in Figure 6-8.

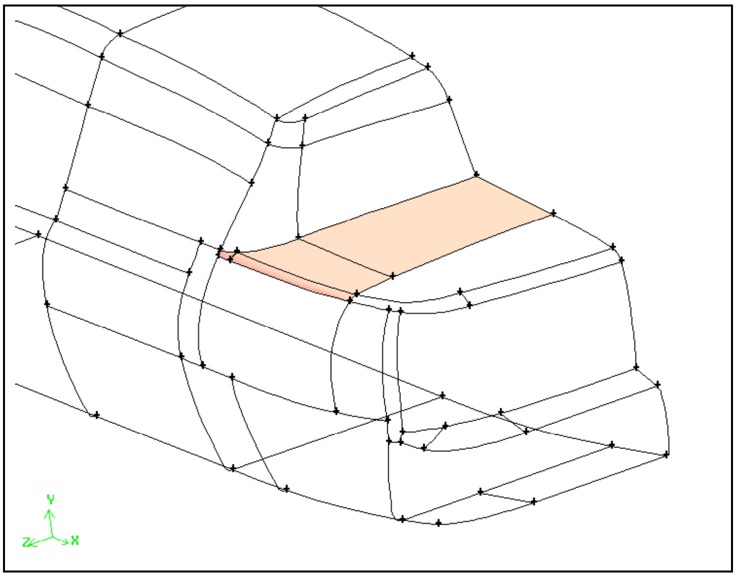


Figure 6-7: Four faces on trunk of sedan

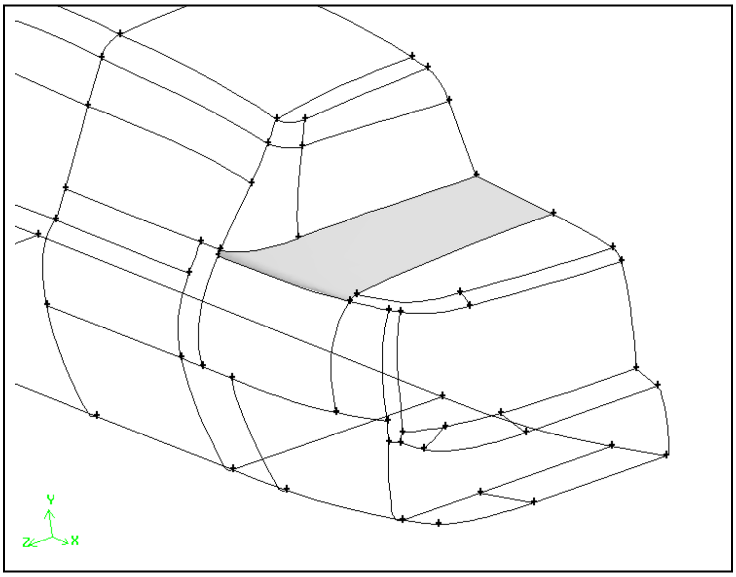


Figure 6-8: Four faces merged on trunk of sedan

3. Merge three faces near the rear end of the trunk of the car (see Figure 6-9) using the method outlined above. The merged faces are shown in Figure 6-10.

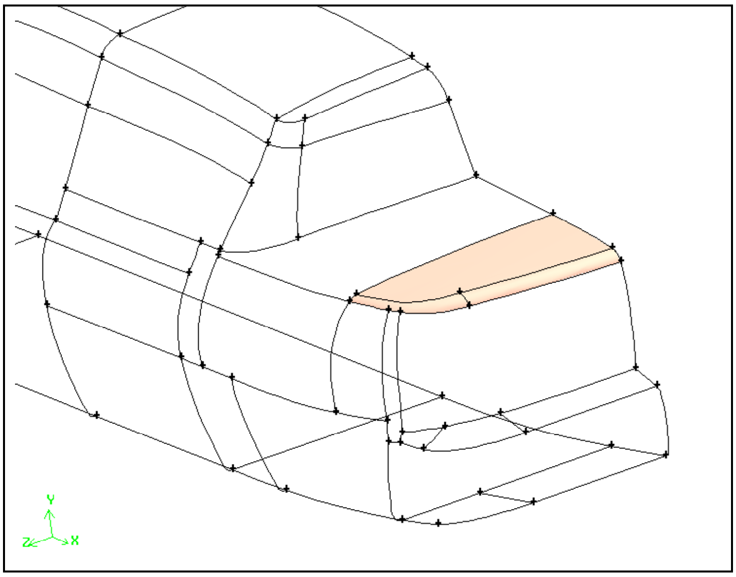


Figure 6-9: Three faces near rear end of trunk

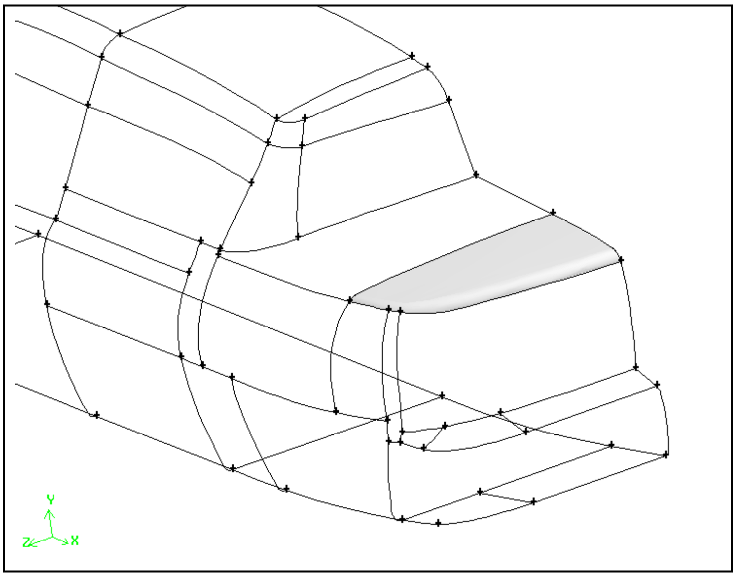


Figure 6-10: Three faces merged near rear end of trunk

4. Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full sedan in the graphics window.

   The top portion of the trunk should now consist of two large faces, as shown in Figure 6-11.

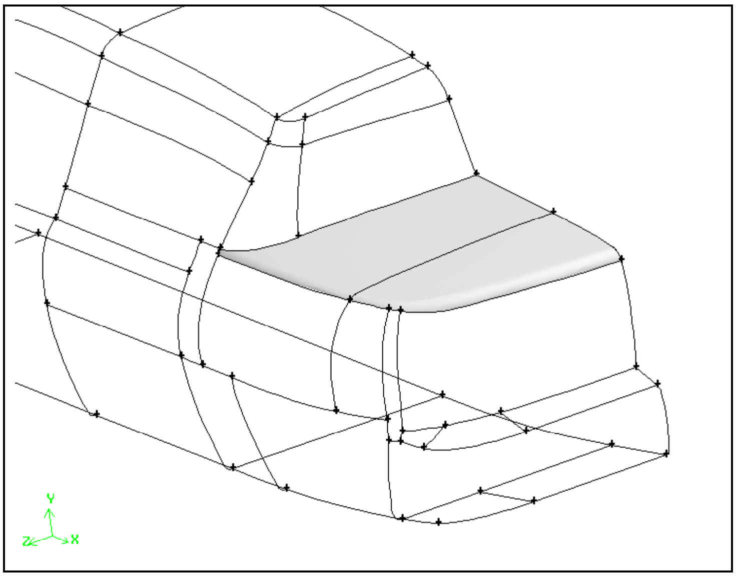
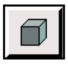
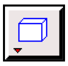


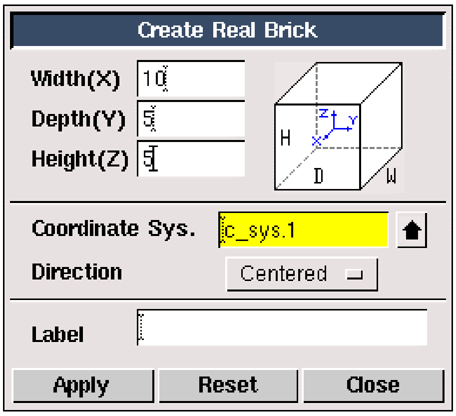
Figure 6-11: Merged faces on sedan

Step 6: Create a Brick Around the Car Body

1. Create a brick.

   GEOMETRY  → **VOLUME**  → **CREATE VOLUME** 

   This command sequence opens the **Create Real Brick** *form.*



   a) Enter a value of `10` for the **Width** of the brick.

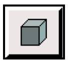
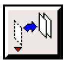
   b) Enter `5` for the **Depth** and `5` for the **Height**.

   c) Retain Centered from the option menu to the right of **Direction**.

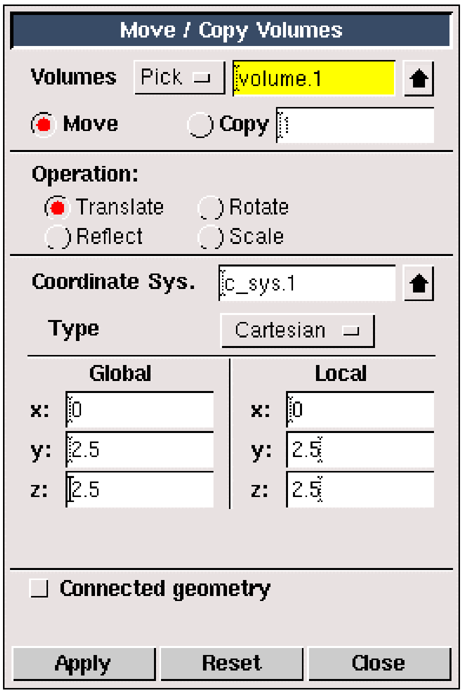
   d) Click **Apply**.

2. Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full sedan and the brick just created in the graphics window.

3. Move the brick to the desired location relative to the sedan.

   GEOMETRY  → **VOLUME**  → **MOVE/COPY/ALIGN VOLUMES** 

   This command sequence opens the **Move / Copy Volumes** *form.*



a) Select (pick) the brick in the graphics window.

b) Retain **Move** (the default) under **Volumes** in the **Move / Copy Volumes** form.

c) Retain Translate (the default) under **Operation**.

d) Enter (`0`, `2.5`, `2.5`) under **Global** to move the brick 2.5 units in the *y* direction and 2.5 units in the *z* direction.

Note that GAMBIT *automatically fills in the values under* **Local** *as you enter values under* **Global**.

e) Click **Apply**.

4. Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full sedan and the brick in the graphics window.

Figure 6-12 shows the brick and sedan geometry (vertices not shown).

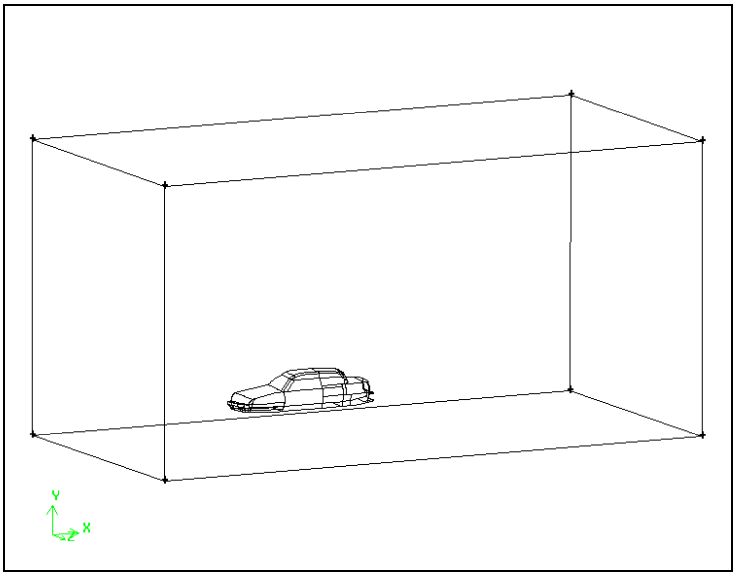


Figure 6-12: Brick and sedan geometry (vertices not shown)

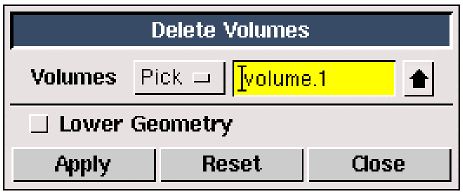
Step 7: Remove Unwanted Geometry

You cannot simply subtract the car from the brick to produce the flow domain around the car, because you used "virtual geometry" to clean up the car body and GAMBIT *cannot perform Boolean operations on virtual geometry. Instead, you must "stitch together" a virtual volume from the virtual faces of the car and the real faces of the brick. To do this you will delete the volume of the brick, leaving the lower geometry (the faces) behind. In the next steps, you will create virtual edges and faces.*

1. Delete the volume of the brick, leaving the faces behind.

   GEOMETRY → **VOLUME** → **DELETE VOLUMES**

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



   a) Select (pick) the brick in the graphics window.

   b) Unselect the **Lower Geometry** option and click **Apply**.

      The brick volume will be deleted, but all its components (faces, edges, and vertices) will remain in the geometry, because you unselected the **Lower Geometry** *option.*

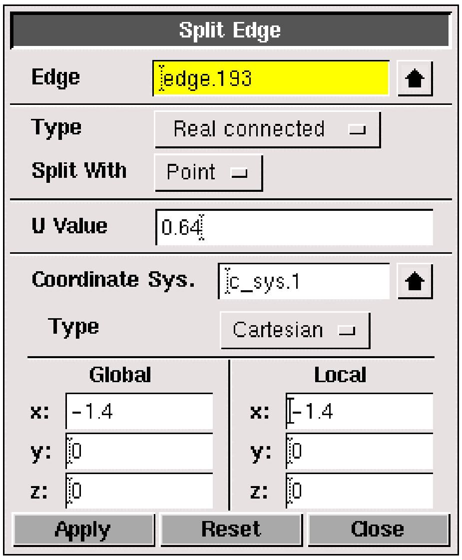
Step 8: Create Straight Edges on the Symmetry Plane

In this step, you will create two straight edges that will be used in the next step to create faces on the symmetry plane.

1. Split the bottom edge of the symmetry plane into three sections.

GEOMETRY → **EDGE** → **SPLIT/MERGE EDGES** R Split Edge

This command sequence opens the **Split Edge** *form.*



a) Retain the Real connected option (the default) next to **Type**.

b) Retain **Split With** Point (the default).

You will split the edge by creating a point on the edge and then using this point to split the edge.

c) Use the mouse to rotate the model and zoom in on the sedan and the edge at the bottom of the symmetry plane (see Figure 6-13).

d) Select (pick) the edge at the bottom of the symmetry plane.

e) Enter a **U Value** of `0.64` in the **Split Edge** form and click **Apply**.

The vertex needs to be close to the front of the sedan. A **U Value** *of* `0.64` *will place the vertex in the correct position, but it is the position relative to the sedan that is important, not the exact* **U Value**.

The edge is split into two parts and a vertex is created near the front bumper of the sedan, as shown in Figure 6-13.

f) Select the longer of the two edges at the bottom of the symmetry plane.

g) Enter a **U Value** of `0.57` in the **Split Edge** form and click **Apply**.

Again, the position of the vertex relative to the sedan is more important than the exact **U Value**.

The edge will be split and a second vertex created near the rear bumper of the sedan, as shown in Figure 6-13.

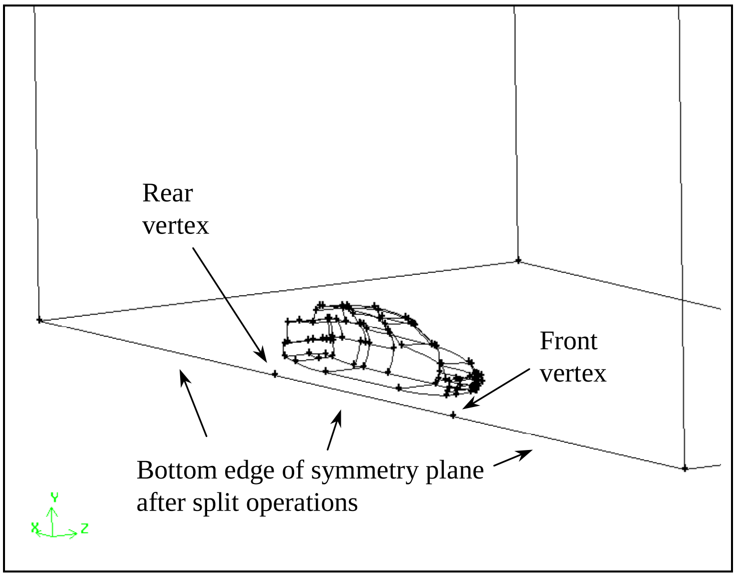
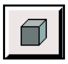
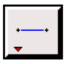
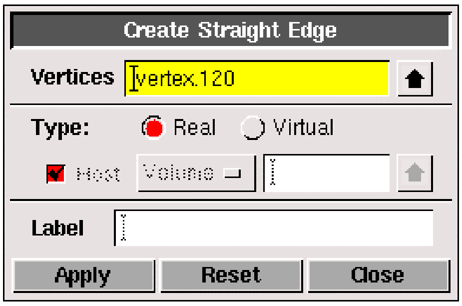


Figure 6-13: Bottom edge of symmetry plane is split into three edges

2. Create straight edges between the two points just created and two points on the sedan.

GEOMETRY  → **EDGE**  → **CREATE EDGE** 

This command sequence opens the **Create Straight Edge** *form.*



a) Retain the Real option to the right of **Type**.

b) Zoom in on the front of the sedan so that you can see the front bumper and the first vertex created by splitting the edge at the bottom of the symmetry plane (see Figure 6-14).

c) Select (pick) the first vertex created by splitting the edge at the bottom of the symmetry plane.

d) Select (pick) the closest vertex on the sedan.

! *Make sure to select the sedan vertex that is on the symmetry plane. The vertex will be on an orange edge if it is on both the symmetry plane and the sedan geometry.*

e) Click **Apply** to create the edge as shown in Figure 6-14.

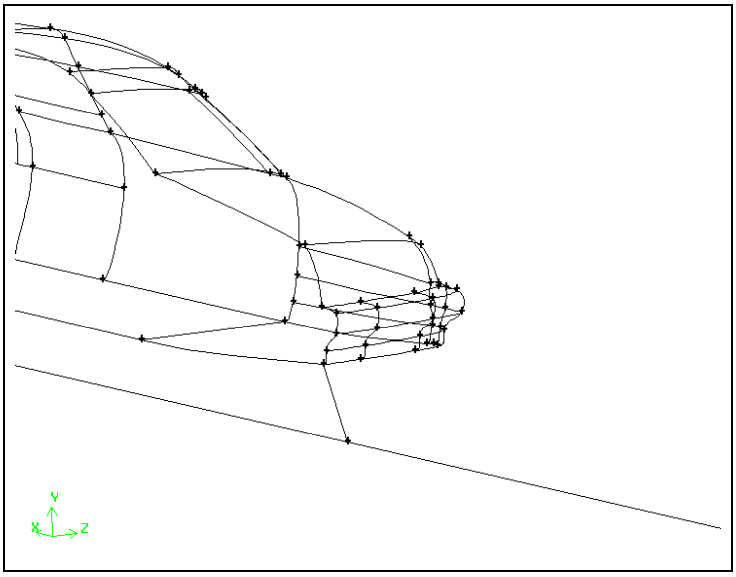


Figure 6-14: Edge from bottom of symmetry plane to front of sedan

3. Create a straight edge from the second vertex created on the bottom edge of the symmetry plane to the rear bumper of the sedan, as shown in Figure 6-15.

   ! *Again, make sure you select the vertex that is on both the sedan geometry and the symmetry plane.*

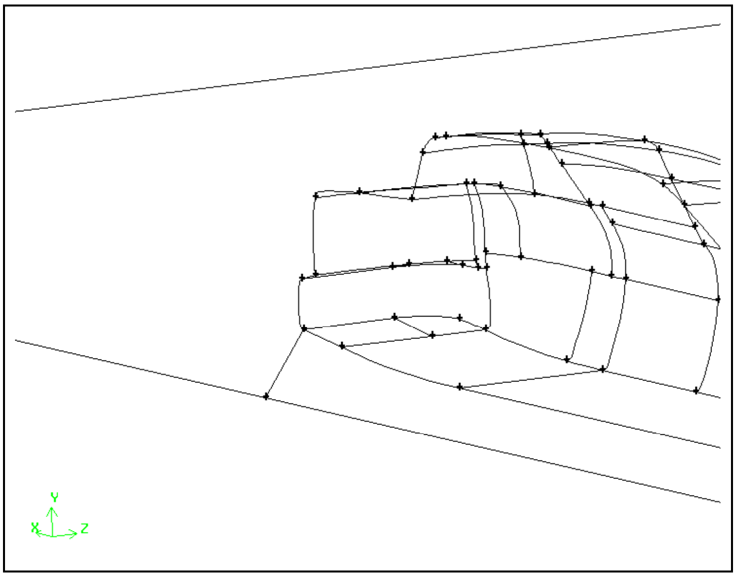


Figure 6-15: Edge from bottom of symmetry plane to rear of sedan

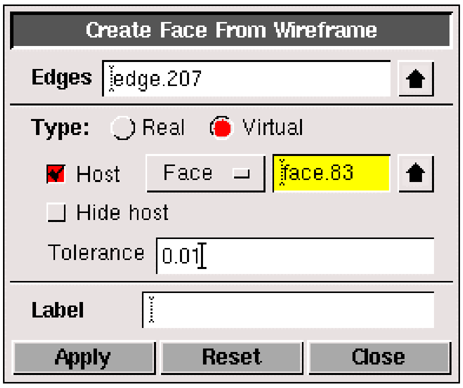
Step 9: Create Faces on the Symmetry Plane

In this step, you will create two new faces on the symmetry plane by stitching edges together. You will use the existing symmetry plane on the brick as a host. The two faces you create in this step will be used to create a volume in the next step.

1. Create a new face on the symmetry plane by stitching edges together.

   GEOMETRY → **FACE** → **FORM FACE**

   This command sequence opens the **Create Face From Wireframe** *form.*



   a) Select the **Type:**Virtual option.

   b) Select (pick) the edges underneath the sedan, the two small diagonal edges on the symmetry plane, and the middle edge at the bottom of the symmetry plane.

      ! *The area under the sedan where the edges to be selected are located is shown in Figure 6-16. You should select seven edges in total. Pay particular attention to any very small edges. If you select an incorrect edge, Shift-middle-click on the edge to unselect it and select the edge adjacent to it.*

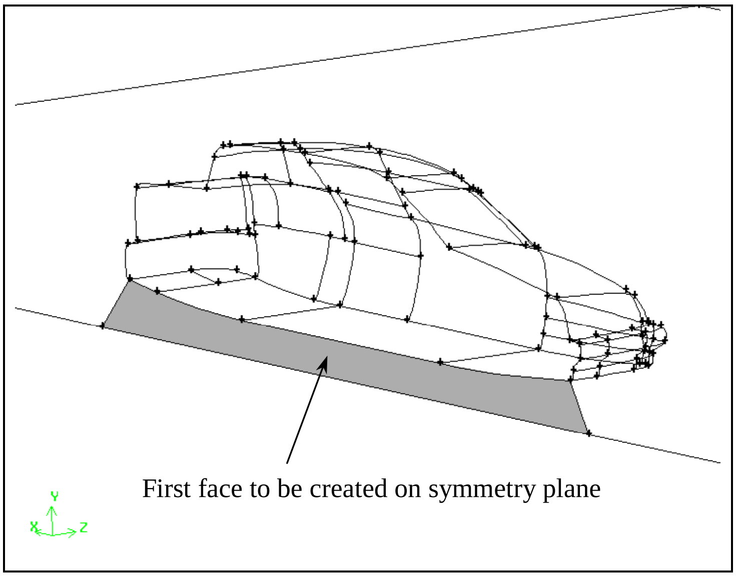


Figure 6-16: Region encompassing first face to be created on symmetry plane

c) Select the Host check box in the **Create Face From Wireframe** form.

d) Select Face from the Host option menu.

e) Select (pick) the symmetry plane of the brick (see Figure 6-17).

If you select the wrong face, Shift-middle-click on the face to unselect it and select the face adjacent to it.

f) Enter `0.01` in the **Tolerance** text entry box.

g) Click **Apply** to accept the selection and create the face.

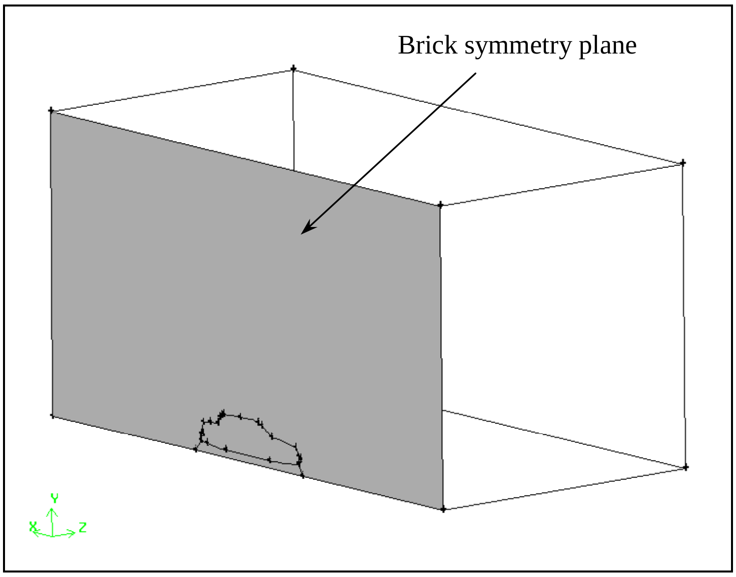


Figure 6-17: Brick symmetry plane

2. Create a second face on the symmetry plane.

   a) Check that the Virtual option is selected next to **Type**.

   b) Left-click in the **Edges** list box in the **Create Face From Wireframe** form.

   c) Select all the edges shown in Figure 6-18.

      ! *You should select 25 edges in total.*

   d) Left-click in the list box to the right of Host in the form.

   e) Select (pick) the symmetry plane of the brick (see Figure 6-17, above).

   f) Click **Apply** to accept the selection and create the face.

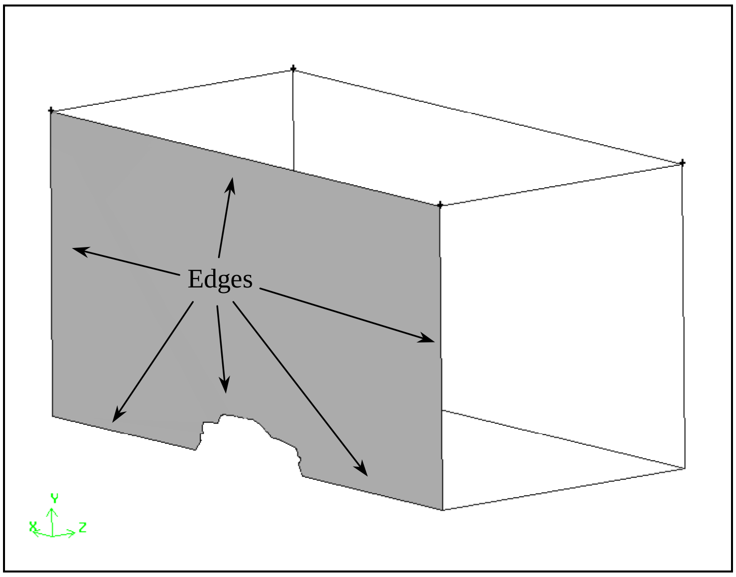
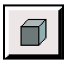
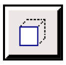
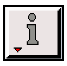
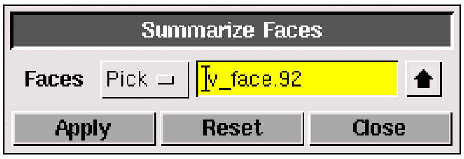


Figure 6-18: Edges used to create face at top of sedan

3. Verify the creation of the faces.

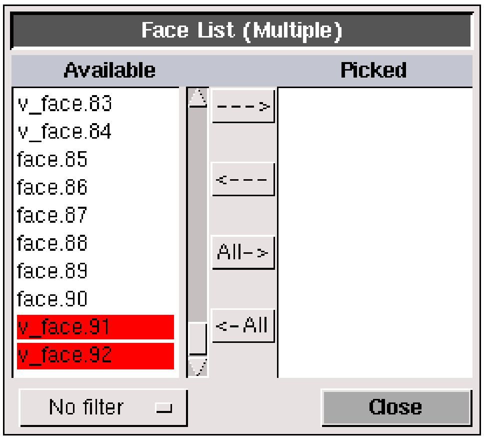
GEOMETRY  → **FACE**  → **SUMMARIZE/QUERY FACES/TOTAL ENTITIES** 

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



a) Left-click the black arrow to the right of the **Faces** list box.

This action opens the **Face List** *form. There are two types of pick-list forms:* **Single** *and* **Multiple**. *In a* **Single** *pick-list form, only one entity can be selected at a time. In a* **Multiple** *pick-list form, you can select multiple entities.*



i. Select the two faces at the bottom of the **Available** list in the **Face List** form.

! *Note that the names of entities in the* **Available** *list may be different in your geometry. In the form shown above, the last two faces in the* **Available** *list are v_face.91 and v_face.92, but you might see faces with different numbers.*

ii. Click the – –> button to pick the two faces.

The two faces will be moved from the **Available** *list to the* **Picked** *list, and they will be highlighted in the graphics window.*

iii. Check that the two faces highlighted in the graphics window are the correct faces that you should have created in the previous steps.

Figure 6-16 and Figure 6-18, above, show the faces that you should have created.

iv. Close the **Face List** form.

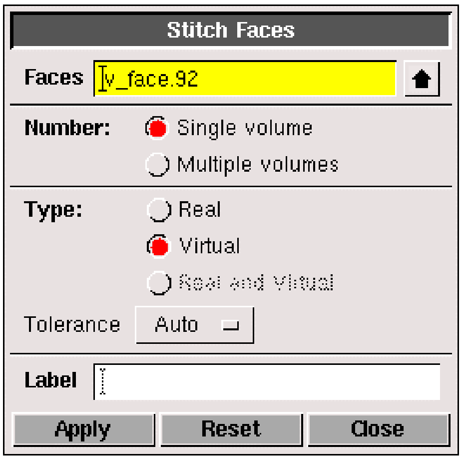
b) Click **Reset** in the **Summarize Faces** form to unselect the two faces in the graphics window.

Step 10: Create a Volume

1. Use the faces to create a volume.

   GEOMETRY → **VOLUME** → **FORM VOLUME**

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



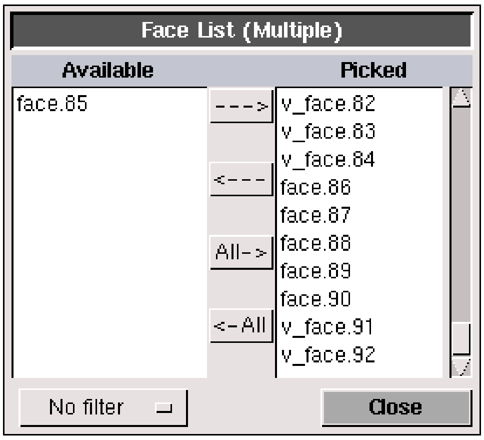
   a) Select the **Number:**Single volume option.

   b) Select the **Type:**Virtual option.

   c) Select the symmetry plane in the graphics window (as shown in Figure 6-17) and remember the label name (for example, *face.85*).

   d) Left-click the black arrow to the right of the **Faces** list box.

      This action opens the **Face List** *form.*



i. Click on the All –> button to move all the faces from the **Available** list to the **Picked** list.

ii. Select the name of the symmetry plane in the **Picked** list.

The symmetry plane face will be highlighted in the graphics window.

iii. Click the <—– button to move the symmetry plane face back into the **Available** list.

iv. Close the **Face List** form.

e) Click **Apply** to create the volume.

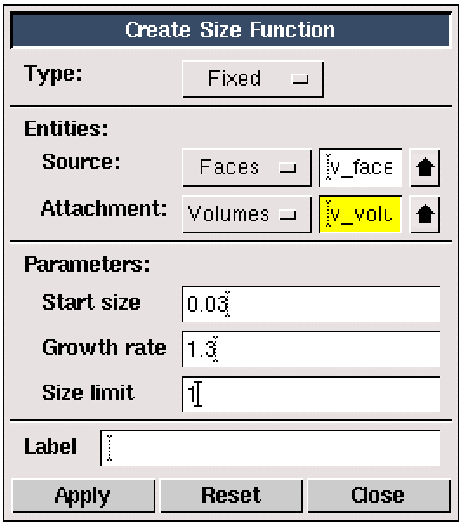
Step 11: Apply Size Functions to Control Mesh Quality

GAMBIT *includes several features that allow you to control the mesh quality, one of which is the application of size functions. For example, size functions can be used to specify the rate at which volume mesh elements change in size in proximity to a specified boundary. In this step, you will apply size functions to the faces that comprise the outer surface of the sedan and to the face you created underneath the sedan on the symmetry plane.*

1. Specify a size function and apply it to four faces of the model.

   TOOLS → **SIZE FUNCTIONS** → **CREATE SIZE FUNCTION**

   This command sequence opens the **Create Size Function** *form.*



   a) Retain the **Type:**Fixed option.

   b) On the **Entities:Source** option button, select the Faces option, and click in the Faces list box to make it active.

   c) Click the **ORIENT MODEL** command button, to orient the model as shown in Figure 6-19.

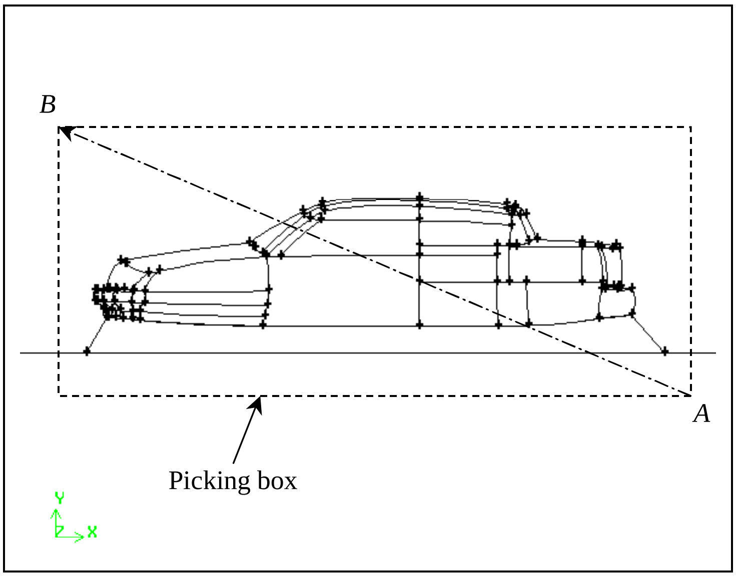


Figure 6-19: Picking box including all faces to which size functions will apply

d) *Shift*-drag the mouse from the lower right (point *A*) to the upper left (point *B*) to create a picking box that encompasses all of the faces on the sedan body as well as the small, symmetry plane face underneath the sedan.

When you Shift-*drag the mouse in an upper diagonal direction to create the picking box,* GAMBIT *picks only those faces that are* completely *enclosed by the box.*

e) On the **Entities:Attachment** option button, retain the Volumes option.

f) In the Volumes list box, select the volume.

g) In the **Start size** text box, enter the value `0.03`.

h) In the **Growth rate** text box, enter the value, `1.3`.

i) In the **Size limit** text box, enter the value, `1`.

j) Click **Apply** to create the size function.

When applying the size function, GAMBIT *displays a message in the* **Transcript** *window indicating that the use of virtual entities as source entities in the size-function definition can cause problems when evaluated during background-grid generation. The message represents a warning only and can be ignored in this case.*

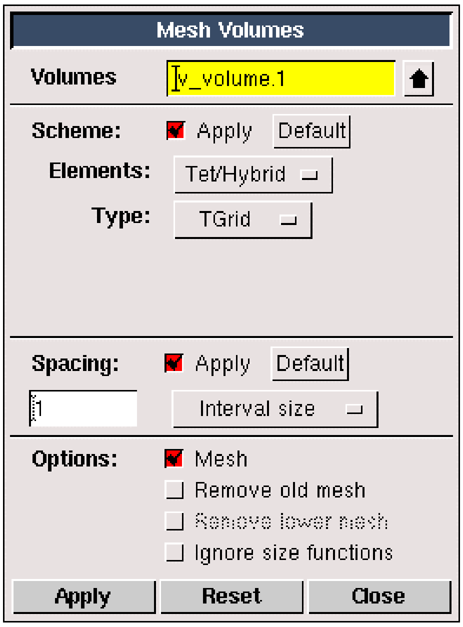
GAMBIT *allows you to view the size function by means of the* **View Size Function** *command on the* **Size Function** *toolpad.*

Step 12: Mesh the Volume

1. Mesh the volume with a coarser mesh than the mesh on the car faces.

   MESH → **VOLUME** → **MESH VOLUMES**

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



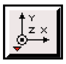
   a) Select the volume in the graphics window.

   b) Select Tet/Hybrid from the **Elements** option menu under **Scheme** in the **Mesh Volumes** form, and select TGrid from the **Type** option menu.

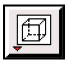
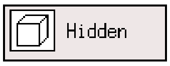
      See the GAMBIT *Modeling Guide for more information on meshing schemes.*

   c) Retain the default **Interval size** of `1` under **Spacing** and click the **Apply** button at the bottom of the form.

      A portion of the volume mesh (looking at the sedan from the symmetry plane side) is shown in Figure 6-20. To achieve this view of the model:

i) *Right-click the* **ORIENT MODEL** *command button*  *on the* **Global Control** *toolpad and select the* –Z *view* 

..

ii) *Turn on hidden-line removal by right-clicking the* **RENDER MODEL** *command button*  *in the* **Global Control** *toolpad and selecting* 

from the resulting list.

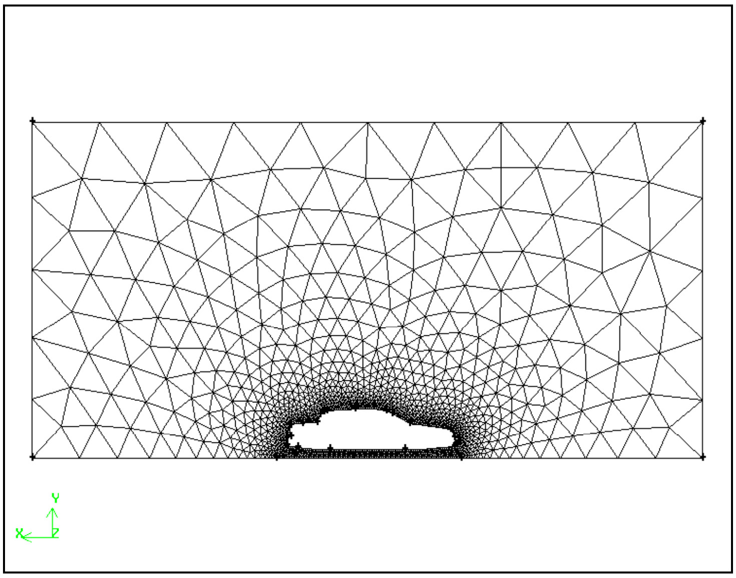
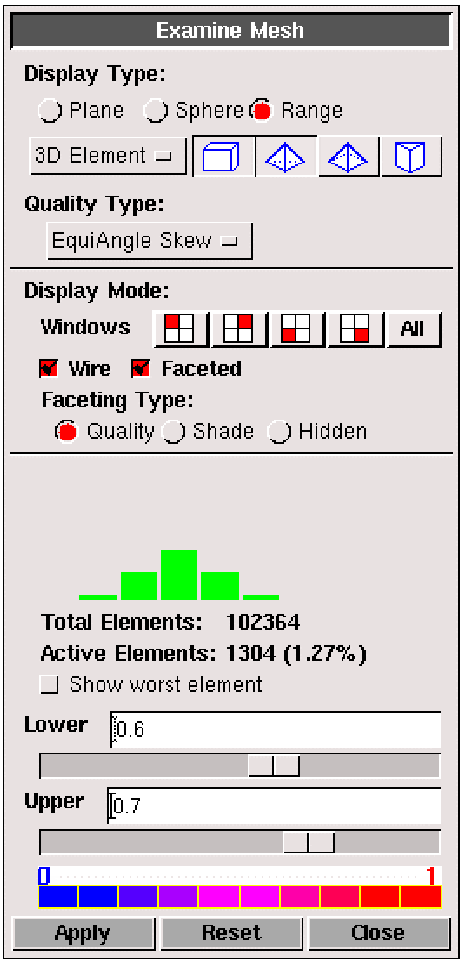


Figure 6-20: A portion of the volume mesh

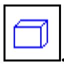
Step 13: Examine the Volume Mesh

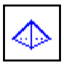
1. Select the **EXAMINE MESH**  command button at the bottom right of the **Global Control** toolpad.

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



   a) Select Range under **Display Type** at the top of the form.

The **3D Element** *type selected by default at the top of the form is a brick* *. You will not see any mesh elements in the graphics window when you first open the* **Examine Mesh** *form, because there are no hexahedral elements in the mesh.*

b) Left-click on the tetrahedron icon  next to **3D Element** near the top of the form.

The mesh elements will now be visible in the graphics window.

c) Select or retain EquiAngle Skew from the **Quality Type** option menu.

This is the default skewness measure for tetrahedra in TGrid.

d) Click with the left mouse button on the histogram bars that appear at the bottom of the **Examine Mesh** form to highlight elements in a particular quality range.

Figure 6-21 shows the view in the graphics window if you click on the fourth bar from the right on the histogram (representing cells with a skewness value between 0.6 and 0.7). These low values for the maximum skewness indicate that the mesh is acceptable.

The histogram consists of a bar chart representing the statistical distribution of mesh elements with respect to the specified **Quality Type**. *Each vertical bar on the histogram corresponds to a unique set of upper and lower quality limits.*

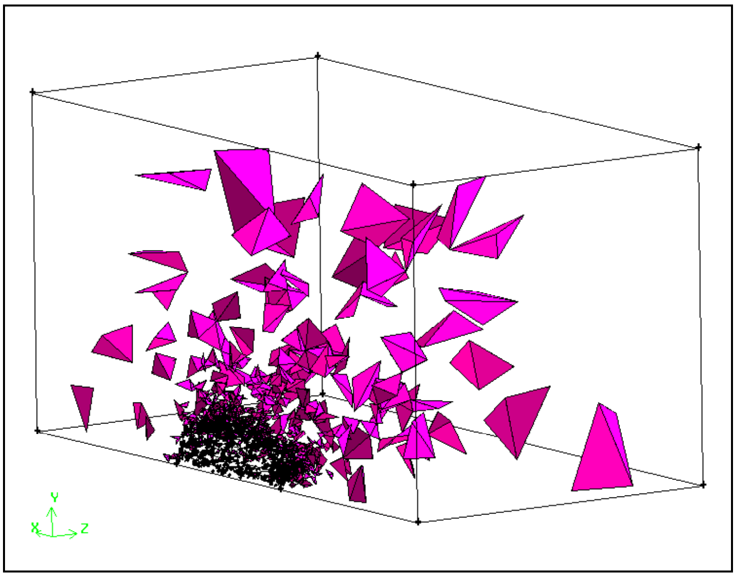


Figure 6-21: Elements EquiAngle Skew = 0.6 to 0.7

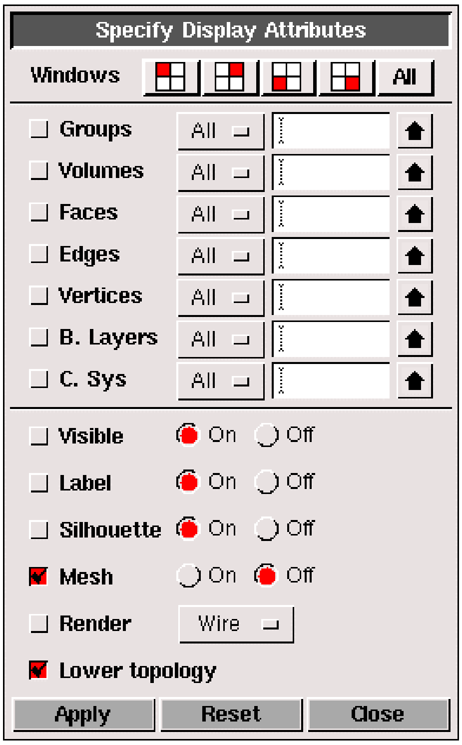
e) Close the **Examine Mesh** form by clicking the **Close** button at the bottom of the form.

Step 14: Set Boundary Types

1. Remove the mesh from the display before you set the boundary types.

   This makes it easier to see the edges and faces of the geometry. The mesh is not deleted, just removed from the graphics window.

   a) Click the **SPECIFY DISPLAY ATTRIBUTES** command button  at the bottom of the **Global Control** toolpad to open the **Specify Display Attributes** form.



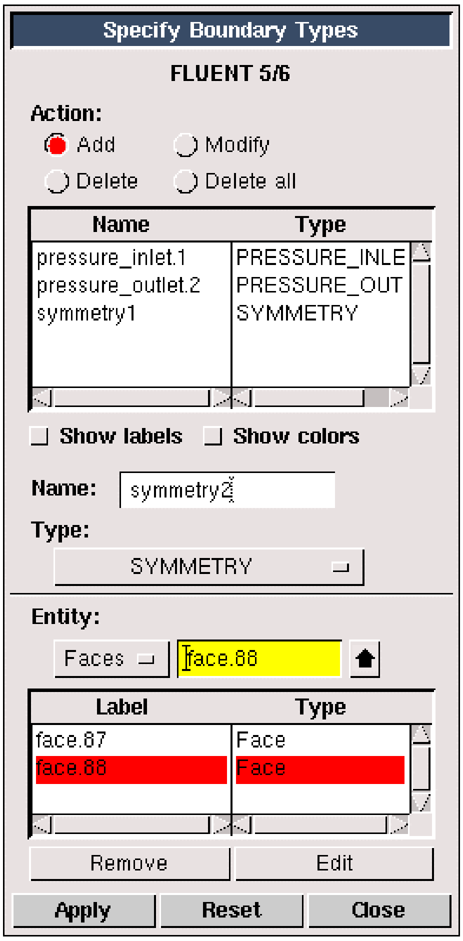
   b) Select the **Mesh:**Off option near the bottom of the form.

   c) Click **Apply** to turn off the mesh display, then **Close** the form.

2. Set boundary types for the sedan.

   ZONES  → **SPECIFY BOUNDARY TYPES** 

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



a) Define the pressure inlet boundary.

   i. Select PRESSURE_INLET in the **Type** option menu.

   ii. Check that Faces is selected as the **Entity**.

   iii. Select (pick) the face on the brick in front of the car in the graphics window (face *A* in Figure 6-22) and click **Apply** to accept the selection.

      This face will be set as a pressure inlet.

GAMBIT *will give the boundary a default name based on what you select in the* **Type** *and* **Entity** *lists (pressure_inlet.1 in this example). You can also specify a name for a boundary by entering a name in the* **Name** *text entry box.*

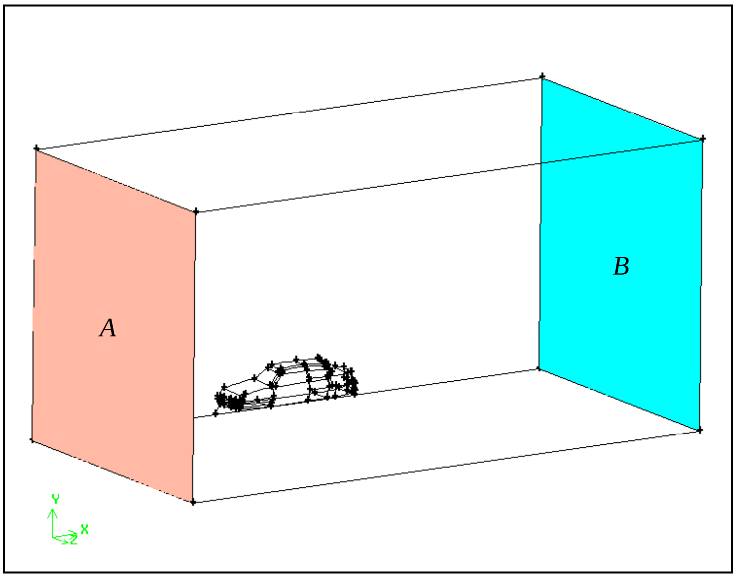


Figure 6-22: Pressure inlet (*A*) and pressure outlet (*B*) for the sedan geometry

b) Define the pressure outlet boundary.

   i. Change the **Type** to PRESSURE_OUTLET by selecting it from the option menu below **Type**.

   ii. Select (pick) the face on the brick behind the car in the graphics window (face *B* in Figure 6-22) and click **Apply** to accept the selection.

c) Define symmetry boundary types for the two faces on the symmetry plane of the brick.

   i. Enter "`symmetry1`" in the **Name** text entry box.

   ii. Select SYMMETRY from the **Type** option menu.

iii. Select (pick) the two faces you created on the symmetry plane of the brick (faces *C* and *D* in Figure 6-23) and click **Apply** to accept the selection.

GAMBIT *will merge the two faces into a single symmetry zone.*

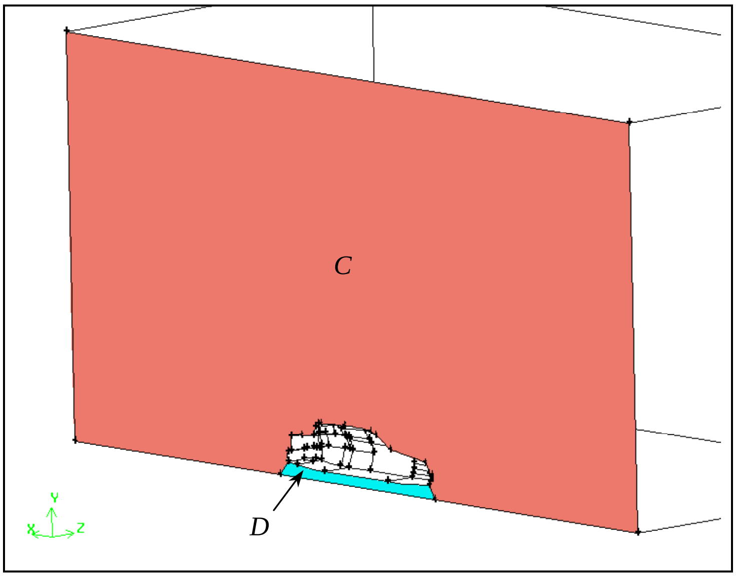


Figure 6-23: Two faces created on the symmetry plane of the brick

d) Define symmetry boundary types for the top face of the brick and the side face opposite the symmetry plane.

i. Enter "`symmetry2`" in the **Name** text entry box.

ii. Check that SYMMETRY is selected in the **Type** option menu.

iii. Select (pick) the faces on the brick that are above and to the side of the sedan (faces *E* and *F* in Figure 6-24) and accept the selection.

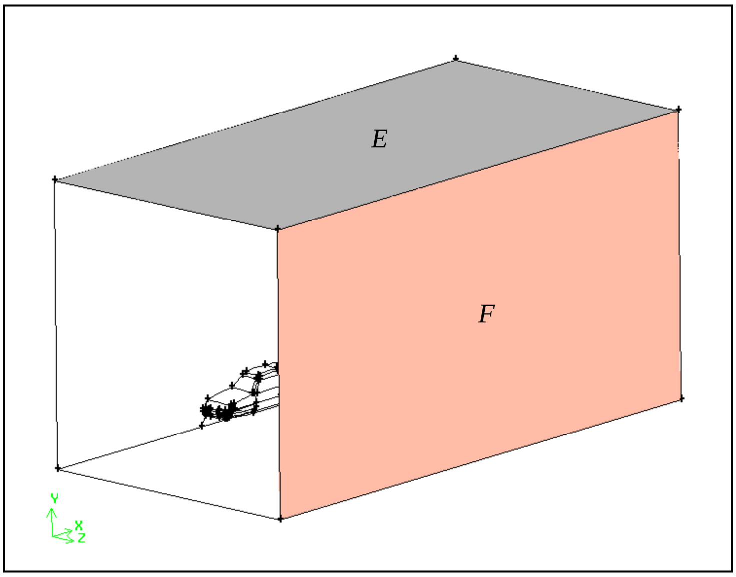


Figure 6-24: Two symmetry boundaries for the sedan geometry

The pressure inlet, pressure outlet, and symmetry boundaries for the sedan geometry are shown in Figure 6-25. (NOTE: To display the boundary types in the graphics window, select the **Show labels** *options on the* **Specify Boundary Types** *form.)*

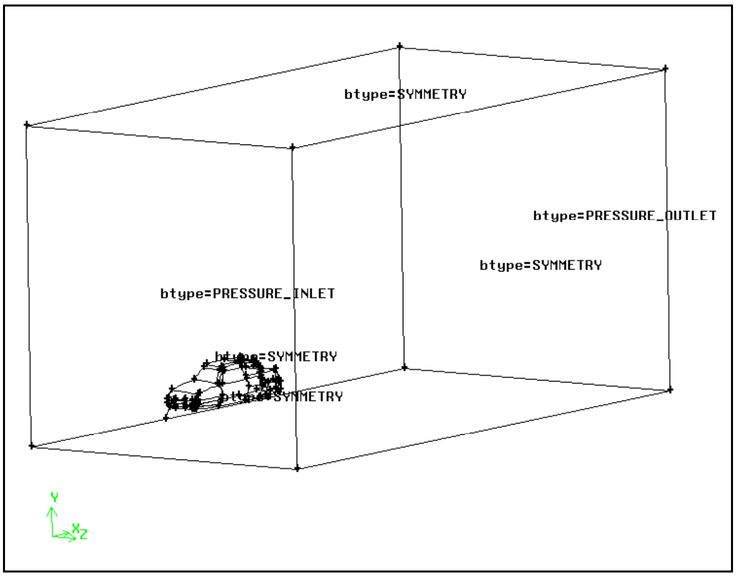


Figure 6-25: Boundary types for the sedan geometry

Note that you could also specify the remaining outer edges of the sedan geometry as wall boundaries. This is not necessary, however, because when GAMBIT *saves a mesh, any faces (in 3D) on which you have not specified a boundary type will be written out as wall boundaries by default.*

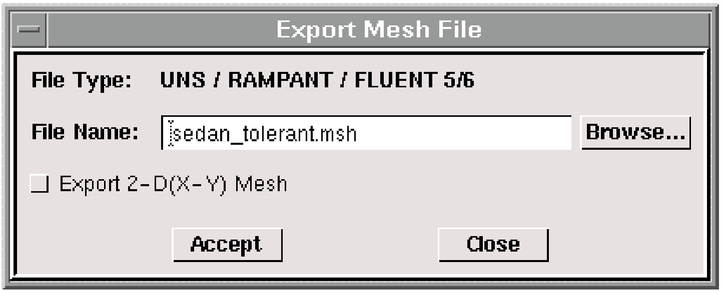
In addition, when GAMBIT *writes a mesh, any volumes (in 3D) on which you have not specified a continuum type will be written as fluid by default. This means that you do not need to specify a continuum type in the* **Specify Continuum Types** *form for this tutorial.*

Step 15: Export the Mesh and Save the Session

1. Export a mesh file for the sedan.

   File → **Export** → **Mesh...**

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



   a) Enter the **File Name** for the file to be exported (`sedan_tolerant.msh`).

   b) Click **Accept** in the **Export Mesh File** form.

      The file will be written to your working directory.

2. Save the GAMBIT session and exit GAMBIT.

   File → **Exit**

   GAMBIT *will ask you whether you wish to save the current session before you exit.*

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

6.5 Summary

This tutorial illustrated how to import geometry from an external CAD package as an IGES file using GAMBIT tolerant modeling to facilitate the creation of a “clean,” meshable model. In addition, several faces were merged, a box was constructed around the carbody geometry, and an unstructured tetrahedral volume mesh was generated.

7. MODELING FLOW IN A TANK

In this tutorial you will utilize the techniques illustrated in the previous tutorials to create a complex pipe junction that represents a real-world example of flow in a process tank.

In this tutorial you will learn how to:

- Create cylinders and bricks by defining their dimensions
- Translate and rotate volumes
- Perform Boolean operations on volumes (unite and subtract)
- Split a volume using another volume
- Align two volumes using a vertex pair
- Specify the distribution of nodes on an edge
- Add boundary layers to your geometry
- Generate an unstructured hexahedral mesh
- Examine the quality of the mesh
- Prepare the mesh to be read into FIDAP

7.1 Prerequisites

This tutorial assumes that you have worked through Tutorials 1, 2, 3, and 4.

7.2 Problem Description

The problem to be considered is shown schematically in Figure 7-1. Due to symmetry, only half of the actual model is shown.

The geometry consists of a large cylindrical tank with an inlet/outlet annular section. This section is connected to the tank at half the length of the tank and at an offset from the center of the tank. In the annular section, the inner pipe is the inlet pipe. There is a small T-junction on the upper end of outer pipe. This is the outlet.

The overall goal is to create a high quality hexahedral mesh including boundary layers and edge meshing to sufficiently capture gradient in solution variables, such as velocity and temperature. The solver selected for this tutorial is FIDAP

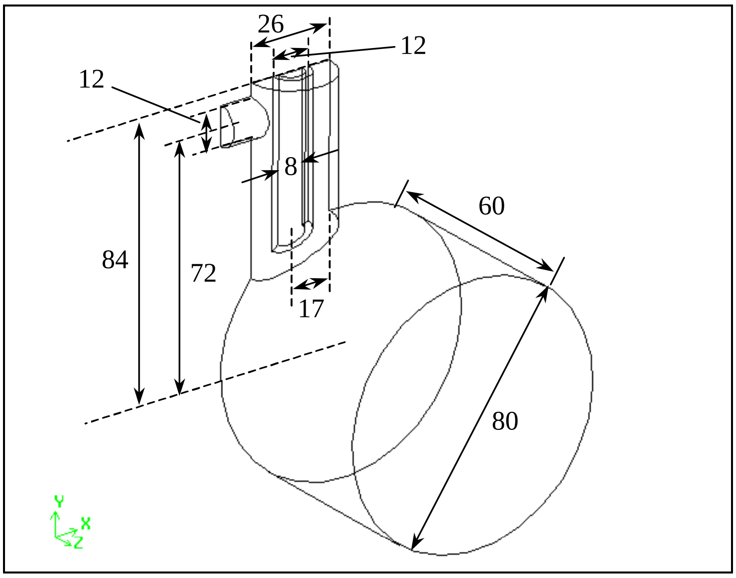


Figure 7-1: Problem specification

7.3 Strategy

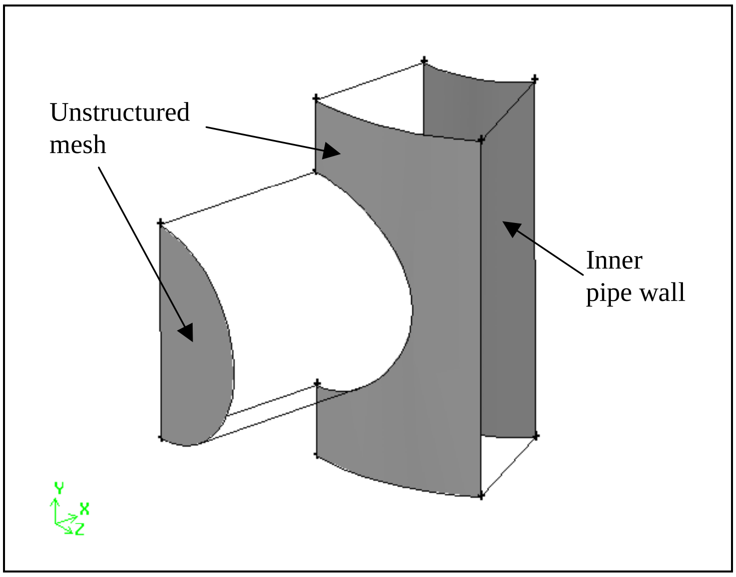
In this tutorial we will combine several of the previously shown tools and strategies and apply them on a real industrial problem. The first thing to find out is if the boundary condition and the physics will allow us to model only half of the geometry. This is a very important step since it immediately reduces the effort of preprocessing and running time. After confirming the symmetry condition, we start building the geometry using primitives and Boolean operations. Although we normally recommend to create the model in the following order:

1. Geometry creation
2. Decomposition
3. Mesh generation

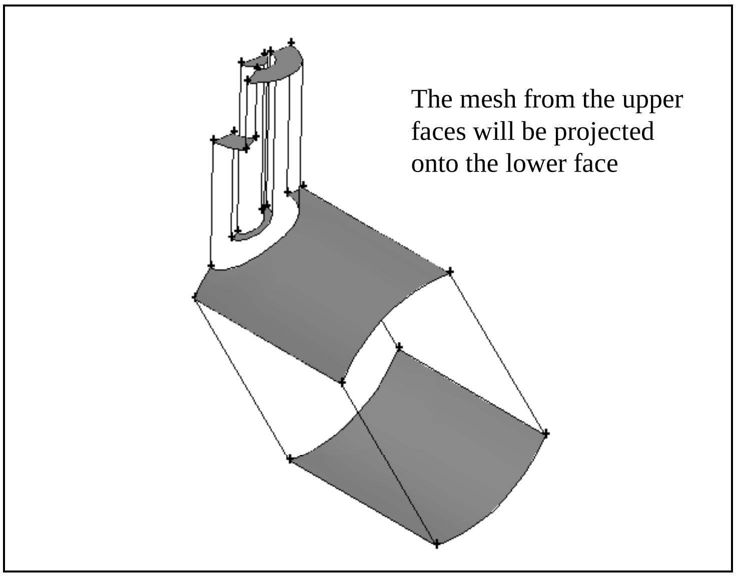
We will illustrate, in this journal, that the order of geometry creation and decomposition is not strict. Mesh generation, though, should in all cases be left to last.

The overall geometry creation is fairly straightforward and based on cylinder primitives, Boolean unites, and subtracts. The model cannot be meshed as is, using hexahedral meshing, since several faces in this model are non-trivial and their normals are facing all three major directions. Essentially there are two pipe-pipe intersections, which needs to be decomposed

The first section is the pipe/annulus intersection at the outlet. In this situation, the recommended strategy is to use a block to split of the outlet pipe. The split has to be made with an angle, such that the unstructured mesh from the pipe will be “projected” to the wall of the inner pipe. (See the following figure.)



The second section is the main intersection of the inlet/outlet pipes with the tank. Again, we are using a block to split of the bigger tank section into a center section. We are tilting the cutting block to optimize mesh quality. This will allow the mesh on all non-trivial source faces to be "projected" into the bottom of the tank as illustrated in the picture below



Edge meshing and boundary layers are applied at several areas to ensure appropriate grading in key areas of the model. The boundary layers are particularly important in areas where the face is being paved—that is, on most source faces, while edge meshing is used where the mesh is being mapped. In some cases edge meshing and boundary layer are combined for full control over the mesh density

Several techniques are used in the face meshing part like; enforce Submap without meshing and multiple source face meshing, where side faces between source faces are also meshed

Finally, the Cooper tool is used to mesh all volumes in this model.

7.4 Procedure

Start GAMBIT using the session identifier "`Tank`".

Step 1: Select a Solver

1. Choose the **FIDAP** solver.

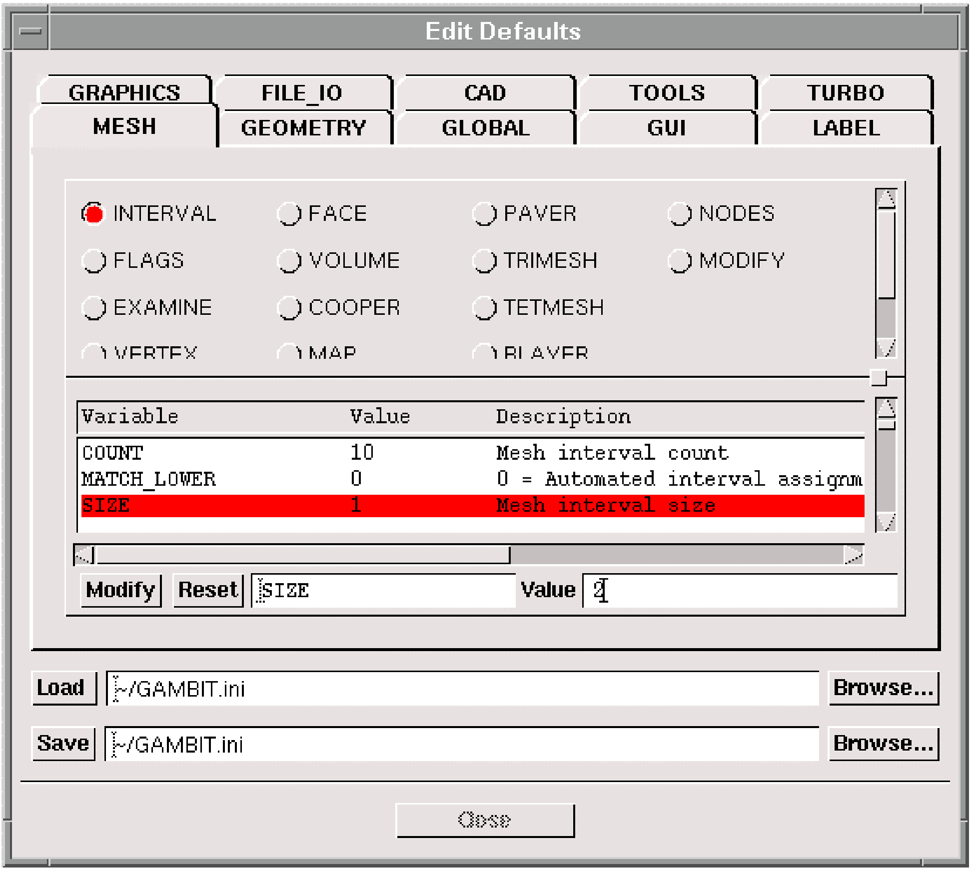
 Solver $\longrightarrow$ **FIDAP**

 The choice of a solver dictates the options available in various forms (for example, the boundary types available in the **Specify Boundary Types** *form). The solver currently selected is indicated at the top of the* GAMBIT *GUI.*

Step 2: Set the Default Interval Size for Meshing

In this tutorial, you will change the default interval size used for meshing. The mesh spacing is by default based on the interval size function, which you will modify in the **Edit Defaults** *form. The value you enter should be the estimated average size of an element in the model. This value will appear as the default* Interval size *on all meshing forms. You will be able to change it on the meshing forms if required.*

Edit $\longrightarrow$ **Defaults...**



1. Select the **MESH** tab at the top of the form.

2. Select the INTERVAL radio button near the top of the form.

3. Select `SIZE` in the `Variable` list.

   `SIZE` *will appear in the space at the bottom of the list and its default value will appear in the* **Value** *text entry box.*

4. Enter a value of 2 in the **Value** text entry box.

5. Click the **Modify** button to the left of `SIZE`.

   The **Value** *of the variable* `SIZE` *will be updated in the list.*

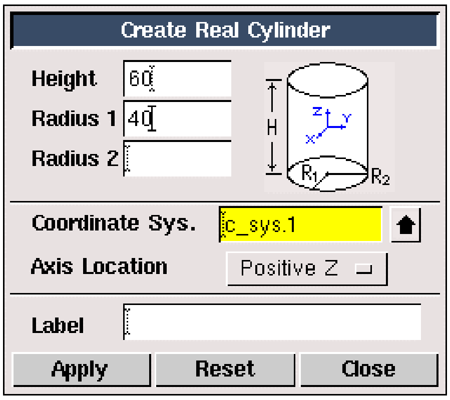
6. Close the **Edit Defaults** form.

Step 3: Create Cylinders

1. Create the first pipe for the pipe junction.

GEOMETRY → **VOLUME** → **CREATE VOLUME** R Cylinder

This command sequence opens the **Create Real Cylinder** *form.*



a) Enter a **Height** of `60`.

b) Enter `40` for **Radius 1**.

The text entry box for **Radius 2** *can be left blank;* GAMBIT *will set this value by default to be the same value as* **Radius 1**.

c) Retain Positive Z (the default) in the list to the right of **Axis Location**.

d) Click **Apply**.

e) Click the **FIT TO WINDOW** command button at the top left of the **Global Control** toolpad to see the cylinder created.

You can rotate the view by holding down the left mouse button and moving the mouse. The cylinder is shown in Figure 7-2.

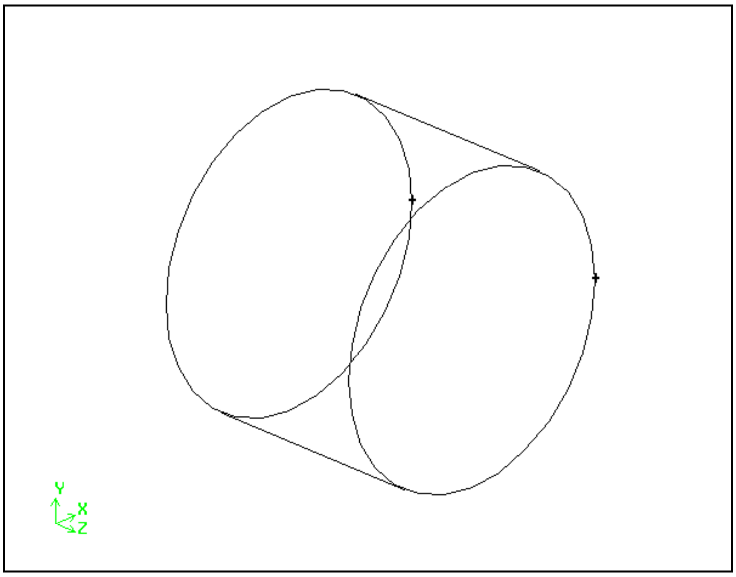


Figure 7-2: First cylinder for the pipe junction

2. Create a second cylinder, with a **Height** of `64`, a **Radius 1** of `13`, in the Centered Y direction.

   The two cylinders are shown in Figure 7-3.

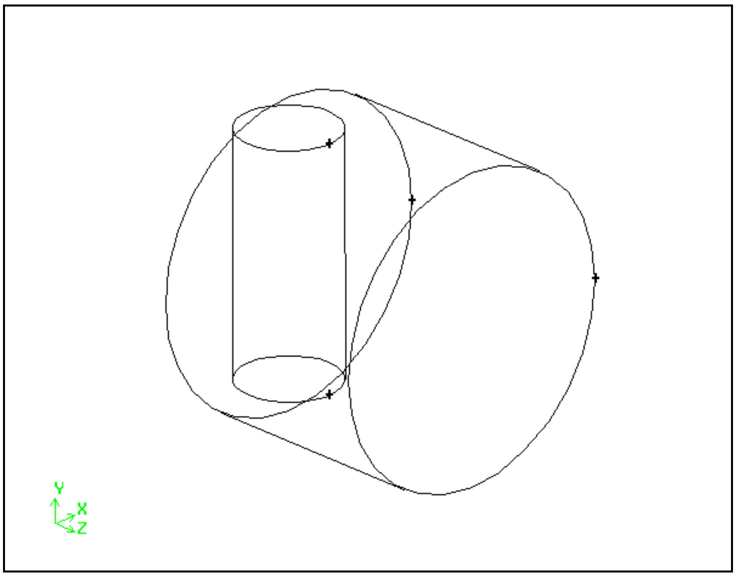


Figure 7-3: Second cylinder for the pipe junction

3. Create a third cylinder, with a **Height** of `64`, a **Radius 1** of `6`, in the Centered Y direction.

4. Create a fourth cylinder, with a **Height** of `64`, a **Radius 1** of `4`, in the Centered Y direction.

   The four cylinders created so far are shown in Figure 7-4.

5. Create a fifth cylinder, with a **Height** of `16`, a **Radius 1** of `6`, in the Centered X direction.

   The five cylinders are shown in Figure 7-5.

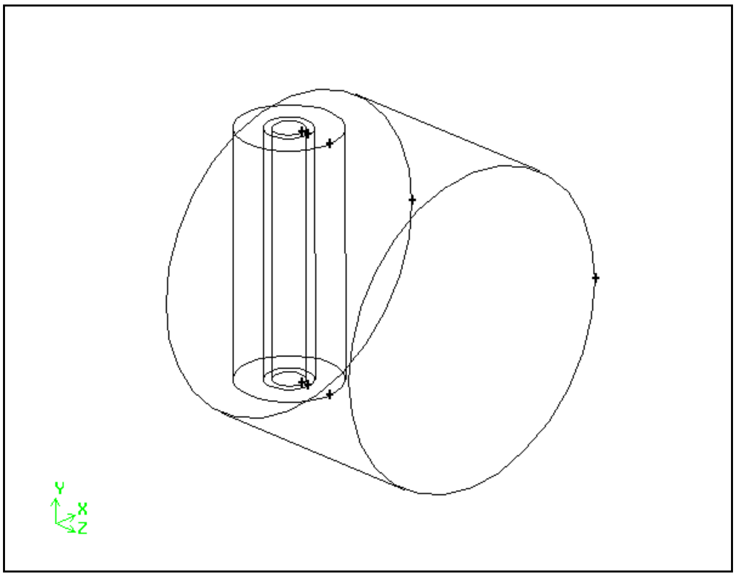


Figure 7-4: Four cylinders for the pipe junction

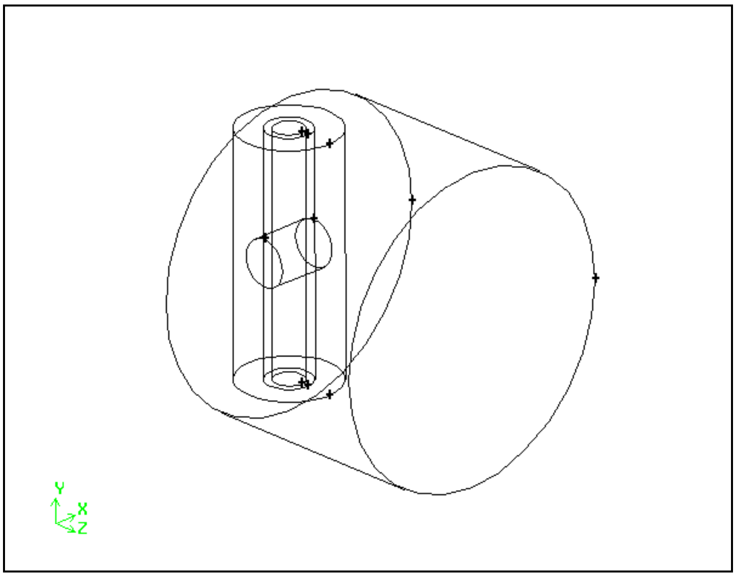


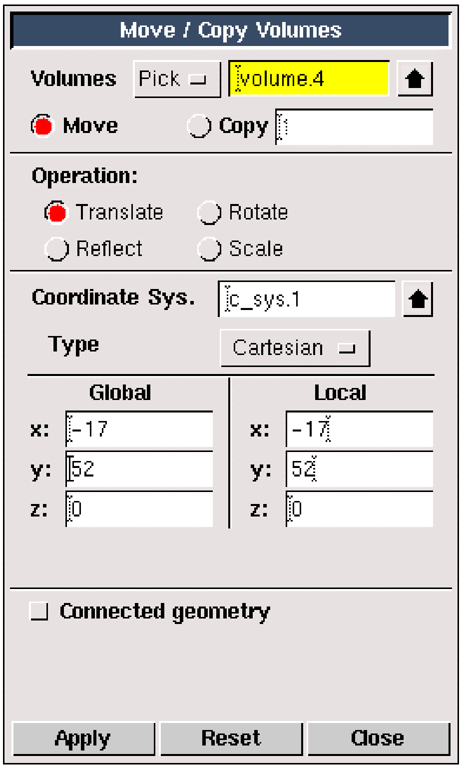
Figure 7-5: Five cylinders for the pipe junction

Step 4: Complete the Geometry Creation

1. Move three of the cylinders to create the geometry of the complex pipe junction.

   GEOMETRY → **VOLUME** → **MOVE/COPY/ALIGN VOLUMES**

   This command sequence opens the **Move / Copy Volumes** *form.*



   a) Select the second, third, and fourth cylinders created, by either selecting them in the graphics window or using the **Volume** query list.

   b) Retain **Move** (the default) under **Volume** in the **Move / Copy Volumes** form.

   c) Retain Translate (the default) under **Operation**.

   d) Enter a **Global** translation vector of (`-17`, `52`, `0`) and click **Apply**.

Note that GAMBIT *automatically fills in the values under* **Local** *as you enter values under* **Global**.

The three cylinders will be moved as shown in Figure 7-6.

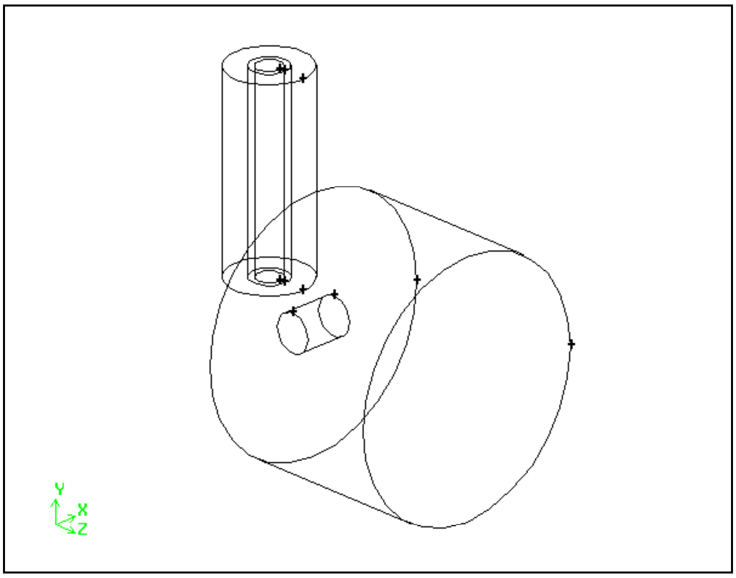


Figure 7-6: Three cylinders moved into position for the complex pipe junction geometry

2. Move the fifth cylinder using a **Global** translation vector of (`-32`, `74`, `0`).

3. Subtract *volume.3* from *volume.2*.

   The order of selecting the volumes is important. For example, Figure 7-7 shows the difference between subtracting volume B from volume A, and vice versa.

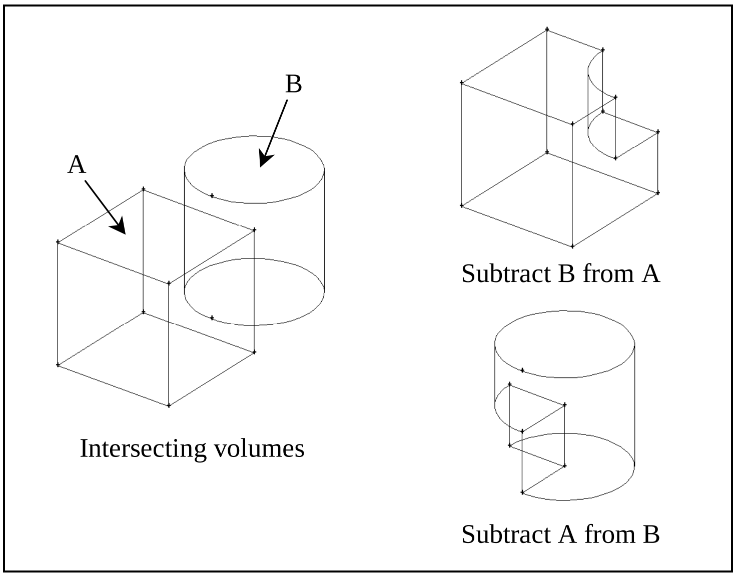
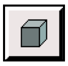
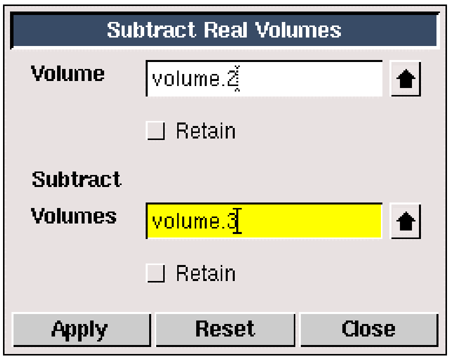


Figure 7-7: Subtracting volumes

GEOMETRY  → **VOLUME**  → **BOOLEAN OPERATION** 

R 

This command sequence opens the **Subtract Real Volumes** *form.*



a) Select *volume.2* in the graphics window and accept the selection of the volume.

b) Select *volume.3* and accept the selection of the volume.

The completed geometry is shown in Figure 7-8.

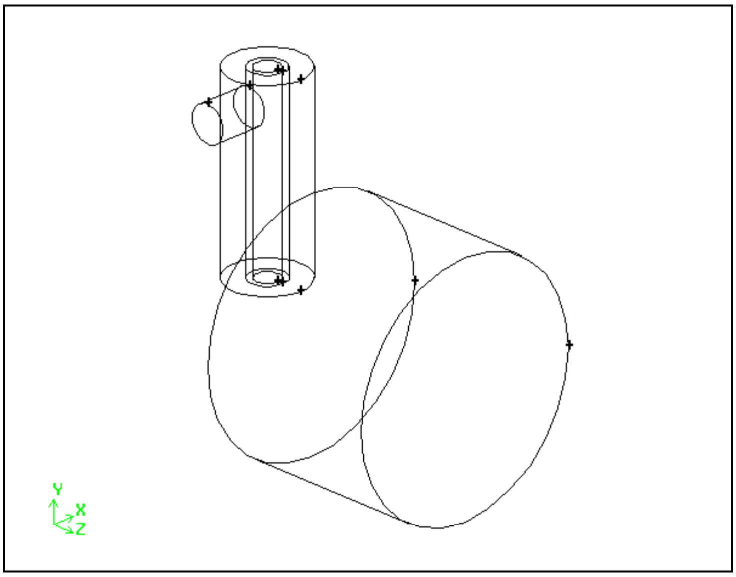


Figure 7-8: Completed geometry

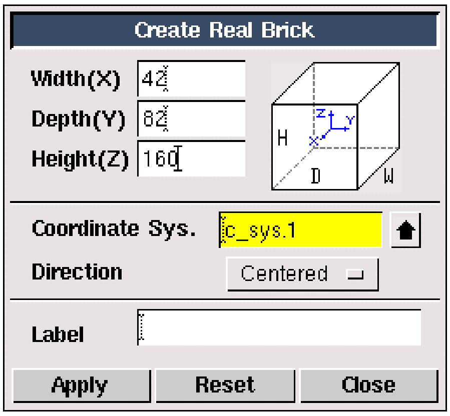
Step 5: Decompose the Geometry

In this tutorial, we are decomposing the tank before uniting it with all of the pipes. If we reversed the order, the pipes would have been split through, which is undesirable. We are also rotating the brick to improve the symmetry of the tank cut. This will increase the edge-angle of one of the faces, which ultimately leads to better mesh quality

1. Create a brick to split the large cylinder.

   GEOMETRY → **VOLUME** → **CREATE VOLUME** R Brick

   This command sequence opens the **Create Real Brick** *form.*



   a) Enter a value of `42` for the **Width** of the brick.

   b) Enter a **Depth** of `82` and a **Height** of `160`.

   c) Retain Centered on the **Direction** option menu and click **Apply**.

   The brick and the pipe junction are shown in Figure 7-9.

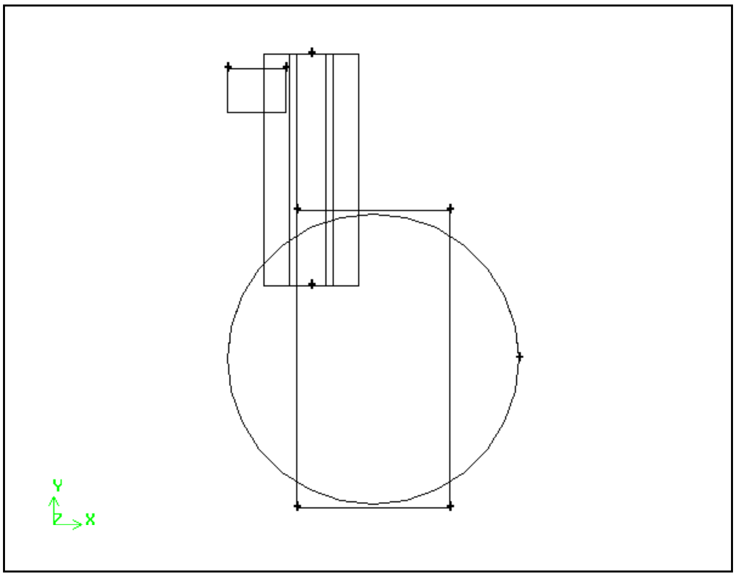
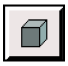
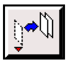
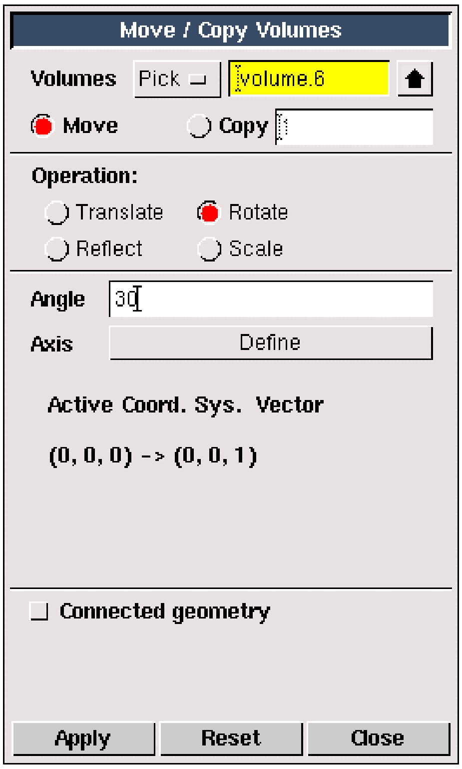


Figure 7-9: Brick and complex pipe junction

2. Rotate the brick relative to the geometry.

GEOMETRY  → **VOLUME**  → **MOVE/COPY/ALIGN VOLUMES** 

This command sequence opens the **Move / Copy Volumes** *form.*



a) Select the brick in the graphics window.

b) Retain **Move** (the default) under **Volume** at the top of the **Move / Copy Volumes** form.

c) Under **Operation**, select Rotate.

d) Enter an **Angle** of `30` for the angle of rotation.

You will use the default **Active Coord. Sys. Vector***. The brick will be rotated around the* z *axis.*

e) Click **Apply**.

The brick will be rotated as shown in Figure 7-10.

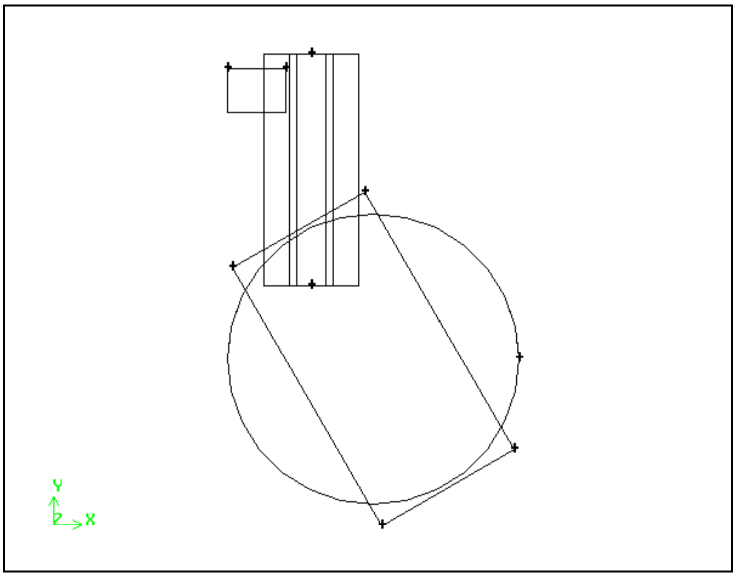


Figure 7-10: Rotated brick

3. Split the largest cylinder using the brick.

If you split one volume with another volume, the following volumes will result:

- *Volumes corresponding to the common region(s) from intersection.*
- *Volumes corresponding to the region(s) defined by subtracting the second volume from the first.*

In other words, splitting a volume results in a combination of the intersection and subtraction Boolean operations. The order of selecting the volumes is important. For example, Figure 7-11 shows the difference between splitting volume A using volume B, and vice versa.

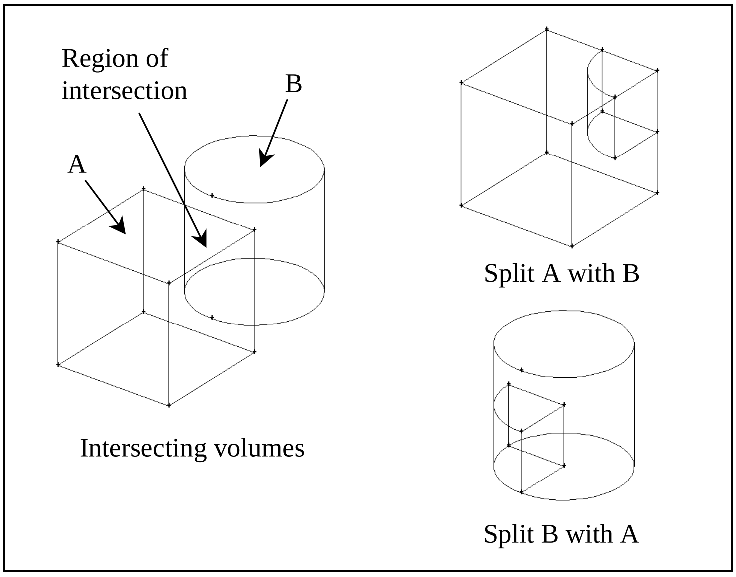
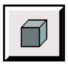
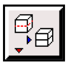
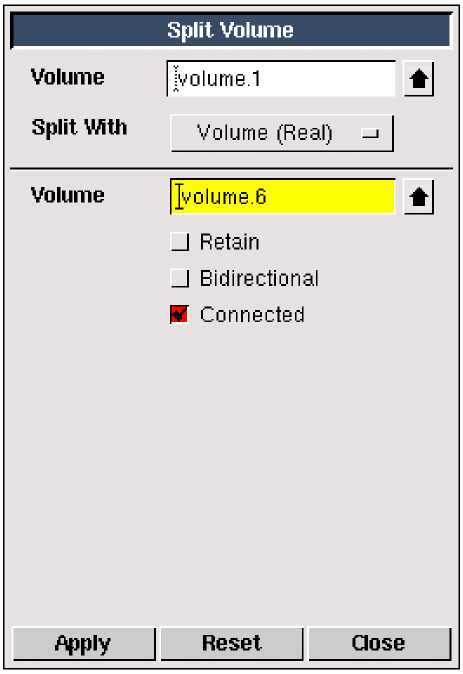


Figure 7-11: Splitting volumes

GEOMETRY  → **VOLUME**  → **SPLIT/MERGE VOLUMES** 

This command sequence opens the **Split Volume** *form.*



a) Select the first cylinder you created in the graphics window (the largest cylinder) and accept the selection of the cylinder.

b) Select Volume (Real) as the **Split With** option.

c) Left-click in the **Volume** list box located below the **Split With** section to make the **Volume** list box active.

d) Select the brick in the graphics window.

e) Unselect the Bidirectional option.

f) Click **Apply**.

The cylinder will be split as shown in Figure 7-12.

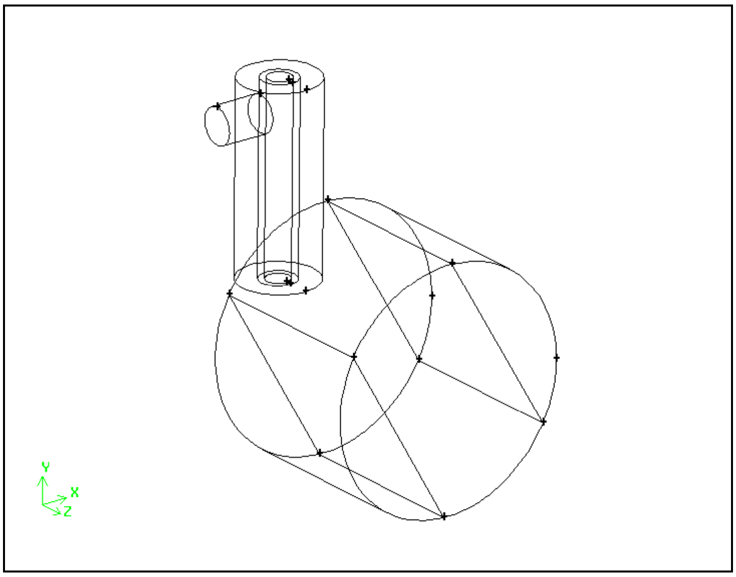


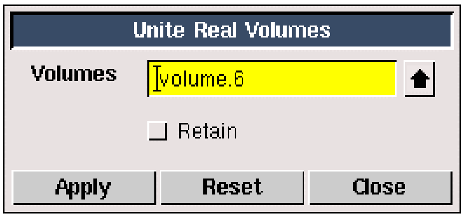
Figure 7-12: Splitting the large cylinder

Step 6: Unite Some Parts of the Geometry

1. Unite two of the volumes into one volume.

GEOMETRY → **VOLUME** → **BOOLEAN OPERATIONS** R Unite

This command sequence opens the **Unite Real Volumes** *form.*



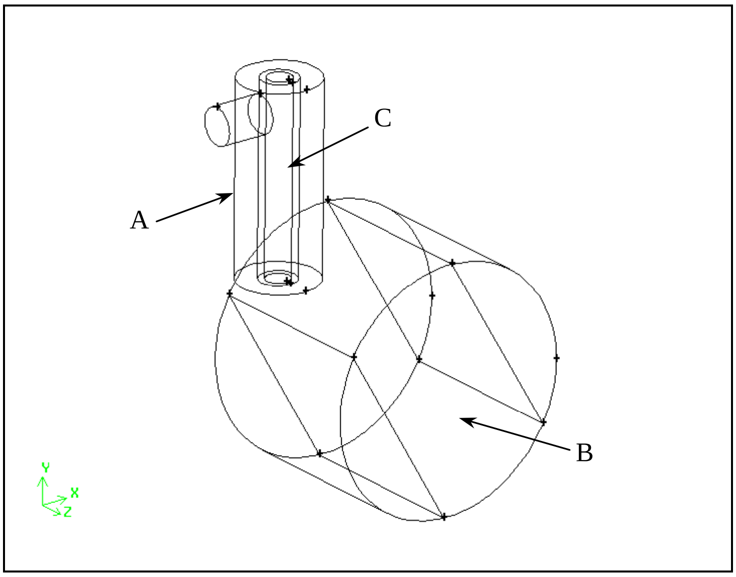


Figure 7-13: Volumes to be united

a) Select the cylinder that you created by subtracting one cylinder from another in Step 2 (the volume marked A in Figure 7-13).

b) Select the middle section of the largest pipe which was created by splitting the pipe with the brick (the volume marked B in Figure 7-13).

c) Click **Apply**.

The two volumes will be united as shown in Figure 7-14. The order in which you select the two volumes is not important when you are uniting them.

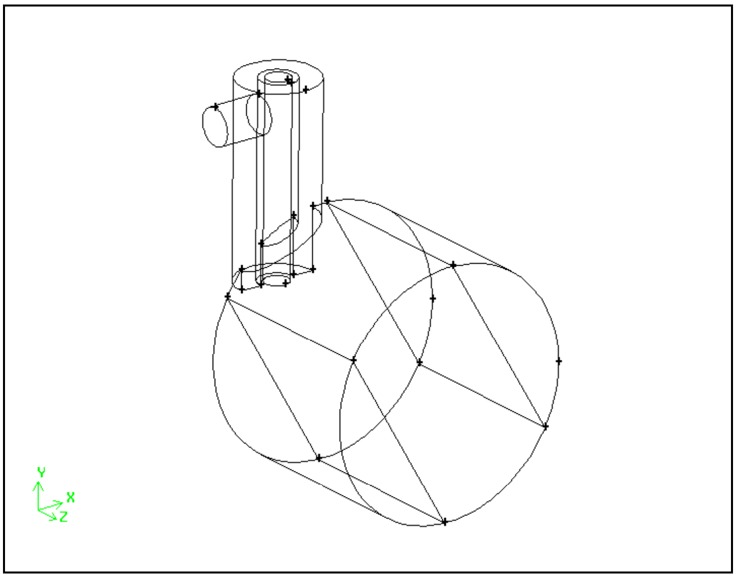


Figure 7-14: Two volumes united into one

2. Unite two more volumes.

a) Select the volume you created in the previous step.

b) Select the middle cylinder of the three concentric cylinders (the volume marked C in Figure 7-13).

c) Click **Apply**.

The united volumes are shown in Figure 7-15.

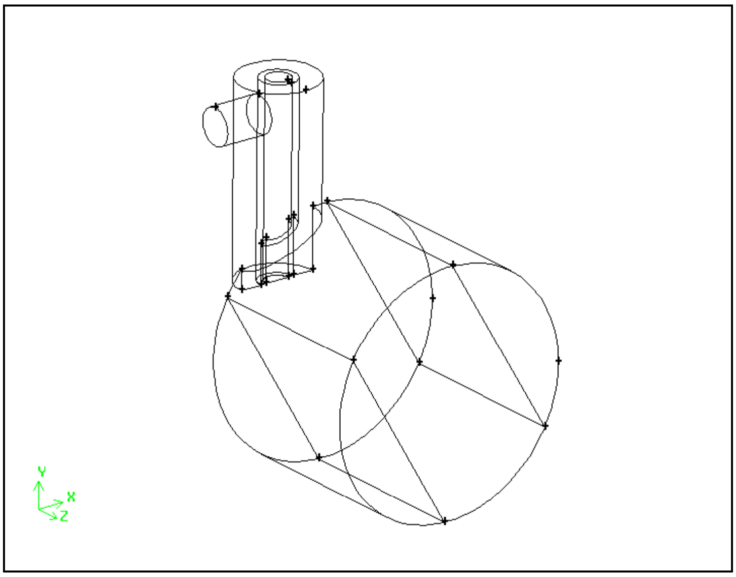


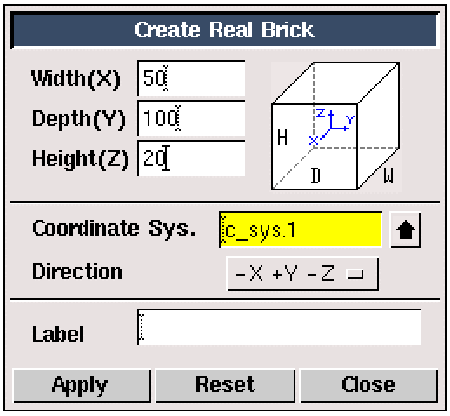
Figure 7-15: United volumes

Step 7: Subtract the Remaining Parts of the Symmetry Plane

1. Create a brick, which will be used to remove some parts of the geometry to create a symmetry plane in the geometry.

   GEOMETRY → **VOLUME** → **CREATE VOLUME**

   This command sequence opens the **Create Real Brick** *form.*



   a) Enter a **Width** of `50`, a **Depth** of `100`, and a **Height** of `20`.

   b) Select -X +Y -Z from the **Direction** option menu.

   c) Click **Apply** to create the brick.

      The brick and the pipe junction are shown in Figure 7-16.

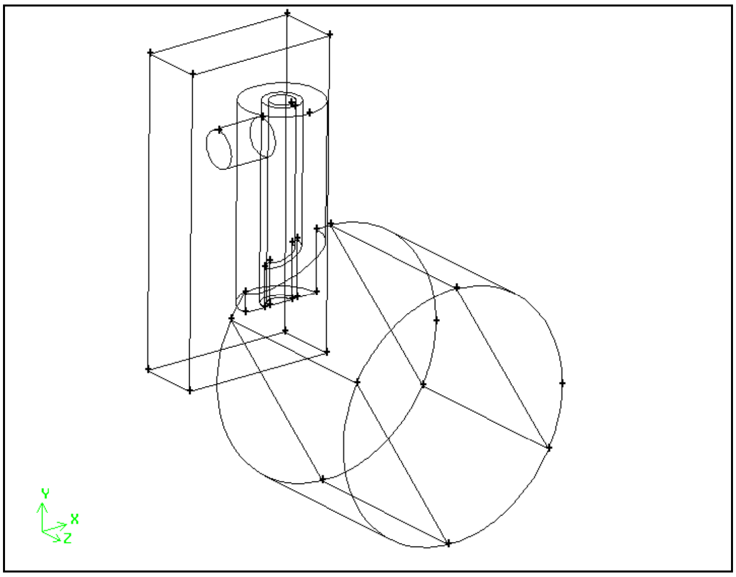
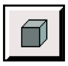
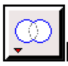
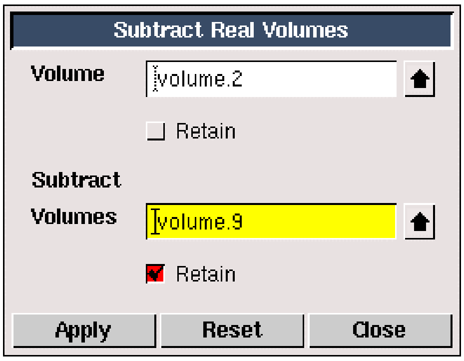


Figure 7-16: Brick and pipe junction

2. Subtract the brick from the complex pipe volume.

GEOMETRY  → **VOLUME**  → **BOOLEAN OPERATIONS**  R

This command sequence opens the **Subtract Real Volumes** *form.*



a) Select the volume that contains most of the pipe sections in the geometry (the volume marked A in Figure 7-17). Accept the selection of the volume.

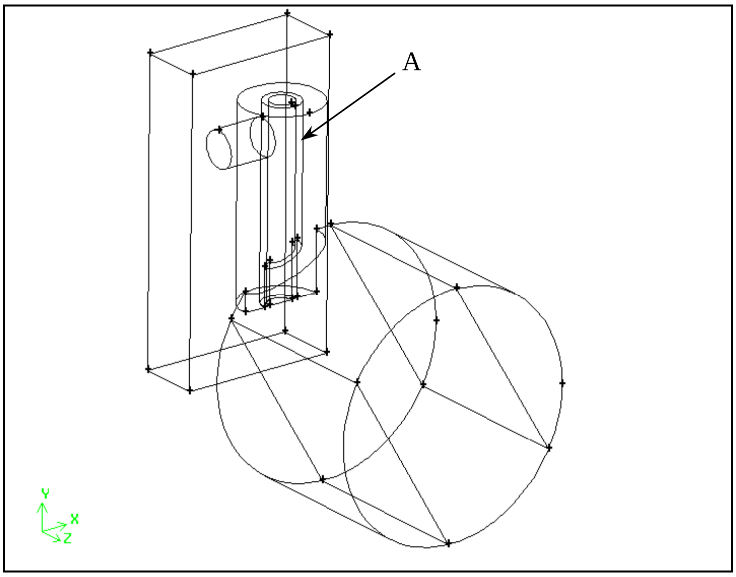


Figure 7-17: Volume to use in subtraction

b) Select the brick in the graphics window.

c) In the **Subtract Real Volumes** form, select the Retain check box below **Subtract Volumes**.

This option instructs GAMBIT *to subtract the brick from the pipe geometry, but retain the brick to be used again in the next step.*

d) Click **Apply**.

The brick will be subtracted from the pipe geometry as shown in Figure 7-18. Notice that the brick is still displayed in this figure, this is because the Retain *check box is selected in the* **Subtract Real Volumes** *form.*

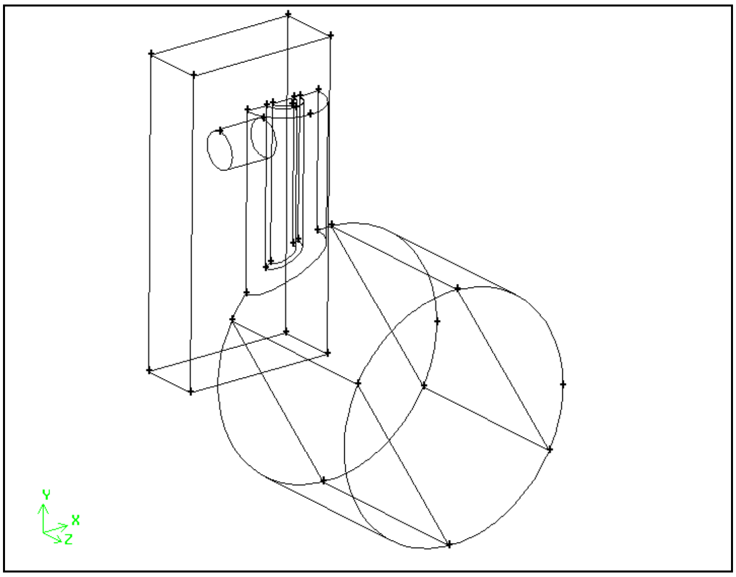


Figure 7-18: Brick subtracted from the pipe geometry

3. Subtract the brick from the small pipe.

   a) Select the smallest cylinder in the graphics window, and accept the selection.

   b) Select the brick in the graphics window.

   c) Unselect the Retain check box.

   d) Click **Apply**.

   Figure 7-19 shows the geometry after the subtraction.

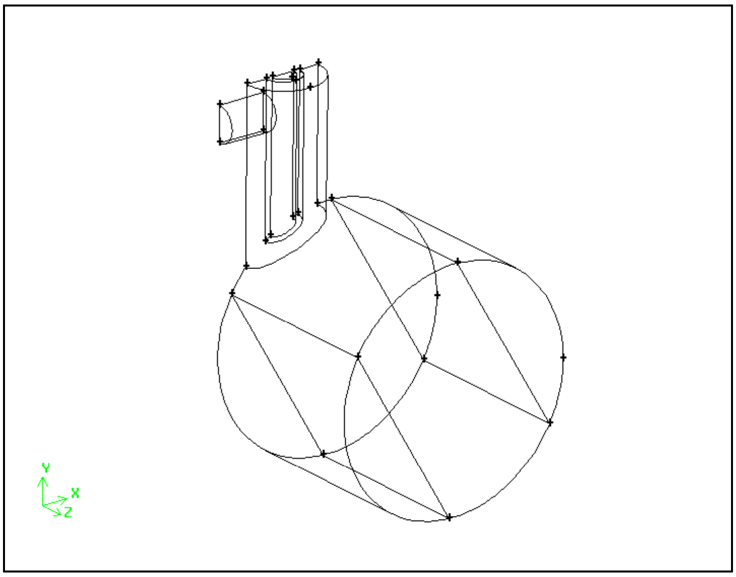


Figure 7-19: Brick subtracted from the small pipe to create a symmetry plane

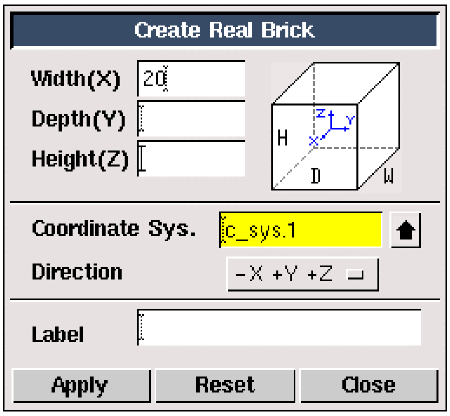
Step 8: Split off Annulus Pipe to Make the Volumes Meshable

1. Create a brick to split off part of the geometry.

   Again the decomposition of the cylinder is done before the full geometry has been created. In this example, we are using **Align** *instead of* **Move/Copy** *to position the tool to the appropriate position before the splitting*

   GEOMETRY → **VOLUME** → **CREATE VOLUME**

   This command sequence opens the **Create Real Brick** *form.*



   a) Enter a value of `20` for the **Width** of the brick. Delete the values in the **Depth** and **Height** text entry boxes.

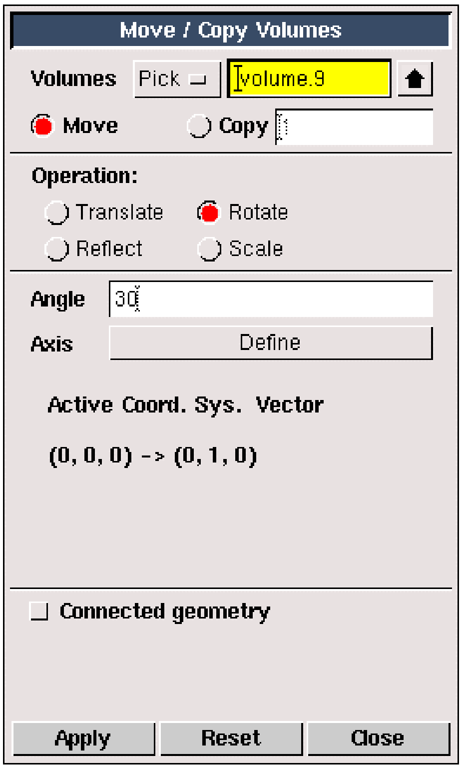
      GAMBIT *will set the* **Depth** *and* **Height** *by default to be the same value as the* **Width**, *to create a cube.*

   b) Select -X +Y +Z from the **Direction** option menu and click **Apply**.

2. Rotate the brick relative to the geometry.

   GEOMETRY → **VOLUME** → **MOVE/COPY/ALIGN VOLUMES**

   This command sequence opens the **Move / Copy Volumes** *form.*



a) Select the brick in the graphics window.

b) Retain **Move** (the default) under **Volume** at the top of the **Move / Copy Volumes** form.

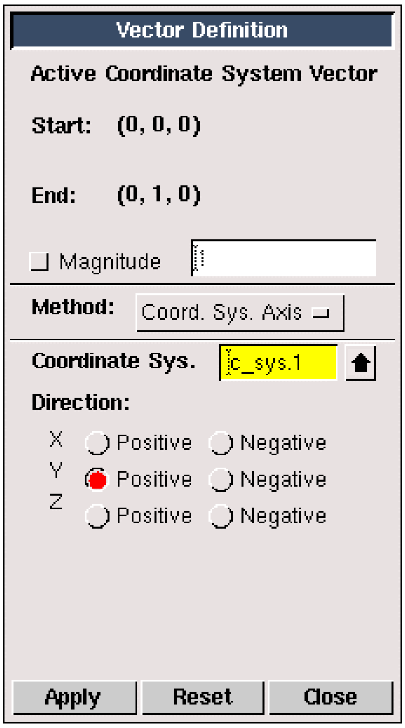
c) Under **Operation**, retain Rotate.

d) Retain the **Angle** of `30` for the angle of rotation.

You will now redefine the **Active Coord. Sys. Vector** *so that* GAMBIT *rotates the brick about the y axis.*

e) Click the Define button to the right of **Axis**.

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

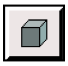
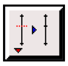


i. Select Y Positive under **Direction** and click **Apply**.

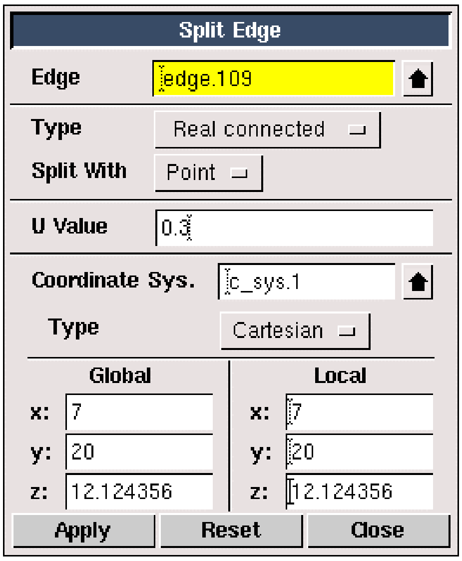
f) Click **Apply** in the **Move / Copy Volumes** form.

3. Create a vertex on the brick.

You will create a vertex on the brick and use it to align the brick correctly relative to the geometry. This is an alternative method to moving the splitting tool (the brick) to the right position in the geometry using coordinates.

GEOMETRY  → **EDGE**  → **SPLIT/MERGE EDGE** 

This command sequence opens the **Split Edge** *form.*



a) Select the edge on the brick marked A in Figure 7-20.

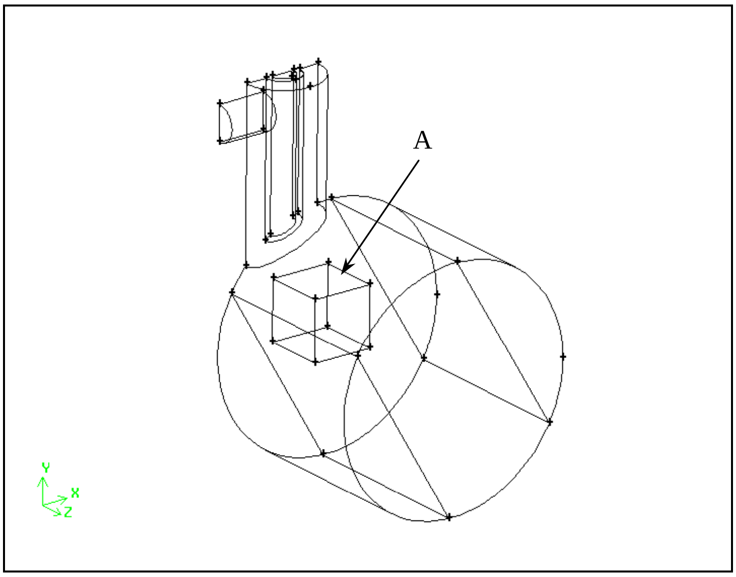


Figure 7-20: Edge to be split on the brick

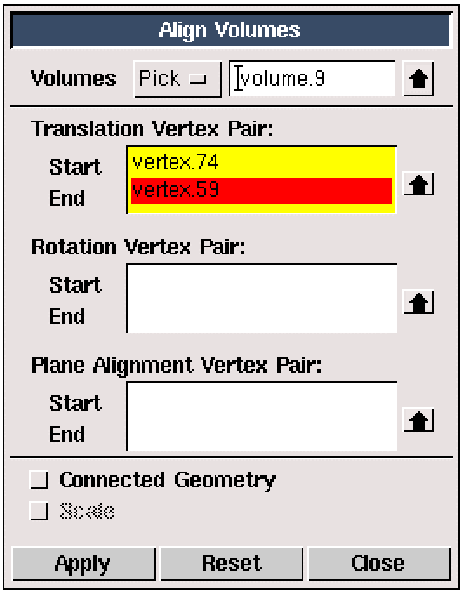
b) Enter a **U Value** of `0.3` in the **Split Edge** form and click **Apply**.

4. Align the brick with the part of the geometry to be split.

GEOMETRY → **VOLUME** → **MOVE/COPY/ALIGN VOLUMES**

R 

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



a) Select the brick in the graphics window and accept the selection.

The **Translation Vertex Pair** *list box in the* **Align Volumes** *form will be highlighted. You will now select the vertex on the object you want to move and then the vertex with which you want to align the object.*

b) Select the vertex you just created on the brick.

c) Select the vertex marked A in Figure 7-21. The vertex is on the end of the long thin pipes near the smallest cylinder.

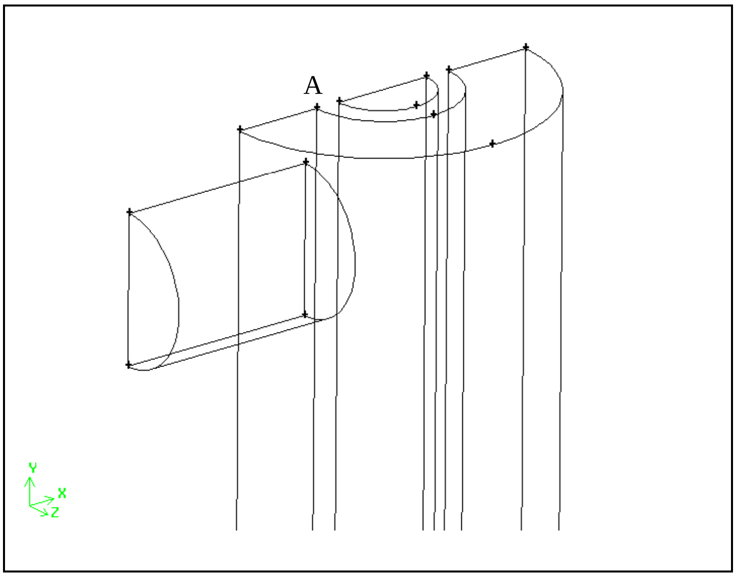


Figure 7-21: Vertex to be selected to align the brick

d) Click **Apply**.

The brick will be aligned with the pipe volume as shown in Figure 7-22.

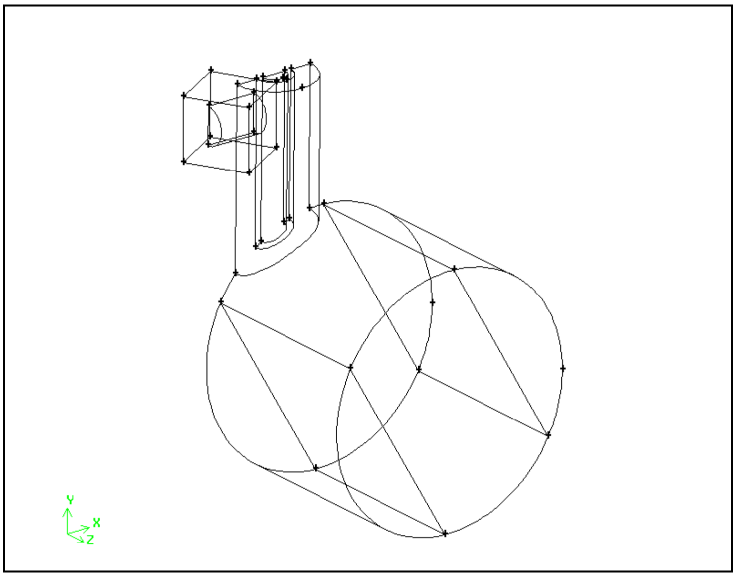
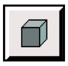
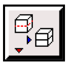
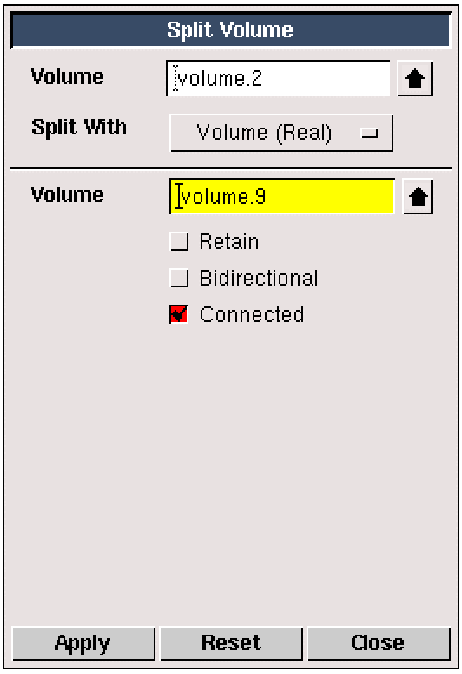


Figure 7-22: Brick aligned with pipe geometry

5. Use the brick to split the volume that contains most of the pipe sections.

GEOMETRY  → **VOLUME**  → **SPLIT/MERGE VOLUMES** 

This command sequence opens the **Split Volume** *form.*



a) Select the volume that contains most of the pipe sections in the geometry (the volume marked A in Figure 7-23) and accept the selection.

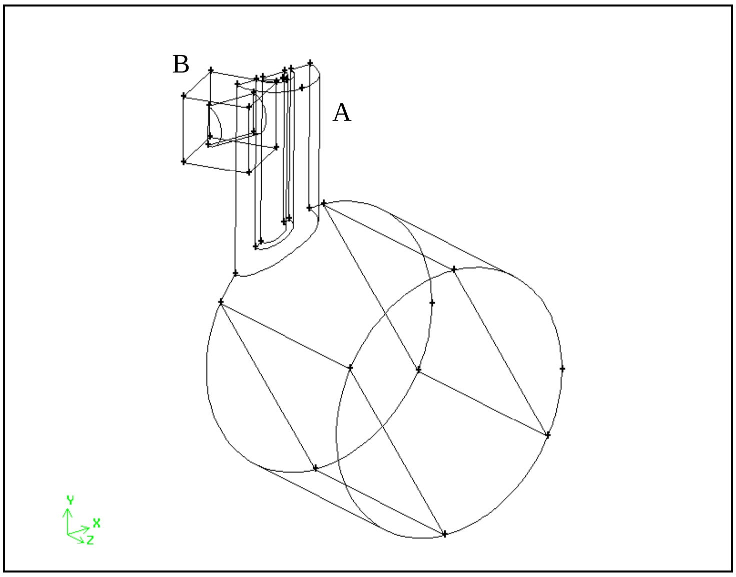


Figure 7-23: Volumes to be used in the split

b) Retain Volume (Real) as the **Split With** option.

c) Left-click in the **Volume** list box located below the **Split With** section to make the **Volume** list box active.

d) Select the brick (marked B in Figure 7-23) in the graphics window.

e) Click **Apply**.

The geometry will be split as shown in Figure 7-24.

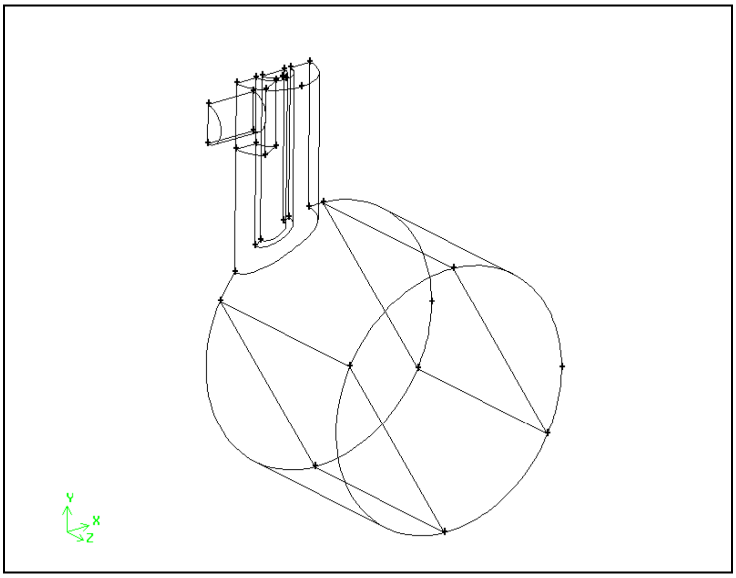


Figure 7-24: Decomposed geometry

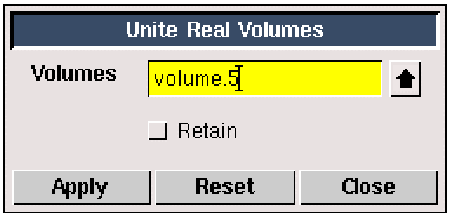
Step 9: Unite the Side Pipe

This is the final unite operation to complete the construction of the geometry

1. Unite two more volumes.

GEOMETRY → **VOLUME** → **BOOLEAN OPERATIONS**
R Unite

This command sequence opens the **Unite Real Volumes** *form.*



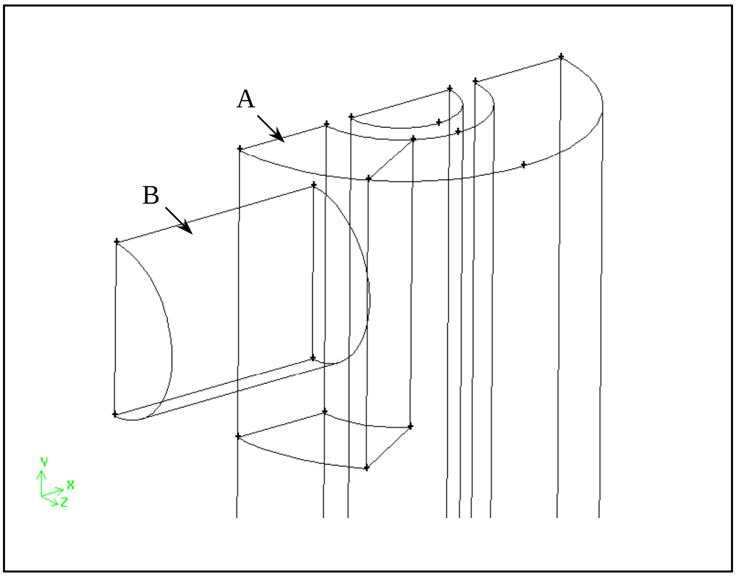


Figure 7-25: Volume to be united

a) Select the volume you created in the previous step (marked A in Figure 7-25).

b) Select the small half-cylinder (the volume marked B in Figure 7-25).

c) Click **Apply**.

The united volumes are shown in Figure 7-26.

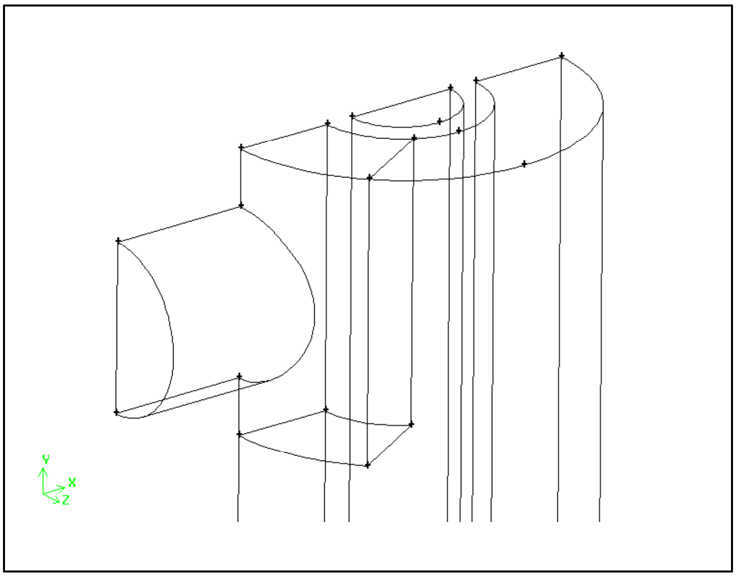


Figure 7-26: United volumes

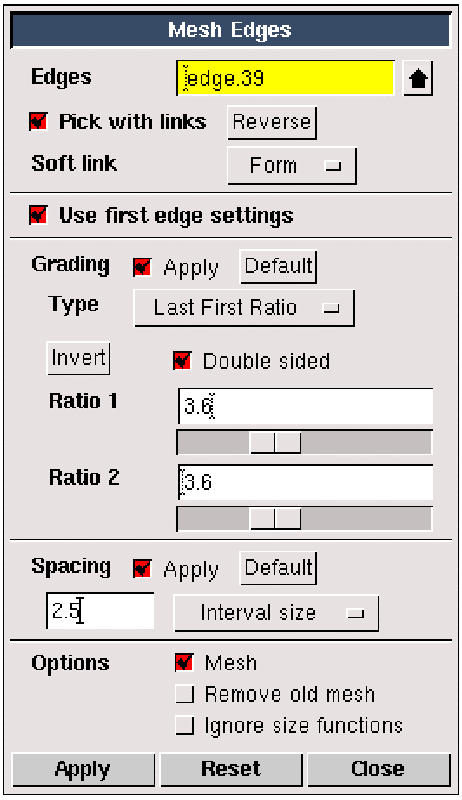
Step 10: Mesh the Edges

In this step, you will define the grid density on some edges of the geometry. You will accomplish this by selecting an edge, assigning the number of nodes, and specifying the distribution of nodes along the edge.

1. Define the grid density on four edges of the geometry.

MESH  → **EDGE**  → **MESH EDGES** 

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



a) Select the edges marked A, B, C, and D in Figure 7-27.

The edges will change color and an arrow and several circles will appear on each edge. The arrow is small and you may have to zoom into the edge to see it. It is located near the center of the edge.

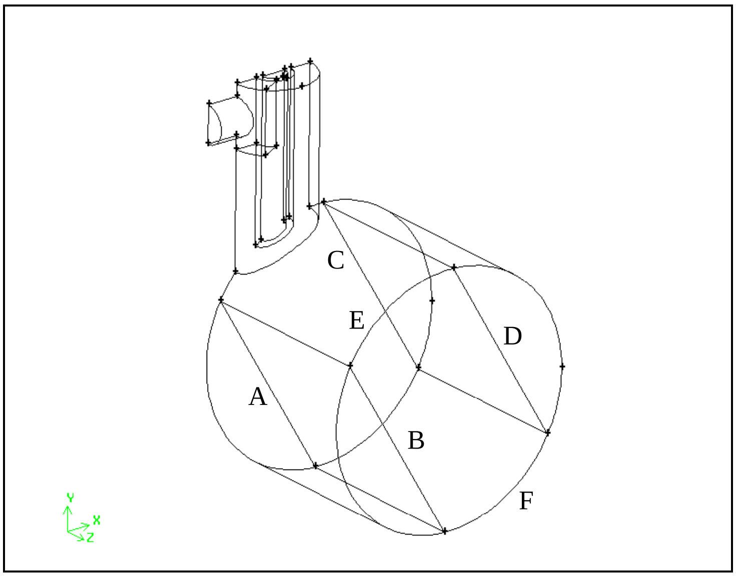


Figure 7-27: Edges to be selected for edge meshing

b) Ensure that **Apply** is selected to the right of **Grading** in the **Mesh Edges** form and select Last First Ratio from the **Type** option menu.

> *The* Last First Ratio *is defined as the size ratio between the first element (or grid distance) on the edge, and the last, based on the direction (sense) of the edge. For double sided grading, the last first ratio is the ratio between the central element and the element at the end of the edge*

c) Enter a value of `3.6` for **Ratio**.

d) Select the Double sided check box

> *If you specify a* Double sided *grading on an edge, the element intervals are graded in two directions from a starting point on the edge.* GAMBIT *determines the starting point such that the intervals on either side of the point are approximately the same length.* **Ratio 2** *is automatically given the value of* **Ratio 1** *(which is equal to the specified value of* **Ratio**).

e) Ensure that the **Apply** check box is selected to the right of **Spacing**. Select Interval size from the option menu under **Spacing** and enter a value of `2.5` in the text entry box.

f) Click **Apply**.

2. Define the grid density on two edges of the geometry.

   a) Select the edges marked E and F in Figure 7-27.

   b) Ensure that **Apply** is selected to the right of **Grading** in the **Mesh Edges** form and select Successive Ratio from the **Type** option menu.

      The Successive Ratio *option sets the ratio of distances between consecutive points on the edge equal to the* **Ratio** *specified in the* **Mesh Edges** *form.*

   c) Retain the default **Ratio** of `1`.

   d) Ensure that the **Apply** check box is selected to the right of **Spacing**. Enter `5` next to Interval size.

   e) Click **Apply**.

      The edge meshing for the six edges is shown in Figure 7-28.

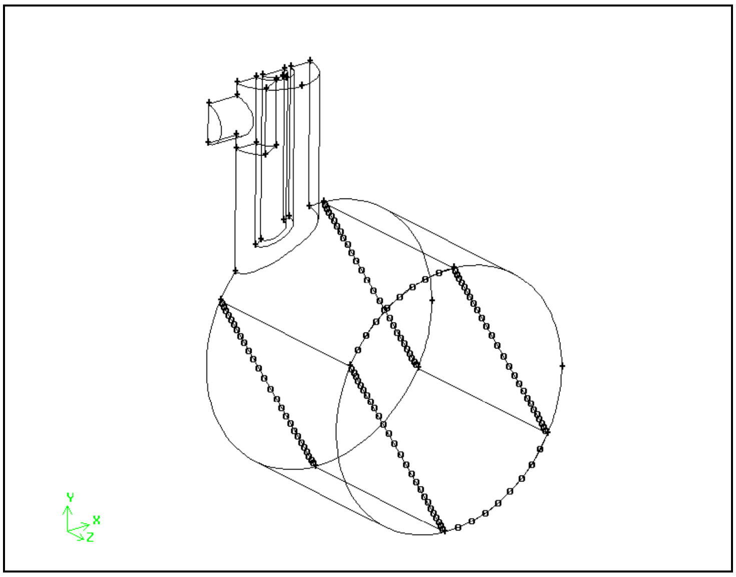
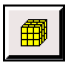


Figure 7-28: Edge meshing on the complex pipe junction geometry

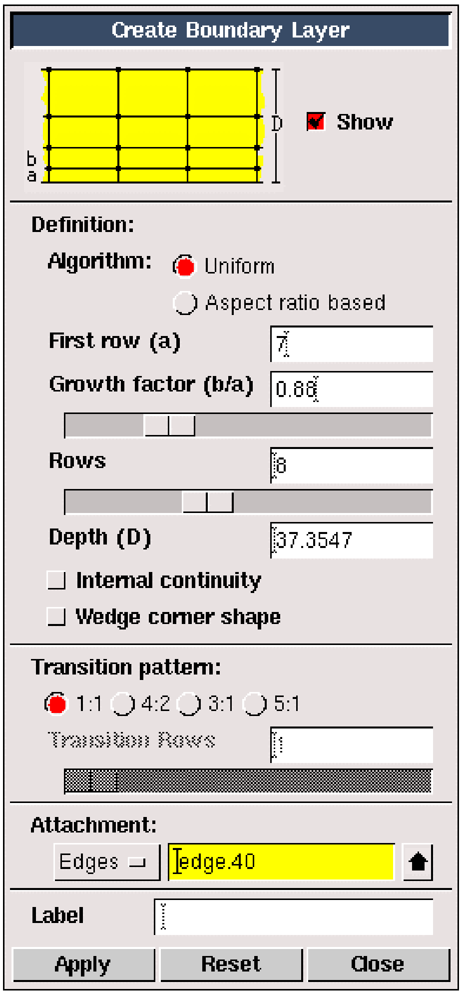
Step 11: Apply Boundary Layers

Boundary layers are layers of elements growing out from a boundary into the domain. They are used to locally refine the mesh in the direction normal to a face or an edge. Boundary layers are used in this example to improve mesh density close to walls on faces that will be paved

1. Create boundary layers on one edge.

   MESH  → **BOUNDARY LAYER**  → **CREATE BOUNDARY LAYER** 

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



a) Under **Definition**, retain the default **Algorithm** (Uniform).

b) Enter `7` in the **First row** text box.

This defines the height of the first row of elements normal to the edge.

c) Enter `0.88` in the **Growth factor** text box.

This sets the ratio of distances between consecutive rows of elements.

d) Move the slider box below **Rows** until the number of rows is equal to `8`.

This defines the total number of element rows. Notice that GAMBIT *updates the* **Depth** *automatically. The depth is the total height of the boundary layer.*

e) Retain the default **Transition pattern** (1:1).

f) Select the edge shown in Figure 7-29 with the boundary layer on it. The boundary layer should appear in the direction shown in Figure 7-29. If it does not, *Shift*-middle-click the edge to change the direction of the boundary layer.

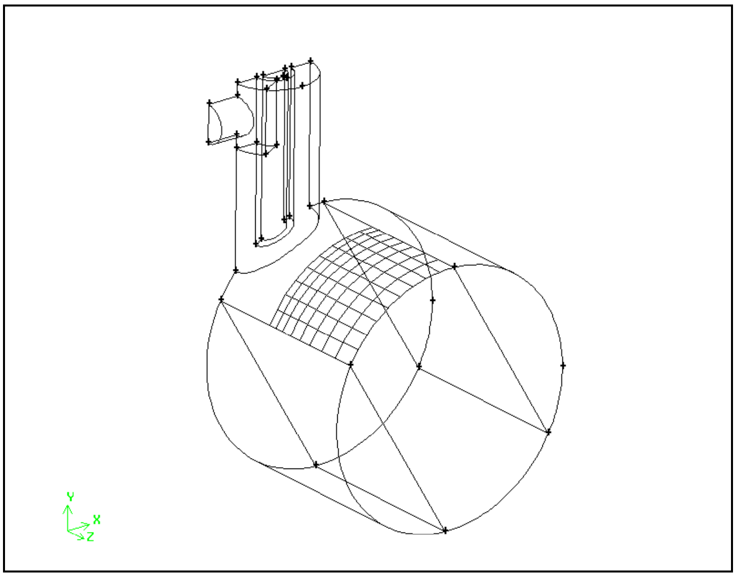


Figure 7-29: Edge on which to apply the boundary layer, showing the direction in which the boundary layer should point

g) Click **Apply** to apply the boundary layer to the edge.

2. Create boundary layers on the edges shown in Figure 7-30.

a) Under **Definition**, retain the default **Algorithm** (Uniform).

b) Enter `0.5` in the **First row** text box.

c) Enter `1.5` in the **Growth factor** text box.

d) Move the slider box below **Rows** until the number of rows is equal to `2`.

e) Retain the default **Transition pattern** (1:1).

f) Select the edges shown in Figure 7-30 with boundary layers on them. The boundary layers should appear in the directions shown in Figure 7-30. If they do not, *Shift*-middle-click an edge to change the direction of the boundary layer.

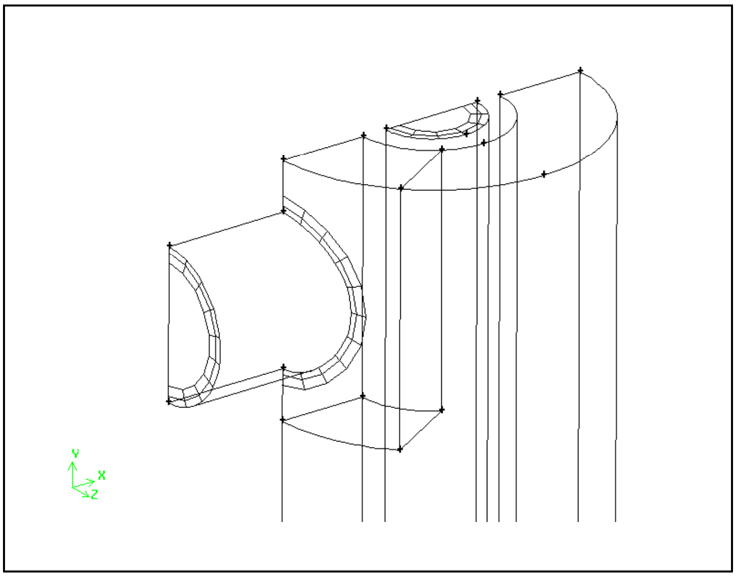


Figure 7-30: Edges on which to apply the boundary layers, showing the directions in which the boundary layers should point

g) Click **Apply** to apply the boundary layers to the edges.

3. Create boundary layers on the edges shown in Figure 7-31.

   a) Enter `1` in the **First row** text box.

   b) Retain `1.5` in the **Growth factor** text box.

   c) Retain the **Rows** value (`2`).

   d) Retain the default **Transition pattern** (1:1).

   e) Select the edges shown in Figure 7-31 with boundary layers on them. The boundary layers should appear in the directions shown in Figure 7-31. If they do not, *Shift*-middle-click an edge to change the direction of the boundary layer.

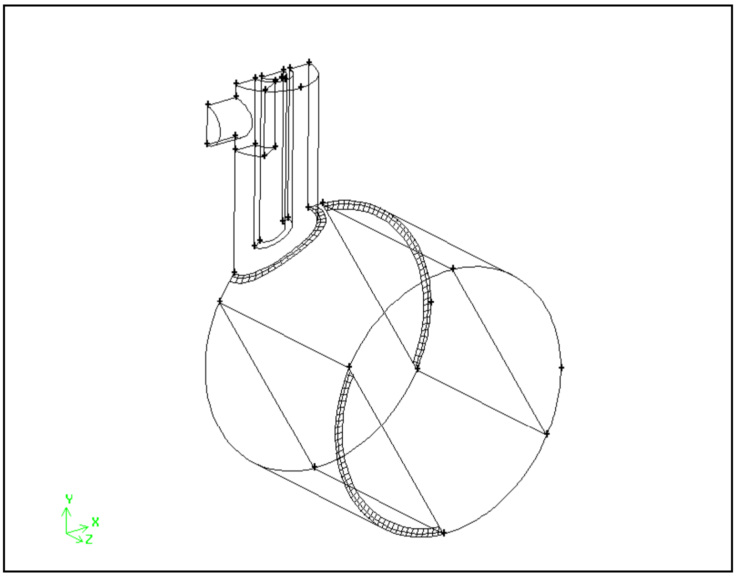


Figure 7-31: Edges on which to apply the boundary layers, showing the directions in which the boundary layers should point

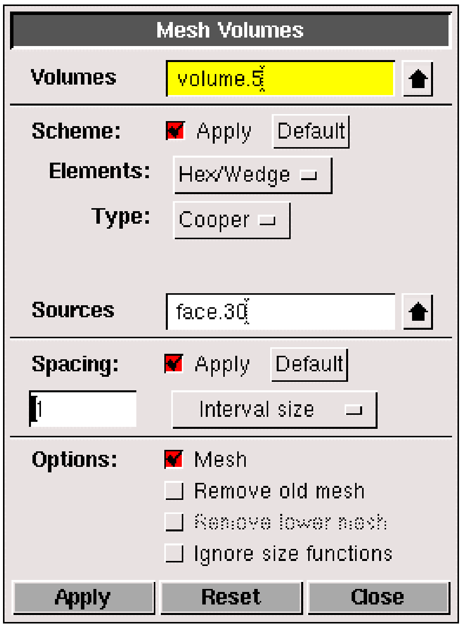
   f) Click **Apply** to apply the boundary layers to the edges.

Step 12: Mesh One of the Volumes

1. Mesh the volume marked C in Figure 7-32.

   MESH → **VOLUME** → **MESH VOLUMES**

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



   a) Select the volume marked C in Figure 7-32 in the graphics window.

      GAMBIT *will automatically select the* Cooper **Scheme** *in the* **Mesh Volumes** *form. See the* GAMBIT *Modeling Guide for more information on the* Cooper *meshing scheme.*

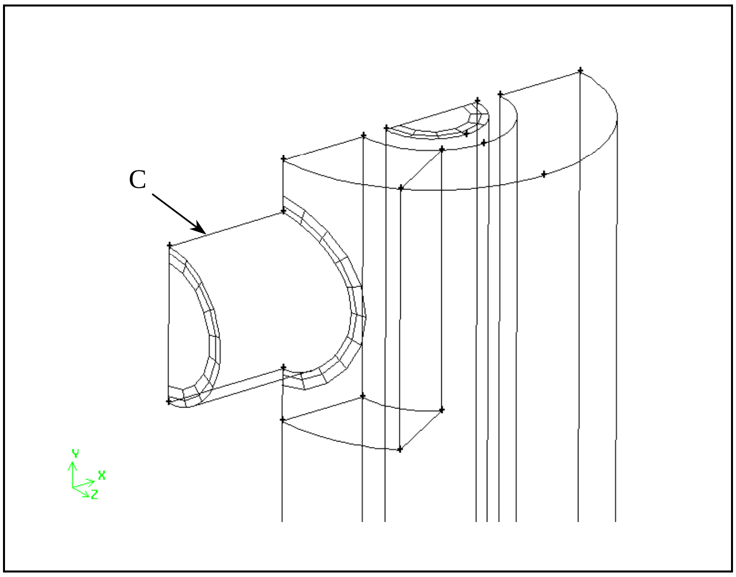


Figure 7-32: Volume to be meshed

b) Under **Spacing**, enter a value of `1` for the Interval size and click the **Apply** button at the bottom of the **Mesh Volumes** form.

This accepts the volume you selected as the one to be meshed. It also accepts the source faces (the faces whose surface mesh is to be swept through the volume to form volume elements) GAMBIT *has chosen for the* Cooper *meshing scheme and starts the meshing. If you need to modify or confirm the source faces, either pick faces from the graphics window or modify the selection of source faces by means of the* **Sources** *face text box. The mesh for the volume is shown in Figure 7-33.*

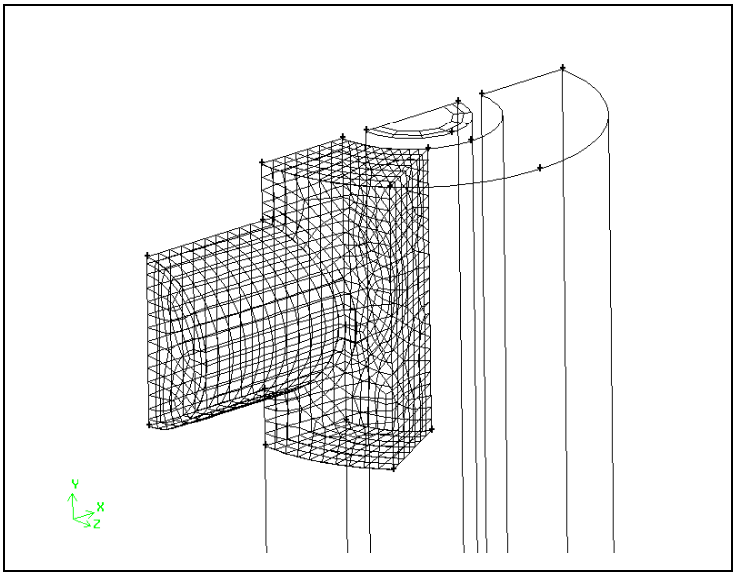


Figure 7-33: Mesh for volume C

It may be useful to remove the mesh from the display before you mesh the faces in the next exercise; it is then easier to see the faces of the geometry. The mesh is not deleted, just removed from the graphics window. To remove the mesh from the display, click the **SPECIFY DISPLAY ATTRIBUTES** *command button*  *at the bottom of the* **Global Control** *toolpad. Select the* Off *radio button to the right of* **Mesh** *near the bottom of the form and click* **Apply**.

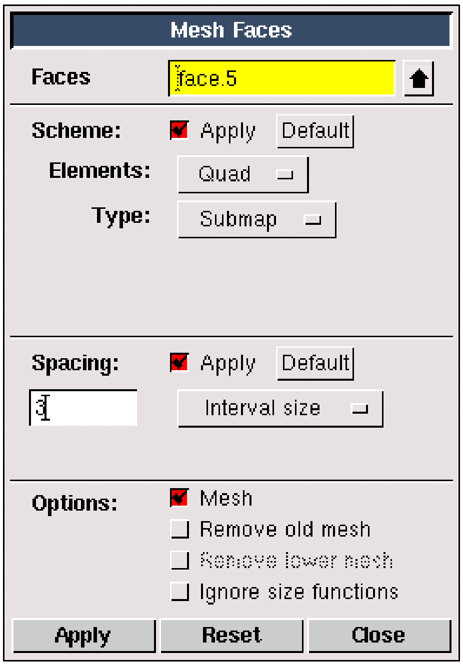
Step 13: Mesh Some Faces

These faces are meshed to ensure a good mesh density around the pipes and mapped meshes on some of the source faces. Some side faces also need to be meshed to assure mesh matching between different source faces

1. Mesh the face marked A in Figure 7-34.

   MESH → **FACE** → **MESH FACES**

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



   a) Select the curved face marked A in Figure 7-34.

      GAMBIT *will automatically select the* Submap **Scheme** *in the* **Mesh Faces** *form. See the* GAMBIT *Modeling Guide for more information on the* Submap *meshing scheme.*

   b) Change the Interval size to 3 under **Spacing** and click the **Apply** button.

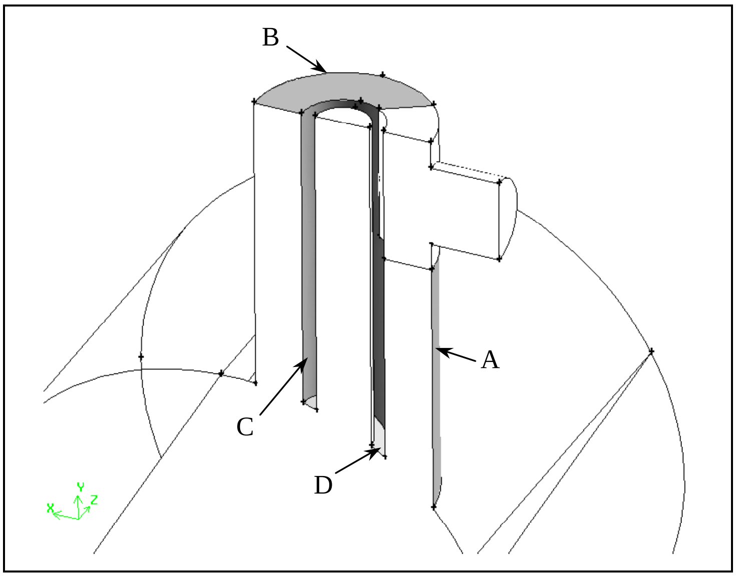


Figure 7-34: Faces to be meshed

The face will be meshed as shown in Figure 7-35.

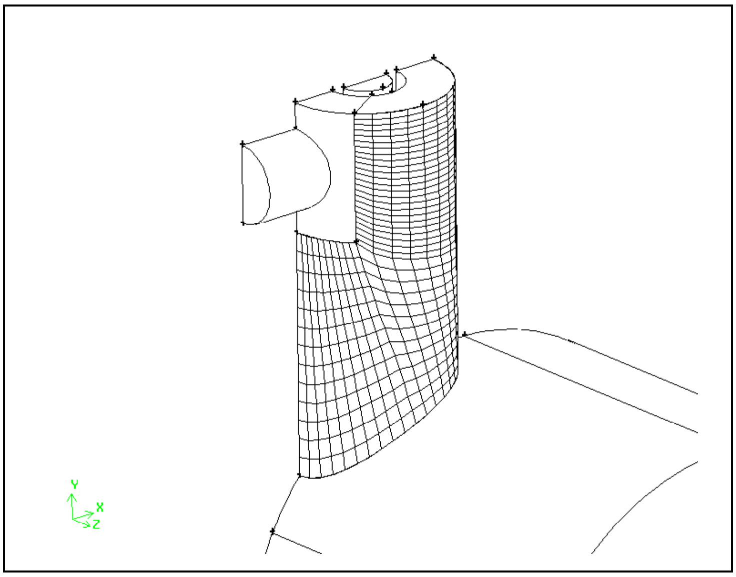


Figure 7-35: Mesh on face A

2. Mesh the face marked B in Figure 7-34

   a) Select the face marked B in Figure 7-34 in the graphics window.

      On most platforms, GAMBIT *will automatically select the* Map *meshing scheme on the* **Mesh Faces** *form. If* GAMBIT *does* not *automatically select the* Map *scheme, select it manually by means of the* **Scheme:Type** *option button.*

   b) Enter `1` as the Interval size, and click the **Apply** button at the bottom of the form.

      The face will be meshed as shown in Figure 7-36.

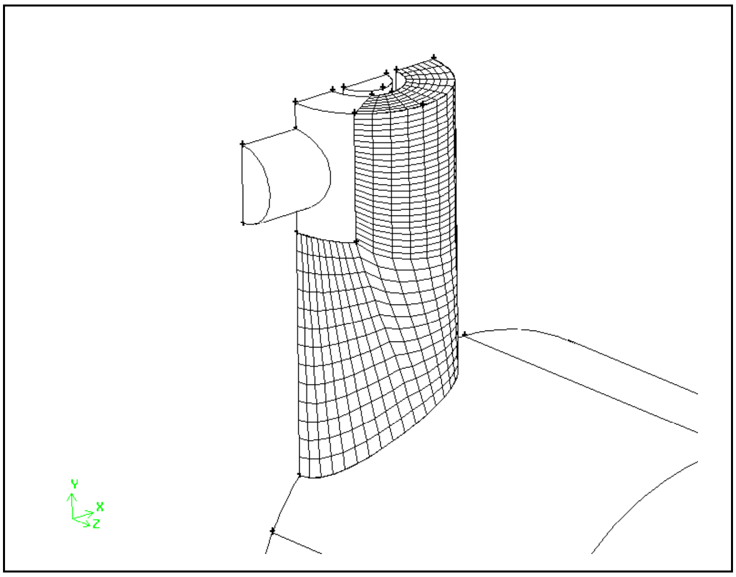


Figure 7-36: Mesh on faces A and B

3. Mesh the face marked C in Figure 7-34

   a) Select the face marked C in Figure 7-34 in the graphics window.

      GAMBIT *will automatically select the* Submap **Scheme** *in the* **Mesh Faces** *form.*

   b) Enter an Interval size of `2.5`, and click the **Apply** button at the bottom of the form.

      The face will be meshed as shown in Figure 7-37.

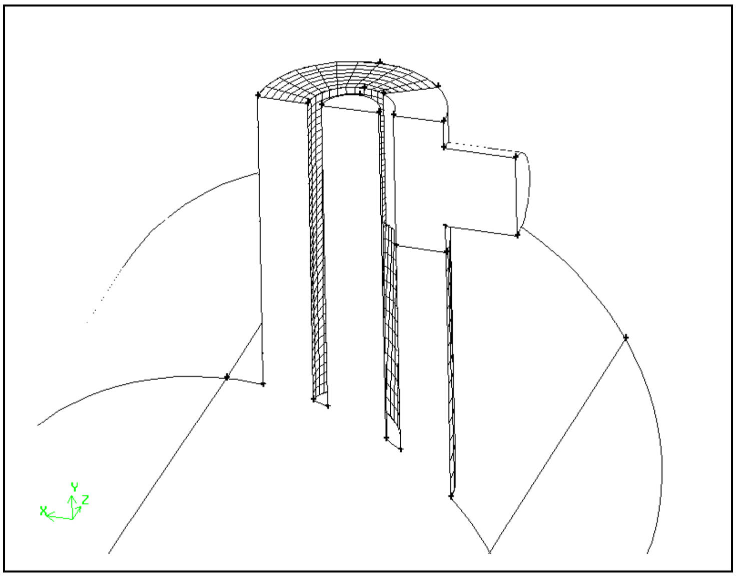


Figure 7-37: Mesh on faces A, B, and C

4. Mesh the face marked D in Figure 7-34.

   a) Select the face marked D in Figure 7-34 in the graphics window.

      GAMBIT *will automatically select the* Map **Scheme** *in the* **Mesh Faces** *form.*

   b) Enter `1` as the Interval size, and click the **Apply** button at the bottom of the form.

      The face will be meshed as shown in Figure 7-38.

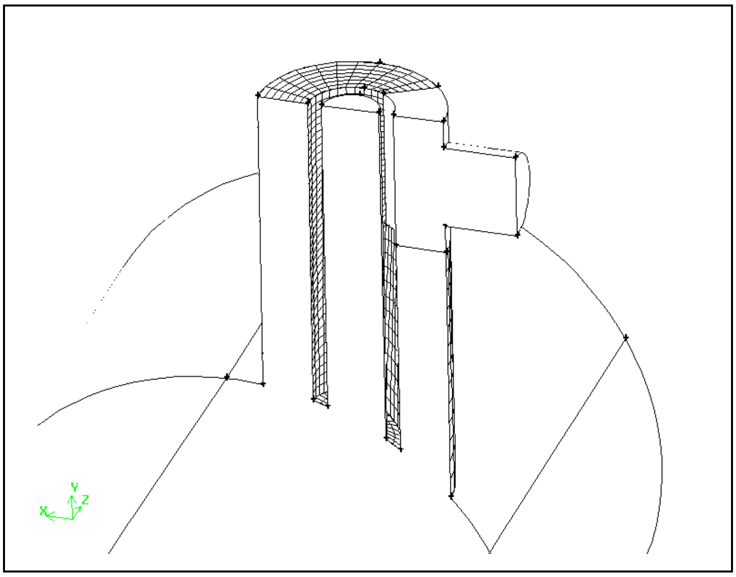


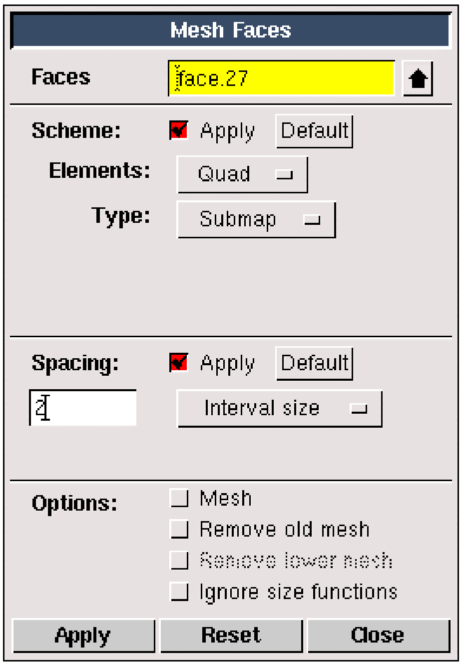
Figure 7-38: Mesh on faces A, B, C, and D

Step 14: Modify Mesh Settings on Some Faces

"Modifying mesh settings" is the same as "applying without meshing". This step illustrates two different applications to this technique:

- *First, you will modify the scheme setting on two faces from* Pave *to* Submap*. This is one way of making the main volume ready for* Cooper *meshing*
- *Second, you will modify the default size of one of the source faces. In this case, you allow the* Cooper *meshing scheme to make sure the mesh is matching with other source faces.*

1. Set the meshing scheme to be Submap for the faces marked F and G in Figure 7-39.



a) Select the faces marked F and G in Figure 7-39 in the graphics window.

b) Select Submap from the **Scheme:Type** option menu.

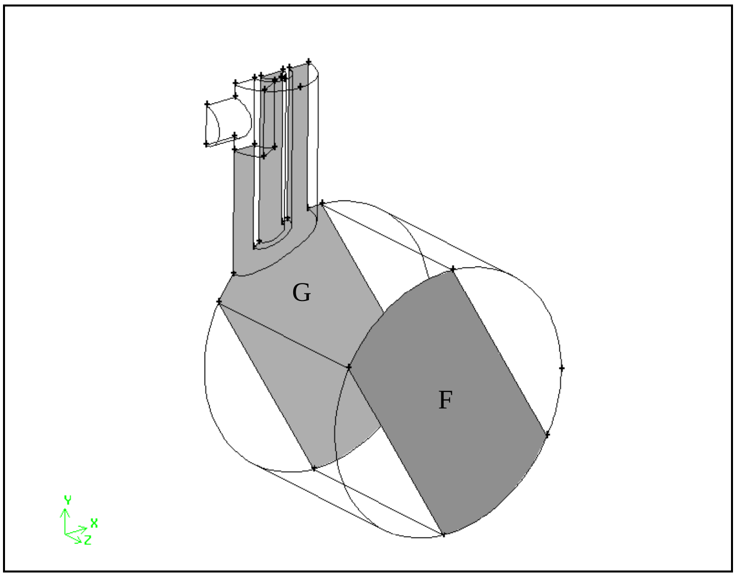


Figure 7-39: Faces to be modified

c) Retain the default Interval size of 2 under **Spacing**.

d) Unselect the Mesh check box under **Options**.

You deselected the Mesh *check box because at this point you do not want to mesh the faces; you only want to apply the* **Scheme** *to the faces.* GAMBIT *will mesh the faces using the* **Scheme** *you specified when it creates a volume mesh.*

e) Click the **Apply** button at the bottom of the form.

2. Set the meshing size on the face marked H in Figure 7-40.

a) Select the face marked H in Figure 7-40 in the graphics window (the face at the end of the pipe).

GAMBIT *will automatically select the* Pave **Scheme** *in the* **Mesh Faces** *form. See the* GAMBIT *Modeling Guide for more information on the* Pave *meshing scheme.*

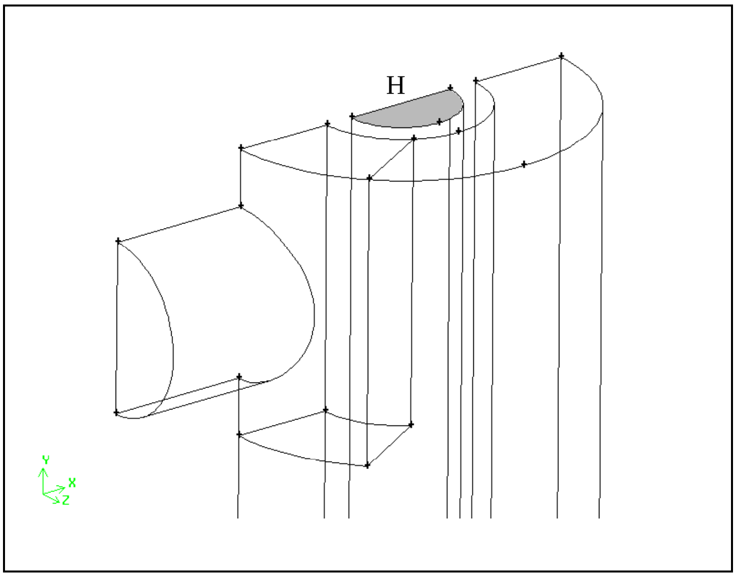


Figure 7-40: Face to be modified

b) Enter `1` as the Interval size under **Spacing**.

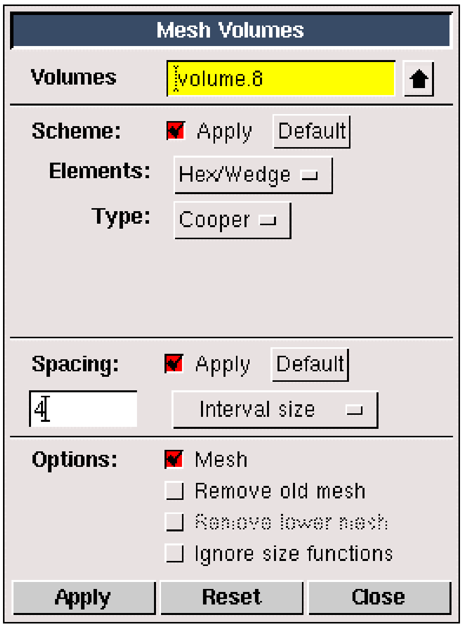
c) Ensure that Mesh is *not* selected under **Options** and click the **Apply** button at the bottom of the form.

Step 15: Mesh the Volumes

1. Mesh the volumes marked J and K in Figure 7-41.

MESH → **VOLUME** → **MESH VOLUMES**

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



a) Select the volumes marked J and K in Figure 7-41 in the graphics window.

GAMBIT *will automatically select the* Cooper **Scheme** *in the* **Mesh Volumes** *form. See the* GAMBIT *Modeling Guide for more information on the* Cooper *meshing scheme.*

b) Enter 4 as the Interval size under **Spacing** in the **Mesh Volumes** form and click the **Apply** button at the bottom of the form.

The volumes will be meshed as shown in Figure 7-42.

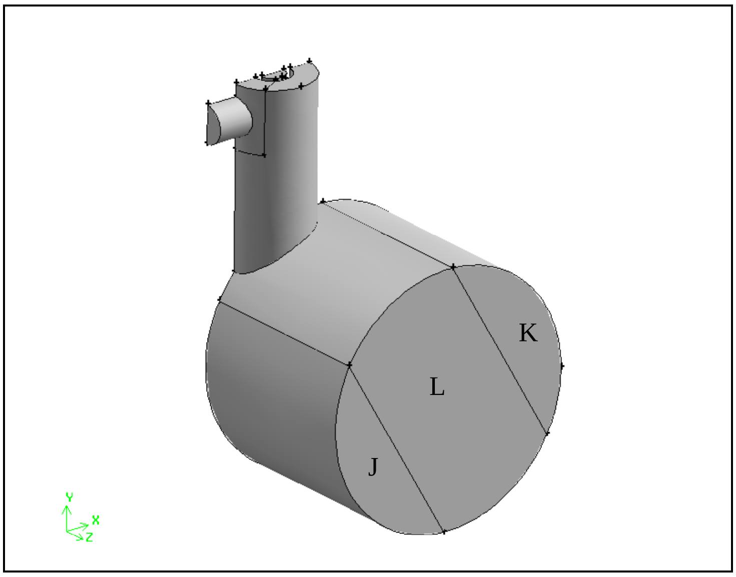


Figure 7-41: Volumes to be meshed

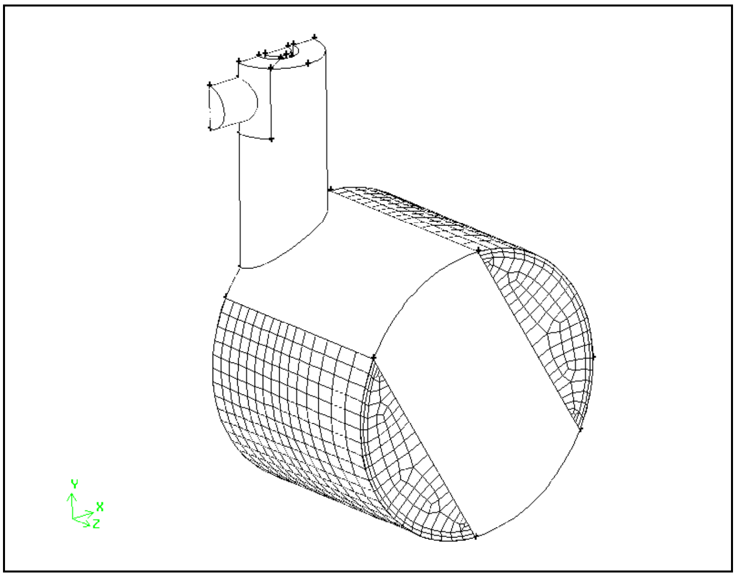


Figure 7-42: Mesh on volumes J and K

2. Mesh the volume marked L in Figure 7-41.

   a) Select the volume marked L in Figure 7-41 in the graphics window.

      GAMBIT *will automatically select the* Cooper **Scheme** *in the* **Mesh Volumes** *form.*

   b) Check that the Remove lower mesh and Remove old mesh check boxes are not selected at the bottom of the form.

   c) Click the **Apply** button again.

   To view the final mesh, click the **SPECIFY DISPLAY ATTRIBUTES** *command button*  *at the bottom of the* **Global Control** *toolpad. Select the* **Mesh:**On *and* **Render:**Hidden *options near the bottom of the form, and click* **Apply**. *The final volume mesh is shown in Figure 7-43.*

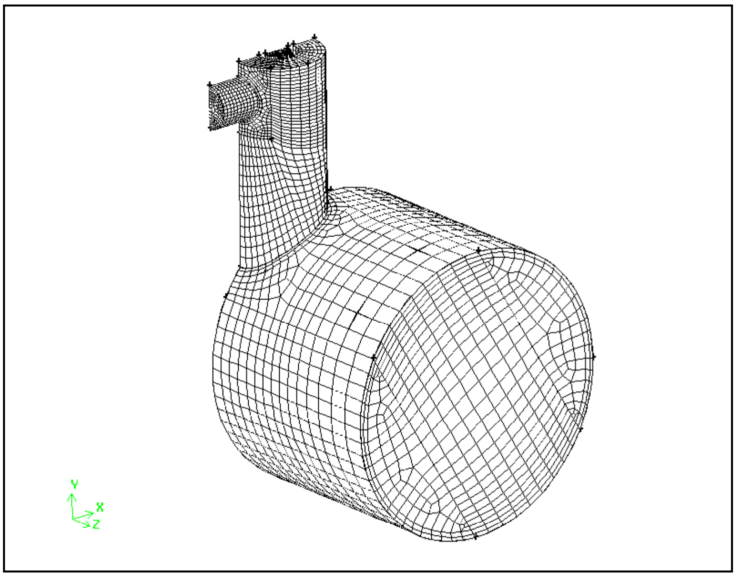
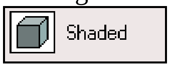


Figure 7-43: Final volume mesh

(<u>NOTE</u>: The Hidden *rendering option has been turned on in Figure 7-43 to make the mesh easier to see.)*

3. You can view the mesh by shading it using the **RENDER MODEL** command button in the **Global Control** toolpad

   a) Hold down the right mouse button on the **RENDER MODEL** command button  and select 

from the resulting list.

   b) Rotate and translate the volume to view the mesh.

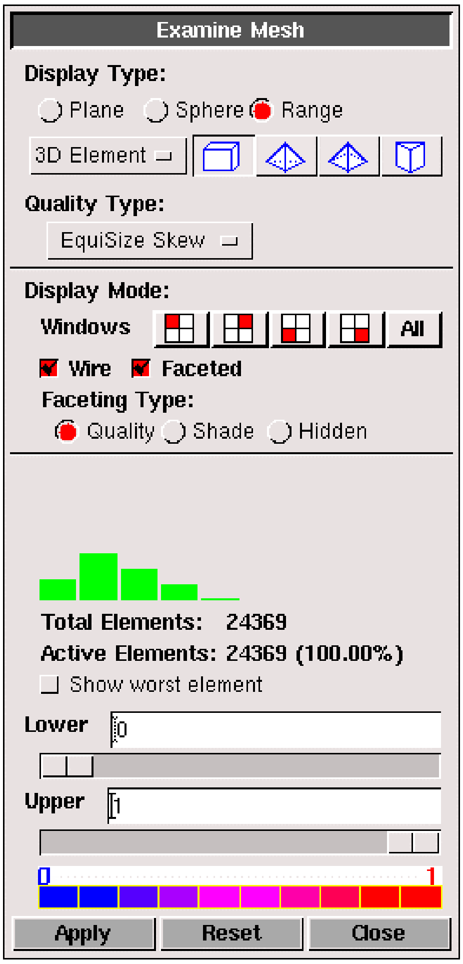
   c) When you are finished, return to the wireframe view of the model, by selecting the following command buttons in the **Global Control** toolpad:  **R**

.

Step 16: Examine the Volume Mesh

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

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



   a) Select Range under **Display Type**.

When you select the **Display Type:**Range *option on the* **Examine Mesh** *form,* GAMBIT *displays the* Show worst element *option immediately below the statistics displayed under the histogram. If you select the* Show worst element *option,* GAMBIT *displays only the "worst" element as determined by the current* **Quality Type** *quality metric.*

b) Select the Show worst element option.

c) Click the **FIT TO WINDOW** command button , at the top left of the **Global Control** toolpad, to see where the worst element is located with respect to the entire geometry.

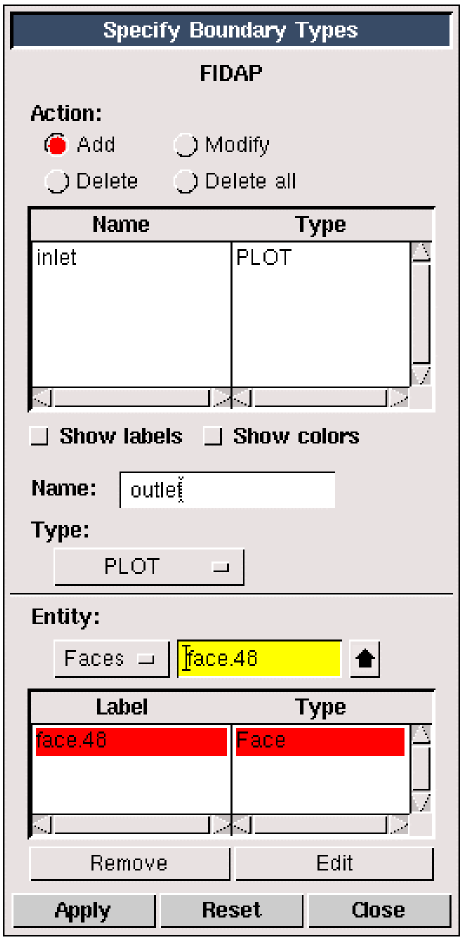
d) Close the **Examine Mesh** form by clicking the **Close** button at the bottom of the form.

Step 17: Set Zone Types and Export the Mesh

1. Set boundary types for the complex pipe junction.

   ZONES  → **SPECIFY BOUNDARY TYPES** 

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



! *It may be useful to remove the mesh from the display before you set the boundary types; it is then easier to see the faces of the geometry. The mesh is not deleted, just removed from the graphics window. To remove the mesh from the display, click the* **SPECIFY DISPLAY ATTRIBUTES** *command button*  *at the bottom of the* **Global Control** *toolpad. Select the* Off *radio button the right of* **Mesh** *near the bottom of the form and click* **Apply**.

a) Define an inlet.

i. Enter the name "`inlet`" in the **Name** text entry box.

ii. Select PLOT in the **Type** option menu.

iii. Check that Faces is selected as the **Entity**.

iv. *Shift*-left-click the face marked A in Figure 7-44 and accept the selection.

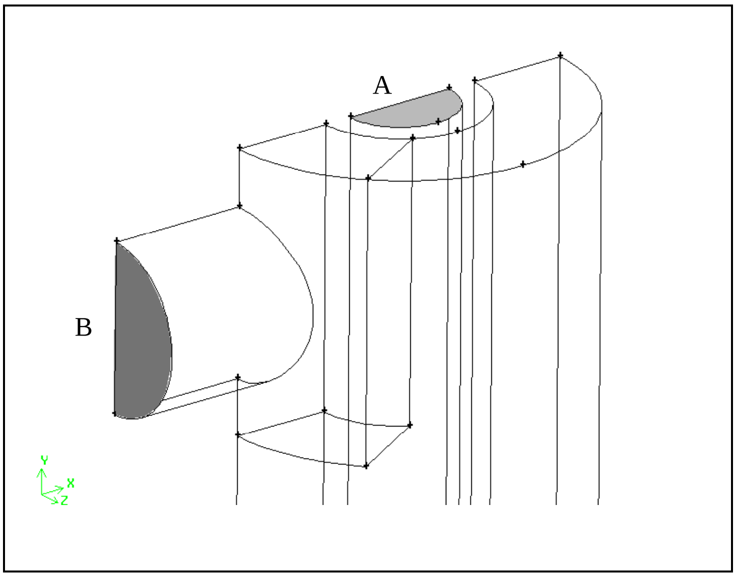


Figure 7-44: Faces to set as inlet, outlet, and wall boundaries

b) Define an outlet.

i. Enter the name "`outlet`" in the **Name** text entry box.

ii. Check that PLOT is still selected in the **Type** option menu.

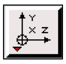
iii. Select the face marked B in Figure 7-44 and accept the selection.

c) Define symmetry boundary types for faces on the x-y plane.

You will pick the faces for this step using a GAMBIT *picking procedure that allows you to pick only entities that are completely within the picking box in the graphics window.*

i. Enter the name "`symmetry`" in the **Name** text entry box.

ii. Select PLOT in the **Type** option menu.

iii. Right-click the **ORIENT MODEL** command button at the lower left corner of the **Global Control** pad and select the -X  option to orient the model as shown in Figure 7-45.

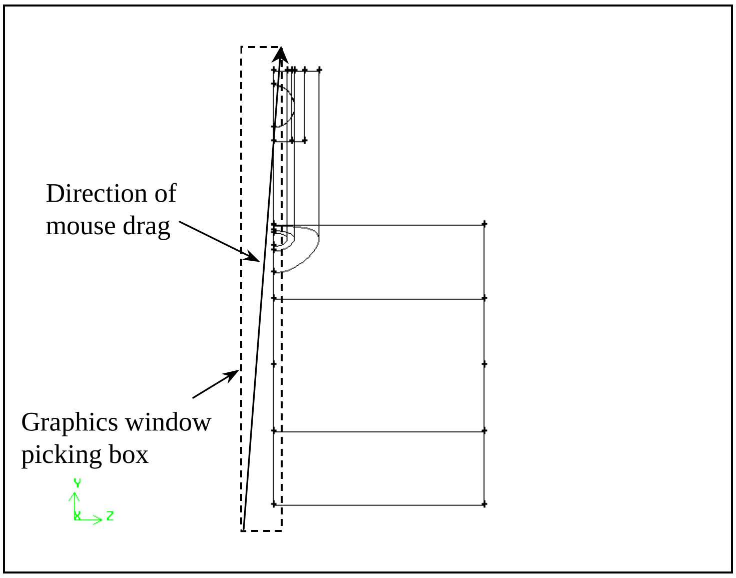


Figure 7-45: Model and picking box, -X view

iv. *Shift*-left-drag the mouse toward the upper right of the graphics window to create the picking box shown in Figure 7-45.

Note that GAMBIT *selects only the faces completely contained in the picking box. If you* Shift-*left-drag the mouse toward the lower left of the graphics window,* GAMBIT *selects all faces touched by the picking box.*

v. Accept the selection of the faces.

Note that you could also specify the remaining external faces of the tank geometry as WALL *boundaries. This is not necessary, however, because when* GAMBIT *saves a mesh, any external faces (in 3-D) for which you have not specified a boundary type will be written out as* WALL *boundaries by default.*

In addition, when GAMBIT *writes a mesh, any volumes (in 3-D) for which you have not specified a continuum type will be written as* FLUID *by default. This means that you do not need to specify a continuum type in the* **Specify Continuum Types** *form for this tutorial.*

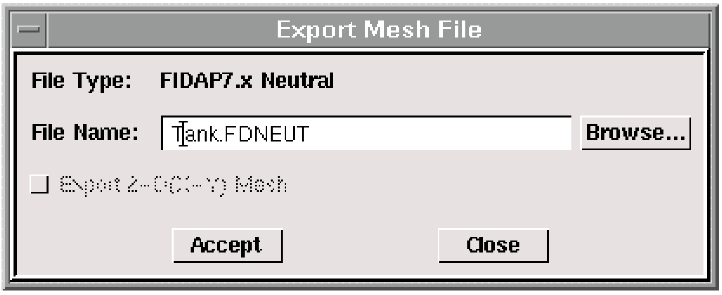
d) Select the **Show colors** option on the **Specify Boundary Types** form and manipulate the model in the graphics window to display the manner in which GAMBIT assigns colors to boundary types.

2. 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 (`Tank.FDNEUT`).

ii. Click **Accept**.

The grid file will be written to your working directory.

3. Save the GAMBIT session and exit GAMBIT

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

   File → **Exit.**

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

7.5 Summary

In this tutorial multiple primitive creations and Boolean operations were used to create the full geometry. Decomposition was embedded in the creation. Edge meshing, boundary layers and face meshing were used to control the mesh density, and type of mesh, in different areas of the model. The Cooper meshing scheme was used for all volumes.

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).

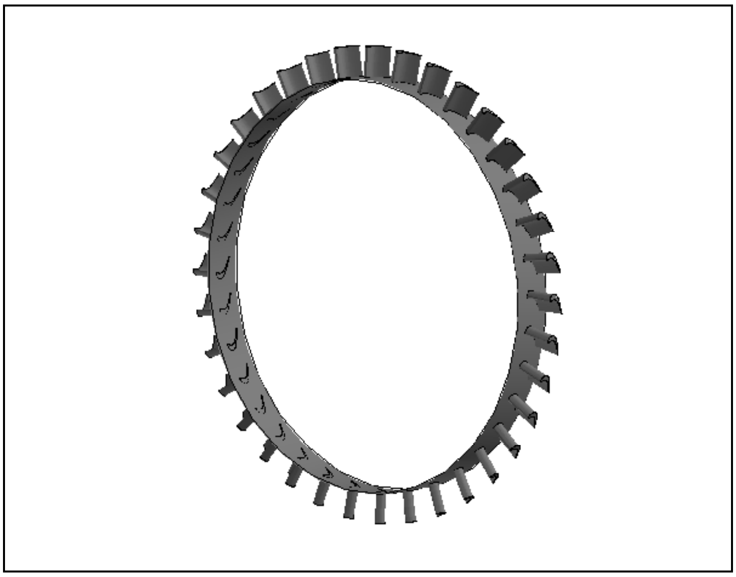


Figure 8-1: 60-blade turbine rotor

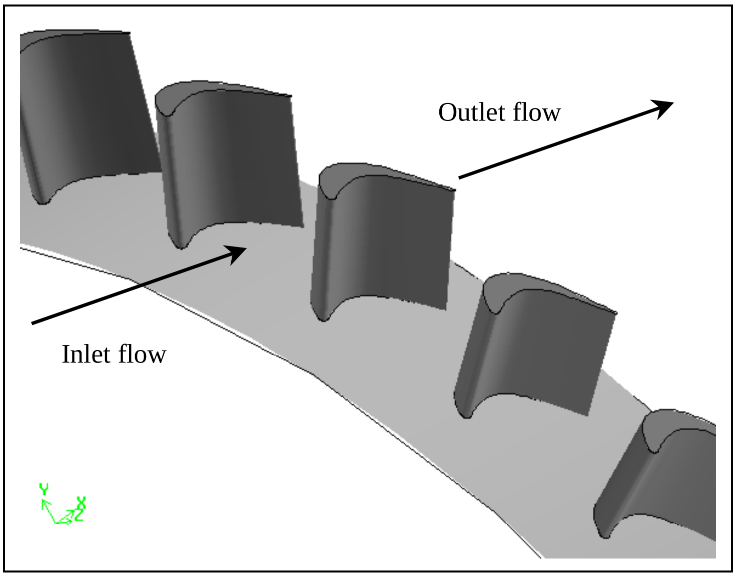


Figure 8-2: Turbine rotor blade configurations

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.*

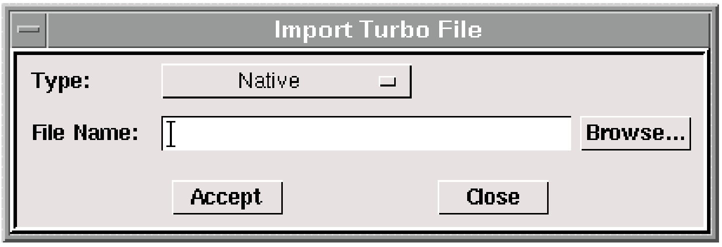
Step 2: Import a Turbo Data File

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.

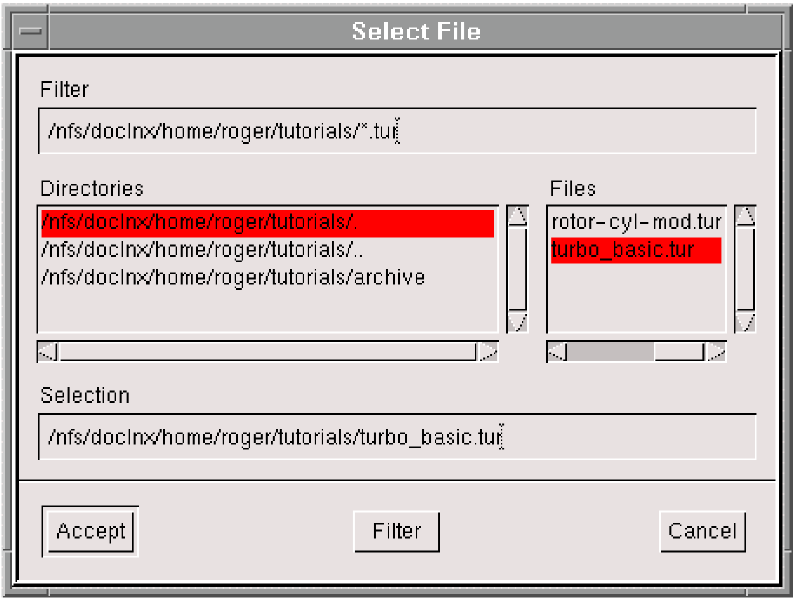
   File $\longrightarrow$ **Import** $\longrightarrow$ **Turbo...**

   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.*

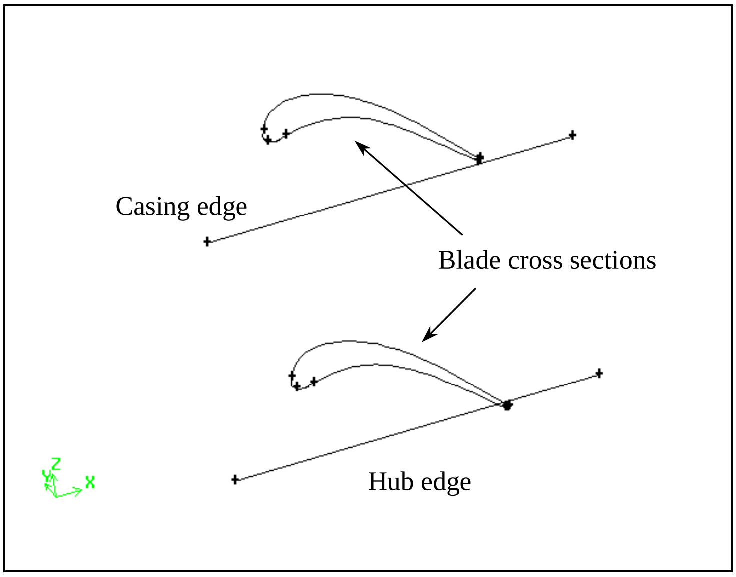


Figure 8-3: Imported turbo geometry

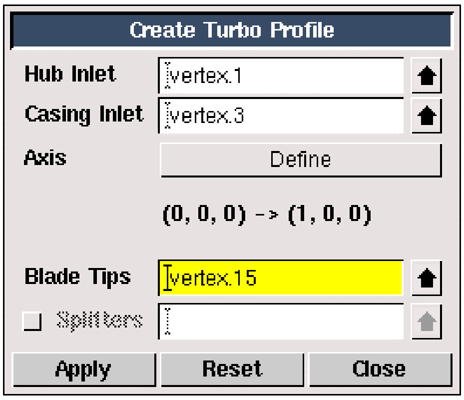
Step 3: Create the Turbo Profile

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.

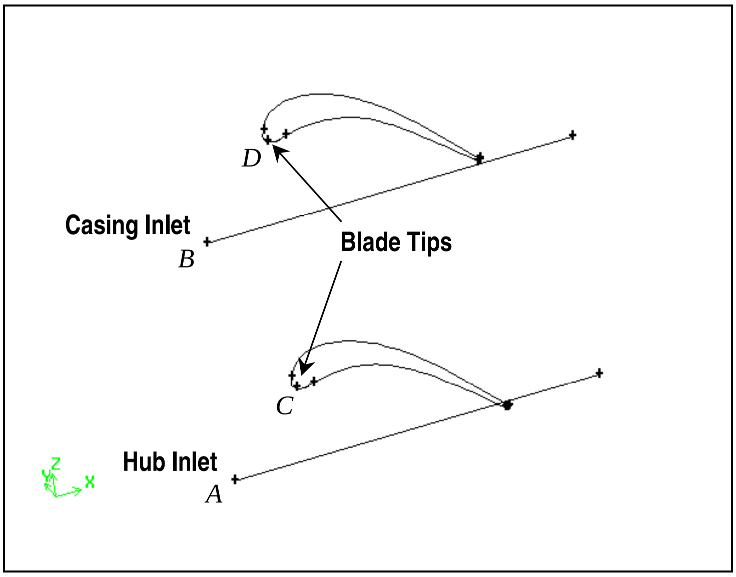


Figure 8-4: Vertices used to specify the turbo profile

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*.

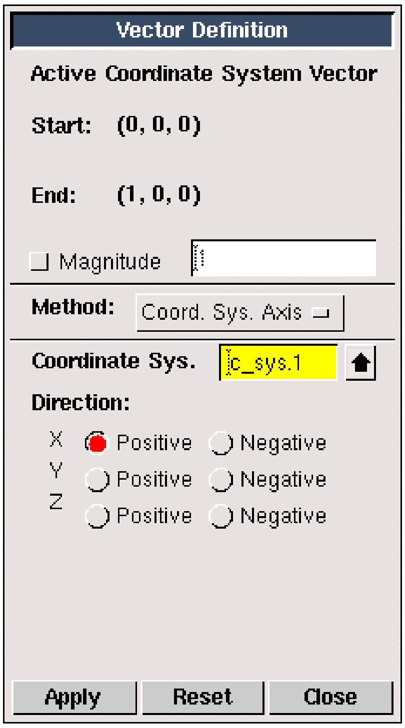
c) Activate the **Casing Inlet** list box.

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.*

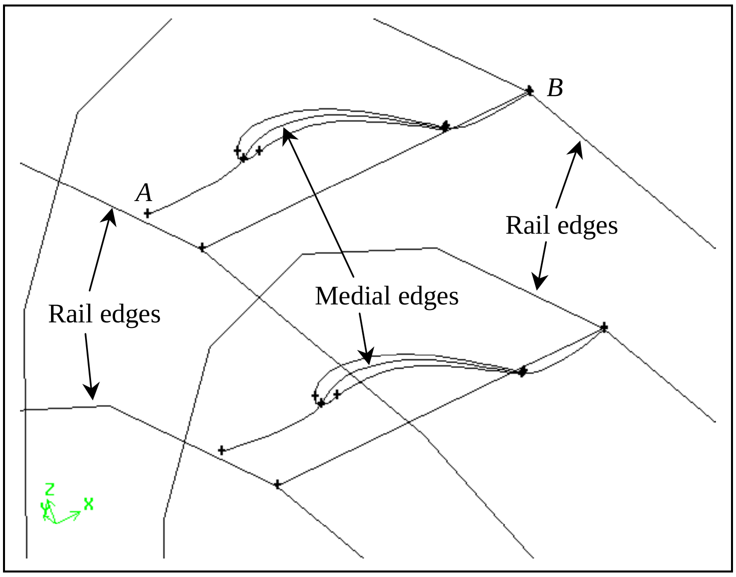


Figure 8-5: Turbo profile

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).

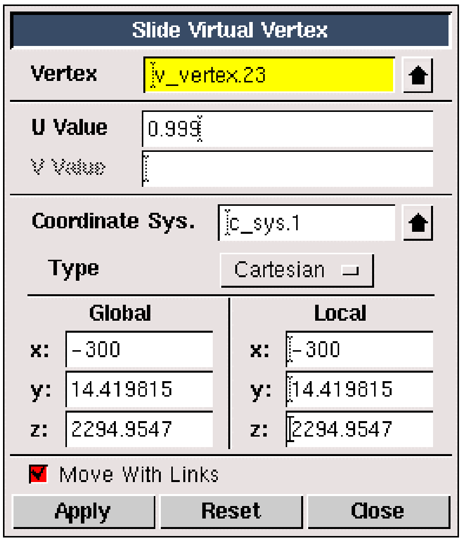
Step 4: Modify the Inlet and Outlet Vertex Locations

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.

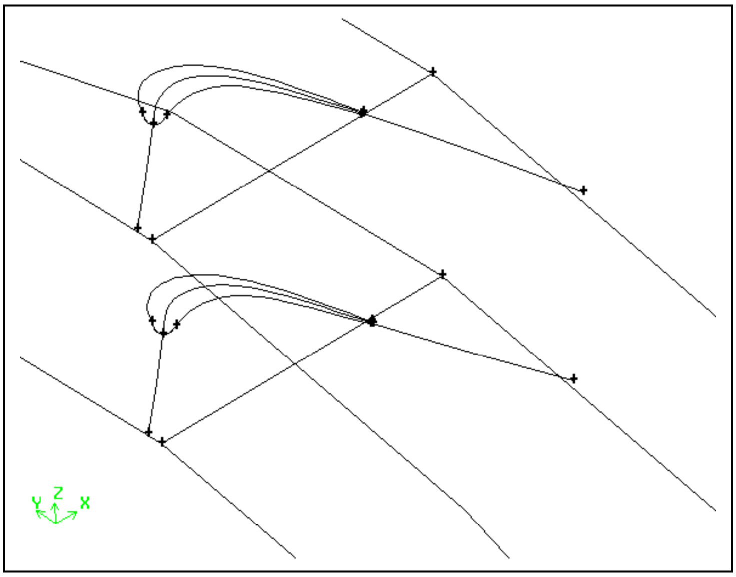


Figure 8-6: Turbo profile with modified inlet and outlet vertex locations

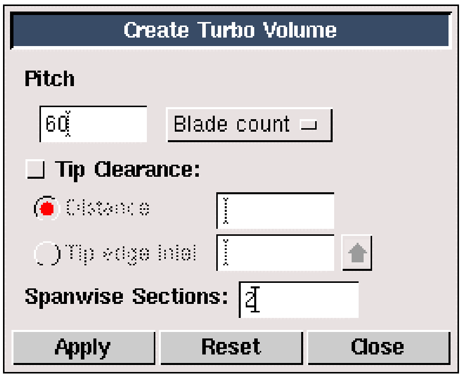
Step 5: Create the Turbo Volume

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.*

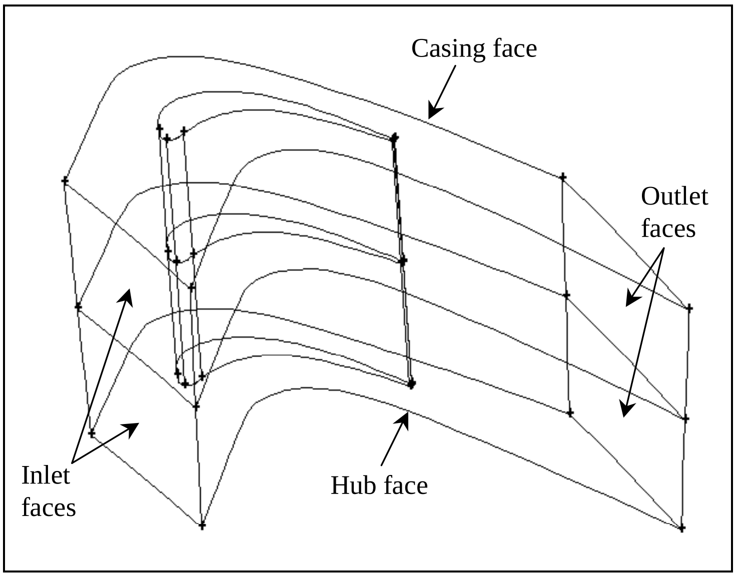


Figure 8-7: Turbo volume—consisting of two geometric volumes

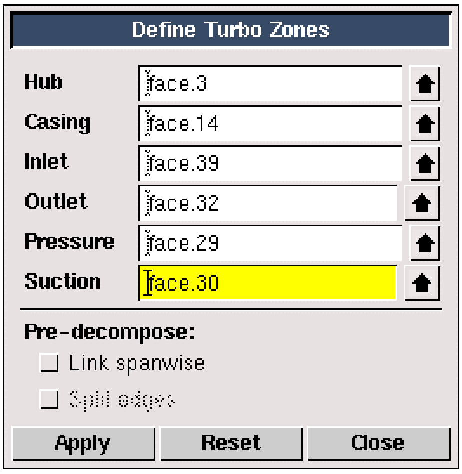
Step 6: Define the Turbo Zones

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.

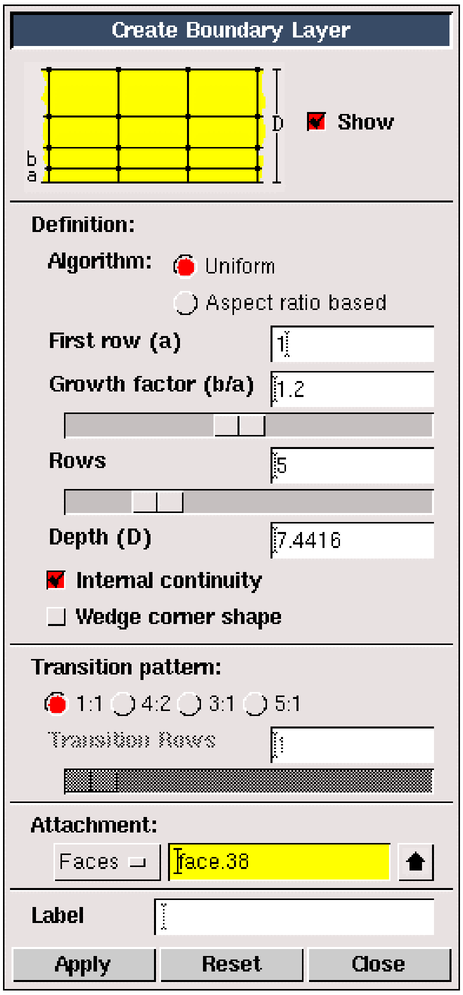
Step 7: Apply 3-D Boundary Layers

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.

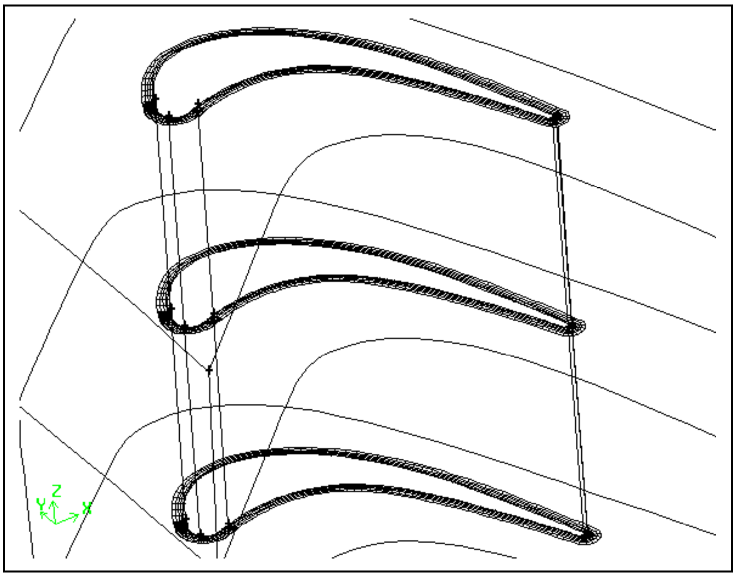
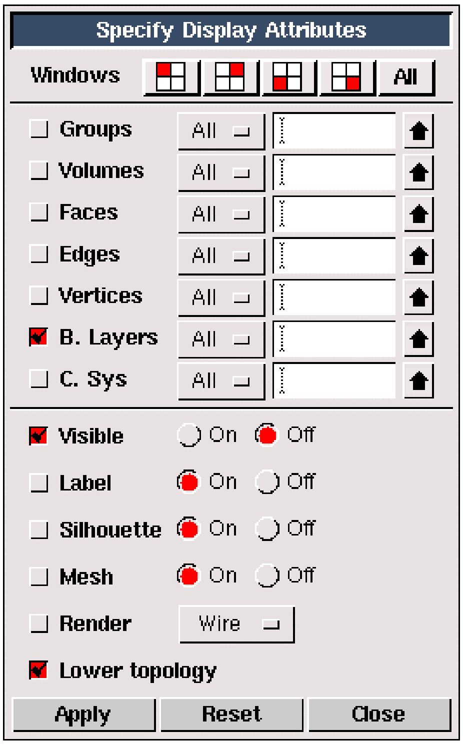


Figure 8-8: Turbo volume with 3-D boundary layers

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.

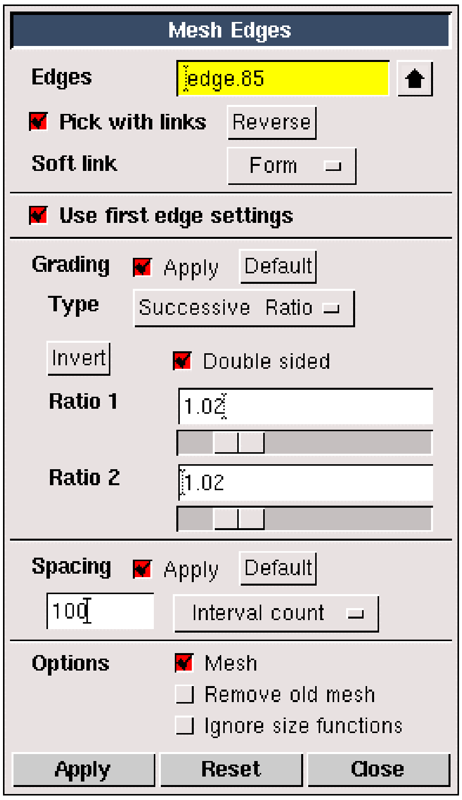
Step 8: Mesh the Blade Cross-Section Edges

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.*

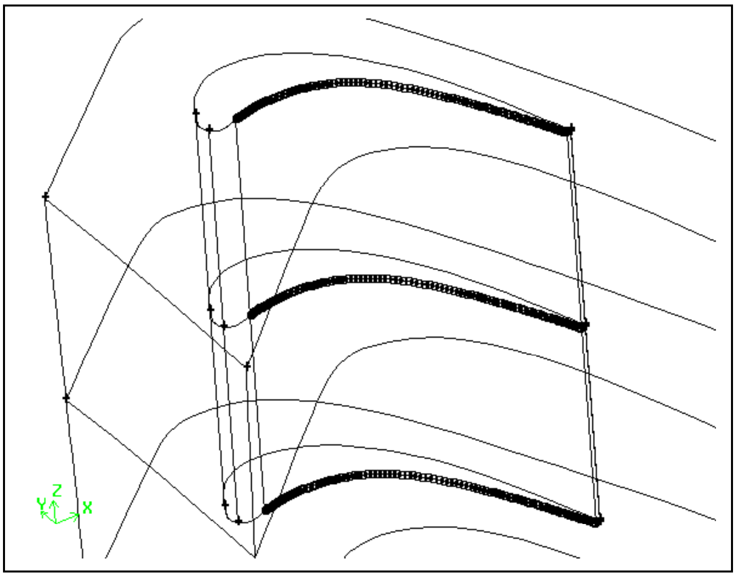


Figure 8-9: Meshed centermost pressure-side edges of the turbo blade

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, 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.

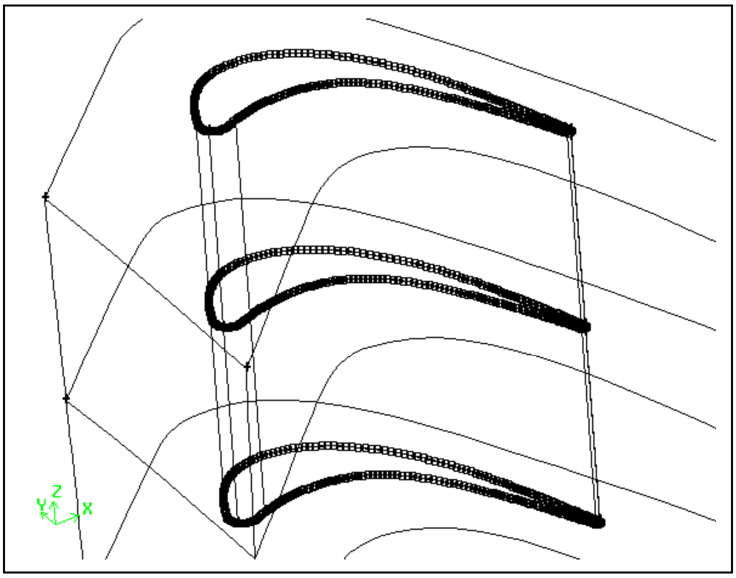


Figure 8-10: Meshed edges of turbo blade cross sections

Step 9: Mesh the Center Spanwise Face

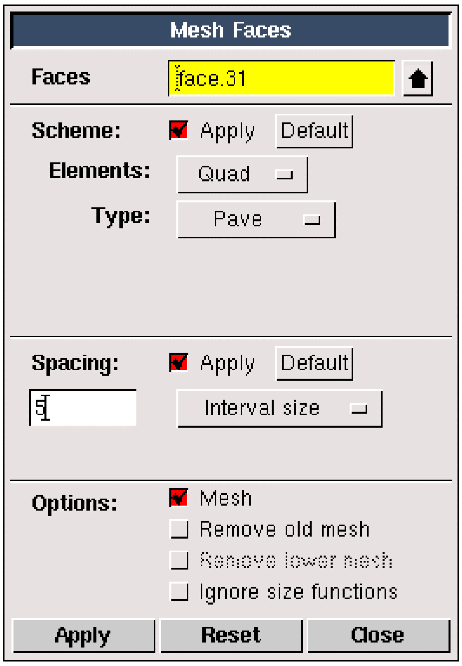
To create an unstructured mesh for this example, it is best to pre-mesh the middle spanwise face and to employ the middle face as a source face for a Cooper *meshing scheme applied to the two geometric volumes. The use of the middle face as a source face ensures that the* Cooper *scheme produces a mesh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

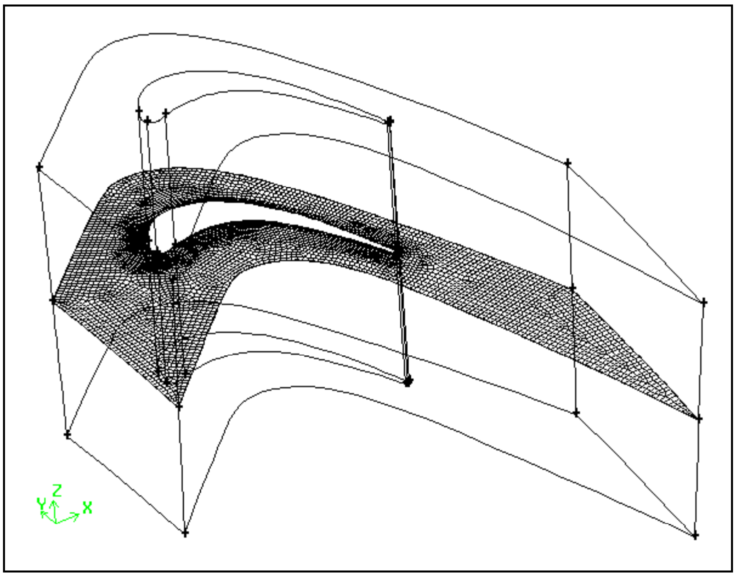


Figure 8-11: Meshed center spanwise face

Step 10: Mesh the Volumes

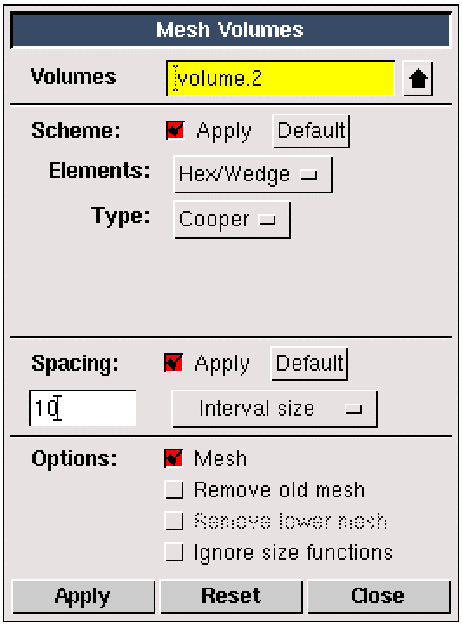
In this step, you will apply a Cooper *meshing scheme to the two geometric volumes that comprise the turbo volume.*

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.*

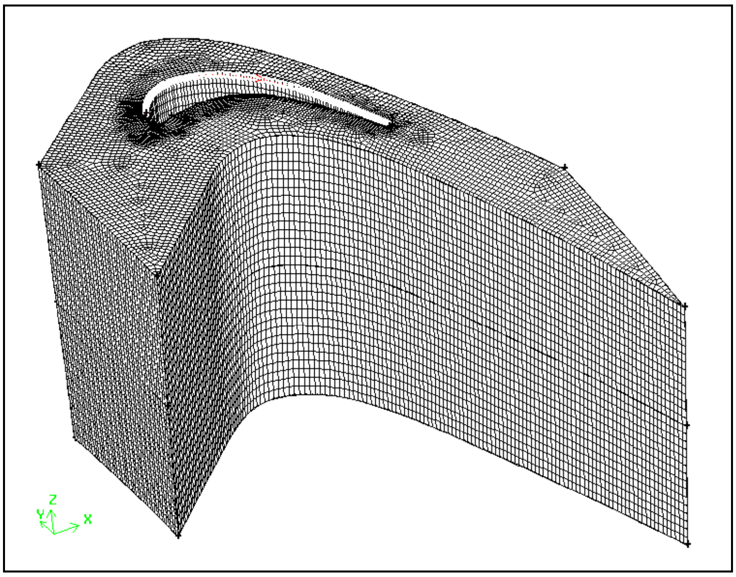
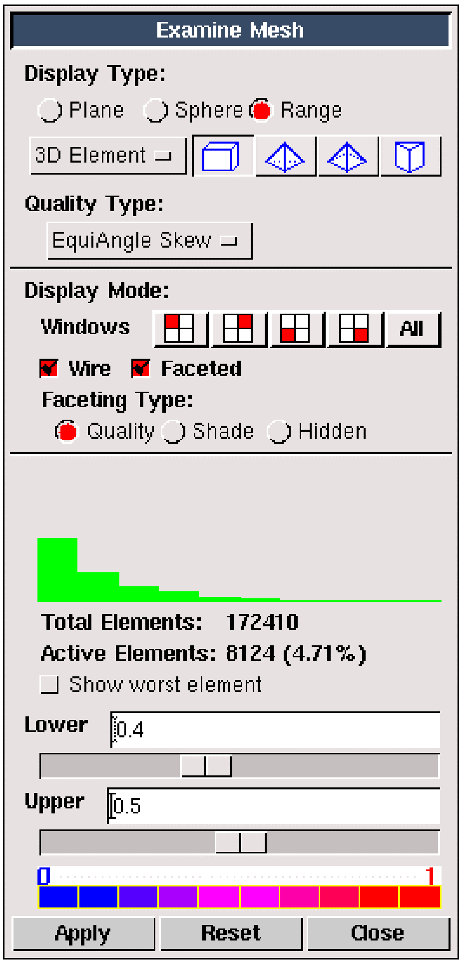


Figure 8-12: Meshed volumes

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* EquiAngle Skew *parameter is between 0.4 and 0.5 for this example.*

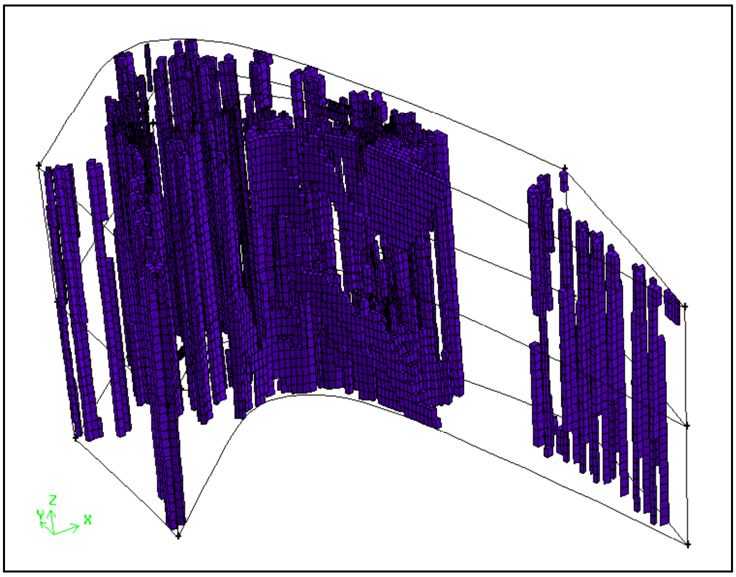


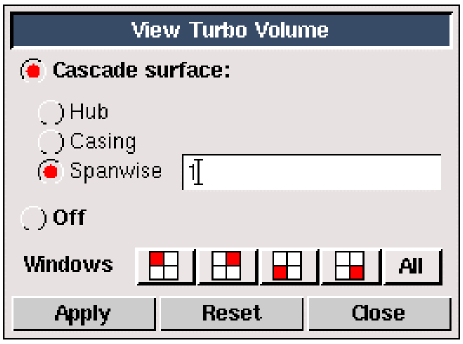
Figure 8-13: Hexahedral mesh elements—**EquiAngle Skew** = 0.4–0.5

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.*

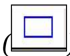


a) Select the **Cascade surface:**Spanwise option.

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 EquiAngle 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.)*

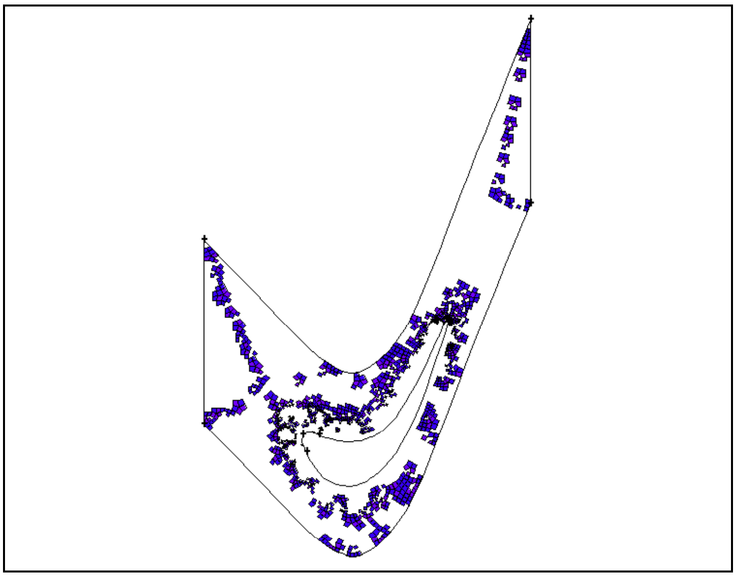


Figure 8-14: Quadrilateral mesh elements—EquiAngle Skew = 0.1–0.3

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

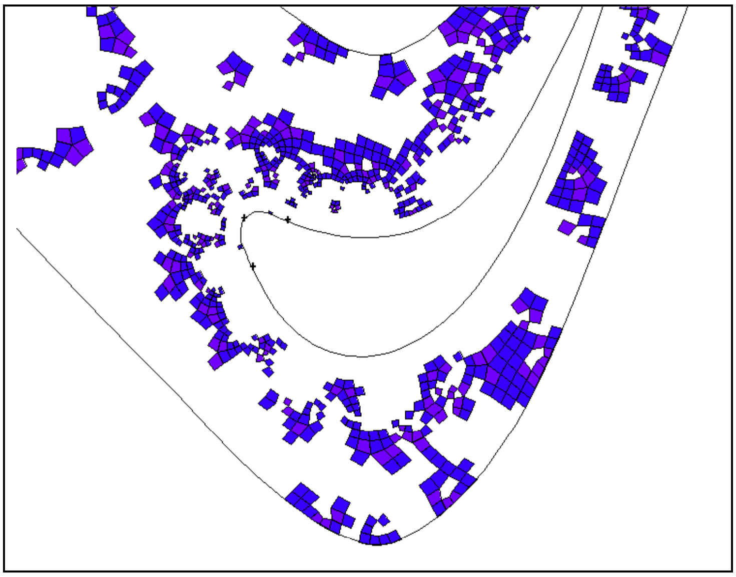


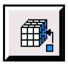
Figure 8-15: Quadrilateral mesh elements—zoomed view near blade tip

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

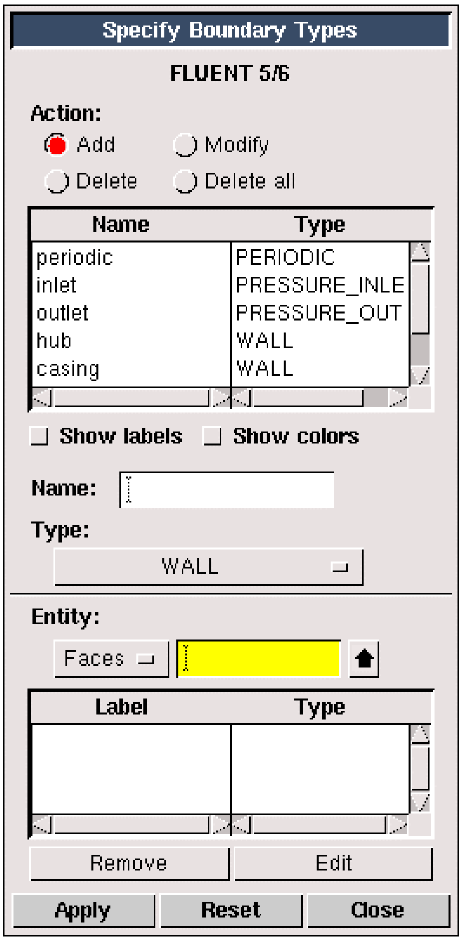
Step 12: Specify Zone Types

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.*



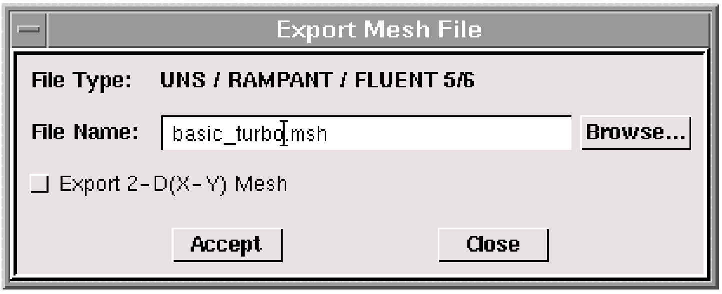
Step 13: Export the Mesh and Exit GAMBIT

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.

9. LOW-SPEED CENTRIFUGAL COMPRESSOR

This tutorial employs the configuration of a low-speed, centrifugal compressor blade to demonstrate the use of imported geometry and the turbo volume decomposition operation. It illustrates how to adjust decomposition split points and employs a structured hexahedral mesh.

In this tutorial, you will learn how to:

- Create a turbo volume based on imported ACIS geometry
- Decompose a turbo volume

9.1 Prerequisites

To understand this tutorial, you should review and understand the steps, principles, and procedures outlined in Tutorials 1, 2, 3, 4, and 8.

9.2 Problem Description

Figure 9-1 shows the turbomachinery configuration to be modeled and meshed in this tutorial. The configuration represents the rotor of a low-speed centrifugal compressor containing 20 identical, highly skewed blades, each of which is spaced equidistant from the others on the rotor hub. The configuration is designed such that the angles of the inlet and outlet flow directions are offset from each other by 90º.

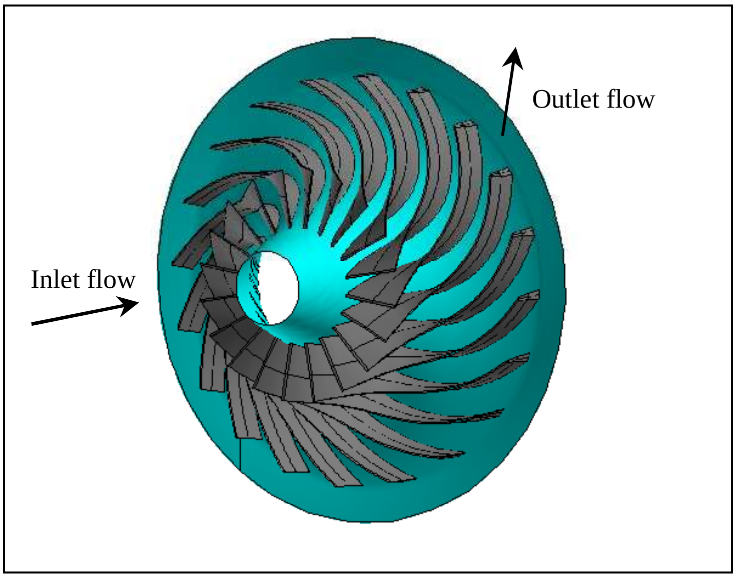


Figure 9-1: Low-speed centrifugal compressor rotor

9.3 Strategy

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 all of the steps listed above. In this example, the edge data that describes the turbo profile is imported from an ACIS file, and edges of the turbo volume are pre-split in the zone-type assignment step (Step 4) to facilitate decomposition (Step 5).

NOTE: In this tutorial, the turbo-volume viewing operation (Step 7, above) is illustrated in conjunction with the mesh examination step (see “Step 10:Examine the Mesh,” below).

9.4 Procedure

1. Copy the file

   path/`Fluent.Inc/gambit2.`*x*`/help/tutfiles/lscc-smooth.sat`

   (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 “`LS_Centrifugal_Comp`”.

Step 1: Select a Solver

1. Choose the solver from the main menu bar:

   Solver $\longrightarrow$ **FLUENT 5/6**

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

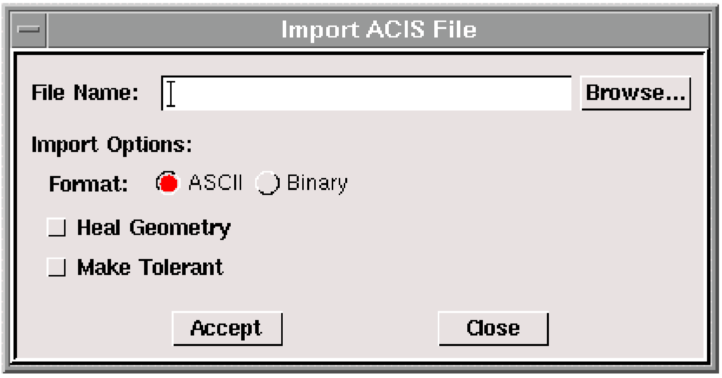
Step 2: Import ACIS Geometry

To create a turbo model, GAMBIT *requires the specification of a set of edges that define the shapes of the turbo hub and casing and the cross-sectional shapes of the turbo blade(s). In this tutorial, the edge specification data is imported from an ACIS file.*

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

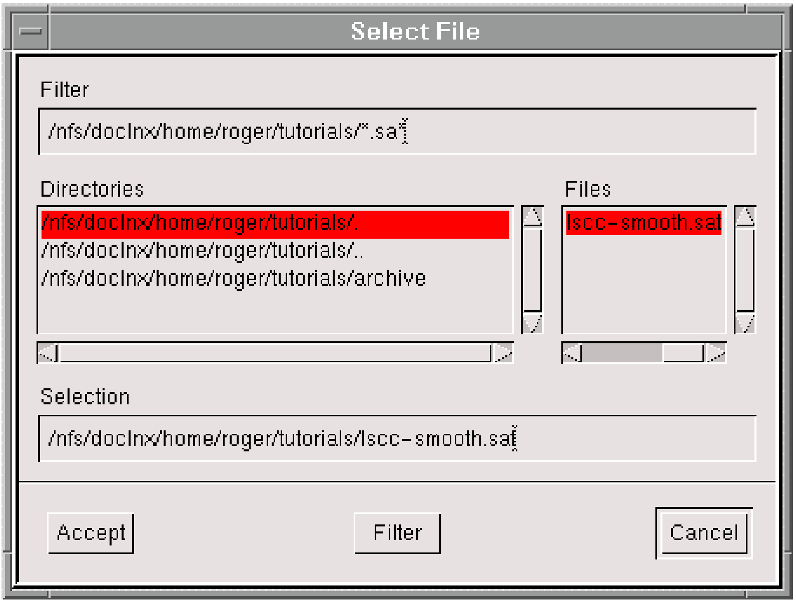
   File $\longrightarrow$ **Import** $\longrightarrow$ **ACIS**

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



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

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



a) In the Files list, select `lscc-smooth.sat`.

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

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

GAMBIT *reads the information contained in the ACIS file and constructs the geometry shown in Figure 9-2.*

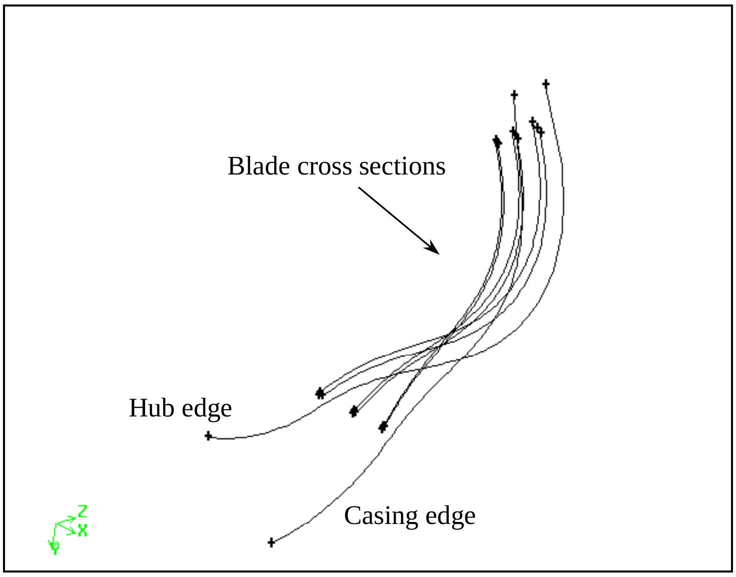


Figure 9-2: Imported ACIS geometry for low-speed centrifugal compressor

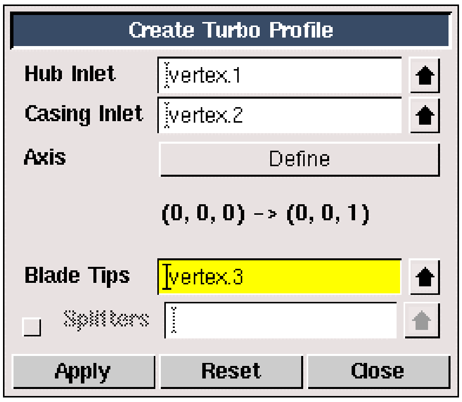
Step 3: Create the Turbo Profile

The turbo profile defines the basic characteristics of the turbo volume. 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 9-3.

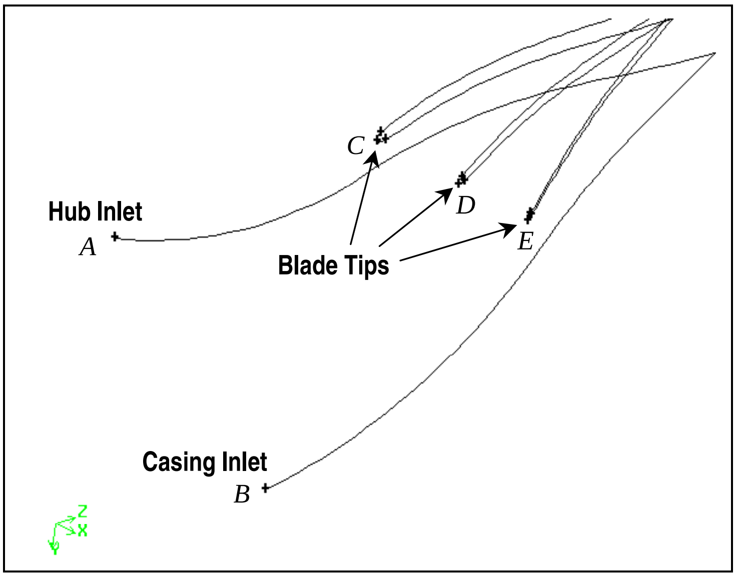


Figure 9-3: Vertices used to specify the turbo profile

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

b) Select vertex *A*.

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

d) Select vertex *B*.

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

f) Select (in order) vertices *C*, *D*, and *E*.

! *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 hub to casing.*

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

GAMBIT *creates the turbo profile shown in Figure 9-4.*

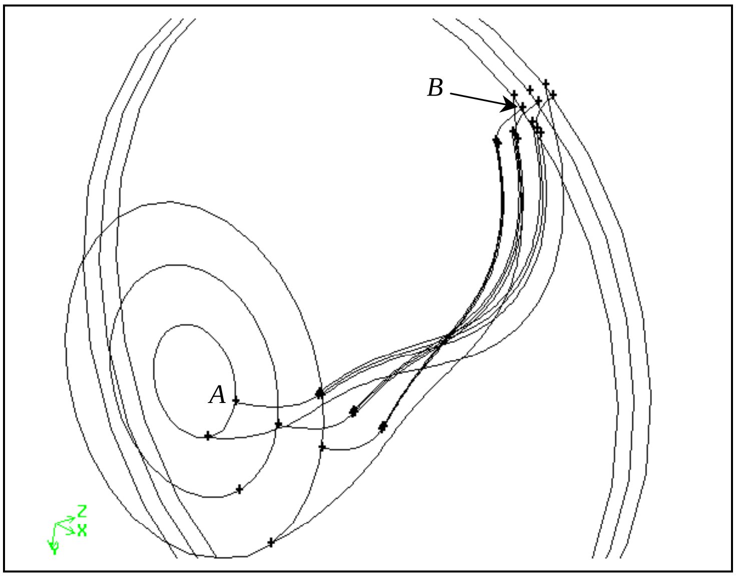


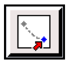
Figure 9-4: Turbo profile for low-speed centrifugal compressor blade

The turbo profile for this tutorial includes six (real) rail edges and three (virtual) medial edges, each of which corresponds to one of the turbo blade cross sections.

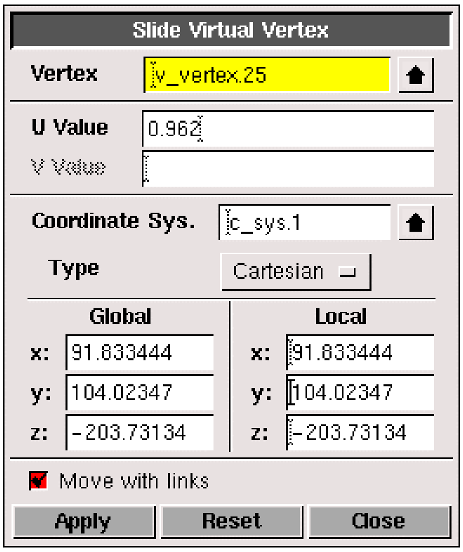
Step 4: Modify the Inlet and Outlet Vertex Locations

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 hub blade cross section (vertex *A* in Figure 9-4, above).

b) In the **U Value** field, enter the value `0.962`.

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 0.962.*

c) Retain the (*default*) Move with links 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 `0.981`.

g) Retain the Move with links 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 9-5.

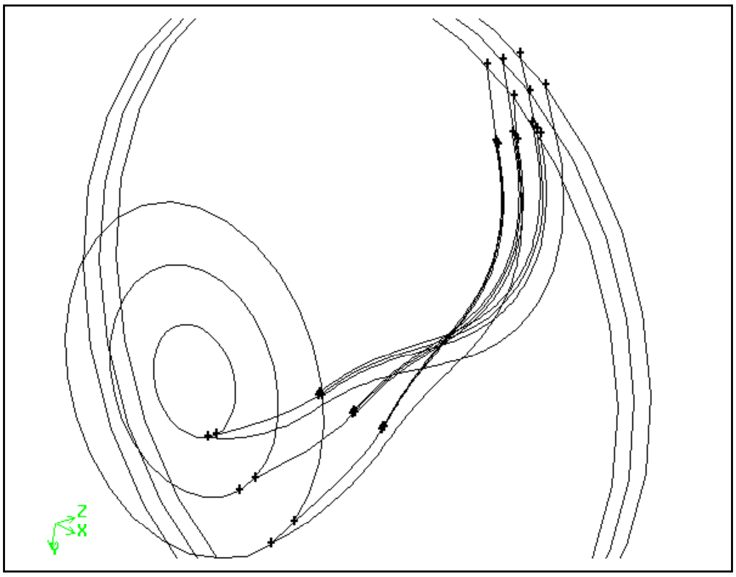


Figure 9-5: Turbo profile with modified inlet and outlet vertex locations

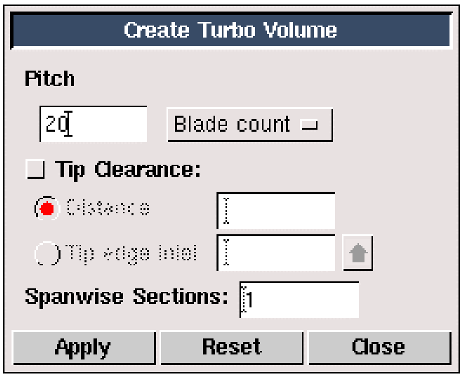
Step 5: Create the Turbo Volume

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 either a tip clearance or spanwise sectioning.

1. Specify the pitch 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 `20`.

   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 `1`.

   d) Click **Apply**.

   Figure 9-6 shows the resulting turbo volume.

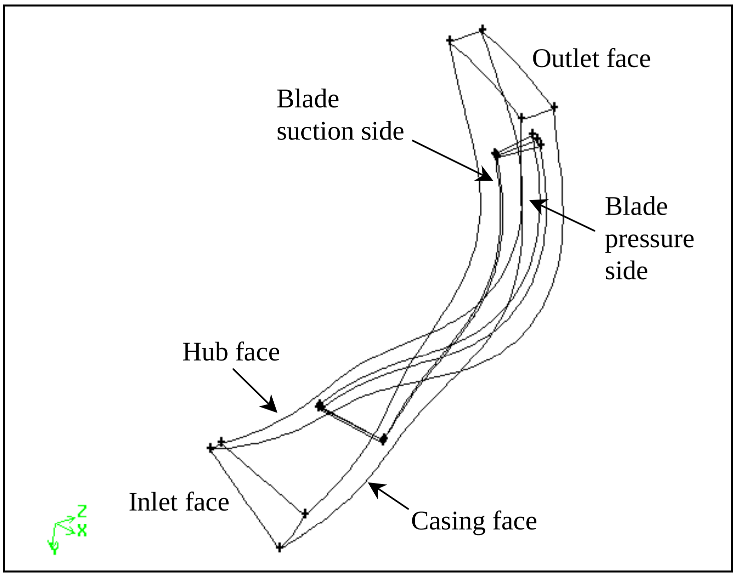


Figure 9-6: Turbo volume for low-speed centrifugal compressor blade

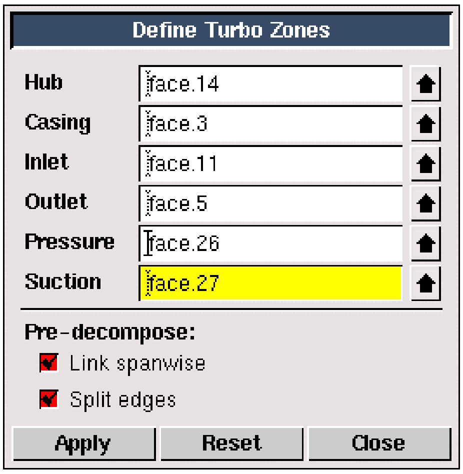
Step 6: Define the Turbo Zones

This step assigns standard zone types to surfaces of the turbo volume. The zone-type specifications determine which faces are linked for meshing. In addition to assigning zone types, this step employs pre-decomposition options that presplit periodic surfaces in order to facilitate turbo volume decomposition (see "Step 8:Decompose the Turbo Volume," below).

1. Specify the faces that constitute the hub, casing, inlet, and 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, and select the bottom (*hub*) face of the turbo volume.

   b) Activate the **Casing** list box, and select the top (*casing*) face of the turbo volume.

   c) Activate the **Inlet** list box, and select the *inlet* face of the turbo volume.

   d) Activate the **Outlet** list box, and select the *outlet* face of the turbo volume.

   e) Activate the **Pressure** list box, and select the *front two* faces (*excluding* the flat, trailing-tip face) on the inner-curve (*pressure* side) of the turbo blade.

   f) Activate the **Suction** list box, and select the *front two* faces (*excluding* the flat, trailing-tip face) on the outer-curve (*suction* side) of the turbo blade.

The flat edges on the trailing tips of the blade cross sections are not included in the definitions of the pressure and suction surfaces; therefore, they will not be merged into their respective surfaces in the decomposition step.

g) In the **Pre-decompose** section, select both the Link spanwise and Split edges options.

The **Pre-decompose** *options specify that* GAMBIT *is to merge the pressure and suction surfaces of the blade, link the spanwise (hub and casing) faces of the turbo volume, and split the periodic edges of the hub and casing faces to facilitate decomposition of the turbo volume. The split locations for the periodic faces are determined by a set of default variables that can be modified by means of the* **Edit Defaults** *form (see Section 4.2.4 in the* GAMBIT *User's Guide).*

h) Click **Apply**.

GAMBIT *assigns the zone types and splits the blade and periodic edges as shown in Figure 9-7.*

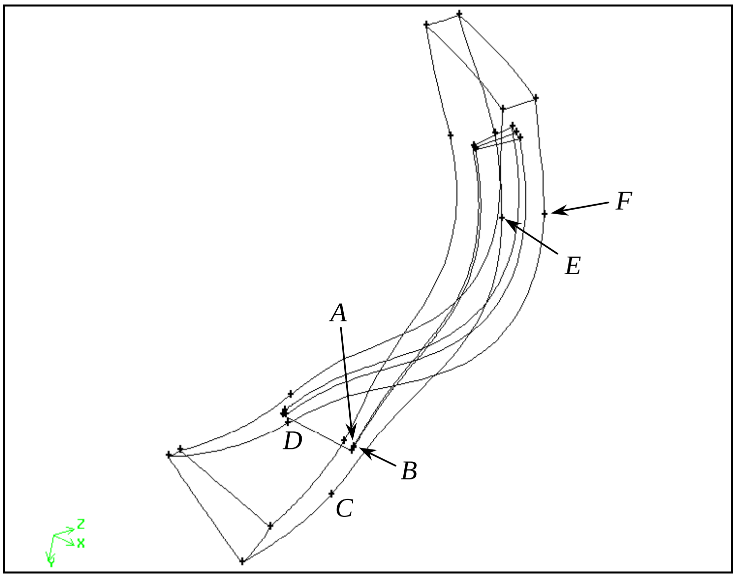


Figure 9-7: Turbo volume with pre-decomposition splits

Because the flat trailing edges are not included in the pressure and suction surface definitions, the sharp edges at the trailing tip of the edge are maintained and are used for the turbo decomposition.

Step 7: Adjust Edge Split Points

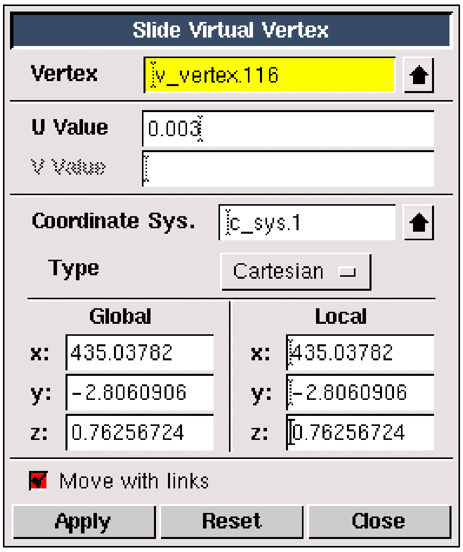
It is often useful to modify the default split-point locations prior to decomposing the turbo volume. Such adjustments can facilitate success of the decomposition operation and the creation of spanwise faces that can be meshed with high-quality elements. You can adjust the split-point locations either before or after decomposition, but the adjustment process is less time-consuming if it is performed prior to decomposition, because it does not involve updating the face and volume configurations associated with each adjustment.

In this step, you will adjust the turbo blade split points such that they are close to, but not coincident with, the leading edge vertex.

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

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

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



   a) Select the suction-side, upstream split-point vertex on the casing face turbo blade cross section (vertex *A* in Figure 9-7, above).

   b) In the **U Value** field, enter the value `0.003`.

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 0.003.*

c) Retain the Move with links option.

The Move with links *option specifies that* GAMBIT *is to apply the current* **Slide Virtual Vertex** *specifications to all linked vertices in addition to the selected vertex. In this case, the suction-side split-point vertex on the casing face turbo blade cross section is linked to a corresponding vertex on the hub face turbo blade cross section.*

d) Click **Apply** to accept the new split-point location.

e) Select the pressure-side, upstream split-point vertex on the casing face turbo blade cross section (vertex *B*).

f) In the **U Value** field, enter the value `0.997`.

g) Click **Apply** to accept the new split-point location.

h) Select the pressure-side, *upstream* split-point vertex on the casing face periodic edge (vertex *C*).

i) Unselect the Move with links option.

Because the leading edge of the blade is swept backwards from hub to casing, it is appropriate to move this vertex independently of the corresponding hub vertex (vertex D). This independent movement is accomplished by unselecting the Move with links *option. (NOTE: In all subsequent* **Slide Virtual Vertex** *operations, the* Move with links *option will remain unselected.)*

j) In the **U Value** field, enter the value `0.238`.

k) Click **Apply** to accept the new split-point location.

l) Select the pressure-side, *upstream* split-point vertex on the hub face periodic edge (vertex *D*).

m) In the **U Value** field, enter the value `0.812`.

n) Click **Apply** to accept the new split-point location.

o) Select the pressure-side, *downstream* split-point vertex on the casing face periodic edge (vertex *E*).

p) In the **U Value** field, enter the value `0.812`.

q) Click **Apply** to accept the new split-point location.

r) Select the pressure-side, *downstream* split-point vertex on the hub face periodic edge (vertex *F*).

s) In the **U Value** field, enter the value `0.156`.

t) Click **Apply** to accept the new split-point location.

Figure 9-8 shows the turbo volume configuration with the adjusted split points.

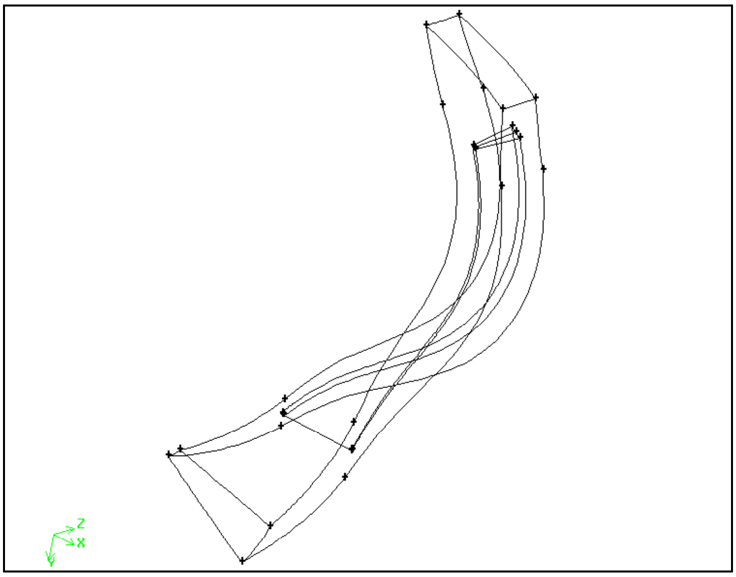


Figure 9-8: Turbo volume with adjusted split points

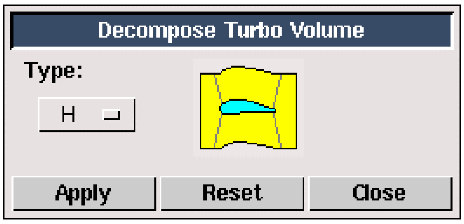
Step 8: Decompose the Turbo Volume

The decomposition step splits the turbo volume into four geometric volumes the topologies of which are suitable for the creation of structured hexahedral meshes.

1. Decompose the turbo volume.

   TOOLS  → **TURBO**  → **DECOMPOSE TURBO VOLUME** 

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



   a) Retain the (*default*) **Type:**H option, and click **Apply**.

   GAMBIT *decomposes the volume as shown in Figure 9-9.*

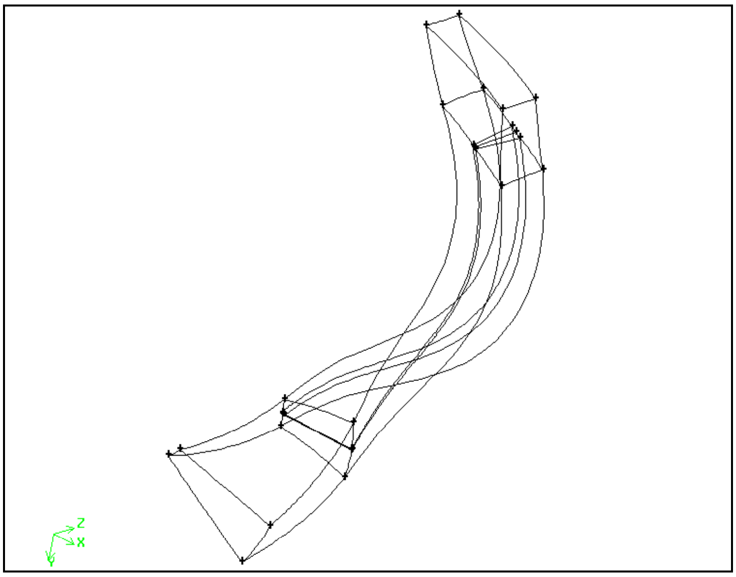


Figure 9-9: Decomposed turbo volume for low-speed centrifugal compressor

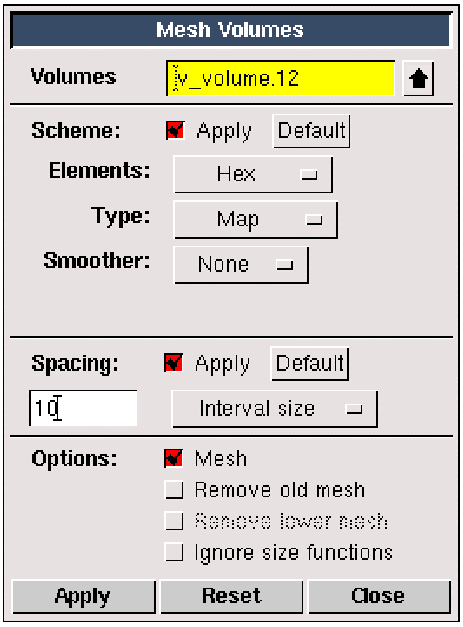
Step 9: Mesh the Volumes

The decomposition step (above) automatically sets the interval count and grading on the edges according to the turbo decomposition defaults. In addition, the decomposition sets face vertex types so that the volume is ready to mesh.

1. Mesh all of the volumes.

TOOLS → **TURBO** → **MESH EDGES/FACES/VOLUMES**
R Mesh Volumes

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



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

b) Select all four volumes.

GAMBIT *automatically selects the* **Scheme:Elements:**Hex *and* **Scheme:Type:** Map *options.*

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

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

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

f) Click **Apply**.

Figure 9-10 shows the final meshed turbo volume.

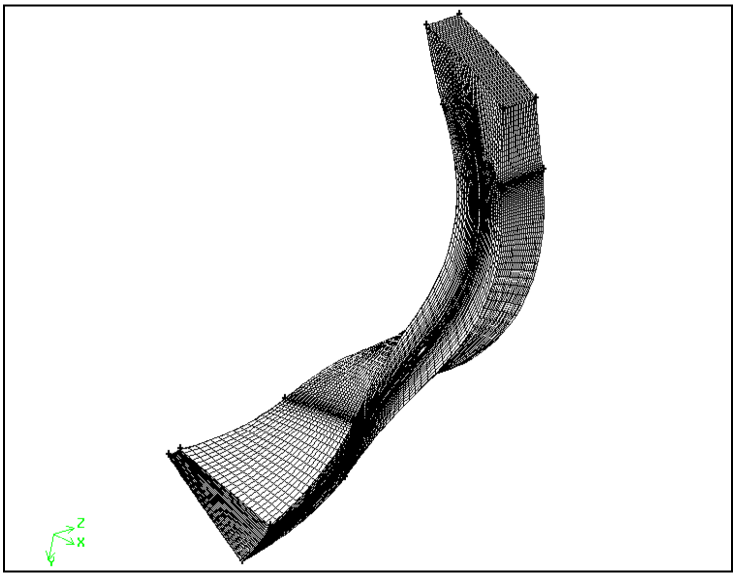
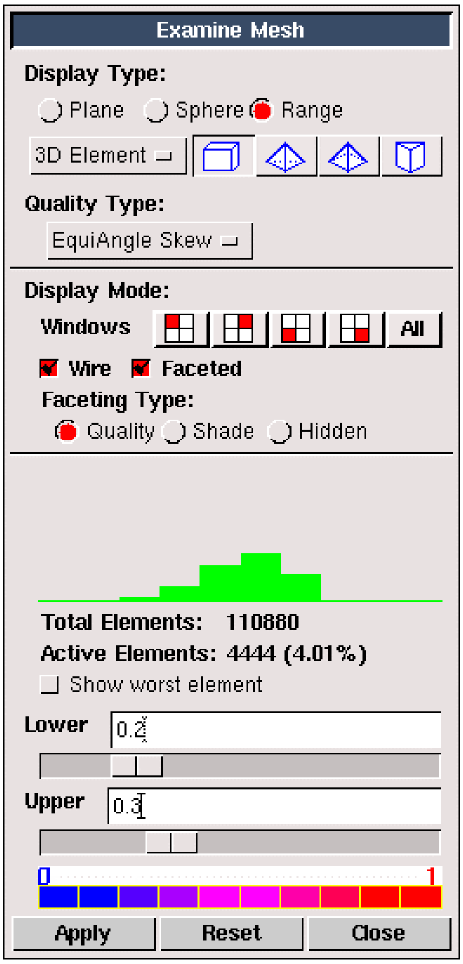


Figure 9-10: Meshed turbo volume for low-speed centrifugal compressor

Step 10: 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 9-11 displays hexahedral volume mesh elements for which the* EquiAngle Skew *parameter is between 0.2 and 0.3 for this example.*

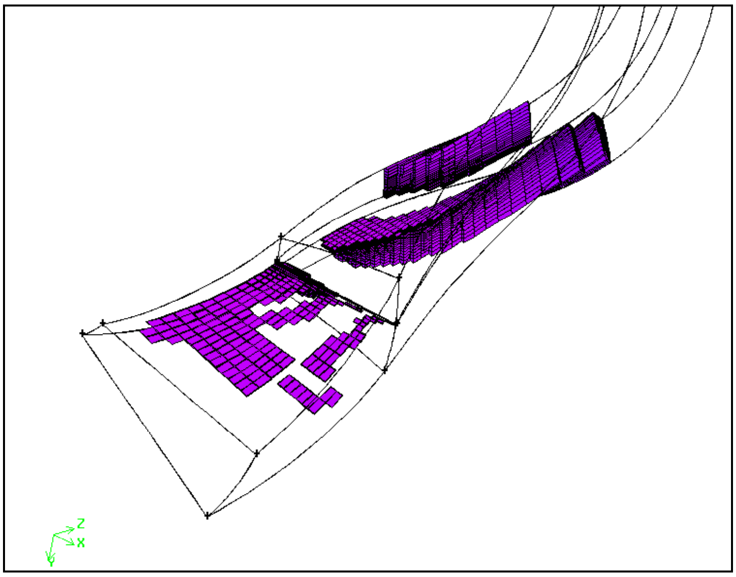
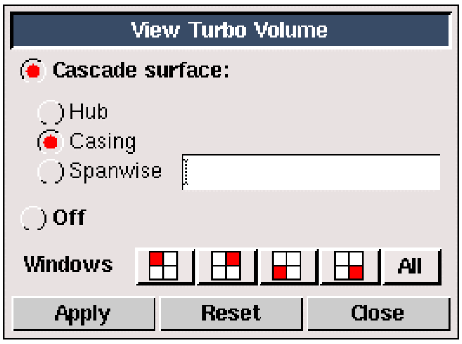


Figure 9-11: Hexahedral mesh elements—EquiAngle Skew = 0.2–0.3

2. Display the casing surface in a cascade turbo view.

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

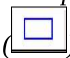
This command sequence opens the **View Turbo Volume** *form.*



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

The **Cascade surface** *specifications described above specify a flattened, 2-D display of the casing surface.*

b) Click **Apply**.

Figure 9-12 displays an enlarged view of the quadrilateral face mesh elements near the blade tip on the casing surface for this example. In this case, the mesh elements are colored to represent the value of the EquiAngle Skew *parameter. (<u>NOTE</u>: To view the 2-D face elements shown in Figure 9-12, select the* **Display Type:** 2D Element *option on the* **Examine Mesh** *form, and specify the display of quadrilateral (**) elements.)*

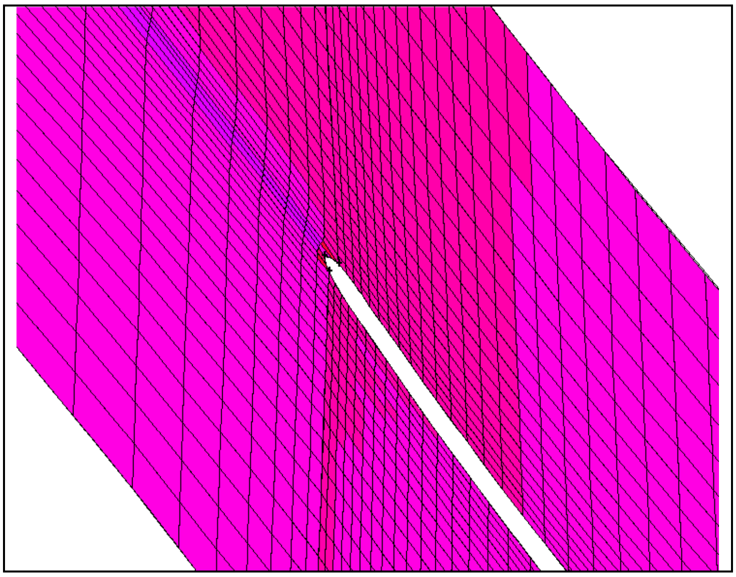


Figure 9-12: Quadrilateral mesh elements near blade tip—EquiAngle Skew = 0–1

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

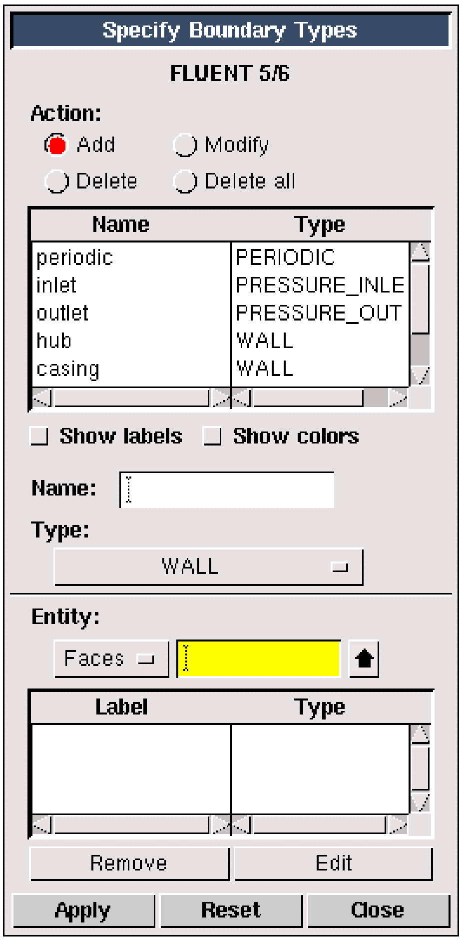
Step 11: Specify Zone Types

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.*



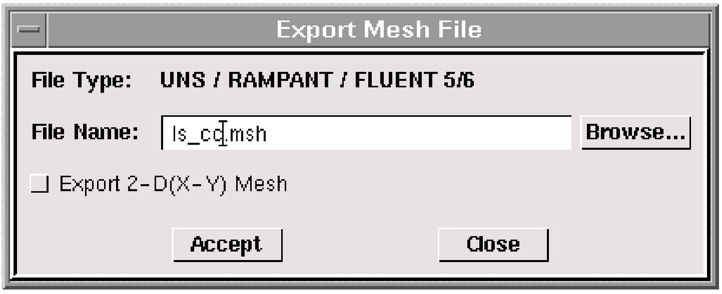
Step 12: Export the Mesh and Exit GAMBIT

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, "`ls_cc.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.

9.5 Summary

This tutorial demonstrates the use of ACIS geometry import and turbo decomposition operations in GAMBIT turbo modeling. In this example, edge data imported from an ACIS file were used to define a turbo profile, which, in turn, was used to create a turbo volume representing the flow region surrounding one blade of a low-speed centrifugal compressor. The turbo zones were assigned, the turbo volume was pre-split, and the split-point locations on the blade and periodic edges were adjusted to facilitate decomposition and meshing. The final, decomposed turbo volume consisted of four volumes, each of which could be meshed using a structured, hexahedral meshing scheme.

10. MIXED-FLOW PUMP IMPELLER

This tutorial employs the configuration of a mixed-flow pump impeller to demonstrate the use of hybrid hexahedral/tetrahedral meshing capabilities in conjunction with GAMBIT turbo modeling. Such capabilities are particularly useful for meshing turbo models that involve highly twisted blades.

10.1 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, 4, and 8.

10.2 Problem Description

Figure 10-1 shows the turbomachinery configuration to be modeled and meshed in this tutorial. The configuration consists of an impeller rotor on which are affixed five identical blades, each of which is spaced equidistant from the others on the rotor hub.

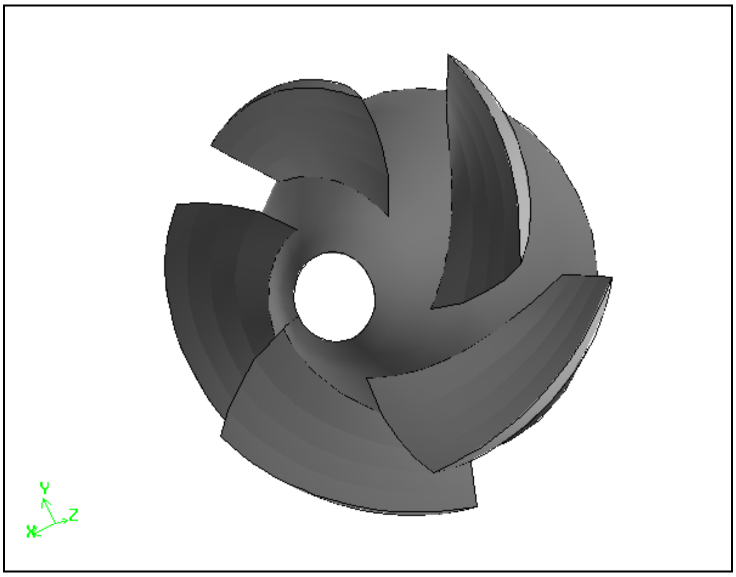


Figure 10-1: Mixed-flow impeller rotor

The overall goal of this tutorial is to create a geometric model of the flow region immediately surrounding one of the impeller blades and to mesh the model using hybrid hexahedral/tetrahedral mesh.

10.3 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 bulk of the turbo volume is to be meshed using unstructured tetrahedral 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 10:Examine the Mesh,” below).

10.4 Procedure

1. Copy the file

   `path/Fluent.Inc/gambit2.x/help/tutfiles/rotor-cyl-mod.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 “`Pump_Impeller`”.

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 below). For some systems,* **FLUENT 5/6** *is the default solver. The currently selected solver is shown at the top of the* GAMBIT *GUI.*

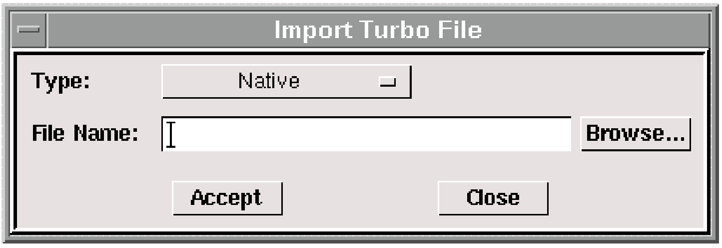
Step 2: Import a Turbo Data File

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.

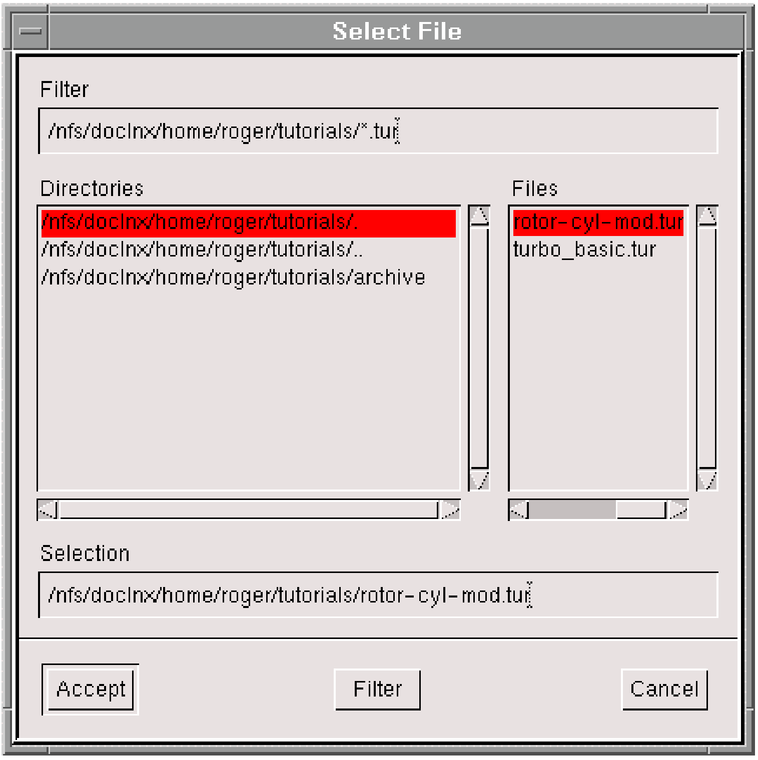
   File → **Import** → **Turbo...**

   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 `rotor-cyl-mod.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 10-2. The two straight edges shown in the figure describe the hub and casing for the turbo volume. The five sets of curved edges constitute cross sections of a single impeller blade.*

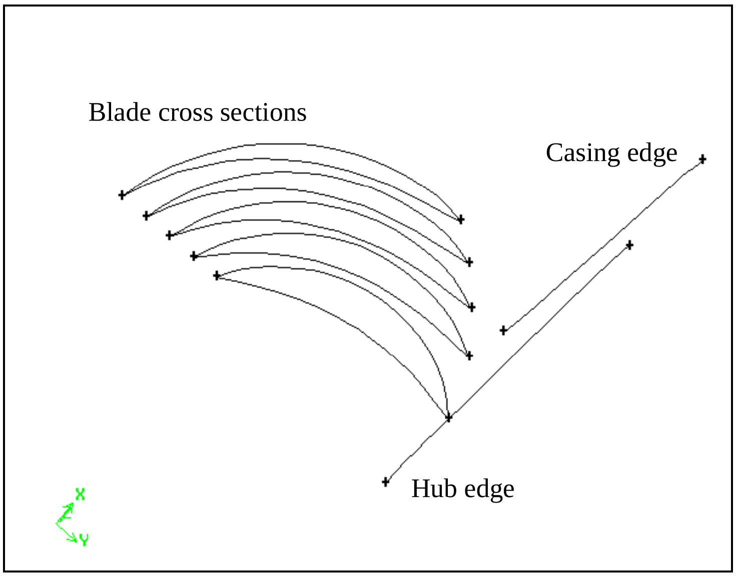


Figure 10-2: Imported impeller geometry

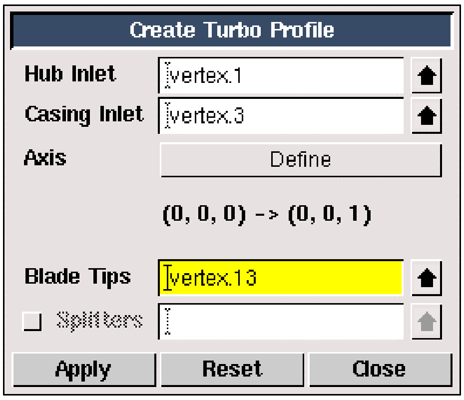
Step 3: Create the Turbo Profile

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. All instructions listed in this step refer to the vertex designations shown in Figure 10-3.

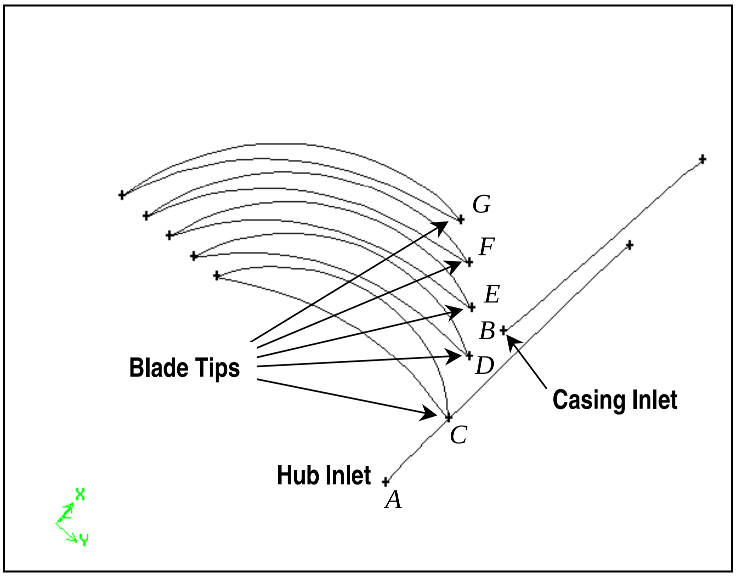


Figure 10-3: Vertices used to specify the turbo profile

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

f) Select (in order) the following vertices: *C*, *D*, *E*, *F*, and *G*.

! *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 to the casing.*

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

GAMBIT *creates the turbo profile shown in Figure 10-4.*

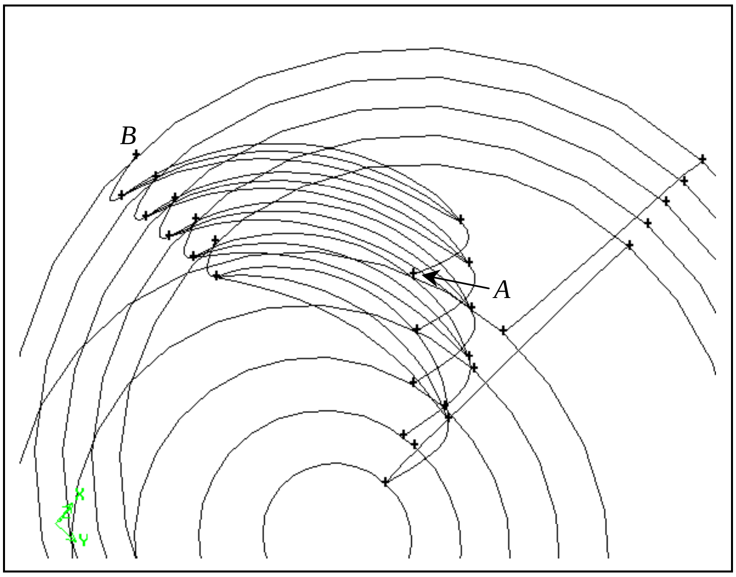


Figure 10-4: Turbo profile

The turbo profile for this tutorial includes 10 (real) rail edges and five (virtual) medial edges, each of which corresponds to one of the blade cross sections.

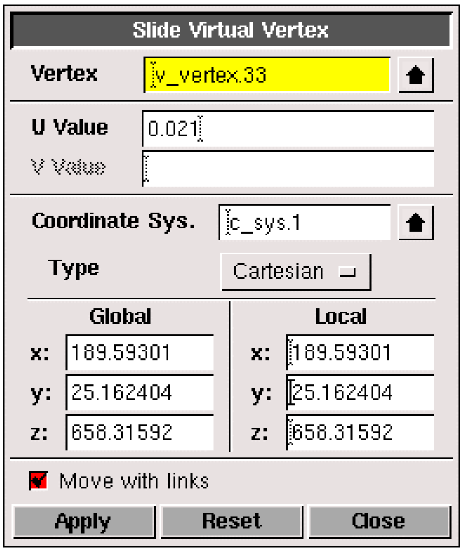
Step 4: Modify the Inlet and Outlet Vertex Locations

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 upper blade cross section (vertex *A* in Figure 10-4, above).

b) In the **U Value** field, enter the value `0.021`.

   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 0.021.*

c) Retain the (*default*) Move with links option.

The Move with links *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 upper blade cross section (vertex *B*).

f) In the **U Value** field, enter the value `0.703`.

g) Retain the Move with links 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 10-5.

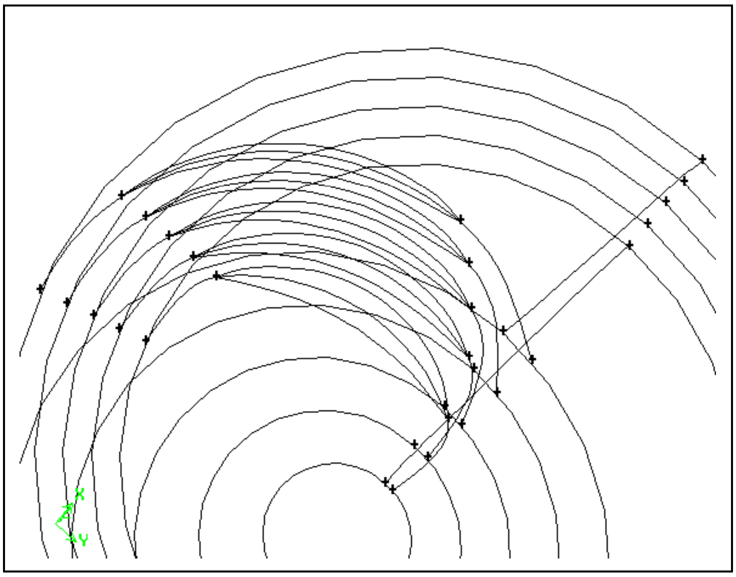


Figure 10-5: Turbo profile with modified inlet and outlet vertex locations

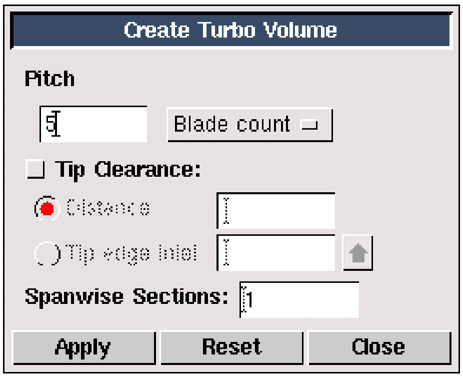
Step 5: Create the Turbo Volume

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 either a tip clearance or spanwise sectioning.

1. Specify the pitch 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 `5`.

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 `1`.

d) Click **Apply**.

GAMBIT *creates the turbo volume shown in Figure 10-6.*

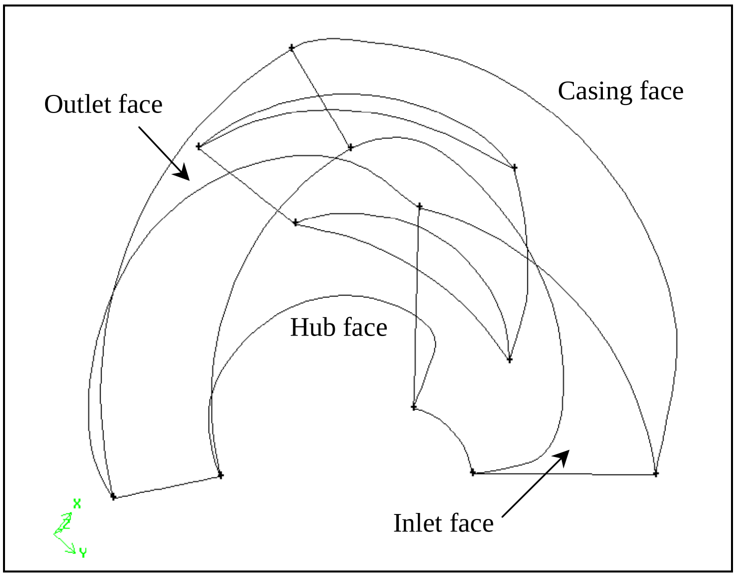


Figure 10-6: Turbo volume for mixed-flow impeller blade

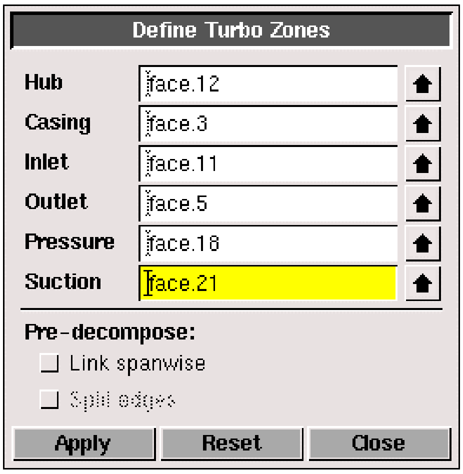
Step 6: Define the Turbo Zones

This step standard zone types to surfaces of the turbo volume. The zone-type definitions 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, and select the bottom (*hub*) face of the turbo volume.

   b) Activate the **Casing** list box, and select the top (*casing*) face of the turbo volume.

   c) Activate the **Inlet** list box, and select the *inlet* face.

   d) Activate the **Outlet** list box, and select the *outlet* face.

   e) Activate the **Pressure** list box, and select the inner-curve (*pressure* side) face of the turbo blade.

   f) Activate the **Suction** list box, and select the outer-curve (*suction* side) face of the turbo blade.

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

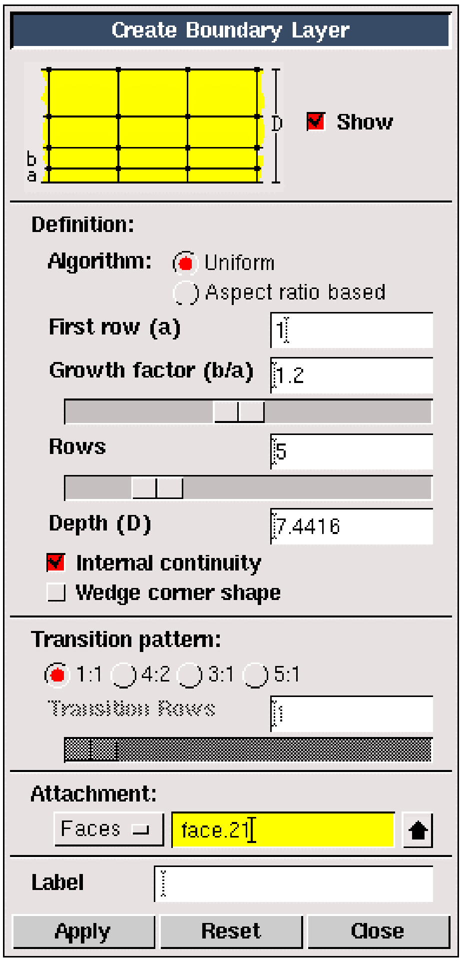
Step 7: Apply 3-D Boundary Layers

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. Apply boundary layers to the faces of the turbo blade.

   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 pressure and suction faces on the turbo blade.

g) Click **Apply**.

Figure 10-7 shows the 3-D boundary layers projected onto the hub and casing surfaces of the turbo volume.

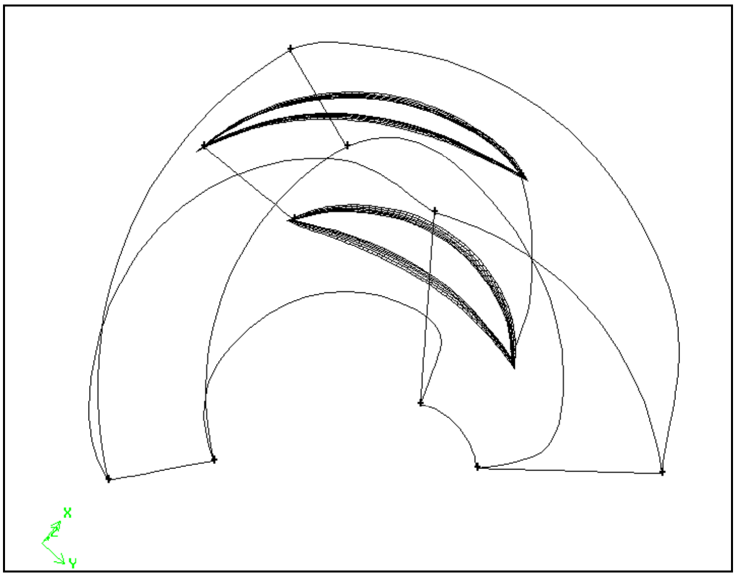
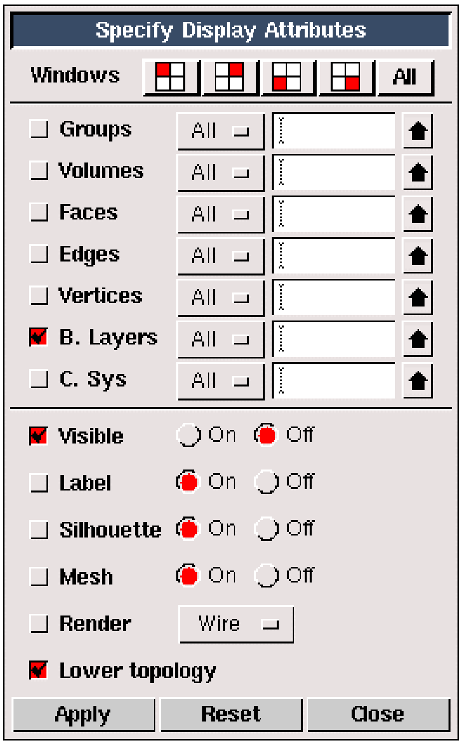


Figure 10-7: Turbo volume with 3-D boundary layers

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**.

   d) Click **Close** to close the **Specify Display Attributes** form.

Step 8: Mesh the Pressure and Suction Faces

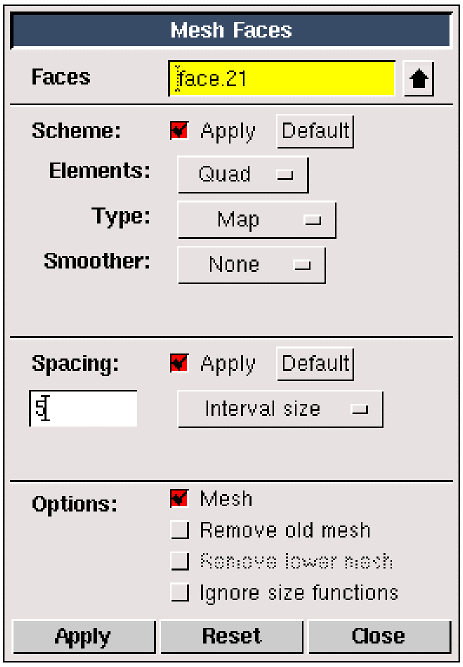
To grow hexahedral cells from the blade surfaces, it is necessary to pre-mesh them using a Quad Map *scheme.*

1. Mesh the pressure and suction surfaces of the turbo blade.

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

R 

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



a) Activate the **Faces** list box, and select the pressure and suction faces on the turbo blade.

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

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

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

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

e) Click **Apply**.

GAMBIT *meshes the pressure and suction faces as shown in Figure 10-8.*

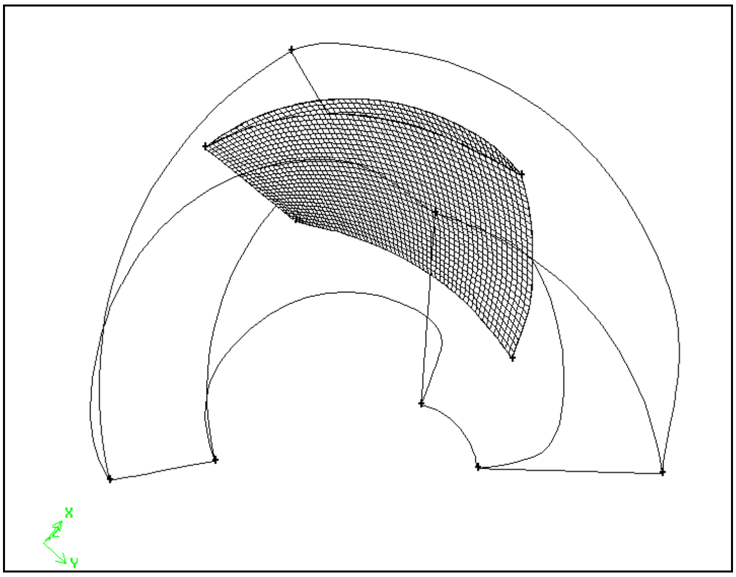


Figure 10-8: Meshed faces of the impeller blade

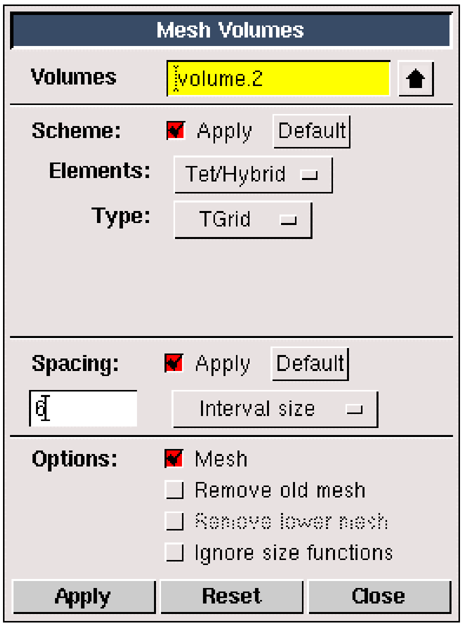
Step 9: Mesh the Volume

In this step, you will mesh the turbo volume using a hybrid scheme that employs hexahedral elements near the blade surface and tetrahedral elements in the bulk of the 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 geometric volume that comprises the turbo volume.

   b) On the **Scheme:Elements** option button, select Tet/Hybrid.

   c) On the **Scheme:Type** option button, select Tgrid.

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

e) In the **Spacing** text box, enter a value of `6`.

f) Click **Apply**.

GAMBIT *meshes the volume as shown in Figure 10-9.*

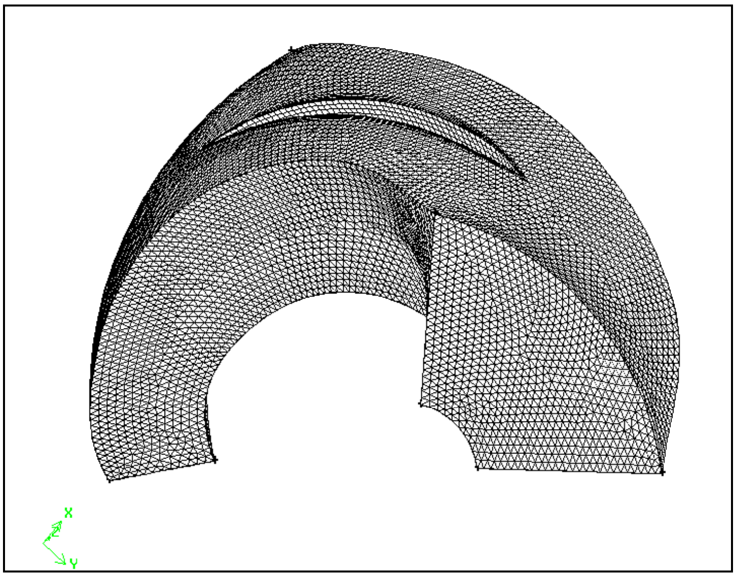
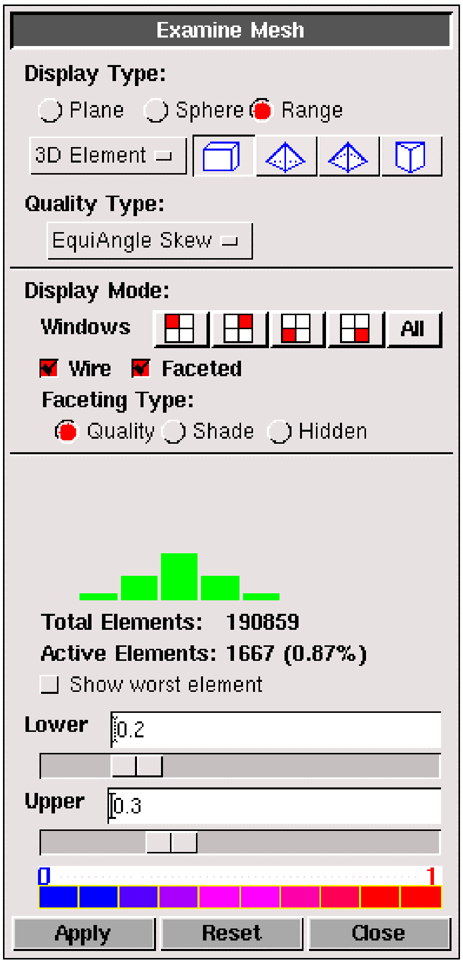


Figure 10-9: Meshed turbo volume for mixed-flow impeller blade

Step 10: 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 10-10, Figure 10-11, and Figure 10-12 display hexahedral, pyramidal, and tetrahedral volume mesh elements, respectively, for which the* EquiAngle Skew *parameter is between 0.2 and 0.3. In this case, the hexahedral elements (Figure 10-10) result from the imposition of the 3-D boundary layer and are confined to the region immediately adjacent to the impeller blade. The pyramidal elements (Figure 10-11) constitute a single transition layer between the hexahedral and tetrahedral (Figure 10-12) elements.*

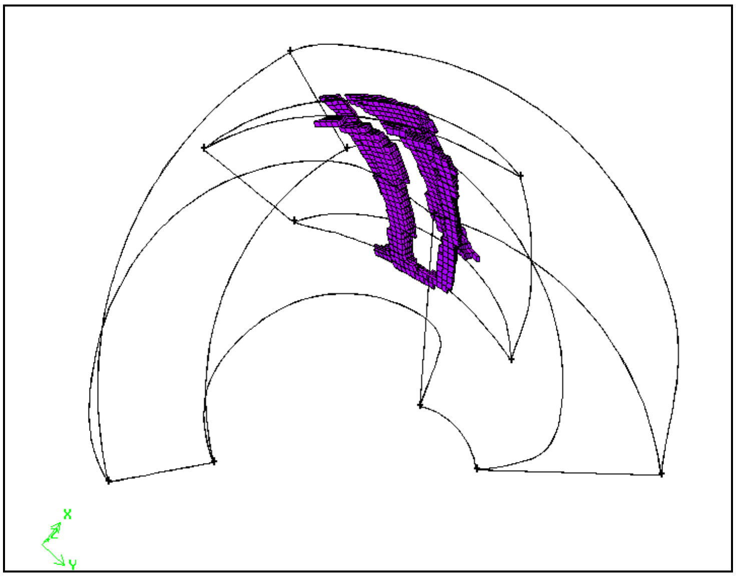


Figure 10-10: Hexahedral mesh elements—EquiAngle Skew = 0.2–0.3

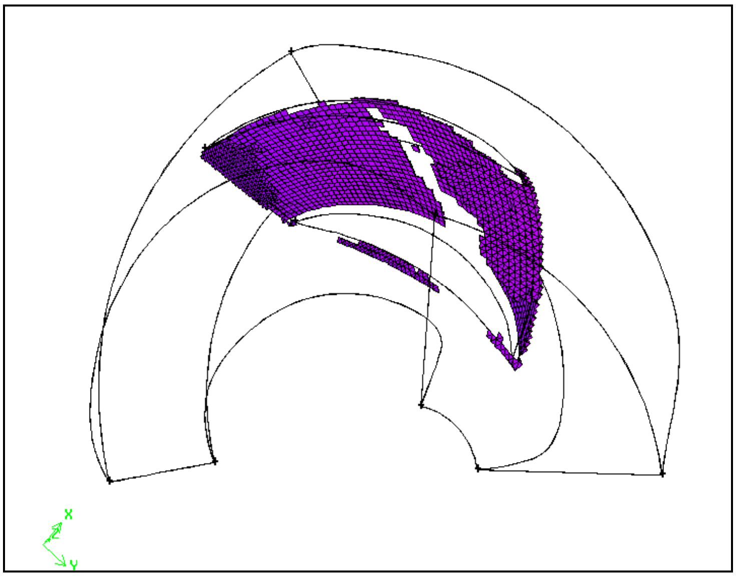


Figure 10-11: Pyramidal mesh elements—EquiAngle Skew = 0.2–0.3

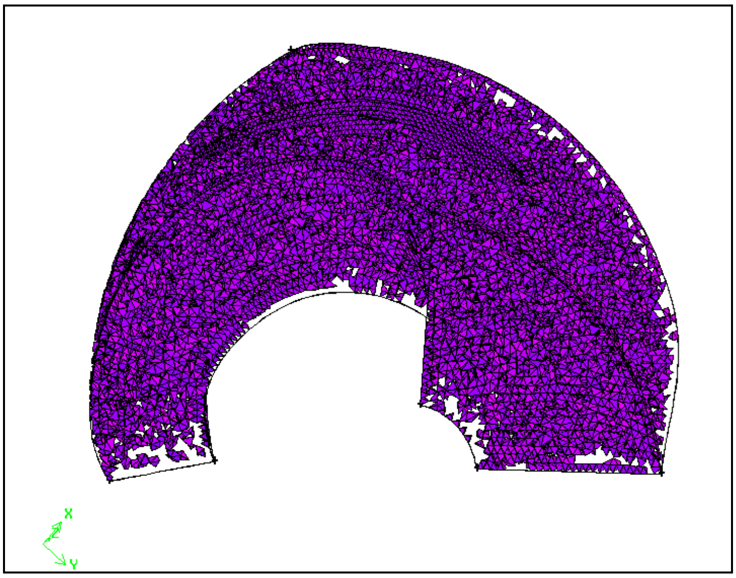


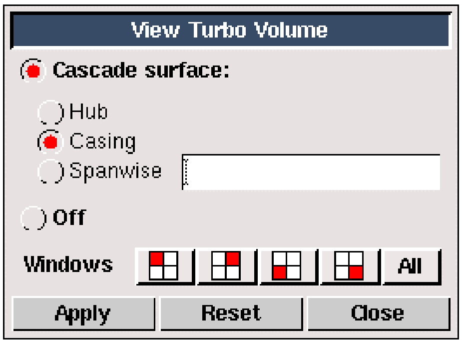
Figure 10-12: Tetrahedral mesh elements—EquiAngle Skew = 0.2–0.3

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 and casing surfaces. Such views are particularly useful when examining the mesh on highly twisted blades.*

2. Display the casing surface in a cascade turbo view.

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

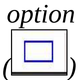
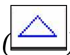
This command sequence opens the **View Turbo Volume** *form.*



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

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

b) Click **Apply**.

Figure 10-13 displays an enlarged view of casing-surface face mesh elements for which the EquiAngle Skew *parameter is between 0.1 and 0.6 in the region surrounding the impeller outlet tip. (NOTE: To view the 2-D face elements shown in Figure 10-13, select the* **Display Type:** 2D Element *option on the* **Examine Mesh** *form, and specify the display of both quadrilateral (**) and triangular (**) elements.)*

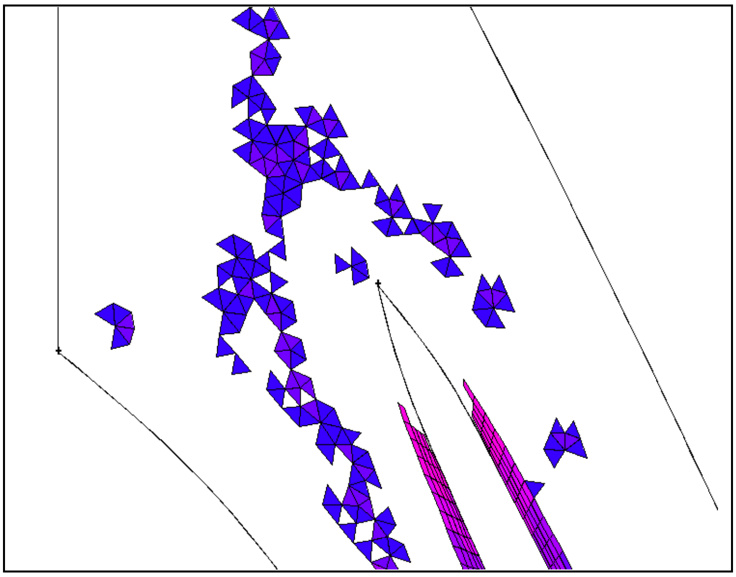


Figure 10-13: Casing-surface face mesh elements—**EquiAngle Skew** = 0.1–0.6

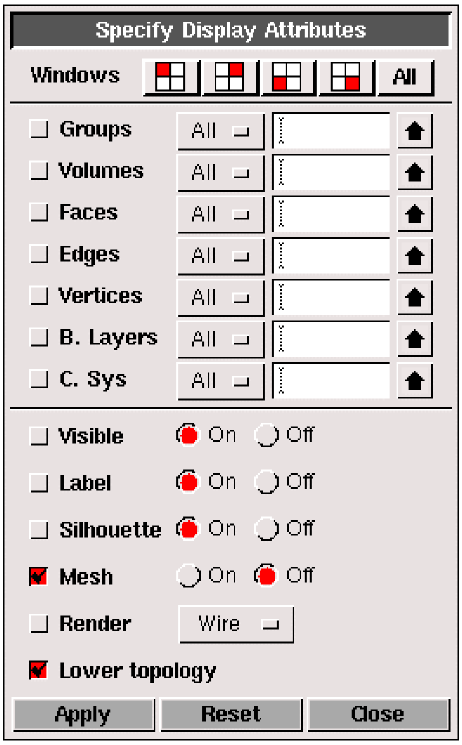
c) Select the **Off** option and click **Apply** to turn off the cascade turbo view before specifying zone types.

Step 11: Specify or Check Zone Types

For some **Solver** *options, including the* **Fluent 5/6** *option used in this tutorial,* GAMBIT *automatically assigns boundary zone specifications to the turbo volume faces when you define the turbo zones (see Step 6: Define the Turbo Zones). You can check such specifications and/or apply solver-specific boundary specifications (for cases in which they are not automatically applied) by means of the* **Specify Boundary Types** *form. It is useful to turn off the mesh display before checking and/or applying the boundary zone specifications.*

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

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



   a) Select the **Mesh:**Off option.

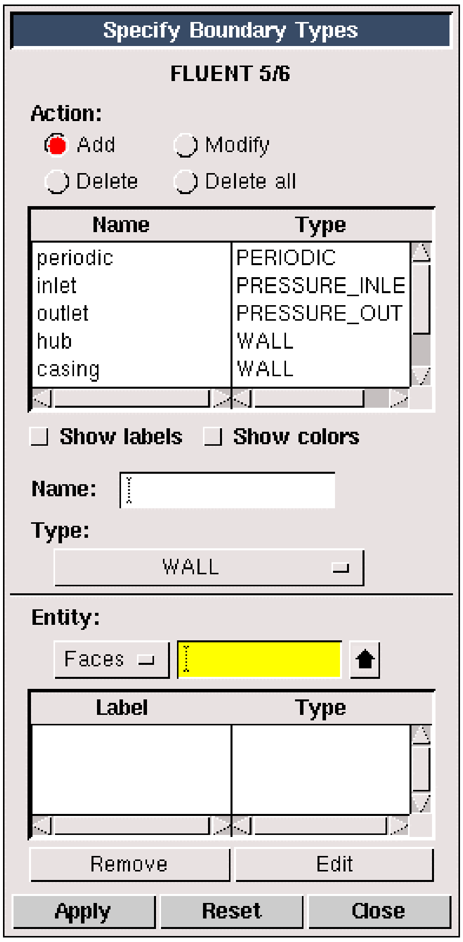
   b) Click **Apply**.

c) Click **Close** to close the **Specify Display Attributes** form.

2. Check the automatically applied boundary zone types.

ZONES  → **SPECIFY BOUNDARY TYPES** 

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



If you select the **Show labels** *option,* GAMBIT *displays labels in the graphics window for all of the assigned boundary zones. If you select the* **Show colors** *option,* GAMBIT *shades and colors the faces for which boundary zones have been defined.*

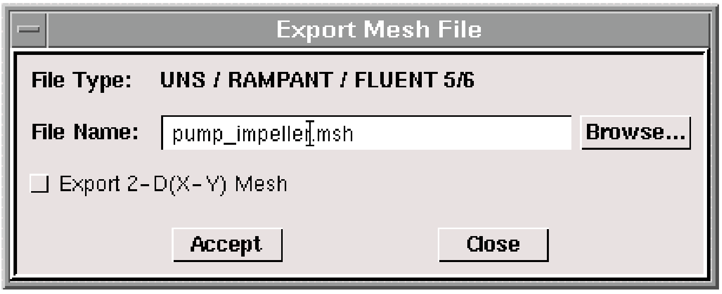
Step 12: Export the Mesh and Exit GAMBIT

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 "`pump_impeller.msh`".

   ii. Click **Accept**.

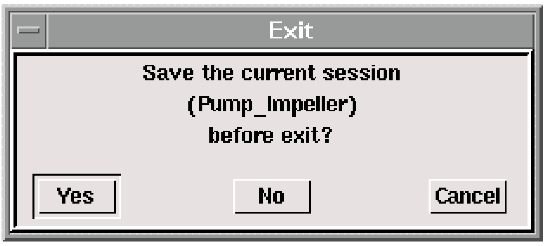
   GAMBIT *writes the mesh file to your working directory.*

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.

10.5 Summary

This tutorial demonstrates the use of the GAMBIT turbo modeling operations as applied to a mixed-flow pump impeller. In this case, 3-D boundary layers were applied to the impeller faces, and the bulk of the turbo volume was meshed using tetrahedral elements. As a result, the meshed turbo volume contained three volume element types: hexahedral (in the region adjacent to the impeller blade), pyramidal (a single transition layer), and tetrahedral (in the bulk of the turbo volume).

11. INDUSTRIAL DRILL BIT—STEP GEOMETRY

This tutorial employs a model of an industrial drill bit to illustrate GAMBIT operations that allow you to import and clean up STEP geometry, modify the model to suppress features that can adversely affect meshing, and control mesh quality. The STEP data file that describes the drill-bit geometry used in this tutorial was created by means of an external CAD package.

In this tutorial, you will learn how to:

- Import STEP geometry
- Employ GAMBIT cleanup operations to clean up imported geometry and suppress model features that inhibit meshing
- Create and apply size functions to control mesh quality

NOTE: You can reproduce the perspectives of the figures in this tutorial by means of `window matrix` commands available in a journal file named "`tg11_figures.jou`," which is included in the "`help/tutfiles`" online help directory. To exactly reproduce the perspective of any figure, you can open the journal file and execute the window matrix command associated with the figure. For example, the following command reproduces the perspective of the model shown in Figure 11-3.

```
window matrix 1 entries \
  0.8298196196556   0.1376460045576  -0.5407903790474 \
 -0.98521900177    -0.3953186273575   0.828989803791  \
 -0.3955990076065  -0.0812062472105   0.3938567638397 \
  0.5420601963997   0.742325425148   -3.794617891312  \
-12.15634346008    12.11377906799    -4.06431388855   \
 15.50736236572   -22.28459358215    22.28459358215
```

11.1 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.

11.2 Problem Description

Figure 11-1 and Figure 11-2 show the drill-bit configuration to be modeled and meshed in this tutorial. Figure 11-1 shows the full model, including the outer face that circumscribes the internal components. Figure 11-2 shows the internal components, themselves.

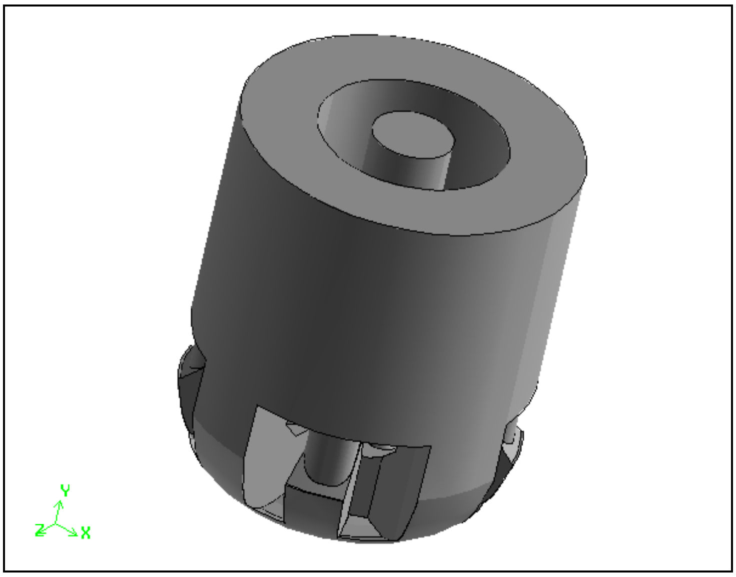


Figure 11-1: Industrial drill bit configuration—full model

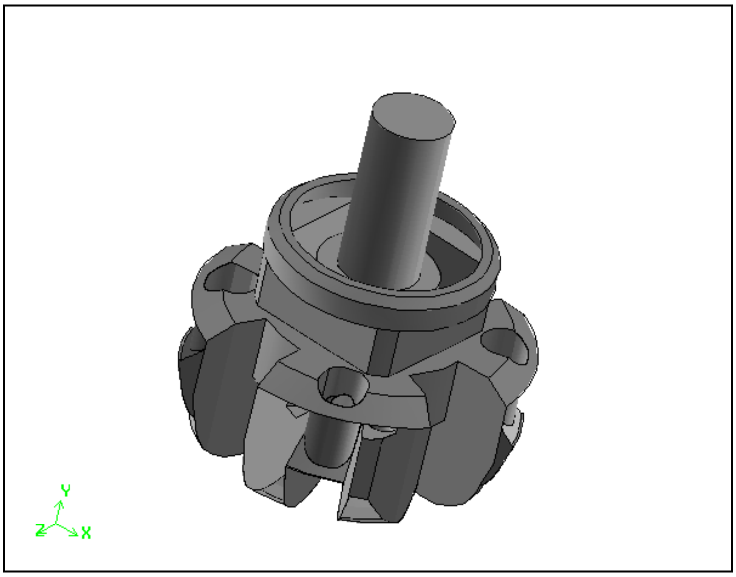


Figure 11-2: Industrial drill bit configuration—internal components

The goals of this tutorial are:

- To import the model by means of a STEP data file
- To use the GAMBIT cleanup tools to identify and eliminate short edges and small faces that can adversely affect meshing
- To mesh the model using unstructured, tetrahedral mesh elements the quality of which is controlled by means of size functions

This tutorial, in conjunction with Tutorial 12, also illustrates the differences between geometry import by means of STEP data files and direct CAD geometry import. Specifically, Tutorial 12 employs a model identical to that shown above but imports the data directly, as a "part" file, from the CAD program that was used to create the drill-bit geometry. (NOTE: The geometry for this example was created by means of the Pro/ENGINEER CAD program.)

11.3 Strategy

The general strategy employed in this tutorial is as follows:

1) Import a STEP file that describes the drill-bit geometry.
2) Use the GAMBIT cleanup tools to identify and eliminate very short edges and small faces that can adversely affect meshing.
3) Apply size functions to control mesh quality.
4) Mesh the model.

11.4 Procedure

1. Copy the file

   path/Fluent.Inc/gambit2.*x*/help/tutfiles/drill_bit.stp

   (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 “Drill_Bit_STEP”.

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. For some systems,* **FLUENT 5/6** *is the default solver. The currently selected solver is shown at the top of the* GAMBIT *GUI.*

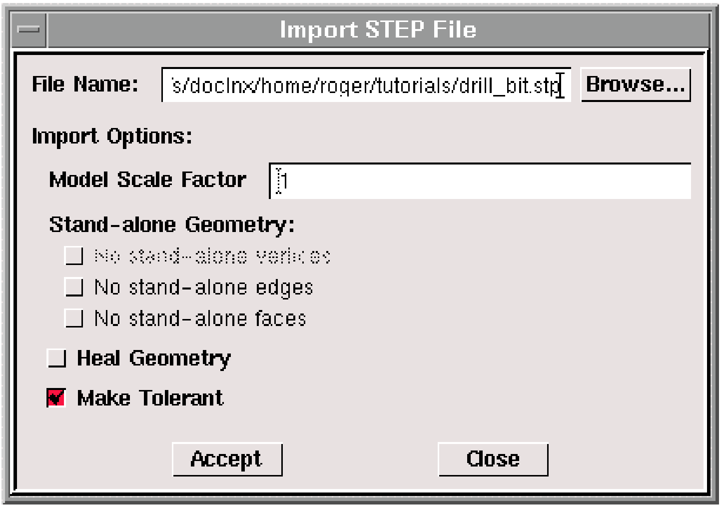
Step 2: Import a STEP File

STEP data files contain geometry data formatted according to a set of industry standards. For this tutorial, the STEP data file was created using the Pro/ENGINEER CAD program.

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

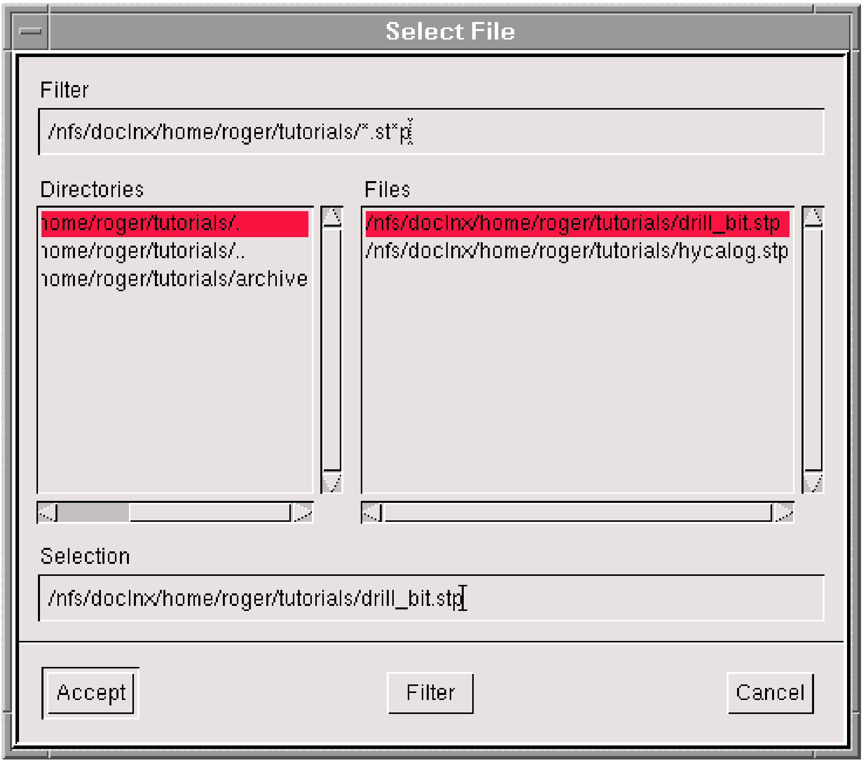
   File → **Import** → **STEP...**

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



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

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



a) In the Files list, select `drill_bit.stp`.

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

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

GAMBIT *reads the information contained in the data file and constructs the geometry shown in Figure 11-3.*

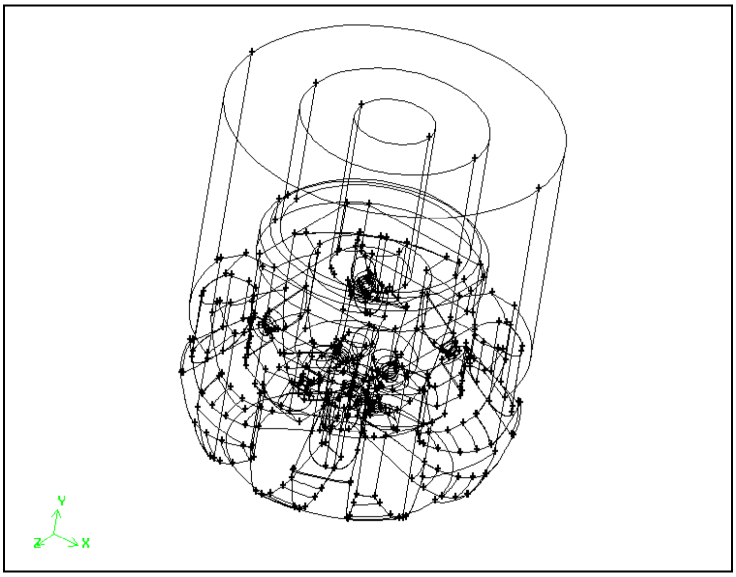


Figure 11-3: Industrial drill bit—imported STEP geometry

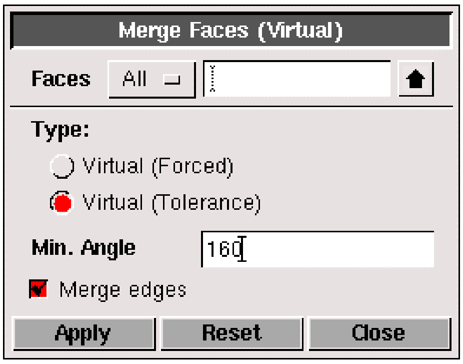
Step 3: Merge Faces and Edges to Suppress Model Features

It is often useful to use the face and edge merging operations available in GAMBIT *to suppress model features that would otherwise inhibit the meshing. In this step, you will use a global face-merge operation to suppress such features.*

1. Perform a global face-merge operation.

GEOMETRY → **FACE** → **SPLIT/MERGE/COLLAPSE/SIMPLIFY FACES** **R** Merge Faces (Virtual)

This command sequence opens the **Merge Faces (Virtual)** *form.*



a) On the **Type** option subset, select the Virtual (Tolerance) option.

b) On the **Faces** pick-list option button, select All.

c) In the **Min. Angle** text box, input `160`.

d) Retain the Merge edges option.

e) Click **Apply** to merge the faces.

GAMBIT *merges the faces to create the model as shown in Figure 11-4.*

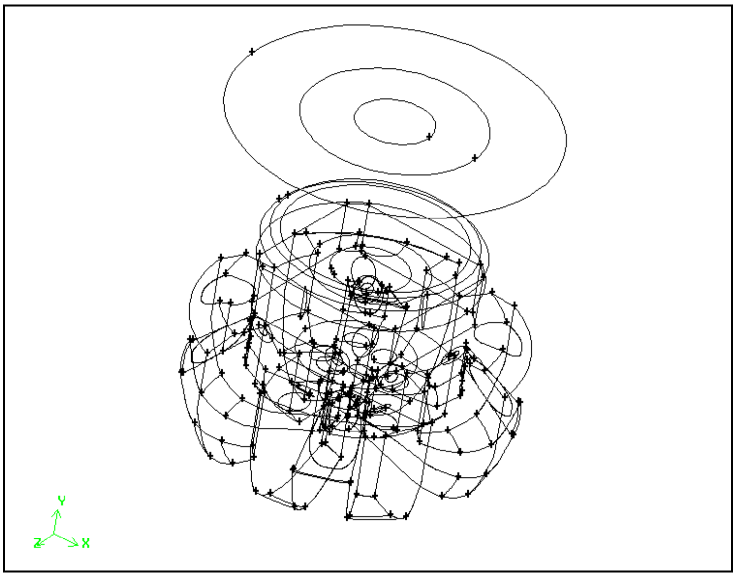


Figure 11-4: Model with merged faces

Step 4: Use Cleanup Tools to Check and Clean Up Geometry

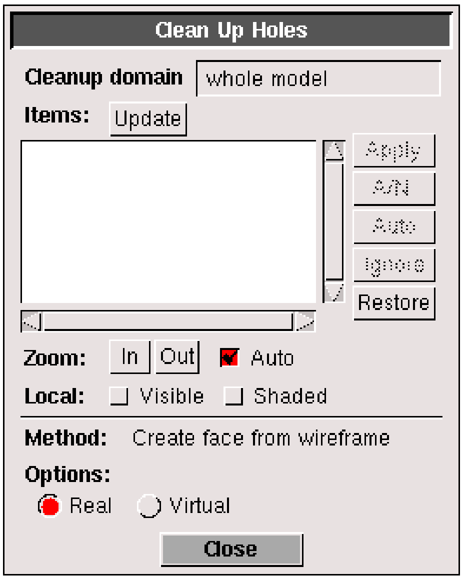
GAMBIT *cleanup tools allow you to easily identify and eliminate individual model features that can inhibit meshing. In this step, you will use the cleanup tools to check for the existence of "holes" and "cracks" in the model and to eliminate small faces.*

1. Check for the existence of holes in the model.

   "Holes" in the model are internal edge loops that do not constitute face boundaries.

TOOLS  → **CLEANUP**  → **CLEAN UP HOLES** 

This command sequence opens the **Clean Up Holes** *form.*



a) Click the **Update** pushbutton located on the right side of the **Items** list heading.

   In this case, GAMBIT *does not populate the* **Items** *list, because no holes exist in the model.*

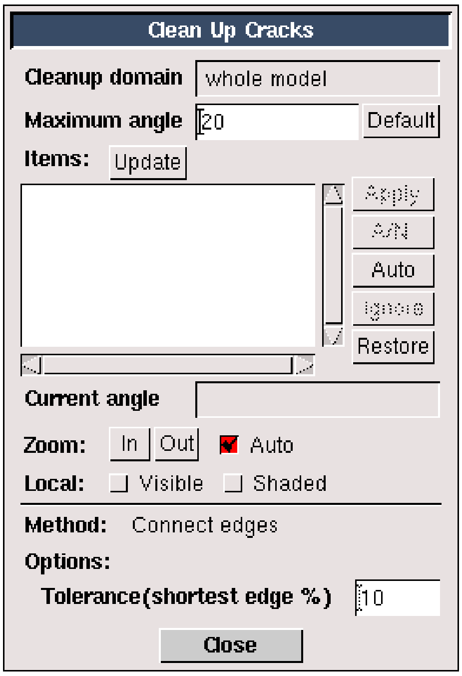
2. Check for the existence of cracks in the model.

For the purposes of the geometry cleanup operations, a "crack" is defined as a geometry consisting of an edge pair that meets the following criteria:

- *Each edge in the pair serves as a boundary edge for a separate face.*
- *The edges share common endpoint vertices at one or both ends.*
- *The edges are separated along their lengths by a small gap.*

TOOLS → **CLEANUP** → **CLEAN UP CRACKS**

This command sequence opens the **Clean Up Cracks** *form.*



a) Click the Default pushbutton located on the right side of the **Maximum angle** text box.

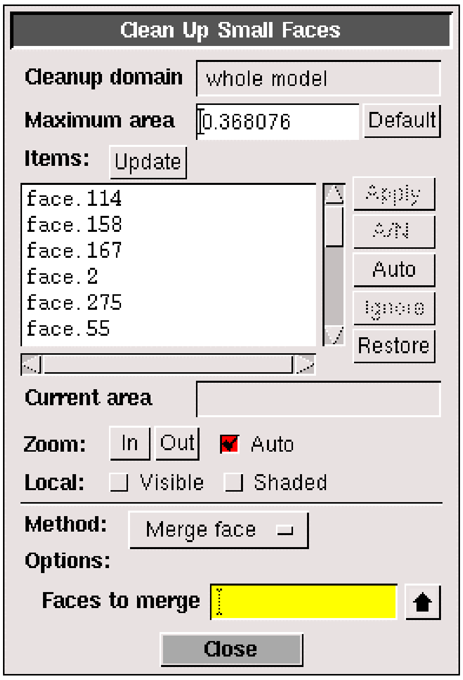
GAMBIT displays the default **Maximum angle** *criterion and automatically populates the* **Items** *list with all cracks existing in the model. In this case,* GAMBIT *does not populate the* **Items** *list, because no cracks exist in the model.*

3. Clean up small faces in the model.

In this substep, you will use the **Clean Up Small Faces** *form to identify and eliminate individual small faces that can adversely affect meshing operations.*

TOOLS → **CLEANUP** → **CLEAN UP SMALL FACES**

This command sequence opens the **Clean Up Small Faces** *form.*



a) Click the Default pushbutton located on the right side of the **Maximum area** text box.

When you click the Default *pushbutton,* GAMBIT *displays the* **Maximum area** *of faces to be included in the* **Items** *list and populates the* **Items** *list with all faces in the* **Cleanup domain** *that meet the* **Maximum area** *criterion. By default, the* **Maximum area** *value is 100 times greater than the area of the smallest face in the* **Cleanup Domain**.

Figure 11-5 shows the three smallest faces in the model, all of which lie at the base of the main drill-bit shaft. These three faces correspond to the first three faces listed in the **Items** *list.*

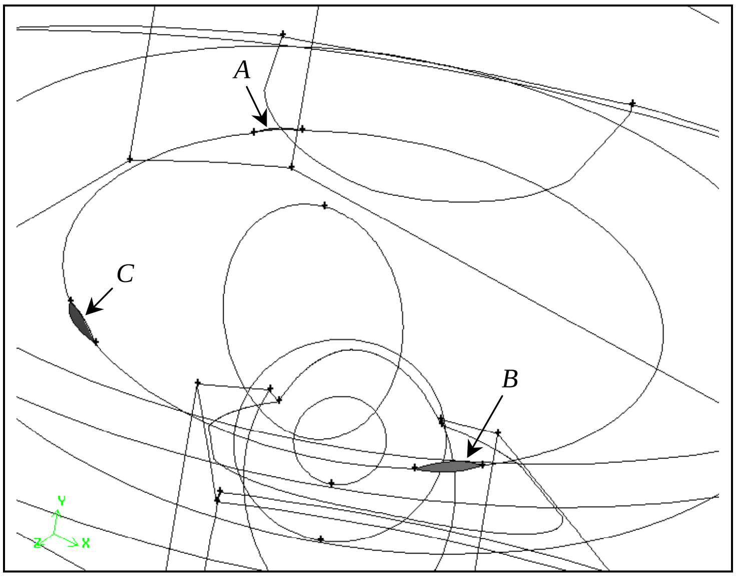


Figure 11-5: Three smallest faces in the model

b) Select the first face in the **Items** list.

GAMBIT *displays the area of the selected face in the* **Current area** *field located below the* **Items** *list and highlights the selected face (face A in Figure 11-5) in the graphics window.*

The **Clean Up Small Faces** *form provides two* **Method** *options for eliminating faces—*Collapse face *and* Merge face. *In this case,* GAMBIT *automatically selects the* Merge face *option and populates the* **Faces to merge** *pick list with suggested faces to merge.*

c) Click the A/N pushbutton in the vertical array of pushbuttons located to the right of the **Items** list.

The A/N *("Apply/Next") pushbutton removes the currently selected face from the model then updates the* **Items** *list and automatically selects the next smallest face in the* **Cleanup domain**. *In this case,* GAMBIT *eliminates the selected face and automatically selects the next smallest face (face B in Figure 11-5).*

d) Click A/N again to eliminate the next smallest face in the **Cleanup domain**.

GAMBIT *eliminates the selected face and automatically highlights the next smallest face (face C in Figure 11-5).*

e) Click Apply to eliminate the third smallest face in the **Cleanup domain**.

Figure 11-6 shows the geometry in the region of the three smallest faces after their removal from the model.

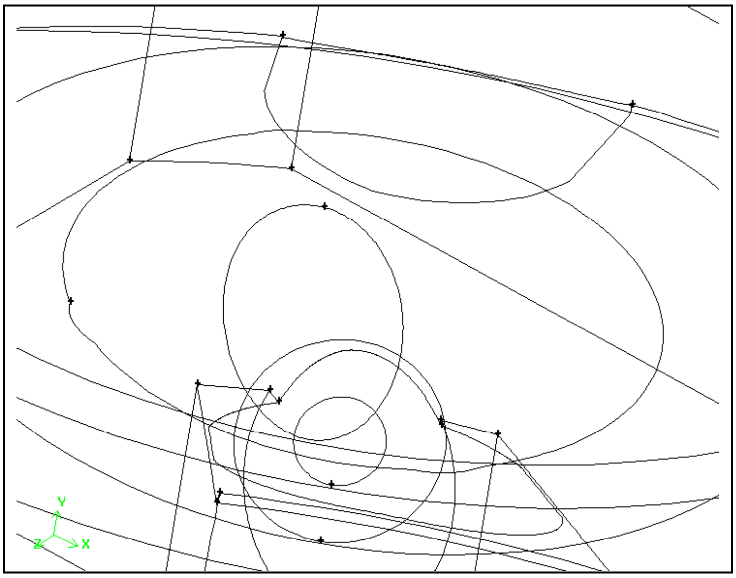


Figure 11-6: Base of main shaft with three smallest faces cleaned up

Having eliminated the three small, ovoid faces at the base of the main shaft, you will now remove four small, rectangular faces that serve as lips to other, larger faces (see Figure 11-7).

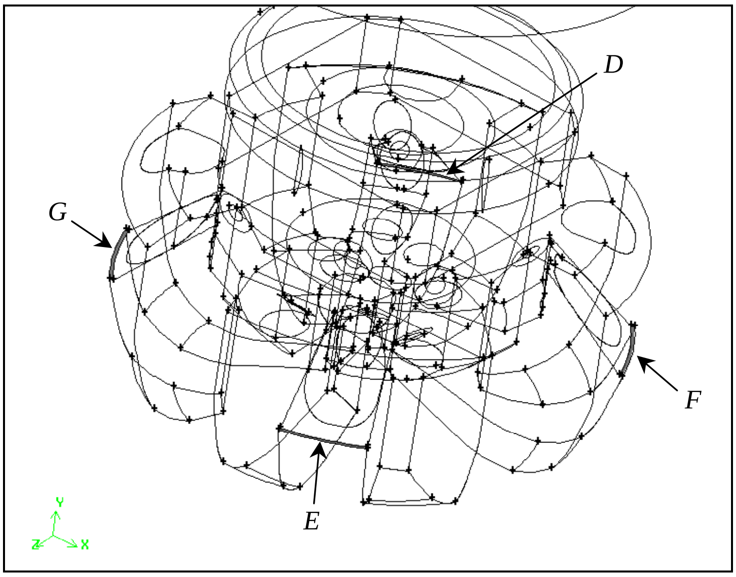


Figure 11-7: Small, rectangular lip faces

f) Select the first face in the **Items** list.

GAMBIT *highlights and zooms in on the smallest of the rectangular lip faces (face D in Figure 11-7) and automatically selects the* Merge faces *option and populates the* **Faces to merge** *pick list with suggested faces to merge.*

g) Click the A/N pushbutton to eliminate the selected face and automatically select the next smallest face (face *E* in Figure 11-7).

h) Click the A/N pushbutton to eliminate the selected face and automatically select the next smallest face (face *F* in Figure 11-7).

i) Click the A/N pushbutton to eliminate the selected face and automatically select the next smallest face (face *G* in Figure 11-7).

j) Click Apply to eliminate the last of the lip faces.

After cleanup of the last lip face, the **Items** *list still contains a list of small faces; however, the remaining faces are not small enough to adversely affect meshing operations.*

Figure 11-8 shows the final, cleaned-up geometry for the drill-bit model.

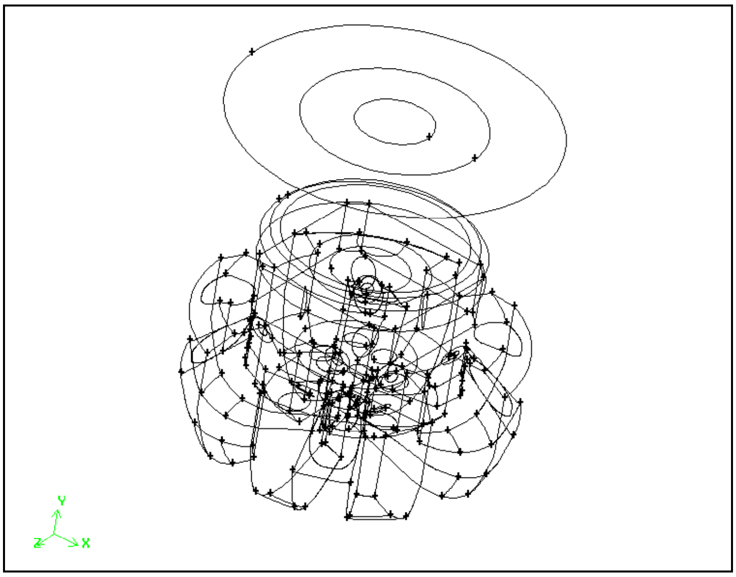


Figure 11-8: Final, cleaned-up model geometry

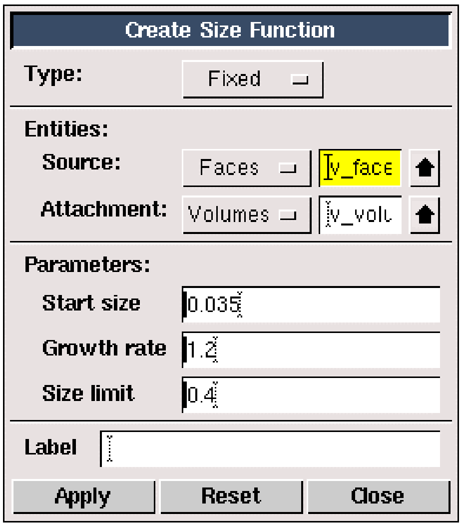
Step 5: Apply Size Functions to Control Mesh Quality

Highly skewed elements adversely affect numerical computations for which the mesh is created. GAMBIT *includes several features that allow you to control the mesh, one of which is the application of size functions. For example, size functions can be used to specify the rate at which volume mesh elements change in size in proximity to a specified boundary. In this step, you will apply size functions to four faces of the drill-bit geometry and, thereby, control the size of the nearby mesh elements to eliminate the skewed elements.*

1. Specify a size function and apply it to four faces of the model.

TOOLS → **SIZE FUNCTIONS** → **CREATE SIZE FUNCTION**

This command sequence opens the **Create Size Function** *form.*



a) Retain the **Type:**Fixed option.

(NOTE: In addition to the "fixed" size function illustrated in this tutorial, GAMBIT *provides "curvature," "proximity," and "meshed" size functions. Curvature and proximity size functions are useful for controlling the mesh in regions of high curvature and small gaps, respectively. Meshed size functions use existing meshes to determine the size-function start size. See Section 5.2.2 of the* GAMBIT *Modeling Guide.)*

b) On the **Entities:Source** option button, select the Faces option.

c) In the Faces list box, select the four faces shown (shaded) in Figure 11-9.

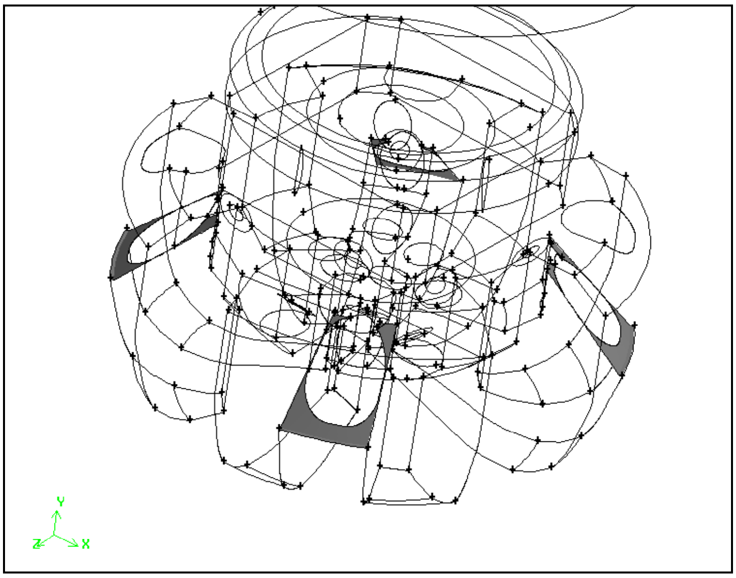


Figure 11-9: Faces on which to apply size functions

d) On the **Entities:Attachment** option button, select the Volumes option.

e) In the Volumes list box, select the volume.

f) In the **Start size** text box, enter the value `0.035`.

g) In the **Growth rate** text box, enter the value, `1.2`.

h) In the **Size limit** text box, enter the value, `0.4`.

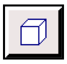
i) Click **Apply** to create the size function.

When applying the size function, GAMBIT *displays a message in the* **Transcript** *window indicating that the use of virtual entities as source entities in the size-function definition can cause problems when evaluated during background-grid generation. The message represents a warning only and can be ignored in this case.* GAMBIT *allows you to view the size function by means of the* **View Size Function** *command on the* **Size Function** *toolpad.*

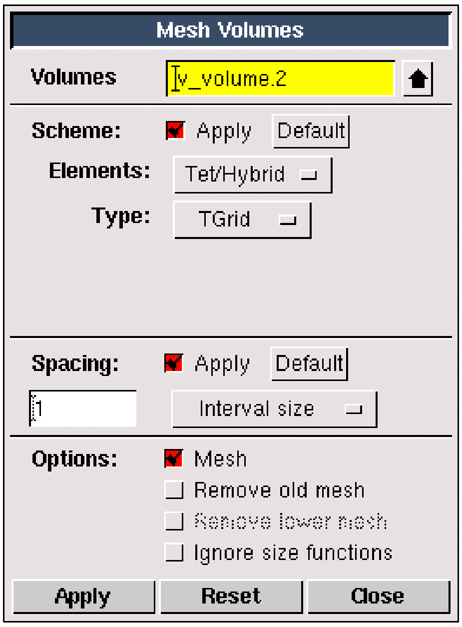
Step 6: Mesh the Volume

After the imported geometry is cleaned up and the size-function is created and attached, you can mesh the geometry using an unstructured, tetrahedral mesh.

1. Mesh the drill-bit volume.

MESH  → **VOLUME**  → **MESH VOLUMES** 

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



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

b) Select the volume.

GAMBIT *automatically selects the* **Scheme:Elements:**Tet/Hybrid *and* **Scheme: Type:**TGrid *options.*

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

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

e) In the **Spacing** text box, retain the default value of `1`.

The size function you attached to the volume in the previous step will override the **Spacing** *specifications.*

f) Click **Apply**.

Figure 11-10 shows the final meshed volume.

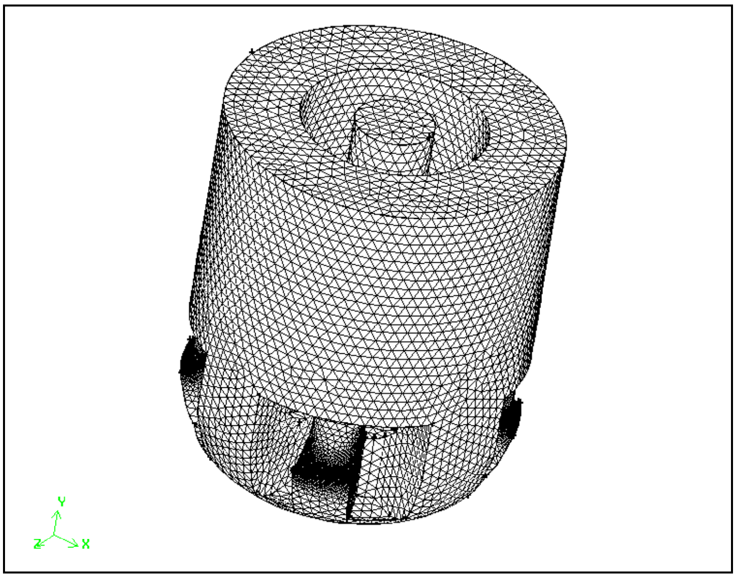
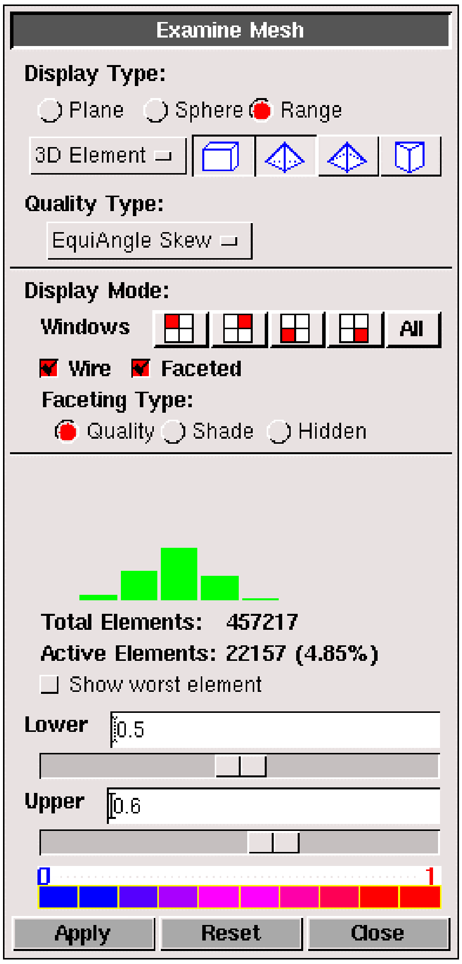


Figure 11-10: Meshed drill-bit volume

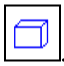
Step 7: Examine the Volume Mesh

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

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



a) Select **Range** under **Display Type** at the top of the form.

The **3D Element** *type selected by default at the top of the form is a brick* *. You will not see any mesh elements in the graphics window when you first open the* **Examine Mesh** *form, because there are no hexahedral elements in the mesh.*

b) Left-click the tetrahedron icon  next to **3D Element** near the top of the form.

The tetrahedral mesh elements will now be visible in the graphics window.

c) Select or retain EquiAngle Skew from the **Quality Type** option menu.

d) Left-click the histogram bars that appear at the bottom of the **Examine Mesh** form to highlight elements in any given quality range.

Figure 11-11 shows the graphics window that results for elements with EquiAngle Skew *values between 0.5 and 0.6.*

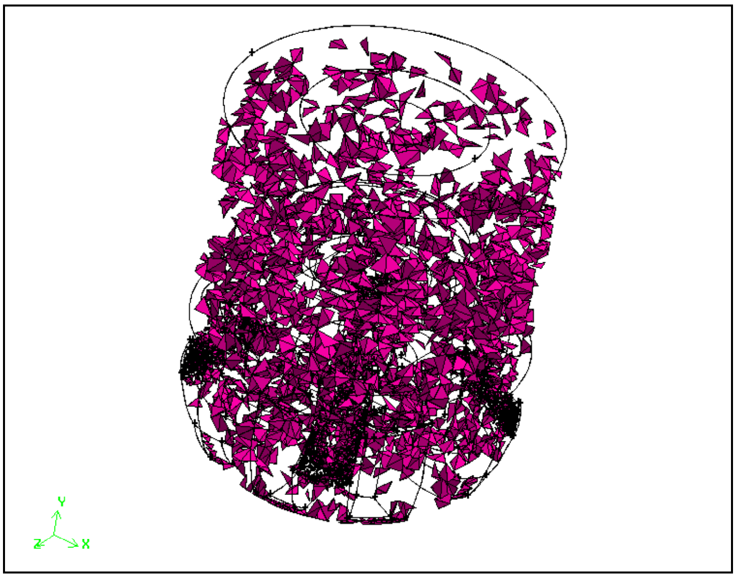


Figure 11-11: Elements with EquiAngle Skew values between 0.5 and 0.6

e) On the **Examine Mesh** form, click **Close** to close the form.

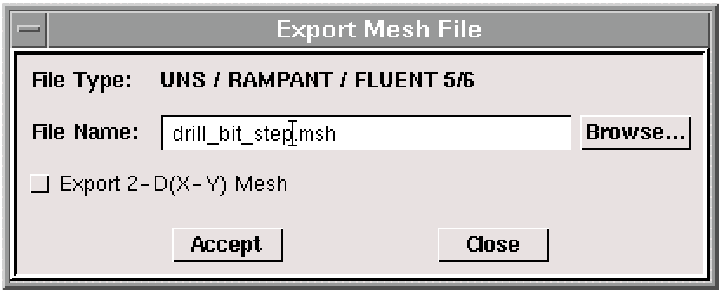
Step 8: Export the Mesh and Exit GAMBIT

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 "`drill_bit_step.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 command sequence opens the **Exit** *form.*

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

11.5 Summary

This tutorial demonstrates the importation of CAD geometry by means of a STEP data file and the operations that are sometimes required to clean up such geometry and render it suitable to GAMBIT meshing operations. In this case, the application of size functions prevented the creation of skewed elements during meshing.

12. INDUSTRIAL DRILL BIT—DIRECT CAD IMPORT

This tutorial employs the industrial drill-bit model described in Tutorial 12 to illustrate the advantages of importing geometry directly from a CAD program rather than importing the geometry by means of an intermediate (STEP) file. The directly imported geometry does not include the very short edges that required elimination in Tutorial 12, however, it does include some small faces that must be merged to facilitate meshing.

In this tutorial, you will learn how to:

- Import geometry directly from the Pro/ENGINEER CAD program
- Use the GAMBIT cleanup tools to identify and eliminate geometry features that can adversely affect meshing operations

---

NOTE (1): The capability of direct geometry import from the Pro/ENGINEER program requires a special GAMBIT license. Without the license, GAMBIT cannot open a database that includes directly imported CAD geometry.

NOTE (2): You can reproduce the perspectives of the figures in this tutorial by means of `window matrix` commands available in a journal file named "`tg12_figures.jou`," which is included in the "`help/tutfiles`" online help directory. To exactly reproduce the perspective of any figure, you must open the journal file and execute the window matrix command associated with the figure. For example, the following command reproduces the perspective of the model shown in Figure 12-3.

```
window matrix 1 entries \
  0.8298196196556   0.1376460045576  -0.5407903790474 \
 -0.98521900177    -0.3953186273575   0.828989803791  \
 -0.3955990076065  -0.0812062472105   0.3938567638397 \
  0.5420601963997   0.742325425148   -3.794617891312  \
-12.15634346008    12.11377906799    -4.06431388855   \
 15.50736236572   -22.28459358215    22.28459358215
```

12.1 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, 4, 8, and 11.

12.2 Problem Description

Figure 12-1 and Figure 12-2 show the drill-bit configuration to be modeled and meshed in this tutorial. Figure 12-1 shows the full model, including the outer face that circumscribes the internal components. Figure 12-2 shows the internal components, themselves. The model shown in these figures is identical to that described in Tutorial 11.

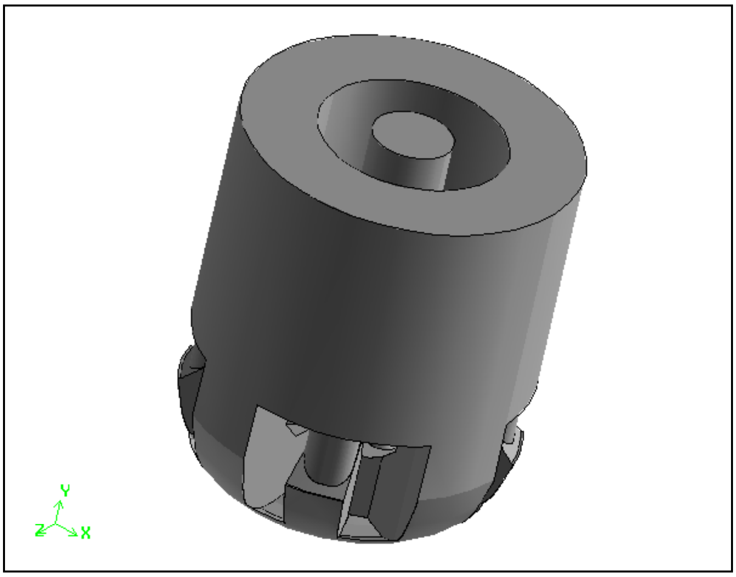


Figure 12-1: Industrial drill bit configuration—full model

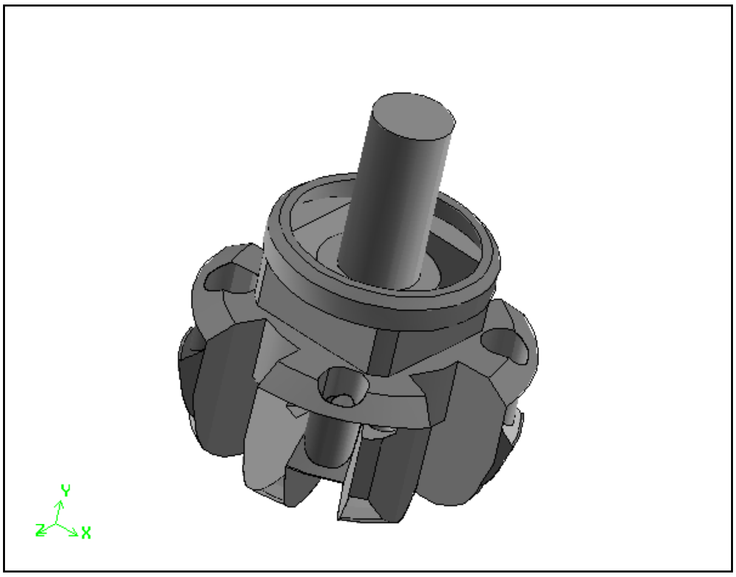


Figure 12-2: Industrial drill bit configuration—internal components

The goals of this tutorial are:

- To directly import the drill-bit geometry from the Pro/ENGINEER CAD program
- To use GAMBIT cleanup operations to render the model suitable for meshing
- To mesh the model using unstructured, tetrahedral mesh elements the quality of which is controlled by means of size functions

This tutorial, in conjunction with Tutorial 11, also illustrates the differences between direct CAD geometry import and geometry import by means of STEP data files. Specifically, this tutorial imports the data directly from the Pro/ENGINEER program as a "part" file. Consequently, the imported geometry does not include the very short edges that otherwise complicate meshing (see Tutorial 11).

12.3 Strategy

The general strategy employed in this tutorial is as follows:

1) Start the Pro/ENGINEER program.
2) Launch GAMBIT from within Pro/ENGINEER.
3) Import to GAMBIT a Pro/ENGINEER part file that describes the drill-bit geometry.
4) Use GAMBIT cleanup operations to eliminate a few small faces that would otherwise complicate meshing.
5) Apply size functions to control mesh quality.
6) Mesh the model.

The operations involved in items 4–6, above, are nearly identical to those described in Steps 4–6 of Tutorial 11.

12.4 Procedure

Step 1: Start Pro/ENGINEER

1. In a terminal window, enter

   `gambit -id Drill_Bit_ProE –proe` *`proe_startup_command`*

   where *`proe_startup_command`* is the system-specific command to start Pro/-ENGINEER.

   This command starts Pro/ENGINEER *and displays the* Pro/ENGINEER *user interface.*

   ! GAMBIT *is available only in a 32-bit version; therefore, the* Pro/ENGINEER *version used for direct CAD import must also be 32-bit.*

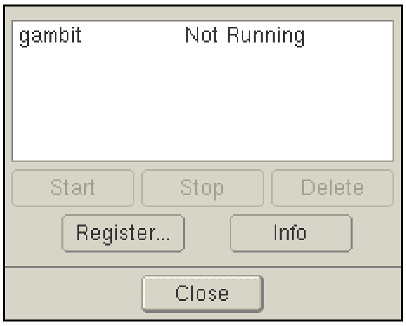
Step 2: Start GAMBIT from within Pro/ENGINEER

1. Start GAMBIT and make it available as an option on the Pro/ENGINEER main menu.

   a) Open the Pro/ENGINEER **Auxiliary Applications** form.

      Tools → Auxiliary Applications…

      This command sequence opens the Pro/ENGINEER **Auxiliary Applications** *form.*

      i. In the list of available auxiliary applications, select gambit, and click Start.

         Pro/ENGINEER *starts* GAMBIT *and displays a new option—titled,* Gambit*—on the* Pro/ENGINEER *main menu.* Pro/ENGINEER *also displays the message, "Application 'gambit' started successfully," to indicate the successful launch of the* GAMBIT *program.*

      ii. Click Close to close the **Auxiliary Applications** form.

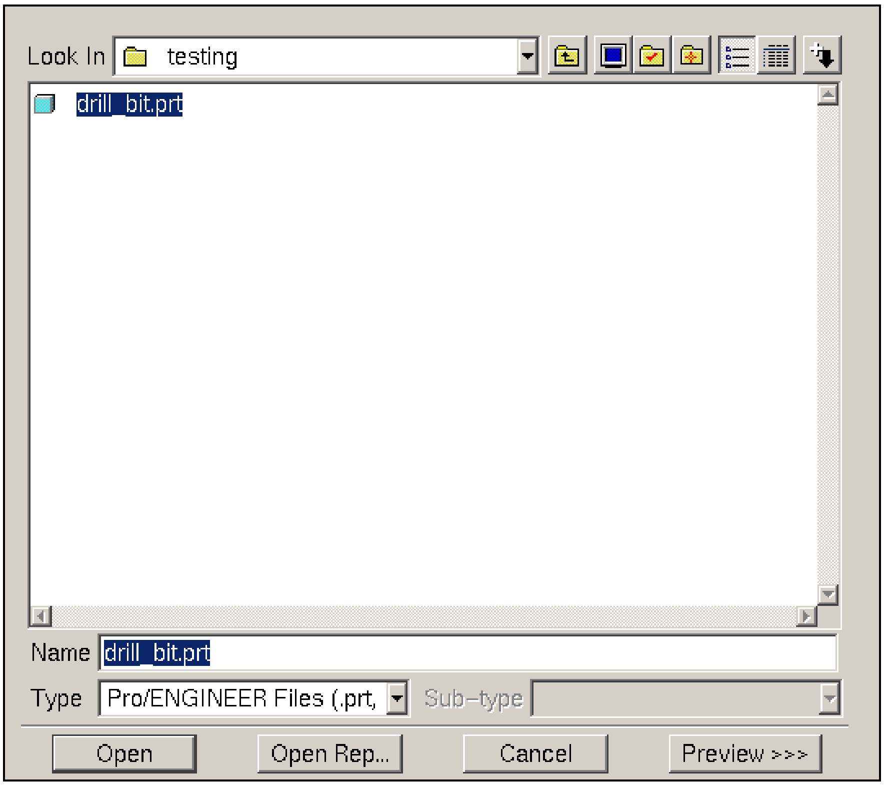
Step 3: Open the Part File

1. Open the Pro/ENGINEER part file.

   a) Open the Pro/ENGINEER part file that describes the drill bit geometry.

   File −→ Open...

   This command sequence opens the Pro/ENGINEER **File Open** *form.*



   i. In the file list, select `drill_bit.prt`, and click Open.

   Pro/ENGINEER *opens the part file and displays it in the* Pro/ENGINEER *GUI graphics window.*

   ! *You cannot operate on parts or assemblies from within* Pro/ENGINEER *while* GAMBIT *is running.*

Step 4: Display the GAMBIT User Interface

1. Display the GAMBIT graphical user interface.

   a) On the Pro/ENGINEER main menu, start the GAMBIT interface.

   Gambit → Start

   Pro/ENGINEER *replaces its foreground user interface with that of* GAMBIT *and remains operational in the background.*

   It is possible to switch between Pro/ENGINEER *and* GAMBIT *operation while* GAMBIT *is running.*

   - *To switch from* GAMBIT *to* Pro/ENGINEER*, you must exit* GAMBIT *by means of the* **File/Close** *option on the* GAMBIT *main menu bar. When you exit* GAMBIT *in this manner, the* GAMBIT *window is iconized, and* GAMBIT *continues to run until you end its execution from within* Pro/ENGINEER.

   - *To switch from* Pro/ENGINEER *to* GAMBIT*, select the* Gambit→Start *option on the* Pro/ENGINEER *main menu bar.*

   To ensure that any GAMBIT *operations are preserved when switching back and forth between* GAMBIT *and* Pro/ENGINEER*, it is advisable to save the* GAMBIT *database before switching from* GAMBIT *to* Pro/ENGINEER *operation.*

Step 5: Select the Solver

1. Choose the solver from the main menu bar:

   Solver $\longrightarrow$ **FLUENT 5/6**

   The choice of solver affects the types of options available in the **Specify Boundary Types** *form. For some systems,* **FLUENT 5/6** *is the default solver. The currently selected solver is shown at the top of the* GAMBIT *GUI.*

Step 6: Import the CAD Geometry

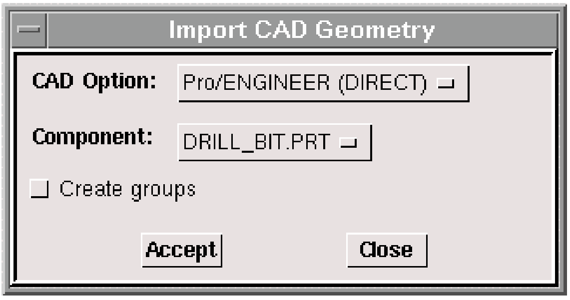
In this step, you will directly import the drill-bit geometry from Pro/ENGINEER. GAMBIT *designates the imported geometry as "CAD" geometry, and assigns its components a "c_" prefix—for example, c_face.123.*

! *To import geometry directly from* Pro/ENGINEER *to* GAMBIT*, you must have a special* GAMBIT *license. Without the license,* GAMBIT *cannot open a database that includes directly-imported CAD geometry.*

1. Select the **Import CAD Geometry** option from the GAMBIT main menu bar.

   File → **Import** → **CAD...**

   This command sequence opens the **Import CAD Geometry** *form.*



   a) On the **CAD Option** option button, select the Pro/ENGINEER (DIRECT) option.

   b) On the **Component** option button, select the DRILL_BIT.PRT option.

      The **Component** *option button includes all part files that are currently open in* Pro/ENGINEER.

2. On the **Import CAD Geometry** form, click **Accept**.

   GAMBIT *imports the part file and displays the geometry shown in Figure 12-3.*

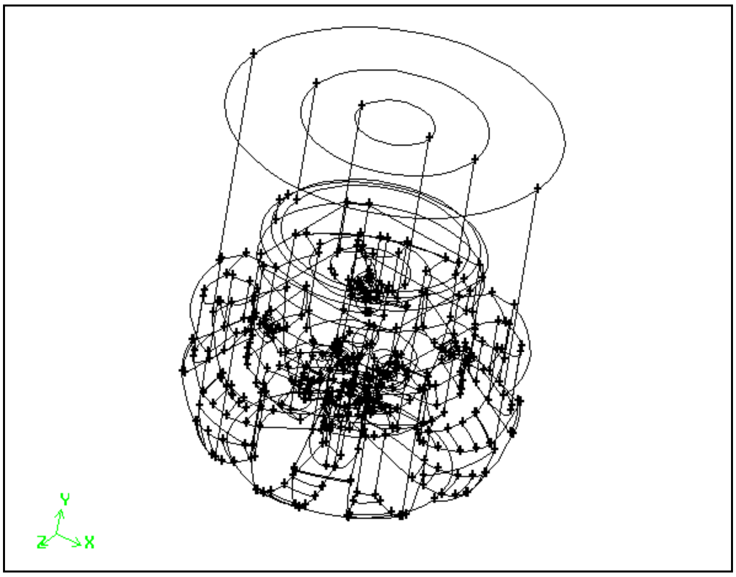


Figure 12-3: Industrial drill bit—directly imported Pro/ENGINEER part file

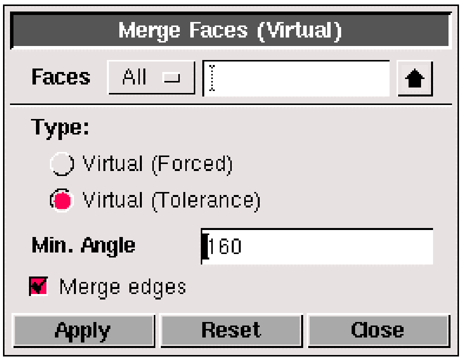
Step 7: Merge Faces and Edges to Suppress Model Features

As a first step in improving the meshing characteristics of the model, you will perform global face and edge merge operations to eliminate many faces that could otherwise adversely affect meshing.

1. Perform a global face-merge operation.

GEOMETRY → **FACE** → **SPLIT/MERGE/COLLAPSE/SIMPLIFY FACES** **R** Merge Faces (Virtual)

This command sequence opens the **Merge Faces (Virtual)** *form.*



a) On the **Type** option subset, select the Virtual (Tolerance) option.

b) On the **Faces** pick-list option button, select All.

c) In the **Min. Angle** text box, input `160`.

d) Retain the Merge edges option.

e) Click **Apply** to merge the faces.

GAMBIT *merges the faces as shown in Figure 12-4.*

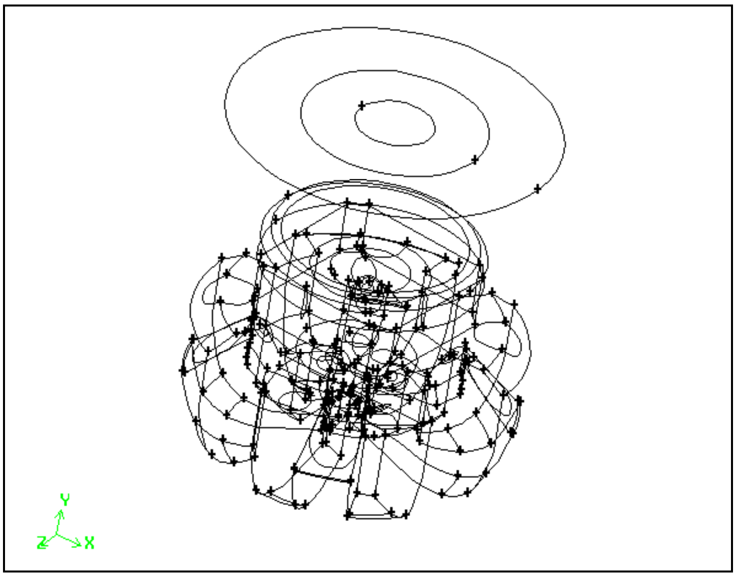
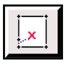


Figure 12-4: Model after face-merge operation

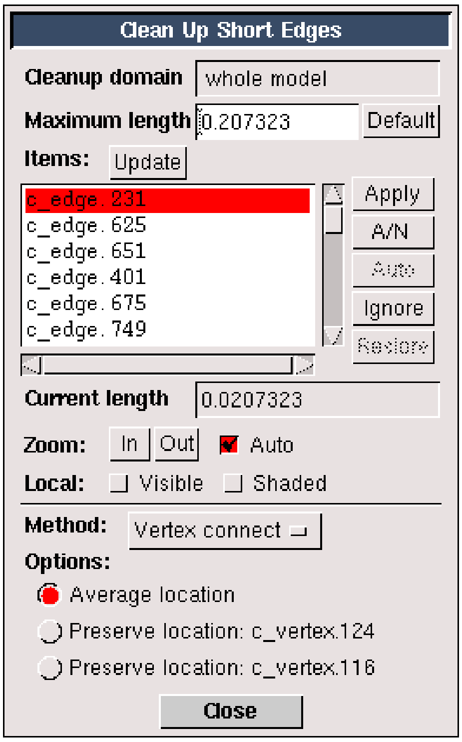
Step 8: Use Cleanup Tools to Check and Clean Up Geometry

GAMBIT *cleanup tools allow you to identify and eliminate individual model features that can inhibit meshing. In this step, you will use the cleanup tools to check for the existence of very short edges, "holes," "cracks," and small faces in the model and to eliminate some of the small faces.*

1. Identify any short edges in the model that might cause meshing problems.

TOOLS  → **CLEANUP**  → **CLEAN UP SHORT EDGES** 

This command sequence opens the **Clean Up Short Edges** *form.*



When you open any of the cleanup forms, such as the **Clean Up Short Edges** *form,* GAMBIT *automatically sets the graphics window color mode to display colors based on connectivity rather than topology. In addition,* GAMBIT *automatically sets the graphics window pivot function to the user-specified pivot mode.*

a) Click the **Default** pushbutton located on the right side of the **Maximum length** text box.

GAMBIT *displays the* **Maximum length** *of edges to be included in the* **Items** *list and populates the* **Items** *list with all edges in the* **Cleanup domain** *that meet the* **Maximum length** *criterion. (In this case, the entire model constitutes the* **Cleanup domain***.) By default, the* **Maximum length** *value is 10 times greater than the arc length of the shortest edge in the* **Cleanup domain**.

b) Select the first edge in the **Items** list.

GAMBIT *displays the arc length of the selected edge in the* **Current length** *field located below the* **Items** *list and highlights and zooms in on the selected edge in the graphics window (see Figure 12-5).*

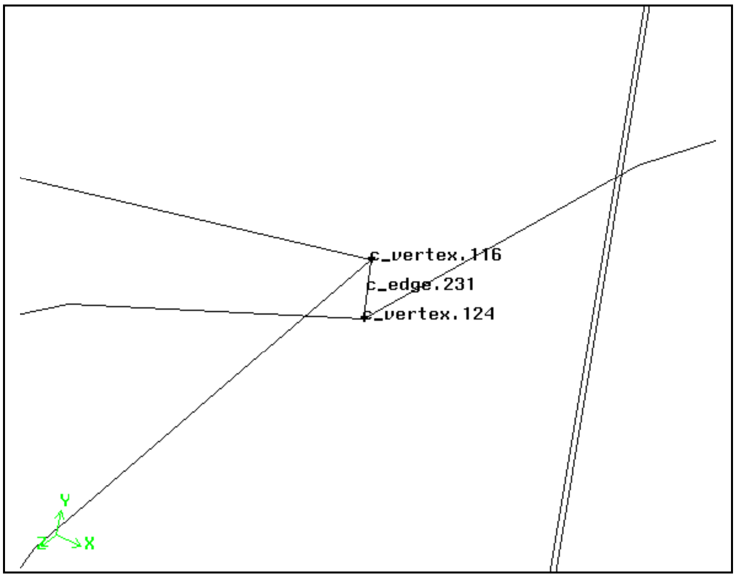


Figure 12-5: Automatic graphics-window display of the first listed edge

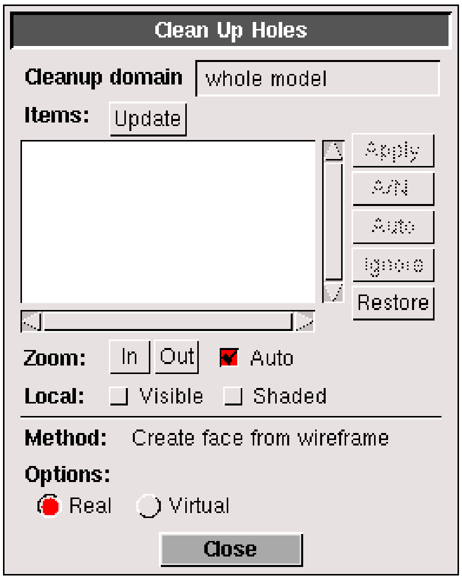
In this case, the shortest edge in the model is not short enough to adversely affect meshing.

2. Check for the existence of holes in the model.

"Holes" in the model are internal edge loops that do not constitute face boundaries.

TOOLS → **CLEANUP** → **CLEAN UP HOLES**

This command sequence opens the **Clean Up Holes** *form.*



a) Click the Update pushbutton located on the right side of the **Items** list heading.

In this case, GAMBIT *does not populate the* **Items** *list, because no holes exist in the model.*

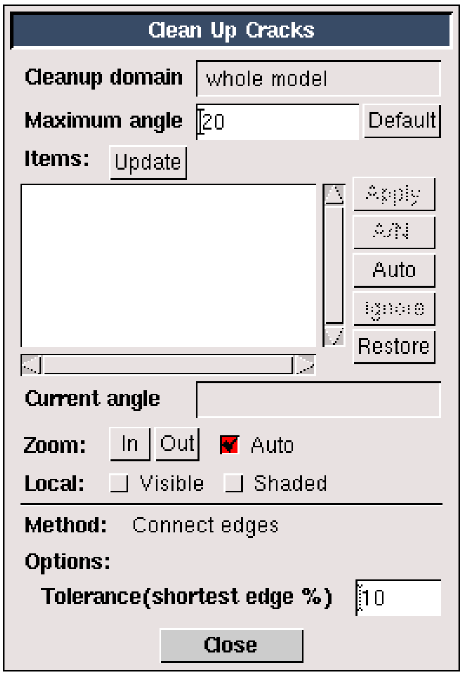
3. Check for the existence of cracks in the model.

For the purposes of the geometry cleanup operations, a "crack" is defined as a geometry consisting of an edge pair that meets the following criteria:

- *Each edge in the pair serves as a boundary edge for a separate face.*
- *The edges share common endpoint vertices at one or both ends.*
- *The edges are separated along their lengths by a small gap.*

TOOLS → **CLEANUP** → **CLEAN UP CRACKS**

This command sequence opens the **Clean Up Cracks** *form.*



a) Click the Default pushbutton located on the right side of the **Maximum angle** text box.

GAMBIT displays the default **Maximum angle** *criterion and automatically populates the* **Items** *list with all cracks existing in the model. In this case,* GAMBIT *does not populate the* **Items** *list, because no cracks exist in the model.*

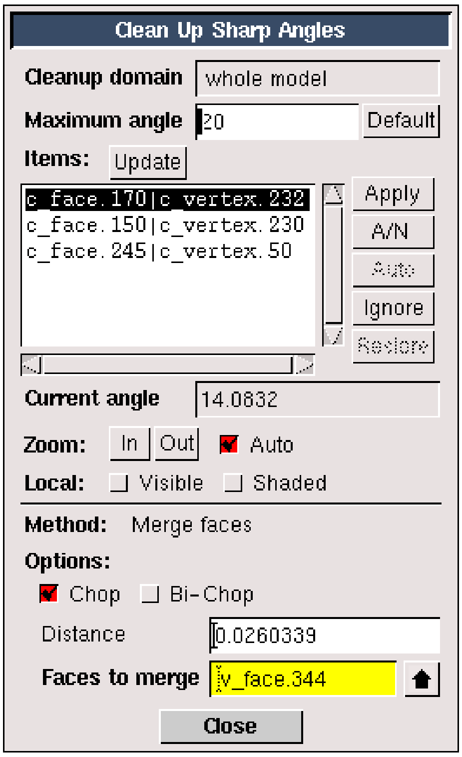
4. Clean up one sharp angle in the model.

In this substep, you will use the **Clean Up Sharp Angles** *form to identify and eliminate an edge pair that constitutes a "sharp angle." For the purposes of the geometry cleanup operations, a sharp angle is defined as an edge pair that meets the following criteria:*

- *The edge pair shares a common endpoint vertex and serves as part of the boundary for an existing face.*
- *One of the edges in the sharp-angle edge pair serves as a common boundary edge between its bounded face and an adjacent face.*
- *The angle between the edges in the pair (computed at their common endpoint vertex) is less than a specified angle.*

TOOLS → **CLEANUP** → **CLEAN UP SHARP ANGLES**

This command sequence opens the **Clean Up Sharp Angles** *form.*



a) Click the Default pushbutton located on the right side of the **Maximum angle** text box.

> GAMBIT *displays the* **Maximum angle** *of angles to be included in the* **Items** *list and populates the* **Items** *list with all face-vertex pairs in the* **Cleanup domain** *that meet the* **Maximum angle** *criterion. By default, the* **Maximum angle** *value is 20.*

b) Select the first face-vertex pair in the **Items** list.

> GAMBIT *highlights and zooms in on the geometry that constitutes the sharp angle (see Figure 12-6).*

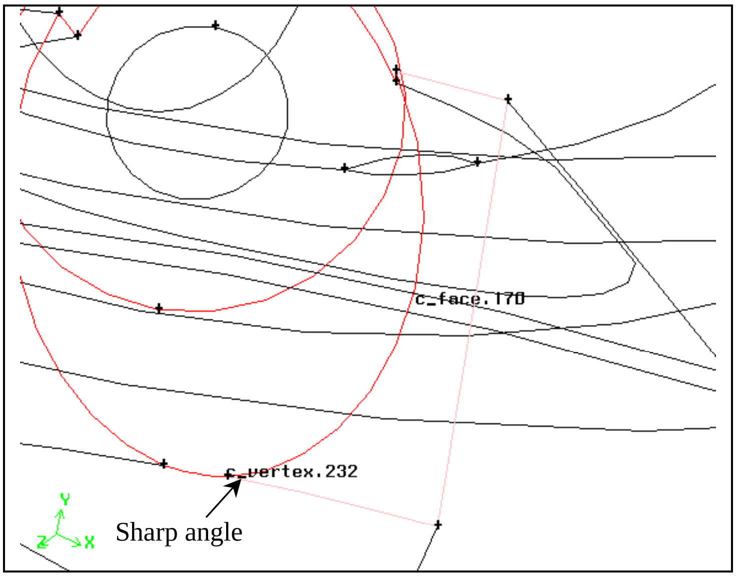


Figure 12-6: Geometry constituting a sharp angle

The **Clean Up Sharp Angles** *operation uses a* Merge faces *procedure to eliminate any sharp angle. In this case,* GAMBIT *automatically populates the* **Faces to merge** *pick list with suggested faces to merge and selects the* Chop *option. (For a complete description of the* **Clean Up Sharp Angles** *form, see "Clean Up Sharp Angles" in Section 5.4.2 of the* GAMBIT *Modeling Guide.)*

c) Click the Apply pushbutton in the vertical array of pushbuttons located to the right of the **Items** list.

GAMBIT merges the highly angular region of one face with the adjacent face to eliminate the sharp angle (see Figure 12-7).

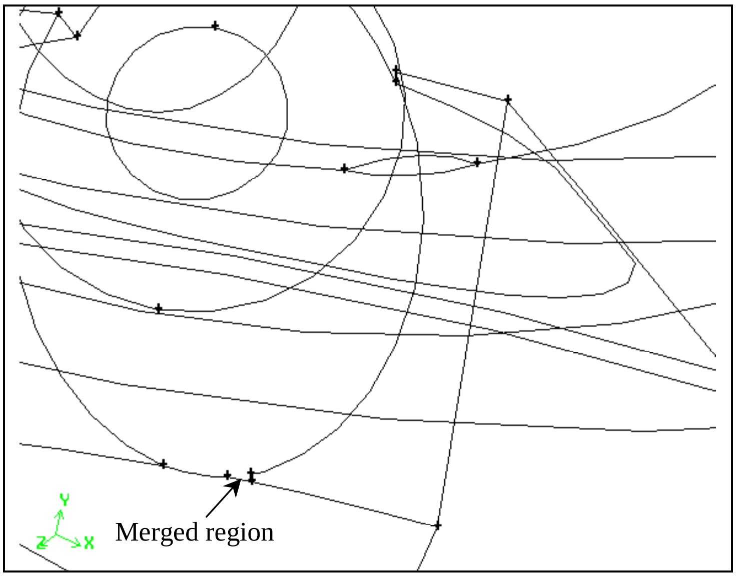
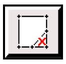


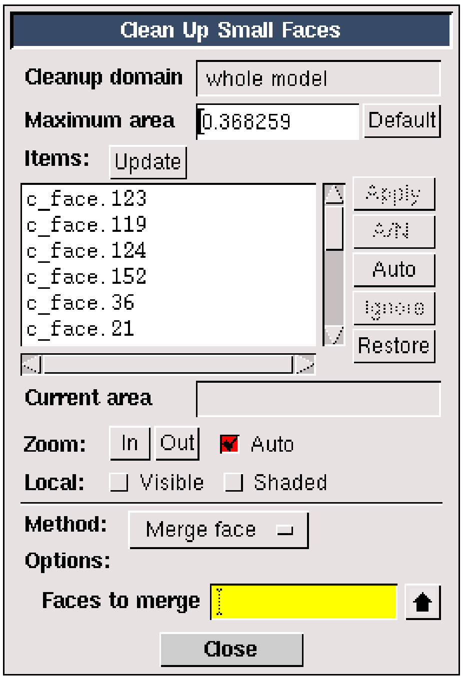
Figure 12-7: Geometry after sharp-angle cleanup operation

5. Clean up small faces in the model.

   In this substep, you will use the **Clean Up Small Faces** *form to identify and eliminate individual small faces that can adversely affect meshing operations.*

TOOLS  → **CLEANUP**  → **CLEAN UP SMALL FACES** 

   This command sequence opens the **Clean Up Small Faces** *form.*



a) Click the **Default** pushbutton located on the right side of the **Maximum area** text box.

GAMBIT *displays the* **Maximum area** *of faces to be included in the* **Items** *list and populates the* **Items** *list with all faces in the* **Cleanup domain** *that meet the* **Maximum area** *criterion. By default, the* **Maximum area** *value is 100 times greater than the area of the smallest face in the* **Cleanup Domain**.

Figure 12-6 shows the three smallest faces in the model, all of which lie at the base of the main drill-bit shaft. These three faces correspond to the first three labels listed in the **Items** *list.*

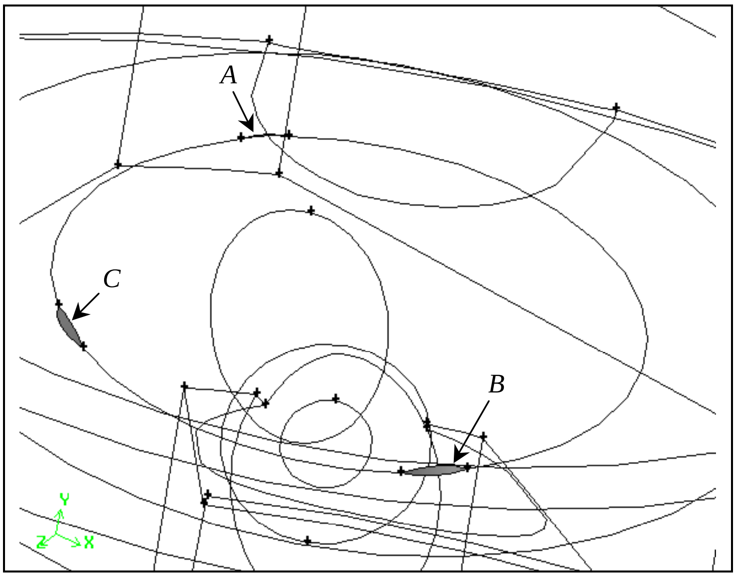


Figure 12-8: Three smallest faces in the model

b) Select the first face in the **Items** list.

GAMBIT *displays the area of the selected face in the* **Current area** *field located below the* **Items** *list and highlights the selected face (face A in Figure 12-8) in the graphics window.*

The **Clean Up Small Faces** *form provides two* **Method** *options for eliminating faces—*Collapse face *and* Merge face. *In this case,* GAMBIT *automatically selects the* Merge face *option and populates the* **Faces to merge** *pick list with suggested faces to merge.*

c) Click the A/N pushbutton in the vertical array of pushbuttons located to the right of the **Items** list.

The A/N *("Apply/Next") pushbutton removes the currently selected face from the model, then updates the* **Items** *list and automatically selects the next smallest face in the* **Cleanup domain**. *In this case,* GAMBIT *eliminates the selected face and automatically selects the next smallest face (face B in Figure 12-8).*

d) Click A/N again to eliminate the next smallest face in the **Cleanup domain**.

GAMBIT *eliminates the selected face and automatically highlights the next smallest face (face C in Figure 12-8).*

e) Click Apply to eliminate the third smallest face in the **Cleanup domain**.

Figure 12-9 shows the geometry in the region of the three smallest faces after their removal from the model.

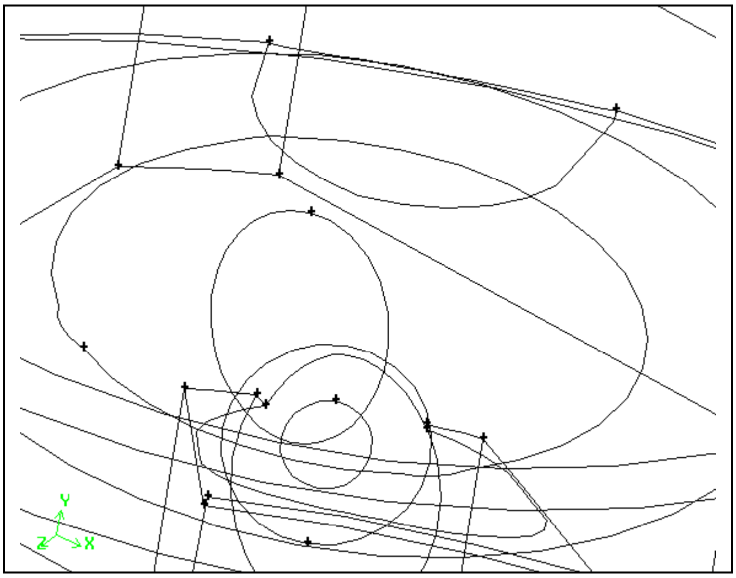


Figure 12-9: Geometry with three smallest faces cleaned up

Having eliminated the three small, ovoid faces at the base of the main shaft, you will now remove four small, rectangular faces that serve as lips to other, larger faces (see Figure 12-10).

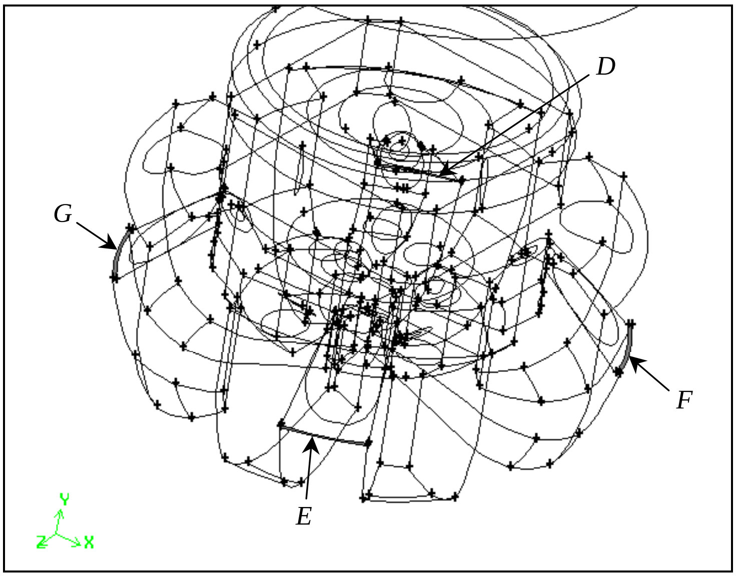


Figure 12-10: Small, rectangular lip faces

f) Select the first face in the **Items** list.

GAMBIT *highlights and zooms in on the smallest of the rectangular lip faces (face D in Figure 12-10) and automatically selects the* Merge faces *option and populates the* **Faces to merge** *pick list with suggested faces to merge.*

g) Click the A/N pushbutton to eliminate the selected face and automatically select the next smallest face (face *E* in Figure 12-10).

h) Click the A/N pushbutton to eliminate the selected face and automatically select the next smallest face (face *F* in Figure 12-10).

i) Click the A/N pushbutton to eliminate the selected face and automatically select the next smallest face (face *G* in Figure 12-10).

j) Click Apply to eliminate the last of the lip faces.

After cleanup of the last lip face, the **Items** *list still contains a list of small faces; however, the remaining faces are not small enough to adversely affect meshing operations.*

Figure 12-11 shows the final, cleaned-up geometry.

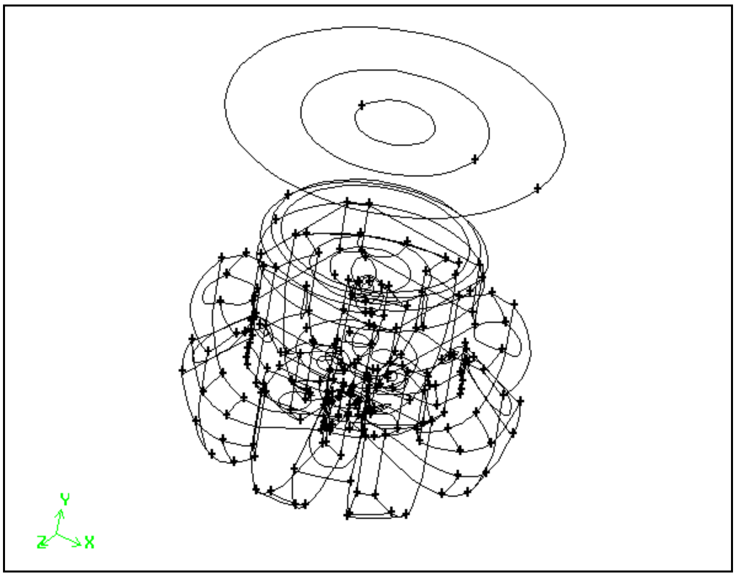


Figure 12-11: Final, cleaned-up model geometry

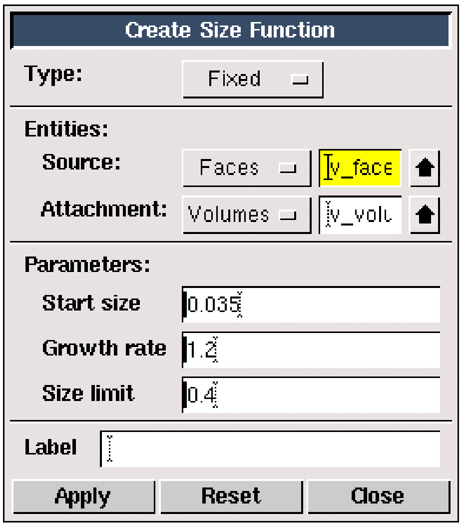
Step 9: Apply Size Functions to Control Mesh Quality

Highly skewed elements adversely affect numerical computations for which the mesh is created. GAMBIT *includes several features that allow you to control the mesh, one of which is the application of size functions. For example, size functions can be used to specify the rate at which volume mesh elements change in size in proximity to a specified boundary. In this step, you will apply size functions to four faces of the drill-bit geometry and, thereby, control the size of the nearby mesh elements to eliminate the skewed elements.*

1. Specify a size function and apply it to four faces of the model.

TOOLS  → **SIZE FUNCTIONS**  → **CREATE SIZE FUNCTION** 

This command sequence opens the **Create Size Function** *form.*



a) Retain the **Type:**Fixed option.

(<u>NOTE</u>: In addition to the "fixed" size function illustrated in this tutorial, GAMBIT provides "curvature," "proximity," and "meshed" size functions. Curvature and proximity size functions are useful for controlling the mesh in regions of high curvature and small gaps, respectively. Meshed size functions use existing meshes to determine the size-function start size. See Section 5.2.2 of the GAMBIT *Modeling Guide.)*

b) On the **Entities:Source** option button, select the Faces option.

c) In the Faces list box, select the four faces shown (shaded) in Figure 12-12.

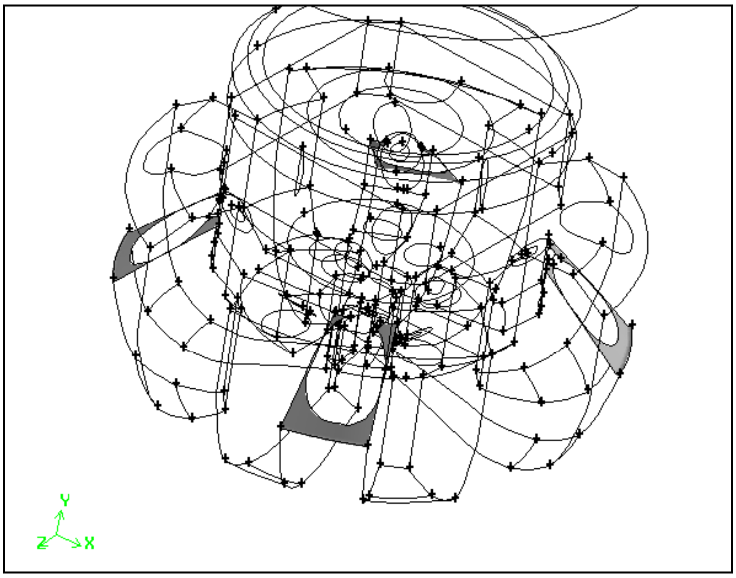


Figure 12-12: Faces on which to apply size functions

d) On the **Entities:Attachment** option button, select the Volumes option.

e) In the Volumes list box, select the volume.

f) In the **Start size** text box, enter the value `0.035`.

g) In the **Growth rate** text box, enter the value, `1.2`.

h) In the **Size limit** text box, enter the value, `0.4`.

i) Click **Apply** to create the size function.

When applying the size function, GAMBIT *displays a message in the* **Transcript** *window indicating that the use of virtual entities as source entities in the size-function definition can cause problems when evaluated during background-grid generation. The message represents a warning only and can be ignored in this case.* GAMBIT *allows you to view the size function by means of the* **View Size Function** *command on the* **Size Function** *toolpad.*

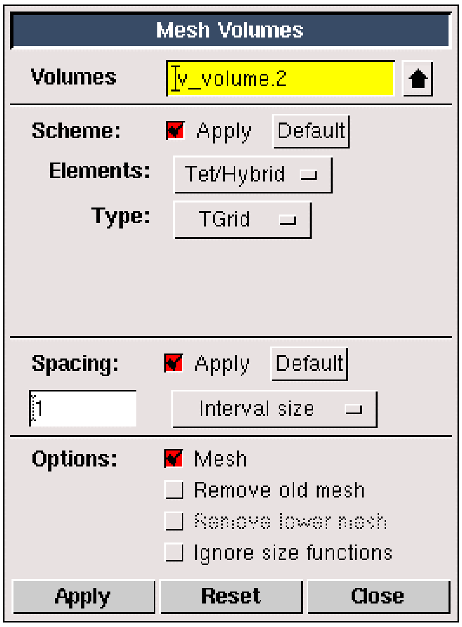
Step 10: Mesh the Volume

After the imported geometry is cleaned up and the size-function is created and attached, you can mesh the geometry using an unstructured, tetrahedral mesh.

1. Mesh the drill-bit volume.

MESH → **VOLUME** → **MESH VOLUMES**

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



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

b) Select the volume.

GAMBIT *automatically selects the* **Scheme:Elements:**Tet/Hybrid *and* **Scheme: Type:**TGrid *options.*

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

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

e) In the **Spacing** text box, retain the default value of `1`.

The size function you attached to the volume in the previous step will override the **Spacing** *specifications.*

f) Click **Apply**.

Figure 12-13 shows the final meshed volume.

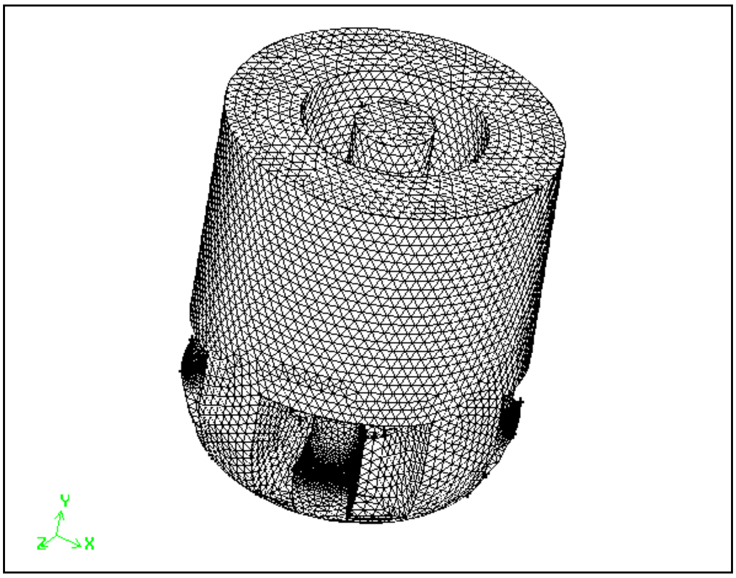
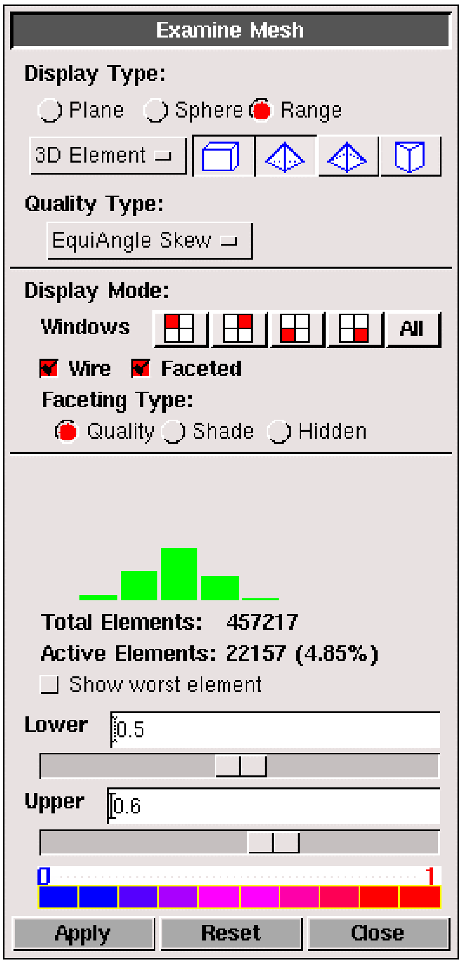


Figure 12-13: Meshed drill-bit volume

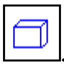
Step 11: Examine the Volume Mesh

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

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



a) Select **Range** under **Display Type** at the top of the form.

The **3D Element** *type selected by default at the top of the form is a brick* . *You will not see any mesh elements in the graphics window when you first open the* **Examine Mesh** *form, because there are no hexahedral elements in the mesh.*

b) Left-click the tetrahedron icon  next to **3D Element** near the top of the form.

The tetrahedral mesh elements will now be visible in the graphics window.

c) Select or retain EquiAngle Skew from the **Quality Type** option menu.

d) Left-click the histogram bars that appear at the bottom of the **Examine Mesh** form to highlight elements in any given quality range.

Figure 12-14 shows the graphics window that results for elements with EquiAngle Skew *values between 0.5 and 0.6.*

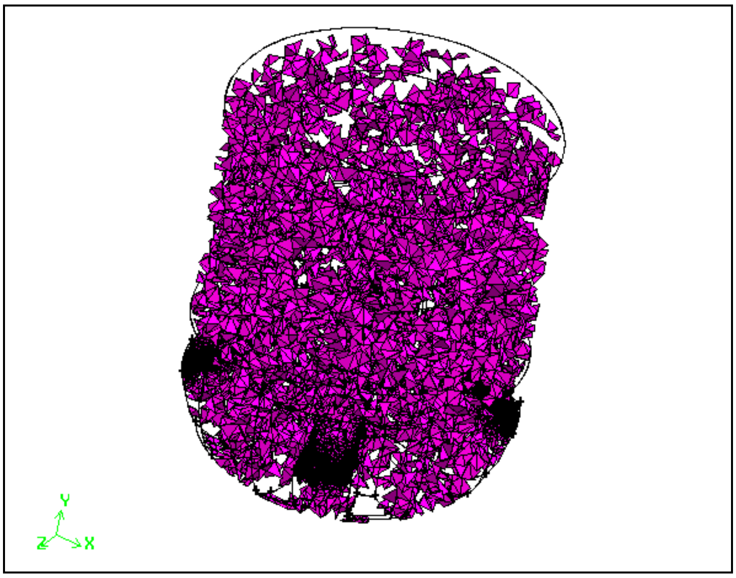


Figure 12-14: Elements with EquiAngle Skew values between 0.5 and 0.6

e) On the **Examine Mesh** form, click **Close** to close the form.

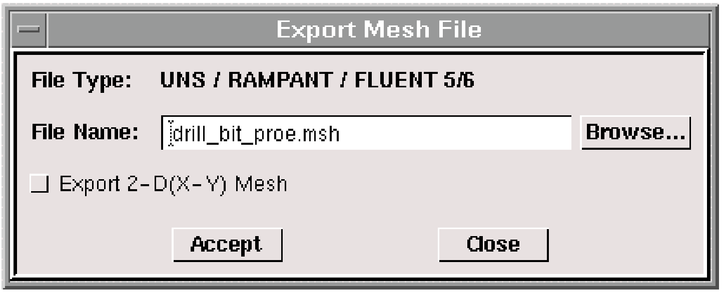
Step 12: Export the Mesh and Close GAMBIT

1. Export a mesh file.

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

   File → **Export** → **Mesh...**

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



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

   iii. Click **Accept**.

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

2. Save the GAMBIT session and close GAMBIT

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

   File → **Close**

   This command sequence opens the **Close** *form.*

The **Close** *option is available only when* GAMBIT *is launched from within the* Pro/ENGINEER *program.*

b) Click **Yes** to save the current session and return to Pro/ENGINEER.

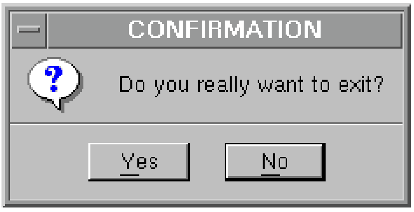
The Pro/ENGINEER *user interface redisplays in the foreground, and* GAMBIT *continues to run in the background.*

Step 13: Exit Pro/ENGINEER and GAMBIT

1. Exit the Pro/ENGINEER program.

File → Exit…

This command sequence opens the Pro/ENGINEER **CONFIRMATION** *form.*



a) Click Yes to exit Pro/ENGINEER.

When you exit Pro/ENGINEER, GAMBIT *is still running, and the* **Close** *form is still open.*

b) On the GAMBIT Close form, click **No** to exit GAMBIT.

12.5 Summary

This tutorial demonstrates the direct import of CAD geometry into GAMBIT and the operations that are sometimes required to render such geometry amenable to GAMBIT meshing operations. A comparison of the procedures described here with those of Tutorial 11 illustrates the advantages of direct CAD import versus import of CAD geometry by means of intermediate files—for example, STEP files. Specifically, the directly imported geometry does not include the very short edges that result from geometry import by means of the STEP file.

13. CATALYTIC CONVERTER

In this tutorial you will import an IGES file describing the geometry of an automotive catalytic converter. You will clean up the geometry using GAMBIT clean-up tools and mesh the geometry using a hex-core meshing scheme.

In this tutorial you will learn how to:

- Import an IGES file
- Use a heal-geometry operation to repair the imported geometry
- Use several clean-up operations to clean up the geometry
- Apply size functions to control mesh quality
- Mesh a volume with a hex-core mesh
- Prepare the mesh to be read into FLUENT 5/6

NOTE: You can reproduce the perspectives of the figures in this tutorial by means of `window matrix` commands available in a journal file named "`tg13_figures.jou`," which is included in the "`help/tutfiles`" online help directory. To exactly reproduce the perspective of any figure, you can open the journal file and execute the window matrix command associated with the figure. For example, the following command reproduces the perspective of the model shown in Figure 13-2.

```
window matrix 1 entries \
  -0.8254742026329   -0.5580788850784   -0.08449375629425 \
  35.9133605957      -0.03749995678663  -0.09513910114765 \
   0.9947574734688  -52.56211853027     -0.563191473484   \
   0.8243152499199    0.05760711431503  16.31116676331    \
 -69.85343933105     93.40626525879    -87.0938949585     \
  44.18941116333   -189.0608520508     189.0608520508
```

13.1 Prerequisites

This tutorial assumes you have worked through Tutorials 1 and 6 and are familiar with use of the GAMBIT GUI and general clean-up operations.

13.2 Problem Description

Figure 13-1 shows the geometry to be imported and meshed in this tutorial. This geometry represents a complex portion of a catalytic converter, including four circular ports through which exhaust gases flow. The imported geometry consists of many edges and faces that you will eliminate before generating the volume mesh. The mesh itself will consist of a core of hexahedral elements surrounded by layers of pyramidal and tetrahedral elements.

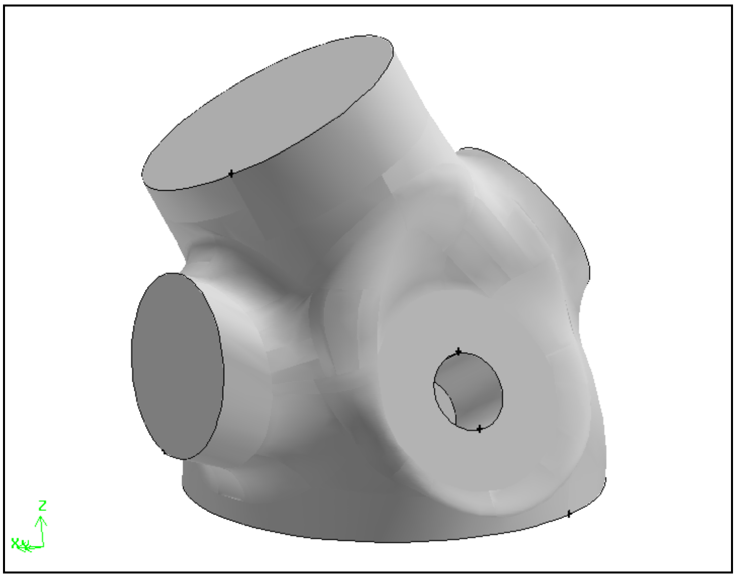


Figure 13-1: Catalytic-converter geometry

13.3 Strategy

In this tutorial, you will create a “hex-core” mesh in a catalytic-converter geometry imported as an IGES file. This tutorial illustrates many of the steps you can take in GAMBIT to prepare imported CAD geometry for meshing. In this case, the imported geometry contains many faces and edges that can complicate meshing by unnecessarily constraining the mesh-generation process. In addition, the imported geometry contains a “bad” face (with convoluted geometry) and an overlapping face. After using a GAMBIT heal-geometry operation to simplify the imported geometry, you will clean up the geometry using the clean-up tools available in GAMBIT. You will then pave four large, circular faces on the geometry and generate a mesh consisting of a core of hexahedral elements surrounded by pyramidal and tetrahedral elements.

13.4 Procedure

1. Copy the file

   path/Fluent.Inc/gambit2.*x*/help/tutfiles/catalytic_conv.igs

   (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 "Catalytic_Converter".

Step 1: Select a Solver

1. Choose the solver from the main menu bar:

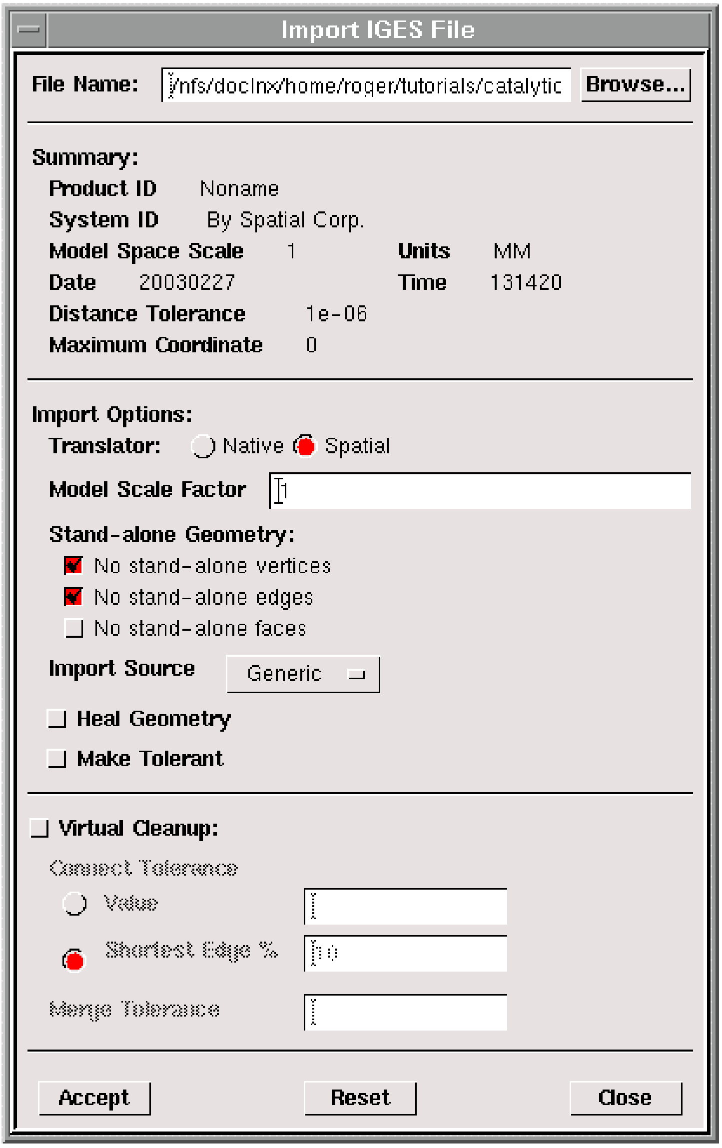
   Solver → **FLUENT 5/6**

   The choice of a solver dictates the options available in various forms (for example, the boundary types available in the **Specify Boundary Types** *form). For some systems,* **FLUENT 5/6** *is the default solver. The solver currently selected is shown at the top of the* GAMBIT *GUI.*

Step 2: Import the IGES File

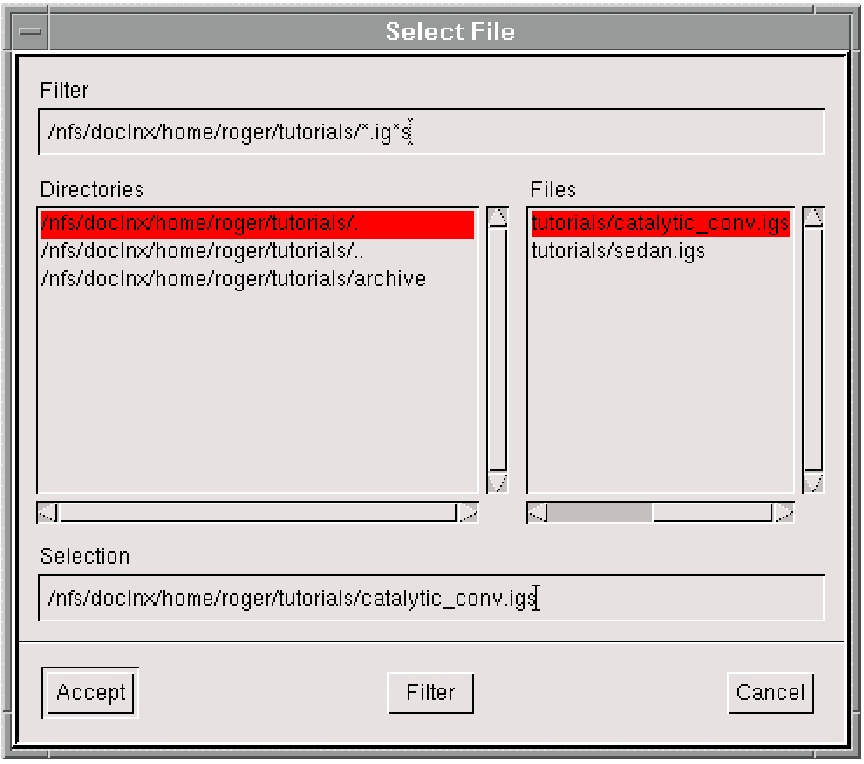
File → **Import** → **IGES ...**

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



1. Click on the **Browse...** button.

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



   a) Select `catalytic_conv.igs` in the **Files** list.

   b) Click Accept in the **Select File** form.

2. On the **Import IGES File** form, under **Stand-alone Geometry**, select the No stand-alone vertices and No stand-alone edges options.

3. Unselect the **Make Tolerant** option.

4. Click **Accept**.

   The IGES file for the catalytic converter will be read into GAMBIT *(see Figure 13-2).*

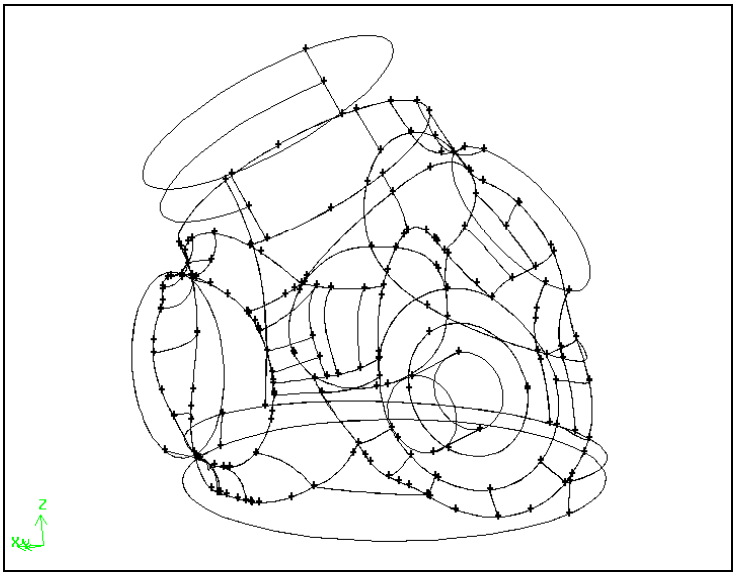
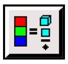


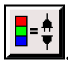
Figure 13-2: Imported catalytic-converter geometry

Step 3: Attempt to Heal the Geometry

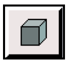
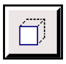
The imported geometry is somewhat "dirty"—that is, it contains gaps between face boundaries that can prevent the creation of a meshable volume. In this step, you will attempt to "heal" the geometry to eliminate the gaps.

1. View entity connectivity in the model.

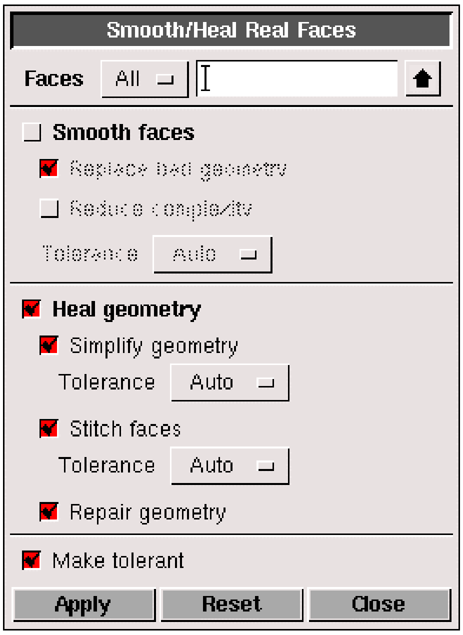
   a) Click the **SPECIFY COLOR MODE** command button  in the **Global Control** toolpad to change to the graphics display to connectivity-based coloring.

      The **SPECIFY COLOR MODE** *command button will change to* . *When* GAMBIT *is in the connectivity display mode, the model is displayed with colors based on connectivity between entities rather than based on entity types. In this case, all edges are displayed as orange, indicating that the faces they bound are not connected to each other.*

2. Attempt the geometry healing operation.

   GEOMETRY  → **FACE**  → **SMOOTH/HEAL REAL FACES** 

   This command sequence opens the **Smooth/Heal Real Faces** *form.*



a) Select All from the option menu to the right of **Faces**.

b) Unselect the **Smooth faces** option.

c) Select the **Heal geometry** option.

d) Select the Simplify geometry option.

This option specifies whether or not GAMBIT *converts NURBS data to analytic data, where possible, to within a specified tolerance.*

e) Retain the Stitch faces option.

This option specifies that GAMBIT *stitches faces to create a volume during the healing operation.*

f) Select the Repair geometry option.

This option specifies that GAMBIT *attempts to repair the model geometry by recomputing and/or extending surface and curve definitions so that the model "fits together" properly.*

g) Click **Apply**.

Most of the edges turn blue, indicating that the healing operation is successful at connecting most faces; however, several edges remain orange. Three sets of orange edges represent circular holes in the model that will be cleaned up in Step 7. The others indicate the existence of one "bad" face and one overlapping face in the model (see Figure 13-3). Before you can successfully heal the geometry, you must eliminate the bad face and the overlapping face.

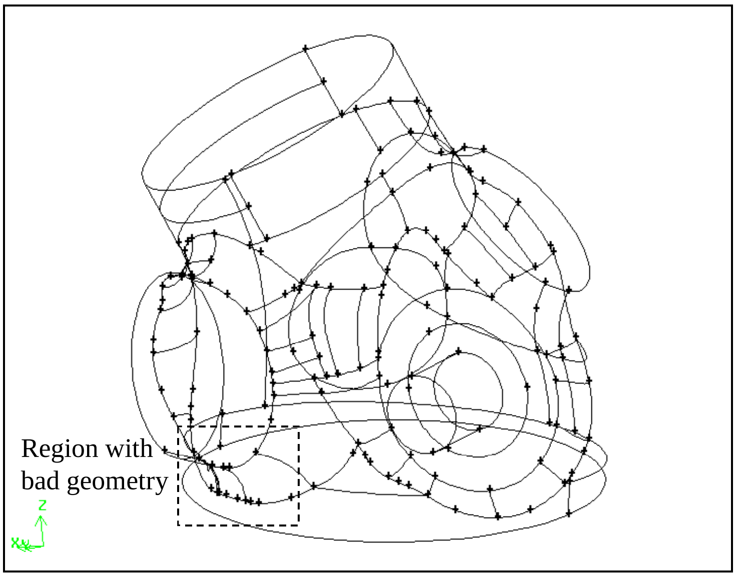


Figure 13-3: Region of the model containing bad and overlapping faces

Step 4: Eliminate the Bad and Overlapping Faces

The imported IGES geometry contains two faces that inhibit the geometry healing operation. In this step, you will identify and eliminate both faces.

1. *Ctrl*-left-drag the mouse to zoom in on the region containing the bad and overlapping faces (shown as shaded faces in Figure 13-4).

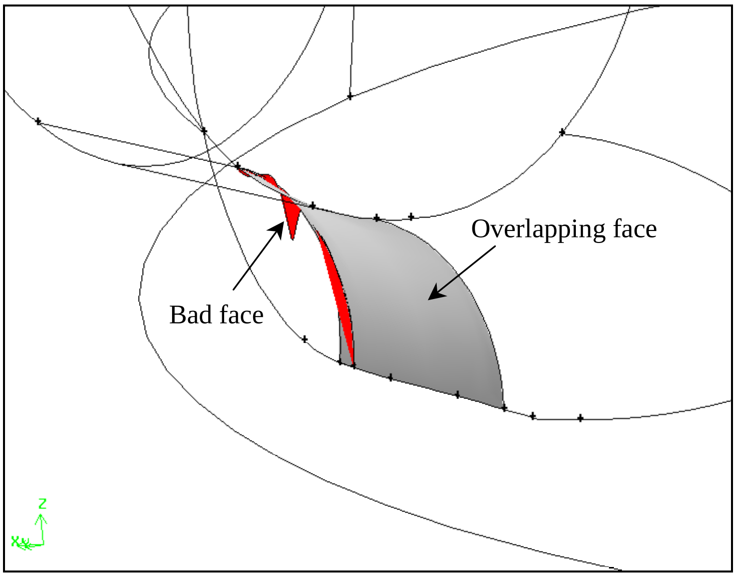


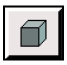
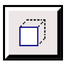
Figure 13-4: Zoomed view of bad and overlapping faces

The shaded faces shown in Figure 13-4 must be deleted, rather than "cleaned up," before the geometry can be correctly healed.

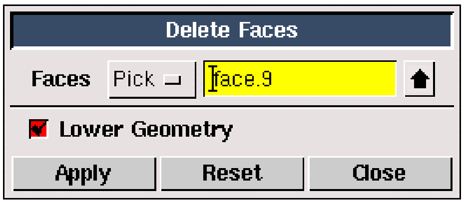
2. Click the **UNDO** command button  on the **Global Control** toolpad to undo the healing operation.

   All edges will turn orange again, indicating that the connections made during the first healing attempt are undone.

3. Delete the bad and overlapping faces.

GEOMETRY  → **FACE**  → **DELETE FACES** 

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



a) Select the bad and overlapping faces shown in Figure 13-4, above.

b) Click **Apply**.

GAMBIT *removes the faces from the model, leaving the local geometry shown in Figure 13-5.*

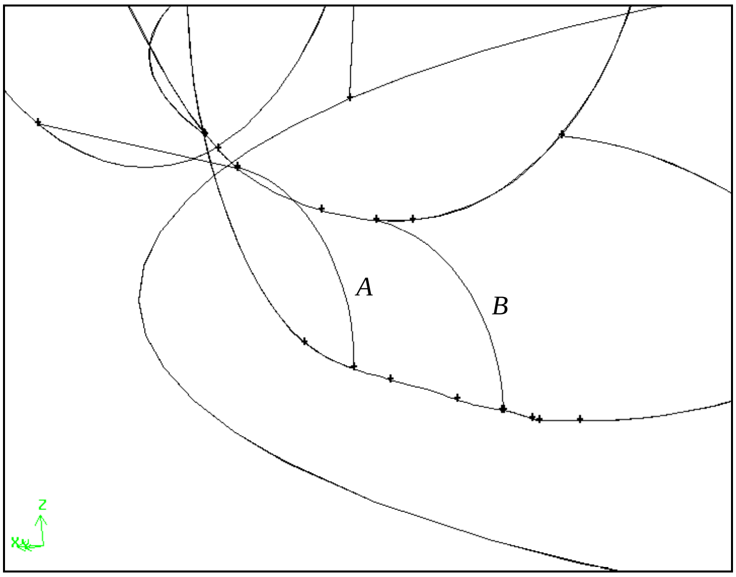


Figure 13-5: Local geometry after face deletion

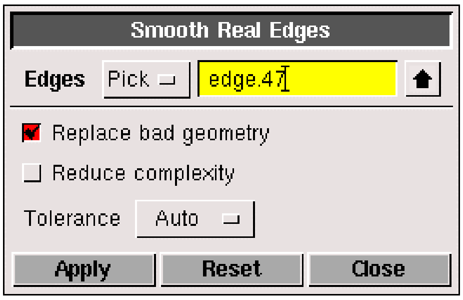
Step 5: Smooth the Discontinuous Edge

Before you can again attempt to heal the geometry, you must create a new face to replace the deleted overlapping face. However, one of the edges remaining after the face-delete operation in Step 4 possesses "G1" discontinuity, which is usually associated with sharp kinks in the curve that defines the edge. Such discontinuities can sometimes complicate operations that employ the edge as a boundary edge of a skin-surface face (see Step 6). In this step, you will remove any discontinuities from the edge.

1. Remove discontinuities from one of the edges shown in Figure 13-5.

   GEOMETRY → **EDGE** → **SMOOTH REAL EDGES**

   This command sequence opens the **Smooth Real Edges** *form.*



   a) In the graphics window, select (pick) edge *B* shown in Figure 13-5, above.

   b) Retain the Replace bad geometry option.

   c) Click **Apply**.

      GAMBIT *removes G1 discontinuities from the edge.*

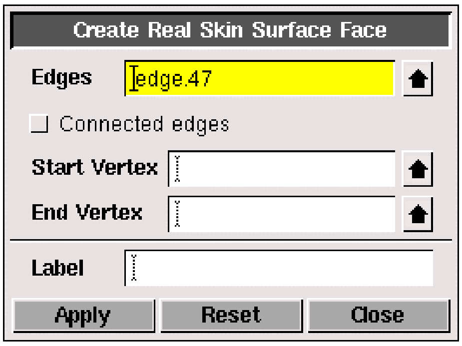
Step 6: Replace the Overlapping Face

After the edge discontinuities are removed, you can create a new face to replace the face deleted in Step 4. The new face will be created using a skin-surface face creation procedure.

1. Create a skin-surface face.

   GEOMETRY → **FACE** → **FORM FACE** R Skin Surface

   This command sequence opens the **Create Real Skin Surface Face** *form.*



   a) In the graphics window, select (pick) edge *A* shown in Figure 13-5, above.

   b) Select (pick) edge *B* shown in Figure 13-5.

      The skin-surface face-creation operation requires that the "sense" directions of all edges used to create a face point in the same direction. In this case, the default senses of edges A and B point in opposite directions. To correctly specify these edges for the procedure, you must reverse the sense direction of one of the two edges.

   c) *Shift*-middle-click edge *B* to reverse its sense direction.

      GAMBIT *indicates the change in the sense direction of edge B by reversing the arrow displayed in the middle of the edge.*

   d) Click **Apply**.

      GAMBIT *creates a new face to replace the deleted overlapping face (see Figure 13-6).*

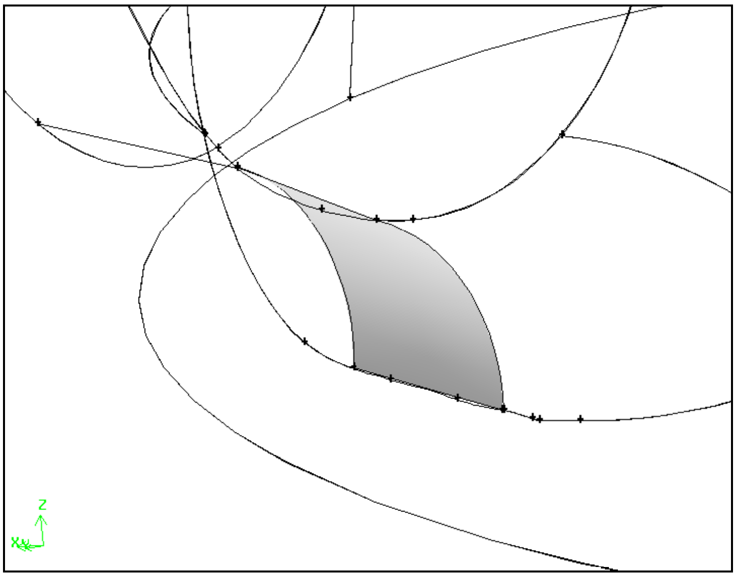


Figure 13-6: New skin-surface face (shaded)

e) Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full catalytic converter in the graphics window.

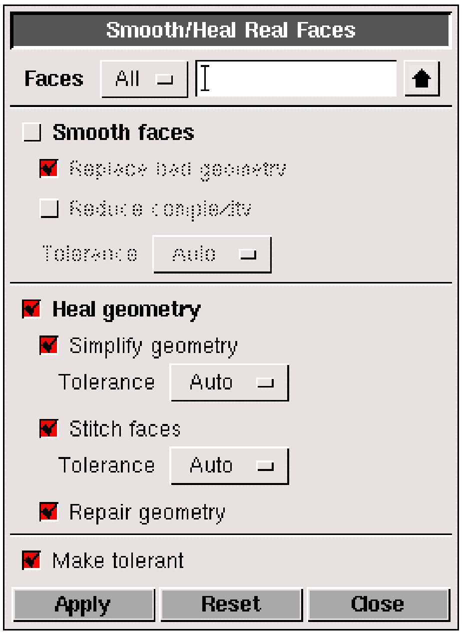
Step 7: Attempt Again to Heal the Geometry

Now that the bad and overlapping faces have been removed from the model, you can attempt again to heal the geometry.

1. Attempt the geometry healing operation on the modified geometry.

   GEOMETRY → **FACE** → **SMOOTH/HEAL REAL FACES**

   This command sequence reopens the **Smooth/Heal Real Faces** *form.*



   a) Retain all options specified on the previous healing attempt.

   b) Click **Apply**.

      This time, GAMBIT *successfully heals the geometry to create the geometry shown in Figure 13-7. The only remaining unconnected edges are those that define the three circular holes in the model (see Step 8: Clean Up Holes in the Model).*

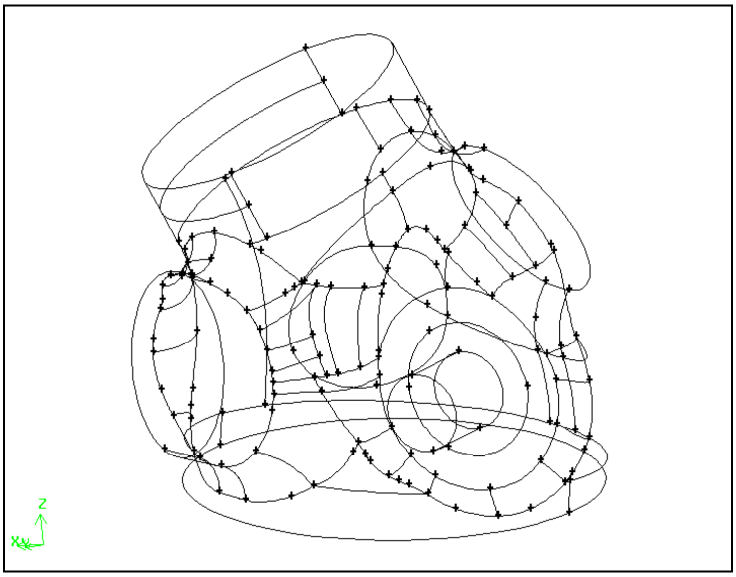


Figure 13-7: Healed catalytic-converter geometry

Although the healing operation has successfully connected the faces, the healed geometry still contains many features that can inhibit meshing—such as short edges and sharp angles. To facilitate meshing, you can remove such features using the GAMBIT *clean-up tools.*

Step 8: Clean Up Holes in the Model

In this step, you will clean up "holes" in the model (see Figure 13-8) to create circular faces that can be stitched together with other faces in the cleaned-up model to create an enclosed, meshable volume.

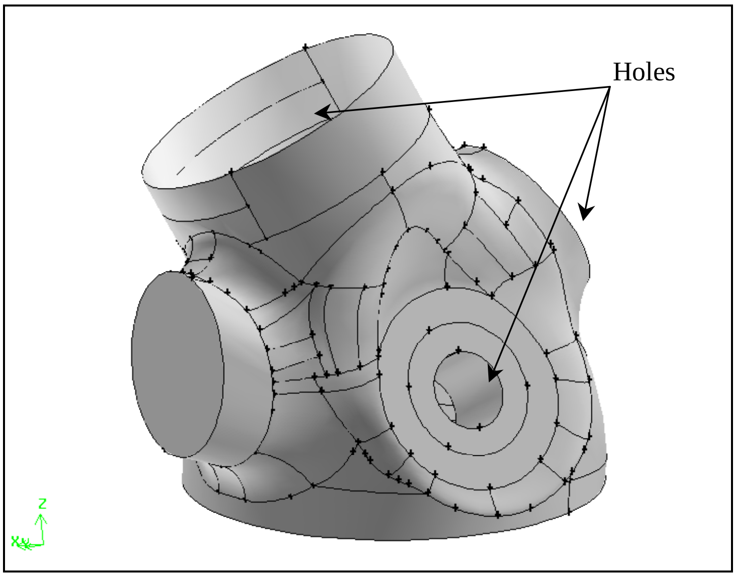
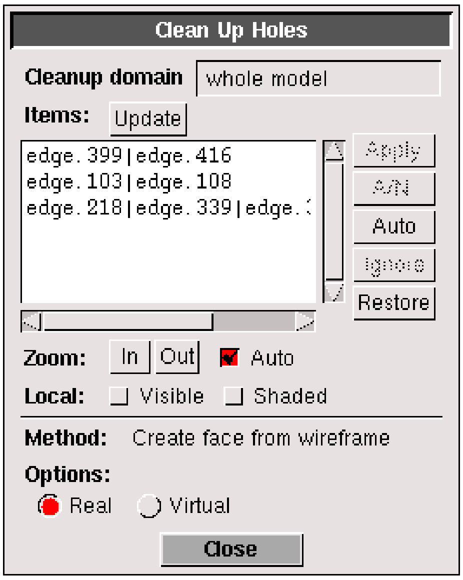


Figure 13-8: Circular holes in catalytic-converter geometry

1. Clean up holes that constitute the catalytic converter inlet and outlet ports.

   TOOLS → **CLEANUP** → **CLEAN UP HOLES**

   This command sequence opens the **Clean Up Holes** *form.*



When you open any of the clean-up forms, such as the **Clean Up Holes** *form,* GAMBIT *automatically sets the graphics window color mode to display colors based on connectivity rather than topology. In addition,* GAMBIT *automatically sets the graphics window pivot function to the user-specified pivot mode.*

GAMBIT *automatically populates the* **Items** *list with the edge sets that constitute holes in the model.*

a) Select the first item (edge set) in the **Items** list.

GAMBIT *automatically highlights the set of edges shown in Figure 13-9. (NOTE: Due to slight differences in entity numbering between computer platforms, the entity numbers shown in Figure 13-9, and in all subsequent figures that include entity labels, might differ from those displayed in the GAMBIT graphics window.)*

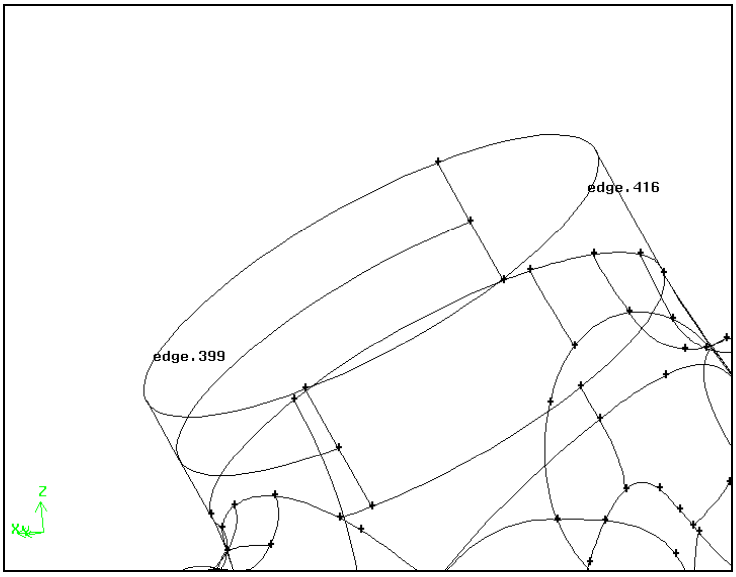


Figure 13-9: Set of edges comprising the first hole listed in the **Items** list

The **Clean Up Holes** *form provides only one method for eliminating holes—* Create face from wireframe.

b) Click the A/N pushbutton in the vertical array of pushbuttons located to the right of the **Items** list.

The A/N *("Apply/Next") pushbutton applies the* Create face from wireframe *method to remove the hole from the model then updates the* **Items** *list and automatically selects the next hole in the* **Cleanup domain**.

c) Click A/N again to eliminate the next hole in the **Cleanup domain**.

GAMBIT *eliminates the selected face and automatically highlights the next hole in the* **Cleanup domain**.

d) Click Apply to eliminate the last remaining hole in the **Cleanup domain**.

e) Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full catalytic converter in the graphics window.

Figure 13-10 shows a shaded view of the full geometry, illustrating that the holes have been converted to faces.

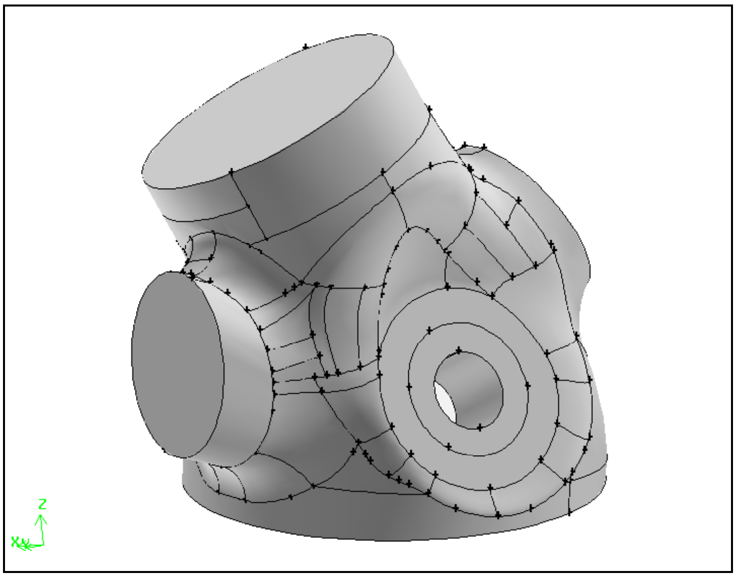
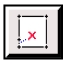


Figure 13-10: Geometry after hole clean-up operations

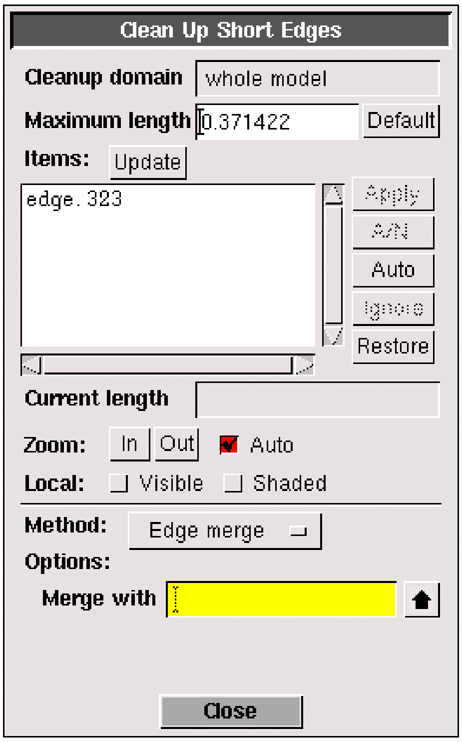
Step 9: Clean Up Short Edges

In this step, you will use manual and automatic clean-up procedures to eliminate short edges in the model.

1. Manually eliminate the shortest edge in the model.

   TOOLS  → **CLEANUP**  → **CLEAN UP SHORT EDGES** 

   This command sequence opens the **Clean Up Short Edges** *form.*



   a) Click the Default pushbutton located on the right side of the **Maximum length** text box.

      When you click the Default *pushbutton,* GAMBIT *displays the* **Maximum length** *of edges to be included in the* **Items** *list and populates the* **Items** *list with all edges in the* **Cleanup domain** *that meet the* **Maximum length** *criterion. By default, the* **Maximum length** *value is 10 times greater than the arc length of the shortest edge in the* **Cleanup domain**.

b) Select the edge displayed in the **Items** list.

GAMBIT *highlights and displays the edge in the graphics window as shown (in a manually-zoomed view) in Figure 13-11.*

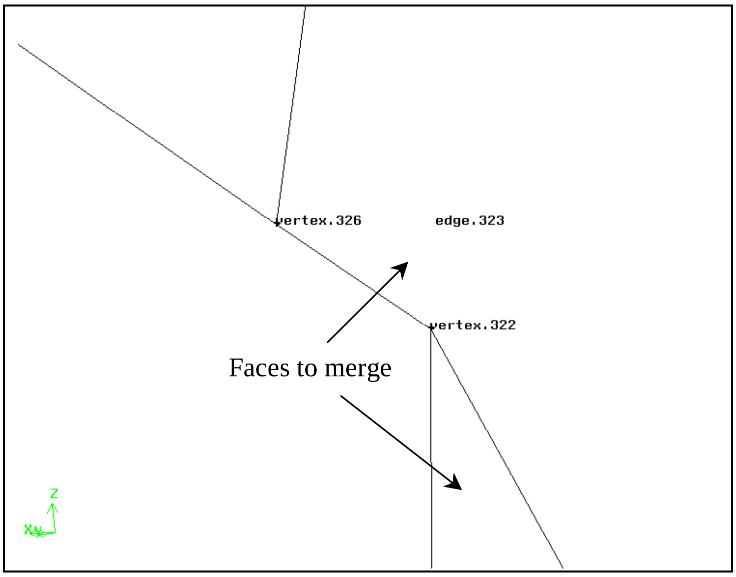


Figure 13-11: Shortest edge in the model

GAMBIT *provides three methods for eliminating short edges*—Vertex connect, Edge merge, *and* Face merge. *In this case,* GAMBIT *automatically selects the* Face merge *option and populates the* **Faces to merge** *pick list with two candidate faces for the merge operation.*

c) Click Apply in the vertical array of pushbuttons located to the right of the **Items** list.

GAMBIT *merges the specified faces to create the configuration shown in Figure 13-12, thereby eliminating the shortest edge in the model.*

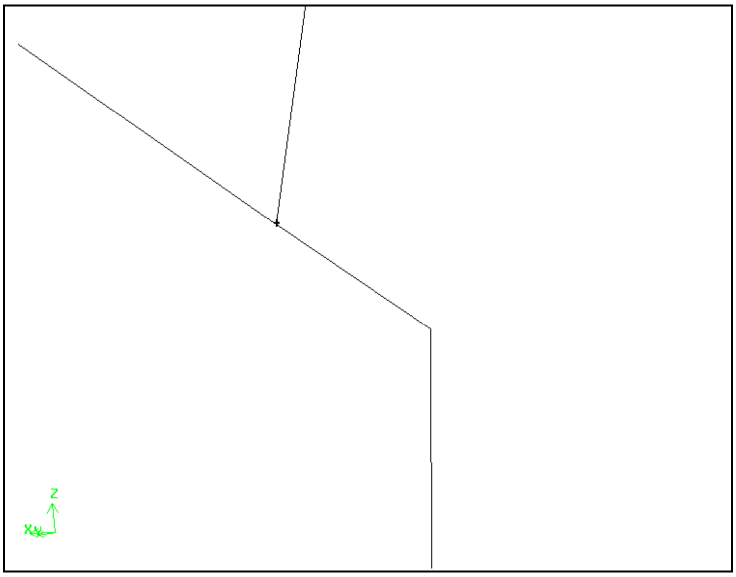


Figure 13-12: Cleaned-up local geometry after the Face merge operation

Now that you have removed the shortest edge in the model, you will use an automatic operation to clean up the remaining short edges.

2. Automatically clean up the remaining shortest edges.

   a) Click the Default pushbutton located next to the **Maximum length** text box.

      GAMBIT *updates the* **Maximum length** *value based on the current set of edges in the* **Cleanup domain** *and updates the* **Items** *list.*

   b) Unselect the **Zoom:**Auto option.

      By default, GAMBIT *automatically zooms in on any edge currently selected for clean-up and displays the labels of the edge and its endpoint vertices in the graphics window. If you retain the* **Zoom:**Auto *option in this case,* GAMBIT *will zoom in on every edge as it is automatically removed from the model, thereby making it difficult to follow the clean-up operation in the graphics window.*

   c) Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full catalytic converter in the graphics window.

d) On the **Clean Up Short Edges** form, click Auto in the vertical array of pushbuttons located to the right of the **Items** list.

GAMBIT *uses edge- and face-merge operations to automatically remove edges from the model. Figure 13-13 shows the full geometry after removing the short edges.*

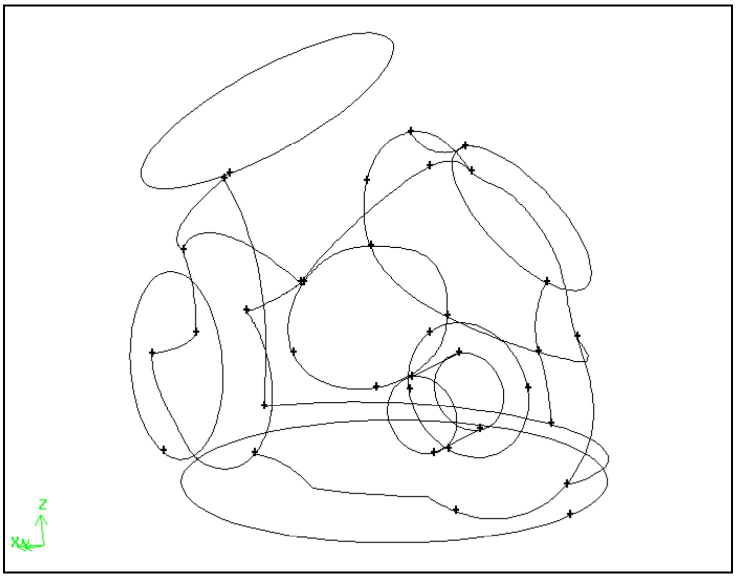


Figure 13-13: Geometry after cleaning up shortest edges

Step 10: Clean Up Sharp Angles

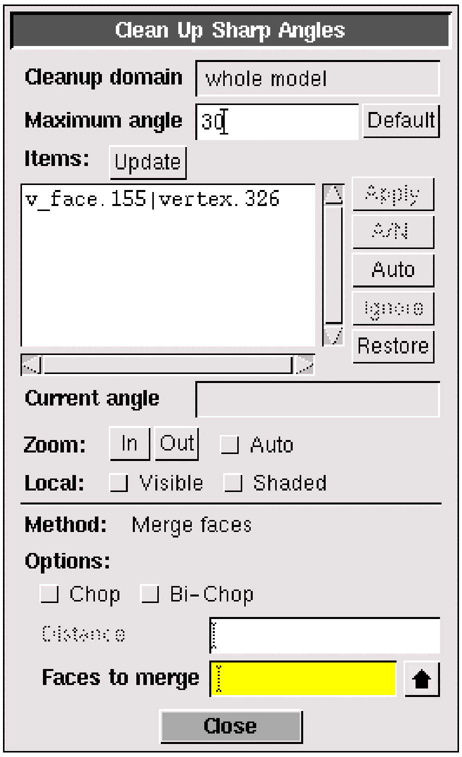
In this step, you will us a clean-up procedures to eliminate a "sharp angle" in the model. For the purposes of the geometry cleanup operations, a "sharp angle" is defined as an edge pair that meets the following criteria:

- *The edge pair shares a common endpoint vertex and serves as part of the boundary for an existing face.*
- *One of the edges in the sharp-angle edge pair serves as a common boundary edge between its bounded face and an adjacent face.*
- *The angle between the edges in the pair (computed at their common endpoint vertex) is less than a specified angle.*

1. Identify and eliminate a sharp angle in the model.

   TOOLS → **CLEANUP** → **CLEAN UP SHARP ANGLES**

   This command sequence opens the **Clean Up Sharp Angles** *form.*



a) In the **Maximum angle** text box, enter the value `30`.

b) Click the **Items:**Update pushbutton.

GAMBIT *populates the* **Items** *list with one face-vertex pair in the* **Cleanup domain** *that meets the* **Maximum angle** *criterion.*

c) Unselect the **Zoom:**Auto option.

d) Select the face-vertex pair displayed in the **Items** list.

GAMBIT *highlights the geometry that constitutes the specified sharp angle (see Figure 13-14).*

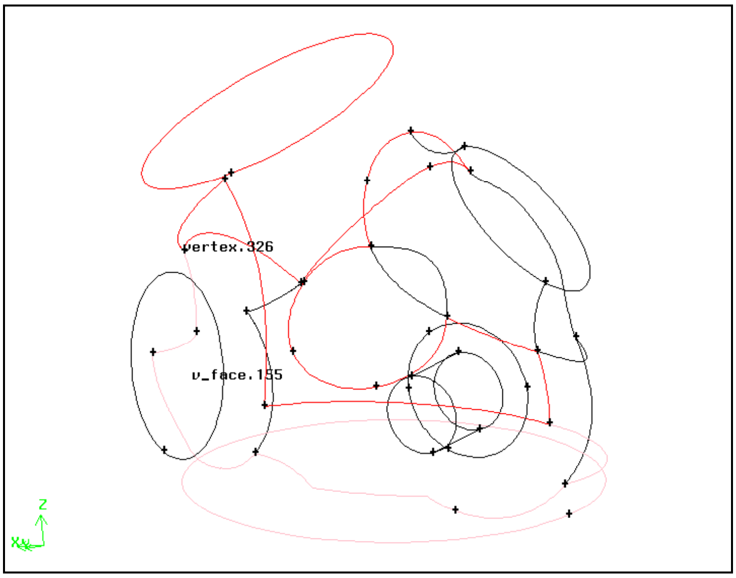


Figure 13-14: Geometry associated with sharp angle to be cleaned up

The **Clean Up Sharp Angles** *operation uses a* Merge faces *procedure to eliminate any sharp angle. In this case,* GAMBIT *automatically populates the* **Faces to merge** *pick list with suggested faces to merge. (For a complete description of the* **Clean Up Sharp Angles** *form, see "Clean Up Sharp Angles" in Section 5.4.2 of the* GAMBIT *Modeling Guide.)*

e) Click the Apply pushbutton in the vertical array of pushbuttons located to the right of the **Items** list.

GAMBIT merges the faces as shown in Figure 13-15.

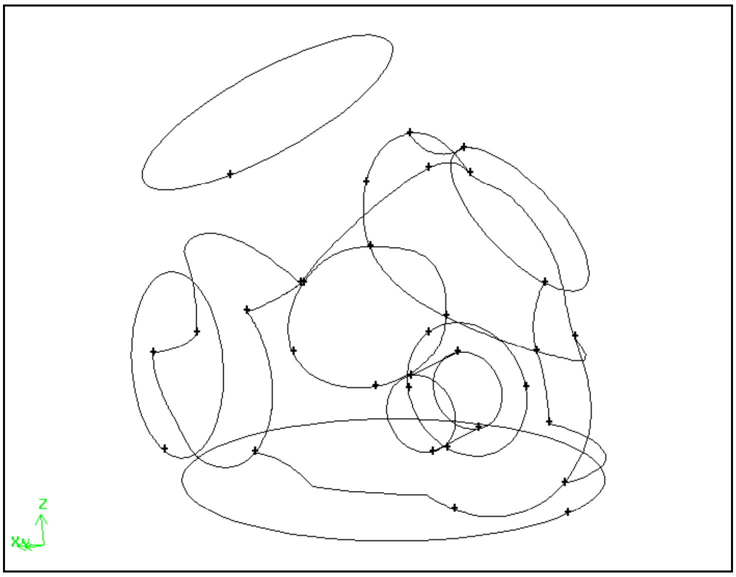


Figure 13-15: Model geometry after sharp-angle clean-up operation

Step 11: Clean Up Large Angles

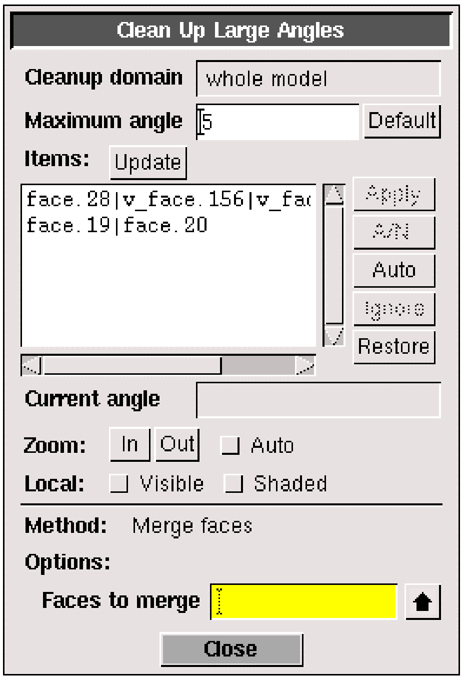
In this step, you will clean-up procedures to eliminate “large angles” in the model. For the purposes of the geometry cleanup operations, a “large angle” is defined by a set of faces that meet the following criteria:

- *All faces in the set are connected to each other.*
- *The angle between the outward-pointing normals for adjacent faces in the set is less than a specified angle.*

1. Identify and eliminate large angles in the model.

   TOOLS  → **CLEANUP**  → **CLEAN UP LARGE ANGLES** 

   This command sequence opens the **Clean Up Large Angles** *form.*



   a) Click the Default pushbutton located on the right side of the **Maximum angle** text box.

GAMBIT *displays the* **Maximum angle** *of angles to be included in the* **Items** *list and populates the* **Items** *list with all face-face sets in the* **Cleanup domain** *that meet the* **Maximum angle** *criterion. By default, the* **Maximum angle** *value is 5.*

b) Unselect the **Zoom:**Auto option.

c) Select the first item in the **Items** list.

GAMBIT *highlights the geometry that constitutes the item to be cleaned up (see Figure 13-16). In this case, the item consists of a set of faces, each of which is connected to the others and forms at least one large angle with others in the set. The* **Clean Up Large Angles** *operation uses a* Merge faces *procedure to eliminate any large angle.*

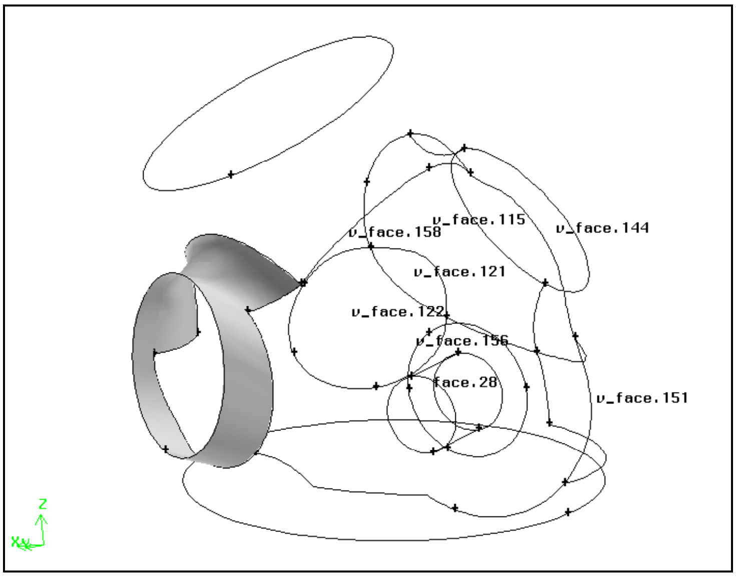


Figure 13-16: Geometry involved in large-angle cleanup operation

d) Click in the **Faces to merge** pick-list field to make it active.

e) Manually select the face shown as shaded in Figure 13-16, above, to include it in the **Faces to merge** list.

f) Click the Apply pushbutton in the vertical array of pushbuttons located to the right of the **Items** list.

GAMBIT *merges the specified faces, thereby creating a continuous surface for the catalytic converter model. Figure 13-17 shows a shaded view of the final, cleaned-up geometry.*

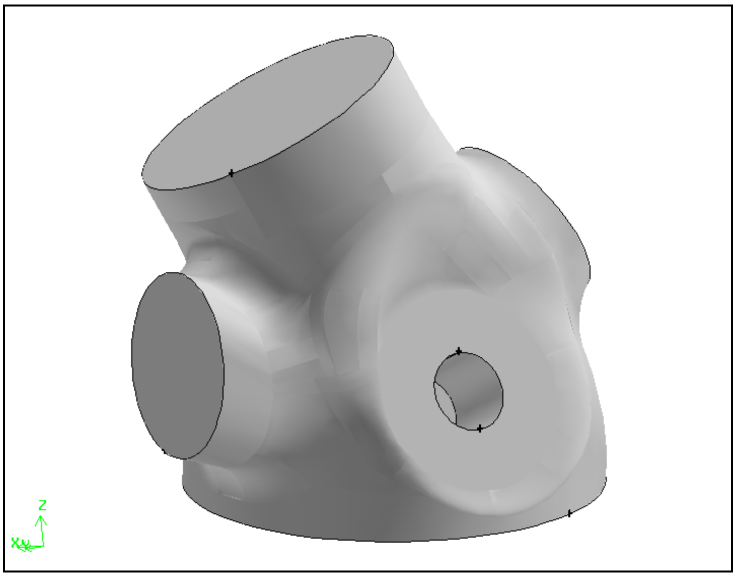
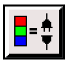


Figure 13-17: Final cleaned-up geometry (shaded view)

g) Click **Close** to close the **Clean Up Large Angles** form.

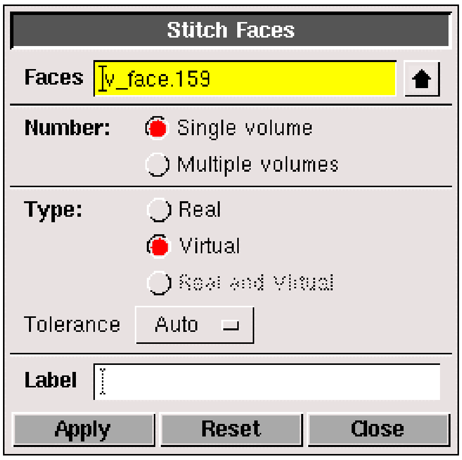
h) Click the **SPECIFY COLOR MODE** command button  in the **Global Control** toolpad to change to the graphics display to entity-based coloring.

Step 12: Stitch the Faces to Create a Volume

1. Stitch the faces in the cleaned-up model to create a volume.

   GEOMETRY → **VOLUME** → **FORM VOLUME**

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



   a) Retain the **Number:**Single volume option.

   b) Select the **Type:**Virtual option.

   c) Select (pick) all remaining faces (eight total) in the model.

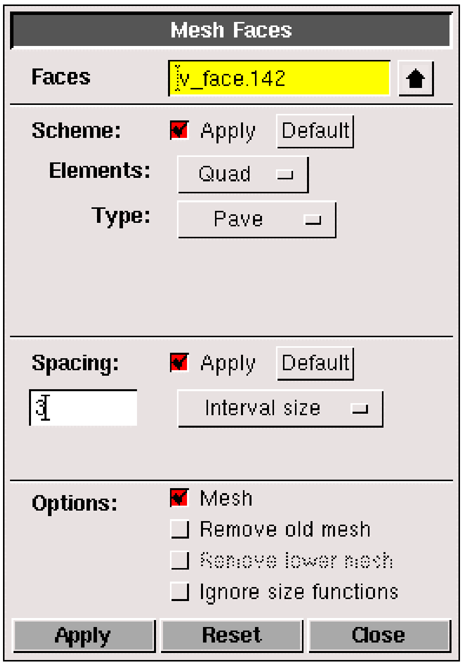
   d) Click **Apply** to create the volume.

Step 13: Mesh the Large Circular Faces

In this step, you will facilitate subsequent volume meshing by paving the four large, circular faces of the model.

1. Create a pave mesh on the four large, circular faces of the catalytic converter.

MESH → **FACE** → **MESH FACES**



a) Select the four large, circular faces (*A*, *B*, *C*, and *D*) shown in Figure 13-18.

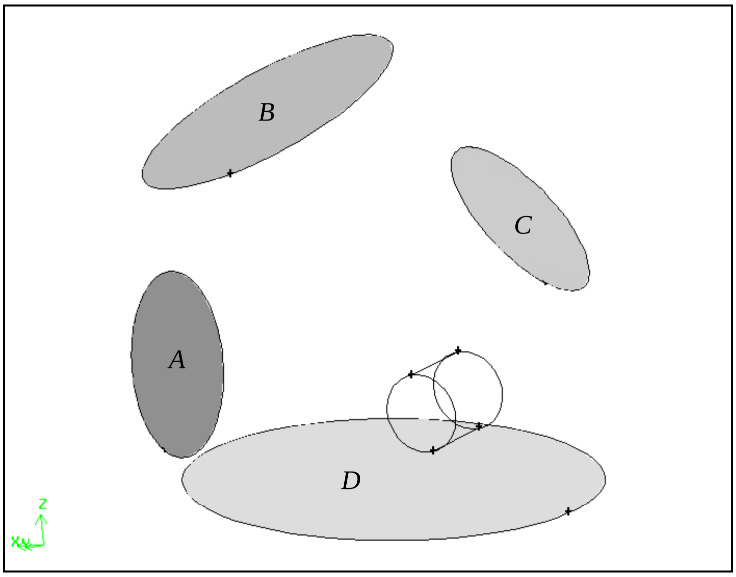


Figure 13-18: Four large, circular faces to be meshed

GAMBIT *automatically selects the* **Elements:**Quad *and* **Type:**Pave *options. For more information on face meshing schemes, see the* GAMBIT *Modeling Guide.*

b) Enter an Interval size of 3 under **Spacing** and click the **Apply** button at the bottom of the form.

GAMBIT *meshes the faces as shown in Figure 13-19.*

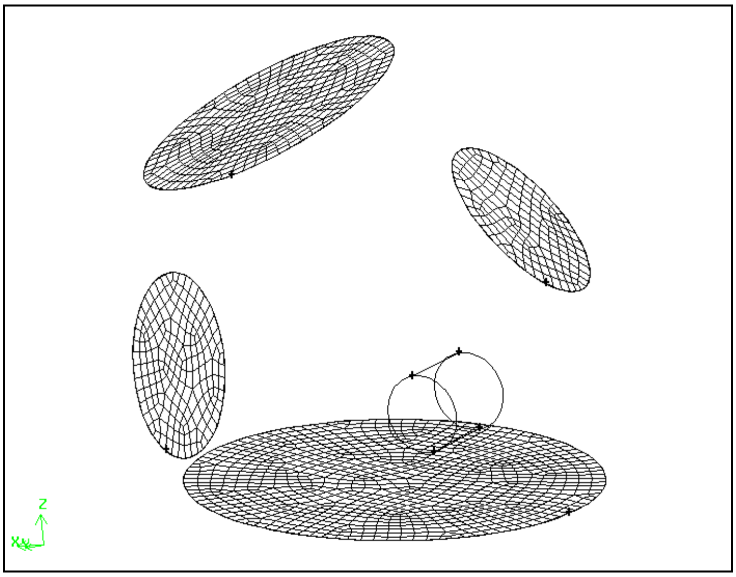


Figure 13-19: Paved meshes on the large, circular faces

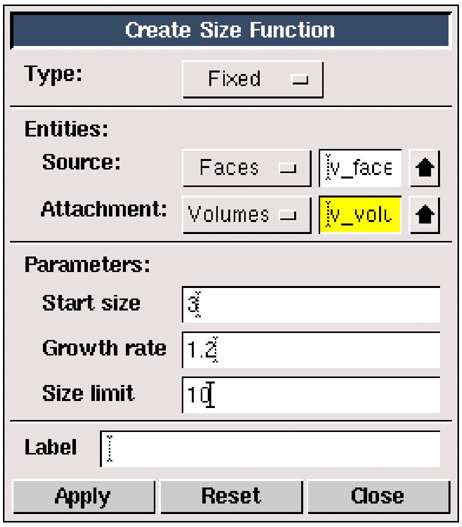
Step 14: Apply Size Functions to Control Mesh Quality

Size functions allow you to control mesh quality and prevent the creation of highly skewed elements. For example, size functions can be used to specify the rate at which volume mesh elements change in size in proximity to a specified boundary. In this step, you will apply size functions to all the faces in the model.

1. Specify a size function and apply it to all faces in the model.

   TOOLS  → **SIZE FUNCTIONS**  → **CREATE SIZE FUNCTION** 

   This command sequence opens the **Create Size Function** *form.*



   a) Retain the **Type:**Fixed option.

      In addition to the "fixed" size function illustrated in this tutorial, GAMBIT provides "curvature" and "proximity" size functions. Curvature and proximity size functions are useful for controlling the mesh in regions of high curvature and small gaps, respectively. See Section 5.2.2 of the GAMBIT Modeling Guide.

   b) On the **Source** option button, select the Faces option.

   c) Click in the Faces list box to make it active and select (pick) all faces in the model.

   d) On the **Attachment** option button, retain the Volumes option.

e) Click in the Volumes list box to make it active, and select the volume.

f) In the **Start size** text box, enter the value `3`.

g) In the **Growth rate** text box, enter the value, `1.2`.

h) In the **Size limit** text box, enter the value, `10`.

i) Click **Apply** to create the size function.

When applying the size function, GAMBIT *displays a message in the* **Transcript** *window indicating that the use of virtual entities as source entities in the size-function definition can cause problems when evaluated during background-grid generation. The message represents a warning only and can be ignored in this case.*

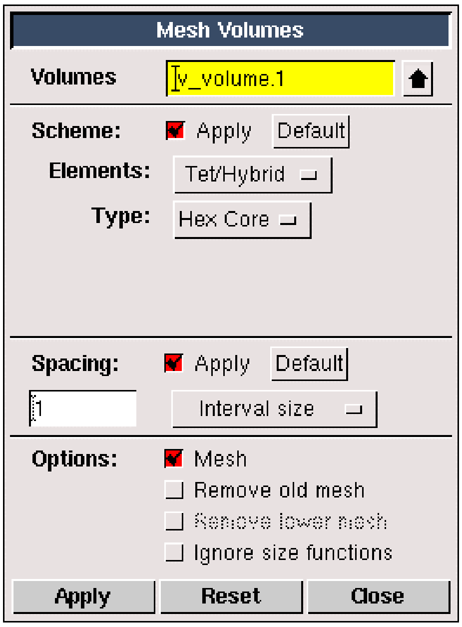
GAMBIT *allows you to view the size function by means of the* **View Size Function** *command on the* **Size Function** *toolpad.*

Step 15: Mesh the Volume

1. Mesh the volume using a hex-core meshing scheme.

   MESH → **VOLUME** → **MESH VOLUMES**

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



   a) Select (pick) the volume in the graphics window.

      GAMBIT *automatically selects the* **Elements:**Tet/Hybrid *and* **Type:**TGrid *options.*

   b) Retain the **Elements:**Tet/Hybrid option.

   c) Select the **Type:**Hex Core option.

   d) Retain the default Interval size of 1 under **Spacing** and click the **Apply** button at the bottom of the form.

      GAMBIT *meshes the volume as shown in Figure 13-20.*

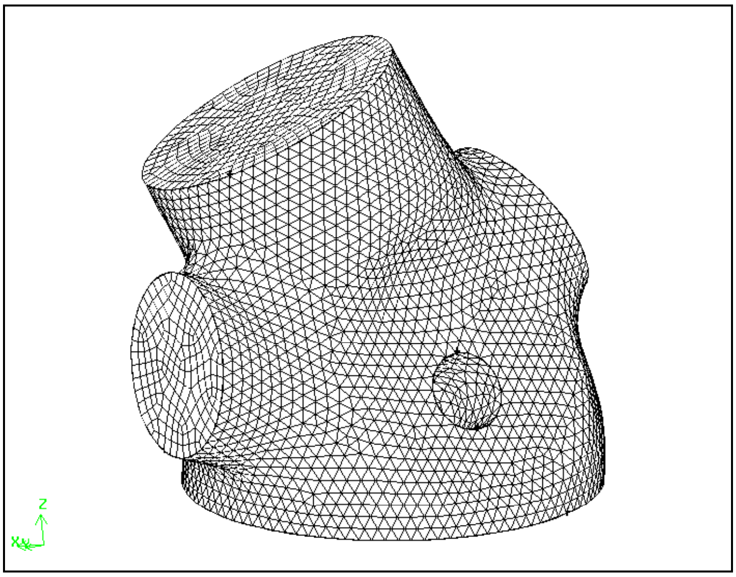
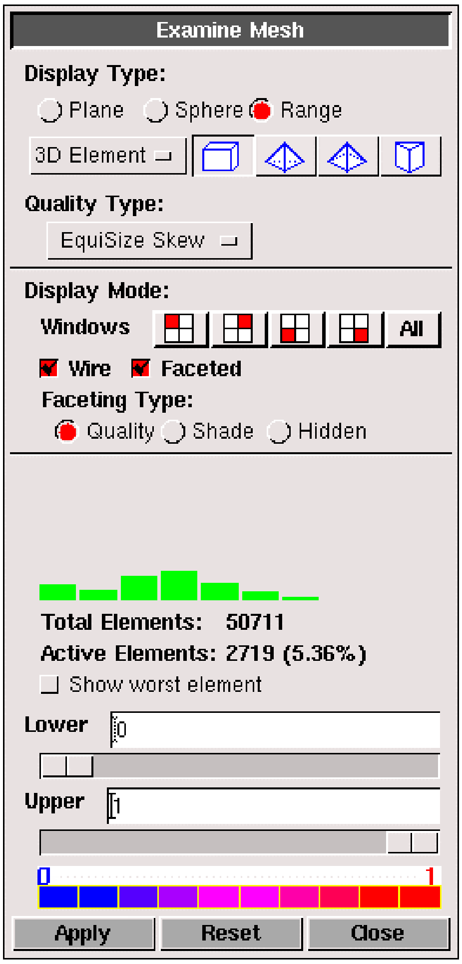


Figure 13-20: Meshed catalytic converter volume

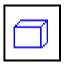
Step 16: Examine the Volume Mesh

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

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



   a) Select the **Display Type:**Range option at the top of the form.

By default, GAMBIT *selects the* 3D Element *brick*  *for display in the graphics window. In this case, the resulting display shows the core of hexahedral elements created by means of the* Hex Core *volume meshing scheme (see Figure 13-21).*

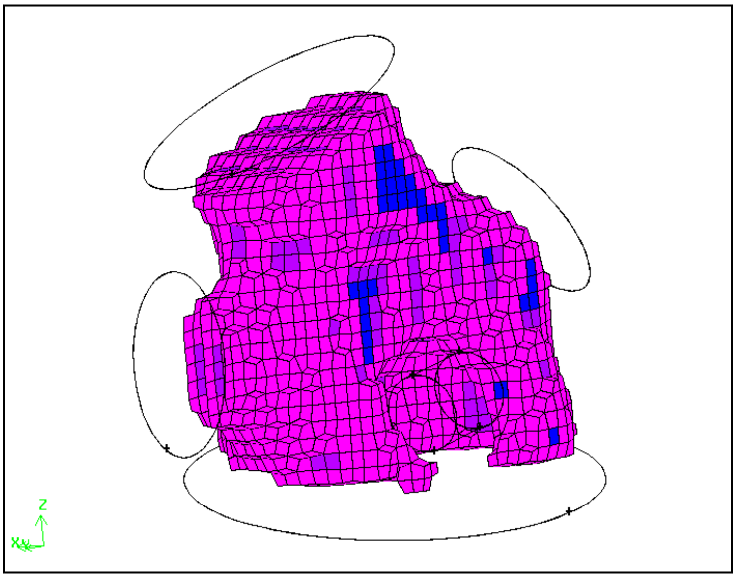


Figure 13-21: Central core of hexahedral mesh elements

b) Select the **Display Type:**Plane option at the top of the form.

The **Examine Mesh** *form allows you to view various mesh characteristics for the 3-D mesh. For example, Figure 13-22 and Figure 13-23 display tetrahedral and pyramidal volume mesh elements, respectively, for an intersecting cutting plane aligned with the x-z coordinate plane. You can generate these views of the model by selecting the*  *and*  *buttons, respectively, at the top of the* **Examine Mesh** *form and clicking on the* Y *slider bar near the bottom of the form.*

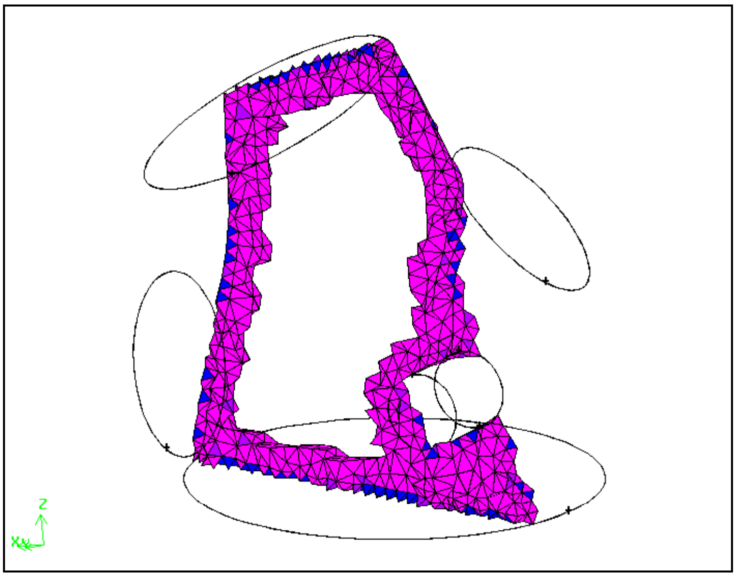


Figure 13-22: Cutting plane showing only tetrahedral mesh elements

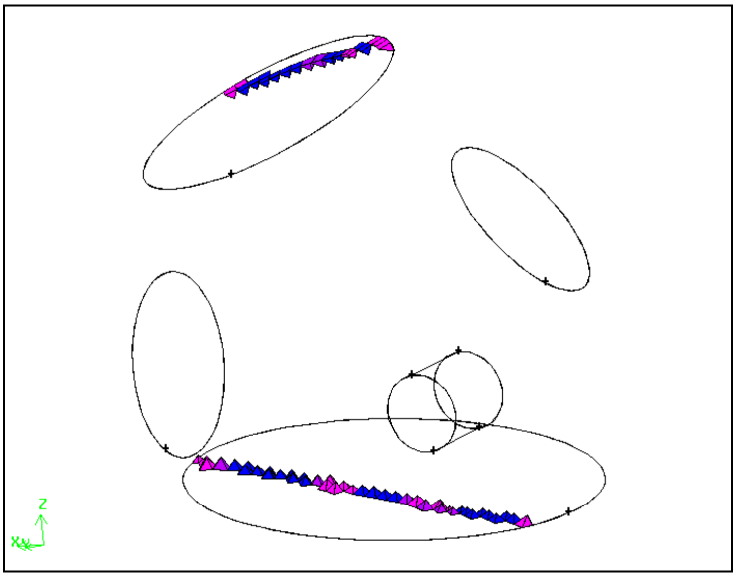


Figure 13-23: Cutting plane showing only pyramidal mesh elements

To view the worst element (as defined by the current quality metric and selected element type(s)), select the **Display Type:**Range *option, and click the* Show worst element *check box near the bottom of the* **Examine Mesh** *form.*

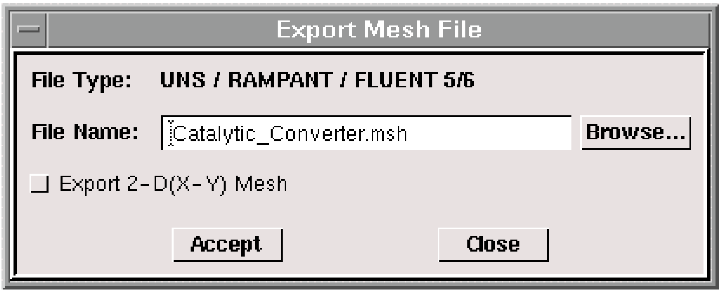
c) Close the **Examine Mesh** form by clicking the **Close** button at the bottom of the form.

Step 17: Export the Mesh and Save the Session

1. Export a mesh file for the catalytic converter.

   File → **Export** → **Mesh...**

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



   a) Enter the **File Name** for the file to be exported (`Catalytic_Converter.msh`).

   b) Click **Accept** in the **Export Mesh File** form.

      The file will be written to your working directory.

2. Save the GAMBIT session and exit GAMBIT.

   File → **Exit**

   GAMBIT *will ask you whether you wish to save the current session before you exit.*

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

13.5 Summary

This tutorial illustrated how to import geometry from an external CAD package as an IGES file, use GAMBIT healing and clean-up tools to make the geometry suitable for meshing, and mesh the geometry.

14. AIRPLANE GEOMETRY

In this tutorial you will import a STEP file that describes the geometry of an airplane body, including the wing and nacelle that houses the engine. You will clean up the geometry using GAMBIT smooth/heal and cleanup tools, apply three different types of size functions, and mesh the geometry using a tetrahedral meshing scheme.

In this tutorial you will learn how to:

- Import a STEP file
- Use a smooth-heal operation to repair the imported geometry
- Use several clean-up operations to clean up the geometry
- Construct a flow volume around the airplane geometry
- Apply size functions to control mesh quality
- Mesh faces using a triangular pave meshing scheme
- Mesh a volume using a tetrahedral meshing scheme
- Prepare the mesh to be read into FLUENT 5/6

14.1 Prerequisites

This tutorial assumes you have worked through Tutorials 1, 6, and 13 and are familiar with use of the GAMBIT GUI and general clean-up operations.

14.2 Problem Description

Figure 14-1 shows the geometry to be imported and meshed in this tutorial. This geometry represents one half of an airplane body, including the wing, strut, and nacelle that houses the engine but does not include the empennage. The imported geometry consists of many edges and faces that you will eliminate before generating the volume mesh. The mesh itself will consist entirely of tetrahedral elements.

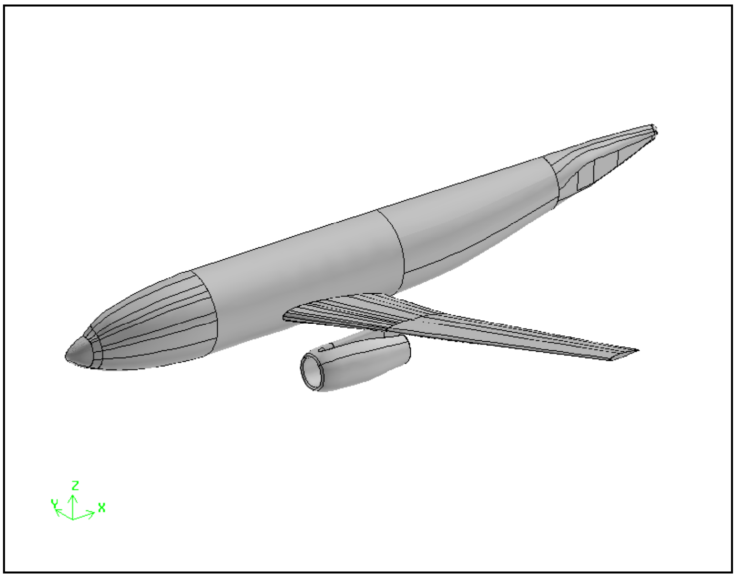


Figure 14-1: Airplane geometry

14.3 Strategy

In this tutorial, you will create a tetrahedral mesh in a flow volume surrounding one half of an airplane body, including the wing, strut, and nacelle housing the engine but excluding the empennage. The geometry will be imported as a STEP file containing many faces and edges that will need to be eliminated before meshing. After creating a flow volume around the imported geometry and using a GAMBIT smooth/heal operation to simplify the geometry, you will clean up the geometry using the cleanup tools available in GAMBIT. You will then create triangular face meshes on the airplane body surfaces and flow-volume symmetry plane and mesh the flow volume with tetrahedral elements.

NOTE: This tutorial employs a relatively course mesh so that the mesh characteristics can be easily examined. In actual practice, the model described in this tutorial would employ a much finer mesh than is used here, especially in the regions adjacent to the airplane body.

14.4 Procedure

1. Copy the file

   `path/Fluent.Inc/gambit2.x/help/tutfiles/nacelle-9.stp`

   (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 “`Airplane`”.

Step 1: Select a Solver

1. Choose the solver from the main menu bar:

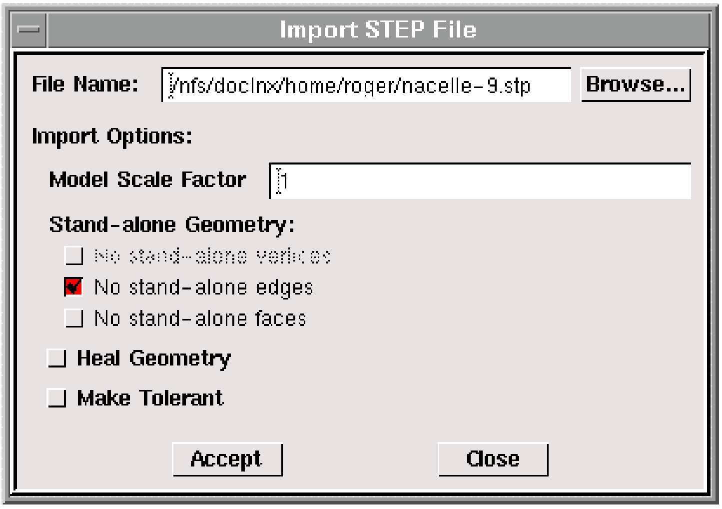
   Solver $\longrightarrow$ **FLUENT 5/6**

   The choice of a solver dictates the options available in various forms (for example, the boundary types available in the **Specify Boundary Types** *form). For some systems,* **FLUENT 5/6** *is the default solver. The solver currently selected is shown at the top of the* GAMBIT *GUI.*

Step 2: Import the STEP File

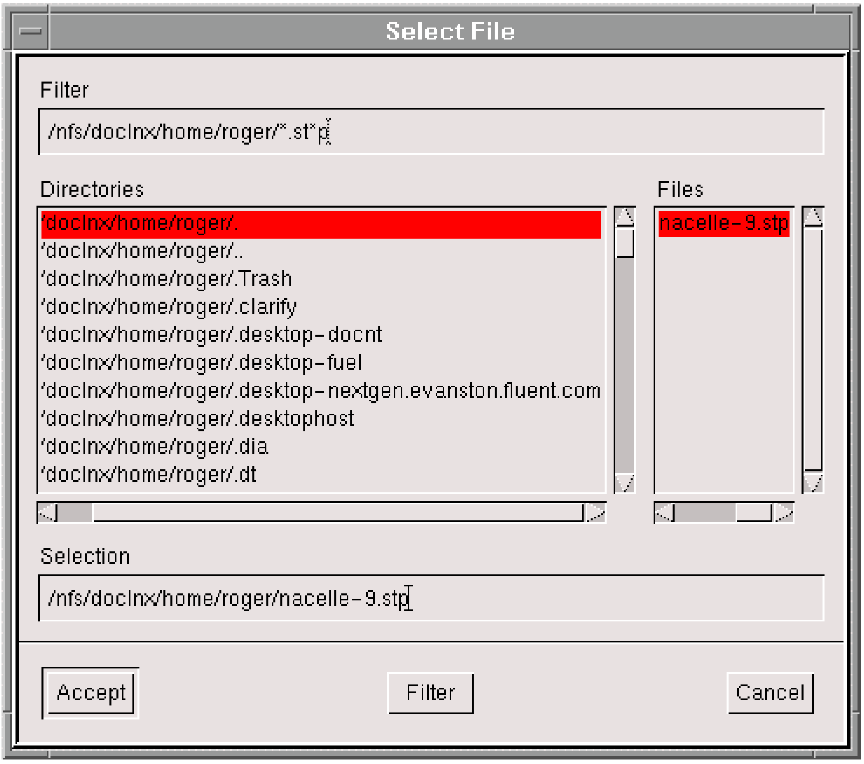
File → Import → STEP ...

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



1. Click on the **Browse...** button.

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



a) Select `nacelle-9.stp` in the **Files** list.

b) Click Accept on the **Select File** form.

2. On the **Import STEP File** form, under **Stand-alone Geometry**, select the No stand-alone edges option.

3. Unselect the **Make Tolerant** option.

   The **Make tolerant** *option improves geometric connectivity and can be invoked either during geometry import or after geometry import—as part of a healing operation. In this tutorial, you will invoke the* **Make tolerant** *option during a healing operation after examining and addressing duplicate-geometry issues (see Steps 3 and 4, below).*

4. Click **Accept**.

   The STEP file for the airplane body will be read into GAMBIT *(see Figure 14-2).*

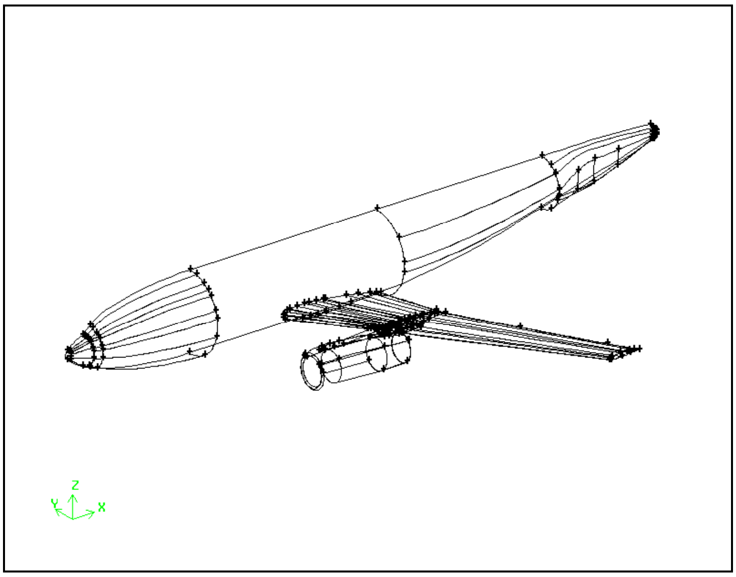


Figure 14-2: Imported airplane geometry

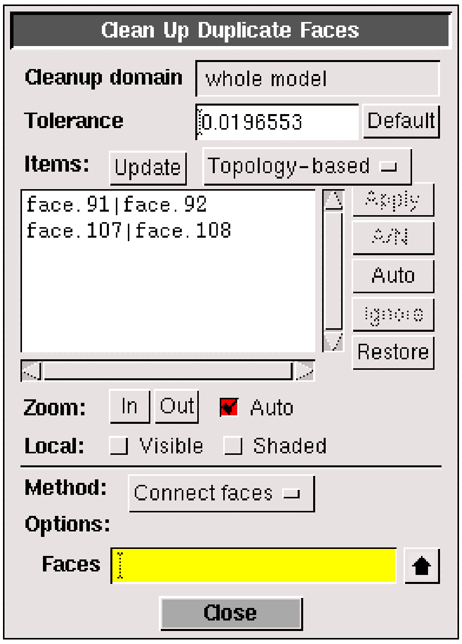
Step 3: Clean Up Duplicate Faces

The imported geometry is “dirty” in that it contains duplicate faces and edges. In this step, you will use a GAMBIT *cleanup operation to eliminate the duplicate faces.*

1. Clean up duplicate faces in the imported geometry.

TOOLS  → **CLEANUP**  → **CLEAN UP DUPLICATE GEOMETRY** 
R 

This command sequence opens the **Clean Up Duplicate Faces** *form.*



When you open any of the cleanup forms, such as the **Clean Up Duplicate Faces** *form,* GAMBIT *automatically sets the graphics window color mode to display colors based on connectivity rather than topology.* GAMBIT *also automatically sets the graphics window pivot function to the user-specified pivot mode.*

a) Retain the Topology-based search option.

b) Click the Default pushbutton located on the right side of the **Tolerance** text box.

When you click the Default *pushbutton,* GAMBIT *displays the* **Tolerance** *default value and populates the* **Items** *list with two sets of faces that meet the tolerance criterion. For the* Topology-based *search option, the* **Items** *list contains sets of faces that are topologically identical to each other and the corresponding boundary edges of which are close to each other to within the* **Tolerance** *value.*

c) Select the first duplicate-face set displayed in the **Items** list.

GAMBIT *zooms in on and highlights the faces shown in Figure 14-3. (NOTE: Due to slight differences in entity numbering between computer platforms, the entity numbers shown in Figure 14-3 and in all subsequent figures that include entity labels might differ from those actually displayed in the* GAMBIT *graphics window.)*

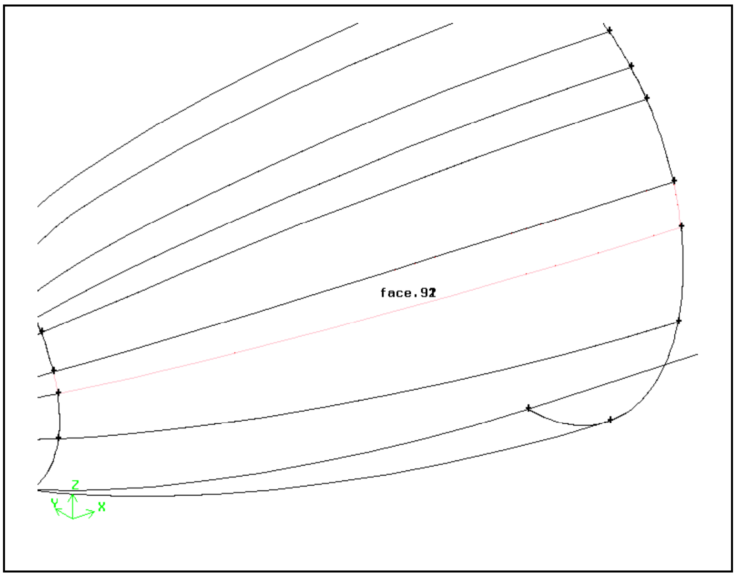


Figure 14-3: Set of faces comprising the first duplicate-face set in the **Items** list

GAMBIT *automatically selects the* **Method:**Connect faces *option and populates the* **Options:Faces** *pick list with the faces to be connected in performing the cleanup operation.*

d) Click the A/N pushbutton in the vertical array of pushbuttons located to the right of the **Items** list.

The A/N *("Apply/Next") pushbutton applies the* Connect faces *method to clean up the duplicate faces and automatically selects the remaining duplicate-face set in the* **Items** *list.*

e) Click Apply to clean up the remaining set of duplicate faces.

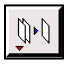
f) Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full model geometry in the graphics window.

The removal of the duplicate faces does not significantly affect the appearance of the airplane geometry.

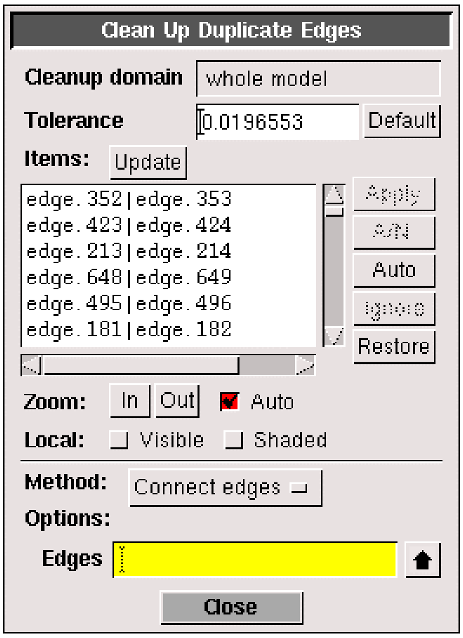
Step 4: View List of Duplicate Edges

In addition to the duplicate faces cleaned up in the previous step, the imported geometry contains many duplicate edges. In this step, you will use a GAMBIT *cleanup operation to view the list the duplicate edges.*

1. List all duplicate edges in the imported geometry.

   TOOLS  → **CLEANUP**  → **CLEAN UP DUPLICATE GEOMETRY** 
   R 

   This command sequence opens the **Clean Up Duplicate Edges** *form.*

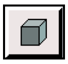
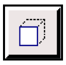


   a) Click the Default pushbutton located on the right side of the **Tolerance** text box.

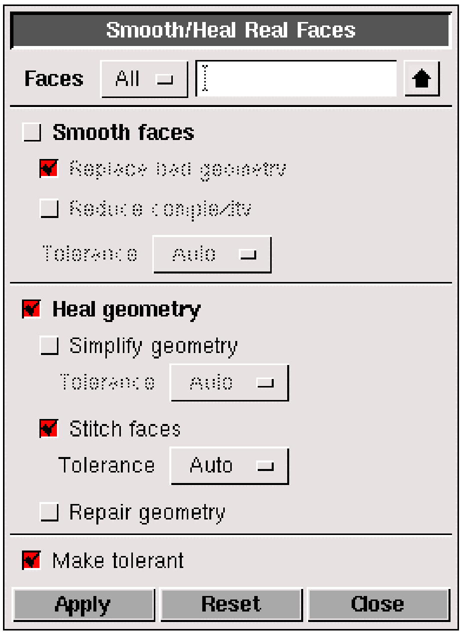
      When you click the Default *pushbutton,* GAMBIT *displays the default* **Tolerance** *value and populates the* **Items** *list with duplicate-edge sets—that is, pairs of edges the maximum distances between which are less than the* **Tolerance** *value. In this tutorial, you will eliminate the duplicate edges by "healing" the geometry (see Step 5: Heal the Geometry, below).*

Step 5: Heal the Geometry

1. Heal the imported geometry.

GEOMETRY  → **FACE**  → **SMOOTH/HEAL REAL FACES** 

This command sequence opens the **Smooth/Heal Real Faces** *form.*



a) Select All from the option menu to the right of **Faces**.

b) Unselect the **Smooth faces** option.

c) Select the **Heal geometry** option.

d) Retain the Stitch faces option.

This option helps ensure connectivity between edges in the healed geometry.

e) Click **Apply**.

GAMBIT *heals the geometry and eliminates duplicate edges in the model. (NOTE: Connected edges are shown as blue in the graphics window.)*

Step 6: Clean Up Holes

In this step, you will clean up "holes" in the model (see Figure 14-4) to ensure that the airplane geometry faces can be stitched together to form a continuous surface. In this case, the geometry includes three holes: one near the tail, one near the cockpit, and one that lies in the symmetry plane for the airplane geometry. The face you create from the "hole" that lies in the symmetry plane will be connected (in a later step) to a rectangular face that represents the symmetry plane of the flow volume.

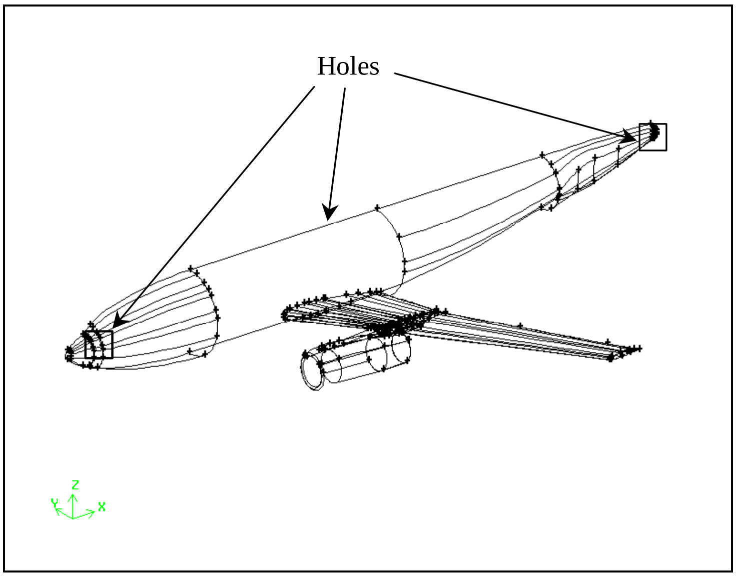
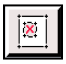
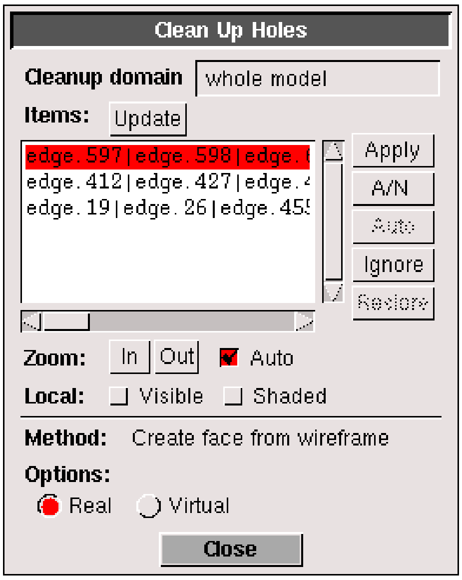


Figure 14-4: Holes in airplane geometry

1. Clean up holes in the geometry.

   TOOLS  → **CLEANUP**  → **CLEAN UP HOLES** 

   This command sequence opens the **Clean Up Holes** *form.*



GAMBIT *automatically populates the* **Items** *list with the edge sets that constitute holes in the model.*

a) Select the first item (edge set) in the **Items** list.

GAMBIT *automatically highlights the set of edges near the airplane tail as shown in Figure 14-5.*

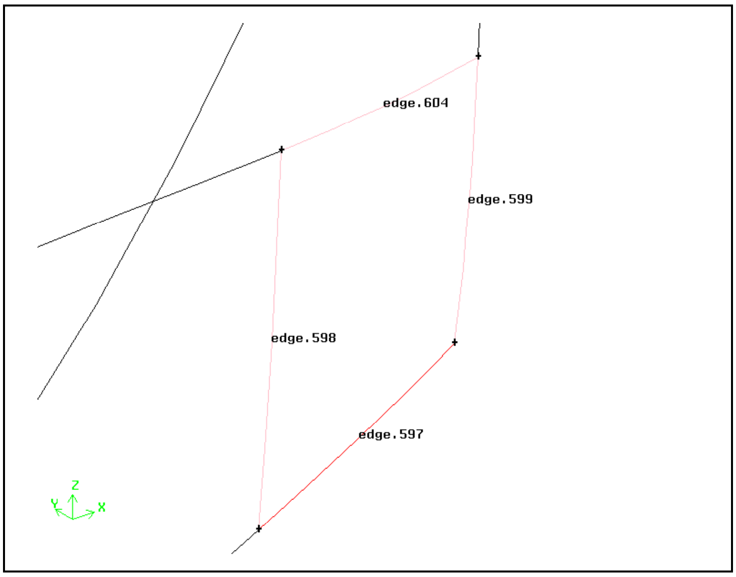


Figure 14-5: Set of edges comprising the first hole listed in the **Items** list

The **Clean Up Holes** *form provides only one method for eliminating holes—* Create face from wireframe.

b) Click the A/N pushbutton in the vertical array of pushbuttons located to the right of the **Items** list.

The A/N *("Apply/Next") pushbutton applies the* Create face from wireframe *method to remove the hole from the tail area then updates the* **Items** *list and automatically selects the next item (the hole near the cockpit) in the* **Cleanup domain**.

c) Click A/N again to eliminate the next hole in the **Cleanup domain**.

GAMBIT *eliminates the hole near the cockpit and automatically highlights the remaining item (corresponding to the hole that lies in the symmetry plane).*

d) Click Apply to eliminate the remaining hole by creating the airplane symmetry face.

e) Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full airplane geometry in the graphics window.

Figure 14-6 shows the airplane geometry with shaded views of the three new faces.

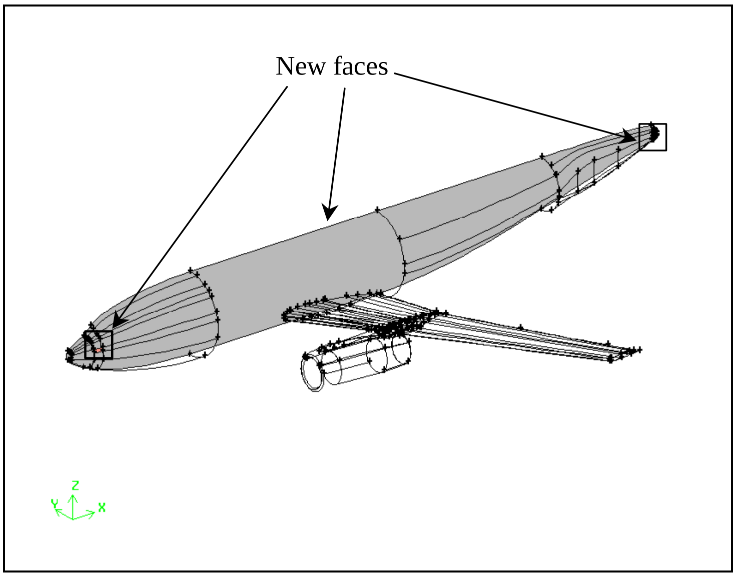
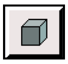
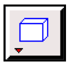


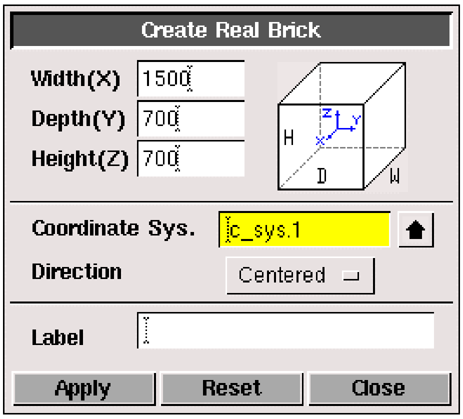
Figure 14-6: Airplane geometry with three new faces

Step 7: Create a Brick around the Airplane Body

1. Create a brick.

   GEOMETRY  → **VOLUME**  → **CREATE VOLUME** 

   This command sequence opens the **Create Real Brick** *form.*



   a) Enter a value of `1500` for the **Width** of the brick.

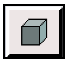
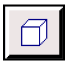
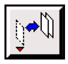
   b) Enter `700` for the **Depth** and `700` for the **Height**.

   c) Retain the **Direction:**Centered option.

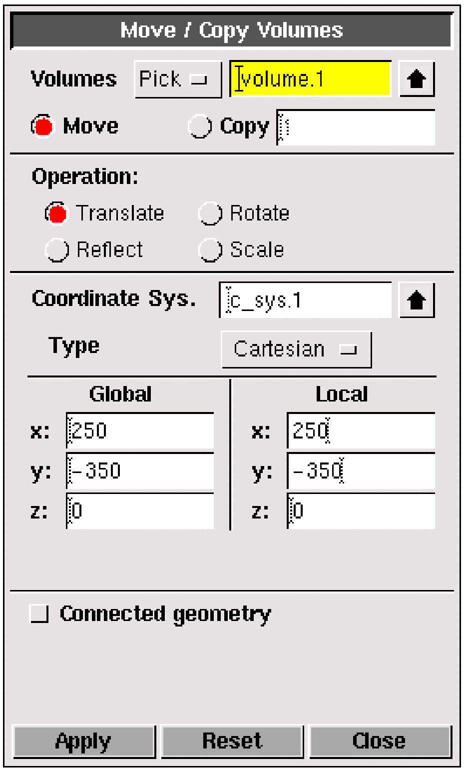
   d) Click **Apply**.

2. Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the brick and full airplane body in the graphics window.

3. Align one face of the brick with the symmetry plane of the airplane geometry.

   GEOMETRY  → **VOLUME**  → **MOVE/COPY/ALIGN VOLUMES** 

   This command sequence opens the **Move / Copy Volumes** *form.*



a) Select (*Shift*-left-click) the brick in the graphics window.

b) Retain **Move** (the default) under **Volumes** in the **Move / Copy Volumes** form.

c) Retain the **Operation:**Translate option.

d) Enter (`250, -350, 0`) under **Global** to move the brick 250 units in the *x* direction and -350 units in the *y* direction.

Note that GAMBIT *automatically fills in the values under* **Local** *as you enter values under* **Global**.

e) Click **Apply**.

4. Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full brick and airplane body in the graphics window.

GAMBIT *aligns the brick and airplane geometry as shown in Figure 14-7.*

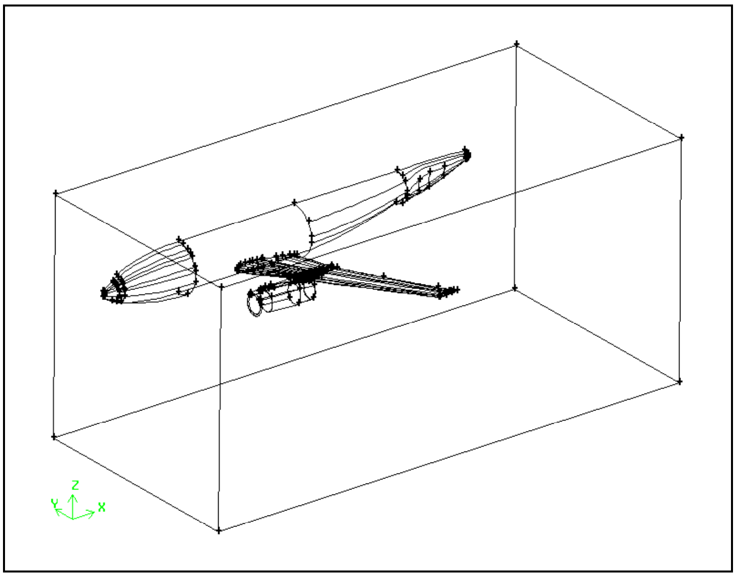
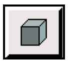


Figure 14-7: Brick and airplane body

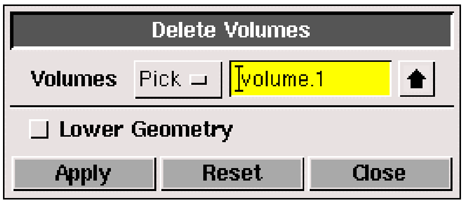
Step 8: Delete the Brick High-level Geometry

In Step 10: "Create the Flow Volume," you will stitch the faces of the brick with those of the airplane geometry to create an enclosed flow volume. You cannot simply subtract the airplane body from the brick to produce the flow volume, because you used "virtual geometry" to clean up the airplane body, and GAMBIT *cannot perform Boolean operations on virtual geometry. Instead, you must create a virtual volume by stitching the virtual faces of the airplane geometry and the real faces of the brick. To do so, you must first delete the brick volume while retaining its lower-level geometry (faces).*

1. Delete the brick while retaining its faces.

GEOMETRY  → **VOLUME**  → **DELETE VOLUMES** 

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



a) Select (*Shift*-left-click) the brick in the graphics window.

b) Unselect the **Lower Geometry** option.

c) Click **Apply**.

GAMBIT *deletes the brick volume, but retains all of its component faces, edges, and vertices.*

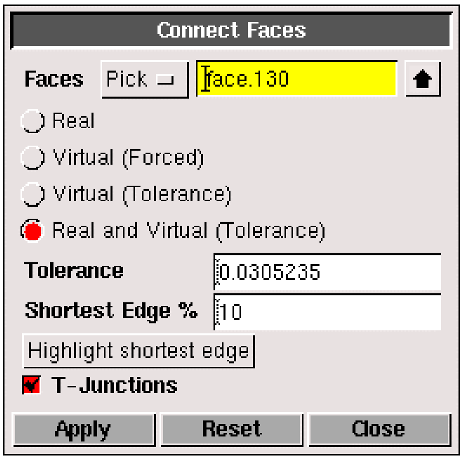
Step 9: Connect Faces on the Symmetry Plane

When you stitch faces to form a volume, it is not always necessary to specify *all* *of the faces to be stitched. In Step 10: "Create the Flow Volume," you will specify only two of the many faces to be used in creating the flow volume, and* GAMBIT *will automatically add the others when performing the stitch operation. To successfully carry out the stitch operation, you must first connect the symmetry face on the airplane geometry to the symmetry face on the brick.*

1. Connect the two symmetry faces.

   GEOMETRY → **FACE** → **CONNECT/DISCONNECT FACES**

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



   a) Click in the **Faces** pick-list field.

   b) Select the two symmetry faces (faces *A* and *B* in Figure 14-8).

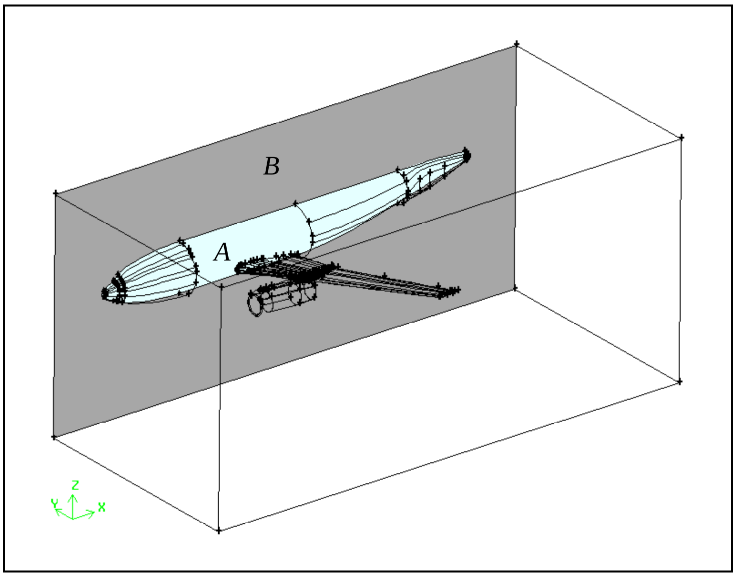


Figure 14-8: Airplane and brick symmetry faces

c) Select the Real and Virtual (Tolerance) option.

d) Select the **T-Junctions** option.

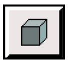
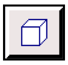
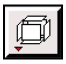
e) Click **Apply**.

GAMBIT *connects the symmetry faces.*

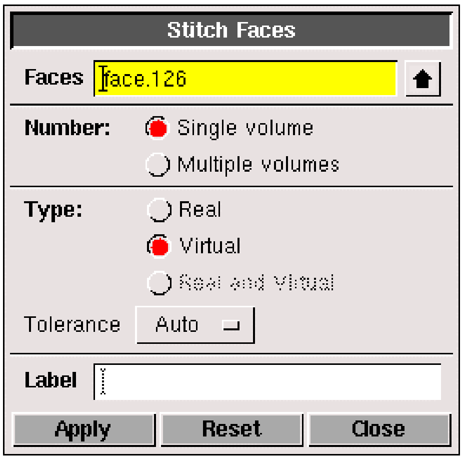
Step 10: Create the Flow Volume

Now that you have connected the symmetry faces, you can create the flow volume by specifying only two of the many faces that will be used to enclose the volume.

1. Stitch the faces to form the flow volume.

GEOMETRY  → **VOLUME**  → **STITCH FACES** 

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



a) Click in the **Faces** pick-list field.

b) Select one face of the flow volume—for example, the bottom face (*C*) shown in Figure 14-9.

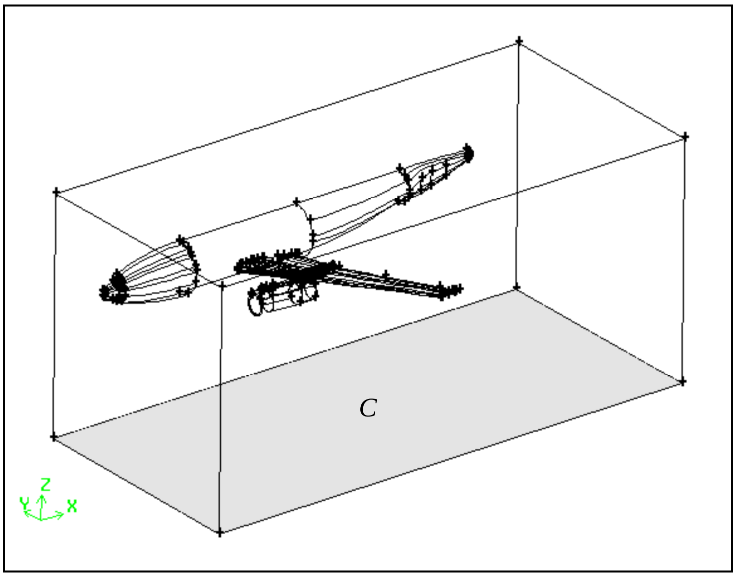


Figure 14-9: Face specified for the face-stitch operation

c) Retain the **Number:**Single volume option.

d) Select the **Type:**Virtual option.

e) Click **Apply**.

GAMBIT *stitches the airplane geometry faces and the brick faces external to the airplane geometry to create the flow volume.*

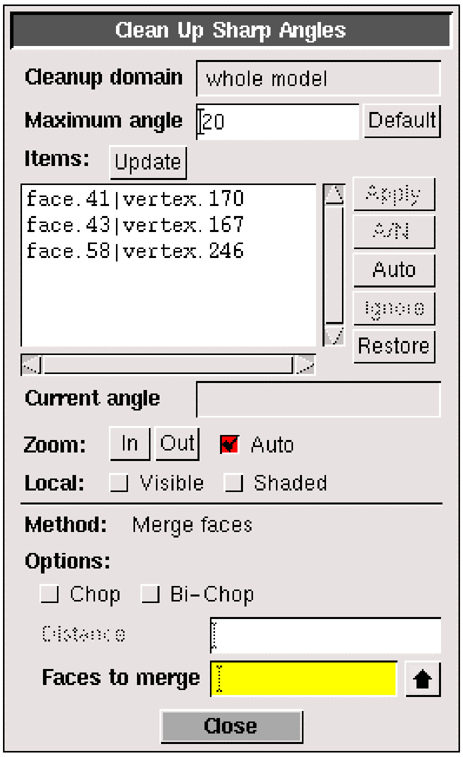
Step 11: Clean Up Sharp Angles

The imported geometry contains several sharp angles and short edges that can complicate your ability to mesh the flow volume. In the next two steps, you will use automatic GAMBIT *cleanup operations to eliminate the sharp angles and short edges.*

1. Automatically eliminate sharp angles in the airplane geometry.

   TOOLS → **CLEANUP** → **CLEAN UP SHARP ANGLES**

   This command sequence opens the **Clean Up Sharp Angles** *form.*



   a) Click the Default pushbutton located on the right side of the **Maximum angle** text box.

When you click the Default *pushbutton,* GAMBIT *displays the* **Maximum angle** *default value (20) and populates the* **Items** *list with three face-vertex pairs that meet the maximum-angle criterion. In this case, you will use an automatic operation to eliminate all three of the sharp angles.*

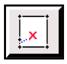
b) Click Auto in the vertical array of pushbuttons located to the right of the **Items** list.

GAMBIT *uses face-merge operations to automatically remove all of the sharp angles from the model. (NOTE: In general practice, you should exercise caution when using the* Auto *pushbutton to execute cleanup operations. Less-experienced* GAMBIT *users should select items one at a time in the* **Items** *list and use the* Apply *and/or* A/N *pushbuttons, instead.)*

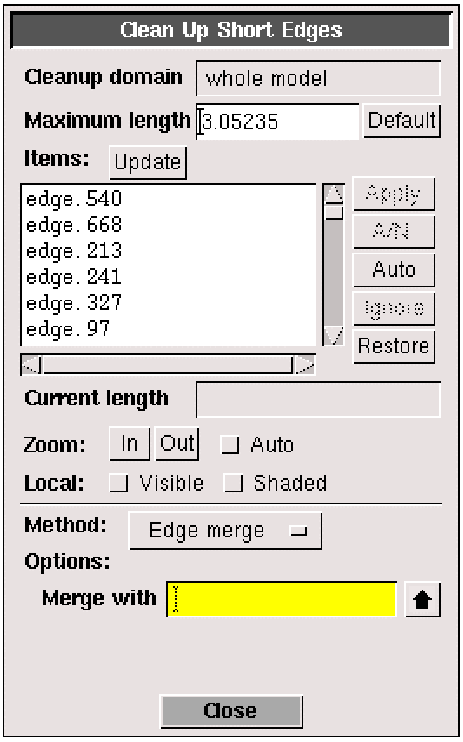
c) Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full flow-volume geometry in the graphics window.

Step 12: Clean Up Short Edges

1. Automatically eliminate the short edges in the model.

TOOLS  → **CLEANUP**  → **CLEAN UP SHORT EDGES** 

This command sequence opens the **Clean Up Short Edges** *form.*



a) Click the Default pushbutton located on the right side of the **Maximum length** text box.

> *When you click the* Default *pushbutton,* GAMBIT *displays the* **Maximum length** *of edges to be included in the* **Items** *list and populates the* **Items** *list with all edges in the* **Cleanup domain** *that meet the* **Maximum length** *criterion. By default, the* **Maximum length** *value is 10 times greater than the arc length of the shortest edge in the* **Cleanup domain**.

b) Unselect the **Zoom:**Auto option.

By default, GAMBIT *automatically zooms in on any item currently selected for a cleanup operation and displays the labels of the entities involved in the operation. If you retain the* **Zoom:**Auto *option in this case,* GAMBIT *will zoom in on every edge as it is removed from the model, thereby making it difficult to follow the cleanup operation in the graphics window.*

c) Click Auto in the vertical array of pushbuttons located to the right of the **Items** list.

GAMBIT *removes all but eight of the short edges from the model. Figure 14-10 shows the full geometry after removing the edges.*

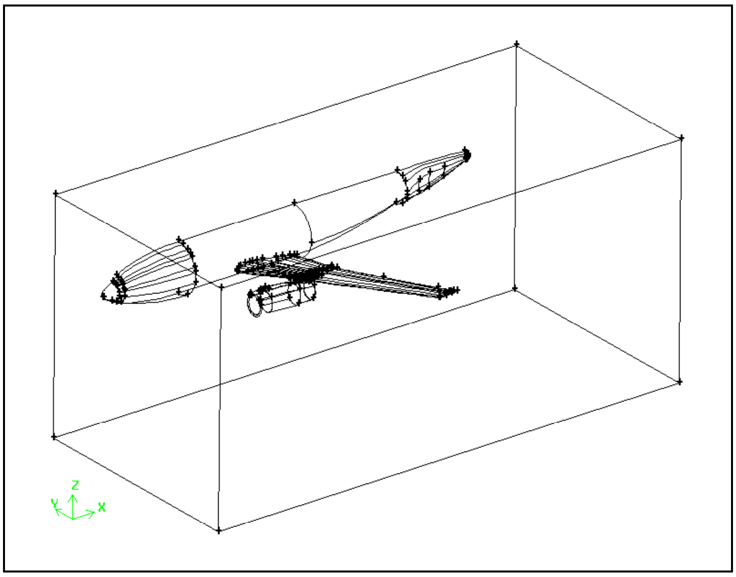


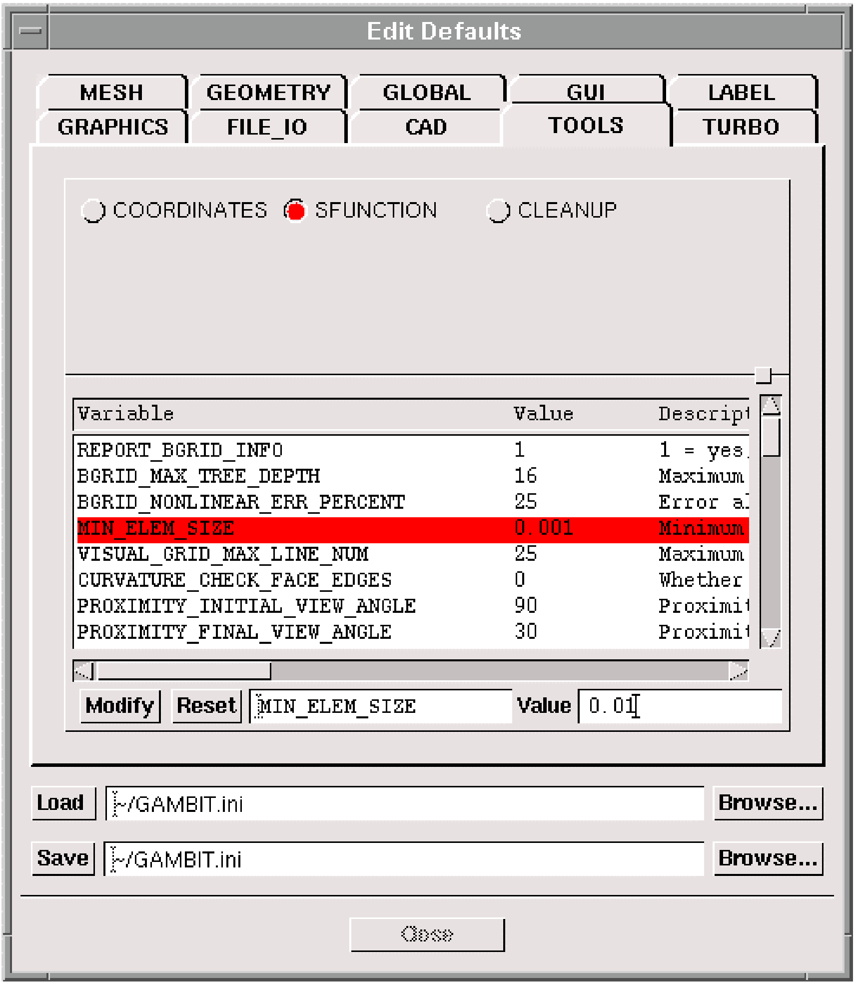
Figure 14-10: Airplane geometry after cleaning up short edges

Step 13: Modify the Size Function Defaults

Size functions allow you to control mesh quality and prevent the creation of highly skewed elements. For example, size functions can be used to specify the rate at which volume mesh elements change in size in proximity to a specified boundary. In this tutorial, you will use size functions to control mesh density in the regions surrounding the airplane geometry surfaces. Before creating and attaching the size functions, you will modify four of the size function defaults.

Edit ⟶ **Defaults...**

This command sequence opens the **Edit Defaults** *form.*



1. Select the **TOOLS** tab at the top of the form.

   GAMBIT *displays the available default settings for three "tools" operations—coordinate system, size-function, and cleanup.*

2. Select the SFUNCTION radio button.

   GAMBIT *displays the size-function defaults variables.*

3. Use the **Modify** pushbutton on the **Edit Defaults** form to modify four size-function defaults.

   a) Set the `MIN_ELEM_SIZE` default variable to `0.01`.

   b) Set the `CURVATURE_CHECK_FACE_EDGES` default variable to `1`.

   c) Set the `BGRID_MAX_TREE_DEPTH` default variable to `20`.

   d) Set the `BGRID_NONLINEAR_ERR_PERCENT` default variable to `15`.

      The `BGRID_MAX_TREE_DEPTH` *and* `BGRID_NONLINEAR_ERR_PERCENT` *values specified here represent moderate, intermediate values. For information concerning the use of such variables to control mesh quality, see "Create Size Function" in Section 5.2.2 of the* GAMBIT *Modeling Guide* .

4. Click **Close** to close the **Edit Defaults** form.

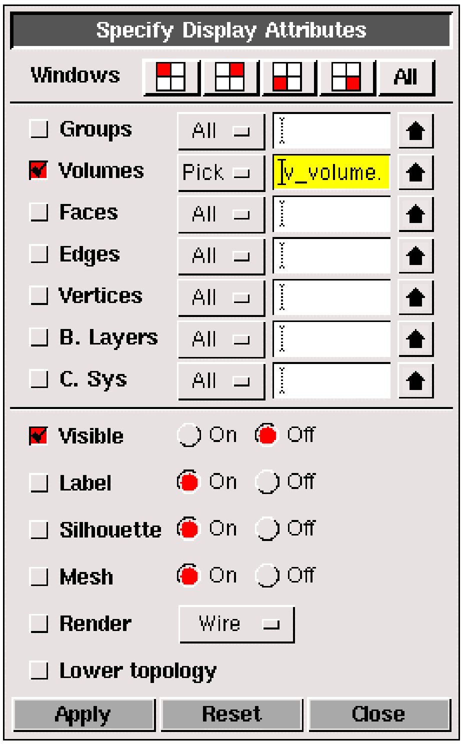
Step 14: Apply Size Functions to Control Mesh Quality

In this step, you will create and attach two types of size functions to control mesh sizes in the regions adjacent to the airplane geometry surfaces. You will apply the size functions to all faces associated with the external airplane surfaces (that is, all but the airplane symmetry face). Before creating the size functions, however, you will modify the graphics display to facilitate picking the size-function source and attachment entities.

1. Render invisible all faces that are not associated with the size functions.

   a) Click the **SPECIFY DISPLAY ATTRIBUTES**  command button on the **Global Control** toolpad.

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



   b) Click in the **Volumes** list box to make it active.

GAMBIT *automatically activates the* **Volumes** *check box, indicating that the display specifications are to apply to any specified volumes.*

c) Select the volume in the graphics window.

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

e) Unselect the **Lower topology** option.

f) Click **Apply**.

GAMBIT *turns off the display of the volume but retains the display of its bounding faces. Now, you will turn off display of all faces other than the symmetry plane and the airplane body.*

g) Click in the **Faces** list box to make it active.

GAMBIT *automatically activates the* **Faces** *check box, indicating that the display specifications are to apply to any specified faces.*

h) Select all six faces of the brick and the airplane symmetry face (see Figure 14-11).

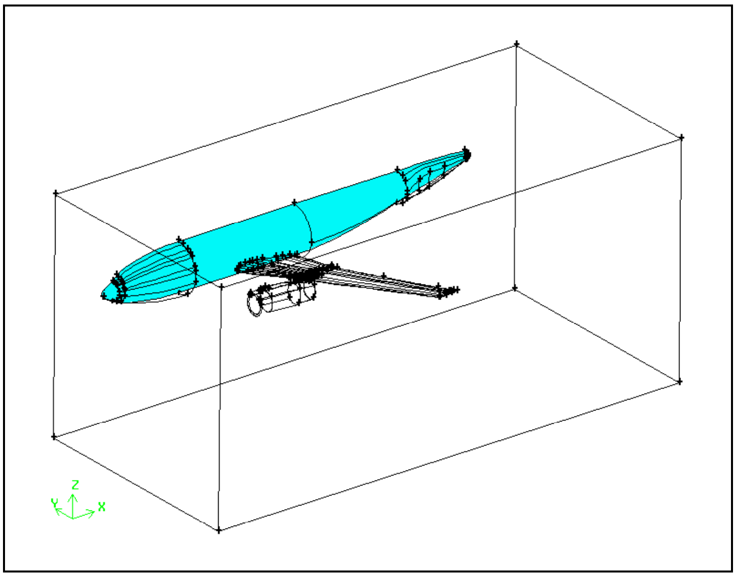


Figure 14-11: Flow volume and airplane symmtery face (shaded)

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

j) Retain the **Lower topology** option.

k) Click **Apply**.

GAMBIT *turns off the display of all faces that are not associated with the airplane surface geometry (see Figure 14-12).*

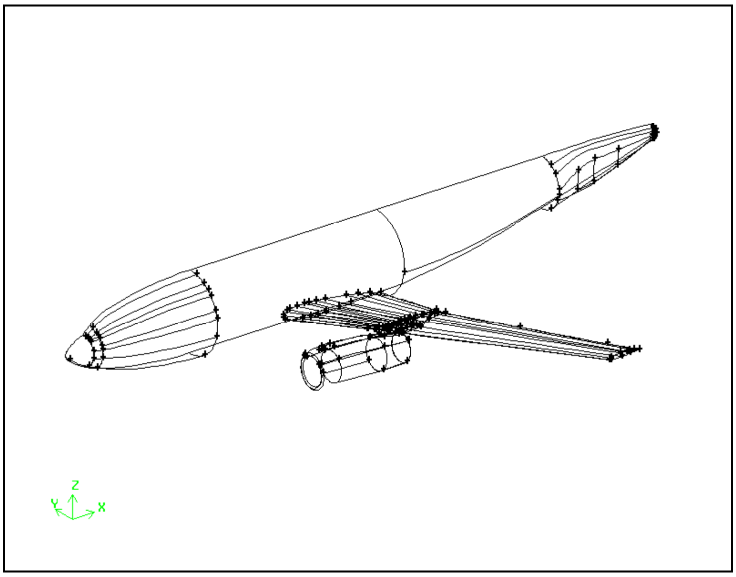


Figure 14-12: Graphics display after change in display attributes

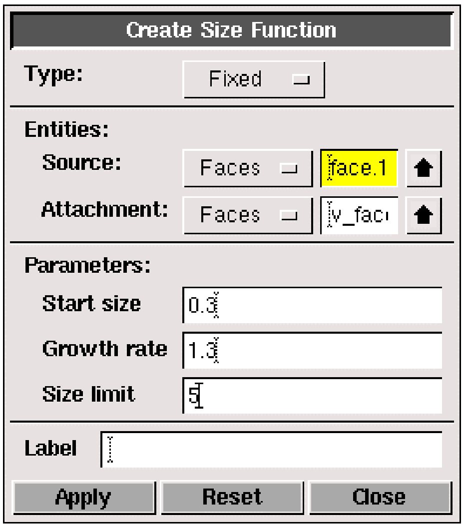
l) On the **Specify Display Attributes** form, click **Close** to close the form.

2. Create and apply a fixed size function to the airplane geometry.

a) Open the **Create Size Function** form.

TOOLS  → **SIZE FUNCTIONS**  → **CREATE SIZE FUNCTION** 

This command sequence opens the **Create Size Function** *form.*



a) Retain the **Type:**Fixed option.

b) On the **Source** option button, select the Faces option.

c) Select the five narrow faces on the trailing edges of the wing and nacelle (see the shaded faces in Figure 14-13).

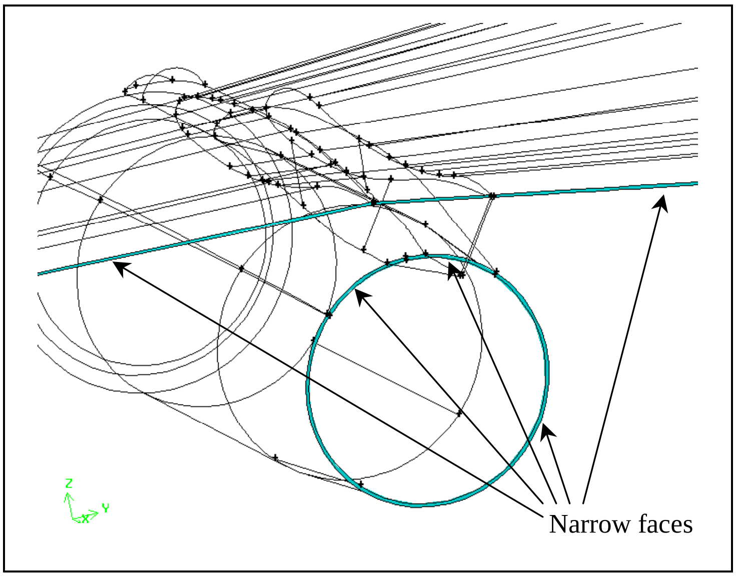


Figure 14-13: Five narrow faces on trailing edges of wing and nacelle

d) On the **Attachment** option button, select the Faces option.

e) Select all of the faces displayed in the graphics window by *Shift*-left-dragging the mouse to create a rectangular selection box around the airplane geometry.

When you Shift-*left-drag the mouse to create a selection box in the graphics window,* GAMBIT *selects all displayed entities touched by or enclosed within the box. In this case,* GAMBIT *populates the* **Attachment:**Faces *list with all of the faces associated with the airplane geometry surfaces.*

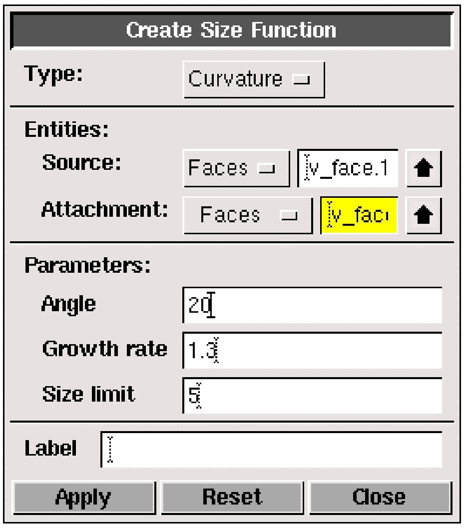
f) In the **Start size** text box, enter the value `0.3`.

g) In the **Growth rate** text box, enter the value, `1.3`.

h) In the **Size limit** text box, enter the value, `5`.

i) Click **Apply** to create and attach the size function.

3. Create a curvature size function and apply it to all faces associated with the airplane geometry.



a) On the **Create Size Function** form, select the **Type:**Curvature option.

b) On the **Source** option button, retain the Faces option.

c) Click in the **Source:**Faces list box to make it active.

d) Select all of the displayed faces by *Shift*-left-dragging the mouse in the graphics window to create a rectangular selection box around the airplane geometry.

e) On the **Attachment** option button, retain the Faces option.

f) Click in the **Attachment:**Faces list box to make it active.

g) Select all of the faces displayed in the graphics window by *Shift*-left-dragging the mouse to create a rectangular selection box around the airplane geometry.

h) In the **Angle** text box, enter the value `20`.

i) In the **Growth rate** text box, retain the value, `1.3`.

j) In the **Size limit** text box, retain the value, `5`.

k) Click **Apply** to create and attach the size function.

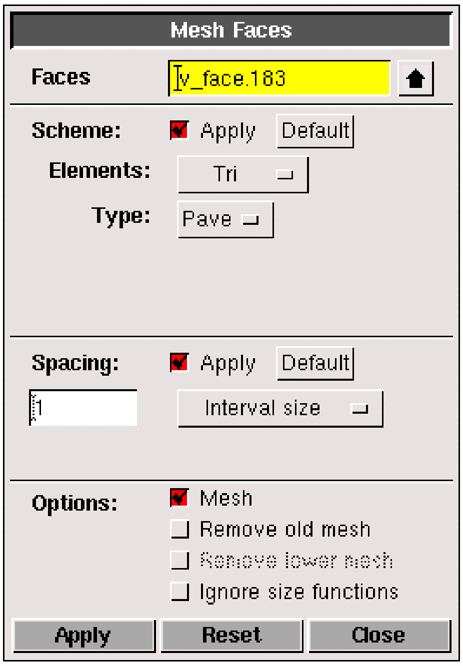
You can view the size functions by means of the **View Size Function** *command on the* **Size Function** *toolpad.*

Step 15: Mesh the Airplane Body Surface

In this step, you will mesh all of the surfaces associated with the airplane geometry (excluding the airplane symmetry face).

1. Create triangular meshes on the surfaces of the airplane geometry.

MESH → **FACE** → **MESH FACES**



a) Click in the **Faces** list box to make it active.

b) Select all of the faces displayed in the graphics window by *Shift*-left-dragging the mouse to create a rectangular selection box around the airplane geometry.

c) Select the **Elements:**Tri option.

d) Retain the **Type:**Pave option.

e) Under **Spacing**, retain the **Interval size** of `1`.

f) Click the **Apply** button at the bottom of the form.

GAMBIT *meshes the faces as shown in Figure 14-14, Figure 14-15, and Figure 14-16.*

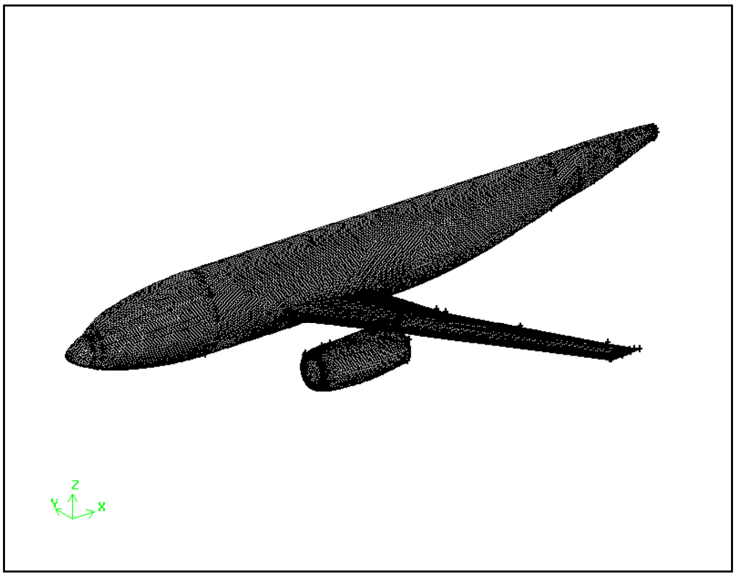


Figure 14-14: Airplane surface mesh—whole airplane

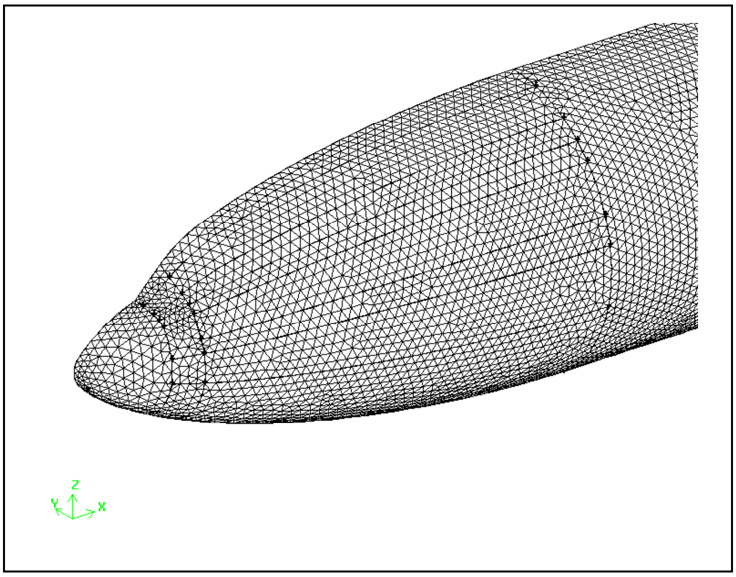


Figure 14-15: Airplane surface mesh—cockpit area

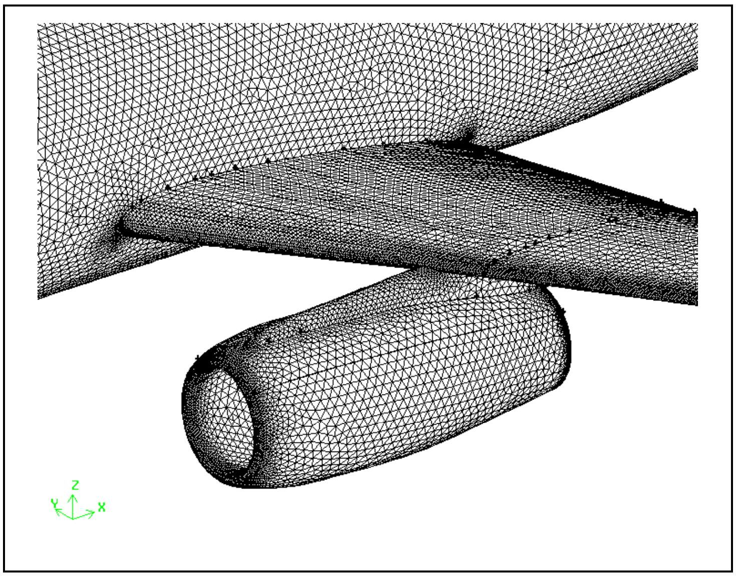


Figure 14-16: Airplane surface mesh—partial wing and nacelle

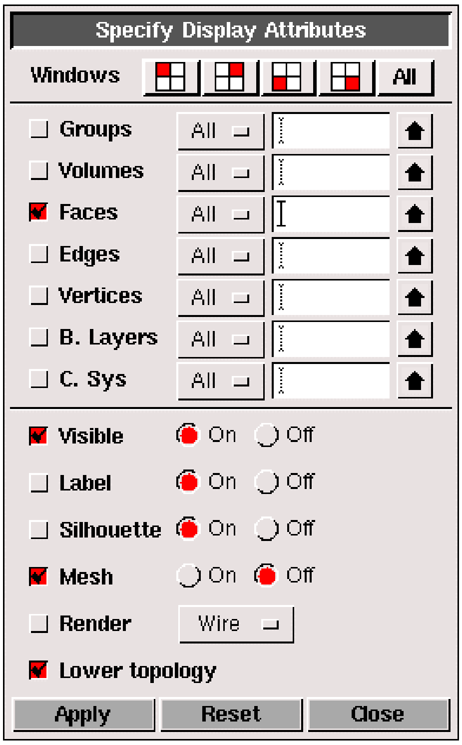
Step 16: Apply a Size Function to the Symmetry Plane

Now that you have meshed the airplane body surfaces, you will use the meshes on the edges that bound the airplane symmetry face to define a size function created on the symmetry plane (that is, the flow-volume symmetry face). To facilitate selecting the appropriate edges when defining the size function, you will first make invisible all faces other than the flow-volume symmetry face.

1. Render invisible all of the faces other than the flow-volume symmetry face.

   a) Click the **SPECIFY DISPLAY ATTRIBUTES**  command button on the **Global Control** toolpad.

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



   b) Select the **Faces** check box and ensure that the All option is selected.

   c) Select the **Visible:**On option.

d) Select the **Mesh:**Off option.

e) Click **Apply**.

GAMBIT *renders the geometry visible and the mesh invisible.*

f) Click the **FIT TO WINDOW** command button  at the top left of the **Global Control** toolpad to see the full model geometry in the graphics window.

g) Click the **Faces** list box to make it active.

h) Select all of the faces associated with the airplane geometry by *Shift*-left-dragging the mouse from the lower right toward the upper left to create a rectangular selection box around the airplane geometry (see Figure 14-17).

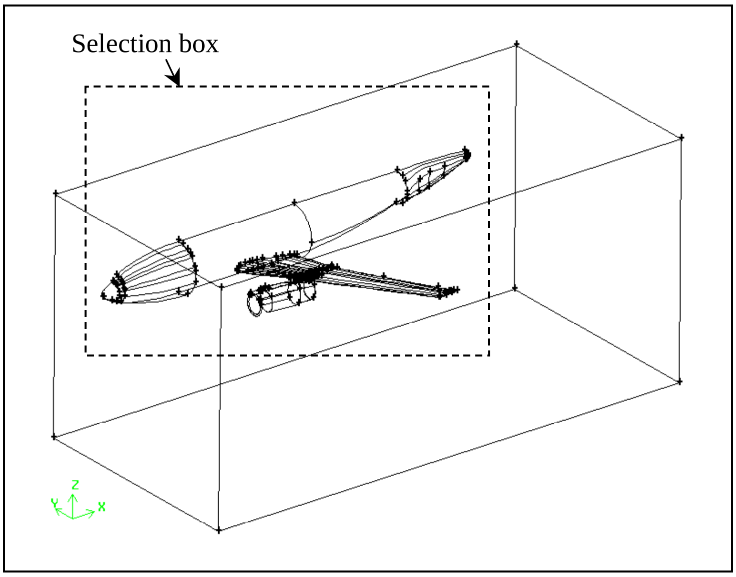


Figure 14-17: Entire model with selection box

When you Shift-*left-drag the mouse from the lower right toward the upper left to create a selection box in the graphics window*, GAMBIT *selects only those entities completely enclosed within the box. In this case,* GAMBIT *populates the* **Faces** *list with all of the faces associated with the airplane geometry.*

i) Manually select (*Shift*-left-click) all faces of the flow volume brick except the symmetry face.

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

k) Click **Apply**.

GAMBIT *renders the model invisible except for the flow-volume symmetry face (see Figure 14-18).*

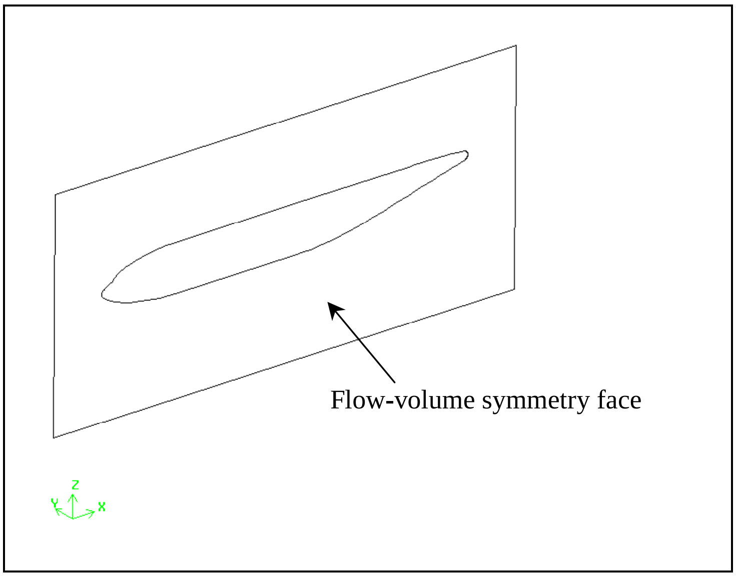


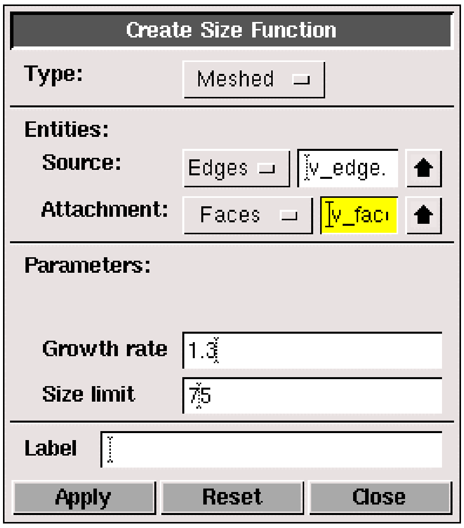
Figure 14-18: Flow-volume symmetry face

l) Click **Close** to close the **Specify Display Attributes** form.

2. Create a meshed size function and apply it to the flow-volume symmetry face.

TOOLS → **SIZE FUNCTIONS** → **CREATE SIZE FUNCTION**

This command sequence opens the **Create Size Function** *form.*



a) Select the **Type:**Meshed option.

b) On the **Source** option button, select the Edges option.

c) Select all of the edges that bound the airplane symmetry face by *Shift*-left-dragging the mouse from the lower right toward the upper left to create a rectangular selection box around the airplane symmetry face (see Figure 14-19).

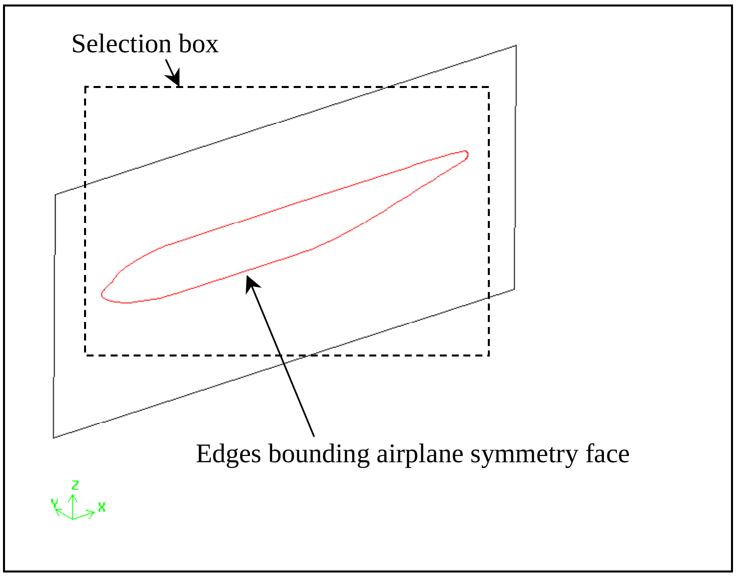


Figure 14-19: Edges that bound airplane symmetry face

d) On the **Attachment** option button, select the Faces option.

e) Click in the **Attachment:**Faces list box to make it active.

f) Select the flow-volume symmetry face (see Figure 14-18, above).

g) In the **Growth rate** text box, enter the value, `1.3`.

h) In the **Size limit** text box, enter the value, `75`.

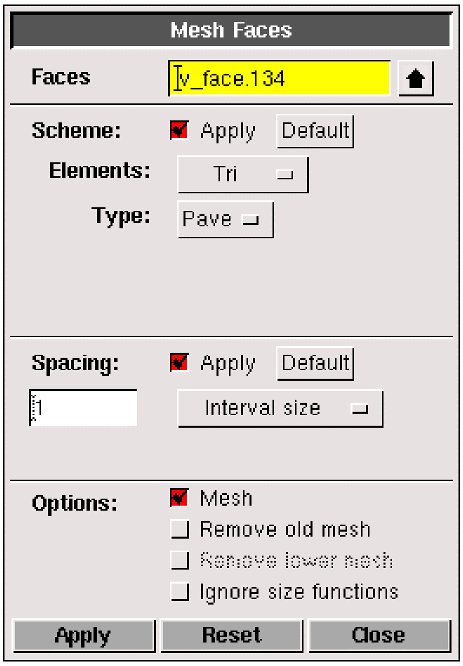
i) Click **Apply** to create and attach the size function.

Step 17: Mesh the Symmetry Plane

In this step, you will mesh the flow-volume symmetry face to facilitate meshing of the flow volume itself.

1. Create triangular meshes on the surfaces of the airplane geometry.

MESH → **FACE** → **MESH FACES**



a) In the graphics window, select the flow-volume symmetry face.

b) Select the **Elements:**Tri option.

c) Retain the **Type:**Pave option.

d) Under **Spacing**, retain the Interval size of `1`.

e) Click the **Apply** button at the bottom of the form.

GAMBIT *meshes the flow-volume symmetry face as shown in Figure 14-20.*

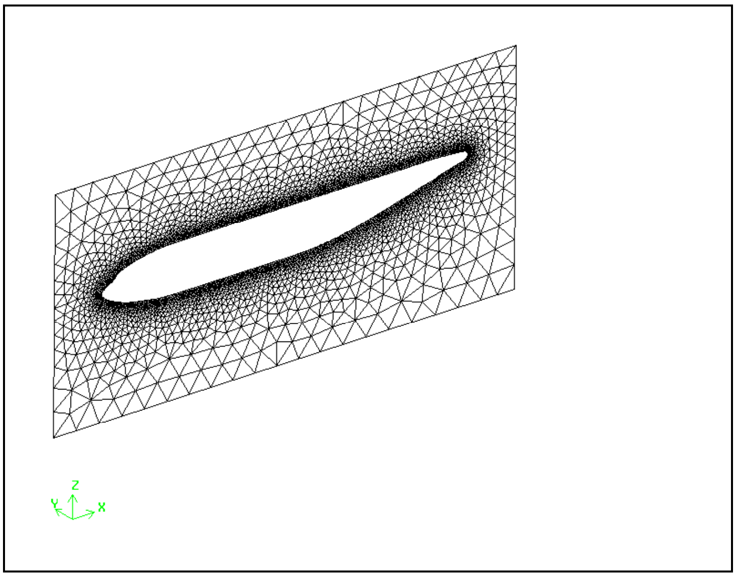
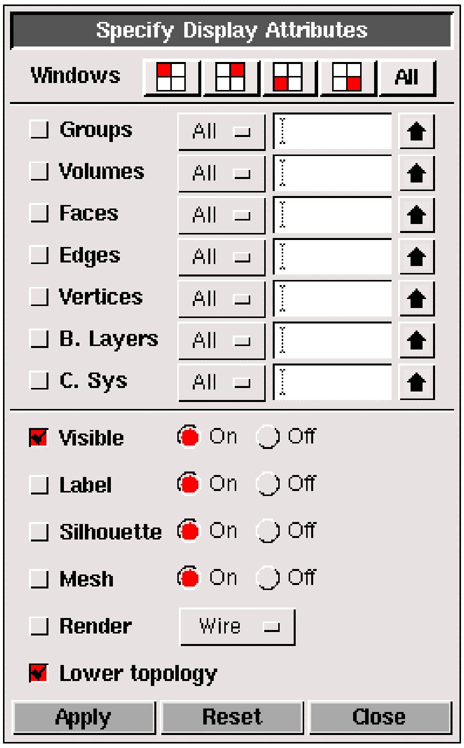


Figure 14-20: Triangular paved mesh on flow-volume symmetry face

2. Render all of the geometry visible.

   a) Click the **SPECIFY DISPLAY ATTRIBUTES**  command button on the **Global Control** toolpad.

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



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

c) Click **Apply**.

GAMBIT *renders the entire model visible, including the mesh (see Figure 14-21).*

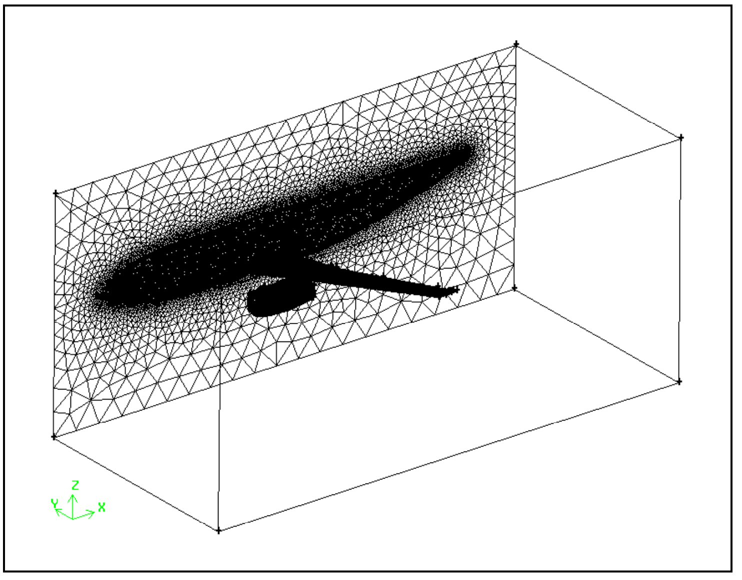


Figure 14-21: Flow volume with meshed faces

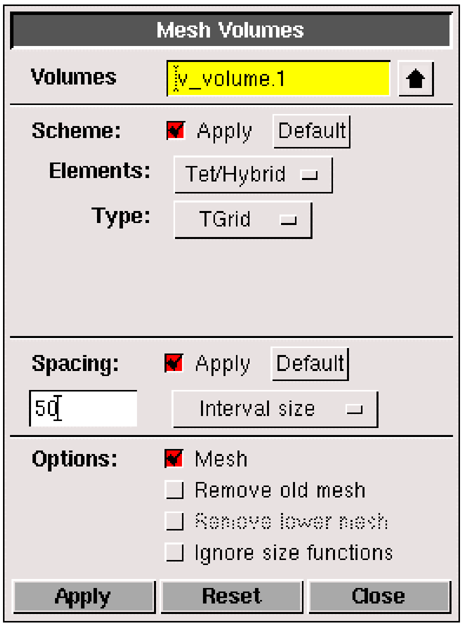
d) Click **Close** to close the **Specify Display Attributes** form.

Step 18: Mesh the Flow Volume

1. Mesh the flow volume using a tetrahedral meshing scheme.

   MESH → **VOLUME** → **MESH VOLUMES**

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



   a) Select (*Shift*-left-click) the flow volume in the graphics window.

   b) Retain the **Elements:**Tet/Hybrid option.

   c) Retain the **Type:**TGrid option.

   d) Under **Spacing**, enter an Interval size of `50`.

   e) Click the **Apply** button at the bottom of the form to mesh the volume.

      GAMBIT *meshes the volume as shown in Figure 14-22.*

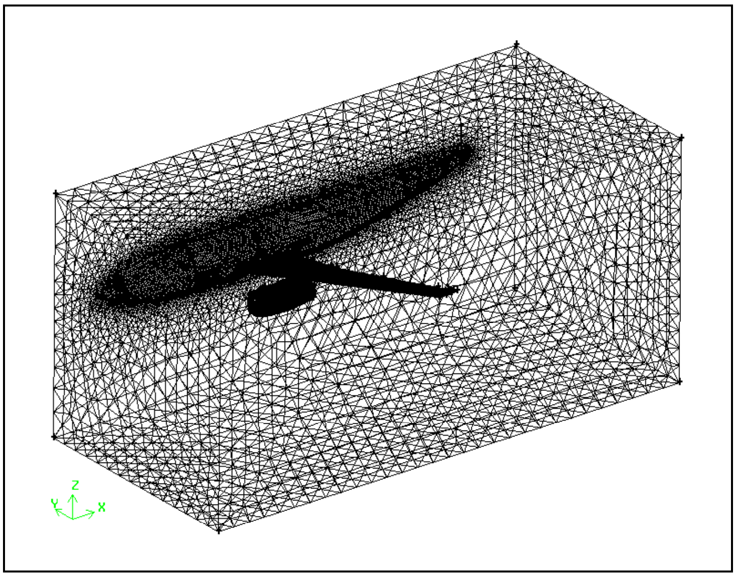


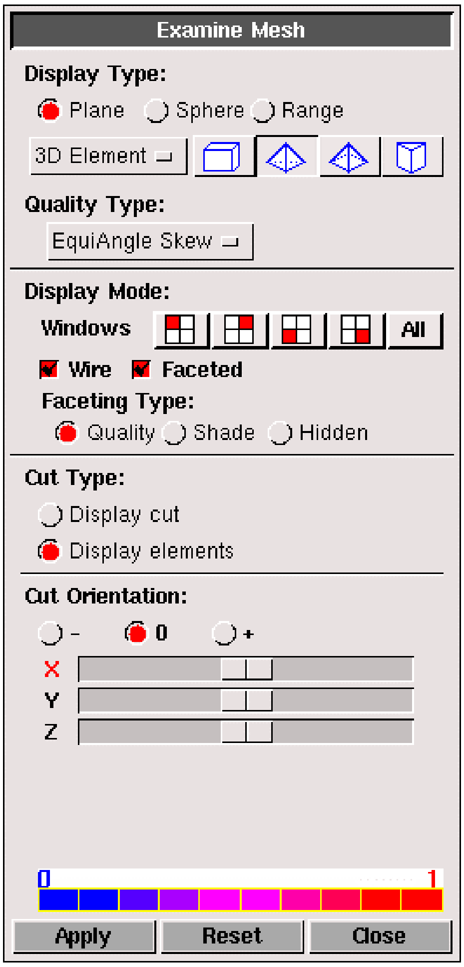
Figure 14-22: Meshed airplane flow volume

As an alternative to the procedure described in Steps 16, 17, and 18, above, you could attach a mesh size function to the flow volume, using the meshed airplane body surfaces as sources for the size function, and mesh the flow volume directly. Such a procedure would produce a smoother distribution of mesh elements within the volume but would significantly increase the meshing time and number of elements created.

Step 19: Examine the Mesh

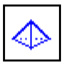
1. Click the **EXAMINE MESH**  command button on the **Global Control** toolpad.

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



   a) Retain the **Display Type:Plane** option at the top of the form.

      The **Examine Mesh** *form allows you to view mesh characteristics for various types of 3-D mesh elements. In this case, the volume mesh consists entirely of tetrahedral elements; therefore, you must specify the viewing of such elements.*

b) Retain the 3-D Element option, and click the tetrahedral button  at the top of the form.

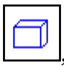
By default, GAMBIT *enables the brick button* *, thereby enabling the viewing of hexahedral elements. In this case, you can retain or disable (by clicking) the brick element button without affecting the mesh display—because the volume mesh does not contain any hexahedral elements.*

Figure 14-23 and Figure 14-24 show x-y and y-z cutting planes through the mesh.

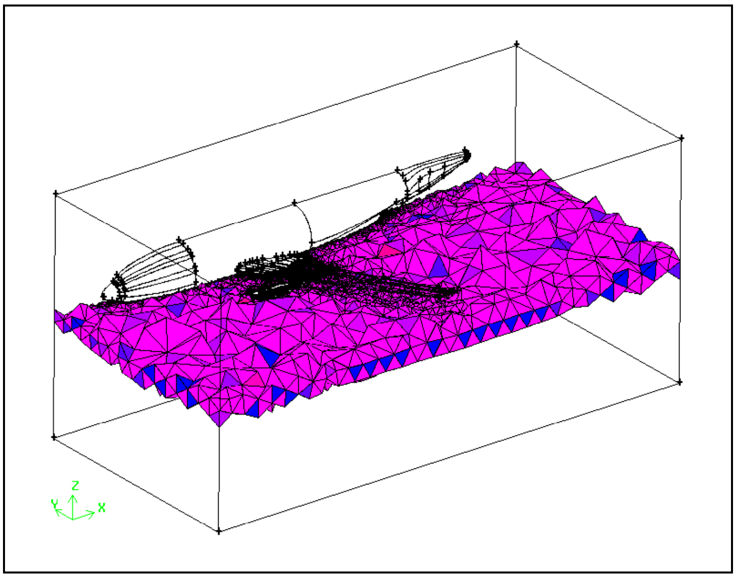


Figure 14-23: Cutting plane (*x*-*y*) showing tetrahedral elements

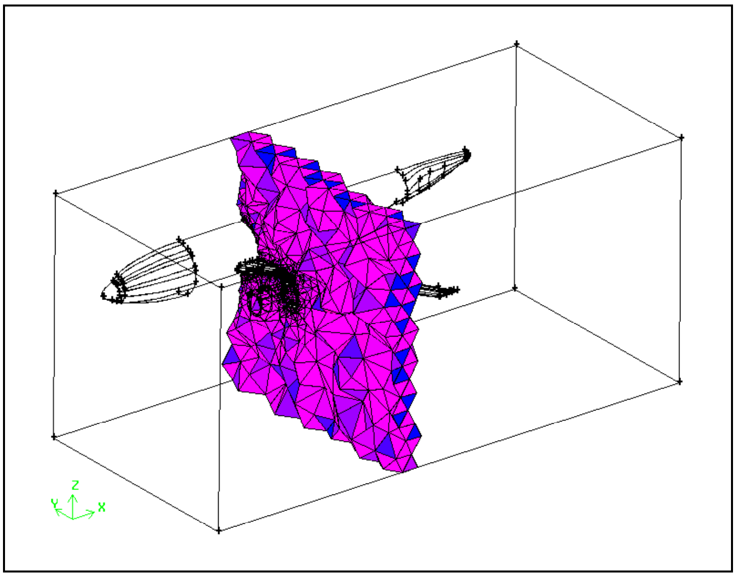


Figure 14-24: Cutting plane (*y*-*z*) showing tetrahedral elements

You can view element quality by range by selecting the **Display Type:**Range *option, and clicking one of the histogram bars near the bottom of the* **Examine Mesh** *form. Figure 14-25 shows the elements with* EquiAngle Skew *values between 0.7 and 0.8.*

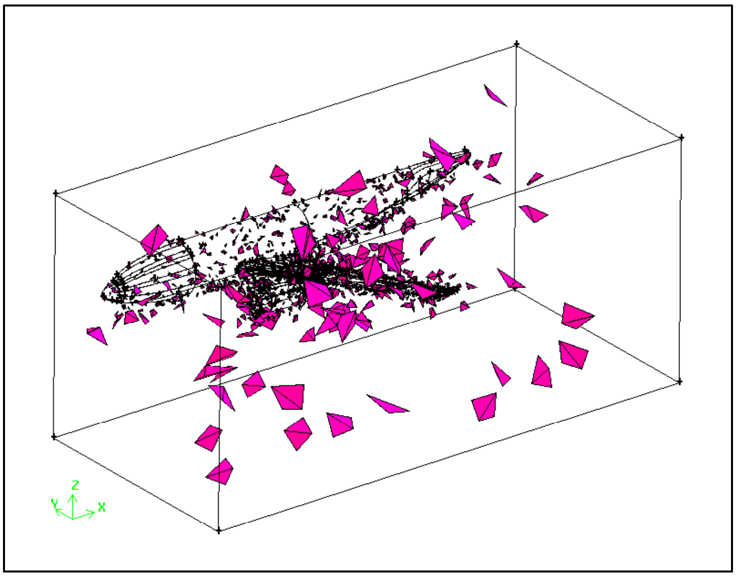


Figure 14-25: Display of elements with EquiAngle Skew values between 0.7 and 0.8

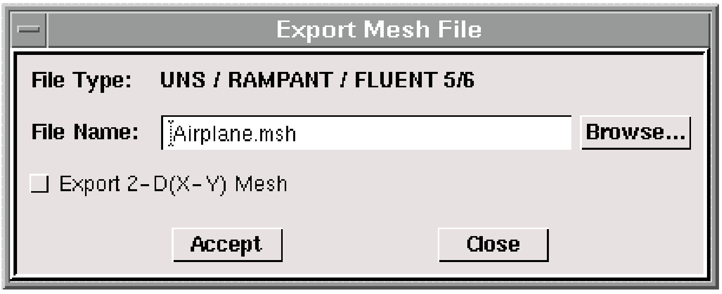
c) Click the **Close** button at the bottom of the **Examine Mesh** form to close the form.

Step 20: Export the Mesh and Save the Session

1. Export a mesh file for the airplane and flow volume.

   File $\longrightarrow$ **Export** $\longrightarrow$ **Mesh...**

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



1. Enter the **File Name** for the file to be exported (`Airplane.msh`).
2. Click **Accept** on the **Export Mesh File** form.

   The file will be written to your working directory.

3. Save the GAMBIT session and exit GAMBIT.

   File $\longrightarrow$ **Exit**

   GAMBIT *will ask you whether you wish to save the current session before you exit.*

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

14.5 Summary

This tutorial illustrated how to import geometry from an external CAD package as a STEP file, use GAMBIT healing and clean-up tools to make the geometry suitable for meshing, apply size functions, and mesh the geometry.