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.

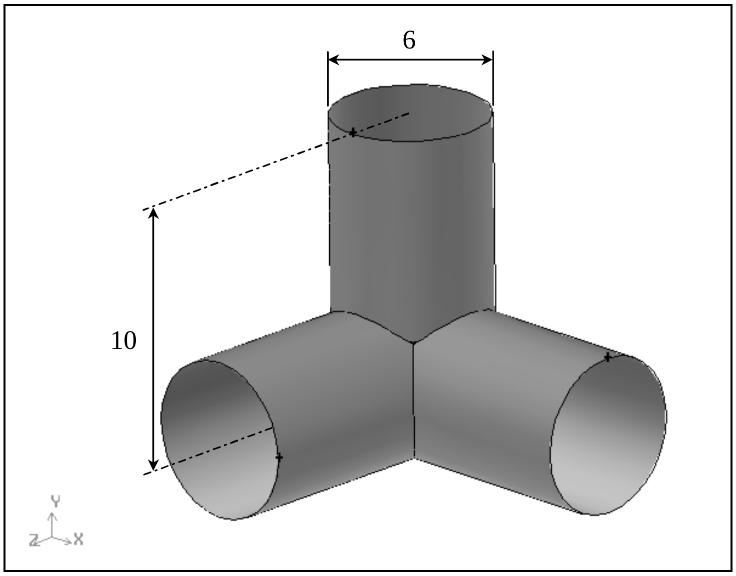


Figure 3-1: Problem specification

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.

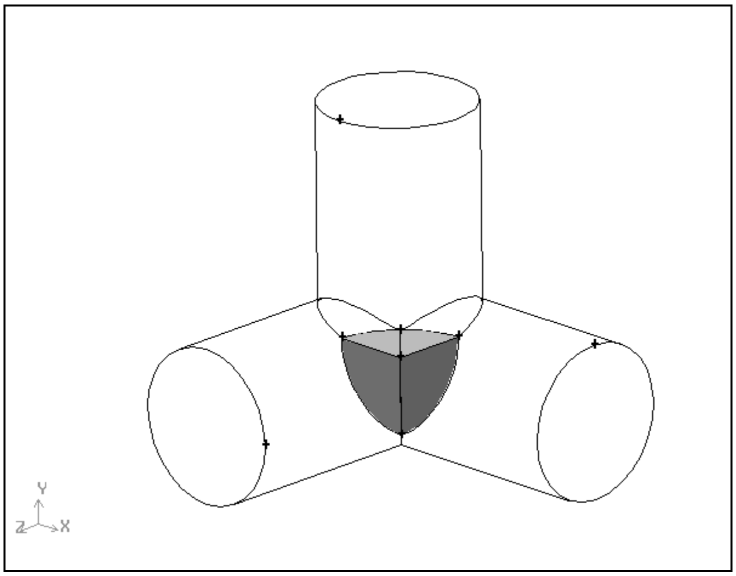


Figure 3-2: Decomposition of the three-pipe intersection geometry

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**

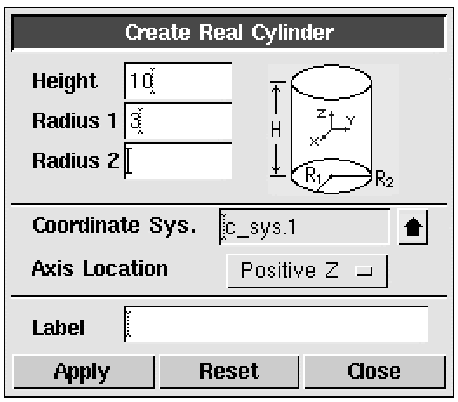
    The choice of a solver dictates the options available in various forms (for example, the boundary types available in the **Specify Boundary Types** *form). The solver currently selected is indicated at the top of the* GAMBIT *GUI.*

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.

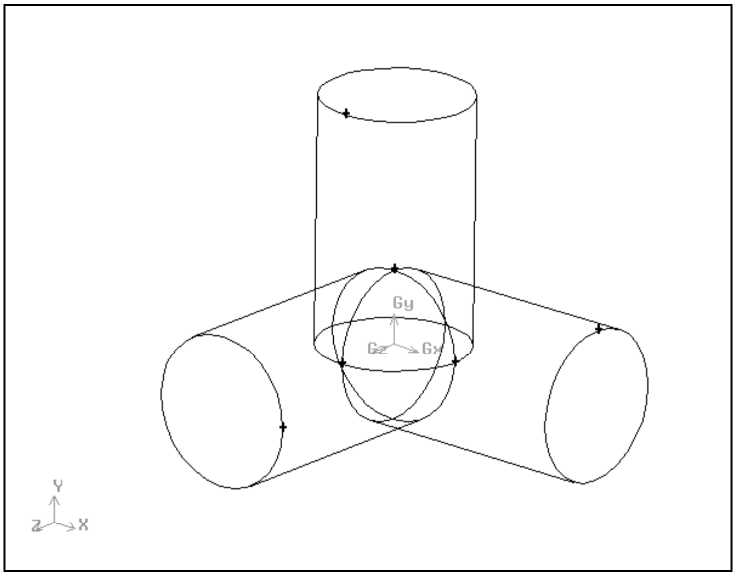
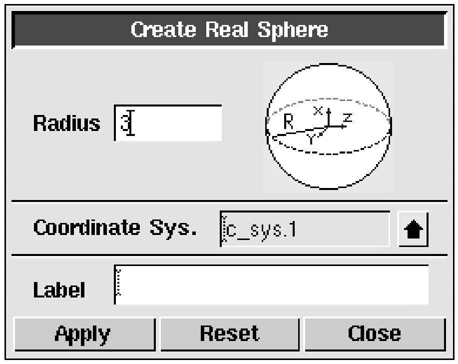


Figure 3-3: Three cylinders for the three-pipe intersection

3. Create a sphere to complete the basic geometry.

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

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



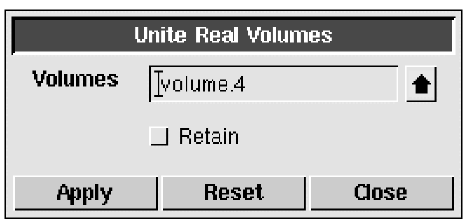
   a) Enter 3 for the **Radius**.

   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.

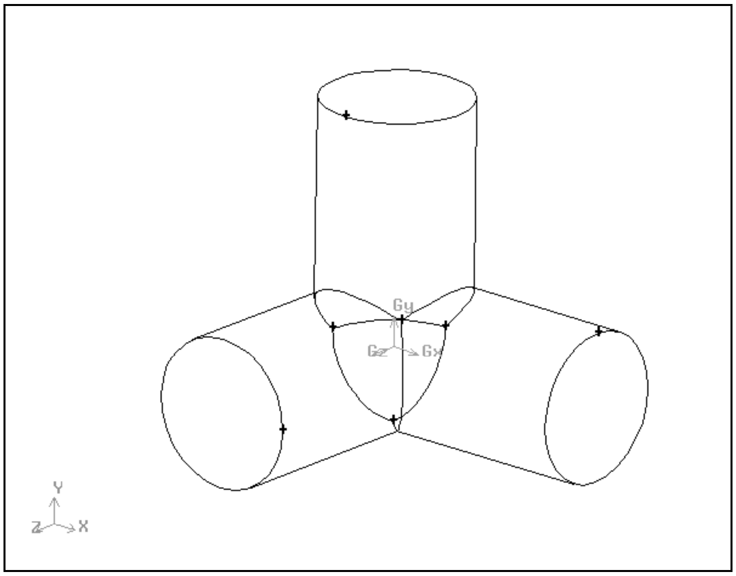
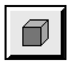
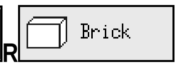


Figure 3-4: The completed geometry

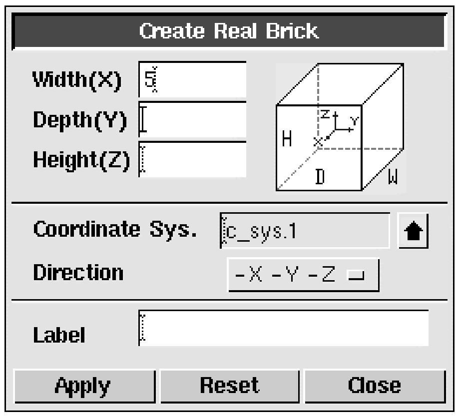
Step 3: Decompose the Geometry

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** 

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.

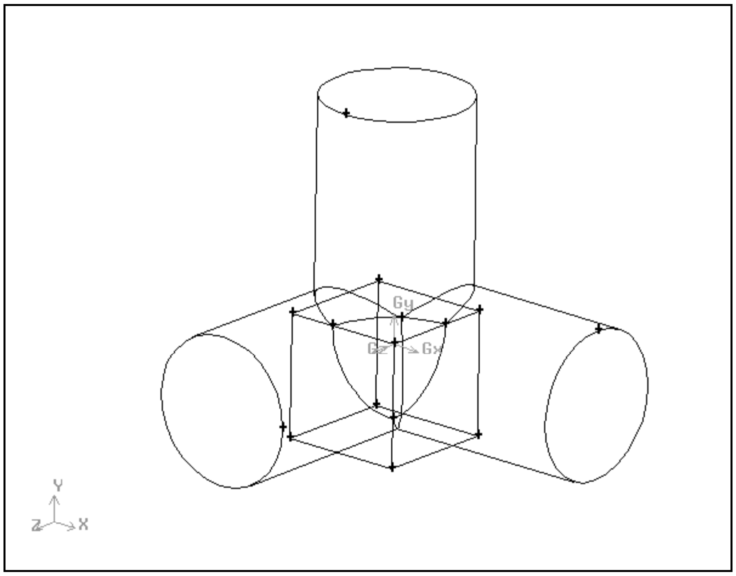


Figure 3-5: Three-pipe geometry and brick

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.

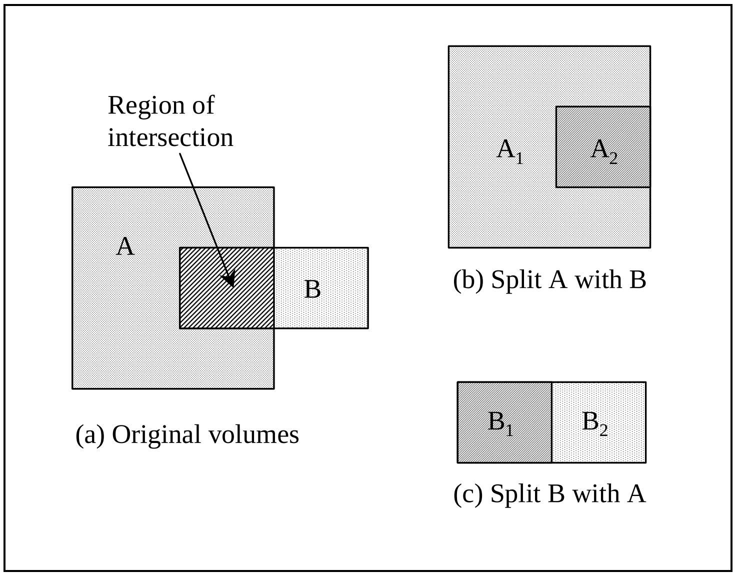
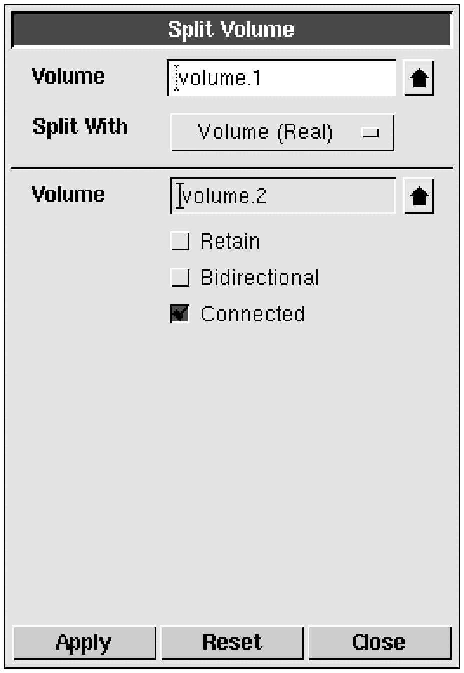


Figure 3-6: Splitting volumes

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.

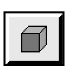
c) Left-click in the **Volume** list box located below the **Split With** section to make the **Volume** list box active.

d) Unselect the Bidirectional option.

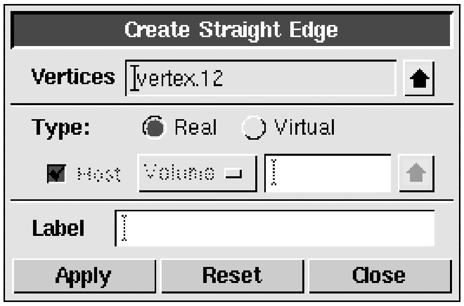
e) Select the brick and click **Apply** to accept the selection.

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.

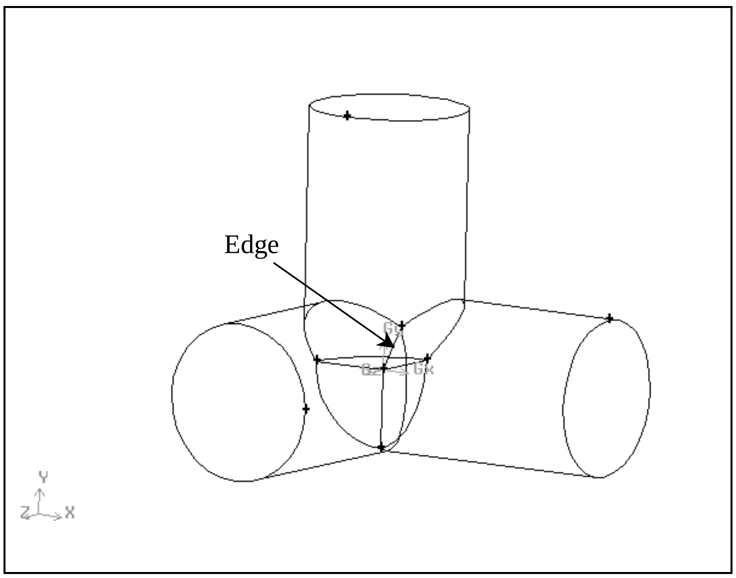
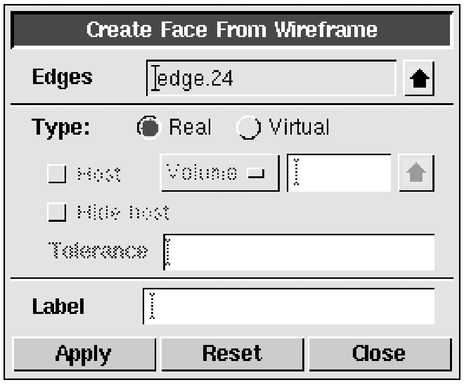


Figure 3-7: Straight edge created inside the volume

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.

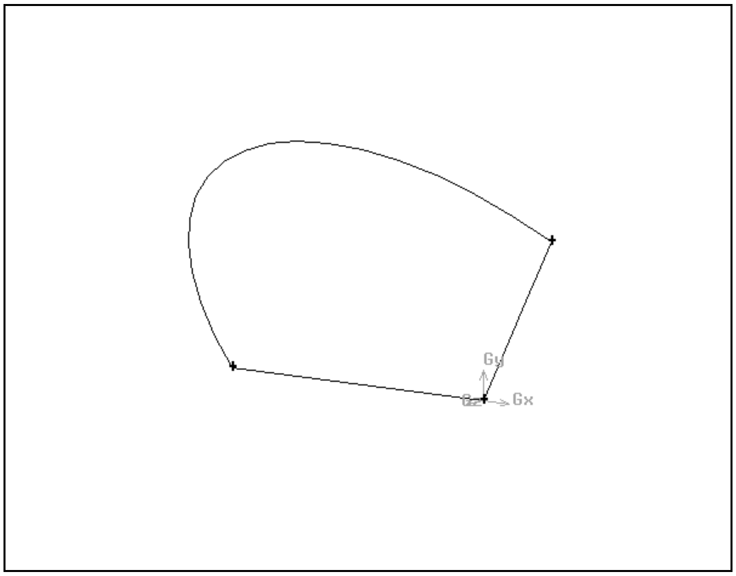


Figure 3-8: Three edges used to create a face

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.*

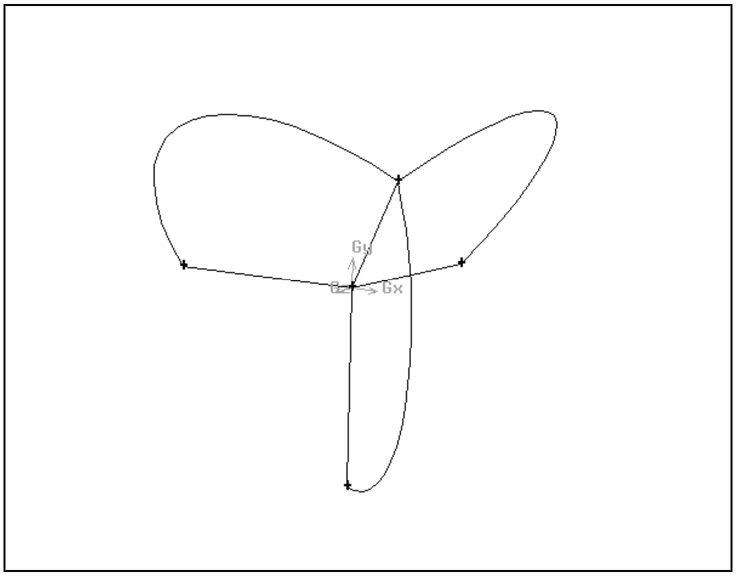
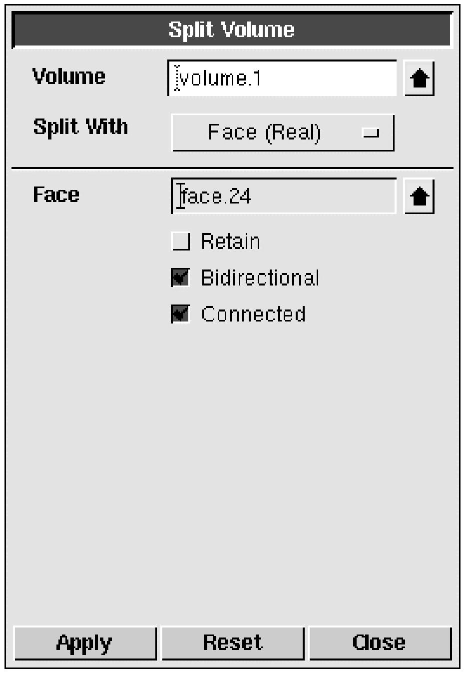


Figure 3-9: Three faces created inside the pipe intersection

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 deselect it and select the face next to it.*

e) Select the Bidirectional option.

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

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

h) Repeat Steps (a) through (f) 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.*

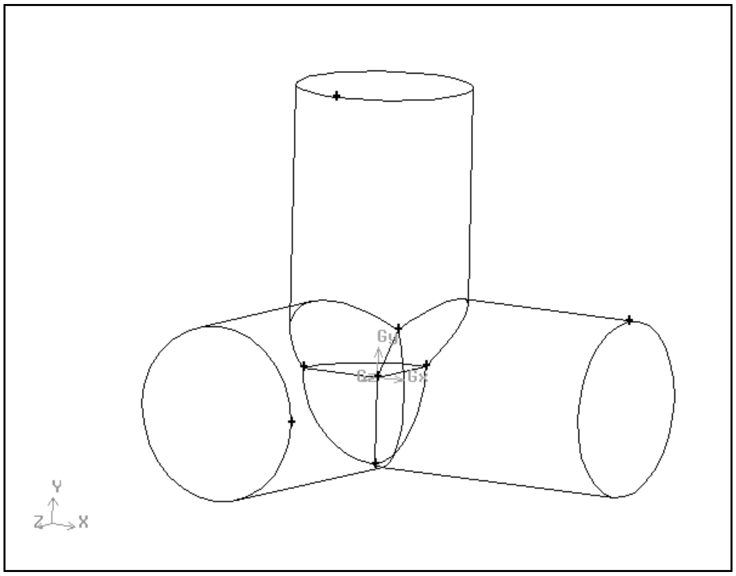


Figure 3-10: Decomposed geometry

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.

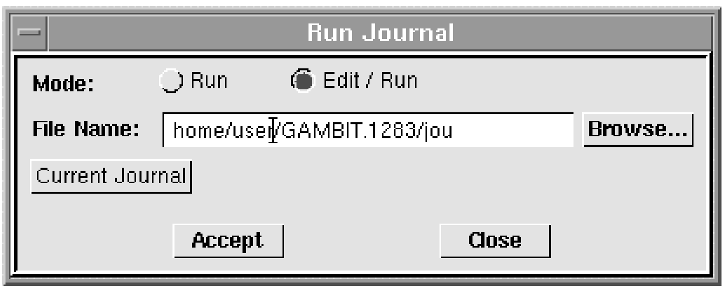
Every time a GUI operation is performed in GAMBIT, *the corresponding commands are automatically written to a journal file. This journal file, therefore, provides a backup copy of all the commands for the current session.*

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.*



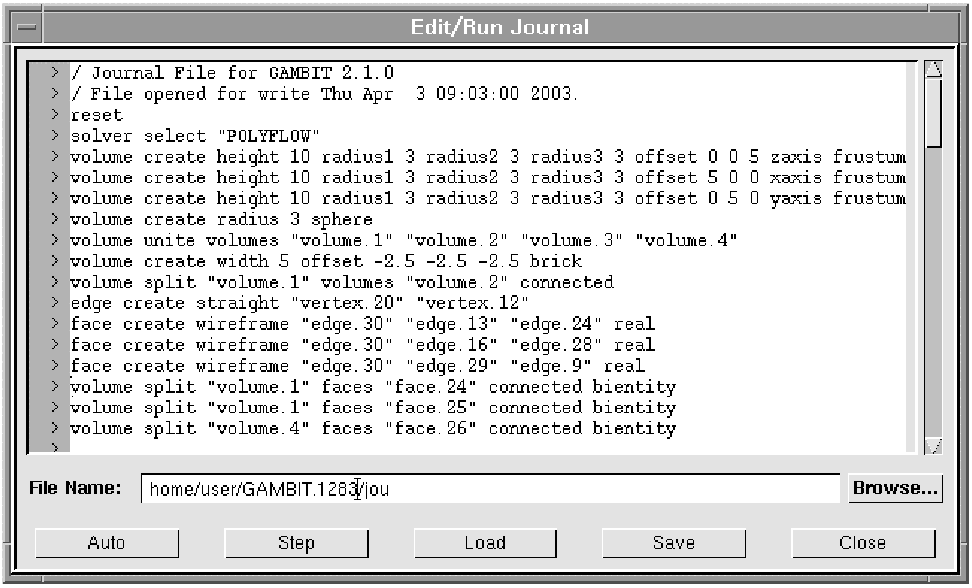
   a) Select the Edit / Run **Mode** option at the top of the form.

   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/deselect 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.*

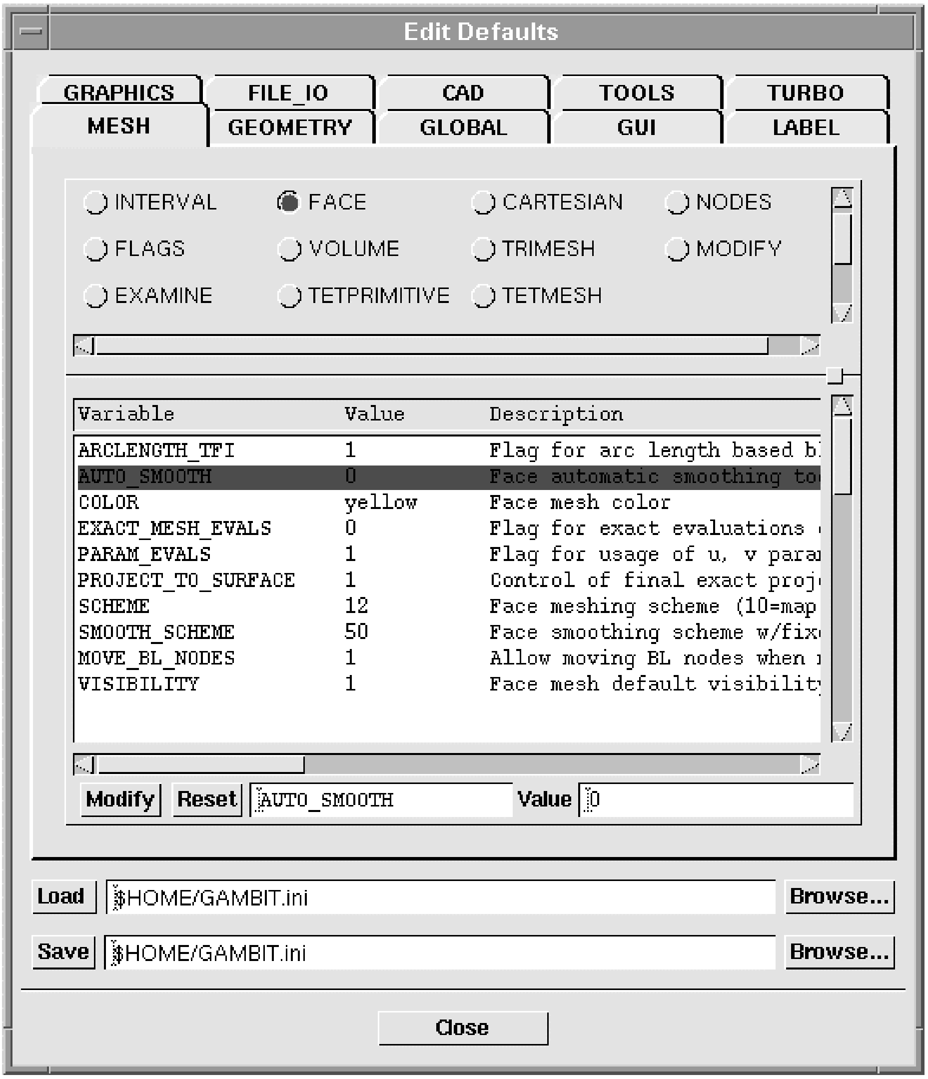
   e) Close the **Edit/Run Journal** form.

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.

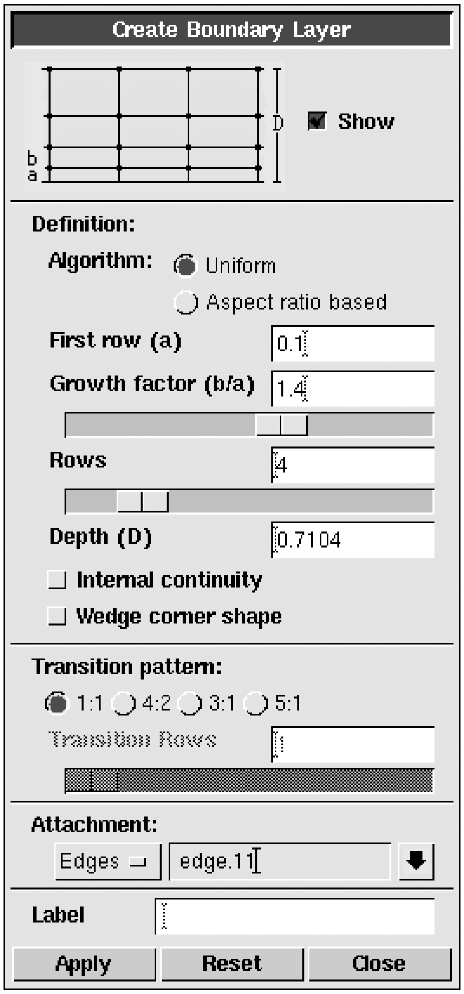
Step 6: Apply Boundary Layers at Walls

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.

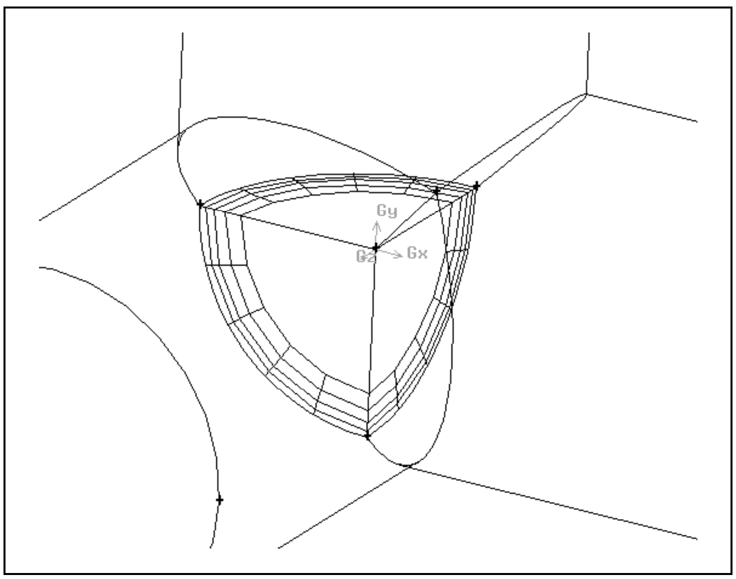


Figure 3-11: Boundary layer on three edges of the sphere octant

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.

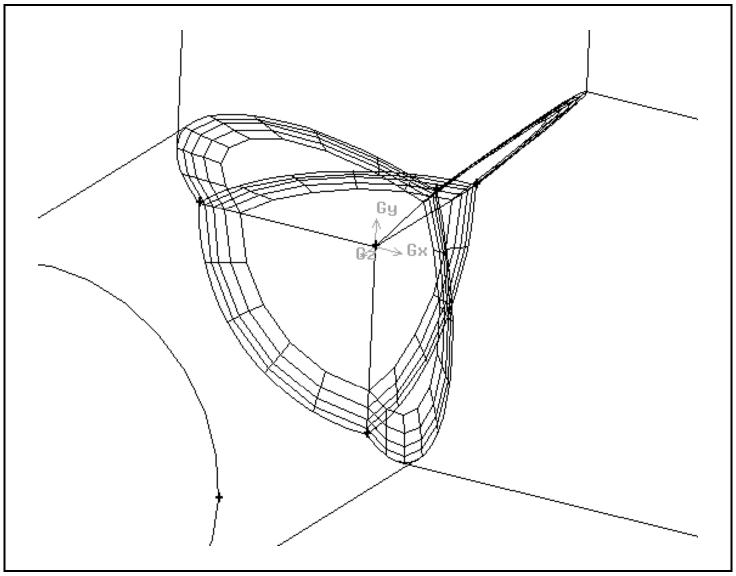
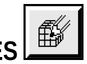


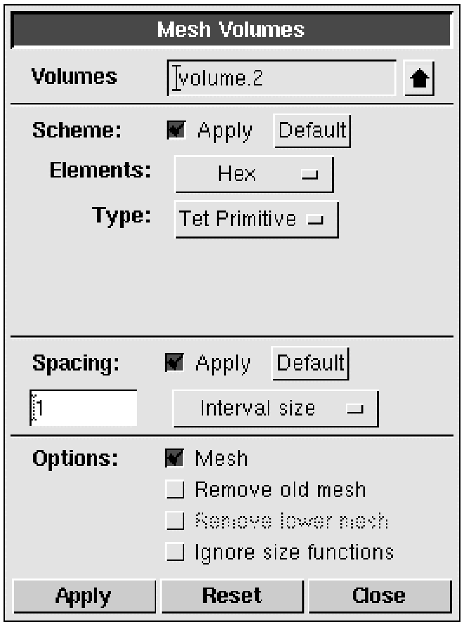
Figure 3-12: Boundary layers on the three edges where the pipes intersect

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. (<u>NOTE</u>: 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.

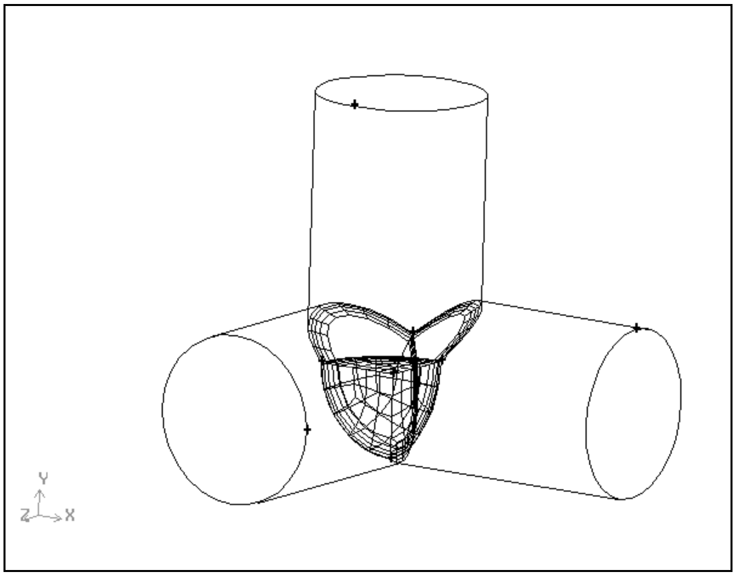


Figure 3-13: Mesh on sphere octant

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

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

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

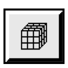
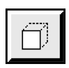
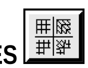
Step 8: Mesh the Pipe Volumes

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.*

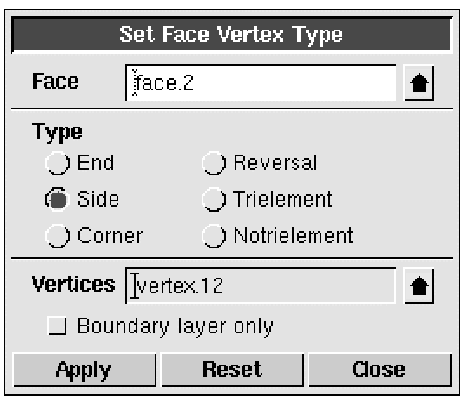
1. Mesh one of the pipes by changing the vertex type on the wall face to Side and then using the Cooper meshing scheme to mesh the volume.

   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 Side (the default) under **Type**.

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).

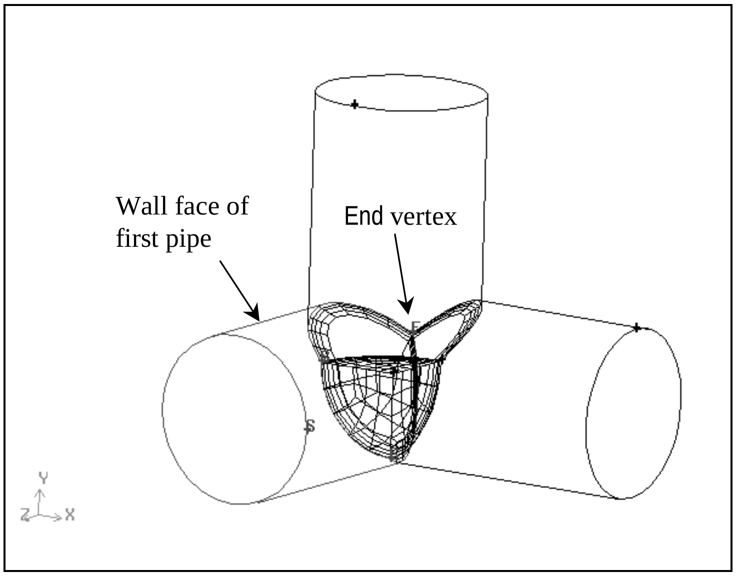


Figure 3-14: Wall face of the first pipe volume showing the End vertex

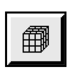
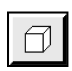
iii. Left-click in the **Vertices** list box.

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).

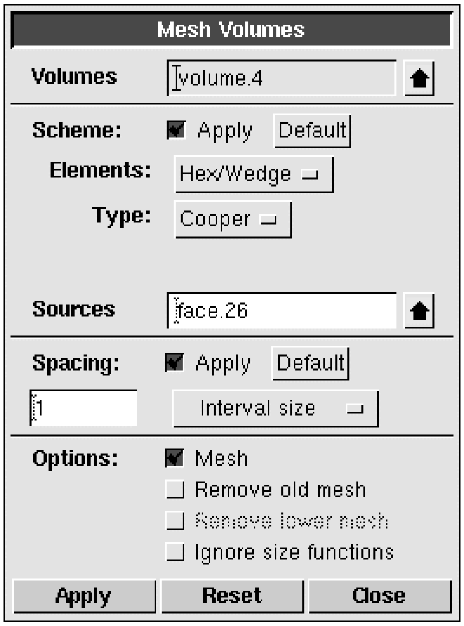
v. Click **Apply** in the **Set Face Vertex Type** form.

The vertex will be changed to **Type** *"S" for* Side. *You will only see the vertex label if you reselect the face. 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.

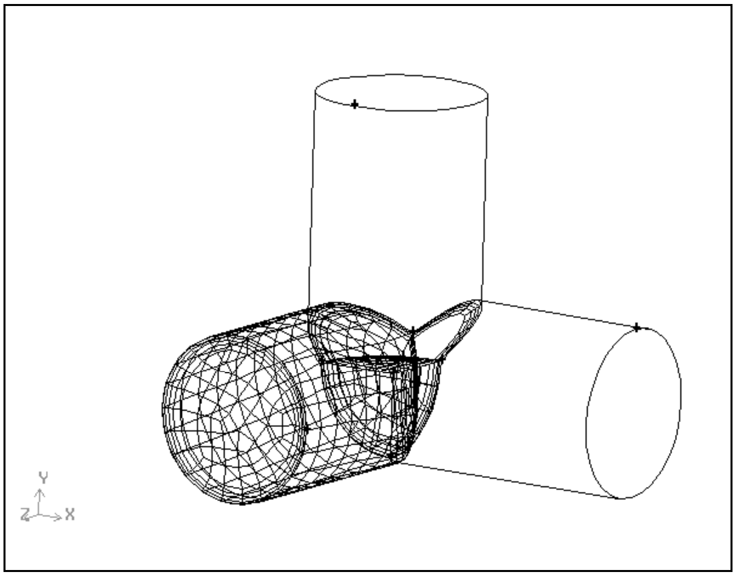


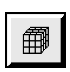
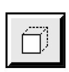
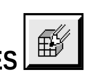
Figure 3-15: Pipe meshed by changing the vertex type on the wall face to Side and using the Cooper meshing scheme

! *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***.*

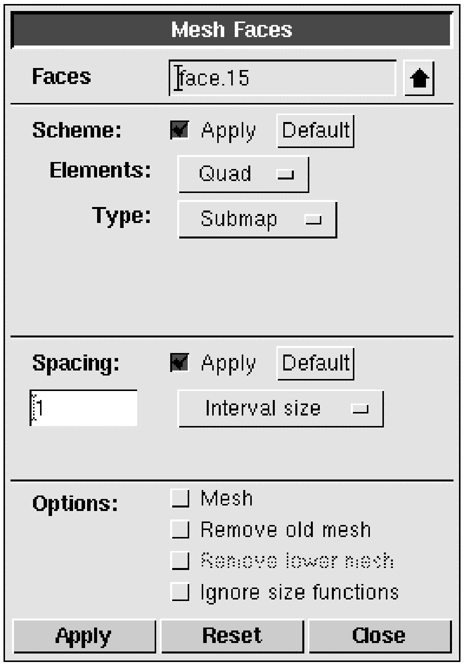
2. Mesh the second pipe using the Submap scheme on the wall face of the pipe and using the Cooper meshing scheme to mesh the volume.

   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.

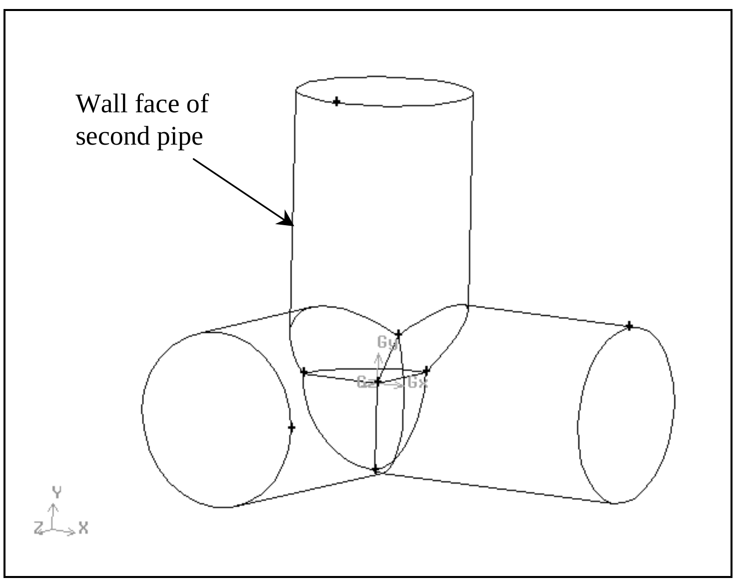


Figure 3-16: Wall face of the second pipe volume

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. Deselect the **Mesh** check box under **Options**.

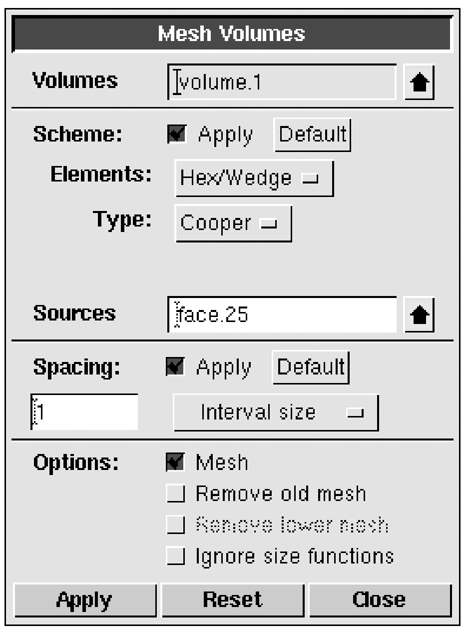
You deselected the **Mesh** *check box because at this point you do not want to mesh the face; you only want to apply the meshing* **Scheme** *to the face.* GAMBIT *will mesh the face using the specified* **Scheme** *when it meshes the pipe volume.*

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

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

MESH → **VOLUME** → **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.

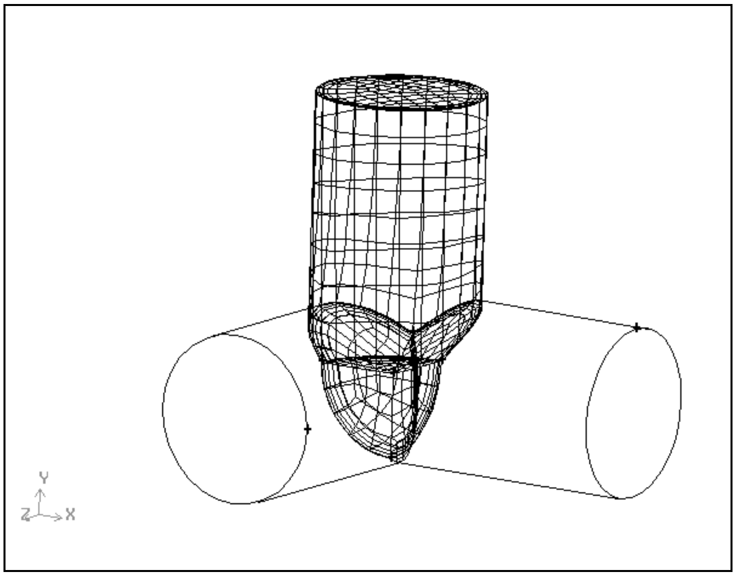
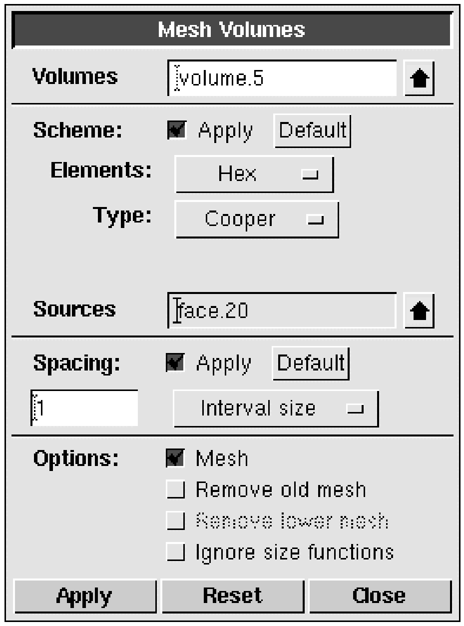


Figure 3-17: Pipe meshed using the Submap scheme for the wall face of the pipe and the project scheme for the volume

! *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 deselect it and select the face next to it. You can also click* **Reset** *in the* **Mesh Volumes** *form to deselect 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.*

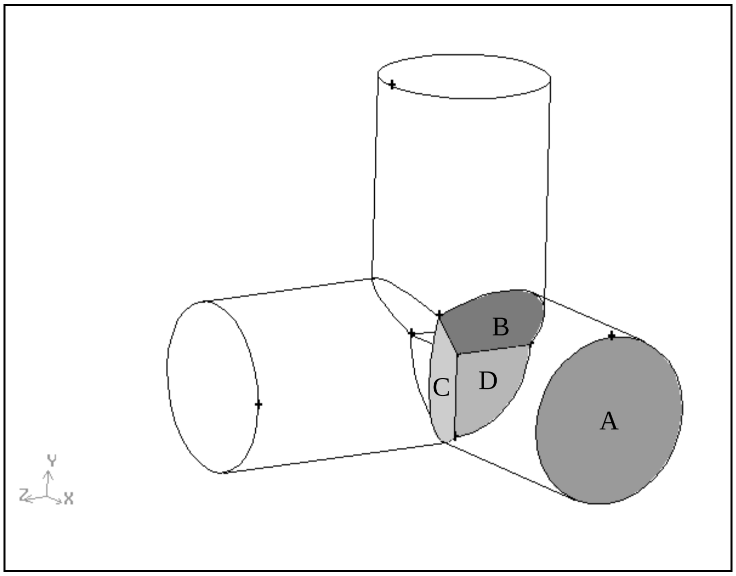


Figure 3-18: Source faces used to mesh one of the pipe volumes using the Cooper meshing scheme

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

4. Display all the face meshes for the three-pipe intersection.

   You will only display the face meshes at this point because displaying the volume meshes would create a cluttered view in the graphics window.

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

   b) Select the check box to the left of **Faces**.

   c) Select the On radio button to the right of **Mesh** near the bottom of the form.

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

The face meshes on the three-pipe intersection are shown in Figure 3-19.

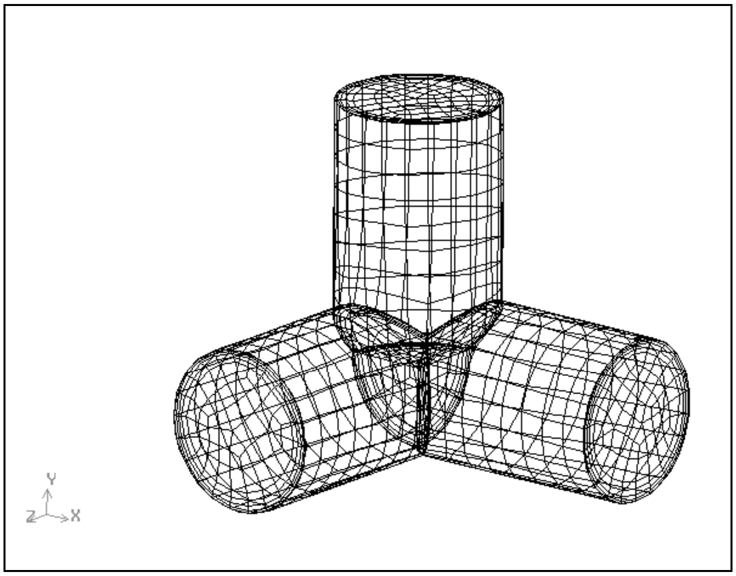
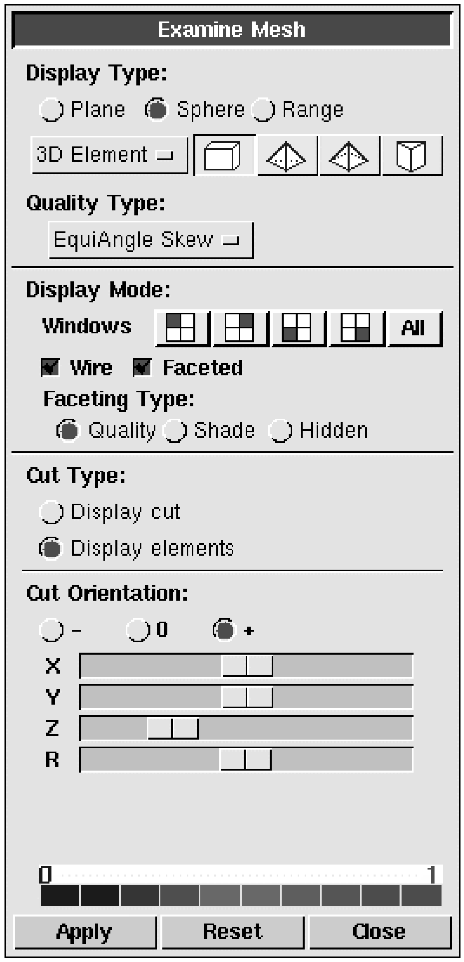


Figure 3-19: Face meshes for the three-pipe intersection

Step 9: Examine the Quality of the Mesh

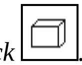
1. Select the **EXAMINE MESH**  command button at the bottom right of the **Global Control** toolpad.

   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 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.

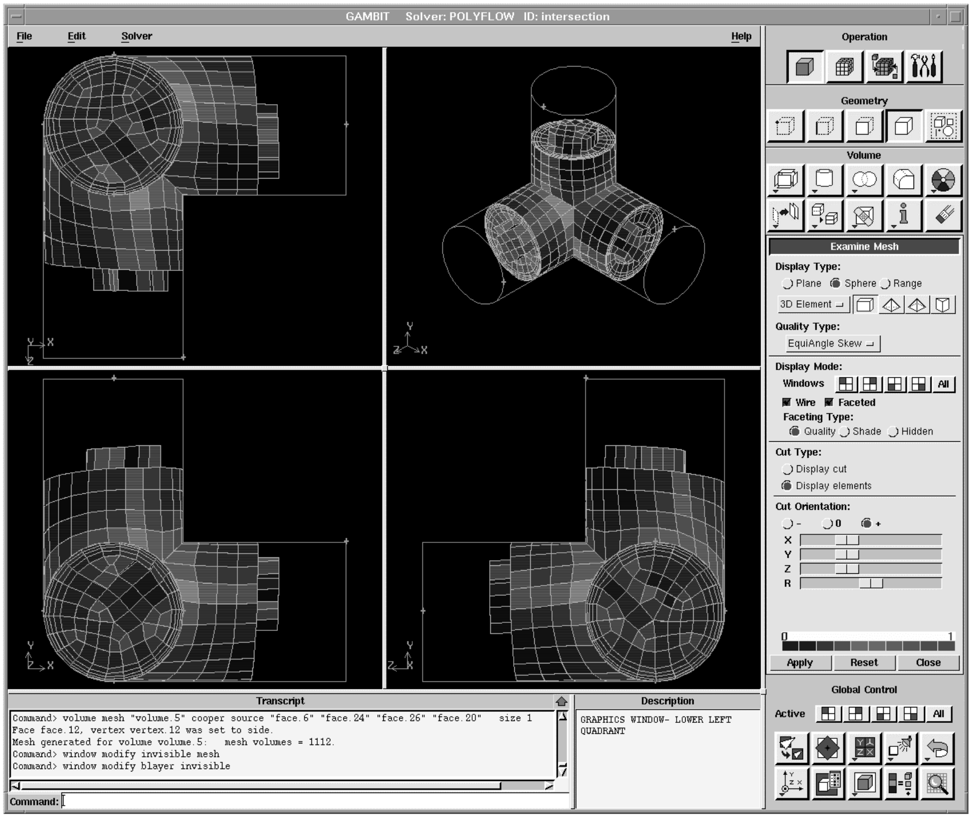


Figure 3-20: Spherical cut centered in the *x*, *y*, and *z* directions

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**.

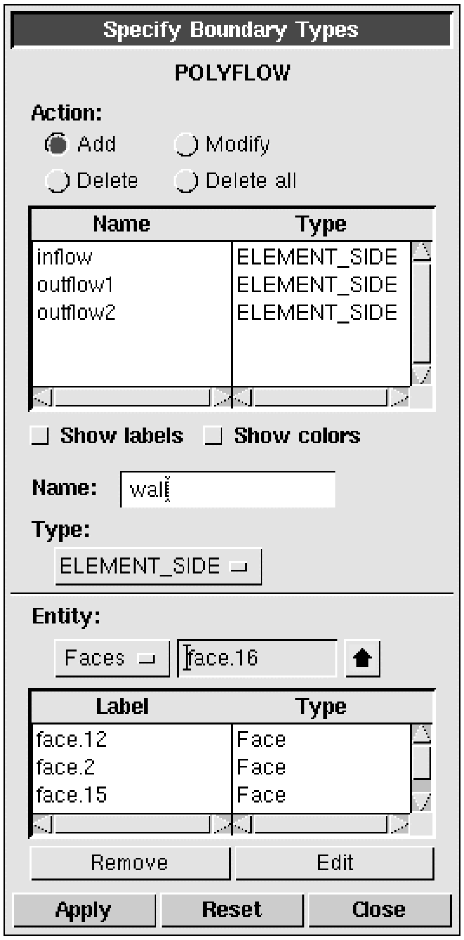
   d) Select the check box to the left of **B. Layers** and select Off from the option menu to the right of **Visible** near the bottom of the form.

   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 `inflow` in the **Name** text entry box.

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.

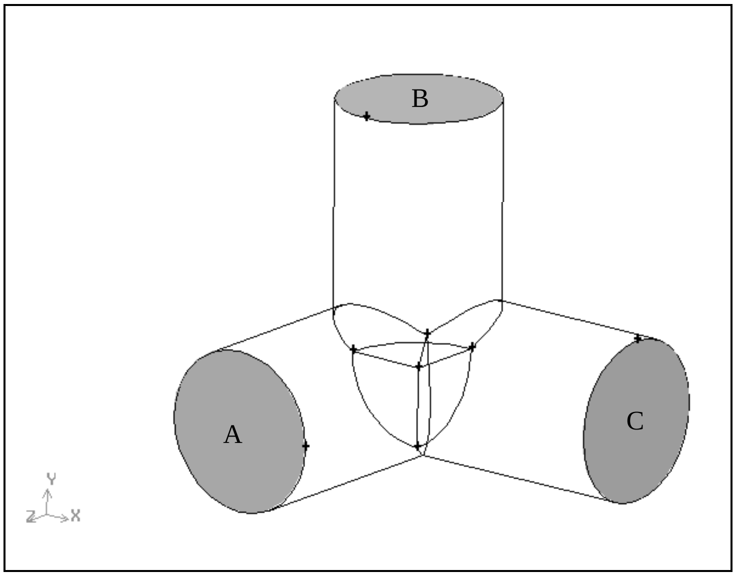


Figure 3-21: Boundary types for faces of the three-pipe intersection

b) Define the two flow outlets.

   i. Enter the name `outflow1` 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** `outflow2`.

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

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

   ii. Check that ELEMENT_SIDE is still selected in the **Type** option menu and pick all 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.)*

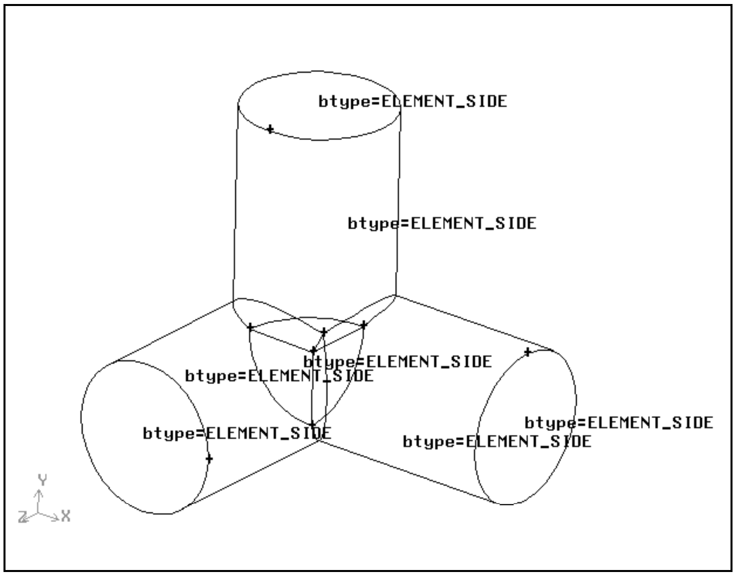


Figure 3-22: Boundary types for the three-pipe intersection

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.