Tutorial: Creating and Meshing Basic Geometry in Gambit

2005 Cornell University
BEE453, Professor Ashim Datta
Authored by Vineet Rakesh
Software: GAMBIT 2.1.6/ FIDAP 8.7.2 /Presto 1.2

Problem Specification

The model consists of an intersecting brick and elliptical cylinder. The basic geometry is shown schematically in Figure 1.1

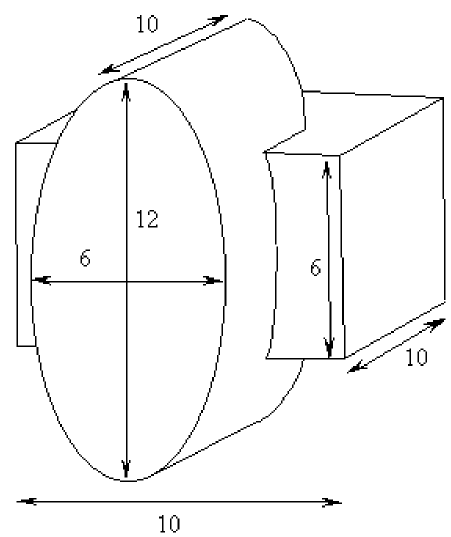


Figure 1.1: Problem specification

Strategy

This 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.

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

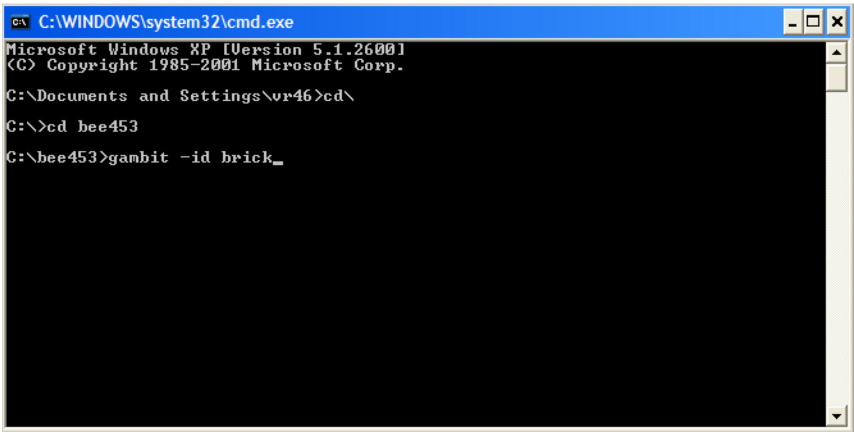
- Create two volumes (a brick and an elliptical cylinder).
- Unite the two volumes.
- Automatically generate the mesh.
- Examine the quality of the resulting mesh.

Step 1: Run the software GAMBIT to create the geometry and to mesh it

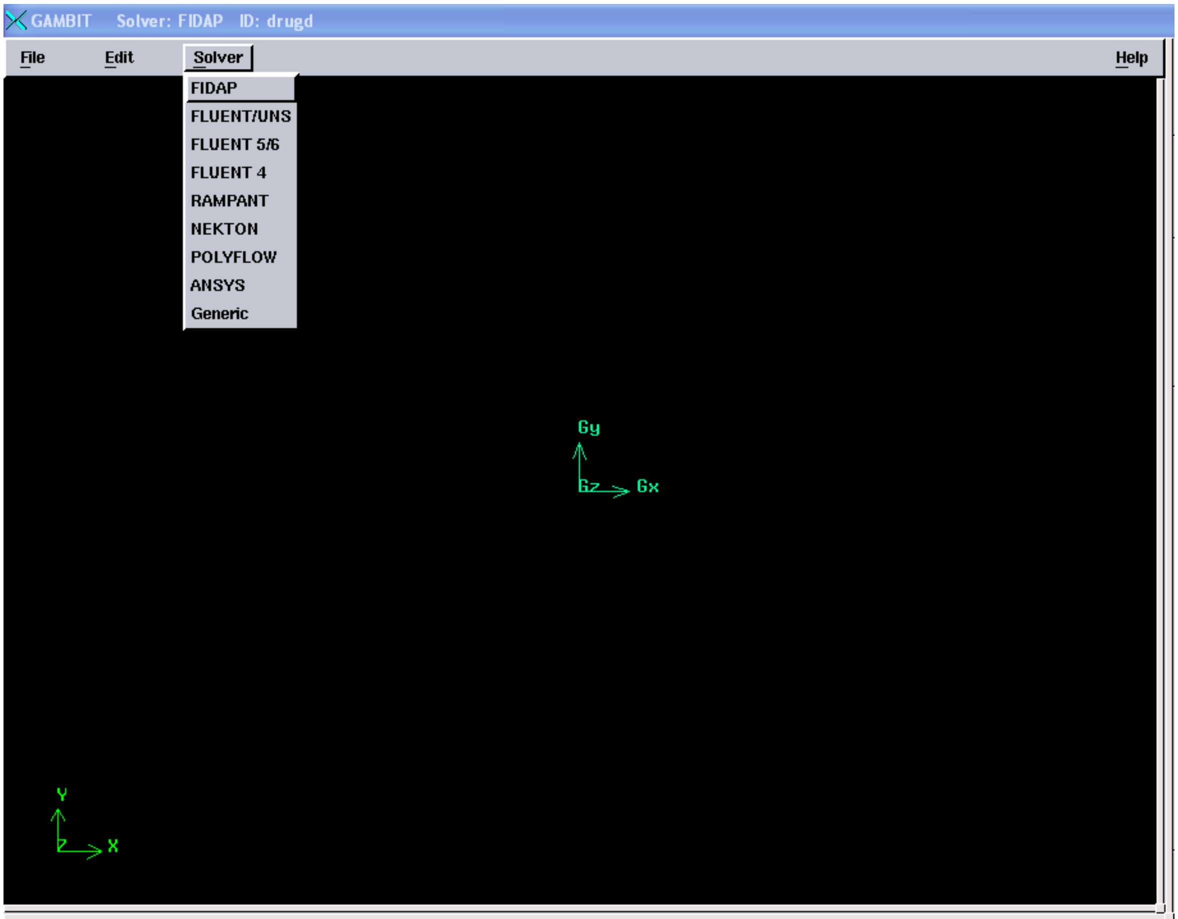
In the Command Prompt, type: **gambit –id brick**

brick is the *filename*

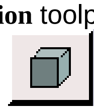
Remember: The *filename* should be at the most 7 characters long.



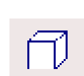
Now Gambit is launched. Click on **Solver** menu at the top of the Gambit window and choose **FIDAP**.



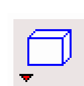
Step 2: Create a Brick

1) 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.

2) 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.

3) 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.

4) 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.
5) Use the *Tab* key on the keyboard to move to the Depth text entry box, and enter `6` for the Depth of the brick.
6) Use the *Tab* key on the keyboard to move to the Height text entry box, and enter `10` for the Height of the brick.

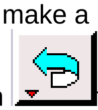
The text entry box for Height can be left blank; GAMBIT will set this value to be the same value as the Width by default.

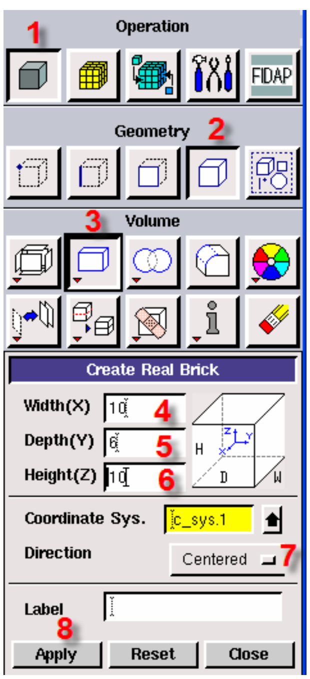
7) Select Centered from the option menu to the right of Direction.

Hold down the left mouse button on the option button to the right of Direction until the option menu appears.
Select Centered from the list.

8) 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 1.2.

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.



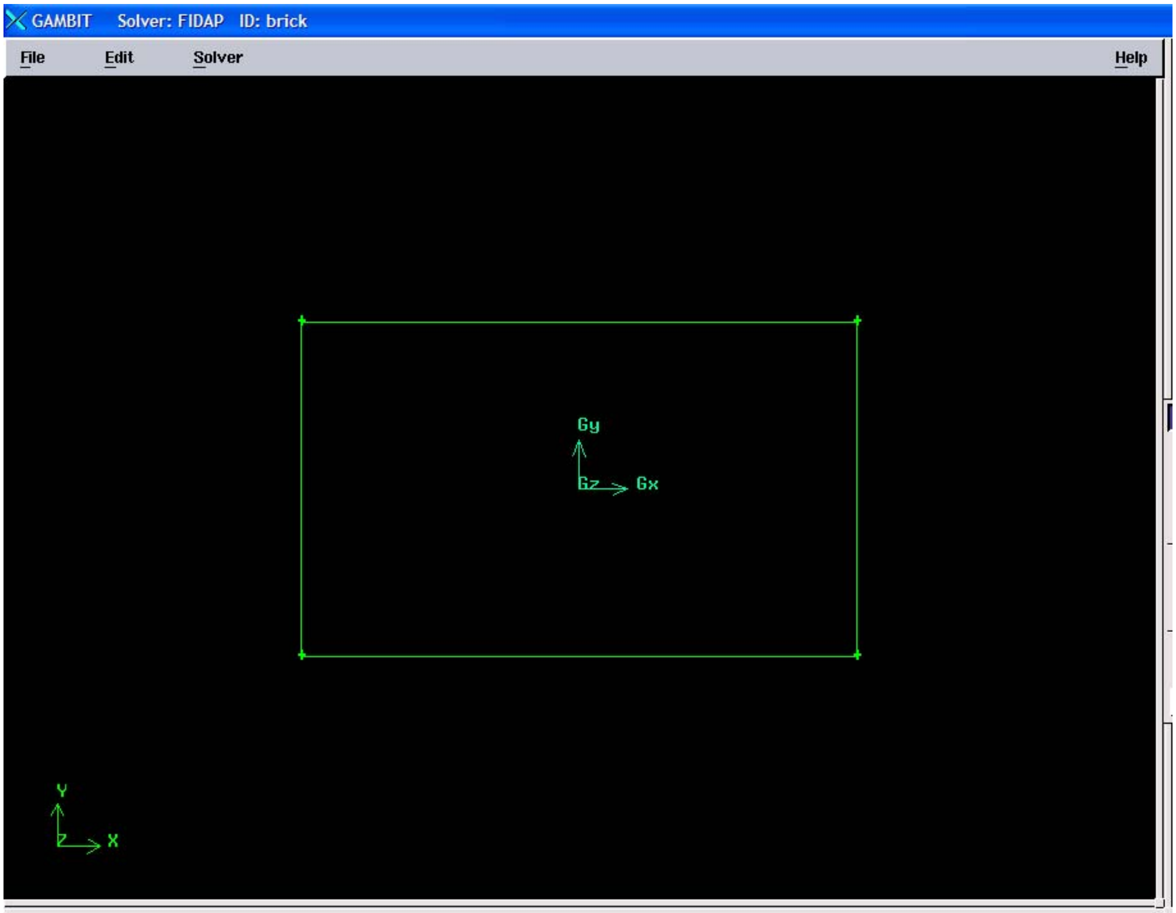
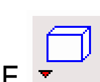
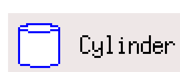


Figure 1.2: Rectangular brick volume (side view)

Step 3: Create an Elliptical Cylinder

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

2) Select the CREATE REAL CYLINDER option 

from the resulting menu.
This action opens the ***Create Real Cylinder*** *form.*

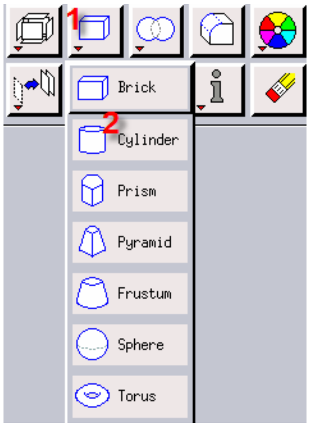
3) In the **Create Real Cylinder** form, enter a Height of 10.

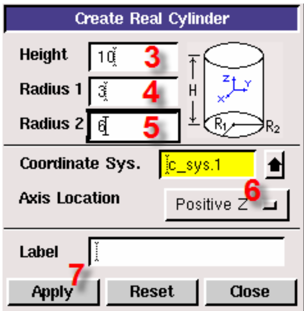
4) Enter a value of 3 for Radius 1.

5) Enter a value of 6 for Radius 2.

6) Retain the default Axis Location of Positive Z.

7) Click Apply.


1
Brick
2
Cylinder
Prism
Pyramid
Frustum
Sphere
Torus


Create Real Cylinder
Height
10
3
Radius 1
3
4
Radius 2
6
5
H
R1
R2
Coordinate Sys.
c_sys.1
Axis Location
Positive Z
6
Label
7
Apply
Reset
Close

The brick and elliptical cylinder are shown in Figure 1.3.

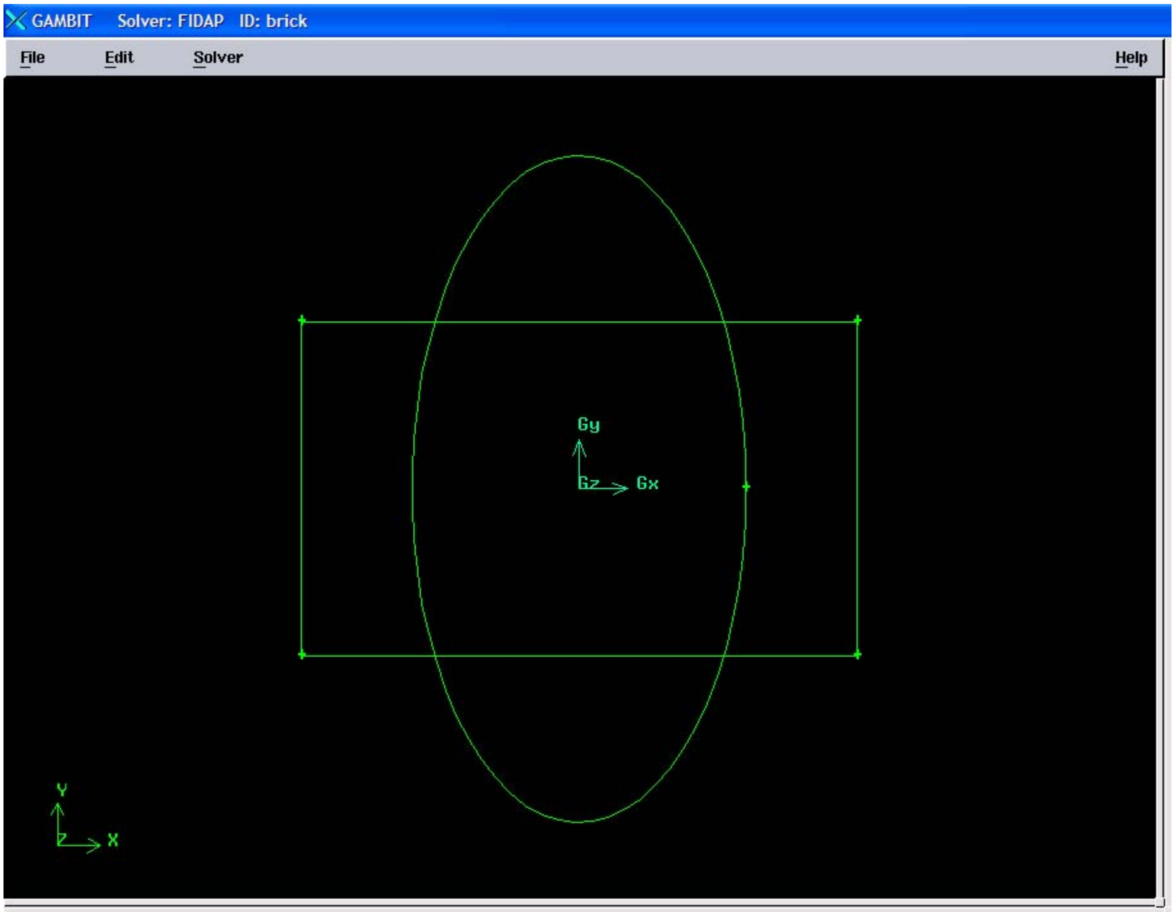
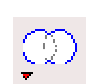
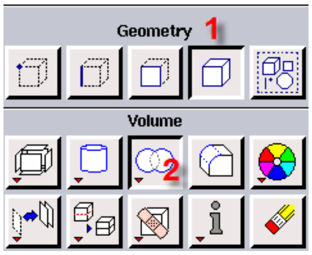


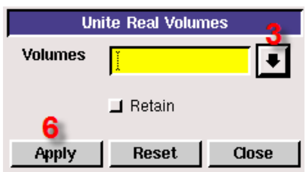
Figure 1.3: Brick and elliptical cylinder

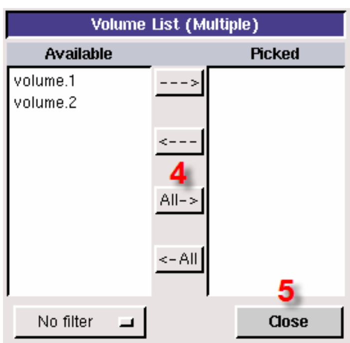
Step 4: Unite the Two Volumes

1) Under the **Geometry** Panel, click on the **Volume** Command Button if it is not already depressed.
2) Under the **Volume** Panel, click BOOLEAN Operations (Unite) Button  *This command sequence opens the* Unite Real Volumes *form.*
3) In the **Unite Real Volumes** Form, click on the arrow next to Volumes.
4) In the **Volume List** Window, click on **All**. This operation selects the two volumes (cylinder and brick). Alternatively, to select the brick, 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. Then, *Shift*-left-click the elliptical cylinder in the graphics window to select it.
5) Click on **Close**.

6) In the **Unite Real Volumes** Form, click on **Apply**. *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 volume is shown in Figure 1.4. You can rotate the display (as shown in Figure 1.4) 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.

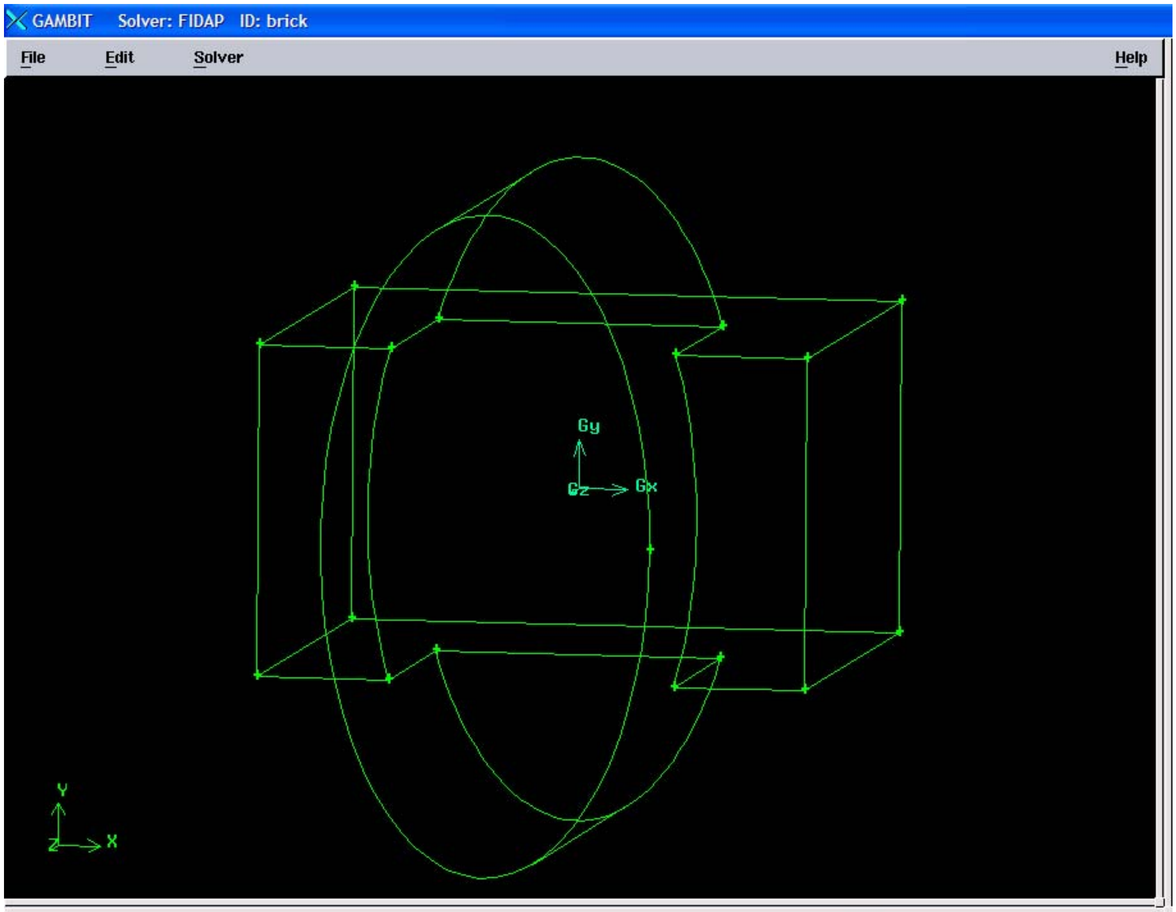


Figure 1.4: Brick and elliptical cylinder united into one volume

---

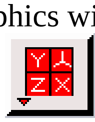
Step 5: Manipulate the Display

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.*

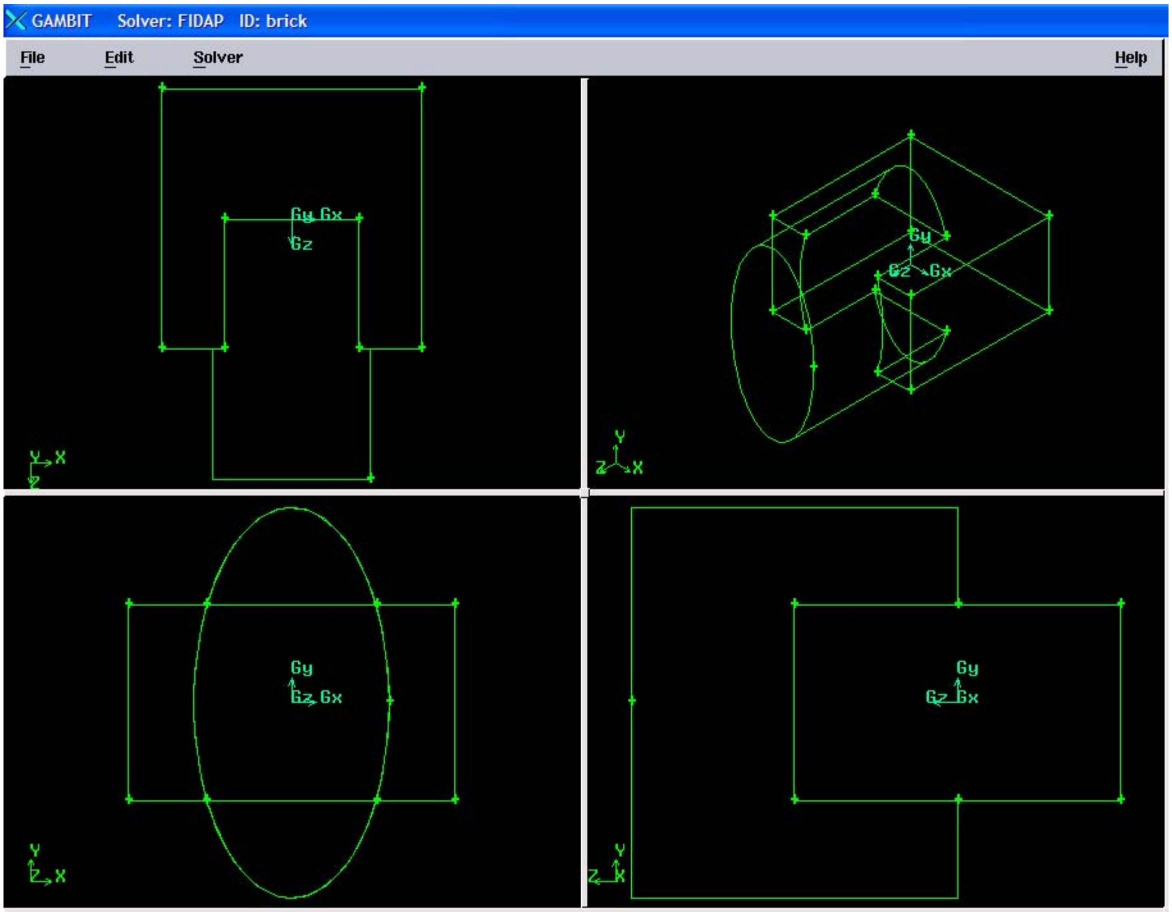
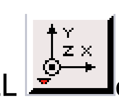


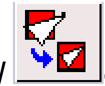
Figure 1.5: GAMBIT GUI-four graphics-window quadrants

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 6: Mesh the Volume

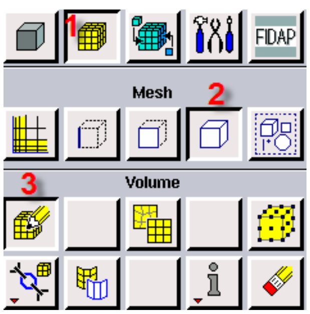
1) Under the **Operations** panel, click on **Mesh**

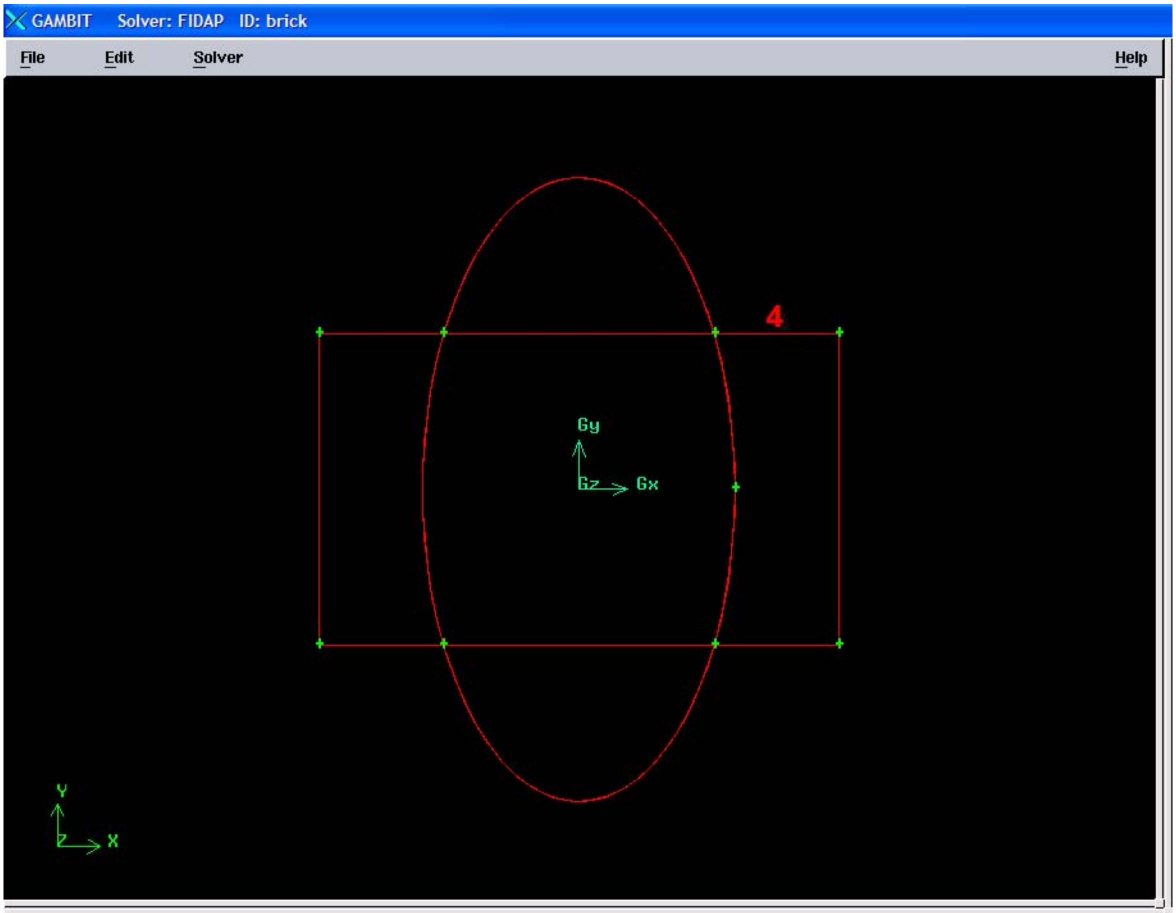
2) Under the **Mesh** panel, click on **Volume**

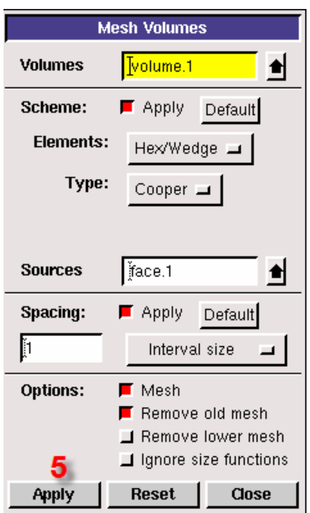
3) Under the **Volume** panel, click on **Mesh Volume**. *This command sequence opens the* **Mesh Volumes** *form.*
4) *Shift*-left-click the volume in the graphics window. GAMBIT *will automatically choose the* Cooper **Scheme Type** *as the meshing tool to be used, and will use an* **Interval size** *of* 1 *(the default) under* **Spacing***.*
5) 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 1.6.




GAMBIT Solver: FIDAP ID: brick
File
Edit
Solver
Help
4
Gy
Gz
Gx
Y
Z
X


Mesh Volumes
Volumes
volume.1
Scheme:
Apply
Default
Elements:
Hex/Wedge
Type:
Cooper
Sources
face.1
Spacing:
Apply
Default
1
Interval size
Options:
Mesh
Remove old mesh
Remove lower mesh
Ignore size functions
5
Apply
Reset
Close

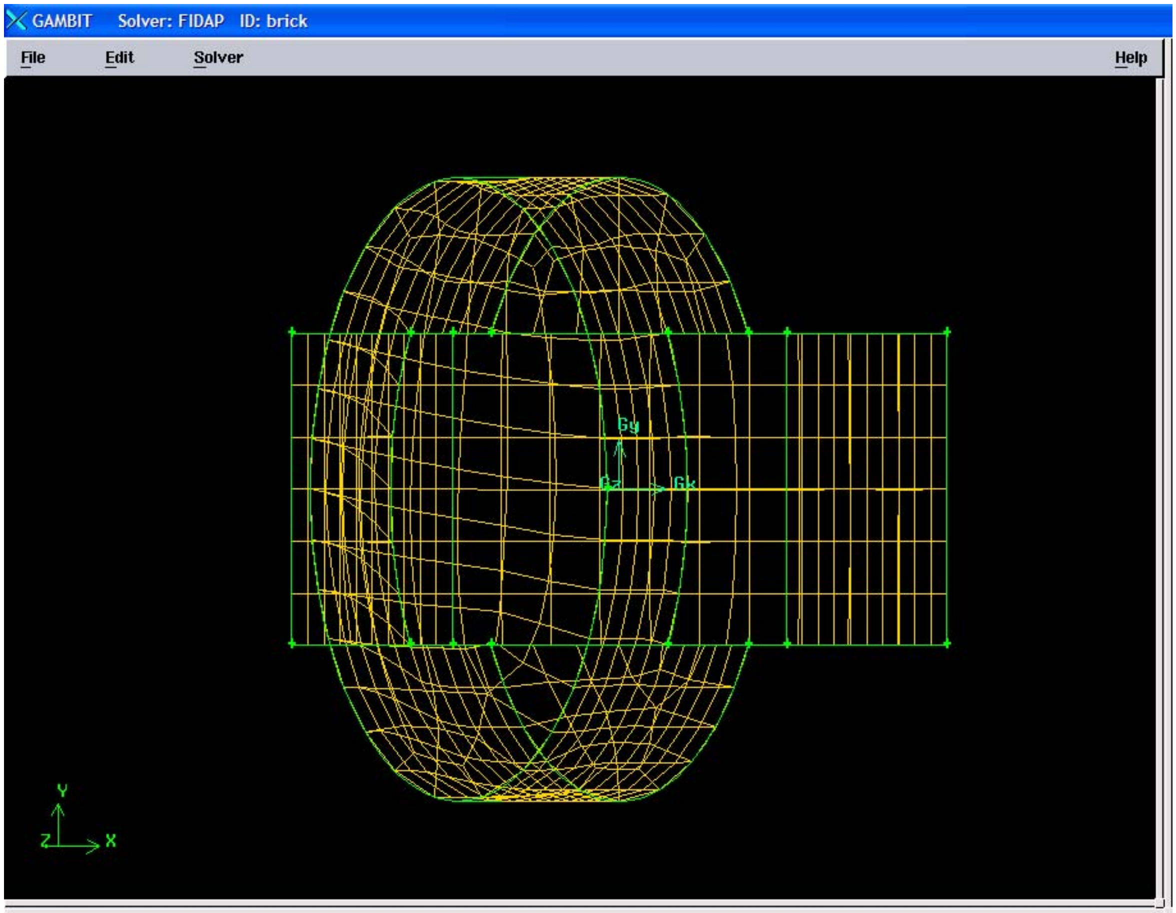
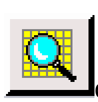


Figure 1.6: Meshed volume

Step 7: Examine the Mesh

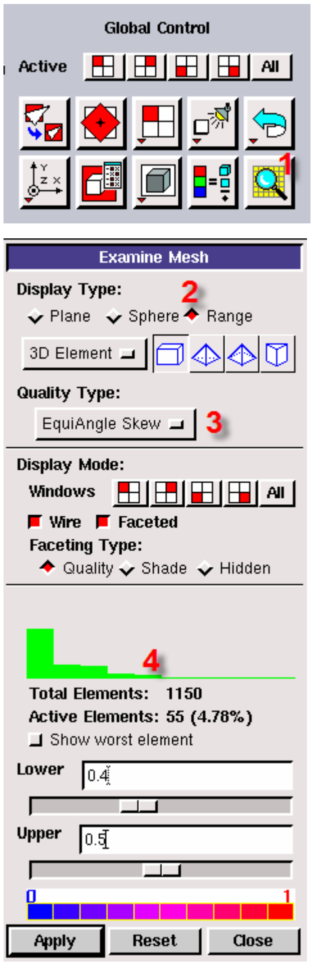
It is important that you check the quality of the resulting mesh, because properties such as skewness can greatly affect the accuracy and robustness of the CFD solution. GAMBIT provides several quality measures (sometimes called "metrics") with which you can assess the quality of your mesh. In the case 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.

1) Select the **EXAMINE MESH**  command button at the bottom right of the **Global Control** toolpad. *This action opens the* Examine Mesh *form.*
2) 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*  *.*

3) Select EquiAngle Skew from the **Quality Type** option menu.
4) 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 1.7 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).

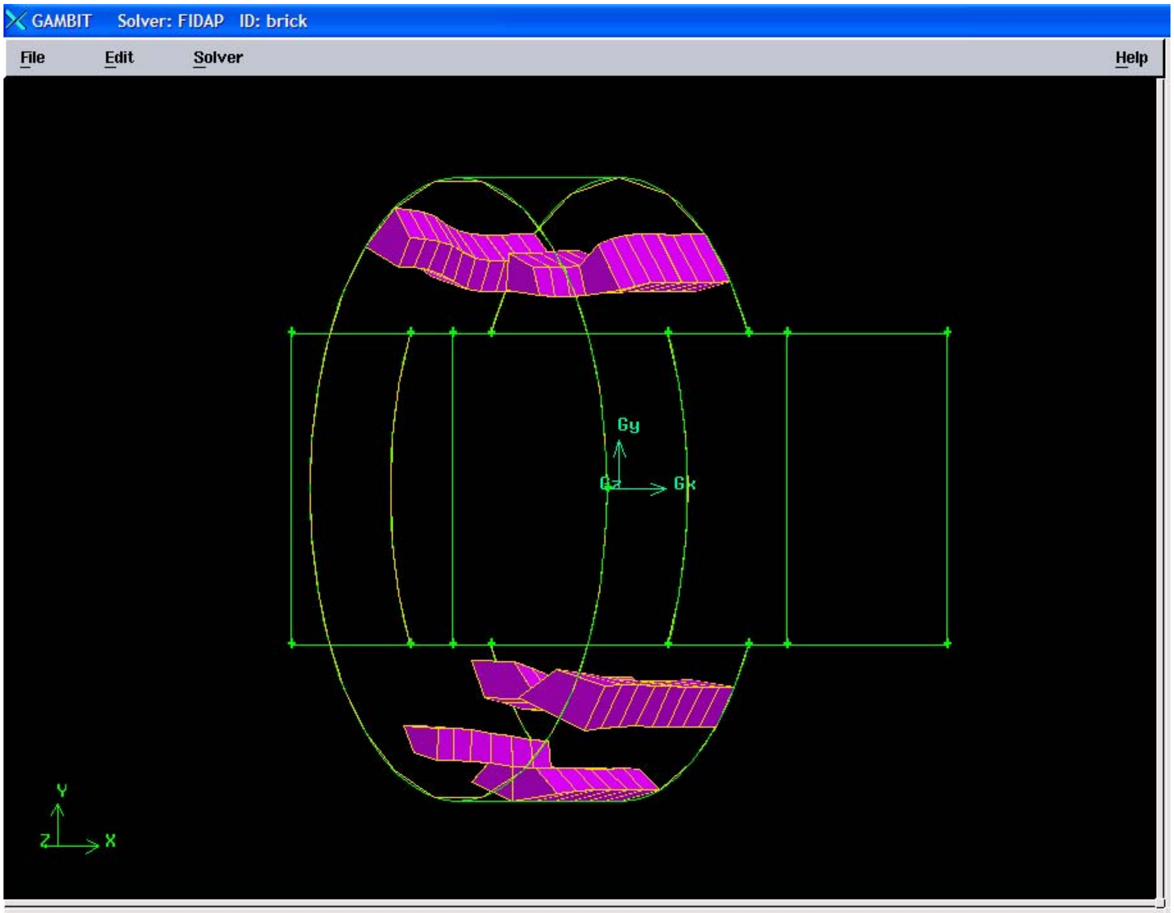
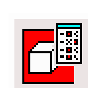
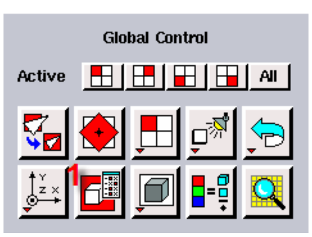


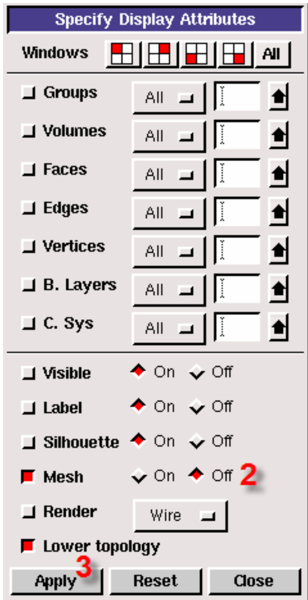
Figure 1.7: Elements of the mesh within a specified quality range

Step 8: Remove the mesh from Display

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.

1) Click the **SPECIFY DISPLAY ATTRIBUTES** command button  at the bottom of the **Global Control** toolpad.
2) Select the Off radio button to the right of **Mesh** near the bottom of the form.
3) Click **Apply** and close the form.





Step 9: Define boundaries and continuum

This is done to define all the boundaries of the problem. For example in our case, we have 14 boundaries. These boundaries are the different faces of the 3D geometry we obtained by uniting the brick and the cylinder. In addition to these, the geometry needs to be defined as a continuum. We name these boundaries and continuum and specify their types depending upon the physics of the problem. For example, in our problem we assume that the geometry is a part of a tissue and so we name it as "TISSUE" and select the type as "SOLID" as shown below.

Define the boundary zones

1) Under the **Operations** panel, click on **Zones**

2) Under the **Zones** panel, click on **Specify Boundary**

3) In the **Specify Boundary Type window**, right click on the button below **Entity**, hold and choose **Faces** from the drop-down list.

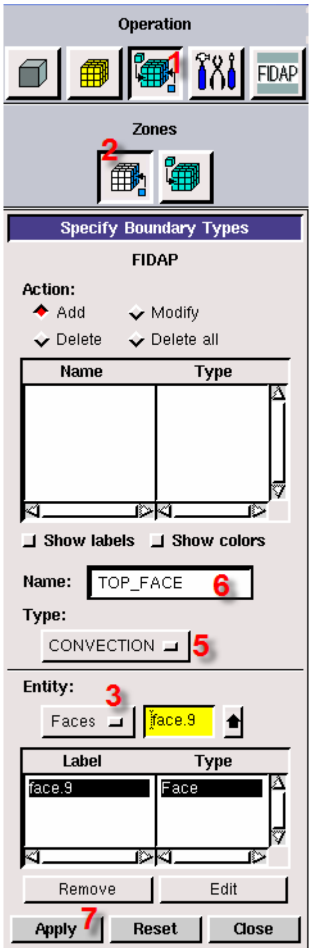
4) To select the front circular face in the **Graphics** window, **Shift left click** on its edge. The color of the face changes to red.

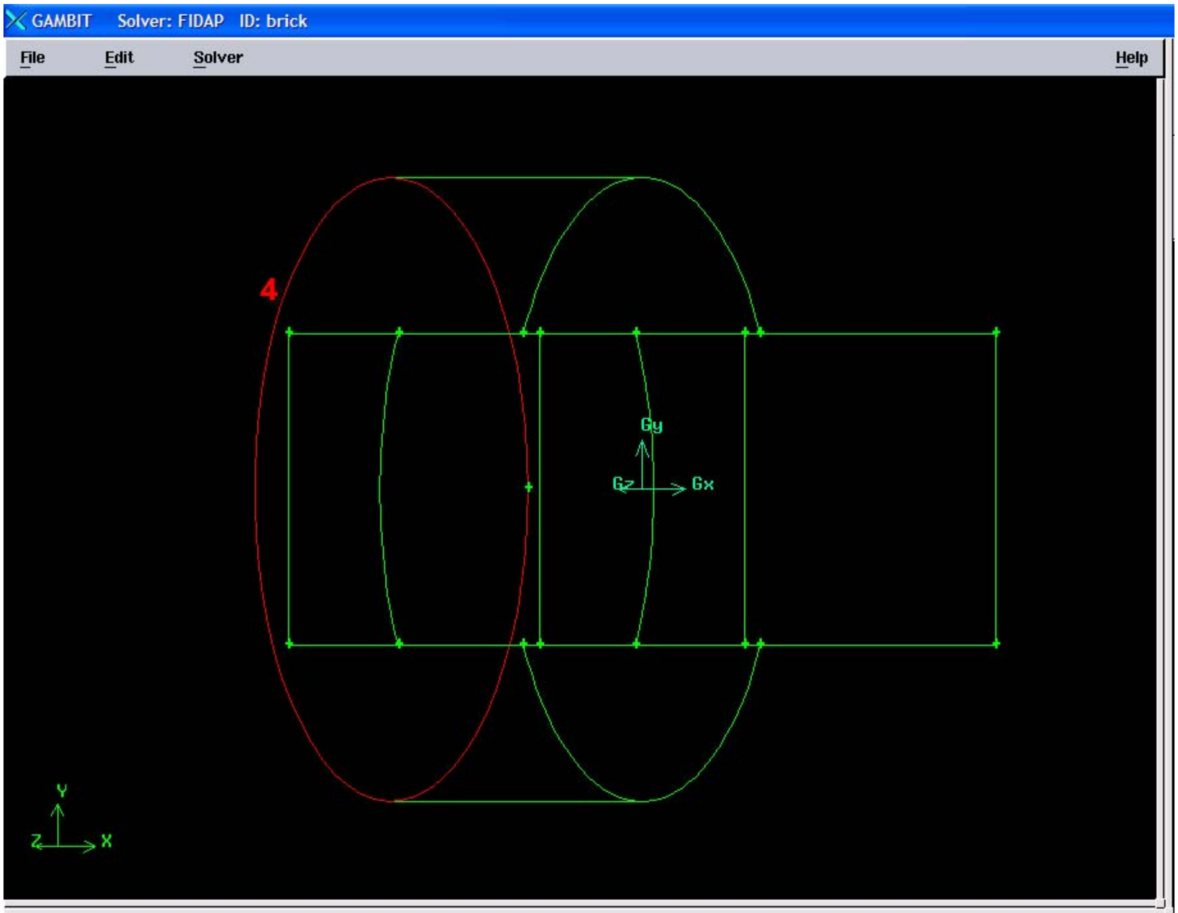
5) Right click on the button below **Type**, hold and choose **CONVECTION** from the drop-down list. We have defined the face as a CONVECTION boundary. The **CONVECTION** keyword specifies that this entity is comprised of boundary convective heat transfer elements. The purpose of a boundary convection entity is to impose convective heat flux boundary conditions. You can define the boundary according to the physics of your problem. There are different options for the boundary type. To know about different boundaries and continuum types click here.

6) Click on the **Name** bar and type TOP_FACE in the text box. *The face has been named as* TOP_FACE.

7) Click on **Apply**

8) Name the other faces similarly.





Specify the continuum

Now, we specify the type of continuum we will use for our analysis. We will specify it as a SOLID. For other options, click here.

1) Under the **Operations** panel, click on **Zones**. You may not need to do this if the **Zones** panel is already open.

2) Under the **Zones** panel, click on the **Specify Continuum Types** Command Button

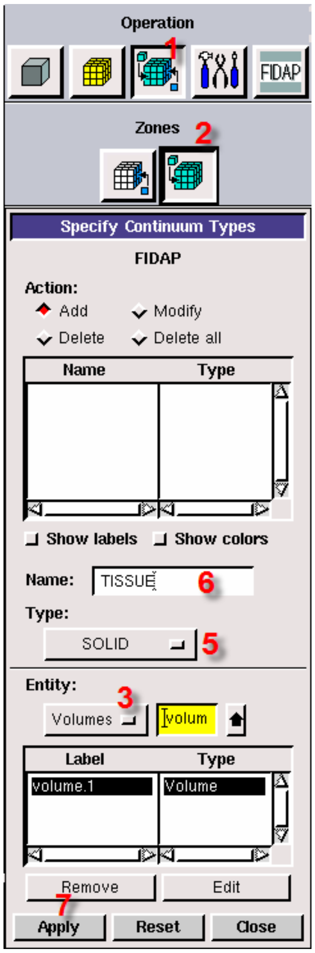
3) In the **Specify Continuum Type** window**,** right click on the button below **Entity**, hold and choose **Volume**

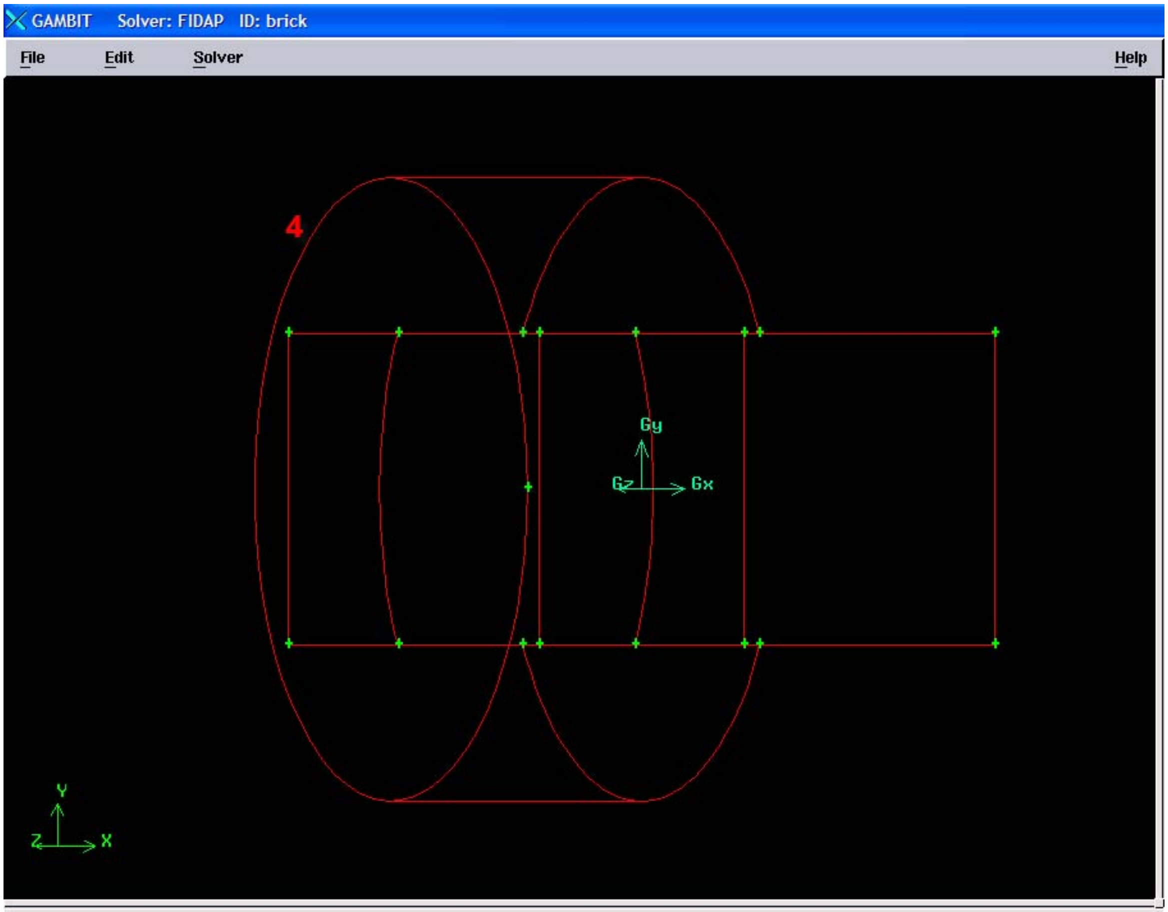
4) To select the volume in the **Graphics** window, **Shift left click** on any one edge of the volume. The selected volume turns red.

5) Right click on the button below **Type**, hold and choose **SOLID** from the drop-down list

6) Click on the **Name** and type TISSUE in the text box. *The volume has been named as TISSUE.*

7) Click on **Apply**


Operation
1
FIDAP
Zones
2
Specify Continuum Types
FIDAP
Action:
Add
Modify
Delete
Delete all
Name
Type
Show labels
Show colors
Name:
TISSUE
6
Type:
SOLID
5
Entity:
3
Volumes
volum
Label
Type
volume.1
Volume
Remove
Edit
7
Apply
Reset
Close



Step 10: Save

To avoid data loss due to unexpected termination of the program save your files time to time using the following procedure.

1) Click on the **File** menu

2) Click on **Save**


GAMBIT Solver: FIDAP ID: ex1
File 1
Edit
Solver
New ...
Open ...
Save 2
Save As ...
Print Graphics ...
Run Journal ...
Clean Journal ...
View File ...
Import
Export
Exit