

## 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. The edges of the combustion chamber are blended on the wall next to the nozzle.

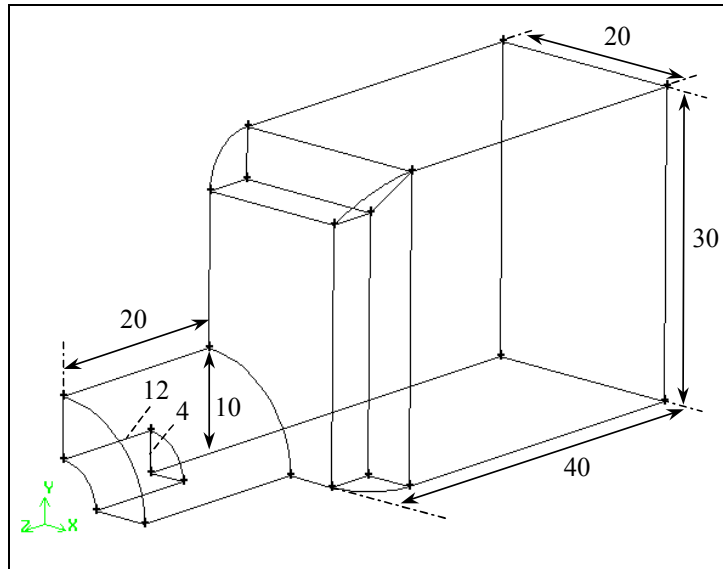


Figure 4-1: Problem specification

### 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.

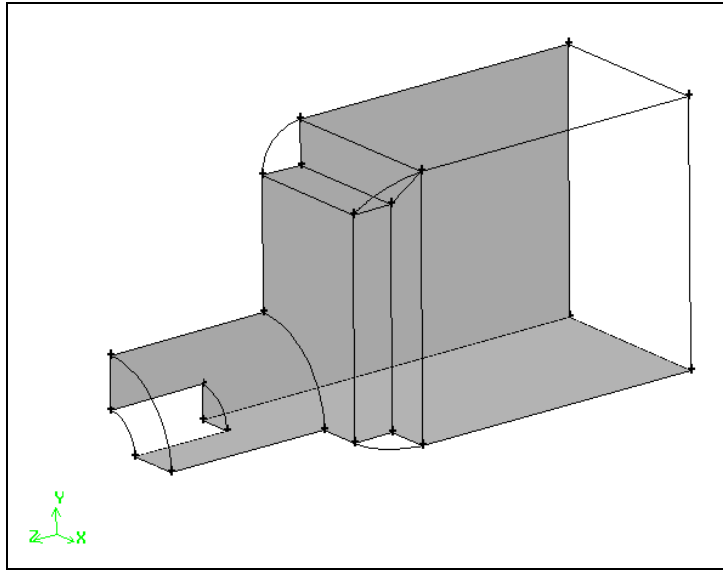


Figure 4-2: The two symmetry planes in the combustion chamber geometry

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.

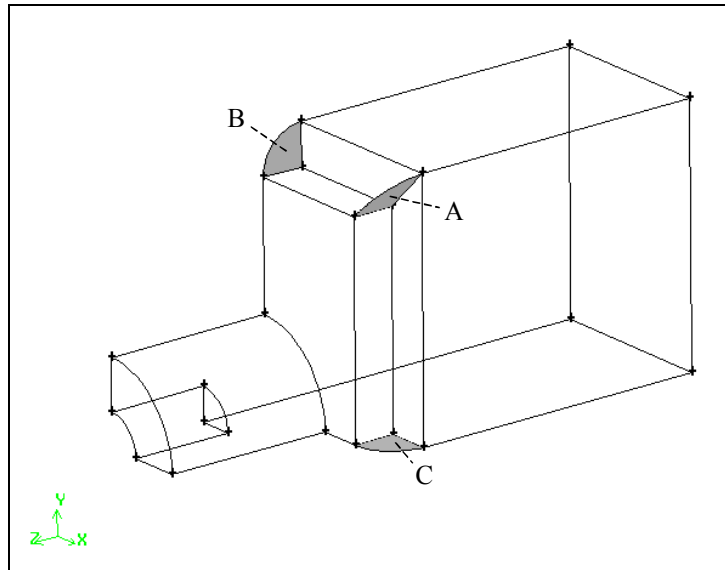


Figure 4-3: Faces created at the blended edges and on the symmetry planes

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:

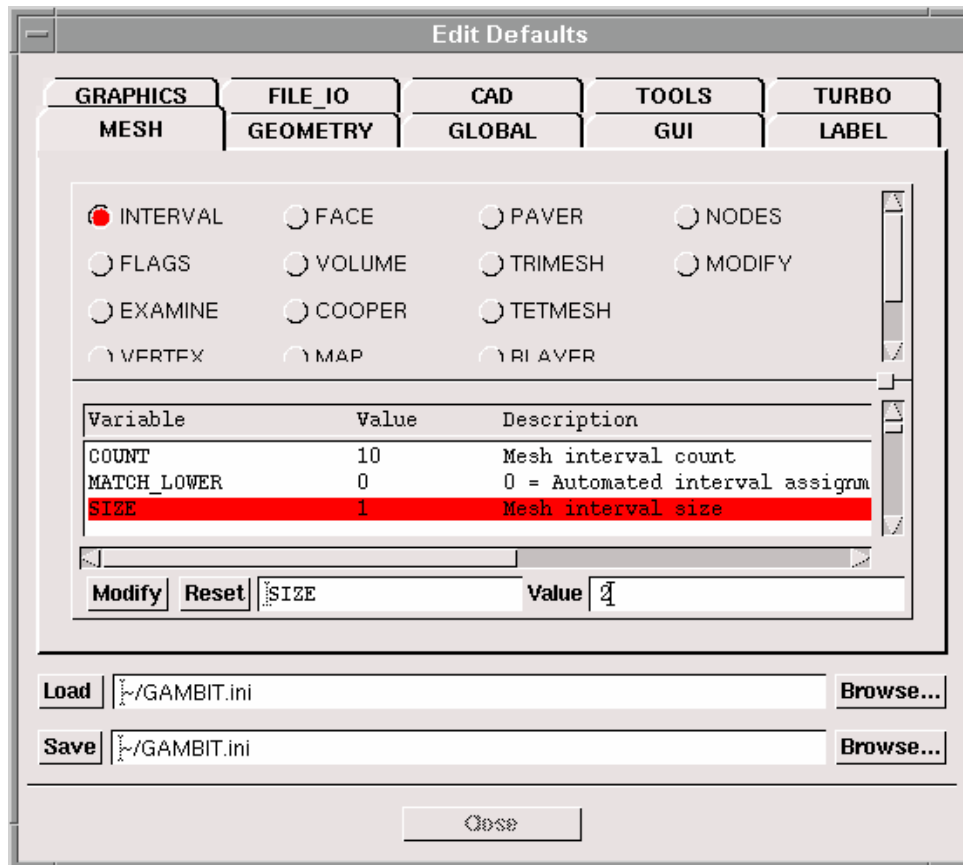
**Solver → FLUENT 5/6**

*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.*

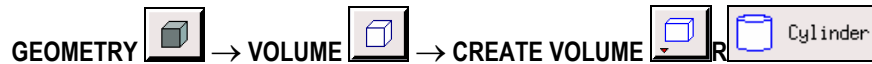
4. Enter a value of 2 in the **Value** text entry box.
5. Click the **Modify** button to the left of **SIZE**.

*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.




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

The 'Create Real Cylinder' dialog box contains the following fields and options:

- Height:** 12
- Radius 1:** 4
- Radius 2:** (blank)
- Coordinate Sys.:** c\_sys.1
- Axis Location:** Positive Z
- Label:** (blank)
- Buttons: Apply, Reset, Close

A diagram of a cylinder is shown with height H, radii R1 and R2, and a coordinate system (X, Y, Z).

- 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.*



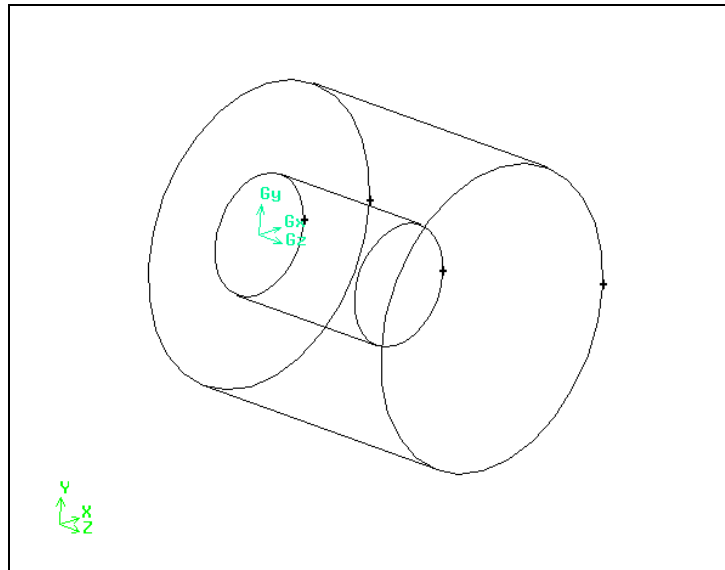





Figure 4-4: Two cylinders

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.*

**Move / Copy Volumes**

Volumes:

☒ Move ☐ Copy

**Operation:**

☒ Translate ☐ Rotate

☐ Reflect ☐ Scale

Coordinate Sys.:

Type:

Global		Local	
x:	<input type="text" value="0"/>	x:	<input type="text" value="0"/>
y:	<input type="text" value="0"/>	y:	<input type="text" value="0"/>
z:	<input type="text" value="8"/>	z:	<input type="text" value="8"/>

☐ Connected geometry

- 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.*

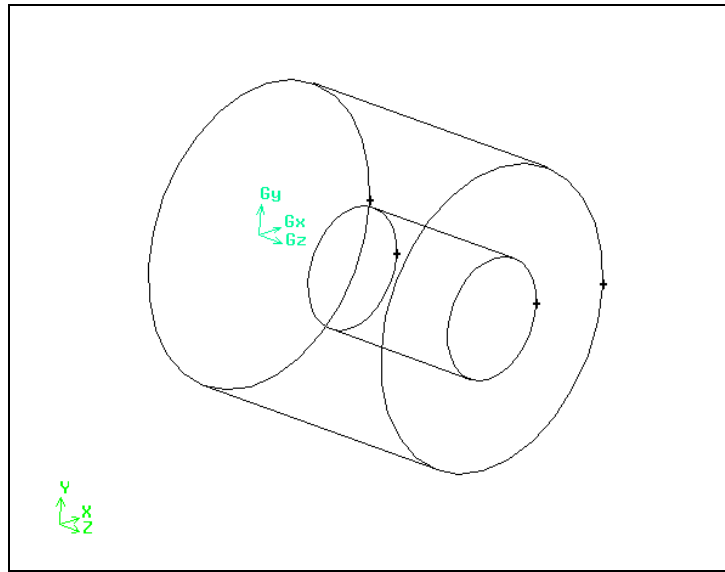


Figure 4-5: Two cylinders after moving the small cylinder

## 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.*

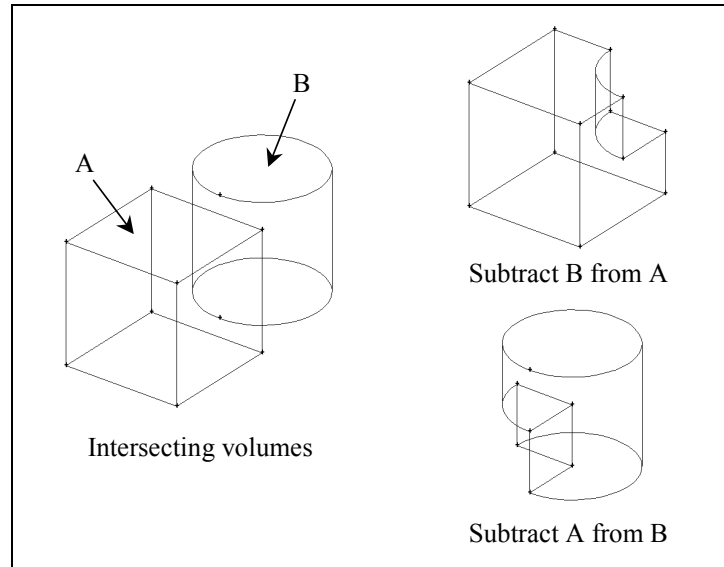







Figure 4-6: Subtracting volumes

GEOMETRY  → VOLUME  → BOOLEAN OPERATIONS  →  Subtract

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

Subtract Real Volumes	
Volume	volume.2 
	<input type="checkbox"/> Retain
Subtract	
Volumes	volume.1 
	<input type="checkbox"/> Retain
<div>Apply    Reset    Close</div>	


- 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).

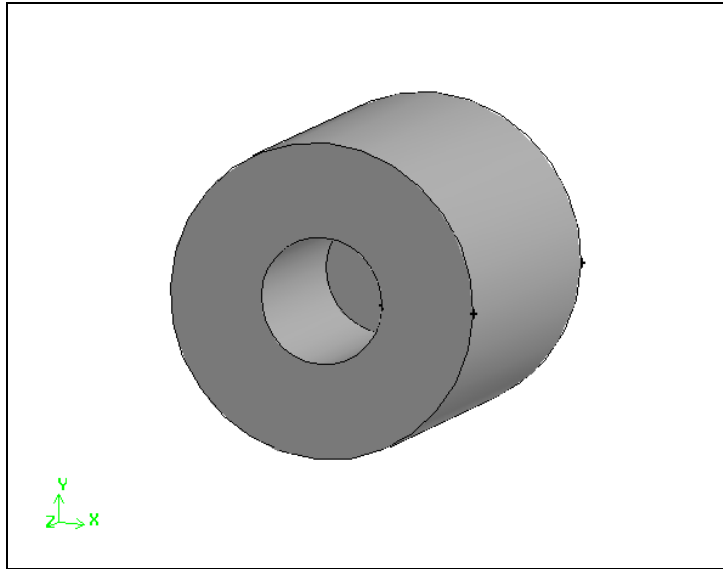




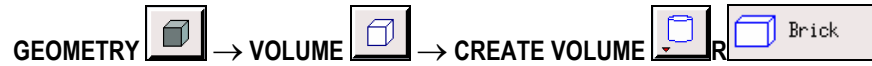
Figure 4-7: Shaded geometry showing hole in large cylinder

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



*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.*

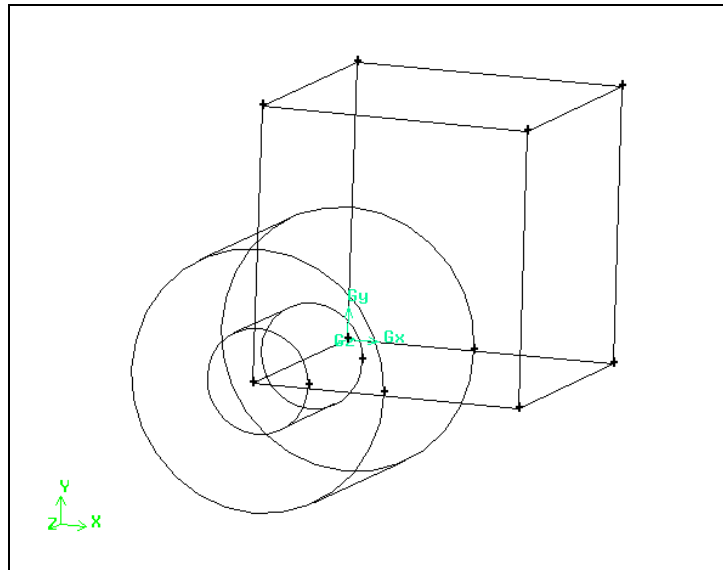
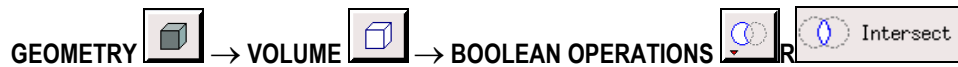
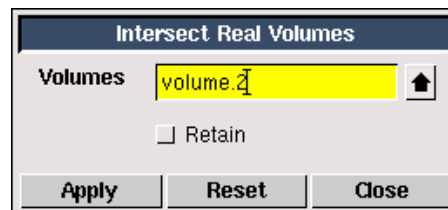


Figure 4-8: Cylindrical volume and brick

2. Intersect the brick and the cylindrical volume.



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



- Shift-left-click the brick in the graphics window.
- Select the cylindrical volume in the graphics window.
- 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.*

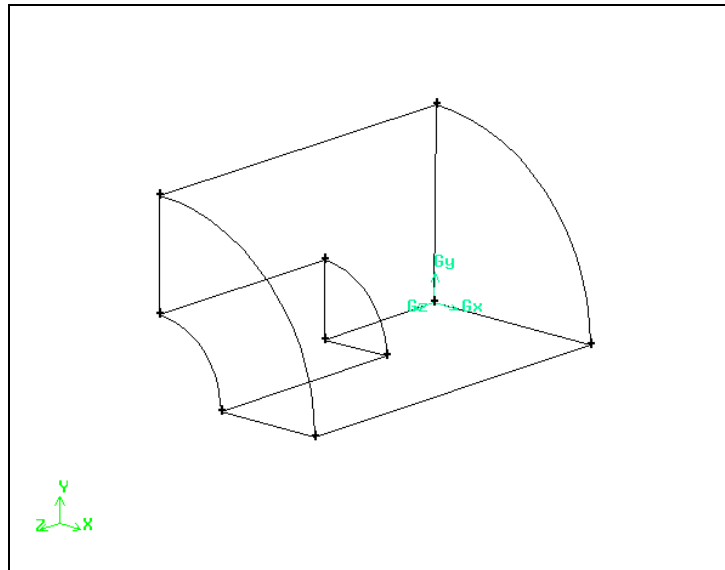


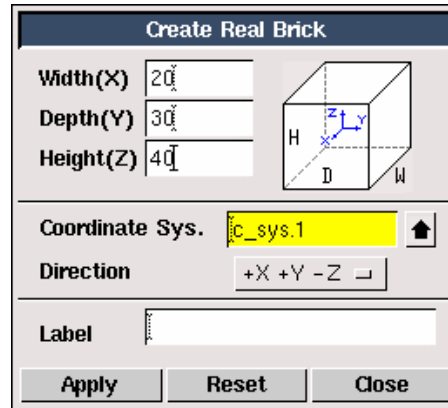
Figure 4-9: One quarter of the cylindrical volume remains

## 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.*



The **Create Real Brick** dialog box contains the following fields and controls:

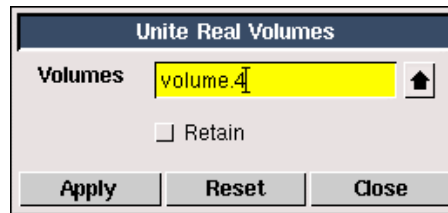
- Width(X)**: 20
- Depth(Y)**: 30
- Height(Z)**: 40
- Coordinate Sys.**: c\_sys.1
- Direction**: +X +Y -Z
- Label**: (empty text field)
- Buttons**: Apply, Reset, Close

A 3D diagram of a brick is shown with dimensions W (width), D (depth), and H (height) labeled.

- 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  →  Unite

*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.*

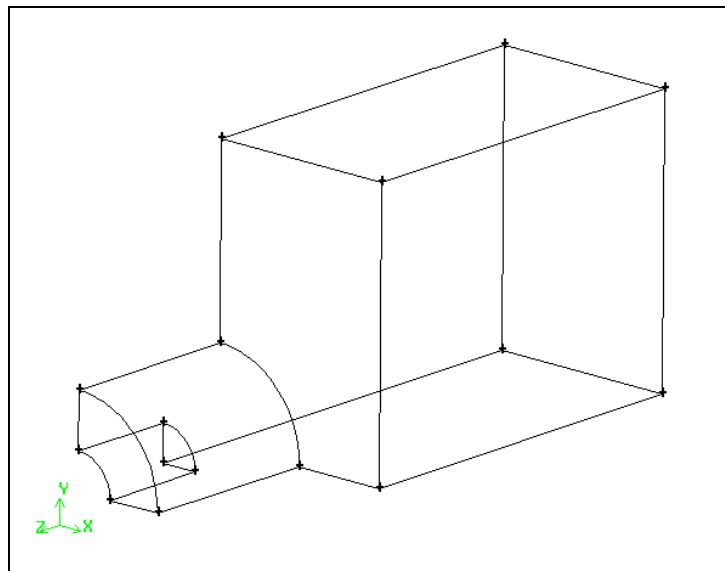


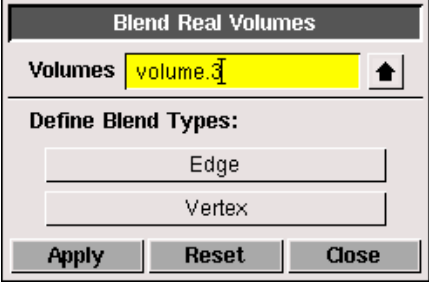
Figure 4-10: Brick and cylindrical volume are united

## 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.



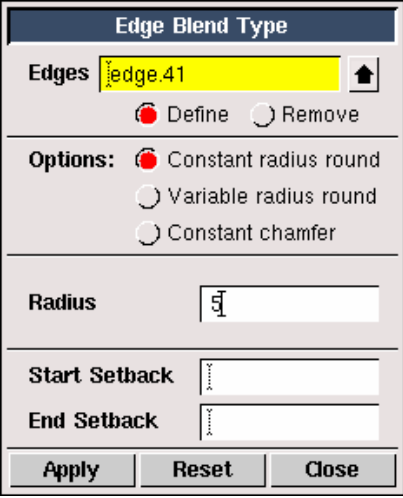
*This command sequence opens the **Blend Real Volumes** form.*



The **Blend Real Volumes** dialog box has a title bar with the same name. It contains a **Volumes** field with the text "volume.3" and a selection icon. Below this is a section titled **Define Blend Types:** with two buttons, **Edge** and **Vertex**. At the bottom are three buttons: **Apply**, **Reset**, and **Close**.

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

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



The **Edge Blend Type** dialog box has a title bar with the same name. It contains an **Edges** field with the text "edge.41" and a selection icon. Below this are two radio buttons: **Define** (selected) and **Remove**. The **Options:** section has three radio buttons: **Constant radius round** (selected), **Variable radius round**, and **Constant chamfer**. Below the options is a **Radius** field with a value of "5". At the bottom are three buttons: **Apply**, **Reset**, and **Close**.

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

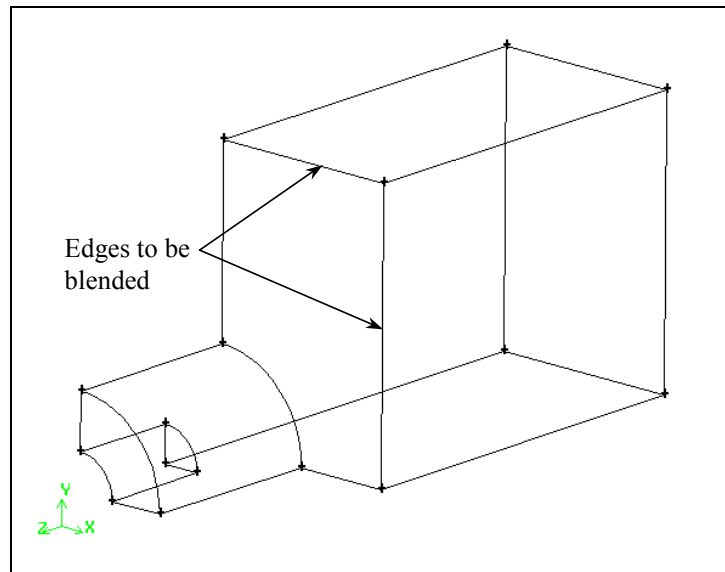


Figure 4-11: Edges to be blended

- 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 Real Volumes** form.

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

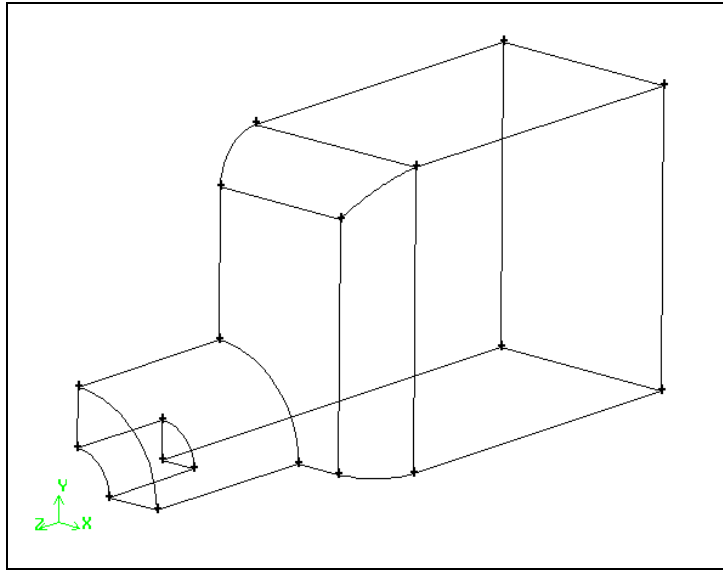
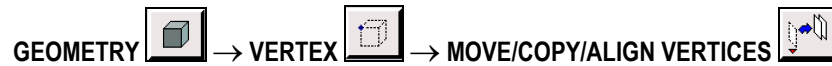


Figure 4-12: Burner with blended edges

## 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.



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

**Move / Copy Vertices**

Vertices  vertex.36

☐ Move ☒ Copy 1

**Operation:**

☒ Translate ☐ Rotate

☐ Reflect ☐ Scale

Coordinate Sys. c\_sys.1

Type Cartesian

Global	Local
x: 0	x: 0
y: 0	y: 0
z: -5	z: -5

- 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.*



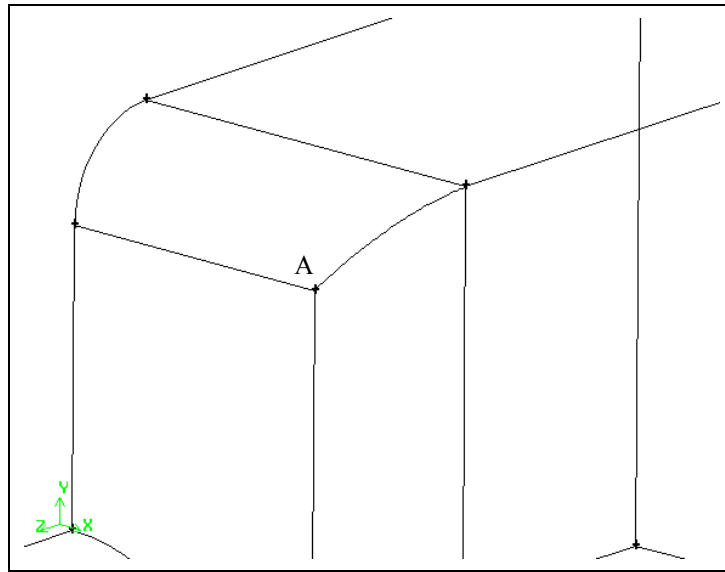


Figure 4-13: Vertex to copy

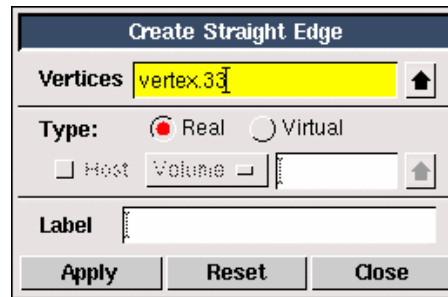
- b) Select **Copy** under **Vertices** in the **Move / Copy Vertices** form.
- c) Select **Translate** (the default) under **Operation**.
- d) Enter the translation vector (0, 0, -5) under **Global**.
- e) Click **Apply**.

*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.

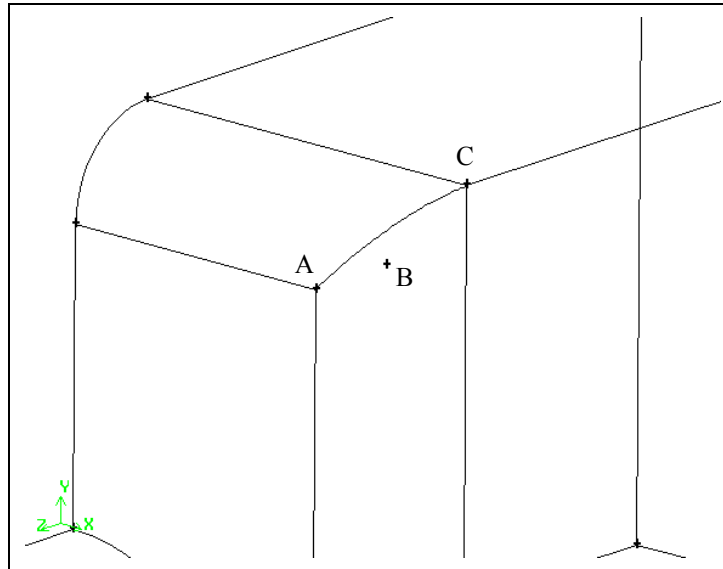


Figure 4-14: Vertices to be selected to create edges

- 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.*

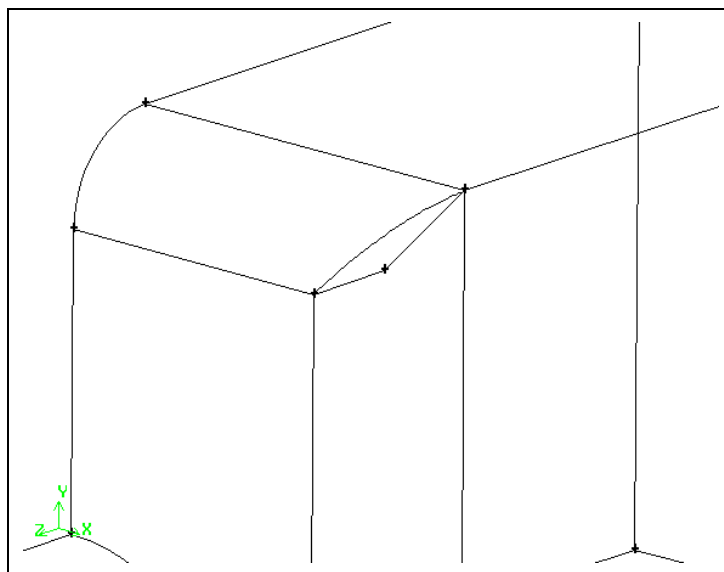


Figure 4-15: Two new straight edges

3. Create a face using the two new edges.

GEOMETRY  → FACE  → FORM FACE 

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

**Create Face from Wireframe**

Edges

Type: ☒ Real ☐ Virtual

☐ Initial Face

☐ Guide Edges

☐ Guide Vertices

Tolerance

Label

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

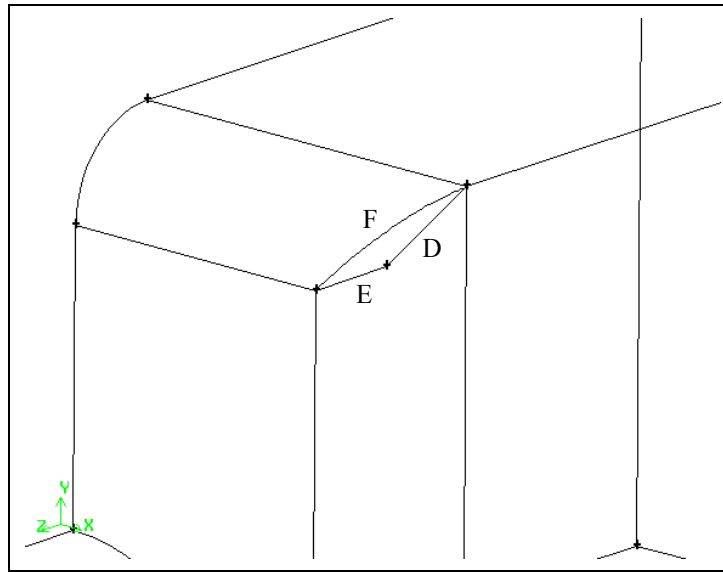
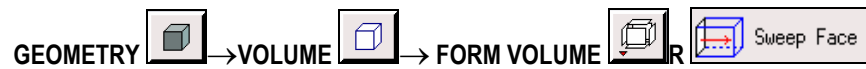


Figure 4-16: Edges used to create the face

- 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.



*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.*

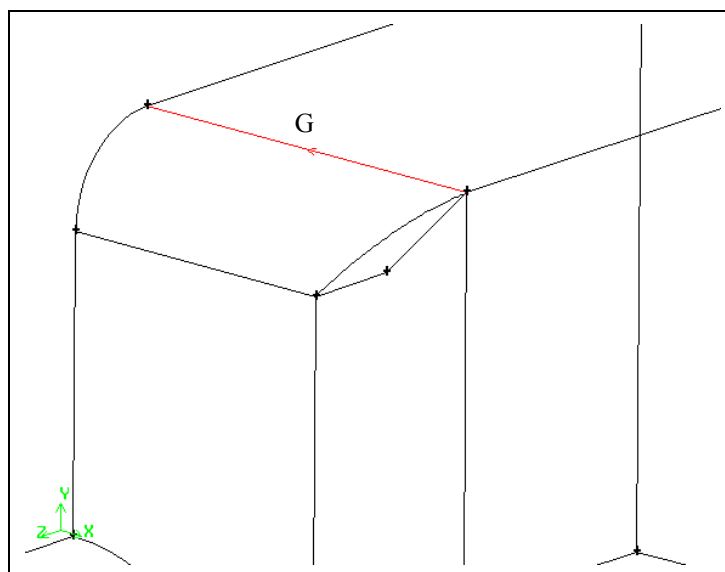


Figure 4-17: Edge to be used for sweep face

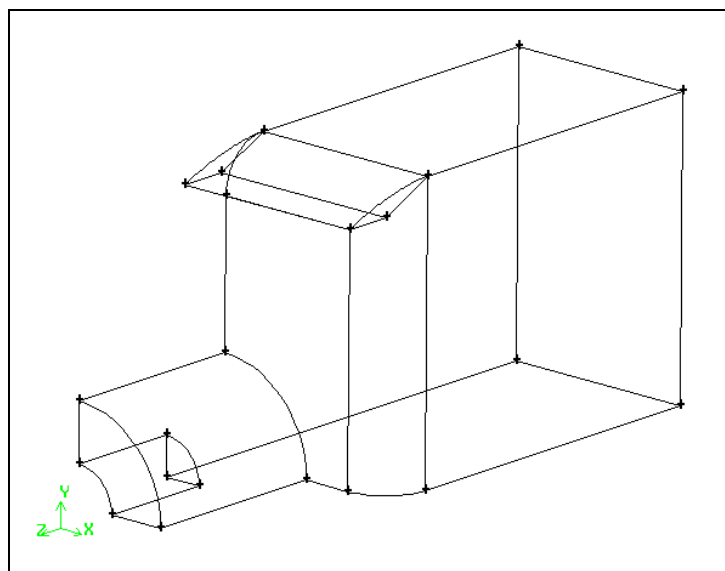


Figure 4-18: Face swept parallel to an edge to form a volume

*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.

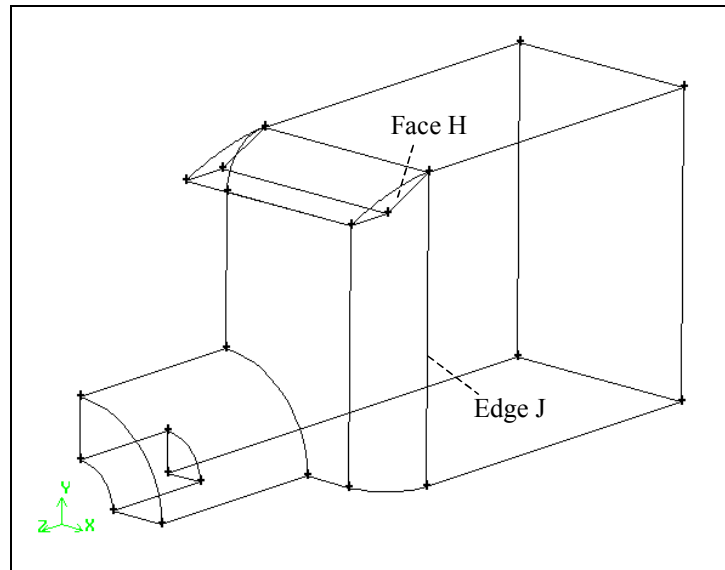


Figure 4-19: Face and edge to be used for sweep face

- 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.*

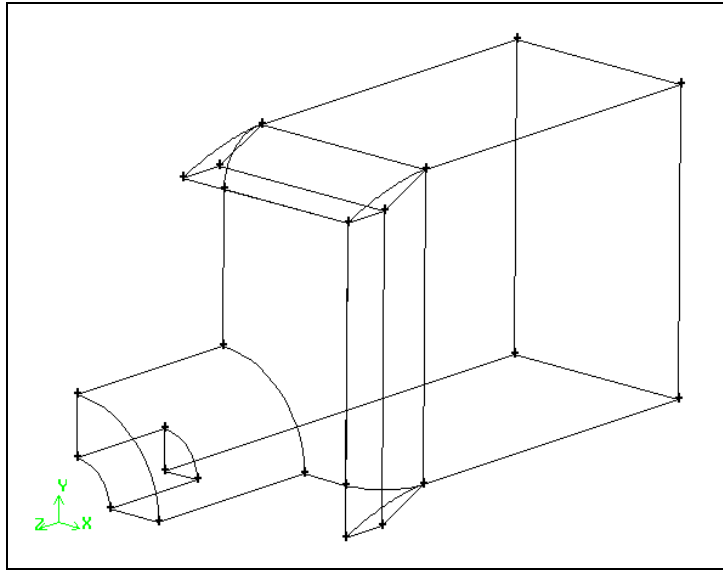


Figure 4-20: Face swept parallel to an edge to form a second volume

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.*



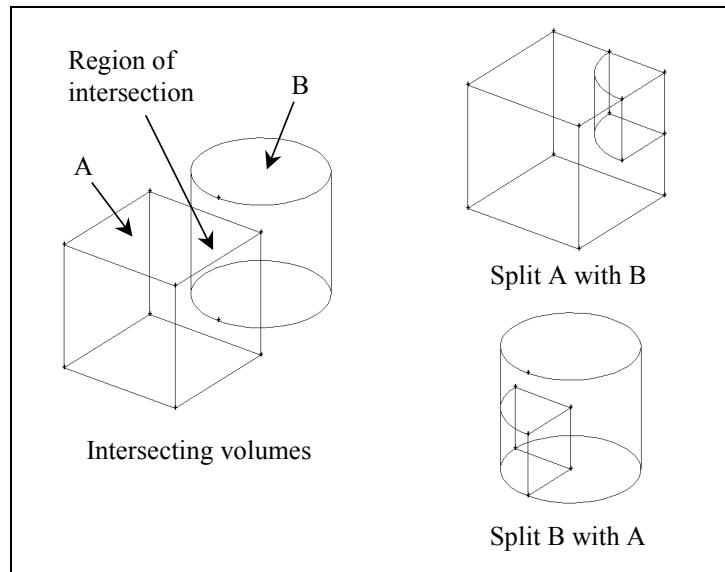



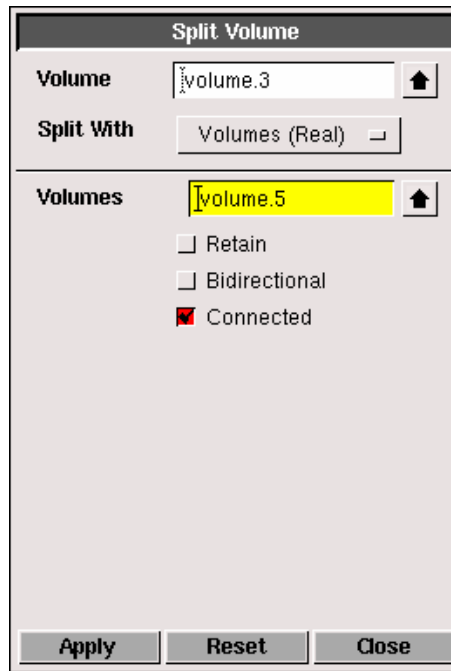


Figure 4-21: Splitting volumes

GEOMETRY  → VOLUME  → SPLIT/MERGE VOLUMES 

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



- a) Select the large burner geometry in the graphics window.
- b) Select Volumes (Real) as the **Split With** option.
- c) Left-click in the **Volumes** list box located *below* the **Split With** section to make it active.
- d) Select the two small volumes created using the sweep face method.
- e) Unselect the Bidirectional option.
- f) Click **Apply**.

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

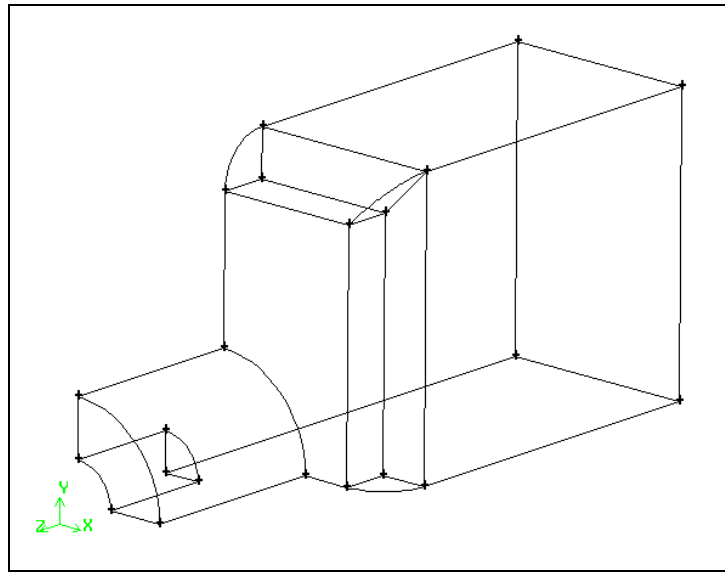


Figure 4-22: Decomposed burner geometry

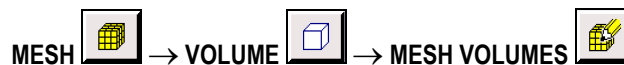
## Step 10: Generate an Unstructured Hexahedral Mesh

*In the meshing section of this tutorial you will use three GAMBIT features: the Cooper tool, face meshing schemes, and 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.



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

**Mesh Volumes**

Volumes: volume.4

Scheme: ☒ Apply Default

Elements: Hex

Type: Cooper

Sources: face.28

Spacing: ☒ Apply Default  
2 Interval size

Options: ☒ Mesh  
☐ Remove old mesh  
☐ Remove lower mesh  
☐ Ignore size functions

Apply Reset Close

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

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

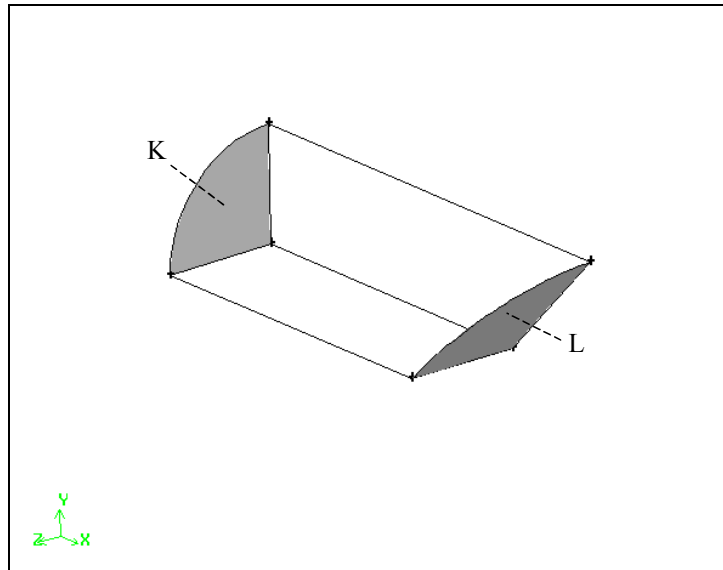


Figure 4-23: Faces to select for first Cooper meshing operation

*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  $90^\circ$  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.*

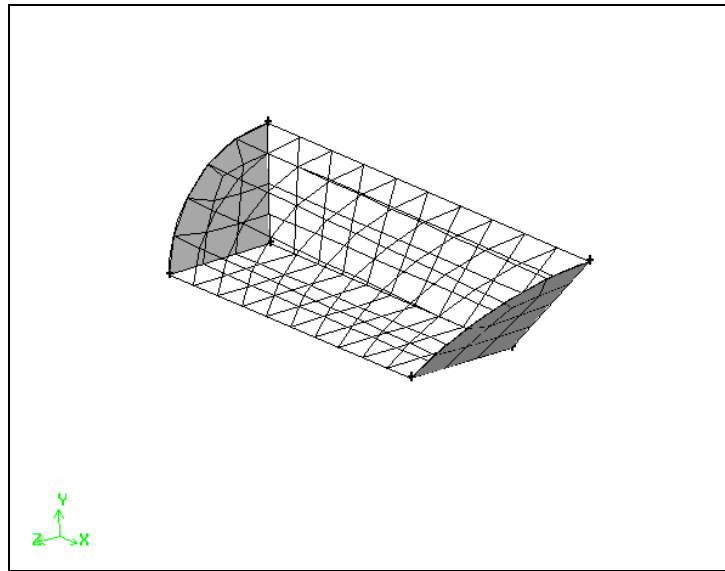


Figure 4-24: Mesh generated for the first small volume in the burner geometry

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.*

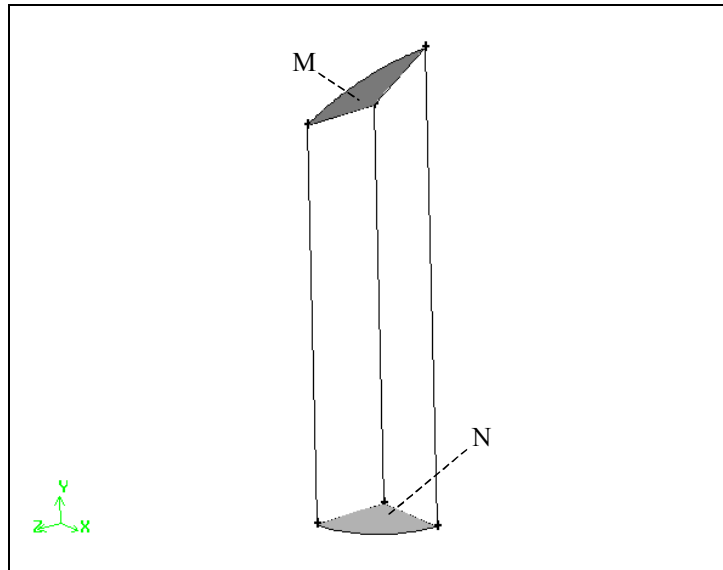


Figure 4-25: Faces to select for cooper meshing on the volume at the side of the burner geometry

- 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.*



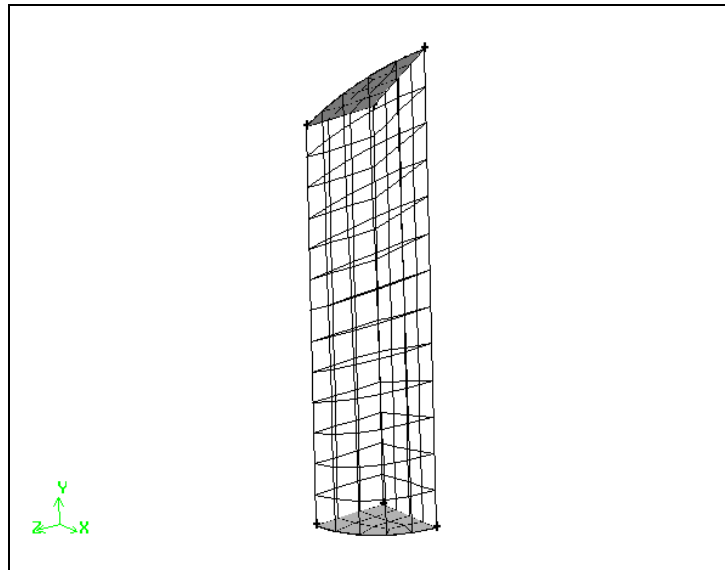


Figure 4-26: Mesh generated for the second small volume in the burner geometry

*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.*

**Mesh Faces**

Faces: face.14

Scheme: ☒ Apply Default

Elements: Quad

Type: Tri Primitive

Spacing: ☒ Apply Default

0.5 Interval size

Options: ☒ Mesh  
☐ Remove old mesh  
☐ Remove lower mesh  
☐ Ignore size functions

Apply Reset Close

- 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.*

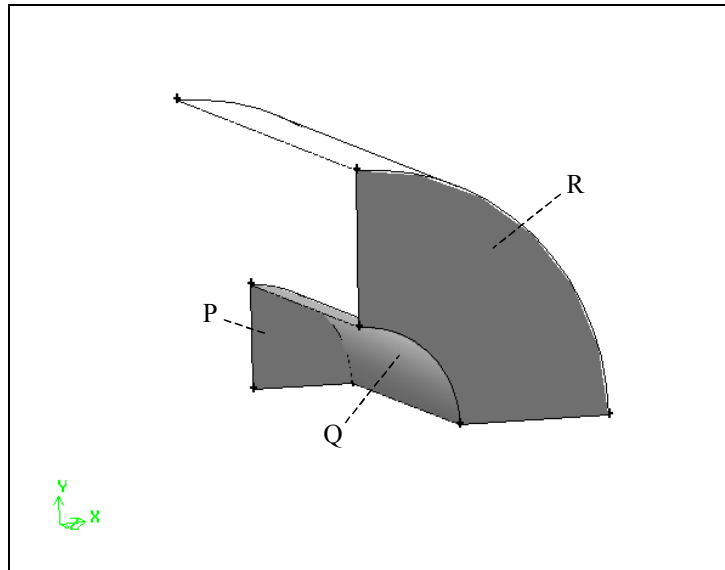


Figure 4-27: Faces to be meshed in the burner geometry

- 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.*

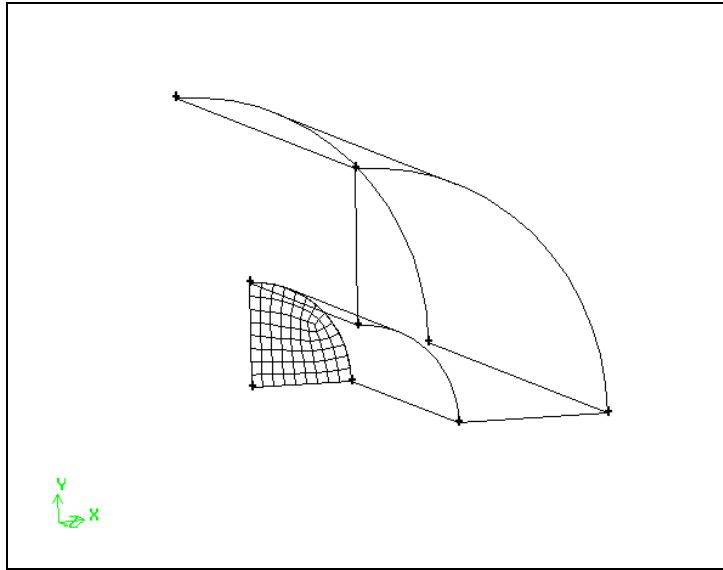


Figure 4-28: Mesh on small face

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.*

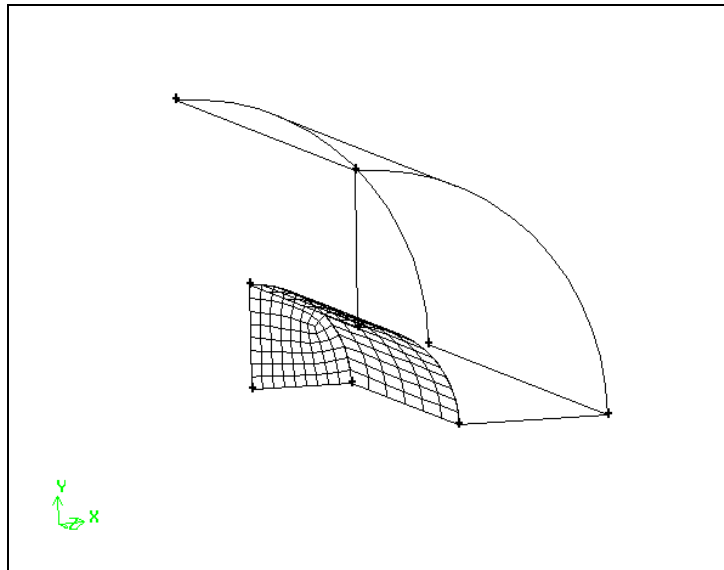


Figure 4-29: Mesh on curved face along the entrance to the burner

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.*

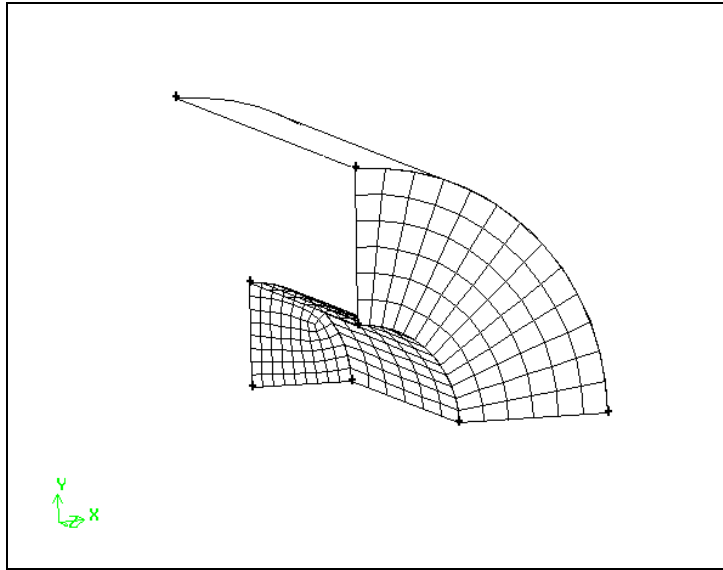


Figure 4-30: Mesh on face at entrance of burner

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.

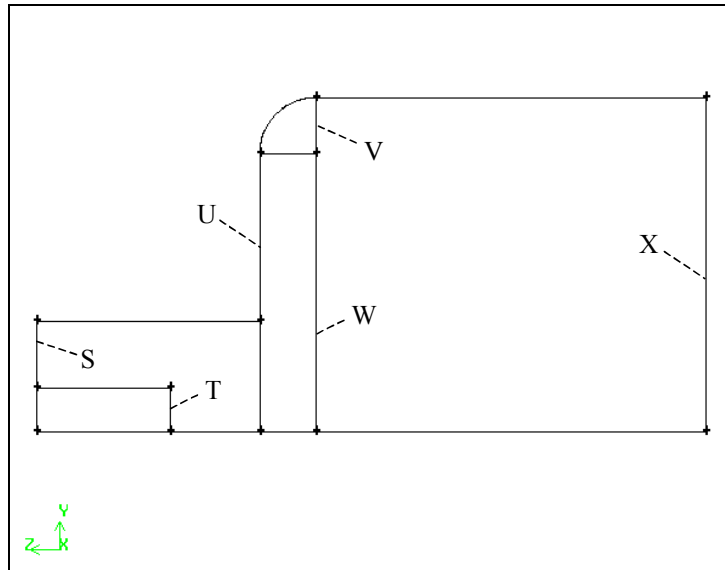





Figure 4-31: Source faces to be used for cooper mesh

- 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  Hidden from the resulting list. To view the mesh without hidden line removal, reselect the  Wireframe option.*

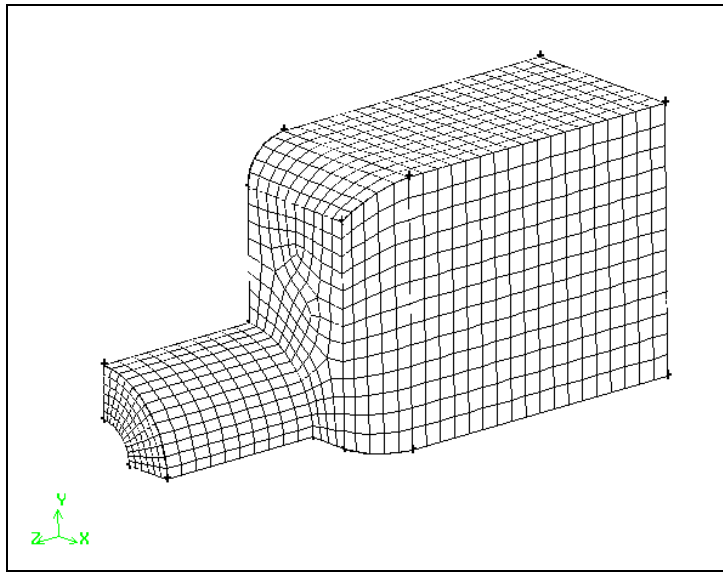







Figure 4-32: Volume mesh for the burner geometry

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

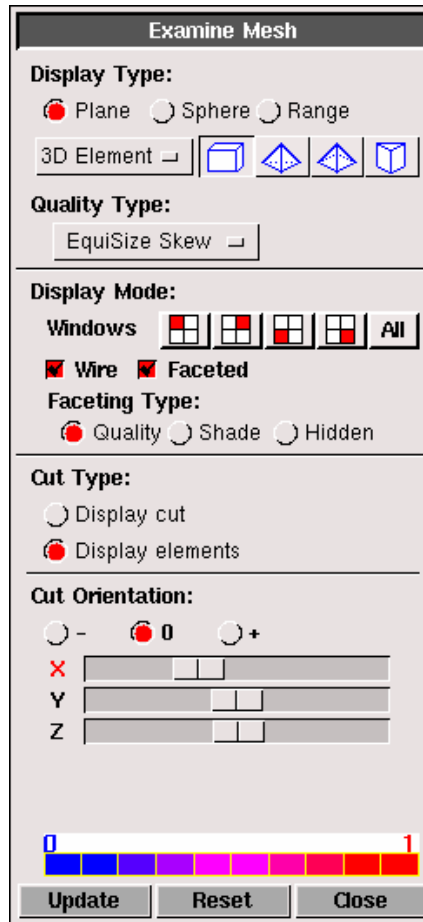
- a) Hold down the right mouse button on the **RENDER MODEL** command button  and select  Shaded from the resulting list.
- b) Rotate and translate the volume to view the mesh.
- c) When you are finished, return to the wireframe view of the model, by selecting the following command buttons in the **Global Control** toolpad:  **R**  Wireframe.



## 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 **Examine Mesh** dialog box contains the following sections and controls:

- Display Type:**
  - Radio buttons: ☒ Plane, ☐ Sphere, ☐ Range
  - 3D Element dropdown menu (set to "3D Element") and four 3D element icons: a brick (selected), a tetrahedron, a hexahedron, and a pyramid.
- Quality Type:**
  - EquiSize Skew dropdown menu.
- Display Mode:**
  - Windows: Four window icons and an "All" button.
  - Checkboxes: ☒ Wire, ☒ Faceted
- Faceting Type:**
  - Radio buttons: ☒ Quality, ☐ Shade, ☐ Hidden
- Cut Type:**
  - Radio buttons: ☐ Display cut, ☒ Display elements
- Cut Orientation:**
  - Radio buttons: ☐ -, ☒ 0, ☐ +
  - Input fields for X, Y, and Z axes, each with a red 'X' icon to the left.
- Color Scale:** A horizontal bar with a gradient from blue (0) to red (1).
- Buttons:** Update, Reset, and Close.

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

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

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

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

- b) Select the Plane option under **Display Type**.
- c) Select or retain EquiSize Skew from the **Quality Type** option menu.
- d) 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 EquiSize 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.

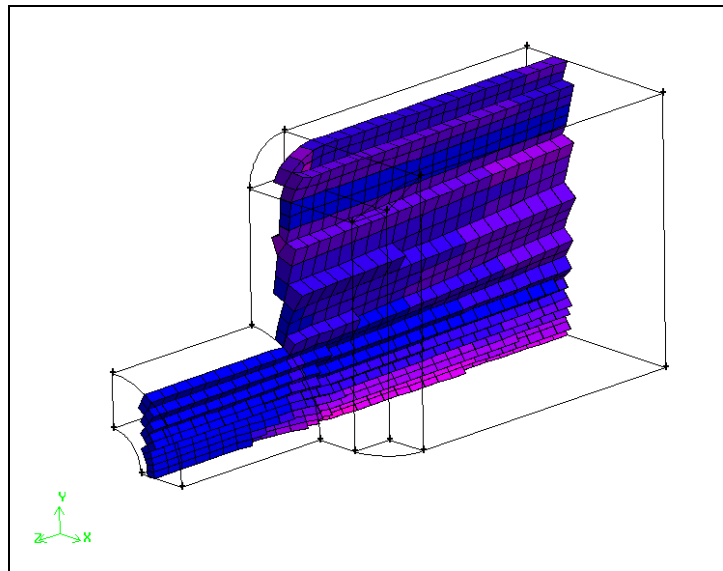


Figure 4-33: Slice of the mesh in the  $x$  direction

- e) Use the **Y** and **Z** sliders to view slices in the  $y$  and  $z$  directions.
- f) 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.*

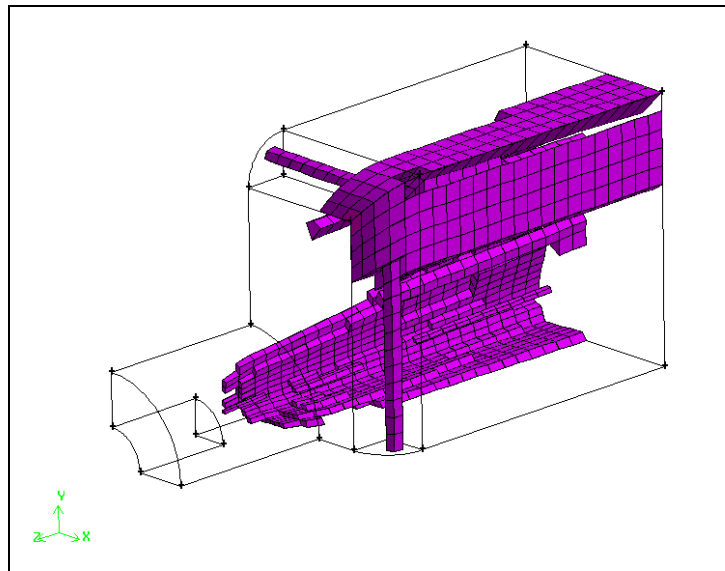



Figure 4-34: Elements within a specified quality range

*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.*


- g) Select the Show worst element option.

- h) 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.
- i) 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.*

**Specify Boundary Types**

**FLUENT 5/6**

**Action:**

☒ Add
 ☐ Modify
 ☐ Delete
 ☐ Delete all

Name	Type
inlet1	VELOCITY_INLE
inlet2	VELOCITY_INLE
outlet	PRESSURE_OUT
symmetryx	SYMMETRY

☐ Show labels
 ☐ Show colors

**Name:**

**Type:**

**Entity:**

Label	Type
face.8	Face
face.40	Face

- 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.

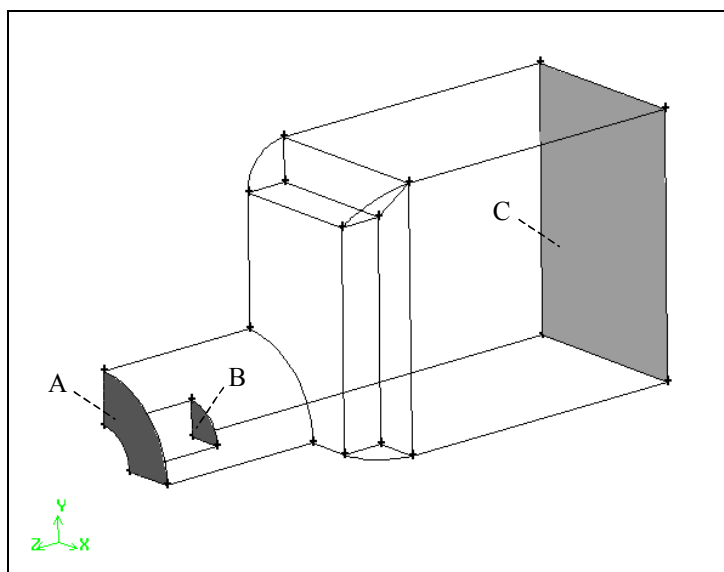


Figure 4-35: Boundary-type faces for the burner—inlet and outlet

*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.

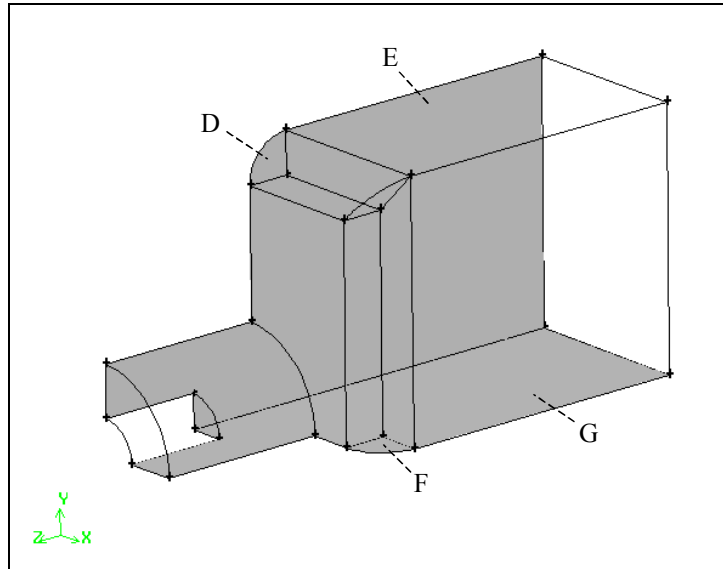


Figure 4-36: Boundary-type faces for the burner—symmetry

- d) Define symmetry boundary types for the two faces normal to the y axis.
  - i. Enter the name `symmetry` 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.)*



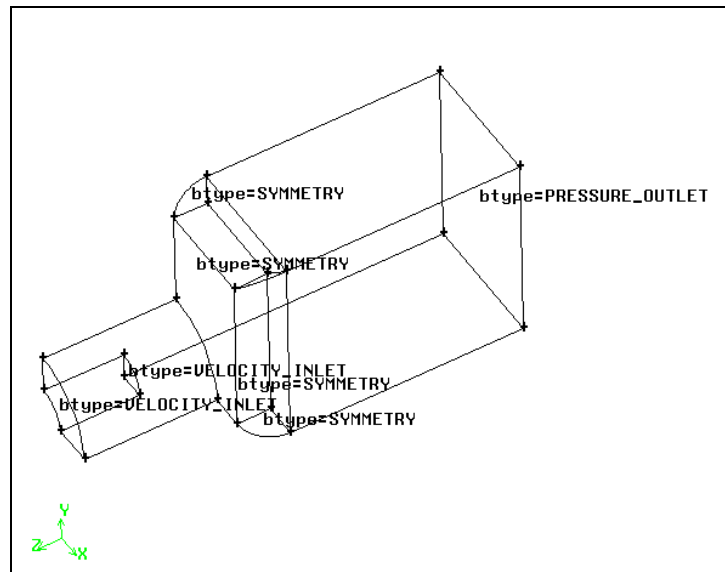


Figure 4-37: Boundary types for the combustion chamber

*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.*

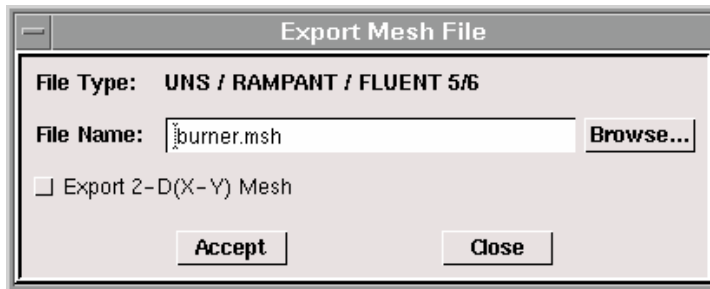
*In addition, when GAMBIT writes a mesh, any volumes (in 3-D) for which you have not specified a continuum type will be written as FLUID by default. This means that you do not need to specify a continuum type in the **Specify Continuum Types** form for this tutorial.*

## 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**.*



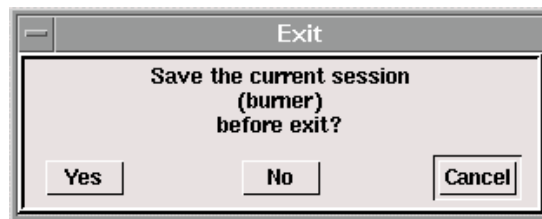
- a) Enter the **File Name** for the file to be exported (burner.msh).
- b) Click **Accept**.

*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.