

Chapter 24

Creating Solid Models

Learning Objectives

After completing this chapter, you will be able to:

- Understand solid modeling.
- Create standard solid primitives.
- Understand regions and create them.
- Understand the use of Boolean operations.
- Use the **EXTRUDE** command to create complex solid models.
- Use the **REVOLVE** command to create complex solid models.
- Use the **FILLET** and the **CHAMFER** commands for the solid models.
- Rotate and mirror the solid models in the 3D space.
- Create an array in 3D space.
- Align the solid models using the **ALIGN** command.
- Slice the solid models and create the cross-sections.

WHAT IS SOLID MODELING?

Solid modeling is the process of building objects that have all the attributes of an actual solid object. For example, if you draw a wireframe or a surface model of a bushing, it is sufficient to define the shape and size of the object. However, in engineering, the shape and size alone are not enough to describe an object. For engineering analysis we need more information, such as volume, mass, moment of inertia, and material properties (density, Young's modulus, Poissons's ratio, thermal conductivity, and so on). When you know these physical attributes of an object, it can be subjected to various tests to make sure that it performs as required by the product specifications. It eliminates the need for building expensive prototypes and makes the product development cycle shorter. Solid models also make it easy to visualize the objects because we always think of and see the objects as solids. With computers getting faster and software getting more sophisticated and affordable, solid modeling will become the core of the manufacturing

process. AutoCAD solid modeling is based on the ACIS solid modeler, which is a part of the core technology.

PREDEFINED SOLID PRIMITIVES

The solid primitives form the basic building blocks for a complex solid. ACIS has six predefined solid primitives that can be used to construct a solid model (box, wedge, cone, cylinder, sphere, and torus). The number of lines in a solid model representation is controlled by the value assigned to the **ISOLINES** variable. These lines are called tessellation lines. The number of lines determines the number of computations needed to generate a solid. If the value is high, it will take significantly more time to generate a solid on the screen. Therefore, the value you assign to the **ISOLINES** variable should be realistic. When you enter commands for creating solid primitives, AutoCAD Solids will prompt you to enter information about the part geometry. The height of the primitive is always along the positive Z axis, perpendicular to the construction plane. Just like surface meshes, solids are also displayed as wireframe models unless you hide, render, or shade them. The **FACETRES** system variable controls the smoothness in the shaded and rendered objects.

Creating a Solid Box (BOX Command)

Toolbar:	Solids > Box
Menu:	Draw > Solids > Box
Command:	BOX



You can use the **BOX** command to create a solid rectangular box or a cube. This command provides a number of options for creating the box. These options are discussed next.

Two Corner Option

This is the default option and using this option you can create a solid box by defining the first corner of the box and the other corner of the box. Note that the length of the box will always be taken along the X axis, the width along the Y axis and the height along the Z axis. Therefore, in this case when you specify the other corner, the value along the X axis will be taken as the length of the box, the value along the Y axis will be taken as the width of the box, and then you will be prompted to specify the height of the box. When you choose this button, the following prompt sequence will be issued:

```
Specify corner of box or [CEnter] <0,0,0>: 2,2,0
Specify corner or [Cube/Length]: @5,4,0    (Length = 5, Width = 4)
Specify height: 3
```

Center-Length Option

The center of the box is the point where the center of the gravity of the box lies. This option is used to create a box by specifying the center of the box, as well as the length, width, and height of the box. The following prompt sequence is issued when you choose the **Box** button:

```
Specify corner of box or [CEnter] <0,0,0>: CE
Specify center of box <0,0,0>: 4,4
```

Specify corner or [Cube/Length]: **L**
 Specify length: **8**
 Specify width: **6**
 Specify height: **3**



Tip

The Corner-Length option is similar to the Center-Length option except that in the Corner-Length option you will define the first corner of the box and the length, width, and height of the box.

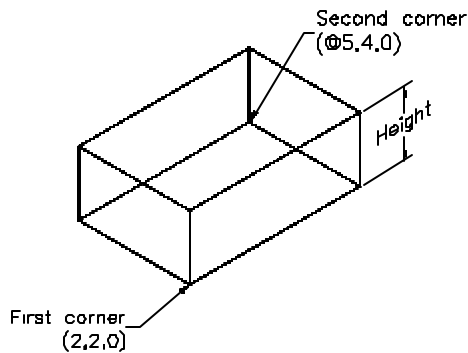


Figure 24-1 Creating the box using the two corner option

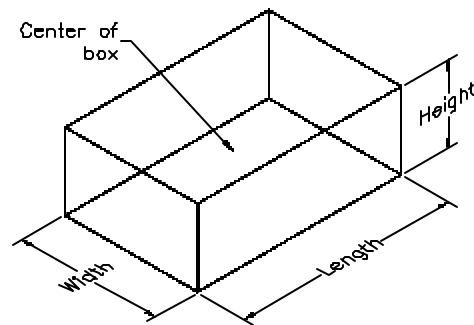


Figure 24-2 Creating the box using the Corner-Length option

Corner-Cube Option

This option is used to create a cube starting from a specified corner. Since you are creating a cube, it will prompt for only the length of the cube that will also be the width and the height. The prompt sequence for creating the cube using the Corner-Cube option is as follows:

Specify corner of box or [Center] <0,0,0>: **2,2**
 Specify corner or [Cube/Length]: **C**
 Specify length: **5**



Tip

The Center-Cube option is similar to the Corner-Cube option, except that in the Center-Cube option you will define the center and the length of the cube.

Creating a Solid Cone (CONE Command)

Toolbar: Solids > Cone
Menu: Draw > Solids > Cone
Command: CONE



The **CONE** command creates a solid cone with an elliptical or circular base. This command provides you with the option of defining the height of the cone or the location of the apex of the cone. Defining the location of the apex will also define the height of the cone and the orientation of the base of the cone from the XY plane. The methods of

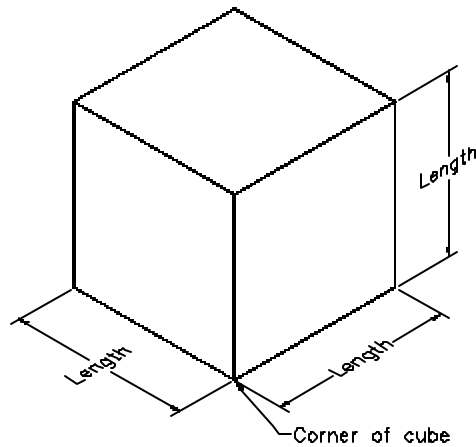


Figure 24-3 Creating the cube using the Corner-Cube option

creating the circular and the elliptical cone are discussed next.

Circular Cone

This method is used to create a cone with a circular base. The prompt sequence that will be issued when you choose the **Cone** button is as follows:

Current wire frame density: ISOLINES=4

Specify center point for base of cone or [Elliptical] <0,0,0>: Specify the center of the base.

Specify radius for base of cone or [Diameter]: Specify radius or Enter **D** to specify the diameter of the cone.

Specify height of cone or [Apex]: Specify the height of the cone.

To specify the apex, enter **A** at the **Specify height of cone or [Apex]** prompt. The prompt sequence that will follow is:

Specify apex point: Specify the location of the apex of the cone.

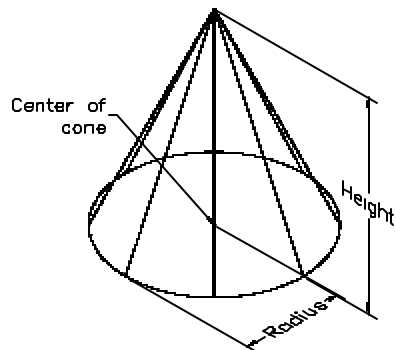


Figure 24-4 Figure showing a circular cone

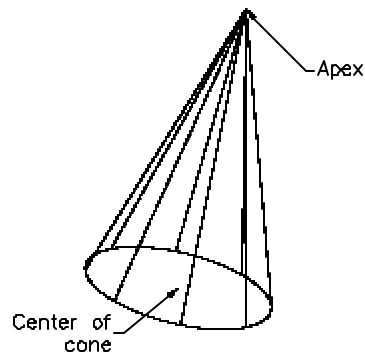


Figure 24-5 Creating a cone using the **Apex** option

Elliptical Cone

Using this method you can create a cone that has the elliptical base. To create an elliptical cone, enter **E** at the **Specify center point for base of cone or [Elliptical] <0,0,0>** prompt. You will be prompted to create the base ellipse for the cone. You can create the base ellipse using any of the methods for drawing the ellipse. However, you can not specify the rotation in this case for defining the other axis. You will have to specify the length of the other axis. Once you have entered all these values, you will be prompted to specify the height of the cone or the apex of the cone. Figure 24-6 shows an elliptical cone.

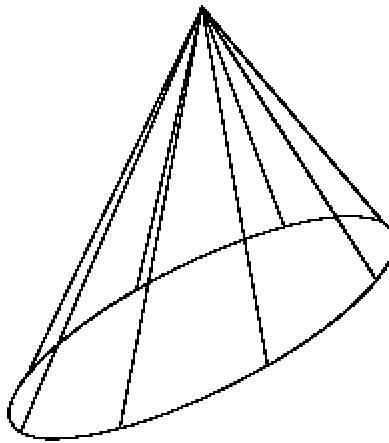


Figure 24-6 Figure showing elliptical cone

Creating a Solid Cylinder (CYLINDER Command)

Toolbar:	Solids > Cylinder
Menu:	Draw > Solids > Cylinder
Command:	CYLINDER



You can use the **CYLINDER** command to create a solid cylinder. Similar to the **CONE** command, this command also provides you with two options for creating the cylinder: **circular cylinder** and **elliptical cylinder**. This command allows you to define the height of the cylinder or the center of the other end for creating an inclined cylinder. The options for creating different types of cylinders are discussed next.

Circular Cylinder

This option is used to create a cylinder with circular base. The prompt sequence that follows when you choose the **Cylinder** button is as follows:

Current wire frame density: ISOLINES=10

Specify center point for base of cylinder or [Elliptical] <0,0,0>: Specify the location of the center point.

Specify radius for base of cylinder or [Diameter]: Specify the radius.

Specify height of cylinder or [Center of other end]: Specify the height of the cylinder.

You can also create an inclined cylinder by specifying the center of the other end. This is done by entering **C** at the **Specify height of cylinder or [Center of other end]** prompt. The prompt sequence that will follow is:

Specify center of other end of cylinder: Specify the location of center of the other end.

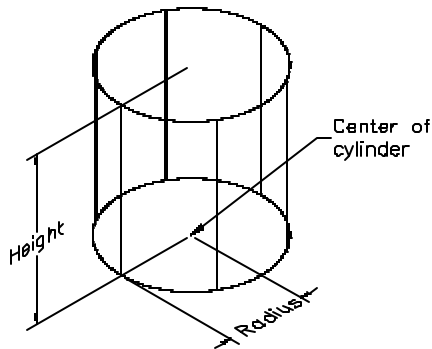


Figure 24-7 Creating circular cylinder

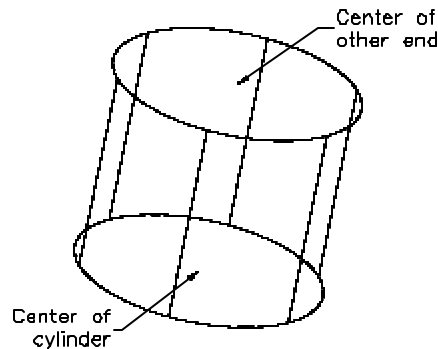


Figure 24-8 Specifying the center of the other end

Elliptical Cylinder

This option is used to create a cylinder with an elliptical base. The elliptical cylinders can be created by entering **E** at the **Specify center point for base of cylinder or [Elliptical] <0,0,0>** prompt. You will be prompted to create the ellipse for the base. You can create the ellipse for the base using any of the methods for creating the ellipse. Note that you can not define the other axis using the rotation method. You can specify the height of the cylinder or the center of the other end of cylinder. See Figure 24-9 and Figure 24-10.

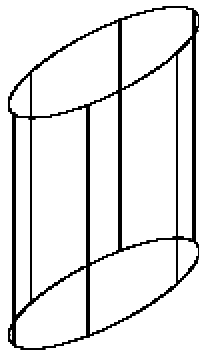


Figure 24-9 Creating an elliptical cylinder

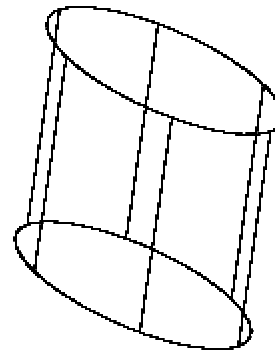


Figure 24-10 Specifying the center of the other end

Creating a Solid Sphere (SPHERE Command)

Toolbar:	Solids > Sphere
Menu:	Draw > Solids > Sphere
Command:	SPHERE



The **SPHERE** command is used to create a solid sphere (Figure 24-11). When you choose the **Sphere** button, you will be asked to specify the center of the sphere. Once you have specified the center of the sphere, you can create the sphere by defining its radius or its diameter. The following prompt sequence will be issued when you choose the **Sphere** button:

Current wire frame density: ISOLINES=current
Specify center of sphere<0,0,0>: Specify the location of center of the sphere.
Specify radius of sphere or [Diameter]: Specify the radius.

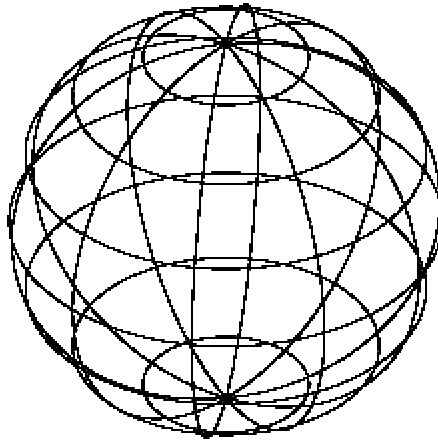


Figure 24-11 Figure showing a sphere

Creating a Solid Torus (TORUS Command)

Toolbar:	Solids > Torus
Menu:	Draw > Solids > Torus
Command:	TORUS



You can use the **TORUS** command to create a torus which is a tire tube like shape, see Figure 24-12. When you select this command, AutoCAD will prompt you to enter the Diameter or the Radius of torus and the Diameter or Radius of tube. The radius of torus is the distance from the center of the torus to the centerline of the tube. This radius can have a positive or negative value. If the value is negative, the torus has a rugby-ball like shape (Figure 24-14). A torus can be self-intersecting. If both the radii of the tube and the torus are positive and the radius of the tube is greater than the radius of the torus, the resulting solid looks like an apple (Figure 24-15). The torus is centered on the construction plane. The top half of the torus is above the construction plane and the other half is below the construction plane. The following prompt sequence is issued when you choose the **Torus** button:

Current wire frame density: ISOLINES=current
Specify center of torus or [Diameter] <0,0,0>: Specify the location of the center of the torus.
Specify radius of torus or [Diameter]: Specify the radius of the torus.

Specify radius of tube or [Diameter]: Specify the radius of the tube.

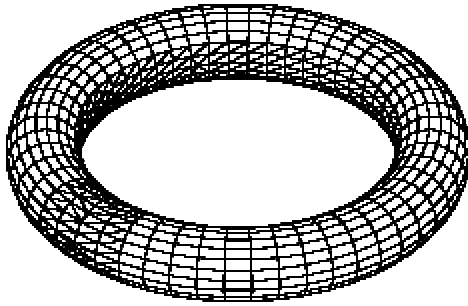


Figure 24-12 Torus with hidden lines suppressed

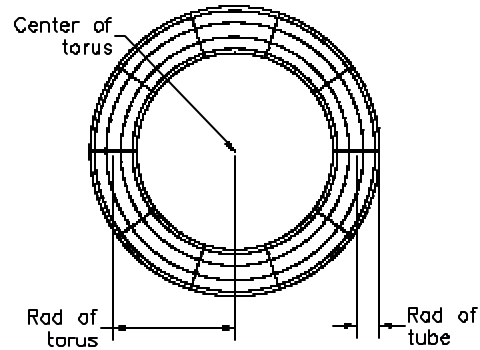


Figure 24-13 Parameters associated with torus

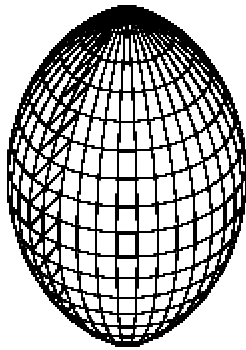


Figure 24-14 Torus with negative value of radius of torus

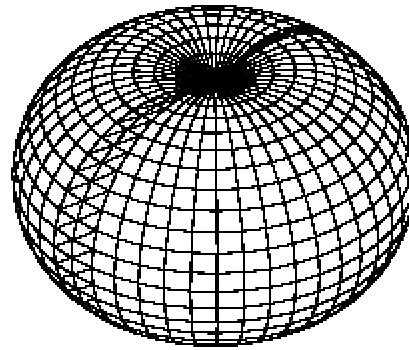


Figure 24-15 Torus with radius of tube more than radius of torus

Creating a Solid Wedge (WEDGE Command)

Toolbar:	Solids > Wedge
Menu:	Draw > Solids > Wedge
Command:	WEDGE



This command is used to create a solid wedge and is similar to the **BOX** command. This means that this command provides you with the options of creating the wedge that are similar to those of the **BOX** command.

CREATING COMPLEX SOLID MODELS

Until now you have learned how to create simple solid models using the standard solid primitives. However, the real time designs are not just the simple solid primitives, but complex solid models. These complex solid models can be created by modifying the standard solid primitives with the Boolean operations or directly by creating complex solid models by extruding

or revolving the regions. All these options of creating complex solid models are discussed next.

REGION COMMAND

Toolbar:	Draw > Region
Menu:	Draw > Region
Command:	REGION



This command is used to create regions from the selected loops or closed entities. Regions are the 2D entities with properties of the 3D solids. You can apply the Boolean operation on the regions and you can also calculate the mass properties of the regions. Bear in mind that the 2D entity you want to convert into a region should be a closed loop. Once you have created regions, the original object is deleted automatically. However, if the value of the **DELOBJ** system variable is set to **0**, the original object is retained. The valid selection set for creating the regions are closed polylines, lines, arcs, splines, circles, or ellipses. The current color, layer, linetype, and linewidth will be applied to the regions.

CREATING COMPLEX SOLID MODELS BY APPLYING THE BOOLEAN OPERATIONS

You can create complex solid models by applying the Boolean operations on the standard solid primitives. The various Boolean operations that can be performed are union, subtract, intersect and interfere. The commands used to apply these Boolean operations are discussed next.

UNION Command

Toolbar:	Solids Editing > Union
Menu:	Modify > Solids Editing > Union
Command:	UNION



This command is used to apply the union Boolean operations on the selected set of solids or regions. You can create a composite solid or region by combining them using this command. You can combine any number of solids or regions. When you invoke this command, you will be asked to select the solids or regions to be added. Figure 24-17 shows the solid model created by uniting the solids shown in Figure 24-16.

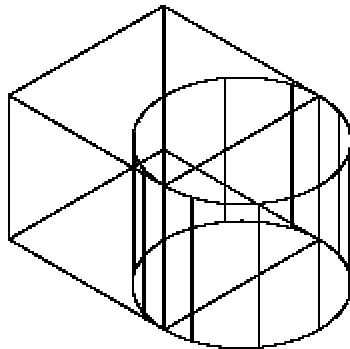


Figure 24-16 Solid models before uniting

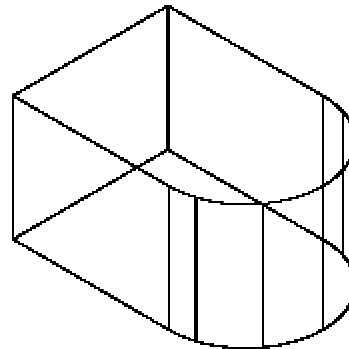


Figure 24-17 Composite solid created after union

SUBTRACT Command

Toolbar:	Solids Editing > Subtract
Menu:	Modify > Solids Editing > Subtract
Command:	SUBTRACT



This command is used to create a composite solid by removing the material common to the selected set of solids or regions. When you invoke this command, you will be prompted to select the set of solids or regions to subtract from. Once you have selected the set of solids or regions to subtract from, you will be prompted to select the solids or regions to subtract. The material common to the first selection set and the second selection set is removed from the first selection set. The resultant object will be a single composite solid. See Figure 24-18 and Figure 24-19.

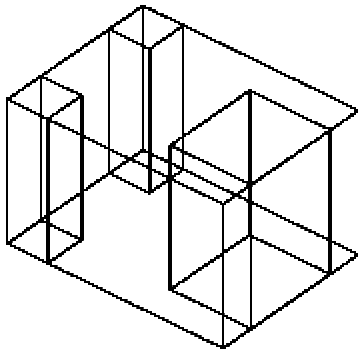


Figure 24-18 Solid models before subtracting

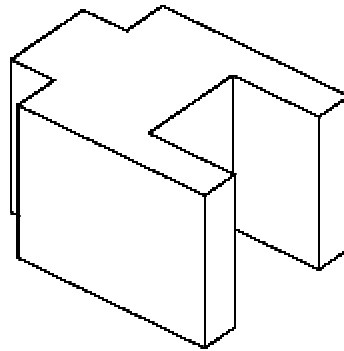


Figure 24-19 Composite solid created after subtracting

INTERSECT Command

Toolbar:	Solids Editing > Intersect
Menu:	Modify > Solids Editing > Intersect
Command:	INTERSECT



This command is used to create a composite solid or region by retaining the material common to the selected set of solids or regions. When you invoke this command, you will be asked to select the solids or regions to intersect. The material common to all the selected solids or regions will be retained to create a new composite solid (Figure 24-20 and Figure 24-21).

INTERFERE Command

Toolbar:	Solids > Interfere
Menu:	Draw > Solids > Interference
Command:	INTERFERE



This command is used to create a composite solid model by retaining the material common to the selected sets of solids. The advantage of using this command is that

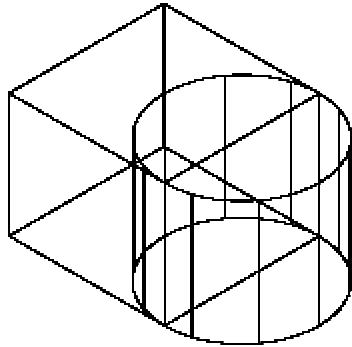


Figure 24-20 Solid models before intersecting

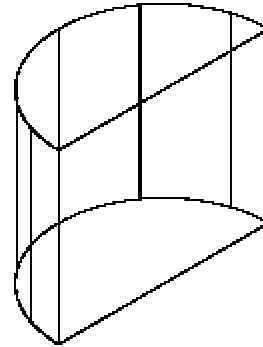


Figure 24-21 Solid created after intersecting

the original objects are also retained. This command prompts you whether you want to create the interference solid or not. If you say yes, it will create the interference solid that can be moved out and used for analyzing the interference. This command is generally used for analyzing the interference between the mating parts of the assembly. Figure 24-22 shows two mating components of the assembly with interference between them and Figure 24-23 shows the interference solid created and moved out.

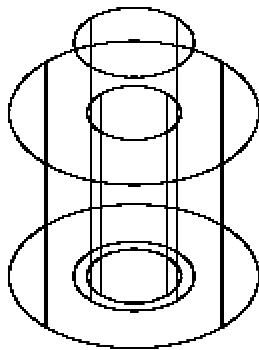


Figure 24-22 Two mating components with interference

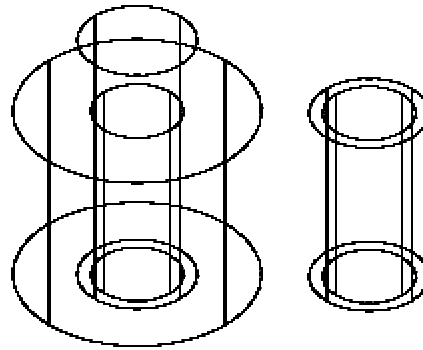


Figure 24-23 Interference solid created using the **INTERFERENCE** command

Example 1

In this example you will create the solid model shown in Figure 24-24.

1. Increase the limits to 100,100. Zoom to the limits of drawing.
2. Choose the **SW Isometric View** button.
3. Enter **UCSICON** at the Command prompt. The prompt sequence is as follows:

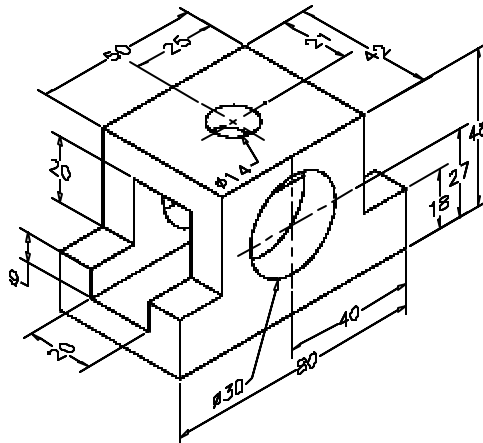


Figure 24-24 Solid model for Example 1

Enter an option [ON/OFF/All/Noorigin/ORigin/Properties] <ON>: N

4. Choose the **Box** button from the **Solids** toolbar. The prompt sequence is as follows:

Specify corner of box or [CEnter] <0,0,0>: **10,10**
 Specify corner or [Cube/Length]: **L**
 Specify length: **80**
 Specify width: **42**
 Specify height: **48**

5. Again choose the **Box** button from the **Solids** toolbar. The prompt sequence is as follows:

Specify corner of box or [CEnter] <0,0,0>: Specify a point on the screen.
 Specify corner or [Cube/Length]: **L**
 Specify length: **15**
 Specify width: **42**
 Specify height: **30**

6. Move the new box inside the old box using the midpoints of both the boxes. Copy the new box to the other side of the box (Figure 24-25).
7. Choose the **Subtract** button from the **Solids Editing** toolbar. The prompt sequence is as follows:

Select solids and regions to subtract from ..
 Select objects: Select the bigger box.
 Select objects: «
 Select solids and regions to subtract ..
 Select objects: Select one of the smaller box.
 Select objects: Select the other smaller box.

Select objects: «

The boxes after subtraction should look similar to the one shown in Figure 24-26.

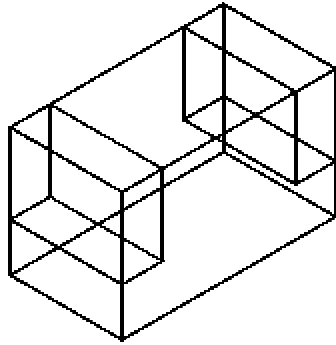


Figure 24-25 Figure showing boxes moved inside the bigger box using the midpoints

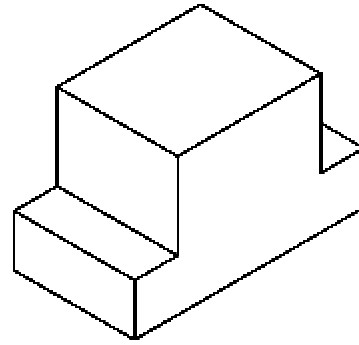


Figure 24-26 Model after subtracting the smaller boxes

8. Again choose the **Box** button from the **Solids** toolbar. The prompt sequence is as follows:

Specify corner of box or [Center] <0,0,0>: Specify a point on the screen.
 Specify corner or [Cube/Length]: **L**
 Specify length: **80**
 Specify width: **20**
 Specify height: **29**

9. Move the new box inside the existing model such that the midpoint of the left edge of the base of the box is 9 units above that of the model, see Figure 24-27. (Use the **From** option.)
10. Choose the **Subtract** button from the **Solids Editing** toolbar. The prompt sequence is as follows:

Select solids and regions to subtract from ..
 Select objects: Select the model.
 Select objects: «
 Select solids and regions to subtract ..
 Select objects: Select the box.
 Select objects: «

The model should look similar to the one shown in Figure 24-28.



Tip

You can control the display of the silhouette lines in the solid model using the **DISPSILH** system variable. Set the value of this variable to **1** to suppress the display of silhouette lines.

11. Choose the **Cylinder** button from the **Solids** toolbar. The prompt sequence is as follows:

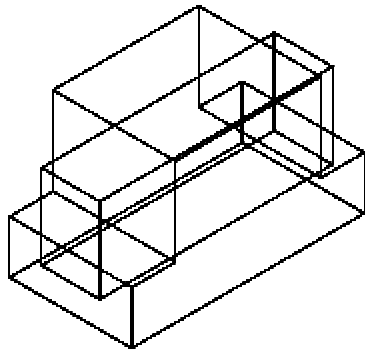


Figure 24-27 Model before subtraction

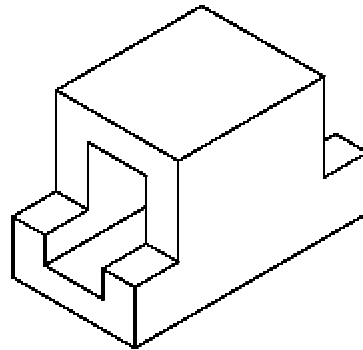


Figure 24-28 Model after subtraction

Current wire frame density: ISOLINES=4

Specify center point for base of cylinder or [Elliptical] <0,0,0>: Specify the center of the cylinder on the top face of the model using either **Object Snap Tracking** or using the **From** option.

Specify radius for base of cylinder or [Diameter]: 7

Specify height of cylinder or [Center of other end]: -10 (The negative value of height creates the cylinder extruded in the negative Z direction, see Figure 24-29.)

12. Subtract this cylinder from the existing model (Figure 24-30).

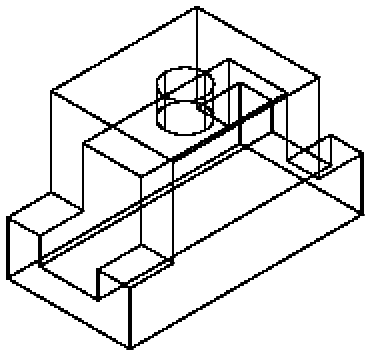


Figure 24-29 Model before subtraction

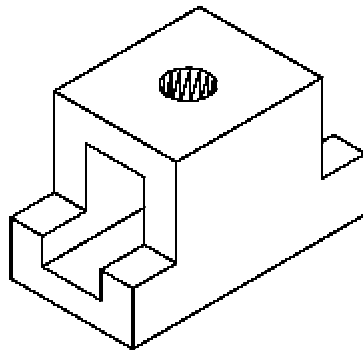


Figure 24-30 Model after subtraction

13. Choose the **X Axis Rotate UCS** button from the **UCS** toolbar. The prompt sequence is as follows:

Specify rotation angle about X axis <90>: «

14. Choose the **Cylinder** button from the **Solids** toolbar. The prompt sequence is as follows:

Current wire frame density: ISOLINES=4

Specify center point for base of cylinder or [Elliptical] <0,0,0>: Use the **From** option to

specify the center point of the cylinder at a distance of 27 units from the midpoint of the horizontal edge of the base.

Specify radius for base of cylinder or [Diameter]: **15**

Specify height of cylinder or [Center of other end]: **-42**

15. Subtract this cylinder from the model. The final model should look similar to the one shown in Figure 24-31.

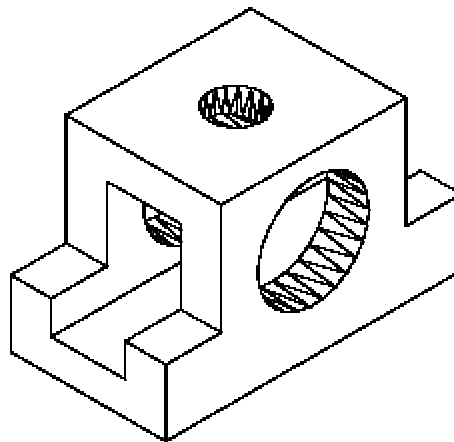


Figure 24-31 Final solid model for Example 1

Exercise 1

Mechanical

In this exercise you will create the solid model shown in Figure 24-32. Save this drawing with the name \Ch-24\Exercise1.dwg

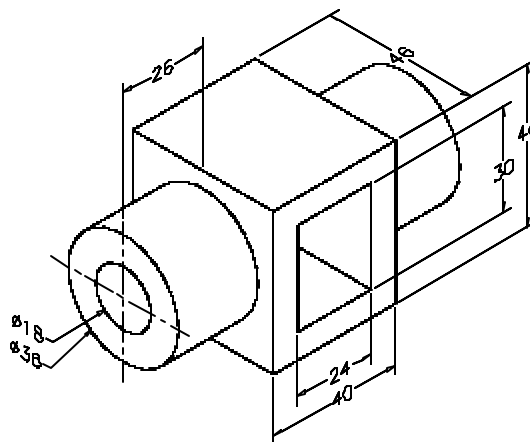


Figure 24-32 Figure showing solid model for Exercise 1

Sometimes the shape of the solid model is such that it can not be created by just applying the Boolean operations on the standard solid primitives. For the situations such as this, AutoCAD provides the commands called **EXTRUDE** and **REVOLVE**. Using these commands you can create the solid models of any complex shape. These commands are discussed in detail.

EXTRUDE COMMAND

Toolbar:	Solids > Extrude
Menu:	Draw > Solids > Extrude
Command:	EXTRUDE



This command is used to create a complex solid model by extruding a closed 2D entity or a region along the Z axis direction or about a specified path. Remember that for extrusion, the original entity must be a closed loop or a region. The solid model can be created by extruding along the Z axis direction or about a specified path. Both of these options of creating the extruded solid are discussed next.

Extruding Along Z Axis

This is the default option and is used to create a solid model by extruding a closed 2D entity or a region along the Z axis direction. Figure 24-33 shows a region to be converted into an extruded solid and Figure 24-34 shows the solid created upon extruding the region. The prompt sequence that will be issued when you choose the **Extrude** button is:

```
Current wire frame density: ISOLINES= 4  
Select objects: Select the region.  
Select objects: «  
Specify height of extrusion or [Path]: 2  
Specify angle of taper for extrusion <0>: «
```

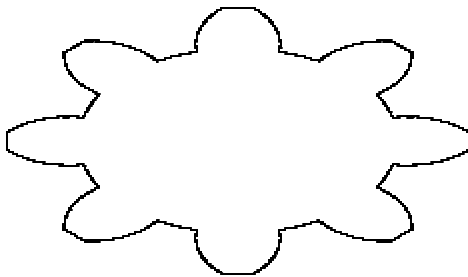


Figure 24-33 Base object for extruding

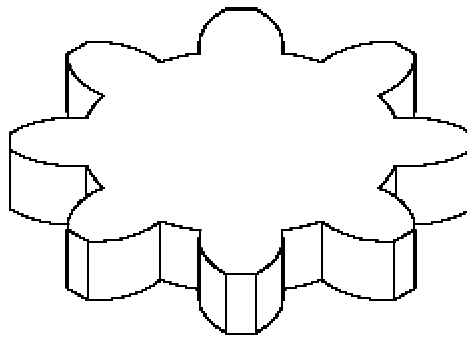


Figure 24-34 Solid created upon extruding

You can also specify the taper angle for the extruded solid. The positive value of the taper angle will taper in from the base object and the negative value will taper out of the base object (Figure 24-35).

REVOLVE COMMAND

Toolbar:	Solids > Revolve
Menu:	Draw > Solids > Revolve
Command:	REVOLVE



This command is used to create a complex solid by revolving closed 2D entities or regions about a specified revolution axis. The entire 2D entity should be on one side of the revolution axis. Self-intersecting and crossed entities can not be revolved using this command. The direction of revolution is determined using the right-hand thumb rule. The axis of rotation can be defined by specifying two points, using the X or the Y axis of the current UCS, or using an existing object. The following prompt sequence will be issued when you choose this button:

Current wire frame density: ISOLINES=4
Select objects: Select the object or region to revolve.
Select objects: «
Specify start point for axis of revolution or
define axis by [Object/X (axis)/Y (axis)]:

Specifying the Start Point for Axis of Revolution

This option is used when you want to define the axis of revolution using two points: the start point and the endpoint of the axis of revolution. The positive direction of the axis will be from the start point to the endpoint and the direction of revolution will be defined using the right-hand thumb rule. Before revolving, make sure the complete 2D entity is on one side of the axis of revolution.

Object

This option is used to create a revolved solid by revolving a selected 2D entity or a region about a specified object. The valid entities that can be used as object for defining the axis of revolution are line, or a single segment of a polyline. If the polyline selected as the object consists of more than one entity, then AutoCAD draws an imaginary line from the start point of the first segment to the endpoint of the last segment. This imaginary line is then taken as the object for revolution.

X (axis)

This option uses the positive direction of the X axis of the current UCS for revolving the selected entity. If the selected entity is not completely on one side of the X axis, it will give you an error message that it can not revolve the object.

Y (axis)

This option uses the positive direction of the Y axis of the current UCS as the axis of revolution for creating the revolved solid.

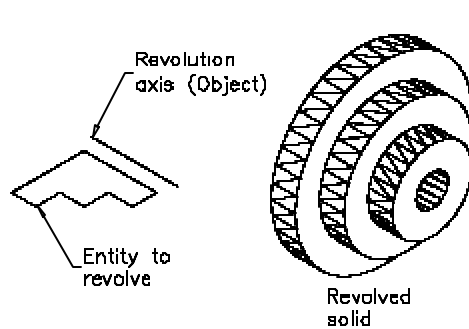


Figure 24-38 Creating a revolved solid

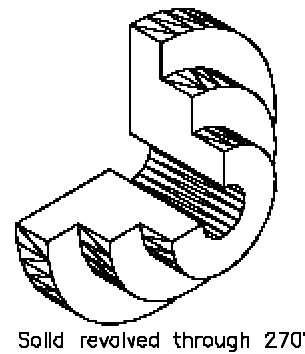


Figure 24-39 Solid revolved to an angle

Example 2

In this example you will create the solid model shown in Figure 24-40.

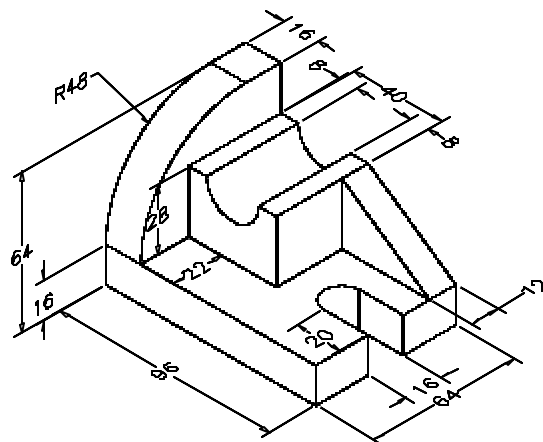


Figure 24-40 Solid model for Example 2

1. Increase the limits and then zoom to the limits of the drawing.
2. Create the base of the model with the dimensions as shown in Figure 24-41.
3. Choose the **Region** button from the **Draw** toolbar. The prompt sequence is as follows:

Select objects: Select the complete base.

Select objects: «

1 loop extracted.

1 Region created.

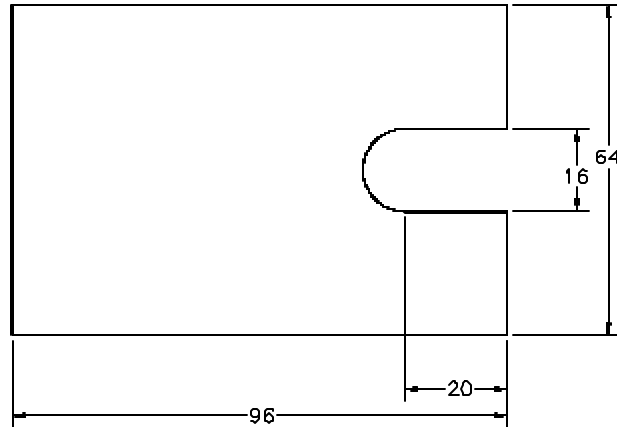


Figure 24-41 Base for the solid model

4. Choose the **Extrude** button from the **Solids** toolbar. The prompt sequence is as follows:

Select objects: Select the region.
Select objects: «
Specify height of extrusion or [Path]: **16**
Specify angle of taper for extrusion <0>: «

5. Change the viewpoint to the SE Isometric view.

6. Choose the **Wedge** button from the **Solids** toolbar. The prompt sequence is as follows:

Specify first corner of wedge or [Center] <0,0,0>: Pick a point on the screen.
Specify corner or [Cube/Length]: **L**
Specify length: **40**
Specify width: **12**
Specify height: **28**

7. Move this wedge using the endpoint so that it is properly aligned with the base, see Figure 24-42.
8. Choose the **X Axis Rotate UCS** button from the **UCS** toolbar. The prompt sequence is as follows:

Specify rotation angle about X axis <90>: «

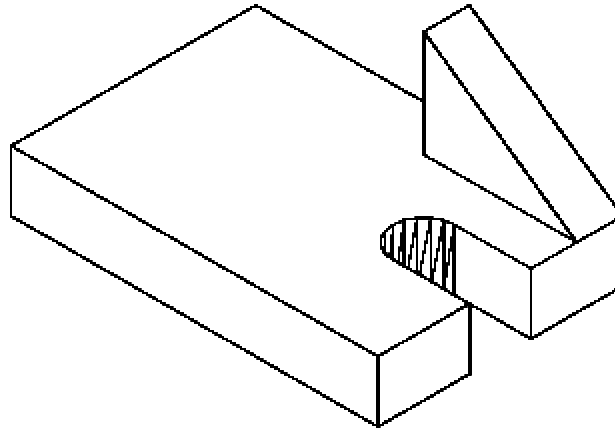


Figure 24-42 Wedge aligned with the base

9. Draw the object shown in Figure 24-43 and then convert it into region.
10. Choose the **Extrude** button from the **Solids** toolbar. The prompt sequence is as follows:

Current wire frame density: ISOLINES= 4
 Select objects: Select the region.
 Select objects: «
 Specify height of extrusion or [Path]: 42
 Specify angle of taper for extrusion <0>: «

11. Move it on the base using the endpoint as shown Figure 24-44.

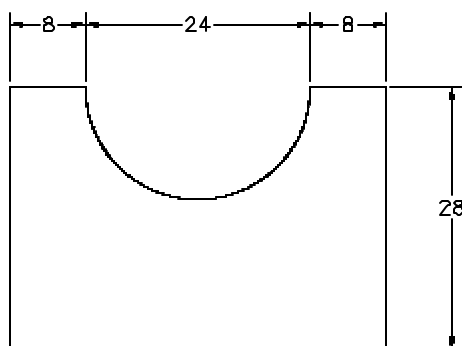


Figure 24-43 Figure showing the region

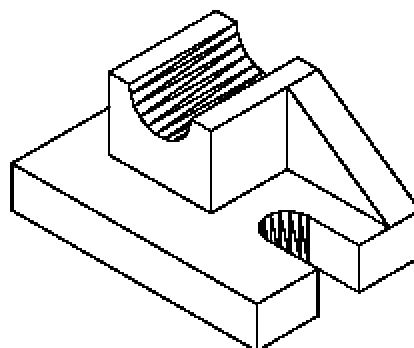


Figure 24-44 Solid aligned with the base

12. Choose the **Y Axis Rotation UCS** button from the **UCS** toolbar. The prompt sequence is as follows:

Specify rotation angle about Y axis <90>: «

13. Draw the object shown in Figure 24-45 and then convert it into a region.
14. Extrude it and move it so that it is properly aligned with the base, see Figure 24-46.

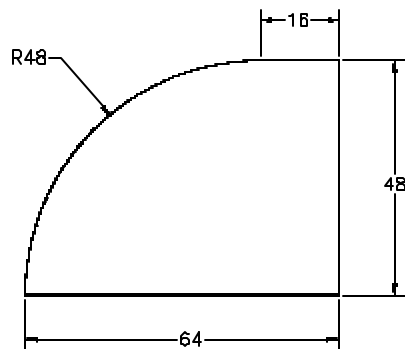


Figure 24-45 Figure showing the region

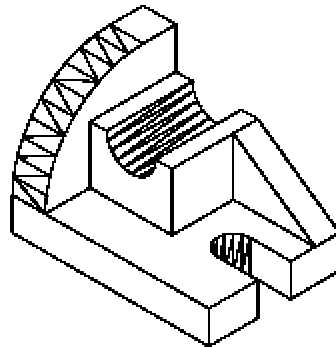


Figure 24-46 Solid aligned with the base

15. Choose the **Union** button from the **Solids Editing** toolbar. The prompt sequence is as follows:

Select objects: Select all the objects.

Select objects: «

The final solid model should look similar to the one shown in Figure 24-47.

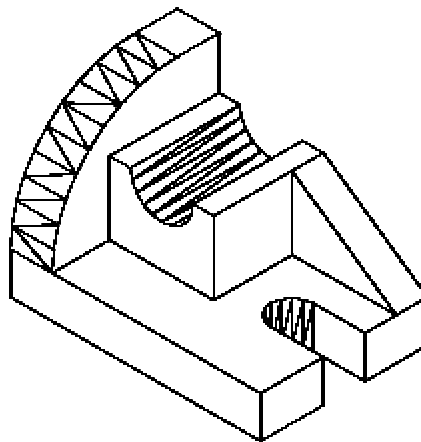


Figure 24-47 Solid model for Example 2

FILLETING THE SOLID MODELS (FILLET COMMAND)

Toolbar:	Modify > Fillet
Menu:	Modify > Fillet
Command:	FILLET



As mentioned earlier, the **Fillet** command is used to round the edges or the corners of the models. This is generally done to reduce the stress concentration area in the model.

The behavior of this command is different while working with 2D entities from the behavior while working with solid models. Therefore, it is very important for the user to first understand the use of this command to fillet the edges of the solid models. Figure 24-48 shows two lines that are selected to fillet. Now, as these lines are nothing but 2D entities, when you select these two lines to fillet, the result will be as shown in Figure 24-49.



Figure 24-48 Lines before filleting

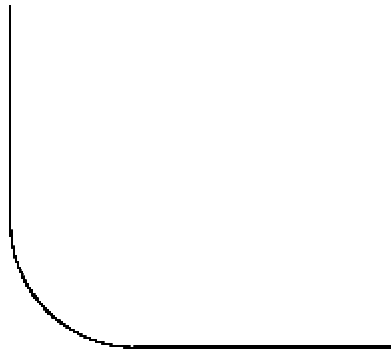


Figure 24-49 Lines after filleting

This shows that if actually there would have been a vertical edge at the corner of the two lines shown in Figure 24-48, then it would have been filleted. However, in the 3D models, you directly have the vertical edges and therefore, you have to select the vertical edge to fillet. Figure 24-50 shows a solid model. To fillet this model you just have to select the vertical edges, see Figure 24-51.

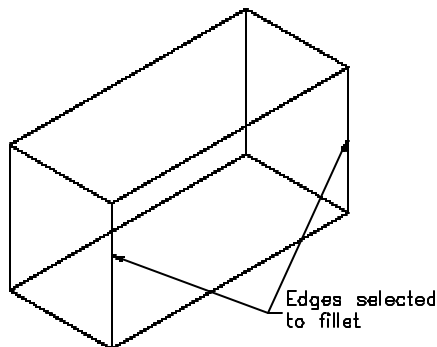


Figure 24-50 Selecting the edges to fillet

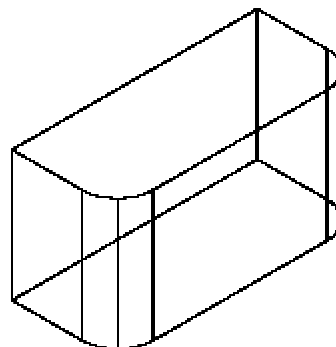


Figure 24-51 Model after filleting the edges

The prompt sequence that will be followed when you choose the **Fillet** button is:

Current settings: Mode = TRIM, Radius = 2.0000

Select first object or [Polyline/Radius/Trim]: Select the edges to fillet. You will be allowed to select only one edge at this moment.

Enter fillet radius <2.0000>: Enter the required fillet radius.

Select an edge or [Chain/Radius]: Select the other edges to fillet or enter an option.

Chain

This option is used to select the other tangential edges on the model. For example, you can select all the tangential edges on the top or the bottom face of the solid model shown in Figure 24-52 for filleting in just a single attempt using the **Chain** option.

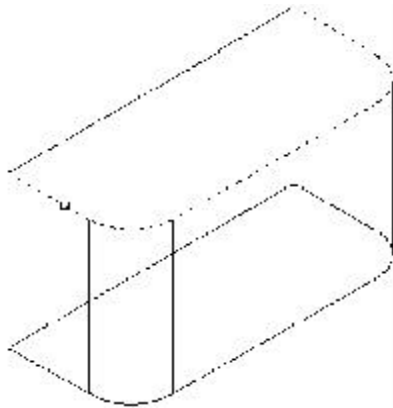


Figure 24-52 Selecting the edges using the Chain option

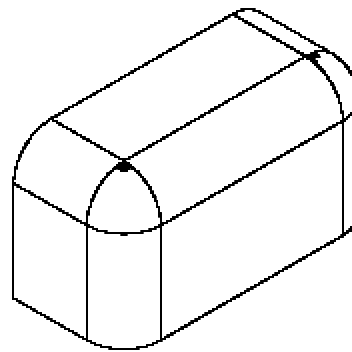


Figure 24-53 The fillet created using the Chain option

Radius

This option is used to redefine the fillet radius.

CHAMFERING THE SOLID MODELS (CHAMFER COMMAND)

Toolbar:	Modify > Chamfer
Menu:	Modify > Chamfer
Command:	CHAMFER



The chamfer command is used to bevel the edges of the solid models. This command is also used to reduce the area of the stress concentration in the solid models. The working of this command is also different while working with the solid models. The prompt sequence that follows when you choose the **Chamfer** button is:

(TRIM mode) Current chamfer Dist1 = 1.0000, Dist2 = 0.5000

Select first line or [Polyline/Distance/Angle/Trim/Method]: Select the edge to fillet. One of the

faces associated with the edge will be selected and highlighted.

Base surface selection...

Enter surface selection option [Next/OK (current)] <OK>: Give a null response if you want to make this face as the base surface. Otherwise enter **N** at this prompt.

Specify base surface chamfer distance <default value>: Specify the distance.

Specify other surface chamfer distance <default value>: Specify the distance.

Select an edge or [Loop]: Select the edge to fillet.

Loop

This option is used to select all the edges comprised of a loop on the selected face of the solid model. To use this option you will have to select any of the edges that comprise the loop at the **Select an edge loop or [Edge]:** prompt.

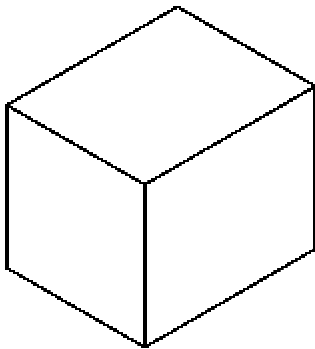


Figure 24-54 Solid model before chamfering

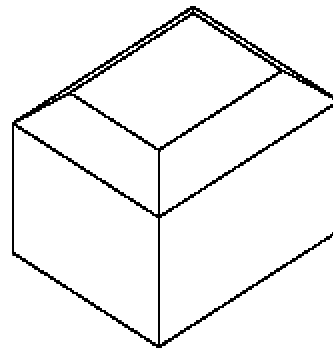


Figure 24-55 Solid model after chamfering

ROTATING SOLID MODELS IN 3D SPACE (ROTATE3D COMMAND)

Menu: Modify > 3D Operations > Rotate 3D
Command: ROTATE3D

This command is used to rotate the selected solid model in the 3D space about a specified axis. Once again the right-hand thumb rule will be used to determine the direction of rotation of solid model in the 3D space. The prompt sequence that will follow when you choose this command from the **Modify** menu is:

Current positive angle: ANGDIR= counterclockwise ANGBASE= 0

Select objects: Select the solid model.

Select objects: «

Specify first point on axis or define axis by

[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]:

2points Option

This is the default option for rotating the solid models. This option allows you to rotate the solid model about an axis specified using two points. The direction of the axis will be from the

first point to the second point. Using this direction of the axis you can calculate the direction of rotation of the solid model by applying the right-hand thumb rule. The prompt sequence that will follow when you invoke this option is:

[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: «

Specify first point on axis: Specify the first point of the rotation axis. See Figure 24-56.

Specify second point on axis: Specify the second point of the rotation axis. See Figure 24-56.

Specify rotation angle or [Reference]: Specify the angle of rotation.

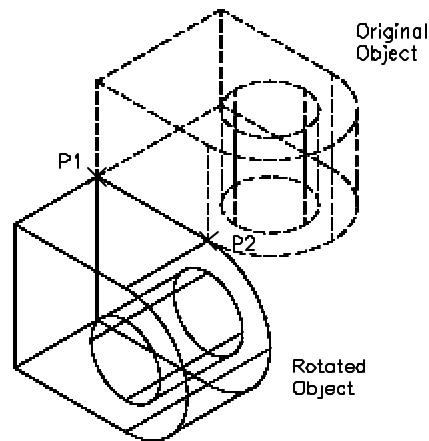


Figure 24-56 Rotating the solid model using the 2points option

Object Option

This option is used to rotate the solid model in the 3D space using a 2D entity. The 2D entities that can be used are lines, circles, arcs, or 2D polyline segments. If the selected entity is a line or a straight polyline segment, then it will be directly taken as the rotation axis. However, if the selected entity is an arc or a circle, then an imaginary axis will be drawn starting from the center and normal to the plane in which the arc or the circle is drawn. The object will be then rotated about this imaginary axis. The prompt sequence that will follow when you invoke this option is:

[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: **O**

Select a line, circle, arc, or 2D-polyline segment: Select the 2D entity as shown in Figure 24-57.

Specify rotation angle or [Reference]: Specify the angle of rotation.

Last Option

This option uses the same axis that was last selected to rotate the solid model.

View Option

This option is used to rotate the solid model about the viewing plane. In this case the viewing plane is the screen of the computer. This option draws an imaginary axis starting from the specified point and continues normal to the viewing plane. The model is then rotated about

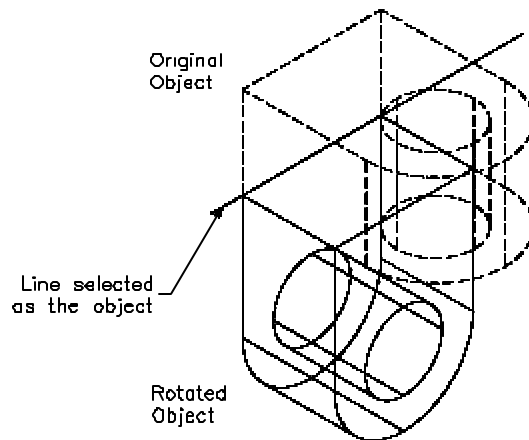


Figure 24-57 Rotating the solid model using the Object option

this axis. The prompt sequence that will follow when you invoke this option is:

Specify first point on axis or define axis by

[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: **V**

Specify a point on the view direction axis <0,0,0>: Specify the point on the view plane as shown in Figure 24-58.

Specify rotation angle or [Reference]: Specify the angle of rotation.

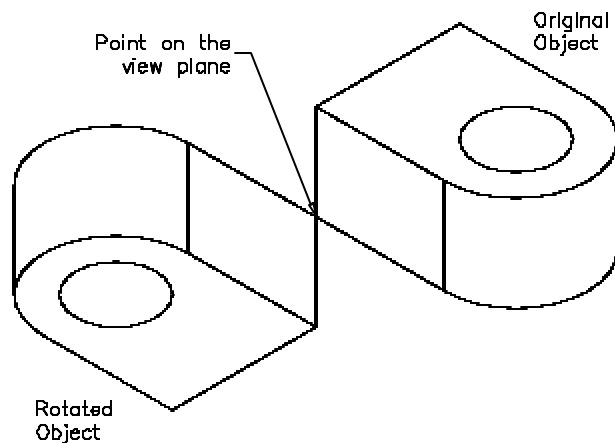


Figure 24-58 Rotating the solid model using the View option by an angle of 180 degrees

Xaxis Option

This option is used to rotate the solid model about the positive X axis of the current UCS.

When you invoke this option, you will be prompted to select a point on the X axis. The prompt sequence that will follow when you invoke this command is:

Specify first point on axis or define axis by
[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: **X**
Specify a point on the X axis <0,0,0>: Specify the point on the X axis.
Specify rotation angle or [Reference]: Specify the angle of rotation.

Yaxis Option

This option is used to rotate the solid model about the positive Y axis of the current UCS. When you invoke this option, you will be prompted to select a point on the Y axis. The prompt sequence that will follow when you invoke this command is:

Specify first point on axis or define axis by
[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: **Y**
Specify a point on the Y axis <0,0,0>: Specify the point on the Y axis.
Specify rotation angle or [Reference]: Specify the angle of rotation.

Zaxis Option

This option is used to rotate the solid model about the positive Z axis of the current UCS. When you invoke this option, you will be prompted to select a point on the Z axis. The prompt sequence that will follow when you invoke this command is:

Specify first point on axis or define axis by
[Object/Last/View/Xaxis/Yaxis/Zaxis/2points]: **Z**
Specify a point on the Z axis <0,0,0>: Specify the point on the Z axis.
Specify rotation angle or [Reference]: Specify the angle of rotation.

MIRRORING THE SOLID MODELS IN THE 3D SPACE (MIRROR3D COMMAND)

Menu:	Modify > 3D Operations > Mirror 3D
Command:	MIRROR3D

This command is used to mirror the solid models about a specified plane in the space. The prompt sequence that will follow when you choose this command from the **Modify** menu is:

Select objects: Select the solid model to be mirrored.
Select objects: «
Specify first point of mirror plane (3 points) or
[Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>:

3points Option

This is the default option for mirroring the solid models. As discussed earlier, a line can be defined by two points from which the line passes. Similarly, a plane can be defined by three points through which it passes. This option allows you to specify the three points from which

the mirroring plane passes. The prompt sequence that follows when you invoke this command is:

Specify first point of mirror plane (3 points) or
 [Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>: «
 Specify first point on mirror plane: Specify the first point on the plane. See Figure 24-59.
 Specify second point on mirror plane: Specify the second point on the plane. See Figure 24-59.
 Specify third point on mirror plane: Specify the third point on the plane. See Figure 24-59.
 Delete source objects? [Yes/No] <N>: «

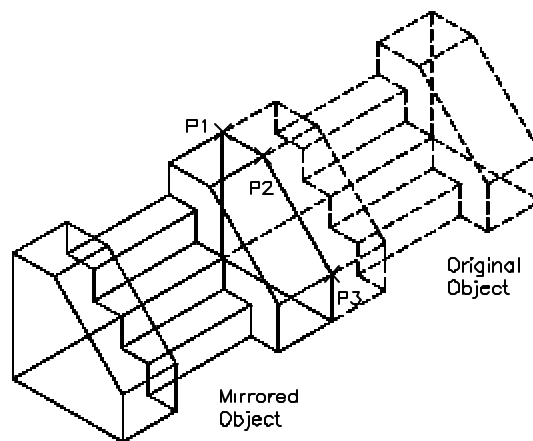


Figure 24-59 Mirroring the solid model using the 3points option

Object

This option is used to mirror the solid model using a 2D entity. The 2D entities that can be used to mirror the solids are circles, arcs, and 2D polyline segments. The prompt sequence that follows when you invoke this option is:

Specify first point of mirror plane (3 points) or
 [Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>: **O**
 Select a circle, arc, or 2D-polyline segment: Select the 2D entity as shown in Figure 24-60.
 Delete source objects? [Yes/No] <N>: «

Zaxis

This option allows you to define a mirroring plane using two points. The first point is the point on the plane and the second point is the point on the positive direction of Z the axis of that plane. The prompt sequence that follows when you invoke this option is:

Specify first point of mirror plane (3 points) or
 [Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>: **Z**
 Specify point on mirror plane: Specify the point on the plane as shown in Figure 24-61.

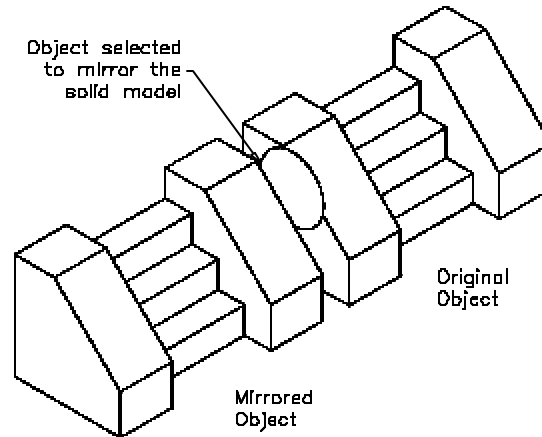


Figure 24-60 Mirroring the solid model using the Object option

Specify point on Z-axis (normal) of mirror plane: Specify the point on the Z direction of the plane as shown in Figure 24-61.

Delete source objects? [Yes/No] <N>: «

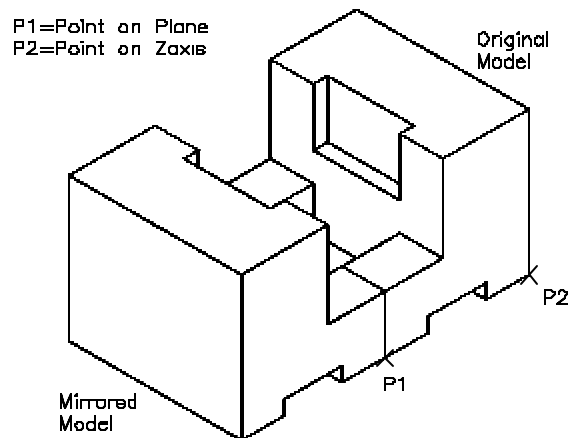


Figure 24-61 Mirroring the solid model using the Zaxis option

View Option

This is one of the most interesting options that is provided for mirroring the solid models. This option is used to mirror the selected solid model about the viewing plane. The viewing plane in this case is the screen of the monitor. The prompt sequence that will follow when you invoke this command is:

Specify first point of mirror plane (3 points) or
 [Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>: **V**
 Specify point on view plane <0,0,0>: Specify the point on the view plane.
 Delete source objects? [Yes/No] <N>: «

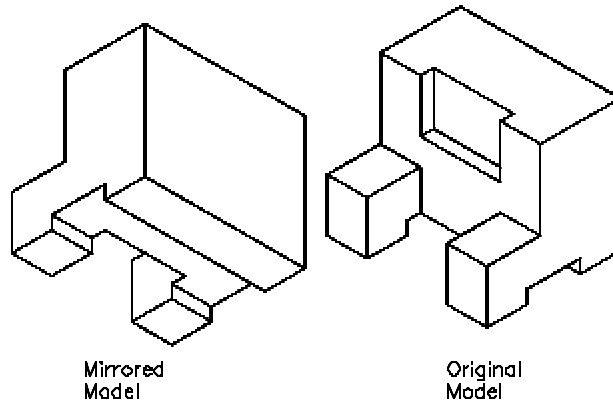


Figure 24-62 Figure showing the solid model mirrored using the View option and then isolated using the MOVE command



Tip

Because the model is mirrored about the view plane, the new model will be placed over the original model when you view it from the current viewpoint. The best option to use to view the mirrored model is to hide the hidden lines using the **HIDE** command. You can also move the new model away from the last model using the **MOVE** command or change the viewpoint for viewing both the models simultaneously.

XY/YZ/ZX

These options are used to mirror the solid model about the XY, YZ or ZX planes of the current UCS.



Tip

All these planes will be considered with reference to the current orientation of the UCS and not with the world position of the UCS. This means that when you select the **XY** option to mirror the solid model, then the XY plane of the current UCS will be considered to mirror the model and not the XY plane of the world UCS.

Example 3

In this example you will create the solid model shown in Figure 24-63. Assume the missing dimensions.

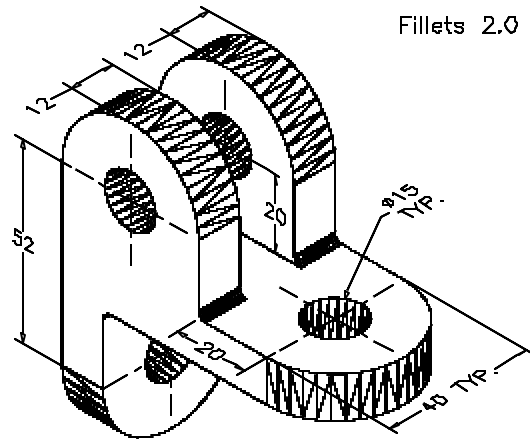


Figure 24-63 Solid model for Example 3

1. Open a new file and then set the limits to **100,100**. Change the overall scale factor to **10** using the **Dimension Style Manager** dialog box.
2. Create the base of the model and then change the viewpoint to **1,-1,1**, see Figure 24-64.

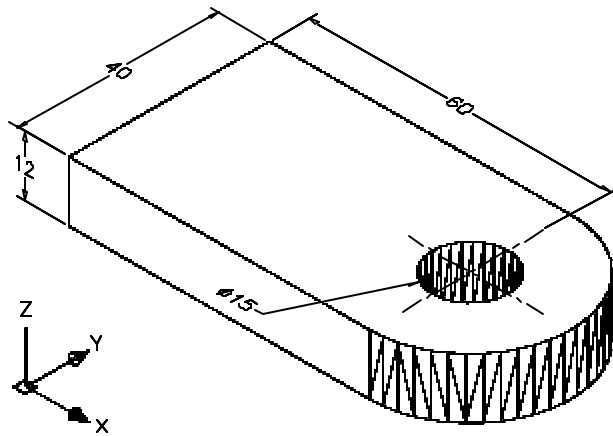


Figure 24-64 Base for the model

3. Change the UCS by rotating it around the X axis by an angle of 90 degrees.
4. Change the viewpoint to the plan view of the current UCS using the **PLAN** command.
5. Create the next object and then move it using the endpoint to align with the base of the model as shown in Figure 24-65.

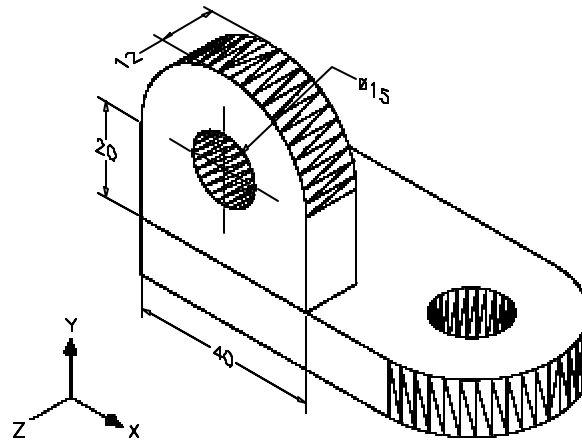


Figure 24-65 After creating the next object and moving it

6. Choose **3D Operations > Mirror 3D** from the **Modify** menu. The prompt sequence is as follows:

Select objects: Select the last object.

Select objects: «

Specify first point of mirror plane (3 points) or
[Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>: **Z**

Specify point on mirror plane: Select P1 as shown in Figure 24-66.

Specify point on Z-axis (normal) of mirror plane: Select P2 as shown in Figure 24-66.

Delete source objects? [Yes/No] <N>: «

7. The object after mirroring should look similar to the one shown in Figure 24-67.

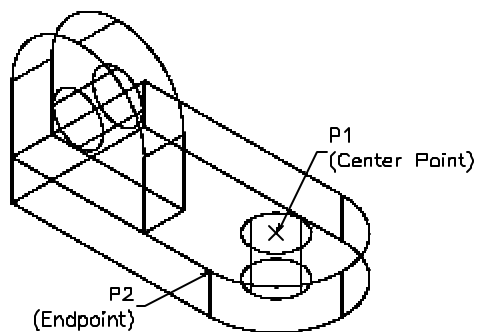


Figure 24-66 Selecting the points to mirror

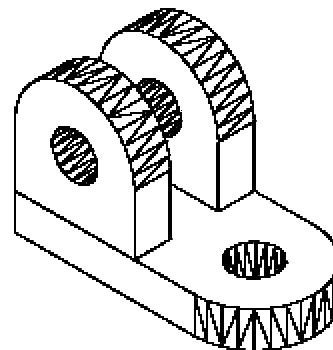


Figure 24-67 Model after mirroring

8. Change the UCS by rotating it around the Y axis through an angle of 90 degrees.

9. Create the next object of the required dimensions and then move it using the endpoints as shown in Figure 24-68.

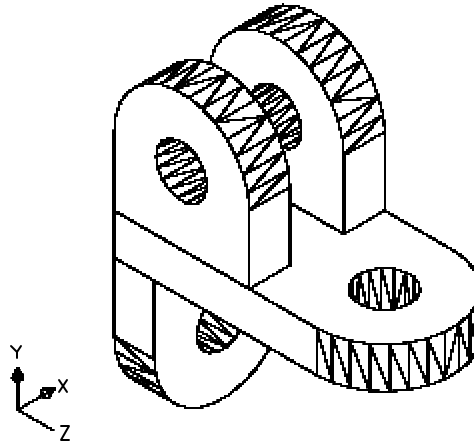


Figure 24-68 Model after creating the next object and moving it

10. Invoke the **UNION** command and then union all the objects, see Figure 24-69.

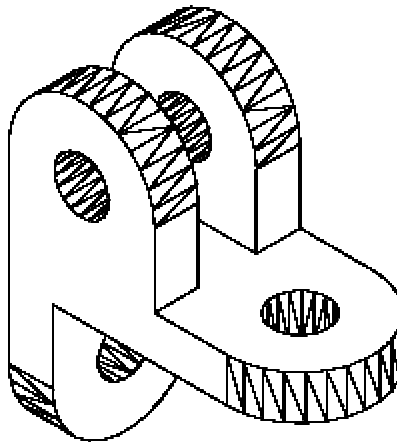


Figure 24-69 Model after union

11. Choose the **Fillet** button from the **Modify** toolbar. The prompt sequence that follows is:

Current settings: Mode = TRIM, Radius = 0.5000

Select first object or [Polyline/Radius/Trim]: Select the first edge as shown in Figure 24-70.

Enter fillet radius <0.5000>: 2

Select an edge or [Chain/Radius]: Select the second edge as shown in Figure 24-70.

Select an edge or [Chain/Radius]: Select the third edge as shown in Figure 24-70.

Select an edge or [Chain/Radius]: «

3 edge(s) selected for fillet.

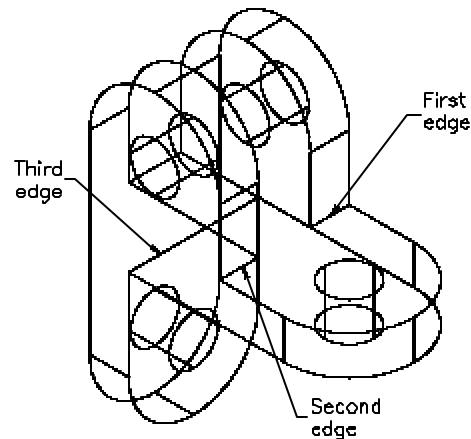


Figure 24-70 Selecting the edges for filleting

12. The final model for Example 3 should look similar to the one shown in Figure 24-71.

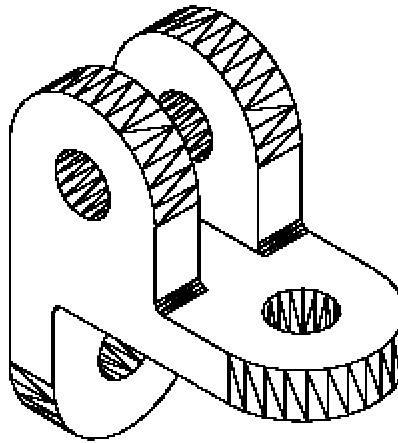


Figure 24-71 The final model for Example 3

CREATING THE ARRAYS IN THE 3D SPACE (3DARRAY COMMAND)

Menu:	Modify > 3D Operations > 3D Array
Command:	3DARRAY

As mentioned in the previous chapters, the arrays are defined as the method of creating the multiple copies of the selected object in the rectangular or polar fashion. The 3D arrays can also be created similar to the 2D arrays. The only difference is that in 3D array, another factor

called the Z axis is also taken into consideration. There are two types of 3D arrays. Both of these types are discussed next.

3D Rectangular Array

This is the method of arranging the solid model along the edges of a box. In this type of array you will have to specify three parameters. They are the rows (along the X axis), the columns (along the Y axis), and the levels (along the Z axis). You will also have to specify the distances between the rows, columns, and the levels. The 3D rectangular array can be easily understood by taking an example shown in Figure 24-72. This figure shows the two floors of a building. Initially, only one chair is placed on the ground floor. Now, if you create a 2D rectangular array then the chairs will be arranged only on the ground floor. However, when you create the 3D rectangular array, then the chairs will be arranged on the first floor (along the Z axis) as well as the ground floor, see Figure 24-73. In this example the number of rows are three, the number of columns are four and the number of levels are two.

The prompt sequence that will follow when you choose this command from the **Modify** menu is:

Select objects: Select the object to array.

Select objects: «

Enter the type of array [Rectangular/Polar] <R>: «

Enter the number of rows (---) <1>: Specify the number of rows along the X axis.

Enter the number of columns (| |) <1>: Specify the number of columns along the Y axis.

Enter the number of levels (...) <1>: Specify the number of levels along the Z axis.

Specify the distance between rows (---): Specify the distance between the rows.

Specify the distance between columns (| |): Specify the distance between the columns.

Specify the distance between levels (...): Specify the distance between the levels.

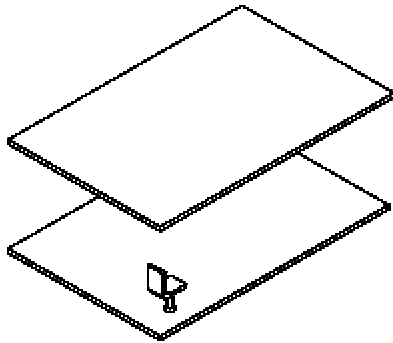


Figure 24-72 Figure showing the model before creating the array

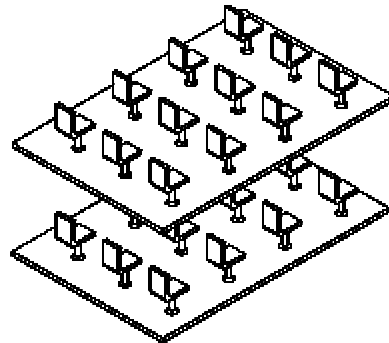


Figure 24-73 Figure showing the model after creating the array

3D Polar Array

The 3D polar arrays are similar to the 2D polar arrays. The only difference is that in 3D you will have to specify an axis about which the solid models will be arranged. For example, consider the solid models shown in Figure 24-74. This figure shows a circular plate that has four holes.

Now, to place the bolts in this plate you can use the 3D polar array as shown in Figure 24-75. The axis for array is defined using the centers at the top and the bottom faces of the circular plate. The prompt sequence that will follow when you invoke this command is as follows:

Select objects: Select the object to array.

Select objects: «

Enter the type of array [Rectangular/Polar] <R>: **P**

Enter the number of items in the array: Specify the number of items.

Specify the angle to fill (+ = ccw, -= cw) <360>: «

Rotate arrayed objects? [Yes/No] <Y>: «

Specify center point of array: Specify the first point on the axis.

Specify second point on axis of rotation: Specify the second point on the axis.

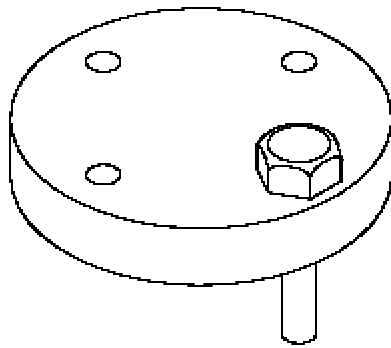


Figure 24-74 Model before creating the array

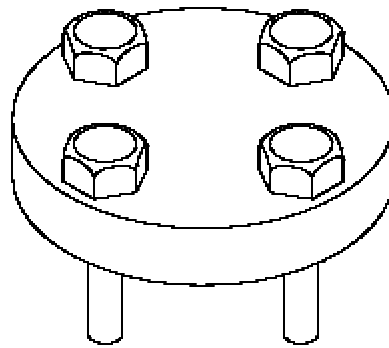


Figure 24-75 Model after creating the array

ALIGNING THE SOLID MODELS (ALIGN COMMAND)

Menu:	Modify > 3D Operations > Align
Command:	ALIGN

The align command is a very versatile and highly effective command. It is extensively used in solid modeling. As the name suggests, this command is used to align the selected solid model with another solid model. In addition to this, it can also be used to translate, rotate, and scale the selected solid model. This command uses pairs of source and destination points to align the solid model. The source point is a point with which you want to align the object. The destination point is a point on the destination object at which you want to place the source object. You can specify one, two, or three pairs of points to align the objects. However, the working of this command will be different for all the three cases. All three cases are discussed individually in the following.

Aligning the Objects Using One Pair of Points

When you align the object using just one pair of points, the working of this command will be similar to the **MOVE** command. That is, the source object will be as it is moved from its original location and will be placed on the destination object. Here, the source point will work as the first point of displacement and the destination point will work as the second point of displacement. A reference line will be drawn between the source and the destination point.

This line will disappear once you exit the command. The prompt sequence is as follows:

Select objects: Select the object to align.

Select objects: «

Specify first source point: Select S1 as shown in Figure 24-76.

Specify first destination point: Select D1 as shown in Figure 24-76.

Specify second source point: «

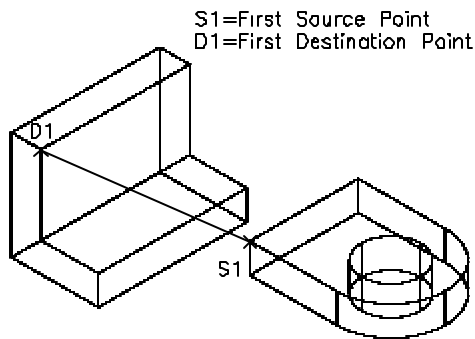


Figure 24-76 Models before aligning

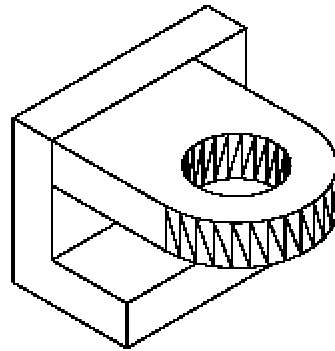


Figure 24-77 Models after aligning

Aligning the Objects Using Two Pairs of Points

The second case is to align the objects using two pairs of source and destination points. This method of aligning the objects forces the selected object to translate and rotate once. You are also provided with an option of scaling the source object to align with the destination object. The prompt sequence that follows is:

Select objects: Select the object to align.

Select objects: «

Specify first source point: Select S1 as shown in Figure 24-78.

Specify first destination point: Select D1 as shown in Figure 24-78.

Specify second source point: Select S2 as shown in Figure 24-78.

Specify second destination point: Select D2 as shown in Figure 24-78.

Specify third source point or <continue>: «

Scale objects based on alignment points? [Yes/No] <N>: **Y**

Aligning the Objects Using Three Pairs of Points

The third case is to align the objects using three pairs of points. This options forces the selected object to rotate twice and then translate. The first pair of source and destination points are used to specify the base point of alignment, the second pair of source and destination points are used to specify the first rotation angle, and the third pair of source and destination points are used to specify the second rotation angle. In this case you will not be allowed to scale the object. The prompt sequence that follows is:

Select objects: Select the object to align.

Select objects: «

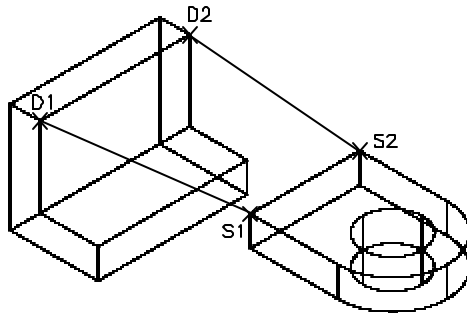


Figure 24-78 Objects before aligning

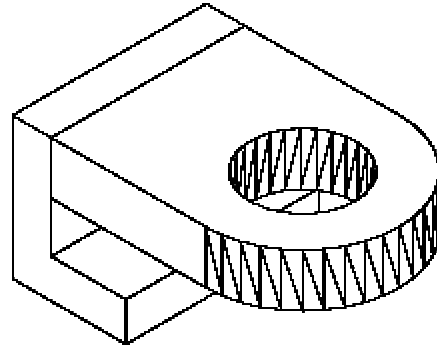


Figure 24-79 Objects after aligning and scaling

Specify first source point: Select S1 as shown in Figure 24-80.

Specify first destination point: Select D1 as shown in Figure 24-80.

Specify second source point: Select S2 as shown in Figure 24-80.

Specify second destination point: Select D2 as shown in Figure 24-80.

Specify third source point or <continue>: Select S3 as shown in Figure 24-80.

Specify third destination point: Select D3 as shown in Figure 24-80.

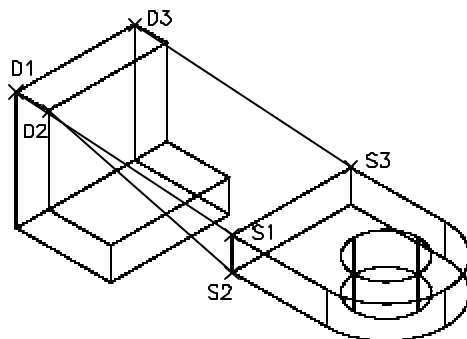


Figure 24-80 Objects before aligning

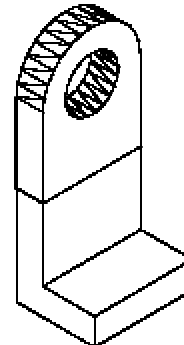


Figure 24-81 Objects after aligning and rotating

SLICING THE SOLIDS (SLICE COMMAND)

Toolbar:	Solid > Slice
Menu:	Draw > Solids > Slice
Command:	SLICE



As the name suggests, this command is used to slice the selected solid with the help of a specified plane. You will be given an option to select the portion of the sliced solid that has to be retained. You can also retain both the portions of the sliced solids. The prompt sequence that follows is:

Select objects: Select the object to slice.

Select objects: «

Specify first point on slicing plane by [Object/Zaxis/View/XY/YZ/ZX/3points] <3points> :

3points

This option is used to slice a solid using a plane defined by three points. The prompt sequence that follows is:

Select objects: Select the object to be sliced.

Select objects: «

Specify first point on slicing plane by [Object/Zaxis/View/XY/YZ/ZX/3points] <3points> :

«

Specify first point on plane: Specify the point P1 on the slicing plane as shown in Figure 24-82.

Specify second point on plane: Specify the point P2 on the slicing plane as shown in Figure 24-82.

Specify third point on plane: Specify the point P3 on the slicing plane as shown in Figure 24-82.

Specify a point on desired side of the plane or [keep Both sides]: Select the portion of the solid to retain.

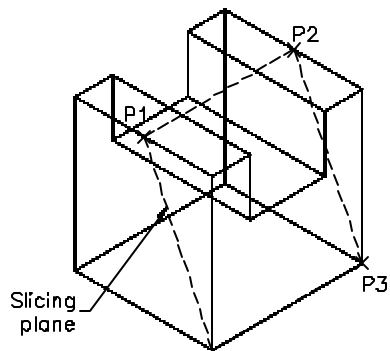


Figure 24-82 Defining the slicing plane

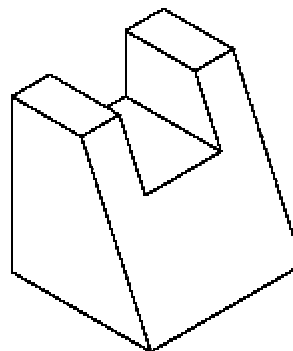


Figure 24-83 Model after slicing

Object

This option is used to slice the solid model using an object. The objects that can be used to slice the solid model include arcs, circles, ellipses, 2D polylines, and splines. The prompt sequence that will follow is:

Select objects: Select the object to slice.

Select objects: «

Specify first point on slicing plane by

[Object/Zaxis/View/XY/YZ/ZX/3points] <3points> : **O**

Select a circle, ellipse, arc, 2D-spline, or 2D-polyline: Select the object to slice the solid.

Specify a point on desired side of the plane or [keep Both sides]: Specify the portion of the solid to retain.

Zaxis

This option is used to slice the solid using a plane defined by two points. The first point is the point on the section plane and the second point is the point in the direction of the Z axis of the

plane. The prompt sequence that follows is:

Select objects: Select the solid to be sliced.
 Select objects: «
 Specify first point on slicing plane by
 [Object/Zaxis/View/XY/YZ/ZX/3points] <3points>: **Z**
 Specify a point on the section plane: Specify the point on the section plane.
 Specify a point on the Z-axis (normal) of the plane: Specify the point in the direction of the Z
 axis of the section plane.
 Specify a point on desired side of the plane or [keep Both sides]: Select the portion to retain.

View

This option is used to slice the selected solid about the viewing plane. The viewing plane in this case will be the screen of the monitor. You will be prompted to specify a point on the solid model through which the viewing plane will pass. The prompt sequence that will follow is:

Select objects: Select the solid model to slice.
 Select objects: «
 Specify first point on slicing plane by [Object/Zaxis/View/XY/YZ/ZX/3points] <3points>: **V**
 Specify a point on the current view plane <0,0,0>: Specify the point P1 as shown in
 Figure 24-84.
 Specify a point on the desired side of the plane or [keep Both sides]: Specify the portion of the
 solid to retain.

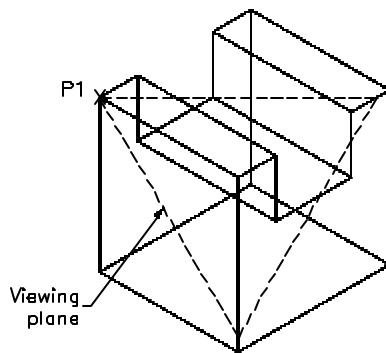


Figure 24-84 Defining the slicing plane

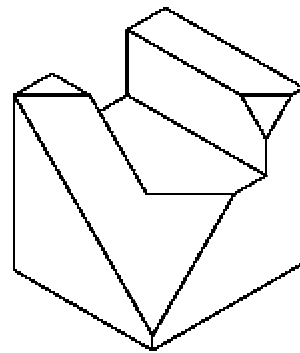


Figure 24-85 Model after slicing

XY, YZ, ZX

These options are used to slice the selected solid about the XY, YZ or the ZX plane respectively. When you invoke this option, you will be prompted to select the point on the plane. The prompt sequence that will follow is:

Select objects: Select the object to be sliced.
 Select objects: «
 Specify first point on slicing plane by [Object/Zaxis/View/XY/YZ/ZX/3points] <3points>:

Select the XY, YZ or the ZX plane.

Specify a point on the XY-plane <0,0,0>: Specify the point on the slicing plane.

Specify a point on desired side of the plane or [keep Both sides]: Specify the portion to retain.

CREATING THE CROSS-SECTIONS OF THE SOLIDS (SECTION COMMAND)

Toolbar:	Solid > Section
Menu:	Draw > Solids > Section
Command:	SECTION



The **Section** command is very similar to the **SLICE** command. The only difference is that this command does not chop the solid. Instead, this command creates a cross section along the selected section plane. The cross-section thus created is a region.

Note that the regions are created in the current layer and not in the layer in which the sectioned solid is stored. The prompt sequence that follows when you choose the **Section** button is:

Select objects: Select the solid to section.

Select objects: «

Specify first point on Section plane by [Object/Zaxis/View/XY/YZ/ZX/3points] <3points> :

3points

This option is used to define the section plane using three points.

Object

This option is used to specify the section plane using a planar object. The objects that can be used to create the sections are arcs, circles, ellipses, splines, or 2D polylines.

View

This option uses the current viewing plane to define the section plane. You will be prompted to specify the point on the current view plane. It will then automatically create a cross-section parallel to the current viewing plane and that passes through the specified point.

XY, YZ, ZX

These option are used to define the section planes that are parallel to the XY, YZ, or the ZX planes respectively. You will be prompted to specify the point through which the selected plane will pass.



Tip

The number of the regions created as cross-sections will be equal to the number of solids selected to create the cross-section.

You can also hatch the cross-section created using the **SECTION** command. Select the entire region as the object for hatching.

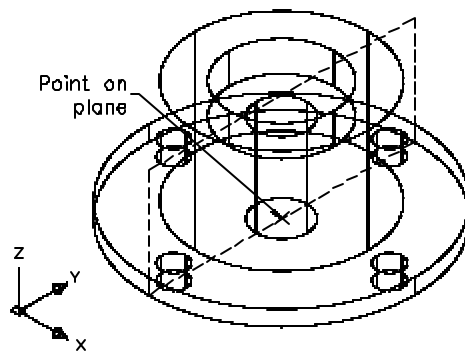


Figure 24-86 Creating the cross-section along the YZ plane

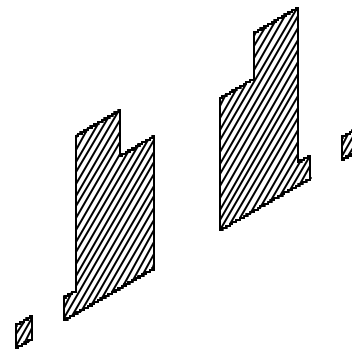


Figure 24-87 Cross-section created, isolated, and hatched for clarity

Example 4

In this example you will draw the solid model shown in Figure 24-88. The dimensions for the model are shown in Figure 24-89 and Figure 24-91. After creating the model, slice it as shown in Figure 24-90.

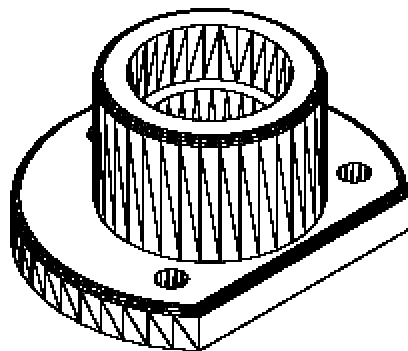


Figure 24-88 Model for Example 4

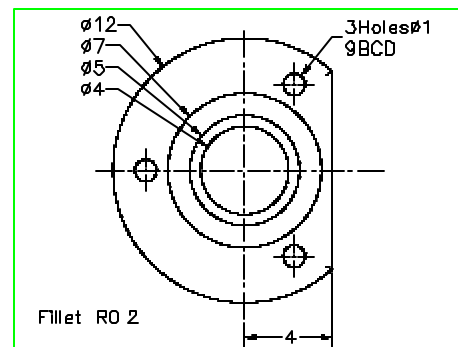


Figure 24-89 Top view of the model

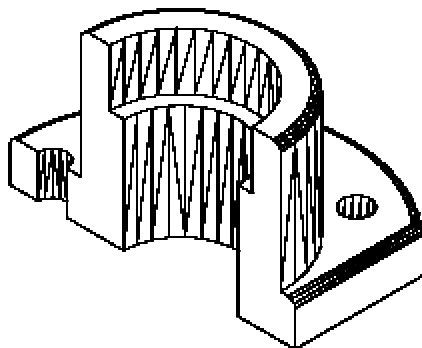


Figure 24-90 Figure showing the sliced solid

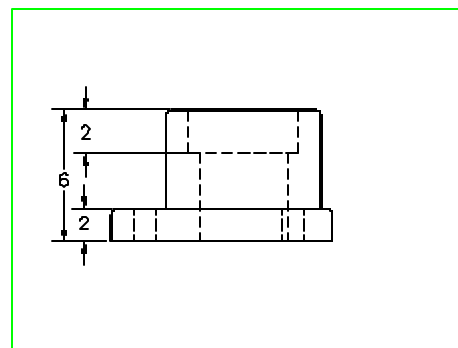


Figure 24-91 Front view of the model

1. Open a new file and then draw the sketch for the base of the model as shown in Figure 24-92. Convert it into a region using the **Region** button in the **Draw** toolbar.
2. Choose the **Extrude** button from the **Solids** toolbar and extrude the region to a distance of 1.5. See Figure 24-93.

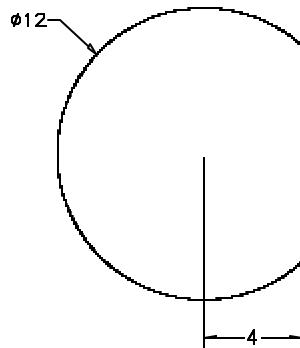


Figure 24-92 Sketch for the base of the model

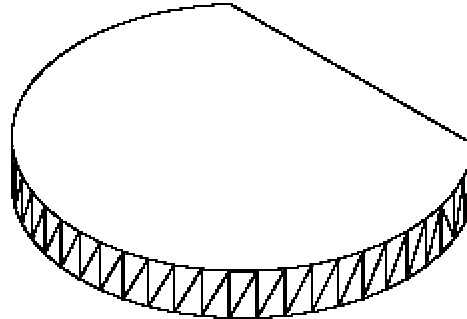


Figure 24-93 Base of the model

3. Rotate the UCS about the X axis through an angle of 90 degrees. Draw the sketch for the next feature and convert into a region, see Figure 24-94.
4. Choose the **Revolve** button from the **Solids** toolbar and then revolve it to an angle of 360 degrees. Move it using the center point of the lower face to the center of the lower face of the base of the model. Choose the **Union** button from the **Solids Editing** toolbar and then union both the objects, see Figure 24-95.

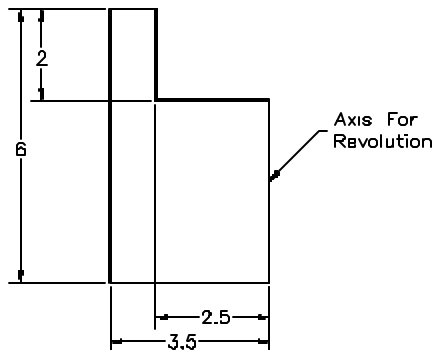


Figure 24-94 Sketch for the next feature

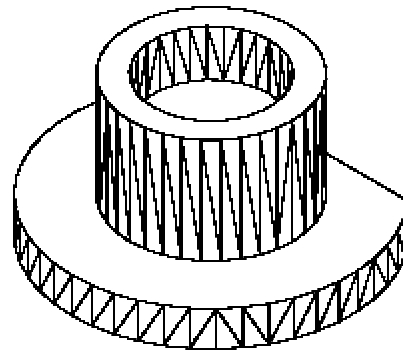


Figure 24-95 Model after union

5. Create a new cylinder of diameter 1 unit. The center of the base of the cylinder should lie at a distance of 1.5 units from the quadrant of the lower face of the base of the model.
6. Choose **3D Operation** > **3D Array** from the **Modify** menu. The prompt sequence is as follows:

Initializing... 3DARRAY loaded.

Select objects: Select the cylinder.

Select objects: «

Enter the type of array [Rectangular/Polar] <R>: **P**

Enter the number of items in the array: **3**

Specify the angle to fill (+ = ccw, -= cw) <360>: «

Rotate arrayed objects? [Yes/No] <Y>: «

Specify center point of array: Select the center of the base of the model.

Specify second point on axis of rotation: Select the center of the top face of the model.

7. Subtract all three cylinders from the model using the **Subtract** button in the **Solids Editing** toolbar.
8. Create the central hole to be subtracted from the cylinder with the diameter 4 units and the height 6 units. Fillet the edges. The model should now look similar to the one shown in Figure 24-96.

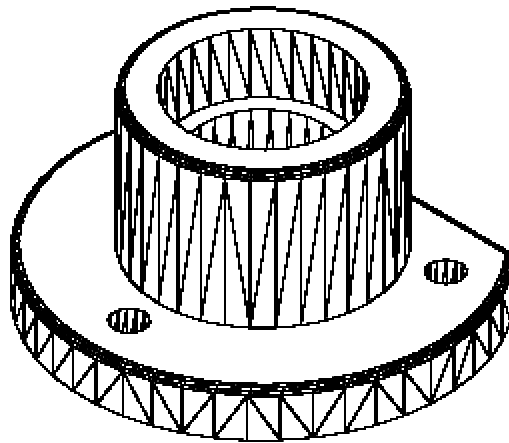


Figure 24-96 Model after creating the fillet

9. Relocate the UCS at the world position.
10. Choose the **Slice** button from the **Solids** toolbar. The prompt sequence is as follows:

Select objects: Select the model.

Select objects: «

Specify first point on slicing plane by [Object/Zaxis/View/XY/YZ/ZX/3points] <3points> : **ZX**

Specify a point on the ZX-plane <0,0,0>: Select the center of the top face of the model.

Specify a point on desired side of the plane or [keep Both sides]: Specify a point on the back side of the model to retain it.
11. The final model for Example 4 should look similar to the one shown in Figure 24-97.

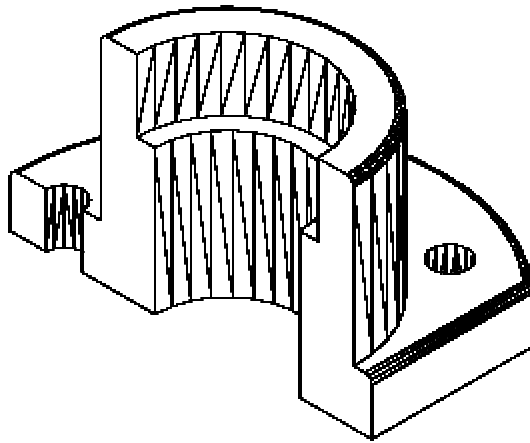


Figure 24-97 Final model after slicing

Self-Evaluation Test

Answer the following questions and then compare your answers to the correct answers given at the end of this chapter.

1. The edges of the solid model can be filleted using the **3DFILLET** command. (T/F)
2. You can select an ellipse as an object to rotate the solid model using the **ROTATE3D** command. (T/F)
3. The **CONE** command can be used to create a solid cone with a circular or elliptical base. (T/F)
4. Open entities can be converted into regions. (T/F)
5. The _____ option of the **CHAMFER** command is used to chamfer all the edges of the selected face of the solid model.
6. The _____ rule is used to determine the direction of rotation of the solid model in the 3D space.
7. The _____ command is used to create a rugby-ball like structure.
8. The _____ command is used to move, rotate, and scale the solid model in a single attempt.
9. The **SECTION** command creates a _____ along the plane of section.
10. You can extrude the selected region about a path using the _____ option of the **EXTRUDE** command.

Review Questions

Answer the following questions.

1. An open entity can be revolved. (T/F)
2. You can not apply Boolean operations on the regions. (T/F)
3. The entity to be revolved should lie completely on one side of the revolution axis. (T/F)
4. You can select all the tangential edges of the selected solid model for filleting using the **Chain** option of the **FILLET** command. (T/F)
5. Which option of the **ROTATE3D** command is used to select a 2D entity as an object for rotating the solid models?
 - (a) **2D**
 - (b) **Last**
 - (c) **Object**
 - (d) **Entity**
6. Which value of the taper angle will taper the extruded model in from the base?
 - (a) Positive
 - (b) Negative
 - (c) Zero
 - (d) None
7. Which command is used to create a cube?
 - (a) **CUBE**
 - (b) **CUBOID**
 - (c) **POLYGON**
 - (d) **CYLINDER**
8. Which command is used to check the interference between the selected solid models?
 - (a) **INTERFERE**
 - (b) **INTERSECT**
 - (c) **INTERFERENCE**
 - (d) **CHECK**
9. Which option of the **MIRROR3D** command is used to mirror the object about the view plane?
 - (a) **Object**
 - (b) **Last**
 - (c) **View**
 - (d) **Zaxis**
10. The _____ option of the **ROTATE3D** command is used to select the same axis that was last selected to rotate the solid model.
11. The _____ direction is known as the extrusion direction.
12. The _____ command is used to create an applelike structure.

13. The _____ command is used to create a revolved solid.
14. In the **3DARRAY** command, the levels are arranged along the _____ axis.
15. The _____ option of the **REVOLVE** command is used to select a 2D entity as the revolution axis.

Exercises

Exercise 2

In this exercise you will create the solid model shown in Figure 24-98. The dimensions for the model are shown in Figure 24-99 and Figure 24-101. After creating the model slice it to get the model shown in Figure 24-100.

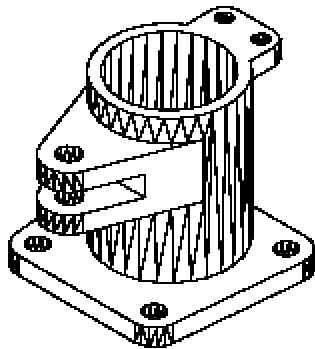


Figure 24-98 Model for Exercise 3

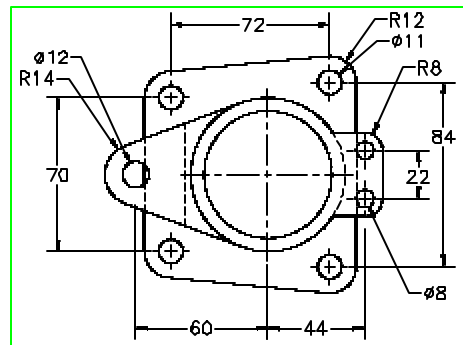


Figure 24-99 Top view of the model

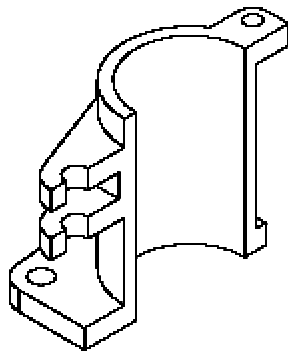


Figure 24-100 Model after slicing

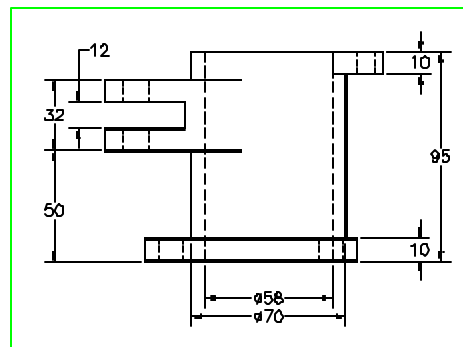


Figure 24-101 Front view of the model

Exercise 3

In this exercise you will create the solid model shown in Figure 24-102. Assume the missing dimensions.

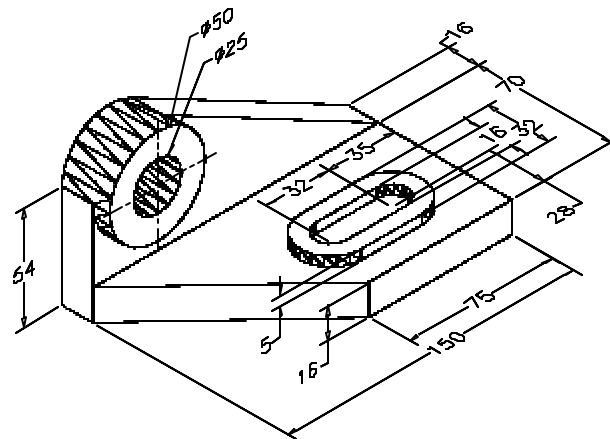


Figure 24-102 Model for Exercise 3

Exercise 4

In this exercise you will create the solid model shown in Figure 24-103. The dimensions for the model are given in the same figure. Assume the missing dimensions.

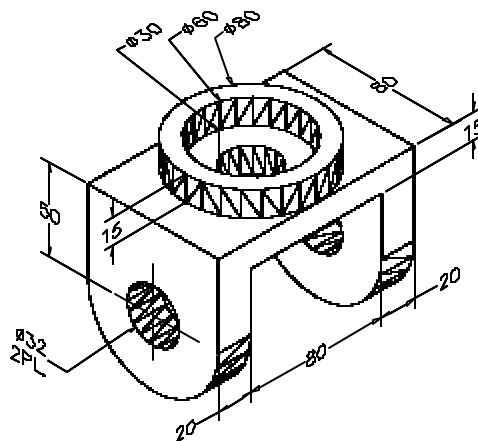


Figure 24-103 Solid model for Exercise 4

Exercise 5

In this exercise you will create the solid model shown in Figure 24-104. The dimensions are given in the drawing. The fillet radius is 5mm.

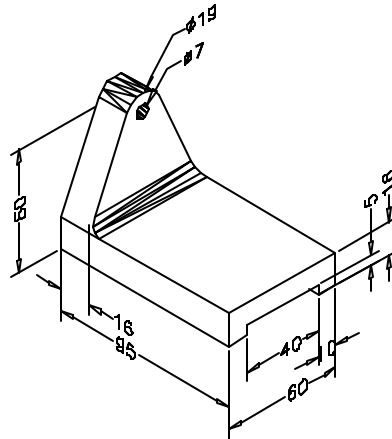


Figure 24-104 Solid model for Exercise 5

Exercise 6

In this exercise you will create the solid model shown in Figure 24-105. The dimensions for the model are given in the same figure. The fillet radius is 0.13 units.

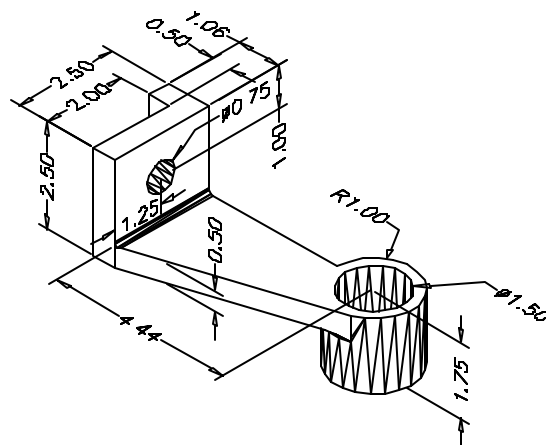


Figure 24-105 Solid model for Exercise 6

Exercise 7

In this exercise you will create the solid model shown in Figure 24-106. The dimensions for the model are given in the same figure. Assume the missing dimensions.

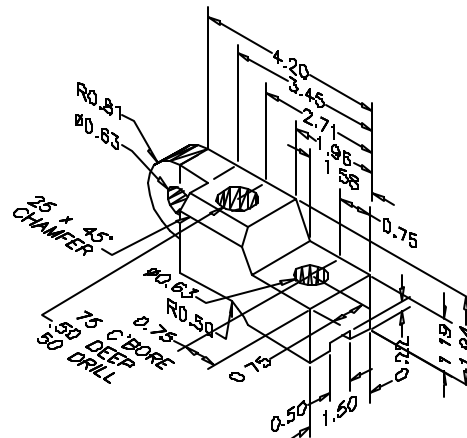


Figure 24-106 Solid model for Exercise 7

Problem Solving Exercise 1

In this exercise you will create the solid model with dimensions shown in Figure 24-107. Next, slice the object, create four viewports and arrange the views as shown in Figure 24-108. Calculate the volume and mass of the part, assuming the part is made of brass.

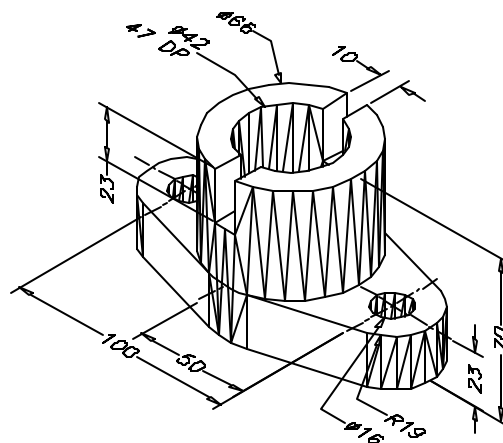


Figure 24-107 Solid model for Problem Solving Exercise 1

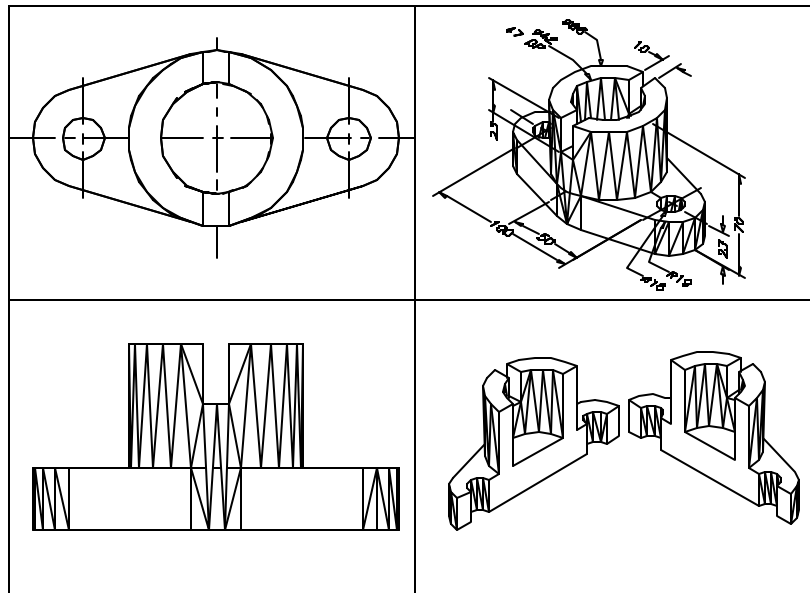


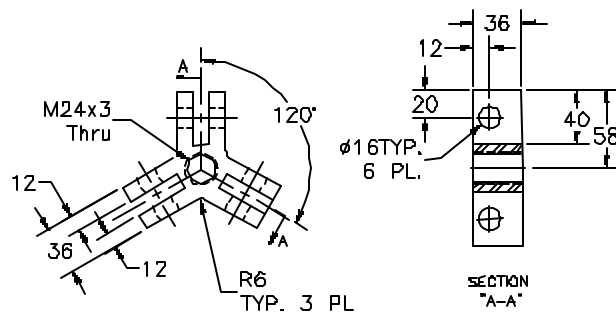
Figure 24-108 Viewports for Problem Solving Exercise 1

Problem Solving Exercise 2

Mechanical

For the Gear Puller shown in Figure 24-115, draw the following:

Piece part drawings shown in Figure 24-109, Figure 24-110, Figure 24-111, Figure 24-112, and Figure 24-113, assembly drawing with Bill of Materials shown in Figure 24-114, solid model of the assembled gear puller shown in Figure 24-115, exploded view of the solid model shown in Figure 24-116.



P1: Yoke

Figure 21-109 Front and side view of Yoke

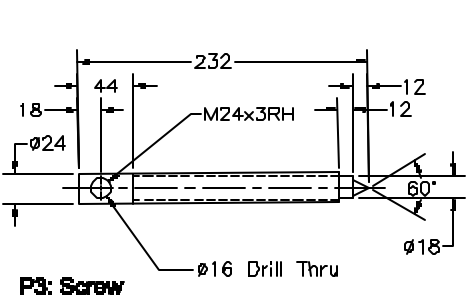


Figure 24-110 Front view of Screw

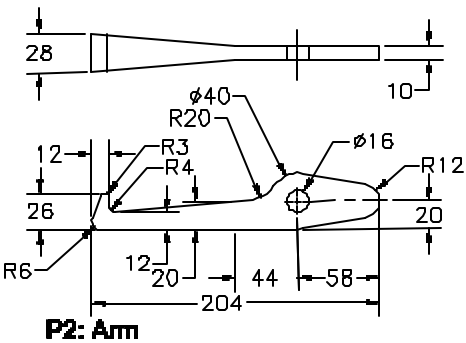


Figure 24-111 Front and top view of Arm

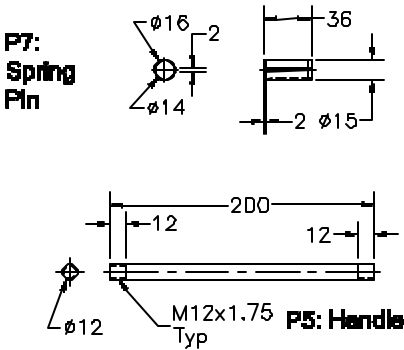


Figure 24-112 Views of Spring Pin and Handle

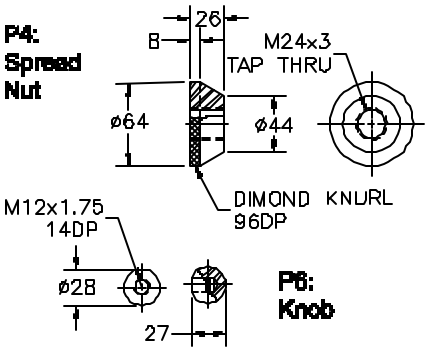


Figure 24-113 Views of Spread Nut and Knob

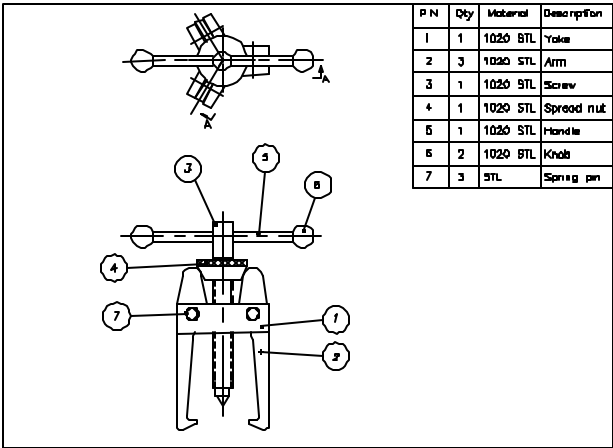


Figure 24-114 Assembled view and BOM of the Gear Puller

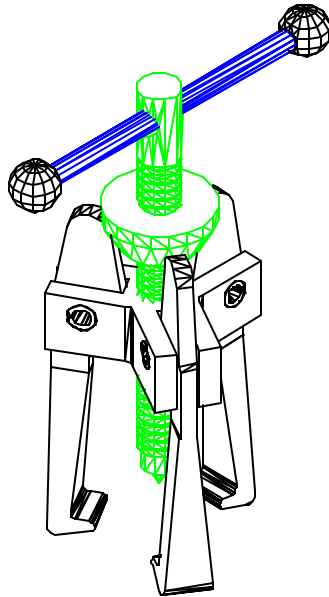


Figure 24-115 Assembled solid model of the Gear Puller

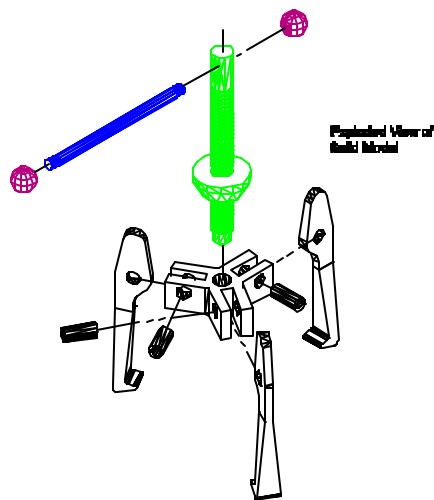


Figure 24-116 Exploded solid model of the Gear Puller

Answers to Self-Evaluation Test

1 - F, 2 - F, 3 - T, 4 - F, 5 - Loop, 6 - right-hand thumb, 7 - TORUS, 8 - Align, 9 - cross-section, 10 - Path