

Chapter 26

Creating Solid Models

CHAPTER OBJECTIVES

In this chapter, you will learn:

- *About solid modeling.*
- *To create standard solid primitives and polysolids.*
- *About regions and create them.*
- *About the use of Boolean operations.*
- *To modify the visual styles of solids.*
- *To define new UCS using the DUCS, ViewCube, and Ribbon.*
- *To use the Extrude and Revolve tools to create solid models.*
- *To use the Sweep and Loft tools to create complex solid models.*
- *To use the Press/Pull tool.*

KEY TERMS

- | | | | |
|--------------------|-----------------|---------------|-------------|
| • Solid Primitives | • Pyramid | • Intersect | • Sweep |
| • Box | • Polysolid | • INTERFERE | • Loft |
| • Cone | • Helix | • 3D Snap | • Solid |
| • Cylinder | • Visual Styles | • Dynamic UCS | • Surface |
| • Sphere | • Region | • Extrude | • Presspull |
| • Torus | • Union | • Mode | |
| • Wedge | • Intersect | • Revolve | |

WHAT IS SOLID MODELING?

Solid modeling is the process of building objects that have all 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, you need more information such as volume, mass, moment of inertia, and material properties (density, Young's modulus, Poisson'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 you always think of and see the objects as solids. With computers getting faster and software getting more sophisticated and affordable, solid modeling has 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.

CREATING PREDEFINED SOLID PRIMITIVES

The Solid primitives form the basic building blocks for a complex solid. ACIS has seven predefined solid primitives that can be used to construct a solid model such as Box, Wedge, Cone, Cylinder, Pyramid, Sphere, and Torus. The number of lines in a solid primitive 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 assigned 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 and perpendicular to the construction plane. Similar to 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 and its value can go up to 10. The tools to create solid primitives are grouped in the **Solid Primitives** drop-down in the **Modeling** panel, as shown in Figure 26-1.

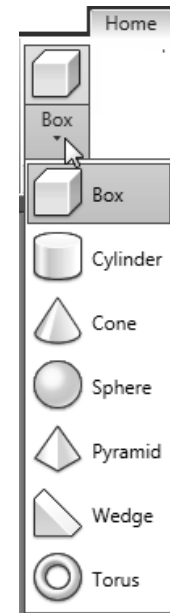


Figure 26-1 Tools in the Solid Primitives drop-down

Creating a Solid Box

Ribbon: Home > Modeling > Solid Primitives drop-down > Box

Command: BOX

Toolbar: Modeling > Box

Menu Bar: Draw > Modeling > Box



The **BOX** command is used to create a solid rectangular box or a cube. Start a new file with the *Acad3d.dwt* template. In 3D drawing templates, you can dynamically preview the operations that you perform. The methods to create a solid box are discussed next.

Creating the Box Dynamically

To create the box dynamically, choose the **Box** tool from the **Modeling** panel. Specify the first corner point of the box and then specify the diagonally opposite corner point for defining the base of the box, as shown in Figure 26-2. Now, specify the height of the box by moving the cursor away from the base. After getting the desired height, click in the drawing window to create the box or specify the height of the box in the dynamic input box, see Figure 26-2.

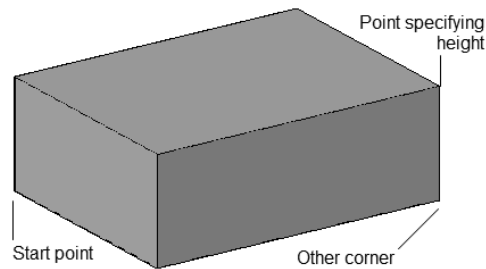


Figure 26-2 Box created dynamically

Two Corner Option

This is the default option. You can use this option to create a solid box by defining the first corner of the box and then its other corner (Figure 26-3). 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 and the value along the *Y* axis will be taken as the width of the box. Next, you will be prompted to specify the height of the box. Given below is the prompt sequence to draw a box of length 5 units and height 4 units.

Specify first corner or [Center] <0,0,0>: *Specify start point.*

Specify other corner or [Cube/Length]: *Drag the cursor in any one of the quadrants and type 5 (Planar face of length 5 units is displayed).*

Specify height or [2Point]: 4

Center-Length Option

The center of the box is the point where the center of gravity of the box lies. This option is used to create a box by specifying the center of the box followed by the length, width, and height of the box (Figure 26-4). The following prompt sequence is displayed when you choose the **Box** button:

Specify first corner or [Center] <0,0,0>: **C**

Specify center <0,0,0>: **4,4**

Specify corner or [Cube/Length]: **L**

Specify length: **8**

Specify width: **6**

Specify height or [2point]: **3**

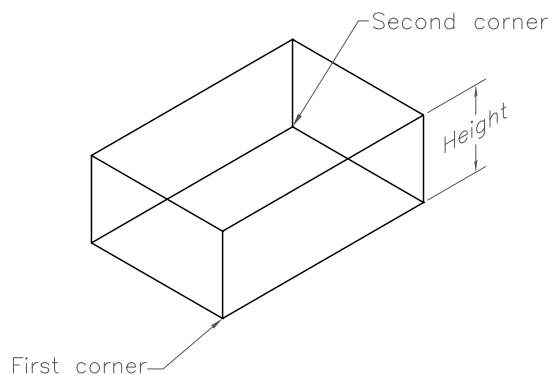


Figure 26-3 Creating a box using the **Two Corner** option

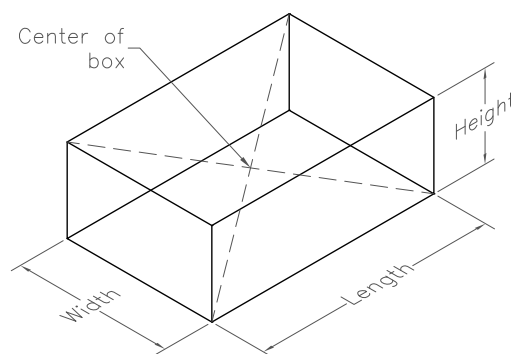


Figure 26-4 Creating a box using the **Center-Length** option

**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 then the length, width, and height of the box. The **2point** option is used to specify the height of the box by specifying 2 points using the pointing device.

Corner-Cube Option

This option is used to create a cube starting from a specified corner, as shown in Figure 26-5. You are creating a cube, and so you will be prompted to enter the length of the cube, that will also act as the width and height. The prompt sequence for creating the cube using the **Corner-Cube** option is given next.

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

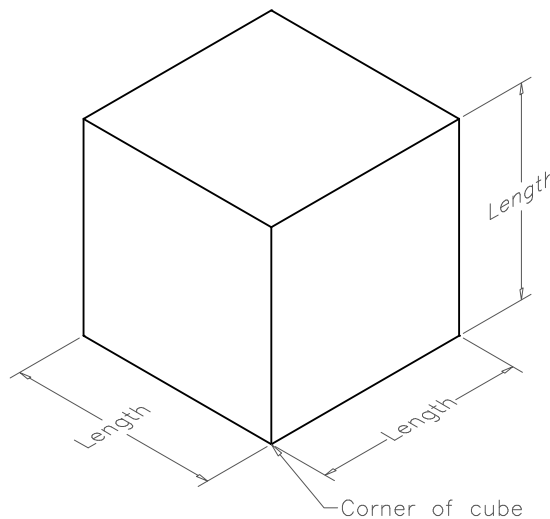


Figure 26-5 Creating a cube using the **Corner-Cube** option

**Tip**

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

Creating a Solid Cone

Ribbon: Home > Modeling > Solid Primitives drop-down > Cone

Command: CONE

Toolbar: Modeling > Cone

Menu Bar: Draw > Modeling > Cone



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

To create a cone, start a new file with the *Acad3d.dwt* template file and choose the **Cone** tool from the **Modeling** panel and specify the center point and the radius of the base, as shown in Figure 26-6. Next, specify the height of cone by moving the cursor away from the base. After getting the desired height, click in the drawing window to create the cone. You can also enter all parameters defining the cone in the dynamic input box. Figure 26-6 shows a cone created dynamically.

Different methods to create a cone discussed next.

Creating a Cone with Circular Base

The options for defining the circular base are **Center Radius**, **Center Diameter**, **3 Point**, **2 Point**, and **Tangent Tangent Radius**. The prompt sequence that will be displayed when you choose the **Cone** tool is given next.

Specify center point for base or [3P/2P/Ttr/Elliptical]: *Specify the center of the base.*

The default option to define a cone with circular base is Center Radius. In this option, after specifying the center point, enter the radius value and then the height of the cone. If you need to enter the diameter value, then enter **D** at the **Specify base radius or [Diameter]** prompt and specify the diameter value followed by the height. The prompt sequence for using this option is given below.

Specify center point for base or [3P/2P/Ttr/Elliptical]: *Specify the center of the base.*

Specify base radius or [Diameter] <default>: *Specify the radius or enter **D** to specify the diameter of the cone.*

Specify height or [2Point/Axis endpoint/Top radius]<default>: *Specify the height of the cone or enter an option or press the ENTER key to accept the default value.*

If you want to create a cone by specifying two ends of the circular base, then type **2P** at the **Specify base radius or [Diameter]** prompt and then specify the two ends of the diameter. Similarly, if the base of the cone is tangent to two circles, type **Ttr** at the **Specify base radius or [Diameter]** prompt, select the circles in succession, and then specify the radius.

Creating a Cone with Elliptical Base

To create a cone with elliptical base, select the **Elliptical** option at the **Specify center point for base of cone or [3P/2P/Ttr/Elliptical]** prompt. On selecting this option, the **Specify endpoint of first axis or [Center]** prompt will be displayed. Now, you can create the elliptical base by specifying the endpoints of axis, the location of the minor axis, and then the height. If you know the centerpoint of the ellipse, enter **C** at the **Specify endpoint of first axis or [Center]** prompt and specify the centerpoint, endpoint of the other two axes in succession, and finally specify the height; a cone with elliptical base will be created, as shown in Figure 26-7.

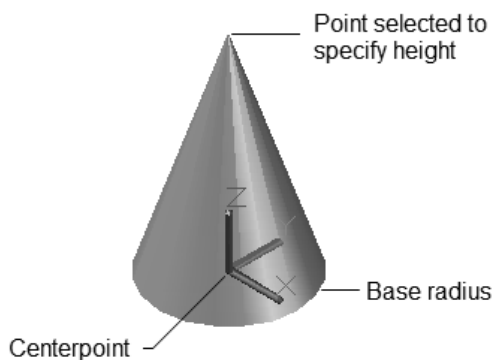


Figure 26-6 Cone created dynamically

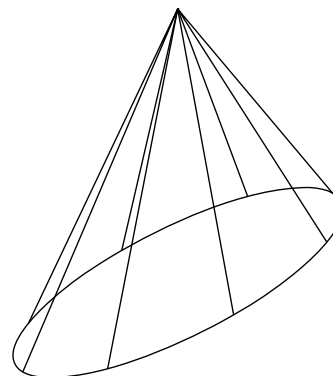


Figure 26-7 An elliptical cone

Creating a Cone with Apex

To create a cone with apex, first specify the base of the cone. Then, specify the height of the cone by choosing an option at the **Specify height or [2Point/Axis endpoint/Top radius]** prompt. By

default, you will be prompted to specify the height of the cone. Enter the height of the cone or specify a location in the drawing area; the cone will be created.

To specify the height using the **2Point** option, enter **2P** at this prompt and specify two points in the drawing area. The distance between these two specified points will be considered as the height of the cone. The prompt sequence that will follow is given next.

Specify height or [2Point/Axis endpoint/Top radius] <current>: **2p**

Specify first point: *Specify the first point.*

Specify second point: *Specify the second point on the screen.*

To specify the height using the **Axis endpoint** option, select the **Axis endpoint** option at the **Specify height or [2Point/Axis endpoint/Top radius]** prompt. On selecting this option, you can specify the endpoint of the axis, whose start point is the center of the base. Apart from fixing the height of the cone, this option is also used to specify the orientation of the cone in 3D space. This means you can also create an inclined cone, see Figure 26-8. The prompt sequence is as follows:

Specify axis endpoint: *Specify the location of the apex or axis endpoint of the right circular cone in 3D space.*

Creating a Frustum of a Cone

To create a frustum of a cone, select the **Top radius** option at the **Specify height or [2Point/Axis endpoint/Top radius]** prompt; you will be prompted to specify the top radius. Specify the radius of the top of the frustum as the top radius; you will be prompted to specify the height. Specify the height as discussed earlier; a frustum of a cone will be created, see Figure 26-9. The prompt to create a frustum of a cone is given next.

Specify top radius <default>: *Specify the radius for the top of the cone frustum.*

Specify height or [2Point/Axis endpoint] <default>: *Specify the height of the frustum or choose an option.*

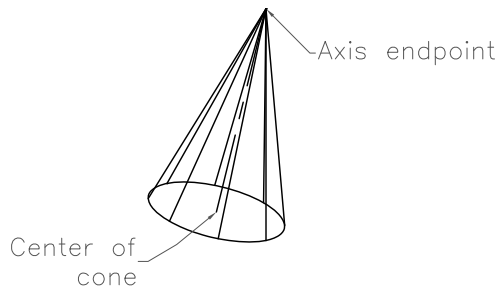


Figure 26-8 Creating a cone using the **Axis endpoint** option

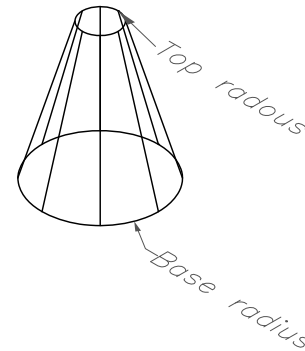


Figure 26-9 Creating a cone using the **Top radius** option



Tip

In AutoCAD 2011, when you select a primitive, grips are displayed on it. You can modify the dimensions of the primitive by dragging these grips.

Creating a Solid Cylinder

Ribbon: Home > Modeling > Solid Primitives drop-down > Cylinder **Command:** CYLINDER
Menu Bar: Draw > Modeling > Cylinder **Toolbar:** Modeling > Cylinder



To create a cylinder, start a new file with the *Acad3d.dwt* template file and choose the **Cylinder** tool from the **Modeling** panel. Next, specify the center point of the base and then specify the radius of the base, as shown in Figure 26-10. Now, specify the height of the cylinder by moving the cursor away from the base. After getting the desired height, click in the drawing window to create the cylinder. You can also enter all parameters in the dynamic input box to define the cylinder.

Similar to the **Cone** tool, this tool provides you with two options for creating the cylinder: **circular cylinder** and **elliptical cylinder** (Figures 26-10 and 26-11). This tool also allows you to define the height of a cylinder dynamically or specify the height by using the **2Point** or **Axis endpoint** options. On selecting the **Axis endpoint** option, you can specify the endpoint of the axis, whose start point is the center of the base. Apart from fixing the height of the cylinder, this option is also used to specify the orientation of the cylinder in 3D space. This means you can also create an inclined cylinder, as shown in Figures 26-12 and 26-13.

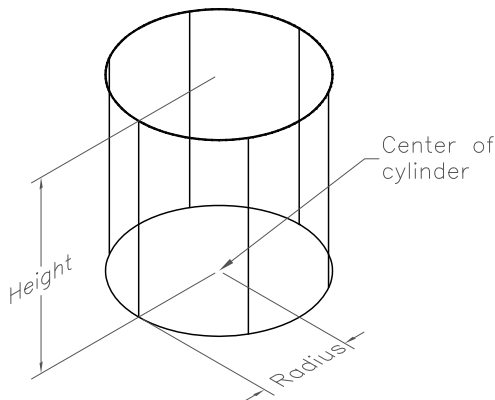


Figure 26-10 Creating a circular cylinder

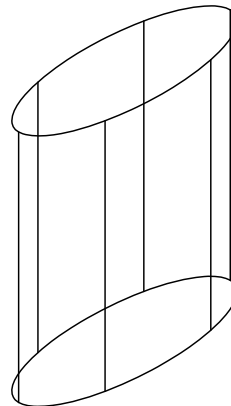


Figure 26-11 Creating an elliptical cylinder

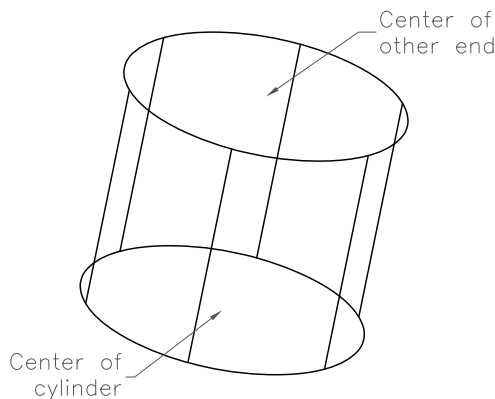


Figure 26-12 Creating an inclined cylinder with circular base

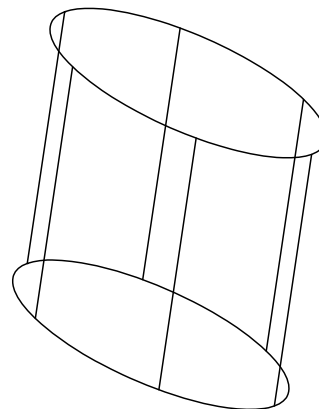


Figure 26-13 Creating an inclined cylinder with elliptical base

Creating a Solid Sphere

Ribbon: Home > Modeling > Solid Primitives drop-down > Sphere

Command: SPHERE

Menu Bar: Draw > Modeling > Sphere

Toolbar: Modeling > Sphere



To create a sphere, choose the **Sphere** tool from the **Modeling** panel; you will be prompted to specify the center of the sphere. After specifying the center, you can create the sphere by defining its radius or diameter, as shown in Figure 26-14. Instead of specifying the center of the sphere, you can also specify its circumference by choosing any one of the **3P**, **2P**, **Ttr** options as discussed earlier.

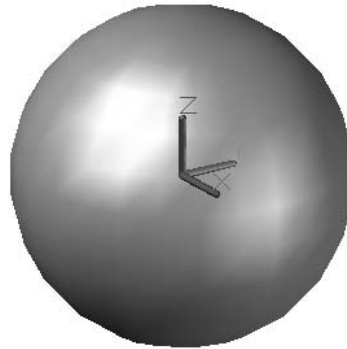


Figure 26-14 Solid sphere created

Creating a Solid Torus

Ribbon: Home > Modeling > Solid Primitives drop-down > Torus

Command: TORUS

Menu Bar: Draw > Modeling > Torus

Toolbar: Modeling > Torus



You can use the **Torus** tool to create a torus (a tyre-tube like shape), as shown in Figure 26-15. A torus is centered on the construction plane. The top half of the torus is above the construction plane and the other half is below it. To create a torus, choose the **Torus** tool from the **Modeling** panel; you will be prompted to enter the diameter or the radius of the torus and the diameter or the radius of tube (Figure 26-16). The radius of torus is the distance from the center of the torus to the center-line of the tube. This radius can have a positive or a negative value. If the value is negative, the torus will have a rugby-ball like shape (Figure 26-17). 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 26-18). Instead of specifying the radius or diameter of the torus, you can also specify the points through which the circumference of the torus will pass by choosing any one of the **3P**, **2P**, and **Ttr** options. These options are discussed earlier.

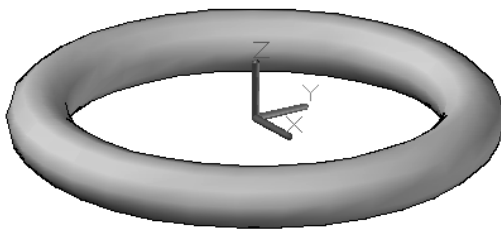


Figure 26-15 Torus created dynamically

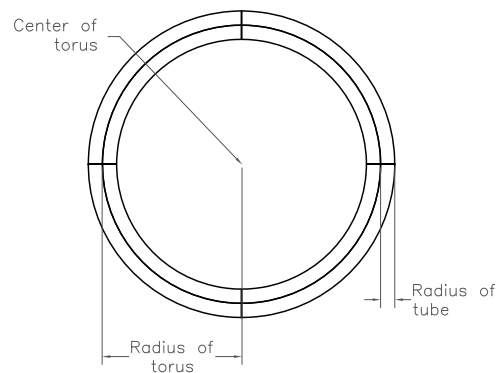


Figure 26-16 Parameters associated with a torus

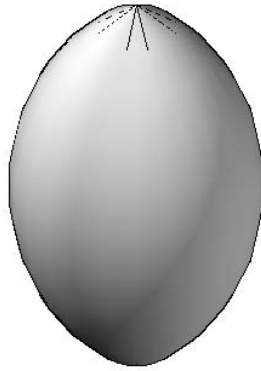


Figure 26-17 Torus with a negative radius value

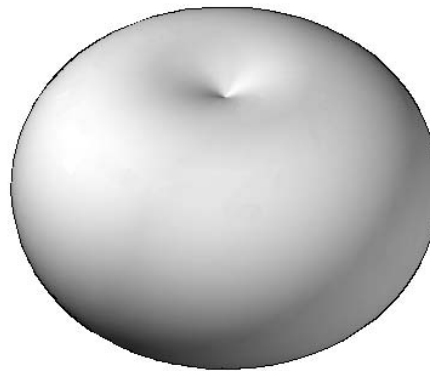


Figure 26-18 Torus with radius of tube more than the radius of torus

Creating a Solid Wedge

Ribbon: Home > Modeling > Solid Primitives drop-down > Wedge

Command: WEDGE

Menu Bar: Draw > Modeling > Wedge

Toolbar: Modeling > Wedge



You can create a solid wedge by using the **Wedge** tool. To create a wedge by using this tool, you need to specify the start point, the diagonally opposite point, and the height of the wedge, as shown in Figure 26-19. The other options in this tool are similar to that of the **Box** tool. Also, the other options to create a box and a wedge are similar.

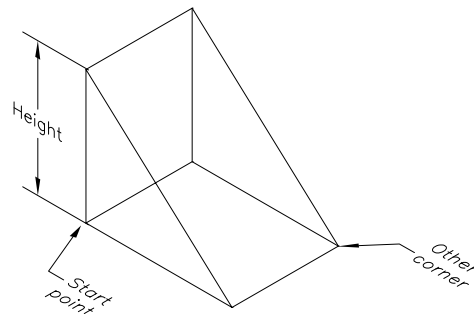


Figure 26-19 Parameters of a wedge

Creating a Pyramid

Ribbon: Home > Modeling > Solid Primitives drop-down > Pyramid

Command: PYRAMID

Menu Bar: Draw > Modeling > Pyramid

Toolbar: Modeling > Pyramid



The **Pyramid** tool is used to create a solid pyramid where all faces, other than the base, are triangular and converge at a point called apex (Figure 26-20). The base of a pyramid can be any polygon, but is typically a square. To create a pyramid, choose the **Pyramid** tool from the **Modeling** panel and follow the prompt sequence given next.

Command: *_pyramid 4 sides circumscribed*

Specify center point of base or [Edge/Sides]: *Specify the center point.*

Specify base radius or [Inscribed] <4>: *Specify the base radius.*

Specify height or [2Point/Axis endpoint/Top radius] <10>: *Specify the height to create the pyramid, refer to Figure 26-20.*

Different methods for creating a solid pyramid are discussed below.

On invoking the **Pyramid** tool, you need to specify the center point of the base polygon or the length of the edges, or the number of sides of the polygon. If you select the **Edge** option in the Command prompt, you will be prompted to pick two points from the drawing area that determine the length and orientation of the edge of the base polygon. If you select the **Sides** option, you will be prompted to specify the number of sides of the base polygon. Note that in AutoCAD, the number of sides of a pyramid can vary from 3 to 32.

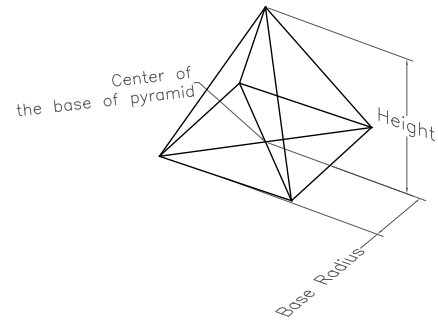


Figure 26-20 Creating a Pyramid dynamically

Next, you need to specify the base radius of the circle in which the base polygon is created. The base polygon is either circumscribed around a circle or inscribed within a circle. By default, the base of the pyramid is circumscribed. After specifying the base radius, you need to specify the height of the pyramid or select an option.

The options to specify the height are same as for cone or cylinder.

You can also create the frustum of a pyramid by selecting the **Top radius** option at the **Specify height or [2Point/Axis endpoint/Top radius]** Command prompt. On selecting this option, you are prompted to specify the radius of the top of the pyramid frustum. Then, you need to specify the height of the frustum either dynamically or by selecting the **2Point** or the **Axis endpoint** option.

Creating a Polysolid

Ribbon: Home > Modeling > Polysolid
Menu Bar: Draw > Modeling > Polysolid

Toolbar: Modeling > Polysolid
Command: POLYSOLID or PSOLID



The **Polysolid** tool is similar to the **Polyline** tool with the only difference that this tool is used to create a solid with a rectangular cross-section of a specified width and height.

This command can also be used to convert existing lines, 2D polylines, arcs, and circles to a polysolid feature. To create a polysolid, choose the **Polysolid** tool from the **Modeling** panel and follow the prompt sequence given next.

Command: `_Polysolid`

Height = current, Width = current, Justification = current

Specify start point or [Object/Height/Width/Justify] <Object>: *Specify the start point for the profile of the polysolid. Else, press the ENTER key to select an object to convert to a polysolid or enter an option.*

The options available at the Command prompt are discussed next.

Next Point of Polysolid

When you specify the start point for the profile of the solid, this option is displayed. This option is used to specify the next point of the current polysolid segment. If additional polysolid segments are added to the first polysolid, AutoCAD automatically makes the endpoint of the previous polysolid, the start point of the next polysolid segment. The prompt sequence is given next.

Command: `_Polysolid`

Height = current, Width = current, Justification = current

Specify start point or [Object/Height/Width/Justify] <Object>: *Specify the start point.*

Specify next point or [Arc/Undo]: *Specify the endpoint of the first polysolid segment.*
 Specify next point or [Arc/Close/Undo]: *Specify the endpoint of the second polysolid segment or press the ENTER key to exit the command.*

Arc. This option is used to switch from drawing linear polysolid segments to drawing curved polysolid segments. The prompt sequence that will follow is given next.

Specify next point or [Arc/Close/Undo]: A
 Specify end point of the arc or [Close/Direction/Line/Second point/Undo]: *Specify the endpoint of the arc or choose an option to create the arc.*

The **Close** option closes the polysolid by creating an arc shaped trajectory from the most recent endpoint to the initial start point and exits the **POLYSOLID** command. Usually, the arc drawn with the **POLYSOLID** command is tangent to the previous polysolid segment. This means that the starting direction of the arc is the ending direction of the previous segment. The **Direction** option allows you to specify the tangent direction of your choice for the arc segment to be drawn. You can specify the direction by specifying a point. The **Line** option switches the command to the **Line** mode. The **Second point** option selects the second point of the arc in the **3P** arc option. The **Undo** option reverses the changes made in the previously drawn polysolid.

Close. This option is available when at least two segments of the polysolid are drawn. It closes the polysolid segment by creating a linear polysolid from the recent endpoint to the initial point.

Undo. This option is used to erase the most recently drawn polysolid segment. You can use this option repeatedly until you reach the start point of the first polysolid segment.

Object

This option is used to convert an existing 2D object into a polysolid. The 2D objects that can be converted into a polysolid include lines, 2D polylines, arcs, and circles. Figure 26-21 shows a polysolid object created by converting a 2D polyline. Note that the polysolid is displayed in the **Realistic** visual style by default, but for clarity, it has been displayed in the **Wireframe** visual style. The prompt sequence for using this option is given next.

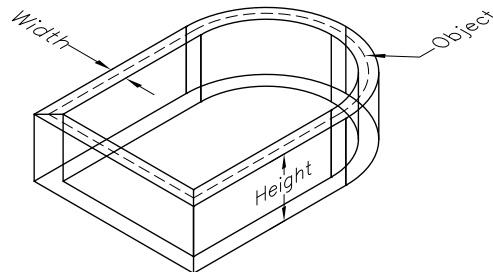


Figure 26-21 Object converted to a polysolid by using the **Object** option

Command: **POLYSOLID**

Height = current, Width = current, Justification = current


Specify start point or [Object/Height/Width/Justify]<Object>:

Select object: *Select an object to convert it into a polysolid.*

Height


This option is used to specify the height of the rectangular cross-section of the polysolid. The

value that you specify will be set as the default value for this command. The prompt sequence for this option is given next.

Command: `_Polysolid`
 Height = current, Width = current, Justification = current
 Specify start point [Object/Height/Width/Justify]<Object>: **H** 
 Specify height <current>: *Specify the value of the height or press the ENTER key to accept the current value.*
 Specify start point [Object/Height/Width/Justify]<Object>: *Specify the start point of the polysolid or choose an option.*

Width

This option is used to specify the width of the rectangular cross-section of a polysolid. The prompt sequence for this option is given next.

Command: `_Polysolid`
 Height = current, Width = current, Justification = current
 Specify start point [Object/Height/Width/Justify]<Object>: **W** 
 Specify width <current>: *Specify the width or press the ENTER key to accept the current value.*
 Specify start point [Object/Height/Width/Justify]<Object>: *Specify the start point of the polysolid or choose an option.*

Justify

The justification option determines the position of the created polysolid with respect to the starting point. The three justifications that are available for a **POLYSOLID** command are **Left**, **Center**, and **Right**.

Left. This justification produces a polysolid with its left extent fixed at the start point and the width added to the right of the start point, when you view the polysolid from the direction of its creation, see Figure 26-22.

Center. This justification produces a polysolid with its center fixed at the start point and the width added equally to both the sides of the start point when you view the polysolid from the direction of its creation, see Figure 26-22.

Right. This justification produces a polysolid with its right extent fixed at the start point and the width added to the left of the start point, when you view the polysolid from the direction of its creation, see Figure 26-22.

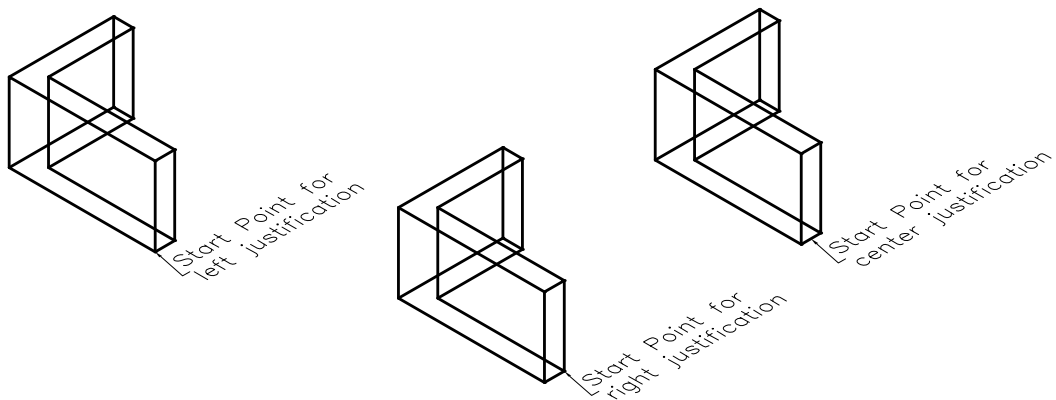


Figure 26-22 Start point for the **Left**, **Right**, and **Center** justifications of a polysolid

Creating a Helix

Ribbon: Home > Draw > Helix
Menu Bar: Draw > Helix

Toolbar: Modeling > Helix
Command: HELIX



The **Helix** tool is used to create a 3D helical curve. To generate a helical curve, you need to first specify the center for the base of the helix. Next, you need to enter the base radius, radius at the top of the helix, and height of the helix. A helix of converging or diverging shape can also be generated by specifying different values for the base and top radius. You can also control the number of turns in a helix and specify whether the twist will be in the clockwise direction or counter clockwise direction. Figure 26-23 shows a helix with the parameters to be defined.

The prompt sequence displayed on choosing the **Helix** tool is given next.

Command: _Helix
 Number of turns = current Twist = CCW
 Specify center point of base: *Specify a point for the center of the helix base.*
 Specify base radius or [Diameter] <1.0000>: *Specify the value for the base radius, enter **D** to specify the diameter, or press the ENTER key to select the default value for the radius.*
 Specify top radius or [Diameter] <current>: *Specify the value for the top radius of the helix, enter **D** to specify the diameter, or press the ENTER key to select the current value of the base radius.*
 Specify helix height or [Axis endpoint/Turns/turn Height/tWist] <default>: *Specify the height of the helix, press the ENTER key to select the default value or choose an option.*

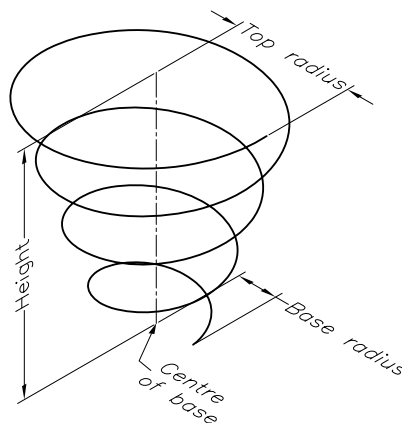


Figure 26-23 Helix with associated parameters

The options available at the command prompt are discussed next.

Axis endpoint

To specify the height using the **Axis endpoint** option, choose the **Axis endpoint** option at the **Specify helix height or [Axis endpoint/Turns/turn Height/tWist]** prompt. Using this option, you can specify the endpoint of the axis, whose start point is assumed to be at the center of the base of the helix. Apart from fixing the height of the helix, this option also specifies the orientation of the helix in 3D space or in other words, you can also create an inclined helix, see Figure 26-24.

Turns

To specify the number of turns in the helix, choose the **Turns** option at the **Specify helix height or [Axis endpoint/Turns/turn Height/tWist]** prompt. Using this option, you can change the

number of turns in the helical curve. The default number of turns for the **HELIX** command is three.

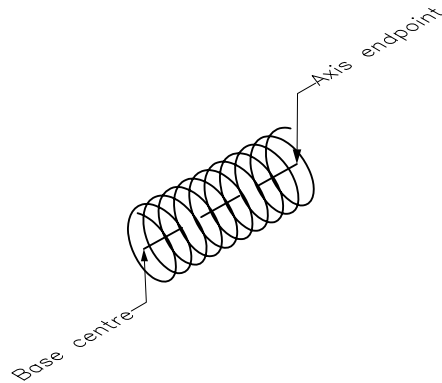


Figure 26-24 Helix created using the **Axis endpoint** option



Note

In a helical curve, the number of turns cannot be more than 500.

turn Height

To specify the pitch of the helix, choose the **turn Height** option at the **Specify helix height or [Axis endpoint/Turns/turn Height/tWist]** prompt. Using this option, instead of specifying the whole height of the helix, you can specify the height of one complete single turn of the helix. The helical curve will be generated with the total height equal to the turn height multiplied by the number of turns.

tWist

To specify the starting direction of the helical curve, choose the **tWist** option from the shortcut menu at the **Specify helix height or [Axis endpoint/Turns/turn Height/tWist]** prompt. By using this option, you can specify whether the curves will be generated in the clockwise or the counterclockwise direction. Figure 26-25 shows a helix with counterclockwise turns and Figure 26-26 shows a helix with clockwise turns. The default value for the starting direction of the helical curve is counterclockwise.

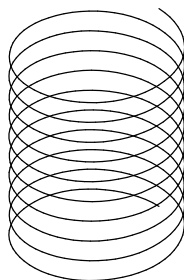


Figure 26-25 Helix created using the **CCW** option

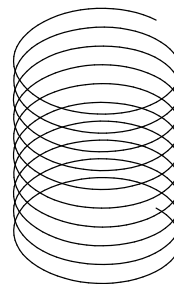


Figure 26-26 Helix created using the **CW** option



Tip

You can use a generated helix as a trajectory for the **SWEEP** command, which will be discussed later in this chapter.

MODIFYING THE VISUAL STYLES OF SOLIDS*

Ribbon: Home > View > Visual Styles > Visual Styles Manager

Command: VISUALSTYLES

Menu Bar: Tools > Palettes > Visual Styles

AutoCAD provides you with various predefined modes of visual styles. These visual styles are grouped together in the **Visual Styles Manager**, see Figure 26-27. To invoke the **Visual Styles Manager**, select the **Visual Styles Manager** option from the **Visual Styles** drop-down list in the **View** panel. Alternatively, click on the inclined arrow at the bottom right of the **Visual Styles** panel in the **View** tab. There are ten visual styles available in the **Available Visual Styles in Drawing** rollout of the **Visual Styles Manager**. To apply a visual style, you need to double-click on it. Alternatively, select the desired visual style from the **Visual Styles** drop-down list in the **View** panel; the model will be update automatically to the new visual style.

The properties of the selected visual style will be displayed below the **Available Visual Styles in Drawing** rollout. You can modify these properties depending upon your requirement. You can also create a user-defined visual style based on the current visual style applied to the model. To do so, choose the **Create New Visual Style** button from the tool strip at the bottom of the **Available Visual Styles in Drawing** rollout; the **Create New Visual Style** dialog box will be displayed. Enter a new name and description for the newly created visual style. Next, choose **OK**; a new visual style will be created. Set the properties of the new visual style by modifying the default properties. Alternatively, set the properties of a visual style and then select **Save as a New Visual Style** from the **Visual Styles** drop-down list on the **View** tab; you will be prompted to specify a name to save the current visual style. Enter a name and press ENTER; the new visual style will be saved.

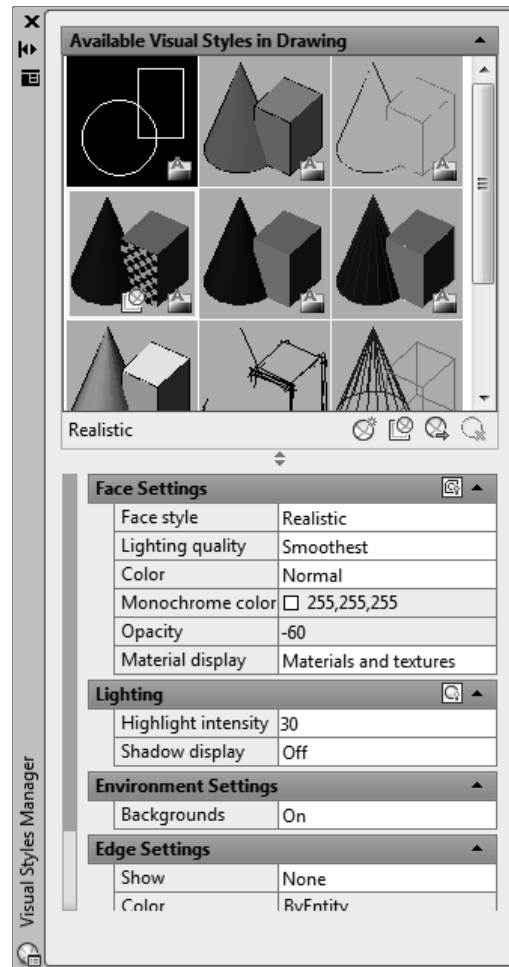


Figure 26-27 The Visual Styles Manager palette

Available Visual Styles in Drawing

The options available in this rollout are discussed next.

2D/3D Wireframe

On applying this visual style, all hidden lines will be displayed along with the visible lines in the model. Sometimes, it becomes difficult to recognize the visible lines and hidden lines, if you set this visual style for complex models.

Conceptual

In this visual style, the model will be displayed as shaded and the edges of the visible faces of the model will also be displayed.

Hidden

On applying this visual style, the hidden lines in the model will not be displayed, and only the edges of the faces that are visible in the current viewport will be displayed.

Realistic

This visual style is the same as **Conceptual**, but with a more realistic appearance. Moreover, if materials are applied to a model, this visual style will display the model along with the materials applied.

Shaded

In this visual style, smooth shading is applied to the faces of a model. In this case, the edges of the visible faces are not visible.

Shaded with Edges

In this visual style, smooth shading is applied to the faces of a model and the edges of the visible faces are visible.

Shades of Grey

In this visual style, a single shade (grey) is applied to all faces of a model.

Sketchy

In this visual style, a model appears as if it is hand sketched.

Wireframe

This visual style is used to display a solid model as a wireframe model. You can see through the solid model while the model is being rotated in the 3D orbit view.

X Ray

This visual style is used to display the model with shading and mild transparency. In this case, the hidden lines will be visible.

2D Wireframe options

The options in this area are discussed next.

Contour Lines

The lines that are displayed on the curved surface of a solid are known as contour lines. The default value for the number of contours to be displayed on each curved surface is 4 and it is controlled by system variable **ISOLINES**. Figure 26-28 shows a cylinder with 4 contour lines and Figure 26-29 shows a cylinder with 8 contour lines.

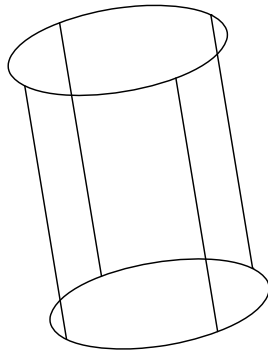


Figure 26-28 Cylinder with 4 contour lines

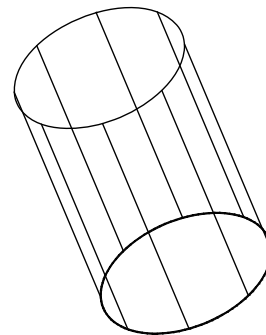


Figure 26-29 Cylinder with 8 contour lines

Draw true silhouettes

This option is used to specify whether or not to display the true silhouette edges of a model when it is hidden by using the **HIDE** command. If you select **Yes** from this drop-down list, the silhouette edges will be displayed when you hide the model, see Figure 26-30. However, when you select **No** from this drop-down list, the silhouette edges will not be displayed when you hide the model, see Figure 26-31.

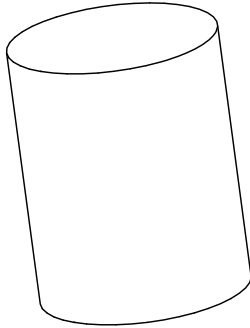


Figure 26-30 Model with the true silhouette edges on

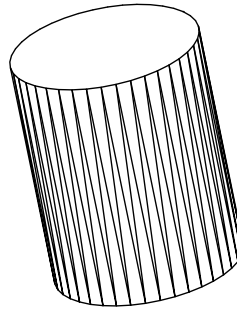


Figure 26-31 Model with the true silhouette edges off

2D Hide - Occluded Lines

The options in this area are used to set the linetype and color of the hidden lines (also called occluded lines).

Linetype

This drop-down list is used to set the linetype for the obscured lines. You can select the desired linetype from this drop-down list. The linetype of the hidden lines can also be modified using the **OBSCUREDLYTYPE** system variable. The default value of this variable is 0. As a result, the hidden lines are suppressed when you invoke the **HIDE** command. You can set any value between 0 and 11 for this system variable. The following table gives the details of the different values of this system variable and the corresponding linetypes that will be assigned to the hidden lines:

Value	Linetype	Sample
0	Off	None
1	Solid	
2	Dashed	
3	Dotted	
4	Short Dash	
5	Medium Dash	
6	Long Dash	
7	Double Short Dash	
8	Double Medium Dash	



9	Double Long Dash	
10	Medium Long Dash	
11	Sparse Dot	

Figure 26-32 shows a model with a hidden linetype changed to dashed lines and Figure 26-33 shows the same model with hidden lines changed to dotted lines.

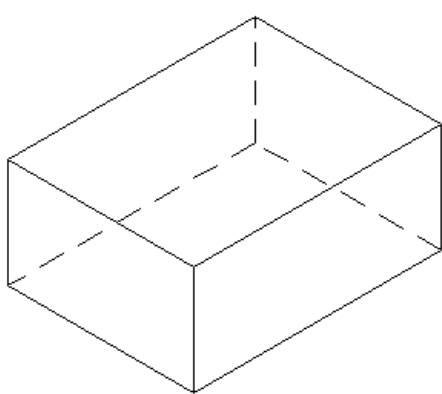


Figure 26-32 Model with hidden lines changed to dashed lines

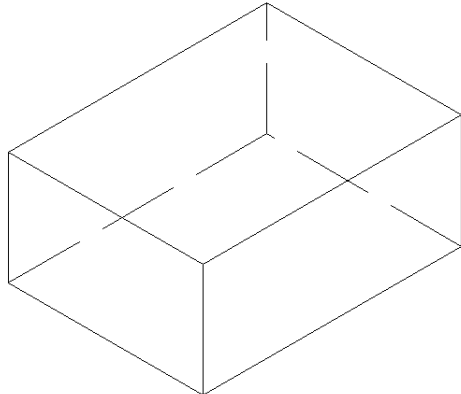


Figure 26-33 Model with hidden lines changed to double long dashed lines

Color

The **Color** drop-down list is used to define the color for the obscured lines. If you define a separate color, the hidden lines will be displayed with that color, when you invoke the **HIDE** command. You can select the required color from this drop-down list. This can also be done using the **OBSCUREDCOLOR** system variable. Using this variable, you can define the color to be assigned to the hidden lines. The default value is 257. This value corresponds to the **ByEntity** color. You can enter the number of any color at the sequence that will follow when you enter this system variable. For example, if you set the value of the **OBSCUREDLTYPE** variable to 2, and that of the **OBSCUREDCOLOR** to 1, the hidden lines will appear in red dashed lines, when you invoke the **HIDE** command.



Note
The linetype and the color for the hidden lines defined using the previously mentioned variables are valid only when you invoke the **HIDE** command. They do not work, if the model is regenerated.

2D - Intersection Edges

The options in this area are used to display a 3D curve at the intersection of two surfaces. Note that the 3D curve is displayed only after invoking the **HIDE** command. These options are discussed next.

Show

If you select **Yes** from the **Visible** drop-down list, a 3D curve will define the intersecting portion of the 3D surfaces or solid models.

Color

The **Color** drop-down list is used to specify the color of the 3D curve that is displayed at the intersection of the 3D surfaces or solid models.

2D Hide - Miscellaneous

This head in the **Visual Styles Manager** is used to set the percentage for halo gap.

Halo gap %

The **Halo gap %** area is used to specify the distance in terms of percentage of unit length by which the edges that are hidden by a surface or solid model will be shortened, see Figures 26-34 and 26-35.

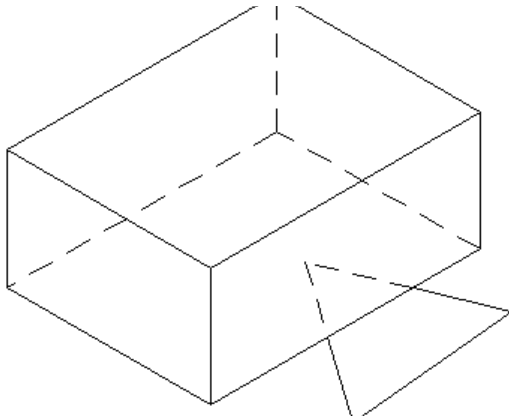


Figure 26-34 Hiding lines with halo gap % as 0

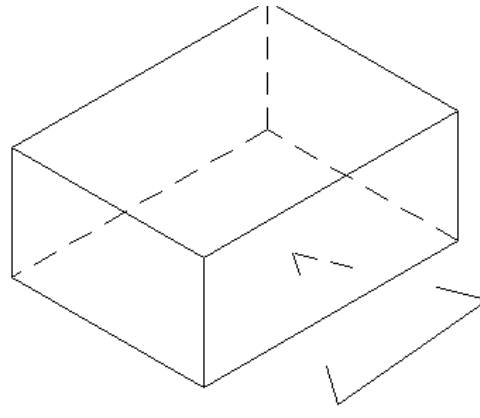


Figure 26-35 Hiding lines with halo gap % as 40



Note

The obscured linetype in Figures 26-34 and 26-35 is changed to dashed.

Similarly, you can specify the settings for faces, edges, and environment required for other visual styles. These settings are discussed next.

Controlling the Settings of Edges

You can control display settings of the edges by using the **Edge Settings** rollout. Figure 26-36 shows different types of edge display. The **Show** field in this rollout has three options: **None**, **Isolines**, and **Facet Edges**. On selecting the **Isolines** option, the contour lines on the surface of the solid will be displayed. Also, the **Always on Top** field will be available in the **Show** field. On selecting the **Yes** option in this field, all isolines in the model will be displayed. If you do not want the edges to be displayed, choose **None** in the **Show** field. On selecting the **Facet Edges** option in the **Show** field, the edges between the planar surfaces will be displayed. You can also set other options in the sub rollout. Note that some of the sub rollouts will not be available for the **Shaded** and **Realistic** visual styles.

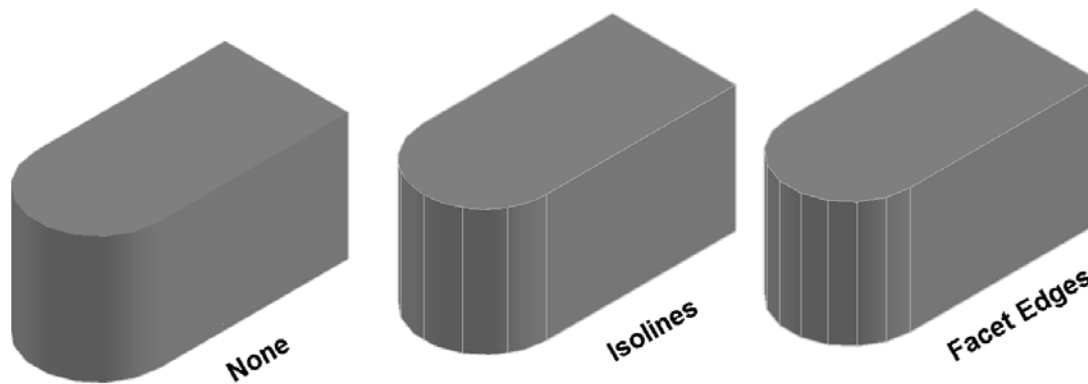


Figure 26-36 Different types of edge display

The **Occluded Edges** sub rollout is used to control the display of hidden edges. Select the **Yes** option in the **Show** filed to display the hidden edges. You can also set the color and the line type of the hidden edges in the **Color** and **Linetype** fields, respectively.

The options in the **Intersection Edges** sub rollout are used to control the display of the resulting curve when two surfaces intersect. Select the **Yes** option in the **Show** filed to display the curve at the intersection. You can also set the color and the line type of the resulting curve in the **Color** and **Linetype** fields, respectively.

The **Silhouette Edges** sub rollout is used to control the display of the silhouette edges. Select the **Yes** option in the **Show** filed to display the silhouette edges. You can set the width of the silhouette edges in the **Width** field. This value ranges from 1 to 25.

The **Edge Modifiers** sub rollout is used to assign special effects such as **Line Extensions** and **Jitter** to edges. Both the effects exhibit a hand-drawn effect with the only difference that the sketches drawn in **Line Extensions** extend beyond the model, whereas in **Jitter**, the edges look as if they were drawn using pencil and has multiple lines. Note that you need to choose the **Line Extension edges** and **Jitter edges** buttons from the title bar of the subrollout to enable these options. You can set the value for the extension line from the vertices by using the **Line Extensions** spinner. Similarly, you can control the effect of jitter by using the options in the **Jitter** drop-down list. Figure 26-37 shows the effect of line extensions and jitter edges. The **Crease angle** field will be displayed only if you choose the **Facets Edges** option in the **Edge Settings** rollout. Note that the Facets edges will be displayed only if the angle between the facets is smaller than the specified crease angle.

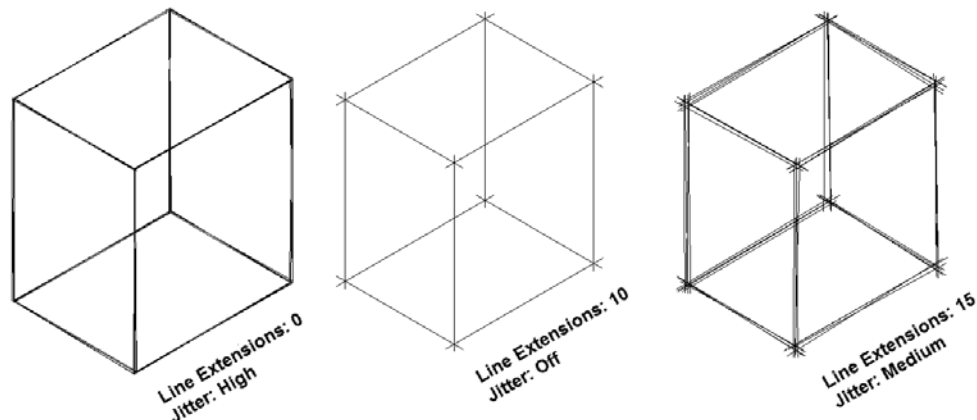


Figure 26-37 The **Line Extensions** and **Jitter** effects of the edges

Controlling the Face Display

AutoCAD enables you to control the display of faces by providing options to apply color effects and shading to solids or surfaces. The shading of models can be controlled by using the **Face style** field. The options in this field are **Realistic**, **Gooch**, and **None**, refer to Figure 26-38. By default, the **Realistic** option is chosen and therefore gives a real world effect to the model. The **Gooch** effect enhances the display by using light colors, instead of dark colors. If the **None** option is chosen, then no style will be applied to the model and only the edges will be displayed.

You can also control the appearance of faces in solids or surfaces using the **Lighting quality** field. The options available in this field are **Faceted**, **Smooth**, and **Smoothest**. On selecting the **Faceted** option, the color is applied to each face, thereby making the object to appear flat. On selecting the **Smooth** option, the color applied is the gradient between the two vertices of the face. This option is selected by default. On selecting the **Smoothest** option, the color is calculated for per pixel lighting.

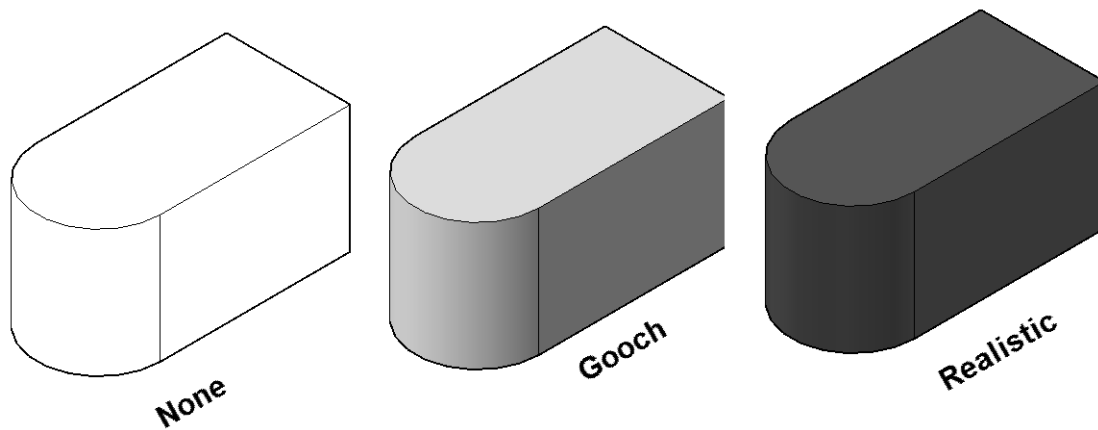


Figure 26-38 Different face styles

You can apply different color settings to faces by selecting an option from the **Color** drop-down list in the **Face Settings** rollout. The options in this drop-down list are **Normal**, **Monochrome**, **Tint**, and **Desaturate**. No face color will be applied if the **Normal** option is selected. If you select the **Monochrome** option, all faces will be shaded with a single color. If you want to use the same color to shade all faces by changing the hue and saturation values of the color, then select the **Tint** option. The hue and saturation values can be changed by using the **Tint** drop-down list. Select the **Desaturate** option to soften the color by reducing the saturation by 30%.

You can also control the opacity of solid models by choosing the **Opacity** button in the title bar of the **Facet Settings** rollout. On doing so, the **Opacity** field will be enabled. Specify the desired transparency level. Figure 26-39 shows the object with and without the transparency set.

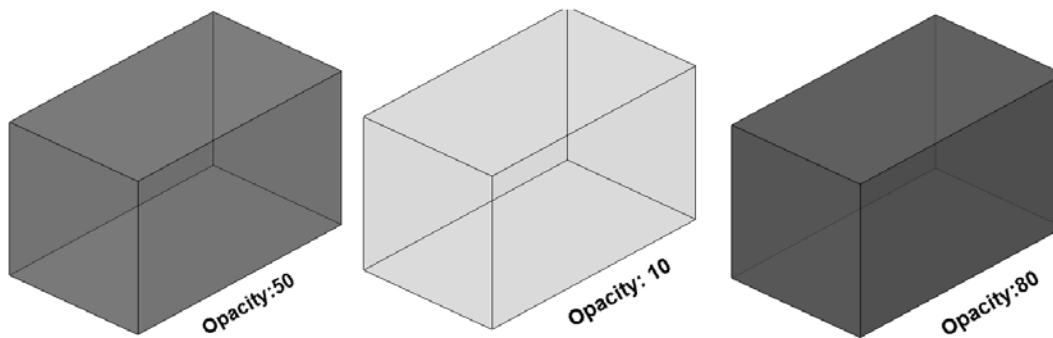


Figure 26-39 Different face transparencies

The shininess of an object is set by using the options in the **Lighting** rollout. To set the shininess, choose the **Highlight Intensity** button on the title bar of this rollout and set the required value in the **Highlight** spinner. Note that you cannot set this value when a material is applied on it.

Shadows provide a real effect at rendering. Shadows are generated when light is applied. Lights can be user-defined lights or the sunlight. The options in this drop-down list are **Off**, **Ground Shadow**, and **Mapped Object shadows**. The Ground Shadows are the shadows that an object casts on the ground. The Mapped Object shadows are the shadows cast by one object on other objects. Shadows appear darker when they overlap. Use of shadows can slow down the system performance. So, you need to turn off the shadows while working, and turn them on when you need shadows. You can set the display of the shadow by using the **Properties** palette. The options under the **Shadow** area of the **Properties** palette are **Casts and Receives Shadows**, **Casts shadows**, **Receives shadows** and **Ignore shadows**. The shadows will be explained in detail in later chapters.

**Note**

The **Mapped Object shadows** cannot be displayed when the **Enhanced 3D Performance** option is off. To turn off this option, enter **3dconfig** at the Command prompt to invoke the **Adaptive Degradation and Performance Tuning** dialog box. Then, choose the **Manual Tune** button; the **Manual Performance Tuning** dialog box will be displayed. Select the **Enable hardware acceleration** check box and click on the **ON** option in the **Value** field of the **Enhanced 3D Performance** and select **OFF** to turn off the effect.

Controlling Backgrounds

You can control the display of background by using the option in the **Environment Settings** rollout. You can turn on or off the background by using the **Backgrounds** drop-down list in this rollout. The background can be a single solid color, a gradient fill, an image, or the sun and the sky. This is explained in detail in Chapter 31 (Rendering and Animating Designs).

**Note**

In AutoCAD 2011, the default visual style applied to a model is **Realistic**. However, in this textbook, the **Wireframe** visual style is applied to models for the clarity of printing.

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.

Creating Regions

Ribbon: Home > Draw > Region
Menu Bar: Draw > Region

Toolbar: Draw > Region
Command: REGION



The **Region** tool is used to create regions from the selected loops or closed entities. Regions are the 2D entities with properties of 3D solids. You can apply the Boolean operation on the regions and you can also calculate their mass properties. 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 lineweight will be applied to the regions.

CREATING COMPLEX SOLID MODELS BY APPLYING 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.

Combining Solid Models

Ribbon: Home > Solid Editing > Solid, Union
Menu Bar: Modify > Solid Editing > Union

Toolbar: Modeling > Union
Command: UNION



The **Solid, Union** tool 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 tool, you will be asked to select the solids or regions to be added. Figure 26-40 shows two solid models before union and Figure 26-41 shows the composite solid created after union.

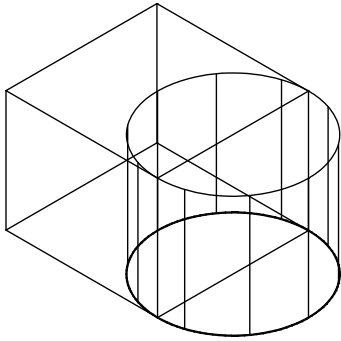


Figure 26-40 Solid models before union

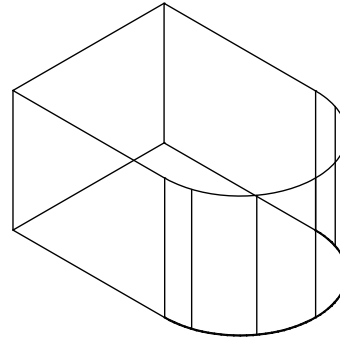


Figure 26-41 Composite solid created after union

Subtracting One Solid From the Other

Ribbon: Home > Solid Editing > Solid, Subtract
Menu Bar: Modify > Solid Editing > Subtract

Toolbar: Modeling > Subtract
Command: SUBTRACT



This tool is used to create a composite solid by removing the material common to the selected set of solids or regions. On invoking this tool, you will be prompted to select the set of solids or regions to subtract from. Once you have selected it, 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 Figures 26-42 and 26-43.

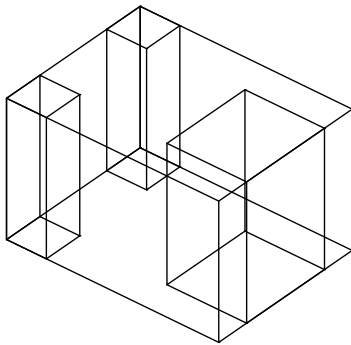


Figure 26-42 Solid models before subtracting

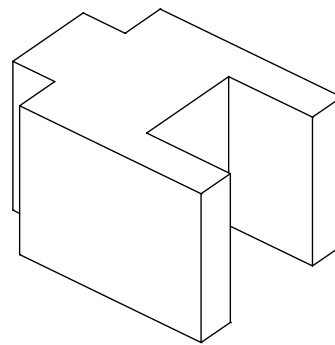


Figure 26-43 Composite solid created after subtracting



Note

If you subtract a single solid from two solids, the two solids join after the subtract operation.

Intersecting Solid Models

Ribbon: Home > Solid Editing > Solid, Intersect
Menu Bar: Modify > Solid Editing > Intersect

Command: INTERSECT
Toolbar: Modeling > intersect



The **Solid, Intersect** tool 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 tool, 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 (Figures 26-44 and 26-45).

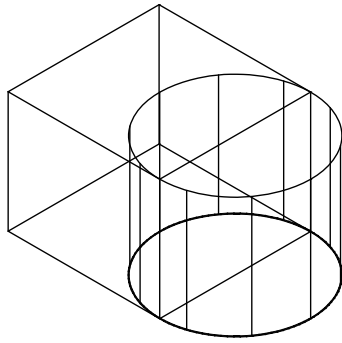


Figure 26-44 Solid models before intersecting

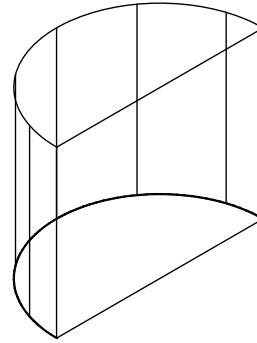


Figure 26-45 Solid model created after intersecting

Checking Interference in Solids

Ribbon: Home > Solid Editing > Interfere

Command: INTERFERE

Menu Bar: Modify > 3d Operations > Interference Checking



The **Interfere** tool is used to create a composite solid model by retaining the material common to the selected sets of solids. This tool is generally used for analyzing interference between the mating parts of an assembly. On invoking this tool, you will be prompted to select the first set of objects. Once you have selected them, you will be prompted to select the second set of objects. Select the second set of objects and press ENTER; the **Interference Checking** dialog box will be displayed. Also, the interference will be highlighted in the drawing area. The **Interfering Objects** area in this dialog box displays the number of objects selected in both the first and second sets and the number of interferences found between them. Choose the **Previous** or the **Next** button in the **Highlight** area to view the previous or next interferences. If the **Zoom to Pair** check box is selected in this area, the interference is zoomed while cycling through the interference objects. You can also use the navigation tools in the **Interference Checking** dialog box to view the interfering area clearly. Clear the **Delete interference objects created on close** check box, if you need to retain the interference objects after closing the dialog box. Close this dialog box by choosing the **Close** button.

If you clear the **Delete interference objects created on close** check box and then close the **Interference Checking** dialog box, you can move the interfering objects by using the **Move** tool. Figure 26-46 shows two mating components of an assembly with interference between them and Figure 26-47 shows the interference solid created and moved out.

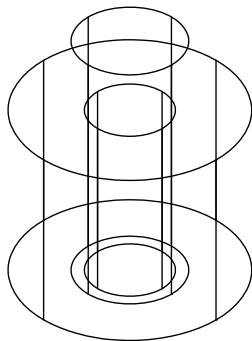


Figure 26-46 Two mating components with interference

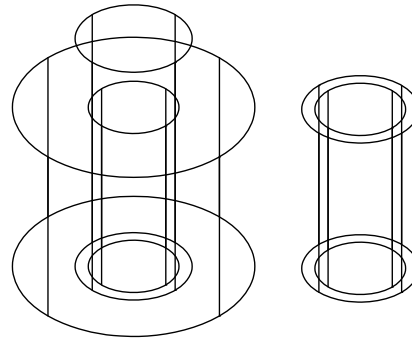


Figure 26-47 Interference solid created using the **INTERFERE** command

You can also specify the visual style and color for the interfering objects. To do so, invoke the **Interfere** tool, type **S** at the **Select first set of Objects or [Nested selection/Settings]** prompt and press ENTER; the **Interference Settings** dialog box will be displayed. In this dialog box, specify the required parameters.

EXAMPLE 1

In this example, you will create the solid model shown in Figure 26-48.

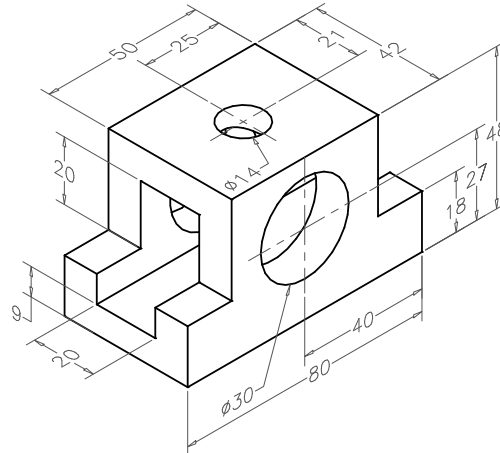


Figure 26-48 Solid model for Example 1

1. Increase the limits to 100,100. Zoom to the limits of the drawing.
2. Enter **UCSICON** at the Command prompt. The following prompt sequence is displayed:
Enter an option [ON/OFF/All/Noorigin/ORigin/Properties] <ON>: N
3. Right-click on the View Cube and choose the **Parallel** option.
4. Invoke the ortho mode, if it is not on.
5. Select **Wireframe** in **Home > View > Visual Styles** drop-down list to set the visual style as Wireframe.
6. Choose the **Box** tool from **Home > Modeling > Solid Primitives** drop-down and follow the prompt sequence given next.

Specify first corner or [Center] <0,0,0>: **10,10**
Specify other corner or [Cube/Length]: **L**
Specify length: (Move the cursor along Y axis) **80**
Specify width: **42**
Specify height: **48**

7. Again, invoke the **Box** tool and follow the prompt sequence given below.

Specify first corner or [Center] <0,0,0>: *Specify a point on the screen.*
Specify other corner or [Cube/Length]: **L**
Specify length: (Move the cursor along X axis) **42**
Specify width: **15**
Specify height: **30**

8. Right-click on the **3D Object Snap** button in the Status Bar and select the **Midpoint on edge** option. Clear the other options in this shortcut menu.
9. Turn **3D Object Snap** on by choosing it, if it is not already on
10. Move the new box inside the old box by snapping the midpoint on edge of both the boxes.
11. By using the **Copy** tool, and the **Vertex** option of the **3D Object Snap**, copy the new box to the other side of the box, refer to Figure 26-49.
12. Choose the **Solid, Subtract** tool from the **Solid Editing** panel and follow the prompt sequence given next.

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 boxes.*

Select objects: *Select other smaller boxes.*

Select objects:

13. Change the visual style to Hidden. The model after subtracting the boxes should look similar to the one shown in Figure 26-50.

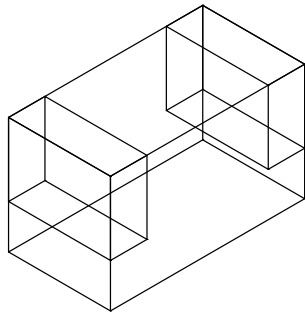


Figure 26-49 Boxes moved inside the bigger box using midpoints

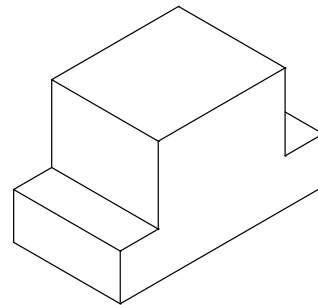


Figure 26-50 Model after subtracting the smaller boxes

14. Again, change the visual style to **Wireframe**. Then, choose the **Box** tool and follow the prompt sequence given next.

Specify first corner or [Center] <0,0,0>: *Specify a point on the screen.*
 Specify other corner or [Cube/Length]: **L**
 Specify length: *(Move the cursor along the X axis)* **20**
 Specify width: **80**
 Specify height: **29**
15. Invoke the **Move** tool and select the box created in the previous step.
16. Specify the midpoint of the lower edge as the base point; the **Specify second point or <use first point as displacement>** prompt is displayed.
17. Enter **From** and press ENTER; you are prompted to specify the base point.

18. Specify the midpoint of the lower edge of the solid created by using the **Subtract** tool and move the cursor vertically upward.
19. Enter **9** at the Command prompt and press ENTER; the box is moved, as shown in Figure 26-51.
20. Choose the **Solid, Subtract** tool from the **Solid Editing** panel and follow the prompt sequence.

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 after changing the visual style to **Hidden** should look similar to the one shown in Figure 26-52.

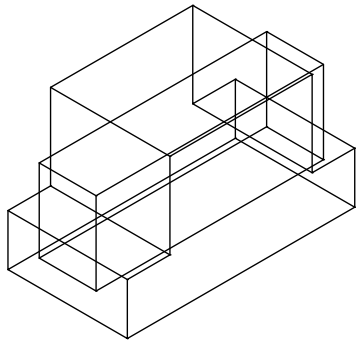


Figure 26-51 Model before subtraction

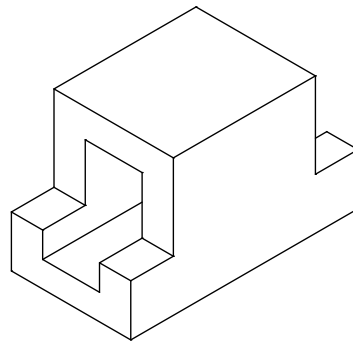


Figure 26-52 Model after subtraction

21. Right-click on the **3D Object Snap** button in the Status Bar and choose the **Center of face** option. Clear the other options in this shortcut menu.
22. Choose the **Cylinder** tool from **Home > Modeling > Solid Primitives** drop-down and follow the command sequence given next to create the cylinder, as shown in Figure 26-53.

Specify center point of base or [3P / 2P / Ttr/ Elliptical]: *Move the cursor on top face of the model and snap the center of the top face.*

Specify base radius or [Diameter]: **7**

Specify height or [2Point / Axis endpoint]: *(Move the cursor vertically downward)* **10**

23. Subtract this cylinder from the existing model, refer to Figure 26-54.
24. Create a new UCS on the front face of the model.
25. Choose **Home > Modeling > Cylinder** from the **Ribbon**. The following command sequence will be displayed:

Specify center point of base or [3P / 2P / Ttr/ Elliptical]: *Use the **From** option to specify the center point of the cylinder at a distance of 27 units from the midpoint of the longer edge of the base.*

Specify base radius or [Diameter]: **15**

Specify height or [2Point / Axis endpoint]: **-42**

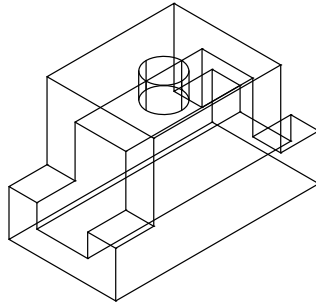


Figure 26-53 Model before subtraction

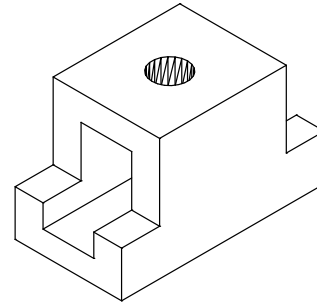


Figure 26-54 Model after subtraction

26. Subtract this cylinder from the model and change the visual style to **Shades of Gray**. The final model should look similar to the one shown in Figure 26-55.

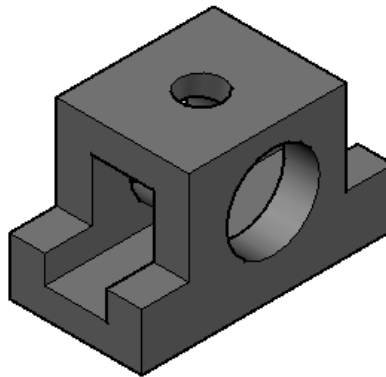


Figure 26-55 Final solid model for Example 1

EXERCISE 1

In this exercise, you will create the solid model shown in Figure 26-56. Save this drawing with the name `|Ch-26|Exercise1.dwg`.

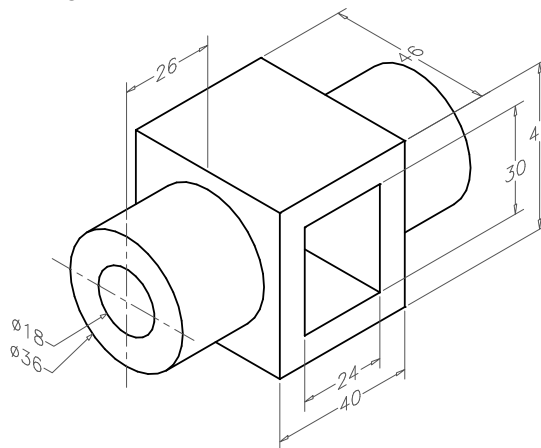


Figure 26-56 Solid model for Exercise 1

DYNAMIC UCS

This option is used to temporarily align the XY plane of the current UCS with the selected face

of an existing solid. Choose the **Allow / Disallow Dynamic UCS** toggle button from the Status Bar to turn it on. Now, invoke a command and move the cursor near the face on which you want to create the new sketch or solid; the face will be highlighted in dashed lines. Click the left mouse button on the desired face; the XY plane of the UCS will automatically be aligned with the selected face. Now, you can create the sketch or solid at the selected face. After the completion of the sketch or feature, the UCS will again move automatically to its original position.

For example, if you want to create a cylinder on the slant face of the model shown in Figure 26-57, choose the **Dynamic UCS** button from the Status Bar to turn it on. Then, choose the **Cylinder** tool from the **Modeling** panel. Next, move the cursor near the slant face of the model; the slant face of the model will be highlighted. Now, left-click on that face and create the cylinder, as shown in Figure 26-58.

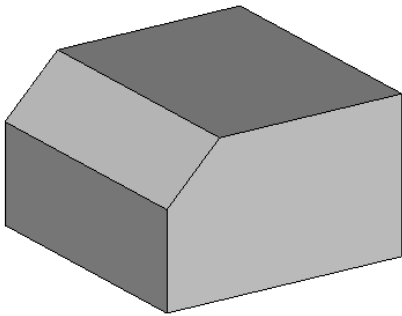


Figure 26-57 Model with UCS at the origin

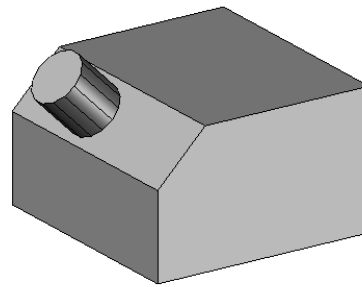


Figure 26-58 UCS dynamically moved on the slant face of the model

If you want a better visualization of how the X, Y, and Z axes of the UCS will be oriented, right-click on the **DUCS** button and choose the **Display crosshair labels** option from the shortcut menu. This will enable you to see the X, Y and Z labels attached with the axes and they will keep on changing as you move the cursor to different faces.



Note

The option of **DUCS** works only when any command is active.



Tip

To toggle between the on and off states of the **DUCS** button, use the F6 function key. To temporarily deactivate the **DUCS** button, press and hold the **SHIFT+Z** keys.

DEFINING THE NEW UCS USING THE ViewCube AND THE RIBBON

In the earlier chapters, you learned how to create new UCS using the **UCS** command. You can also define a new UCS is parallel to the current viewing plane by using the ViewCube and the **Ribbon**. To define a new UCS whose XY plane is parallel to the current viewing plane, click on a hotspot in the ViewCube; the view corresponding to the hotspot will become normal to screen. Next, choose **WCS > New UCS** from the **WCS** flyout in the ViewCube; you will be prompted to specify the origin. Enter **V** at the Command prompt and press **ENTER**; a new UCS will be created at the current viewing plane. Now, if you draw a sketch, it will be in the new UCS. To save the new UCS, choose the **UCS, Named UCS** tool from the **Coordinates** panel in the **View** tab; the **UCS** dialog box will be displayed with the **Unnamed** option. Rename it and choose **OK**; the new name of the UCS will be saved and also displayed in the **WCS** flyout of the ViewCube.

To define a new UCS using the **Ribbon**, choose any one of the predefined orthographic views

from the **Views** drop-down list in the **Views** panel of the **View** tab; the model will orient to that view. Also, you will notice that the ViewCube is also oriented according to the selected view, but **Top** is displayed as the view and **Unnamed** is displayed at the **WCS** flyout. Now, if you draw a sketch, it will be in the new UCS. However, if there are different faces parallel to the new UCS, you can draw sketches on those faces.

CREATING EXTRUDED SOLIDS*

Ribbon: Home > Modeling > Solid Creation drop-down > Extrude Or Solid > Solid > Extrude

Command: EXTRUDE

Menu Bar: Draw > Modeling > Extrude

Toolbar: Modeling > Extrude



Sometimes, the shape of a solid model is such that it cannot be created by just applying the Boolean operations on the standard solid primitives. In such cases, you can use the tools from the **Solid Creation** drop-down, as shown in Figure 26-59. These tools are also available in the **Solid** panel of the **Solid** tab. Using these tools, you can create solid models of any complex shape.

The **Extrude** tool is used to create a complex solid/surface model by extruding a 2D entity or a region along the Z axis or any specified direction or about a specified path. On invoking this tool, following prompt sequence will be displayed.

Command: `_extrude`

Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid

Select objects to extrude or [MOde]: `_MO` Closed profiles creation mode

[Solid/Surface] <Solid>: `_SO`

Select objects to extrude or [MOde]:

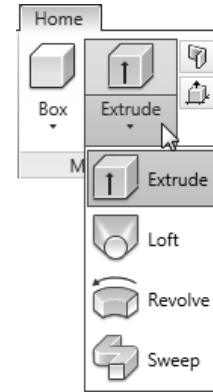


Figure 26-59 Tools in the **Solid Creation** drop-down

In AutoCAD 2011, you can specify mode to extrude an object. There are two modes available: **Solid** and **Surface**. If you need to change the mode, enter **MO** at the **Select objects to extrude or [MOde]** prompt and specify the mode. Note that if the original entity is a closed loop or a region, you can extrude it as a solid/surface model. If the 2D entity is an open loop, then you can extrude it as a surface. After specifying the mode, select the objects to be extruded and press ENTER; the **Specify height of extrusion or [Direction/Path/Taper angle/Expression]** prompt will be displayed. Specify the depth of extrusion or select other options to specify the depth. Different methods to extrude an object by using the options at the Command prompt are discussed next.

Extruding along the Z Axis

This is the default option and is used to create a model by extruding a 2D entity or a region along the Z axis direction. Figure 26-60 shows the region to be converted into an extruded solid and Figure 26-61 shows the solid created upon extruding the region. The prompt sequence that will be displayed when you choose the **Extrude** tool is given next.

Select objects to extrude: *Select the region.*

Select objects to extrude:

Specify height of extrusion or [Direction / Path / Taper angle]: *Specify the height.*

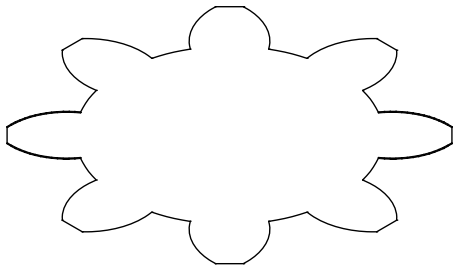


Figure 26-60 Base object for extruding

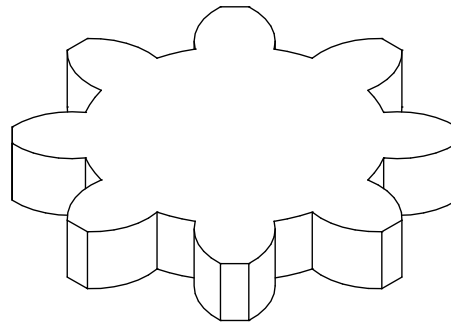


Figure 26-61 Solid created upon extruding

Extruding with a Taper Angle

You can specify the taper angle for an extruded solid/surface by selecting the **Taper angle** option at the **Specify height of extrusion or [Direction/Path/Taper angle]** prompt. 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 26-62).

Extruding along a Direction

This option is used to create a solid by extruding a 2D entity or a region in any desired direction by specifying two points. You can create an inclined extruded object by selecting a start point and an endpoint. The distance between the start point and endpoint acts as the height of the extrusion. Figure 26-63 shows an object extruding along a direction.

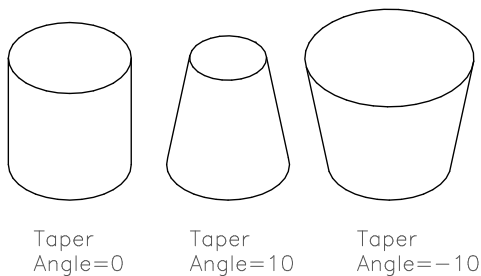


Figure 26-62 Results of various taper angles

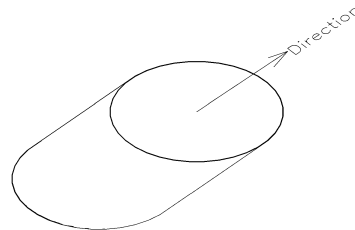


Figure 26-63 Object extruding along a direction

Extruding along a Path

This option is used to extrude a 2D entity or a region about a specified path. Bear in mind that the path of extrusion should be normal to the plane of the base object. If the path consists of more than one entity, all of them should be first joined using the **PEDIT** command so that the path remains a single entity. This option is generally used for creating complex pipelines and also by architects and interior designers for creating beadings. The path used for extrusion can be a closed entity or an open entity and the valid entities that can be used as path are lines, circles, ellipses, polygons, arcs, polylines, or splines. You cannot specify the taper angle when you use a path. Figure 26-64 shows a base object and a path about which it has to be extruded and Figure 26-65 shows the solid created upon extruding the base entity about the specified path.

Extruding using Expressions

This option is used to specify the extrusion depth in terms of formula or equations, as you have created parametric drawings.

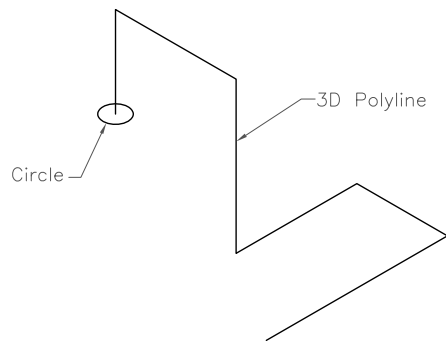


Figure 26-64 The base object and the path for extrusion

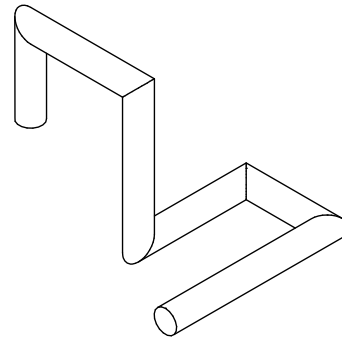


Figure 26-65 Solid model created upon extruding the base entity about the specified path

CREATING REVOLVED SOLIDS*

Ribbon: Home > Modeling > Solid Creation drop-down > Revolve
Or Solid > Solid > Revolve

Command: REVOLVE

Menu Bar: Draw > Modeling > Revolve **Toolbar:** Modify > Revolve



The **Revolve** tool is used to create a complex model by revolving 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 cannot be revolved using this tool. Remember that the direction of revolution is determined using the right-hand thumb rule. The axis of rotation is defined by specifying two points, using the X or Y axis of the current UCS, or using an existing object. Similar to the **Extrude** tool, you can create a solid/surface model by specifying the mode after invoking the **Revolve** tool. On specifying the mode and selecting the object, the **Specify axis start point or define axis by [Object/X/Y/Z]** prompt will be displayed. Specify the axis of revolution. After setting the revolution axis option, you will be prompted to specify the angle of revolution. The default value is 360°. You can enter the required value at this prompt.

The options that are used to specify the axis are discussed next.

Start Point for the Axis of Revolution

This option is used 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 model by revolving a selected 2D entity or a region about a specified object. The valid entities that can be used as an object for defining the axis of revolution can be a 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

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 cannot revolve the object.

Y

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

Z

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

Figure 26-66 shows the entity to be revolved and the revolution axis along with the revolved solid. Figure 26-67 shows the same solid revolved through an angle of 270-degree.

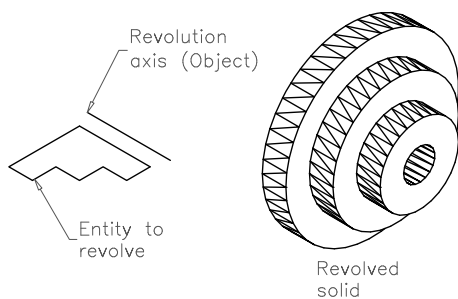


Figure 26-66 Creating a revolved solid

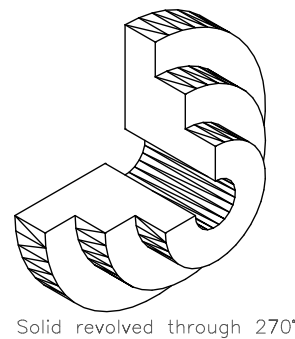


Figure 26-67 Solid revolved to an angle

CREATING SWEEP SOLIDS*

Ribbon: Home > Modeling > Solid Creation drop-down > Sweep
Or Solid > Solid > Sweep/Loft Drop-down > Sweep

Command: SWEEP **Menu Bar:** Draw > Modeling > Sweep

Toolbar: Modeling > Sweep



The **Sweep** tool is used to sweep an open or a closed profile along a path. This tool is also available in the **Solid** panel of the **Solid** tab. To create this model by using this tool, you need an object and a path. The object is a cross-section for the sweep feature and the path is the course taken by the object while creating the swept solid. Figure 26-68 shows the object to be swept and the path to be followed and Figure 26-69 shows the resulting sweep feature.

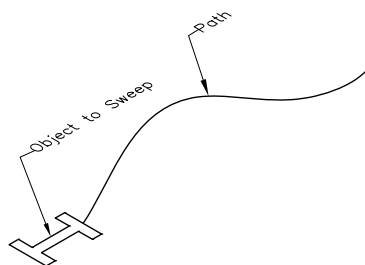


Figure 26-68 Object to be swept and path

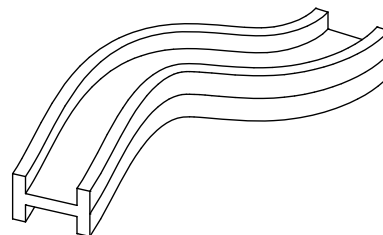


Figure 26-69 Resulting sweep feature

The path specified for the profile and the object for the cross-section of the swept model can be open or closed. If the object for the cross-section is open, or closed but not forming a single region, the generated feature will result in a surface sweep. If the object for the section is closed

and it forms a single region, the generated feature will be a solid. Figure 26-70 shows the object and the closed path and Figure 26-71 shows the resulting sweep feature.

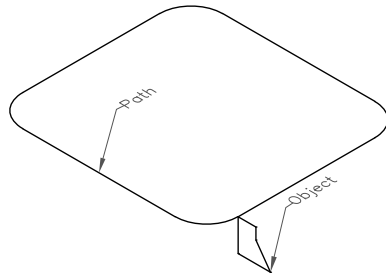


Figure 26-70 Object to be swept with a closed path

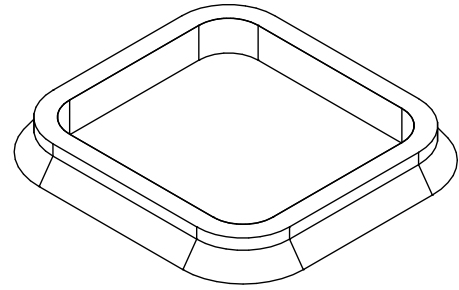


Figure 26-71 Resulting sweep feature



Note

The path selected for the **Sweep** tool should not form a self-intersecting loop.

You can select more than one section at a time to be swept along a path, but all these sections should be drawn on the same plane. To create a sweep draw the cross section profile, path, and select the Sweep tool. Then, follow the prompt sequence given next.

Command: `_sweep`

Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid

Select objects to sweep or [MMode]: *Select the object to be swept.*

Select objects to sweep or [MMode]: *Select more objects to be swept or press ENTER.*

Select sweep path or [Alignment/Base point/Scale/Twist]: *Select the path to be followed by the object or enter an option.*

Depending on your requirement, you can invoke the following options at the **Select sweep path or [Alignment/Base point/Scale/Twist]** prompt:

Alignment

This option is used to specify whether the object will be oriented normal to the curve at the start point. By default, the sweep feature is created with a section normal to the path at the start point. To avoid this, choose **No** from the shortcut menu at the **Align sweep object perpendicular to path before sweep [Yes/No]** prompt. Figure 26-72 shows the object and the path for the sweep command. Figure 26-73 shows the resulting sweep feature with the aligned option **Yes** and Figure 26-74 shows the resulting sweep feature with the aligned option **No**.

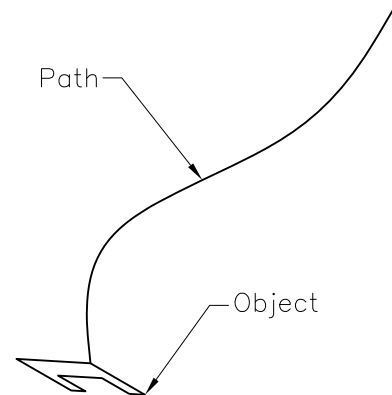


Figure 26-72 Object and path for sweep command

Base Point

This option is used to specify a point on the object that will be attached to the path while sweeping it. You can specify the location of the cross-section while sweeping it when prompted to specify the base point. Figure 26-75 shows the path and the cross-section with base points P1 and P2. Figure 26-76 shows the resulting sweep feature with P1 as the base point and Figure 26-77 shows the resulting sweep feature with P2 as the base point.

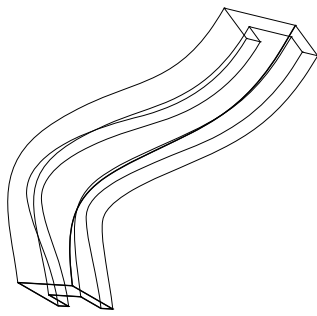


Figure 26-73 Swept solid with the aligned option **Yes**

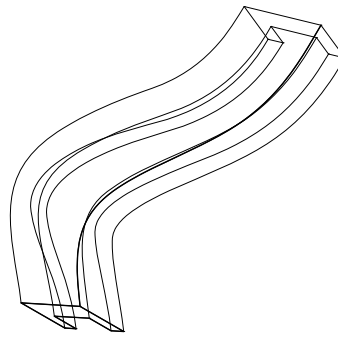


Figure 26-74 Swept solid with the aligned option **No**

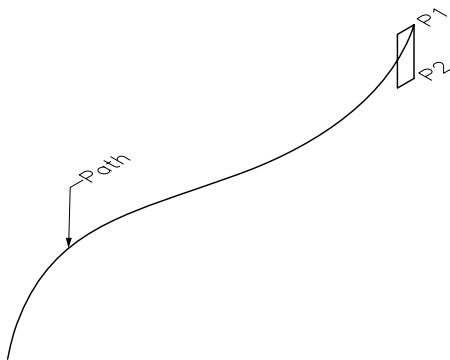


Figure 26-75 Path with object and P1 and P2 as base points

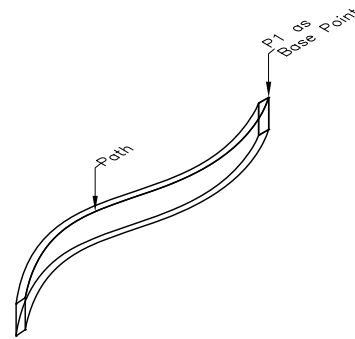


Figure 26-76 Sweep with P1 as base point

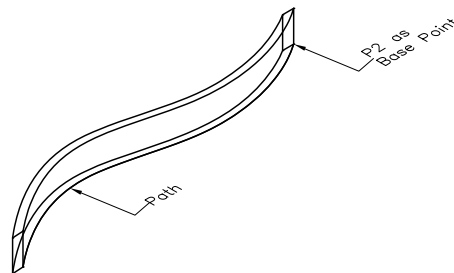


Figure 26-77 Sweep with P2 as the base point

Scale

This option is used to scale an object uniformly from the start of the path to the end of the path while sweeping it. Note that the size of the start section will remain the same and then end section will be scaled by the specified value. Also, the transition from the beginning to the end is smooth. Figure 26-78 shows a pentagon swept along a path with a scale factor of 0.25.

Twist

You can rotate an object about a path uniformly from the start to the end with a specified angle. The value of the twist should be less than 360 degrees. Figure 26-79 shows a sweep feature with 300-degree of twist.



Note

A helix can also be used as the path for the **Sweep** tool. Figure 26-80 shows the object and a helical path generated using the **Helix** tool and Figure 26-81 shows the resulting swept solid.

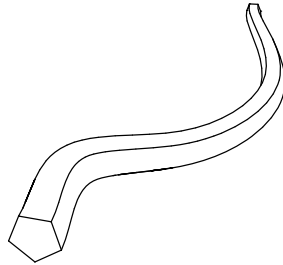


Figure 26-78 Sweep with the **Scale** option

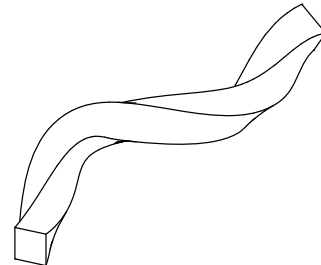


Figure 26-79 Sweep with the **Twist** option

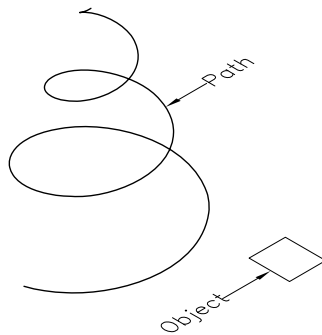


Figure 26-80 Helical path and object to be swept

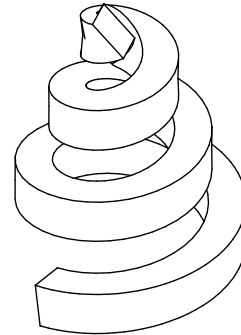


Figure 26-81 Sweep feature with the helical path

CREATING LOFTED SOLIDS*

Ribbon: Home > Modeling > Solid Creation drop-down > Loft
Or Solid > Solid > Sweep/Loft Drop-down > Sweep

Command: LOFT

Menu Bar: Draw > Modeling > Loft

Toolbar: Modeling > Loft

The **Loft** tool is used to create a feature by blending two or more similar or dissimilar cross-sections together to get a free form shape. These similar or dissimilar cross-sections may or may not be parallel to each other. To create a loft feature, choose the **Loft** tool from the **Solid** panel (Figure 26-82); the following prompt sequence will be displayed:

Command: _loft

Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid

Select cross sections in lofting order or [POint/Join multiple edges/MOde]: _MO

Closed profiles creation mode [SOlid/SUrface] <Solid>:

_SO

Select cross sections in lofting order or [POint/Join multiple edges/MOde]

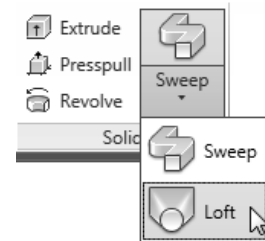


Figure 26-82 Selecting the **Loft** tool from the **Sweep/Loft** drop-down in the **Solid** panel

Next, select at least two sections in a sequence in which you want to blend them. The prompt sequence that will be followed after choosing the **Loft** button is given next.

Select cross sections in lofting order or [PPoint/Join multiple edges/MODE]: 1 found
 Select cross sections in lofting order or [PPoint/Join multiple edges/MODE]: 1 found, 2 total
 Select cross sections in lofting order or [PPoint/Join multiple edges/MODE]:
 2 cross sections selected
 Enter an option [Guides/Path/Cross sections only/Settings] <Cross sections only>:



Note

*As you are prompted to select cross-sections while creating a loft, you can select open or closed sections. Note that if you select an open section first, all subsequent sections should be open. The model thus created will be a surface. Similarly, if you select a closed section, all subsequent sections should be closed. The model created using these sections can be solid/surface. By default, a loft model is solid. Therefore, if cross sections selected are closed profiles and you need a surface model, then first you need to change the mode by entering **MO** at the **Select cross sections in lofting order or [PPoint/Join multiple edges/MODE]** prompt.*

After specifying the mode, you can select the cross sections, an imaginary point as the start point of the loft, or multiple edges on a solid model. These options are discussed next.

Point

In AutoCAD 2011, you can start a loft such that it begins from an imaginary point and passes through different sections. To do so, enter **PO** at the **Select cross sections in lofting order or [PPoint/Join multiple edges/MODE]** prompt and specify the start point of the loft, as shown in Figure 26-83. Next, select the cross sections in succession and press ENTER after selecting all cross sections; the preview of the loft feature will be displayed. Also, a Look up grip near the last cross section and the **Enter an option [Guides/Path/Cross sections only/Settings/CContinuity/Bulge magnitude] <Cross sections only>** prompt will be displayed. Select the **Cross sections only** option and press ENTER; the loft will be created, as shown in Figure 26-84.

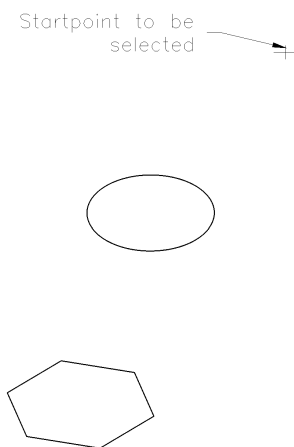


Figure 26-83 Sections and the imaginary start point to be selected

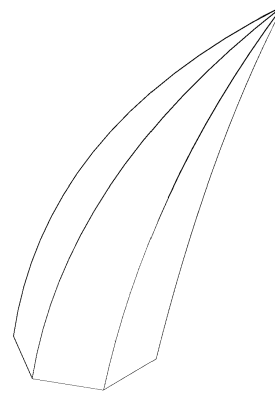


Figure 26-84 Resulting loft feature by using the **PO** option

Join multiple edges

In AutoCAD 2011, you can create a loft feature between the edges of a solid model and a profile. To do so, enter **J** at the **Select cross sections in lofting order or [PPoint/Join multiple edges/MODE]** prompt; the cursor will have the symbol of solid and you will be prompted to select

edges that are into be joined to a single cross section. Select the edges in succession (See Figure 26-85) and press ENTER. Then, select the cross sections and press ENTER; the preview of the loft will be displayed. One look up grip at the start of the loft surface and other look up grip near the base of the model will be displayed. The Look up grip at the start of the loft will have the surface continuity symbol (See Figure 26-86) and the other Look up grip will have the Loft symbol. Click on the Look up grip that has the surface continuity symbol and select the required option from the flyout displayed. The options in the other look up grip will be discussed later.

Given below is the prompt sequence to select the edges on the top face of the cuboid and a circle as the cross section.

Select cross sections in lofting order or [POint/Join multiple edges/MODE]: J
Select edges that are to be joined into a single cross section: (Select the first edge) 1 found
Select edges that are to be joined into a single cross section: (Select the second edge) 1 found, 2 total
Select edges that are to be joined into a single cross section: (Select the third edge) 1 found, 3 total
Select edges that are to be joined into a single cross section: (Select the fourth edge) 1 found, 4 total
Select edges that are to be joined into a single cross section:
Select cross sections in lofting order or [POint/Join multiple edges/MODE]: 1 found
Select cross sections in lofting order or [POint/Join multiple edges/MODE]:
2 cross sections selected
Enter an option [Guides/Path/Cross sections only/Settings/COntinuity/Bulge magnitude] <Cross sections only>: (Preview of the loft will be displayed with grips (See Figure 26-86). Press ENTER; the loft surface will be created, (Figure 26-87).

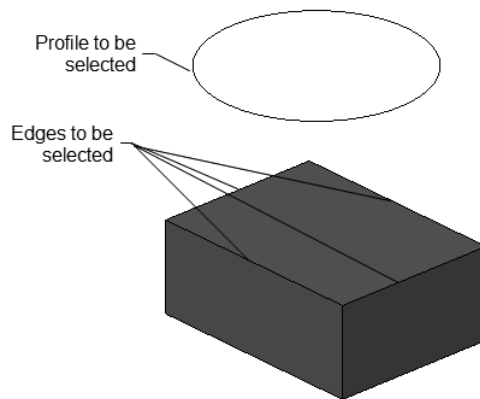


Figure 26-85 Edges and the profile to be selected

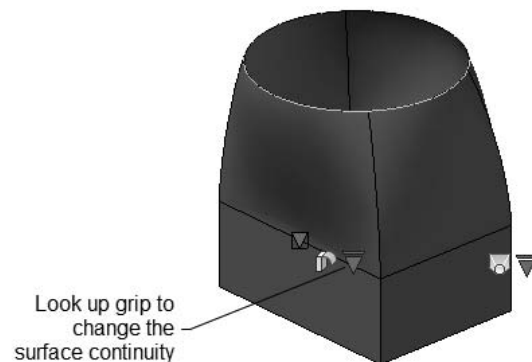


Figure 26-86 Look up grips displayed

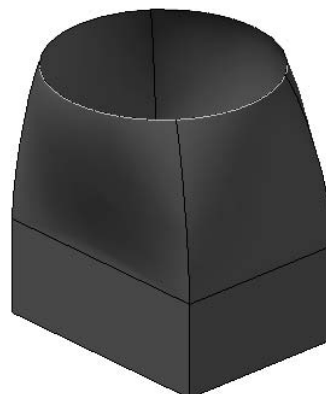


Figure 26-87 Loft surface created

After selecting all cross sections and on pressing ENTER, the **Enter an option [Guides/Path/Cross sections only/Settings/COntinuity/Bulge magnitude] <Cross sections only>** prompt will be displayed. The **C**Ontinuity and **B**ulge magnitude options will be displayed in the Command prompt only if you have selected the edges of a model, as one of the cross sections. Depending on requirement, you can invoke any one of those options. These options are discussed next.

Guide

AutoCAD enables you to select guide curves between the sections of the loft feature. These guide curves define the shape of the material addition between the selected sections for the loft feature. Guide curves also specify the points of different sections to be joined while adding material between them. This reduces the possibility of an unnecessary twist in the resulting feature. You can select as many guide curves as you require. But these guide curves should pass through or should touch each section that you selected for loft feature. Also the guides should start from the first section and terminate at the last section. Figure 26-88 shows three sections for the loft feature. Figure 26-89 shows the resulting loft feature. Figure 26-90 shows the sections to be lofted and the guide curves and Figure 26-91 shows the resulting model.

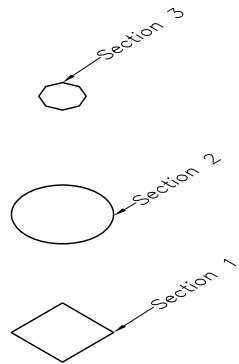


Figure 26-88 Three section curves for the loft feature



Figure 26-89 Resulting loft feature

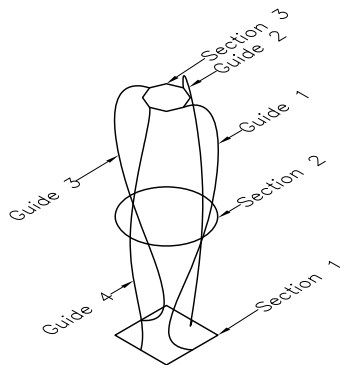


Figure 26-90 Sections and guide curves for the loft feature



Figure 26-91 Resulting loft feature by using the guide curve

Path

The path option is used to create a loft feature by blending more than one section along the direction specified by the path curve. Note that you can select only one path curve. The selected path curve may or may not touch all sections, but it should pass through all sketching planes of the sections. Figure 26-92 shows the sections and the path along which the blending of material will propagate and Figure 26-93 shows the resulting loft feature.

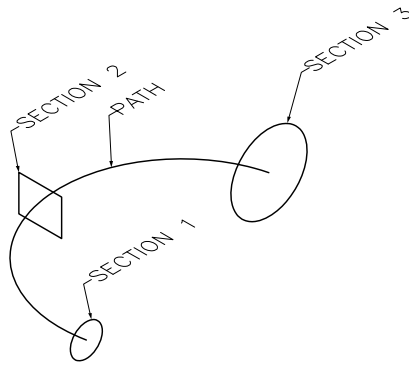


Figure 26-92 Sections and path to be used for the **Loft** option

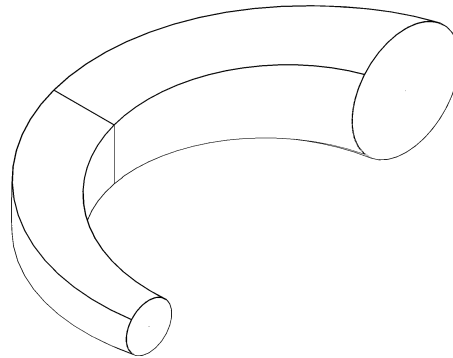


Figure 26-93 Resulting loft feature after using the **Path** option

Cross sections only

This is the default option displayed at the **Enter an option [Guides/Path/Cross sections only/Settings/COntinuity/Bulge magnitude] <Cross sections only>** prompt. If you need to create a loft without a guide or path, press ENTER; the loft feature will be created by using the cross sections only.

Settings

When you select **Settings**, the **Loft Settings** dialog box will be displayed, as shown in Figure 26-94. This dialog box is used to control the shape of the material addition between sections without specifying any guide curve or path. You can also select these options from the Look up grip displayed along with the preview. Various options in the **Loft Settings** dialog box are discussed next.

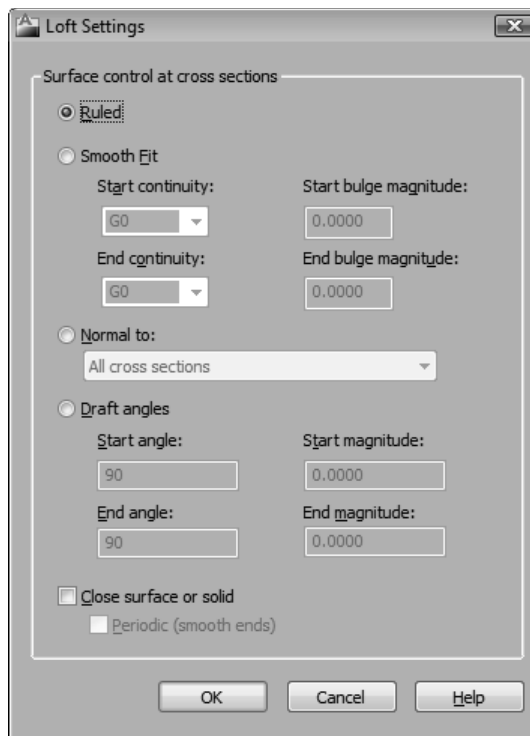


Figure 26-94 The **Loft Settings** dialog box

Ruled

If you select the **Ruled** radio button, a straight blend will be created by connecting different sections with straight lines. The loft feature created by this option will have sharp edges at the sections. Figure 26-95 shows the loft feature when the **Ruled** radio button is selected.

Smooth Fit

If you select the **Smooth Fit** radio button, a smooth loft will be created between sections. The loft feature created by this option will have smooth edges at the intermediate section. Figure 26-96 shows the loft feature when the **Smooth Fit** radio button is selected.

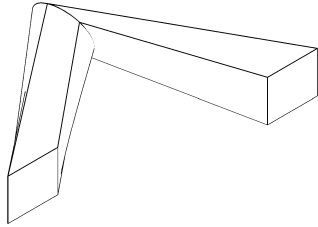


Figure 26-95 Loft solid with the **Ruled** radio button selected

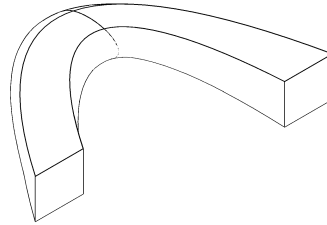


Figure 26-96 Loft solid with the **Smooth Fit** radio button selected

Normal to

This option controls the shape of the loft between the sections. When you select this radio button, the drop-down list will be enabled. The following options are available in this drop-down list:

Start cross section. In this option, the normal of the lofted feature is normal to the start section. The loft feature starts normal to the start section and follows a smooth polyline as it approaches the next section. Figure 26-97 shows the section selected to create the loft and Figure 26-98 shows the loft created by specifying the **Start cross section** option.

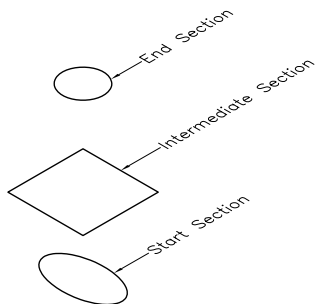


Figure 26-97 Start and end sections to create loft

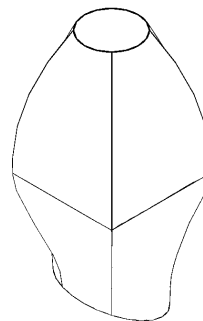


Figure 26-98 Loft solid with blending material normal to the start section

End cross section. In this option, the normal to the blending material is normal to the end section. At the second last section, the blending material starts along a spline and while reaching up to end section, it becomes normal to it, as shown in Figure 26-99.

Start and End cross sections. In this option, the normal to the blending material is normal to both the start and end section.

All cross sections. In this option, the normal to the blending material always remains normal to all cross-sections, as shown in Figure 26-100.

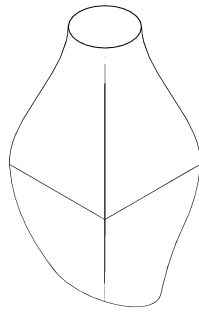


Figure 26-99 Loft solid with blending material normal to the end section

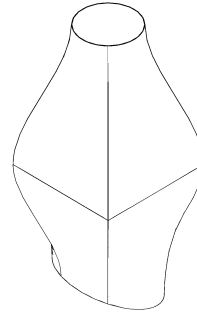


Figure 26-100 Loft solid with blending material normal to all sections

Draft angles

In this option, you can define the angle at the start section and the end section of the loft. Remember that you cannot define an angle for the intermediate sketches. You can also specify the distance up to which the blending material will follow this draft angle before bending toward the intermediate sections. The various options for specifying the draft angle are discussed next.

Start angle. This edit box is used to specify the draft angle at the start section of the loft. Figure 26-101 shows the two sections to be blended and Figure 26-102 shows the loft with the start and end angles as 0-degree. Figures 26-103 and 26-104 show the loft feature with the start angle as 90-degree and 180-degree, respectively.

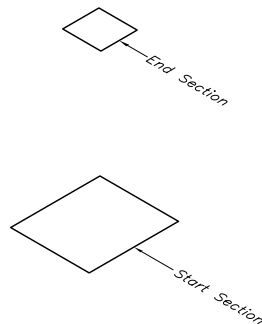


Figure 26-101 Start and end sections of the loft feature

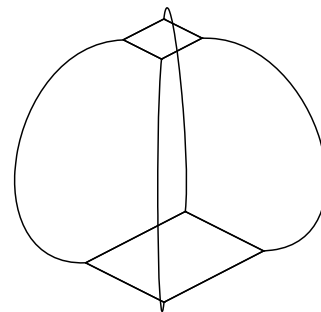


Figure 26-102 Loft feature with 0-degree start and end angles

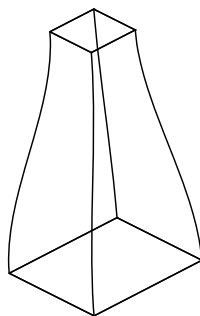


Figure 26-103 Loft feature with 90-degree start and end angles

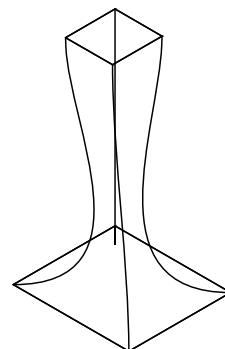


Figure 26-104 Loft feature with 180-degree start angle and 90-degree end angle

Start magnitude. The value specified in this region signifies the distance up to which the blending material will follow the start angle before bending toward the next section. Figure 26-105 shows a loft feature with 180-degree as the start and end angles, and 15 as the start magnitude. Figure 26-106 shows a loft feature with 180-degree as the start and end angles and 2 as the start magnitude. Notice the difference in the extent up to which the blended material follows the start angle in the two figures.

End angles. This edit box is used to specify the draft angle at the end section of the loft. Figure 26-101 shows two sections to be blended and Figure 26-102 shows the loft with the start and end angles as 0-degree. Figure 26-103 and Figure 26-104 show the loft feature with the end angle as 90-degree.

End magnitude. The value specified in this region signifies the distance up to which the blending material will follow the end angle before approaching the last section. Figure 26-105 shows a loft feature with 180-degree as the start and end angles, and 2 as the end magnitude. Figure 26-106 shows a loft feature with 180-degree as the start and end angles, and 10 as the end magnitude. Notice the difference in the extent up to which the blended material follows the end angle in the two figures.

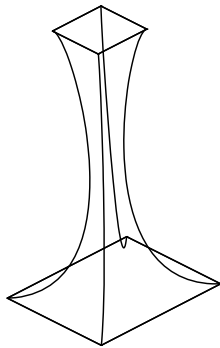


Figure 26-105 Loft feature with 180-degree as the start and end angles, 15 as the start magnitude, and 2 as the end magnitude

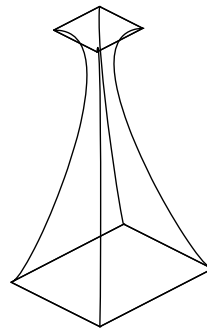


Figure 26-106 Loft feature with 180-degree as the start and end angles, 2 as the start magnitude, and 10 as the end magnitude

Close surface or solid

The **Close surface or solid** check box is selected to close a loft feature by joining the end section with the start section. Figure 26-107 shows a loft feature created with this check box cleared and Figure 26-108 shows the same sections lofted by this check box selected.

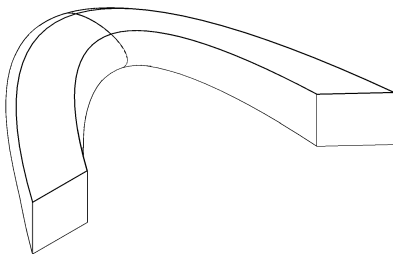


Figure 26-107 Loft solid with the **Close surface or solid** check box cleared

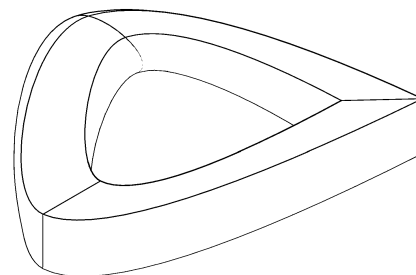


Figure 26-108 Loft solid with the **Close surface or solid** check box selected

**Note**

To visualize the effect of the **Close surface or solid** option, the loft feature should contain at least three sections. Otherwise, the extra blending material will be added at the same location where the loft feature would have been created without selecting this check box, thereby resulting in no change.

Periodic (smooth ends). If this check box is selected, the seam of the closed surface will not buckle, if you alter the shape of the surface.

**Tip**

You can also use a point as a section for the loft feature, but it should always be the start section or the end section of the loft feature.

Continuity

Continuity defines the smoothness of the loft surface between two cross sections, if one of the cross sections is an edge of a model. Continuity can be set to **G0** (position continuity), **G1** (tangency continuity), or **G2** (curvature continuity). **G0** maintains the position of the selected edges, **G1** the maintains tangency of the loft surface where it meets edges, and **G2** maintains the same curvature. By default, the continuity is set to G1. To edit the continuity setting of a loft, enter **CON** at the **Enter an option [Guides/Path/Cross sections only/Settings/COntinuity/Bulge magnitude] <Cross sections only>** prompt and press ENTER; you will be prompted to specify the continuity. Enter the required option and press ENTER.

You can also set the surface continuity by using the Continuity grip.

Bulge magnitude

The bulge magnitude determines the roundness or the amount of bulge between two cross sections, if one of the cross sections is an edge of a model. By default, the bulge magnitude is set to 0.5. If you want to edit the bulge magnitude, enter **B** at the **Enter an option [Guides/Path/Cross sections only/Settings/COntinuity/Bulge magnitude] <Cross sections only>** prompt and press ENTER; you will be prompted to enter the loft bulge magnitude at the start. Enter a value and press ENTER. If there is only one solid edge as a cross section, then the bulge will be created. If there are two solid edges as cross sections (one at the start and the other at the end), then you will be prompted to enter the bulge magnitude at the start and end. Type a value and press ENTER; the bulge will be created. If you need to change the bulge magnitude while creating, then click the Bulk magnitude Look up grip; a dot with a leader will be displayed. Drag the dot dynamically to change the bulk magnitude.

You can modify the bulk magnitude value after the loft is created. To do so, select the loft; three grips will be displayed on it. Double-click on any grip; the **Properties** palette will be displayed with the settings of the loft feature. Using this palette, you can modify the bulge magnitude and other parameters.

Figure 26-109 shows two solids whose edges on the top face and the region are to be selected for creating a loft feature. Figure 26-110 shows the preview of the loft feature with grips. Figure 26-111 shows the loft feature with bulge magnitude added at the start and the end.

**Note**

You can also set the surface continuity and the bulge continuity in the **Loft setting** dialog box also.



Figure 26-109 Two solids and a region to create a loft

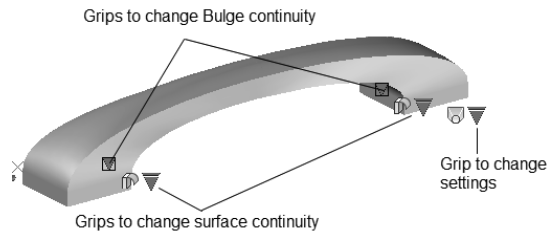


Figure 26-110 Preview of the loft and the grips displayed

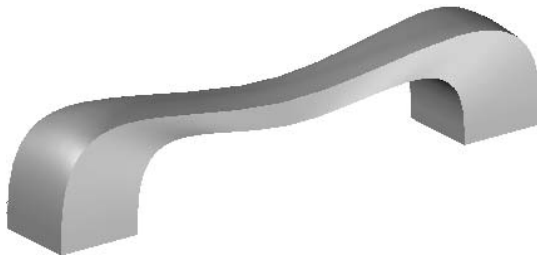


Figure 26-111 Loft surface created

CREATING PRESSPULL SOLIDS

Ribbon: Home > Modeling > Press/Pull

Toolbar: Modeling > Press/Pull

Command: PRESSPULL



The **Press/Pull** tool is used to remove or add material of any desired shape in a model. As the name presspull indicates, if you press a closed boundary inside an existing model, it will remove the material of the shape of the closed boundary. However, if you pull a closed boundary outside an existing model, it will add material to the model. To perform this operation, choose the **Press/pull** tool from the **Modeling** panel. Next, click inside a closed boundary that you want to press or pull and then move the mouse in the desired direction. You can specify the desired height by entering a value or by clicking the mouse. Figure 26-112 shows the solid model with the closed area to be presspulled. Figure 26-113 shows the resulting model by pressing the closed region and Figure 26-114 shows the resulting model by pulling the closed area.

If the sketch that you select to press or pull has nested loops, as shown in Figure 26-115, the inner closed loops will be deleted from the outer loop, as shown in Figure 26-116. The closed area you select for the presspull should consist of lines, polylines, 3D face, edges, regions, and faces of a 3D solid. But all these entities should form a coplanar closed area.



Tip

To perform the presspull operation without choosing the **Presspull** button, press and hold the **CTRL+ALT** keys and then click inside the closed area to be pressed or pulled.

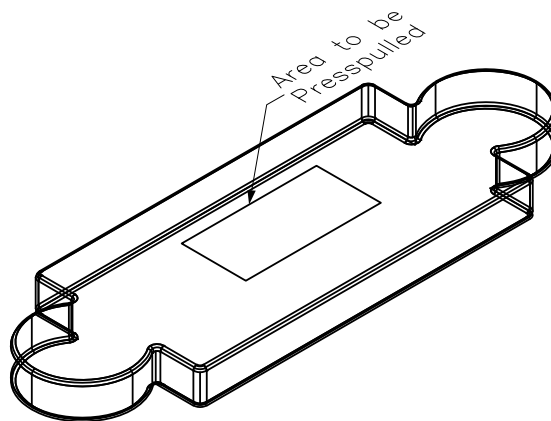


Figure 26-112 Model with closed area to be presspulled

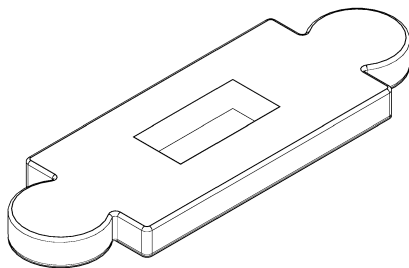


Figure 26-113 Model with the closed area pressed

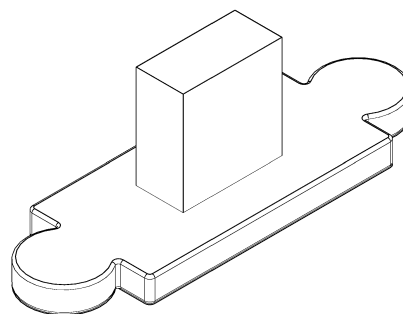


Figure 26-114 Model with closed area pulled

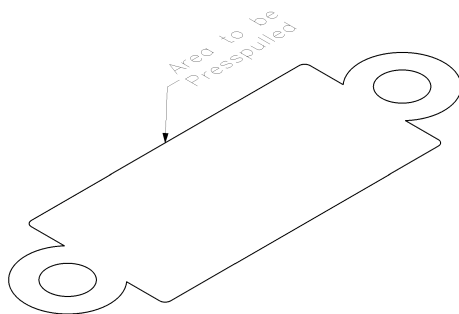


Figure 26-115 Area to be presspulled with closed boundary inside it

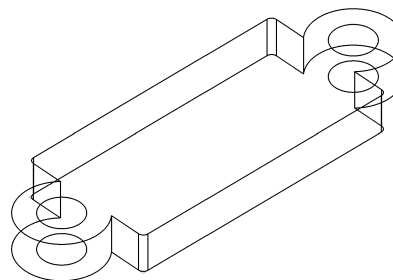


Figure 26-116 Resulting model after the presspull operation is performed

EXAMPLE 2

In this example, you will create the solid model shown in Figure 26-117.

1. Start a new file using the *Acad3d.dwt* template.
2. Turn off the grid display, increase the limits to 120,120 and then zoom to the limits of the drawing.

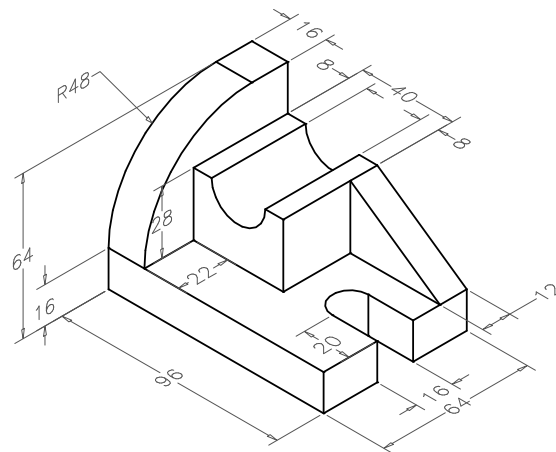


Figure 26-117 Solid model for Example 2

3. Set the view to SE Isometric by using the View cube. Then, right-click on the View Cube and choose the **Parallel** option.
4. Switch on the ortho mode. Using lines and arc, create the base of the model with the dimensions shown in Figure 26-118.

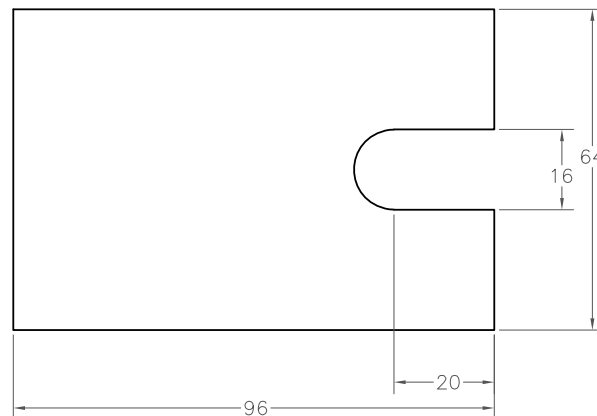


Figure 26-118 Base for the solid model

5. Choose the **Region** tool from the **Draw** panel in the **Home** tab and follow the prompt sequence given next to convert the sketch drawn into region.

Select objects: *Select the complete base.*

Select objects:

1 loop extracted.

1 Region created.



Note

You can also use the **PLINE** command to create this sketch. In that case, you do not need to convert it into a region.

6. Choose the **Extrude** tool from **Home > Modeling > Solid Creation** drop-down and follow the prompt sequence given next to extrude the region.

Command: `_extrude`

Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid

Select objects to extrude or [MOde]: `_MO` Closed profiles creation mode [SOlid/SURface]

<Solid>: `_SO`

Select objects to extrude or [MOde]: *Select the region*

1 found

Select objects to extrude or [MOde]:

Specify height of extrusion or [Direction/Path/Taper angle/Expression]: **16**

7. Change the visual style to Wireframe by choosing the **Wireframe** option from the **Visual Styles** drop-down list in the **View** panel.
8. Choose the **Wedge** tool from **Home > Modeling > Solid Primitives** drop-down and follow the prompt sequence given next to create the next feature.

Specify first corner or [Center]: *Pick a point on the screen.*

Specify other corner or [Cube/Length]: **L**

Specify length: **40**

Specify width: **12**

Specify height or [2Point]: **28**

9. Move this wedge and align it with the base. Use the **Vertex** option of the **3D Object Snap** to snap the vertex. The model after applying the **Hidden** visual style is shown in Figure 26-119.

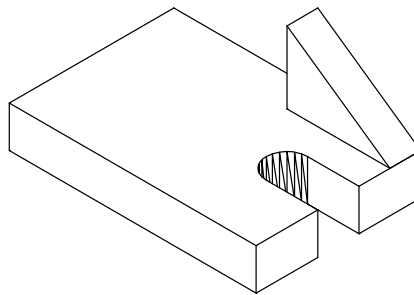


Figure 26-119 Wedge aligned with the base

10. Orient the view to **Front** by choosing Front in the View cube and choose **WCS > New UCS** from the drop-down list below it; you are prompted to specify the origin.
11. Enter **V** at the **Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis] <World>** prompt and press ENTER.
12. Right-click on the View Cube and choose the **Parallel** option.
13. Draw the profile shown in Figure 26-120 at any place in the drawing area and then convert it into a region.
14. Choose the **Extrude** tool from **Home > Modeling > Solid Creation** drop-down and extrude the region to a depth of 42 units.
15. Move this extrude feature and align it with the base. Use the **Vertex** option of the **3D Object Snap** to snap the vertex.

The isometric view of the model after aligning the extrude feature is shown in Figure 26-121.

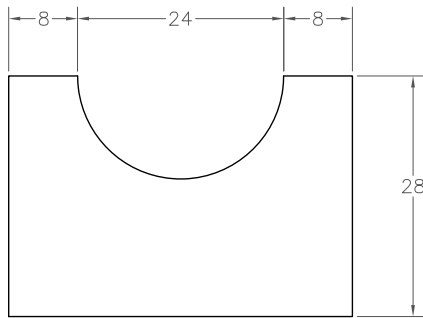


Figure 26-120 The region to be extruded

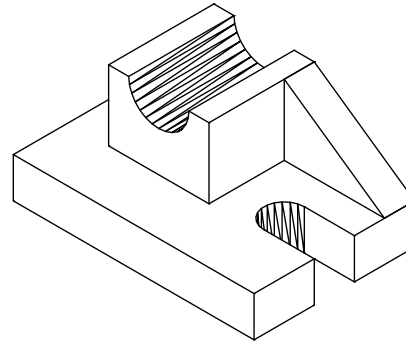


Figure 26-121 Solid aligned with the base

16. Orient the view to Right by choosing **Right** in the View cube and choose **WCS > New UCS** from the drop-down list below it; you are prompted to specify the origin.
17. Enter **V** at the **Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis] <World>** prompt and press ENTER.
18. Right-click on the View Cube and choose the **Parallel** option.
19. Draw the profile shown in Figure 26-122 at any place in the drawing area and then convert it into a region.
20. Extrude the region to a depth of 16 units and then move it so that it is properly aligned with the base, see Figure 26-123.

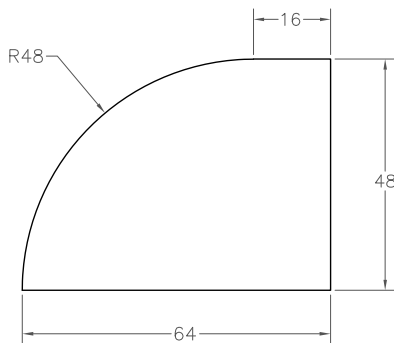


Figure 26-122 The region before extrusion

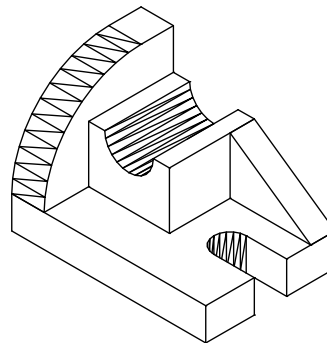


Figure 26-123 Next solid aligned with the base

21. Choose the **Union** tool from the **Solid Editing** panel and follow prompt sequence given next:
 Select objects: *Select all objects.*
 Select objects:
22. Change the visual style by selecting the **Shades of Grey** option from the **Visual Styles** drop-down list in the **View** panel. The final solid model should look similar to the one shown in Figure 26-124.

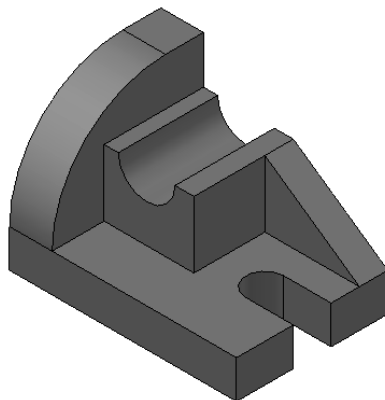


Figure 26-124 Solid model for Example 2

EXAMPLE 3

In this example, you will create the solid model shown in Figure 26-125.

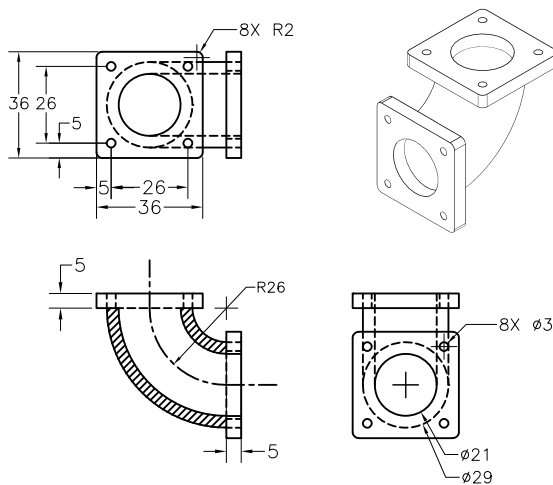


Figure 26-125 Solid model for Example 3

1. Start a new file in the *Acad3d.dwt* template.
2. Turn off the grid display, increase the limits to 75,75, and then zoom to the limits of the drawing.
3. Orient the view to Left by choosing **Left** in the View cube and choose **WCS > New UCS** from the drop-down list below it; you are prompted to specify the origin.
4. Enter **V** at the **Specify origin of UCS or [Face/NAMED/Object/Previous/View/World/X/Y/Z/ZAxis] <World>** prompt and press ENTER.
5. Right-click on the View Cube and choose the **Parallel** option.
6. Draw the path and cross section of the sweep feature, as shown in Figure 26-126.
7. Choose the **Region** tool from the **Draw** panel in the **Home** tab and follow the prompt sequence given next to convert the sketch into a region.

Select objects: *Select both the circles.*

Select objects:

2 loop extracted.

2 Region created.

8. Choose the **Subtract** tool from the **Solid Editing** panel in the **Home** tab and follow the prompt sequence:

Select objects: *Select the region of diameter 29.*

Select objects:

Select solids and regions to subtract.

Select objects: *Select the region of diameter 21.*

Select objects:

9. Change the visual style to Wireframe by choosing the **Wireframe** option from the **Visual Styles** drop-down list in the **View** panel.
10. Choose the **Sweep** tool from **Home > Modeling > Solid Creation** drop-down and follow the prompt sequence given next to sweep the region along the path.

Current wire frame density: ISOLINES=4, Closed profiles creation mode = Solid

Select objects to sweep or [MOfde]: *_MO* Closed profiles creation mode [SOLid/SURface]

<Solid>: *_SO*

Select objects to sweep or [MOfde]: *Select the region created in Step 8*

1 found

Select objects to sweep or [MOfde]:

Select sweep path or [Alignment/Base point/Scale/Twist]: *Select the arc created in Step 6.*

11. Set the view to **SE Isometric** by choosing the appropriate hotspot in the ViewCube. Also, select the **Realistic** option in the **Visual Styles** drop-down list in the **View** panel. The resulting model should look similar to the one shown in Figure 26-127.

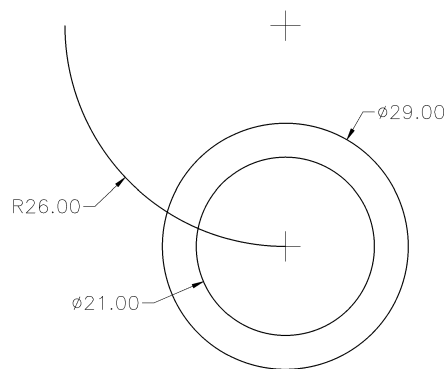


Figure 26-126 Sketch for base pipe feature

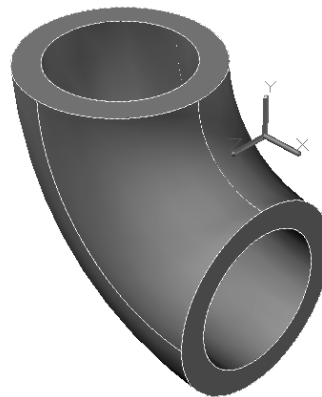


Figure 26-127 Resulting sweep feature

12. Choose the **Top** option from the **Views** panel in the **View** tab and draw the sketch with the dimensions shown in Figure 26-128.
13. Change the view to isometric. The model after drawing the sketch and turning off the dimension is shown in Figure 26-129.

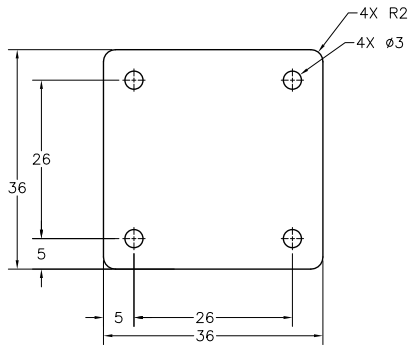


Figure 26-128 Sketch for the area to Press/Pull

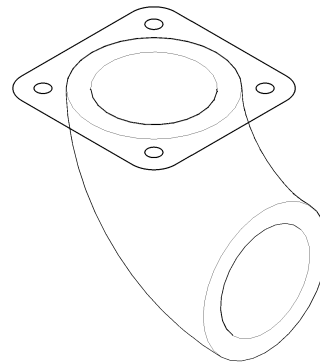


Figure 26-129 Location of the sketch

14. Choose the **Press/Pull** tool from the **Modeling** panel and then click inside the rectangular area at the location shown as Area A in Figure 26-130. Next, move the mouse in the upward direction, enter the value **5** at the Command prompt, and press ENTER.
15. Again, choose the **Press/Pull** tool and pull the area shown as Area B in Figure 26-131 by the same value as specified in Step 8. The resulting model should be similar to the one shown in Figure 26-131.

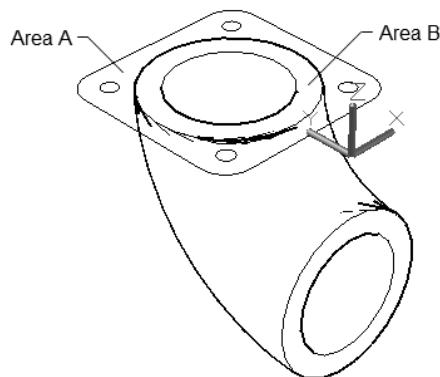


Figure 26-130 Two areas to presspull

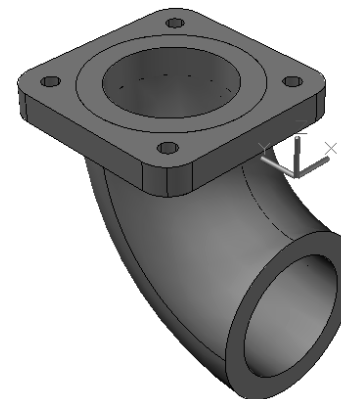


Figure 26-131 Resulting model after Step 10

16. Similarly, create the flange for the other face of the pipe. The resulting model should be similar to the one shown in Figure 26-132.
17. Choose the **Union** tool from the **Solid Editing** panel in the **Home** tab and follow the prompt sequence given next.

Select objects: *Select all objects.*

Select objects:

The final solid model should look similar to the one shown in Figure 26-133.

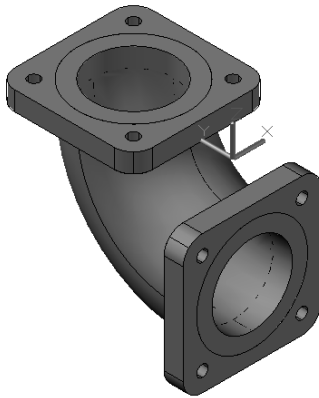


Figure 26-132 Resulting model after Step 12

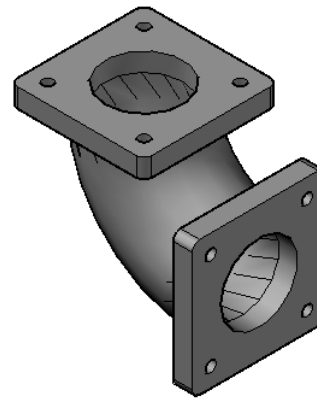


Figure 26-133 Final model after Union

Self-Evaluation Test

Answer the following questions and then compare them to those given at the end of this chapter:

1. The **Cone** tool can be used to create a solid cone with a circular and an elliptical base. (T/F)
2. Open entities can be converted into regions. (T/F)
3. An ellipse can be converted into a polysolid. (T/F)
4. You can twist a cross-section while sweeping it. (T/F)
5. The _____ tool can be used to add as well as remove material from a model.
6. In AutoCAD, you can create a pyramid of maximum 42 sides. (T/F)
7. The _____ tool is used to generate a helical curve.
8. The number of paths for a loft feature cannot be more than _____.
9. At least _____ cross-sections are required to create a loft feature.
10. You can extrude the selected region about a path using the _____ option of the **Extrude** tool.

Review Questions

Answer the following questions:

1. You can revolve an open entity. (T/F)
2. You cannot apply Boolean operations on regions. (T/F)
3. The entity to be revolved should lie completely on one side of the revolution axis. (T/F)
4. You cannot apply the **Press/pull** tool on an open object. (T/F)

5. There cannot be more than one guide for the loft feature. (T/F)
6. A curve generated by the **Helix** tool can be used as a path for the swept solids. (T/F)
7. Which of the following values for the taper angle is used to taper the extruded model from the base?
 - (a) Positive
 - (b) Negative
 - (c) Zero
 - (d) None of these
8. Which of the following tools is used to create a cube?
 - (a) **Box**
 - (b) **Cuboid**
 - (c) **Polygon**
 - (d) **Cylinder**
9. Which of the following commands is used to check the interference between the selected solid models?
 - (a) **INTERFERE**
 - (b) **INTERSECT**
 - (c) **INTERFERENCE**
 - (d) **CHECK**
10. The **Press/pull** tool is used to
 - (a) Add material
 - (b) Remove material
 - (c) Both add and remove material
 - (d) Neither add nor remove material
11. The _____ direction is known as the extrusion direction.
12. The _____ command is used to create an apple-like structure.
13. The _____ command is used to create a revolved solid.
14. Guides for the **Loft** tool should always _____ the cross-section.
15. The _____ option of the **Revolve** tool is used to select a 2D entity as the revolution axis.

EXERCISE 2

In this exercise, you will create the solid model shown in Figure 26-134. Assume the missing dimensions.

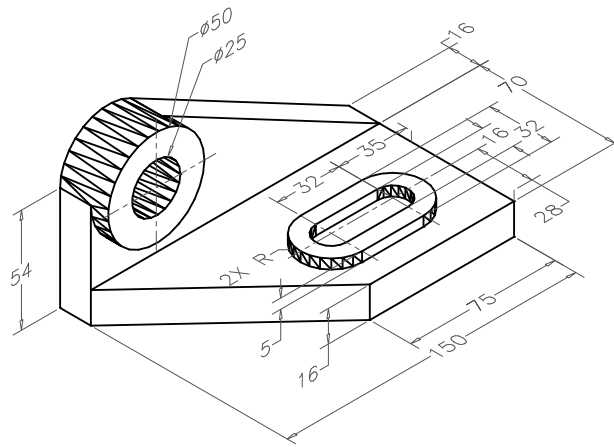


Figure 26-134 Model for Exercise 2

EXERCISE 3

In this exercise, you will create the solid model shown in Figure 26-135. The dimensions for the model are given in the same figure. Assume the missing dimensions.

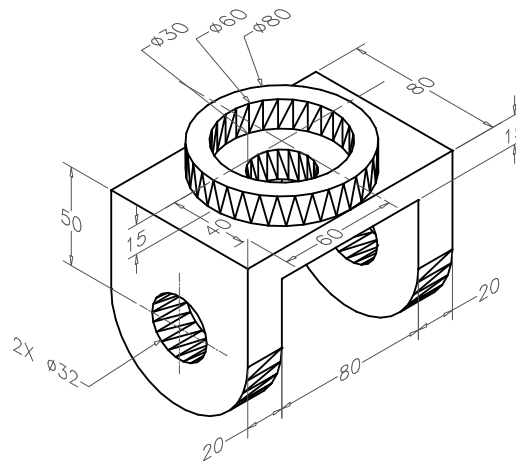


Figure 26-135 Solid model for Exercise 3

EXERCISE 4

In this exercise, you will create the solid model shown in Figure 26-136. The dimensions for the model are shown in Figures 26-137 and 26-138.

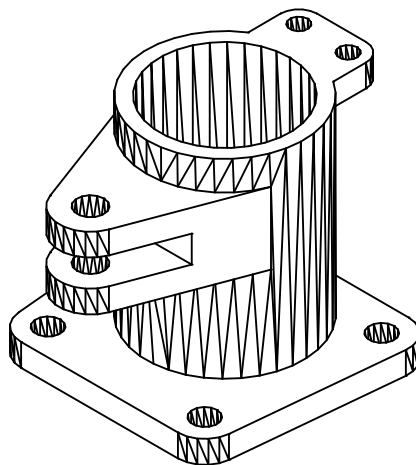


Figure 26-136 Model for Exercise 4

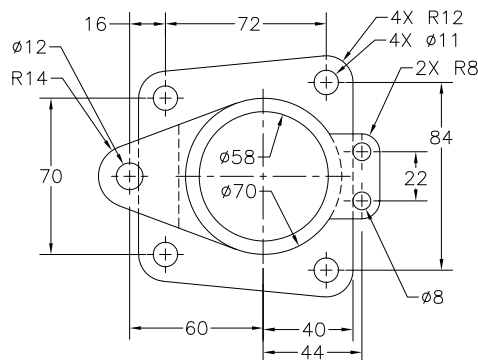


Figure 26-137 Top view of the model

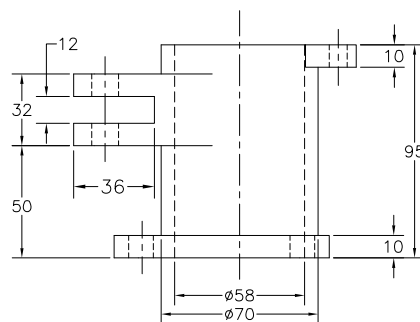


Figure 26-138 Front view of the model

Answers to Self-Evaluation Test

1. T, 2. F, 3. F, 4. T, 5. PRESSPULL, 6. F, 7. HELIX, 8. one, 9. two, 10. Path