Chapter 2

Getting Started with 3D

Learning Objectives

After completing this chapter, you will be able to:

- *Start 3D workspace in AutoCAD for creating models*
- *Use three-dimensional (3D) drawing*
- *Use various types of 3D models*
- *Know conventions used in AutoCAD*
- *Use the Viewpoint Presets tool to view objects in 3D space*
- *Use the ViewCube to view objects easily in 3D space*
- *Know various types of 3D coordinate systems*
- *Set thickness and elevation for objects*
- *Dynamically view 3D objects using the Orbit tool*
- *Use the SteeringWheel for viewing a model*

Key Terms

- *Wireframe*
- *Surface*
- *Solids*
- *ViewCube*
- *DDVPOINT*
- *3D Coordinate System*
- *ELEV*
- *Hide*
- *3D Polyline*
- *3D Face*
- *PFACE*
- *3DMESH*
- *Edit Polyline*
- *SteeringWheel*
- *Orbit*
- *3DCLIP*
- *Nudge*

STARTING THREE DIMENSIONAL (3D) MODELING IN AutoCAD

In AutoCAD, you can start the 3D Modeling in a separate workspace. All tools required to create the 3D design are displayed in this workspace by default. To start a new file in the 3D workspace, invoke the **Select template** dialog box and select the *acad3D.dwt* template. This template file supports the 3D environment. The other template files that support the 3D workspace are *acadiso3D.dwt*, *acad -Named Plot Styles3D.dwt*, and *acadISO -Named Plot Styles3D.dwt*. Next, select the **3D Modeling** option from the **Workspace** drop-down list available at the right of the **Status Bar**. Figure 2-1 shows the **3D Modeling** workspace of AutoCAD.

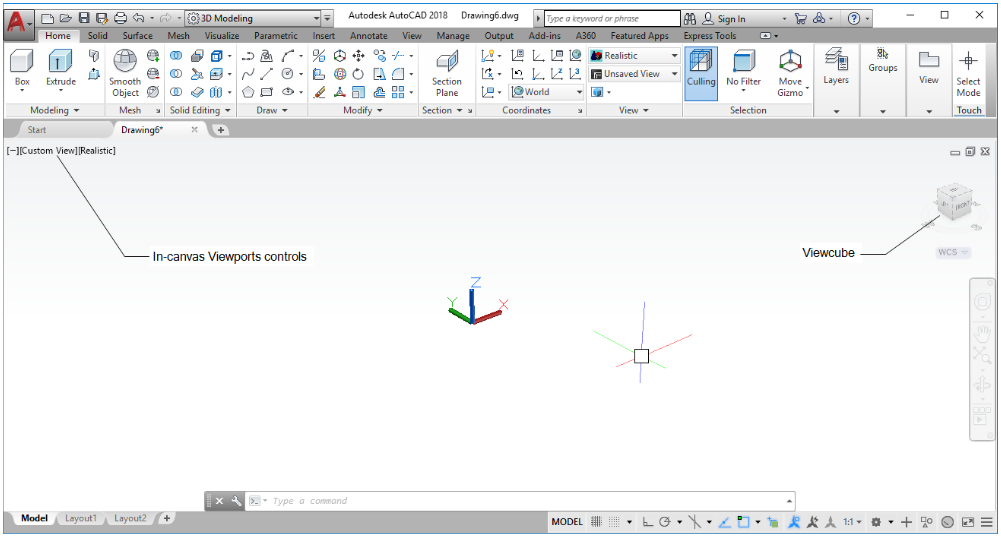


Figure 2-1 *The **3D Modeling** workspace of AutoCAD*

USE OF THREE-DIMENSIONAL DRAWING

The real world, and literally whatever you see around it, is three-dimensional (3D). Each and every design idea you think of is 3D. Before the development of 3D software packages, it was not possible to materialize these ideas in the design industry due to the lack of the third dimension. This is because the drawings were made on the drawing sheets, which are two dimensional objects with only *X* and *Y* coordinates. Therefore, it was not possible to draw the 3D objects. As prototyping is a very long and costly affair, therefore, the designers had to suppress their ideas and convert the 3D designs into 2D drawings.

The use of computers and the CADD (Computer Aided Design & Drafting) Technology has brought a significant improvement in materializing the design ideas and creating the engineering drawings. You can create a proper 3D object in the computer using the CADD Technology. This technology allows you to create models with the third coordinate called the *Z* coordinate, in addition to the *X* and *Y* coordinates. Apart from materializing the design ideas, the 3D models have the following advantages:

1. **Generate the drawing views.** Once you have created a 3D model, its 2D drawing views can be automatically generated.

2. **Provide realistic effects**. You can provide realistic effects to the 3D models by assigning a material and providing light effects to them. For example, an architectural drawing can be made more realistic and presentable by assigning the material to the walls and interiors and adding lights to it. You can also create its bitmap image and use it in presentations.

3. **Create the assemblies and check them for interference**. You can assemble various 3D models and create the assemblies. Once the components are assembled, you can check them for interference to reduce the errors and material loss during manufacturing. You can also generate the 2D drawing views of the assemblies.

4. **Create an animation of the assemblies**. You can animate the assemblies and view the animation to provide the clear display of the mating parts.

5. **Apply Boolean operations**. You can apply Boolean operations to the 3D models.

6. **Calculate the mass properties**. You can calculate the mass properties of the 3D models.

7. **Cut Sections**. You can cut sections through the solid models to view the shape at that cross-section.

8. **NC Programs**. You can generate an NC program with the help of 3D models by using a CAM software.

TYPES OF 3D MODELS

In AutoCAD, depending on their characteristics, the 3D models are divided into the following three categories:

Wireframe Models

Wireframe models are created using simple AutoCAD entities such as lines, polylines, rectangles, polygons, or some other entities such as 3D faces, and so on. To understand the wireframe models, consider a 3D model made up of match sticks or wires. These models consist of edges only, therefore you can see through them. You cannot apply the Boolean operations on these models and cannot calculate their mass properties. Wireframe models are generally used in frame building of vehicles. Figure 2-2 shows a wireframe model.

Surface Models

Surface models are made up of one or more surfaces. They have a negligible wall thickness and are hollow inside. To understand these models, consider a wireframe model with a cloth wrapped around it. You cannot see through it. These models are used in the plastic molding industry, such as shoe manufacturing, utensils manufacturing, and so on.

You can directly create a surface or a mesh. Alternatively, you can create wireframe model and then convert it into a mesh or surface model. Remember that you cannot perform the Boolean operations in surface models. Figure 2-3 shows a surface model created using a single surface or a mesh and Figure 2-4 shows a surface model created using a combination of surfaces.

Solid Models

Solid models are the solid filled models having mass properties. To understand a solid model, consider a model made up of metal or wood. You can perform Boolean operations on these models such as cutting a hole through them, or even cutting them into slices. Figure 2-5 shows a solid model.

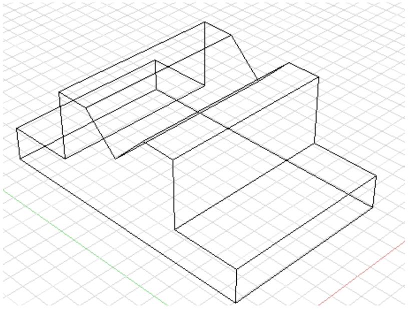


Figure 2-2 *A wireframe model*

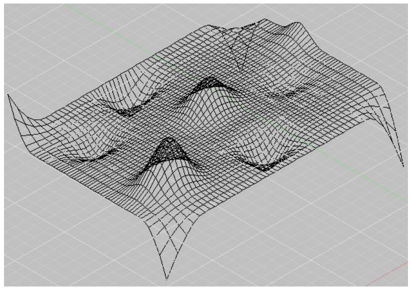


Figure 2-3 *A surface model created using a single surface*

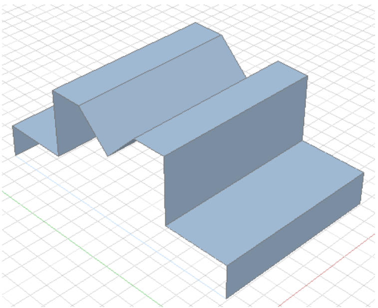


Figure 2-4 *A semi-finished surface model created using more than one surface*

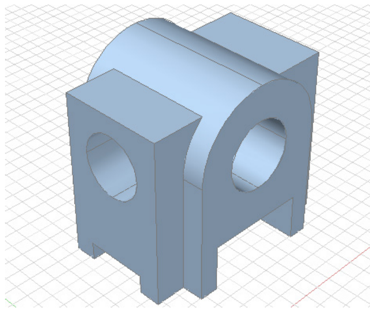


Figure 2-5 *A solid model*

CONVENTIONS FOLLOWED IN AutoCAD

It is important for you to know the conventions in AutoCAD before you proceed with 3D. AutoCAD follows these three conventions:

1. By default, any drawing you create in AutoCAD will be in the world XY plane.

2. The right-hand thumb rule is followed in AutoCAD to identify the *X*, *Y*, and *Z* axis directions. This rule states that if you keep the thumb, index finger, and middle finger of the right hand mutually perpendicular to each other, as shown in Figure 2-6(a), then the thumb of the right hand will represent the direction of the positive *X* axis, the index finger will represent the direction of the positive *Y* axis, and the middle finger of the right hand will represent the direction of the positive *Z* axis, refer to Figure 2-6(b).

3. The right-hand thumb rule is followed in AutoCAD to determine the direction of rotation or revolution in the 3D space. The right-hand thumb rule states that if the thumb of the

right hand points in the direction of the axis of rotation, then the direction of the curled fingers will define the positive direction of rotation, refer to Figure 2-6(c).

This rule will be used during the rotation of 3D models or the UCS and also, during the creation of revolved surfaces and revolved solids.

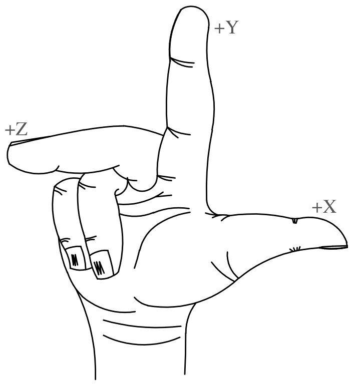


Figure 2-6(a) *The orientation of fingers as per the right-hand thumb rule*

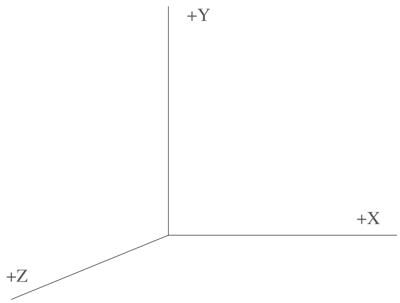


Figure 2-6(b) *Orientation of the X, Y, and Z axes*

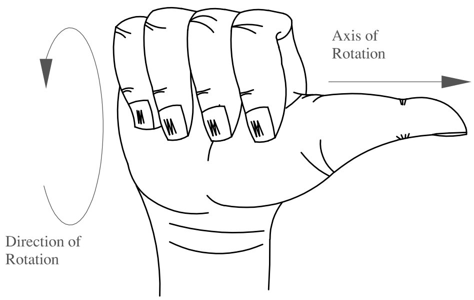


Figure 2-6(c) *Right-hand thumb rule showing the axis and direction of rotation*

CHANGING THE VIEWPOINT TO VIEW 3D MODELS

Until now, you have been drawing only the 2D entities in the *XY* plane and in the Plan view. But in the Plan view, it is very difficult to find out whether the object displayed is a 2D entity or a 3D model, refer to Figure 2-7. In this view, the cuboid displayed appears as a rectangle. The reason for this is that, by default, you view the objects in the Plan view from the direction of the positive *Z* axis. You can avoid this confusion by viewing the object from a direction that also displays the *Z* axis. In order to do that, you need to change the viewpoint so that the object is displayed in the space with all three axes, as shown in Figure 2-8. In this view, you can also see the *Z* axis of the model along with the *X* and *Y* axes. It will now be clear that the original object is not a rectangle but a 3D model. The viewpoint can be changed by using the ViewCube available in the drawing area or by using the options in the **View** panel.

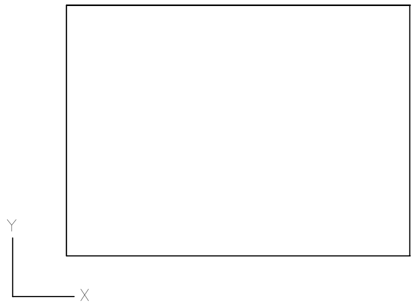


Figure 2-7 *Cuboid looking like a rectangle in the Plan view*

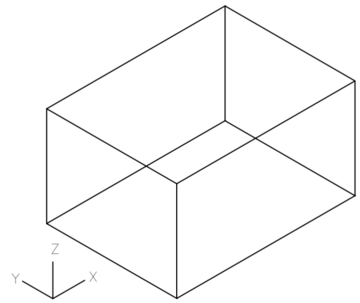


Figure 2-8 *Viewing the 3D model after changing the viewpoint*

Tip
Remember that changing the viewpoint does not affect the dimensions or the location of the model. When you change the viewpoint, the model is not moved from its original location. Changing the viewpoint only changes the direction from which you view the model.

Changing the Viewpoint Using the ViewCube

You can change the view of the models easily and quickly using the ViewCube. The ViewCube is displayed when the **3D Modeling** workspace is enabled and it allows you to switch between the standard and isometric views, roll the current view, or return to the **Home** view of a model. ViewCube consists of Cube, Home, Compass, and WCS menu. When you move the cursor over the ViewCube, it becomes active and the area at the pointer tip gets highlighted. The highlighted area can be a face, a corner, or an edge of the ViewCube. These highlighted areas are called hotspots. There are 6 faces, 12 edges, and 8 corners on a cube, refer to Figure 2-9. So, you can get 26 views by using the ViewCube. You can select the required hotspot to restore a view. You can also go back to the Home view by clicking on the Home icon of the ViewCube. You can set any existing view as the Home view by choosing the **Set Current View as Home** option from the shortcut menu that is displayed on right-clicking on the ViewCube. By default, a model is displayed in the perspective view. To display a model in parallel projection, right-click on the ViewCube and choose **Parallel** from the shortcut menu.

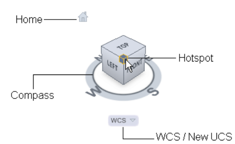


Figure 2-9 *The ViewCube*

The compass on the ViewCube indicates the geographic location of a model. The **N**, **E**, **S**, and **W** alphabets on the compass indicate the North, East, South, and West directions of a 3D model. You can pick and drag the compass ring to rotate the current view in the same plane. You can choose between the **UCS** and **WCS** from the WCS menu and even create a new UCS from the menu. When you right-click on the ViewCube, a shortcut menu will be displayed. Choose the **ViewCube Settings** option from it; the **ViewCube Settings** dialog box will be displayed, refer to Figure 2-10. This dialog box allows you to adjust the display of the compass ring and UCS menu, size, appearance, and location of the ViewCube.

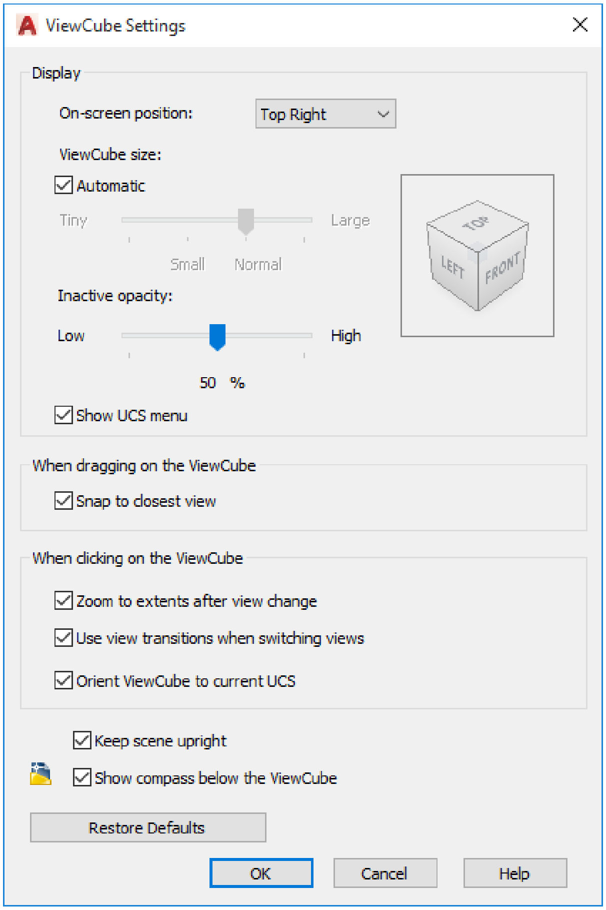


***Figure 2-10** The **ViewCube Settings** dialog box*

Note

*In AutoCAD, you can change the current viewport by using the **In-canvas Viewport controls**, displayed as **[-][Custom View][Realistic]**. It is available at the top left corner of the drawing area. When you choose the **[Custom View]** option, a flyout is displayed, refer to Figure 2-1. You can select the desired view from the flyout.*

Changing the Viewpoint Using the Ribbon or the Toolbar

You can also set the viewpoint either by using the **Views** panel in the **Visualize** tab of the **Ribbon** or by using the **View** toolbar.

The **View** drop-down list in the **Views** panel consists of six preset orthographic views: **Top**, **Bottom**, **Left**, **Right**, **Front**, and **Back**; and four preset isometric views: **SW Isometric**, **SE Isometric**, **NE Isometric**, and **NW Isometric**, refer to Figure 2-11. This drop-down list also allows you to retrieve the previously set views or any previously created named views. All the above-mentioned controls are also available in the **View Manager** dialog box that is invoked on selecting the **View Manager** option from the **Views** drop-down list.

Choose the **New** button from the **View Manager** dialog box to create a new view; the **New View** / **Shot Properties** dialog box will be displayed, as shown in Figure 2-12.

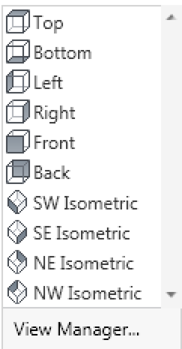


Figure 2-11 The ***View*** *drop-down list*

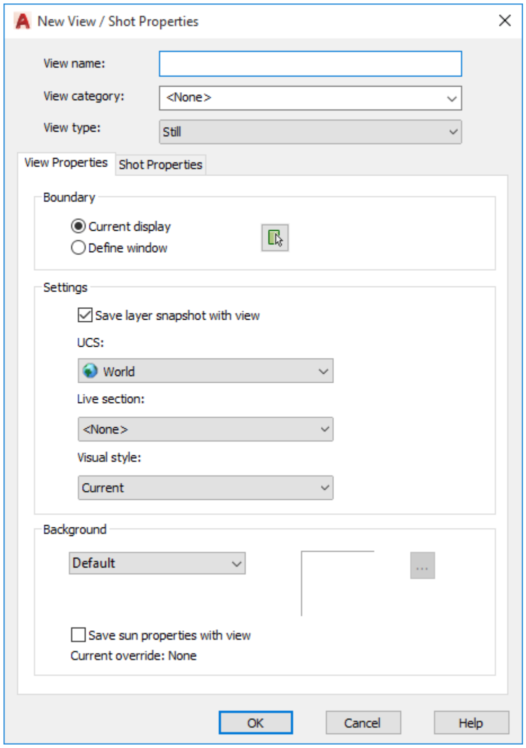


Figure 2-12 The ***New View / Shot Properties*** *dialog box*

Enter a name in the **View name** edit box to save the view. You can save different types of views into different categories using the options in this dialog box. The different types of views available in the **View type** drop-down list are **Still**, **Cinematic**, and **Recorded Walk**. But only **Still** is used for creating views and the other two are used for creating shots. The **View Properties** tab of this dialog box has three main areas: **Boundary**, **Settings** and **Background**. The **Boundary** area allows you to save the current display or you can select an area of display by choosing the **Define window** radio button. In the **Settings** area, you can control the position of UCS to save the view, select the section of the view, and control the display of the model. The drop-down list in the **Background** area is used to select the type of background to save the view. The preview of the selected background is displayed in the preview area next to the drop-down list. You need to select the **Save sun properties with view** check box to save the view with the sun properties.

Tip

When you select any of the preset orthographic views from the ***View*** *drop-down list, the UCS is also aligned to that view.*

Changing the Viewpoint Using the Viewpoint Presets Dialog Box

Menu Bar: View > 3D Views > Viewpoint Presets **Command:** DDVPOINT (VPOINT)

You can invoke the **Viewpoint Presets** dialog box using the **VPOINT** command for setting the viewpoint to view 3D models. The viewpoint is set with respect to the angle from the X axis and the angle from the XY plane. Figure 2-13 shows different view direction parameters.

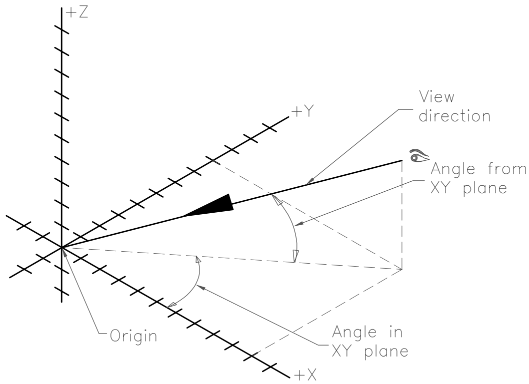


Figure 2-13 *View direction parameters*

Viewpoint Presets Dialog Box Options

The options in the **Viewpoint Presets** dialog box are discussed next, refer to Figure 2-14.

Absolute to WCS. This radio button is selected by default and used to set the viewpoint with respect to the world coordinate system (WCS).

Relative to UCS. This radio button is selected to set the viewpoint with respect to the current user coordinate system (UCS).

Note
The WCS and UCS have already been discussed in Chapter 1.

From: X Axis. This edit box is used to specify the angle in the *XY* plane from the *X* axis. You can directly enter the required angle in this edit box or select the desired angle from the image tile, refer to Figure 2-15. There are two arms in this image tile, and by default, they are placed one over the other. The gray arm displays the current viewing angle and the black one displays the new viewing angle. On selecting a new angle from the image tile, it is automatically displayed in the **X Axis** edit box. This value can vary between **0** and **359.9**.

From: XY Plane. This edit box is used to specify the angle from the *XY* plane. You can directly enter the desired angle in it or select the angle from the image tile, as shown in Figure 2-16. This value can vary from -90 to +90.

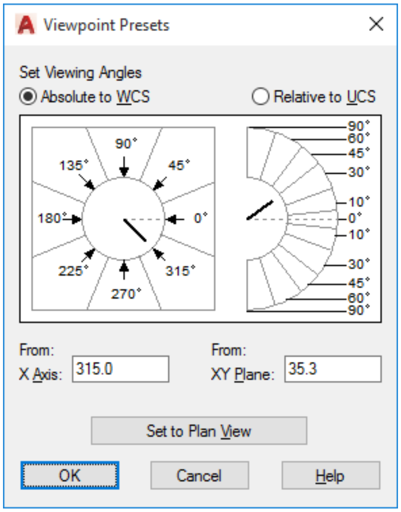


*Figure 2-14 The **Viewpoint Presets** dialog box*

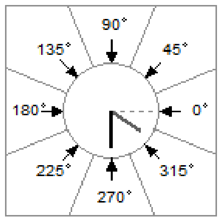


Figure 2-15 The image tile for selecting the angle from X axis

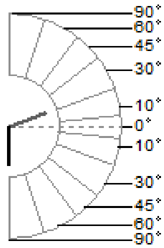


Figure 2-16 The image tile for selecting the angle from the XY plane

Set to Plan View. This button is chosen to set the viewpoint to the Plan view of the WCS or the UCS. If the **Absolute to WCS** radio button is selected, the viewpoint will be set to the Plan view of the WCS. If the **Relative to UCS** radio button is selected, the viewpoint will be set to the Plan view of the current UCS. Figures 2-17 and 2-18 show the 3D models from different viewing angles.

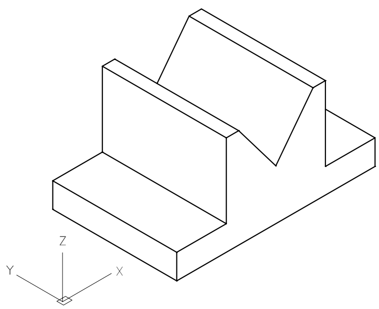


*Figure 2-17 Viewing the 3D model with the angle in the XY plane as **225** and the angle from the XY plane as **30***

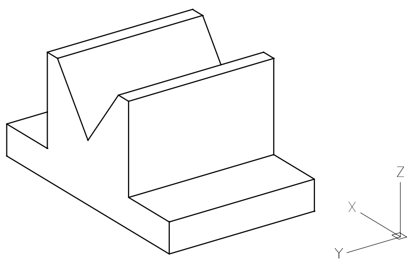


*Figure 2-18 Viewing the 3D model with the angle in the XY plane as **145** and the angle from the XY plane as **25***

Tip

1. If you set a negative angle from the XY plane, the Z axis will be displayed as a dotted line, indicating a negative Z direction, refer to Figures 2-19 and 2-20.

2. In case of confusion in identifying the direction of X, Y, and Z axes, use the right-hand thumb rule.

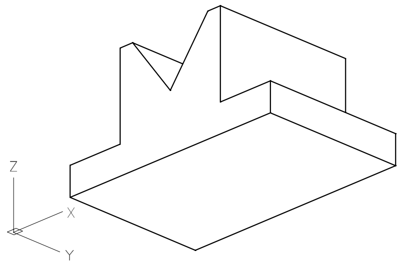


*Figure 2-19 Viewing the 3D model with the angle in the XY plane as **315** and the angle from the XY plane as **-25***

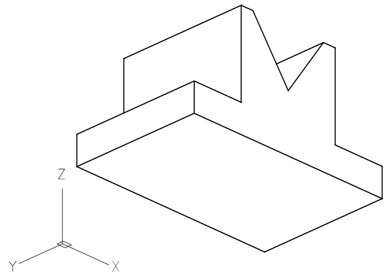


*Figure 2-20 Viewing the 3D model with the angle in the XY plane as **225** and the angle from the XY plane as **-25***

Changing the Viewpoint Using the -VPOINT Command

Menu Bar: View > 3D Views > Viewpoint **Command:** -VPOINT

The **-VPOINT** command is used to set the viewpoint for viewing the 3D models. Using this command, the users can specify a point in the 3D space that will act as the viewpoint.

Specifying a Viewpoint

This is the default option and is used to set a viewpoint by specifying its location (of viewer) using the *X*, *Y*, and *Z* coordinates of that particular point. AutoCAD follows a convention of the sides of the 3D model for specifying the viewpoint. The convention states that if the UCS is at the World position (default position), then

1. The side at the positive *X* axis direction will be taken as the right side of the model.
2. The side at the negative *X* axis direction will be taken as the left side of the model.
3. The side at the negative *Y* axis direction will be taken as the front side of the model.
4. The side at the positive *Y* axis direction will be taken as the back side of the model.
5. The side at the positive *Z* axis direction will be taken as the top side of the model.
6. The side at the negative *Z* axis direction will be taken as the bottom side of the model.

Some standard viewpoint coordinates and the view they display are given next.

Value	View	Value	View	Value	View
1,0,0	Right side	-1,0,0	Left side	0,1,0	Back
0,-1,0	Front	0,0,1	Top view	0,0,-1	Bottom view
1,1,1	NE Isometric	-1,1,1	NW Isometric	1,-1,1	SE Isometric
-1,-1,1	SW Isometric	1,1,-1	Right, Back, Bottom	-1,1,-1	Left, Back, Bottom
1,-1,-1	Right, Front, Bottom	-1,-1,-1	Left, Front, Bottom		

You can also enter the values in decimal, but the resultant views will not be the standard views. Figures 2-21 through 2-24 show a 3D model from different viewpoints.

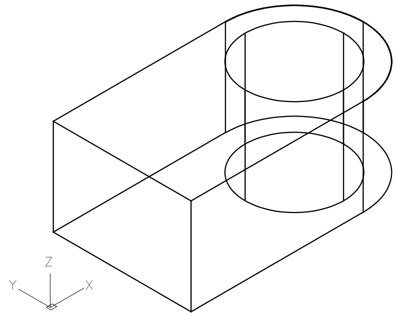


Figure 2-21 *Viewing the model from the viewpoint -1,-1,1*

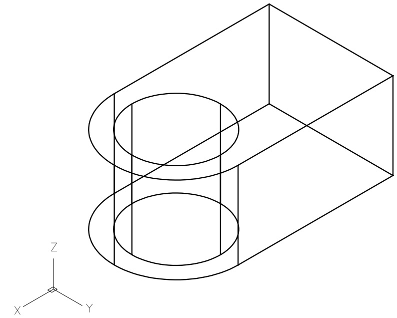


Figure 2-22 *Viewing the model from the viewpoint 1,1,1*

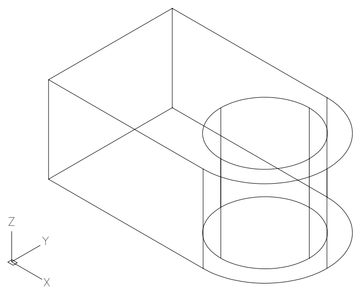


Figure 2-23 *Viewing the model from the viewpoint 1,-1,1*

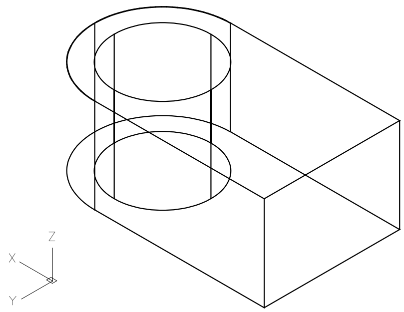


Figure 2-24 *Viewing the model from the viewpoint -1,1,1*

Compass and Tripod

When you choose **View > 3D Views > Viewpoint** from the menu bar, a compass and an axis tripod are displayed, as shown in Figure 2-25. You can directly set the viewpoint using this compass and can select any point on this compass to specify the viewpoint. The compass consists of two circles, a smaller circle and a bigger circle. Both these circles are divided into four quadrants: first, second, third, and fourth. The resultant view will depend upon the quadrant and the circle on which you select the point. In the first quadrant, both the X and Y axes are positive; in the second quadrant, the X axis is negative and the Y axis is positive; in the third quadrant, both the X and Y axes are negative; and in the fourth quadrant, the X axis is positive and the Y axis is negative. Now, if you select a point inside the inner circle, it will be in the positive Z axis direction. If you select the point outside the inner circle and inside the outer circle, it will be in the negative Z axis direction. Therefore, if the previously mentioned statements are added, the following is concluded.

1. If you select a point inside the smaller circle in the first quadrant, the resultant view will be the Right, Back, or Top view. If you select a point outside the smaller circle and inside the bigger circle in this quadrant, the resultant view will be the Right, Back, or Bottom view, refer to Figure 2-26.

2. If you select a point inside the smaller circle in the second quadrant, the resultant view will be the Left, Back, or Top view. If you select a point outside the smaller circle and inside

the bigger circle in this quadrant, the resultant view will be the Left, Back, or Bottom view, refer to Figure 2-26.

3. If you select a point inside the smaller circle in the third quadrant, the resultant view will be the Left, Front, or Top view. If you select a point outside the smaller circle and inside the bigger circle in this quadrant, the resultant view will be the Left, Front, or Bottom view, refer to Figure 2-26.

4. If you select a point inside the smaller circle in the fourth quadrant, the resultant view will be the Right, Front, or Top view. If you select a point outside the smaller circle and inside the bigger circle in this quadrant, the resultant view will be the Right, Front, or Bottom view, refer to Figure 2-26.

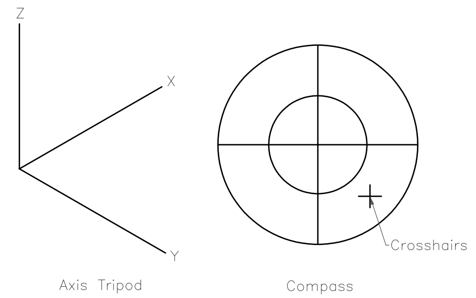


***Figure 2-25** The axis tripod and the compass*

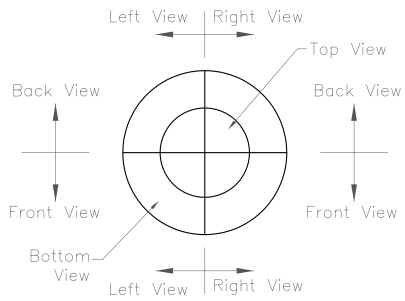


***Figure 2-26** The directions of the compass*

Rotate

This option is similar to setting the viewpoint in the **Viewpoint Presets** dialog box, which is displayed by invoking the **-VPOINT** command. When you select this option, you will be prompted to specify the angle in the *XY* plane and the angle from the *XY* plane. The angle in the *XY* plane will be taken from the *X* axis.

IN-CANVAS VIEWPORT CONTROL

The **In-canvas Viewport control** enables you to control the viewport as well as visual style. By default, the **In-canvas Viewport control** is displayed as **[-][Custom View][Realistic]** in the top left corner of the drawing area. The tools available in this viewport control are discussed next.

[-]

When you choose the **[-]** button in the **In-canvas Viewport control**, a flyout will be displayed, as shown in Figure 2-27. Options in this flyout are discussed next.

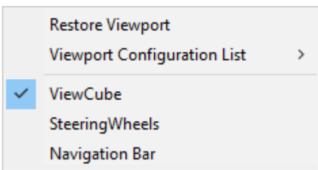


***Figure 2-27** The [-] flyout*

Restore Viewport

When you choose this option, the drawing screen is displayed with four equal partitions. In each of these partitions, a **[+]** button is displayed in the upper left corner. When you click on the **[+]** button, a flyout will be displayed with the **Maximize Viewport** option. Using this option, you can maximize the partition window.

Viewport Configuration List

When you choose this option, a flyout will be displayed, as shown in the Figure 2-28. In this flyout, 12 options are available for viewport settings. Using these options, you can set as many partitions as needed to be created and their position in the drawing area.

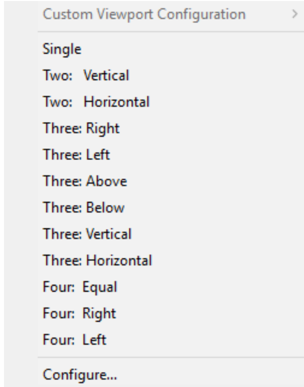


*Figure 2-28 The **Viewport Configuration List** flyout*

Viewcube, SteeringWheels, and Navigation Bar

ViewCube is used to minimize the time required in changing the view of the model, while the SteeringWheels and Navigation Bar are used to minimize the navigation time. Using the ViewCube from the Viewport Controls flyout, you can toggle the display of the ViewCube. As you choose the **SteeringWheels** option, SteeringWheels will get attached to the cursor. SteeringWheels will be discussed in detail later in this chapter. Using the **Navigation Bar** option, you can toggle the display of the Navigation Bar in the right of the drawing screen.

View Controls

The **View Controls** flyout contains various options to change your current view or to create a new view. When you choose the **Custom View** button from the **In-canvas Viewport controls**, this flyout will be displayed, as shown in Figure 2-29. The options in this flyout are used for setting view controls and are discussed next.

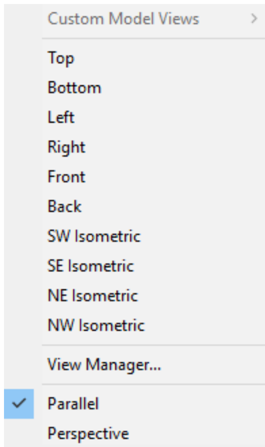


*Figure 2-29 The **View Controls** flyout*

- **Custom Model Views:** This flyout consists of custom views created by using the **View Manager**.
- **Views:** There are 10 view options available in this flyout to set 10 different views. When you change the view, the UCS gets reoriented according to the view selected.
- **View Manager:** Using this option, you can create a new view or edit the view settings of an existing view.
- **Parallel and Perspective:** These options are used to control the projection. If you choose the **Parallel** option, the drawing will be displayed parallel to the screen. If you choose the **Perspective** option, the drawing will be displayed in a perspective view.

Visual Style Controls

The **Visual Style Controls** flyout is used to control the visual style of the model. This flyout is displayed when you select the **Realistic** button from the **Visual Style Controls**, as shown in Figure 2-30. The options in this flyout are used for setting visual styles and are discussed next.

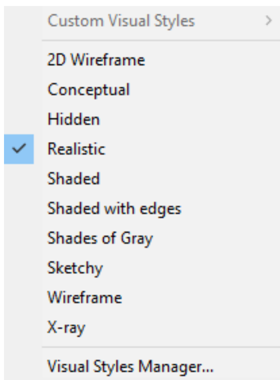


*Figure 2-30 The **Visual Style Controls** flyout*

- **Custom Visual Styles**. When you click on this option, a flyout is displayed. This flyout displays the visual styles that you have created.

- **Visual Styles**. There are 10 options that enable you to select the required visual style.

- **Visual Style Manager**. This option is used to create a new visual style as well as to change the settings of the current visual style.

3D COORDINATE SYSTEMS

Similar to 2D coordinate systems, there are two types of 3D coordinate systems. They are discussed next.

Absolute Coordinate System

This type of coordinate system is similar to the 2D absolute coordinate system in which the coordinates of the point are calculated from the origin (0,0). The only difference here is that, along with the *X* and *Y* coordinates, the *Z* coordinate is also included. For example, to draw a line in 3D space starting from the origin to a point say 10,6,6, the procedure to be followed is given next.

Command: *Choose the **Line** tool from the **Draw** panel.*
Specify first point: **0,0,0**
Specify next point or [Undo]: **10,6,6**
Specify next point or [Undo]: Enter

Figure 2-31 shows the model drawn using the absolute coordinate system.

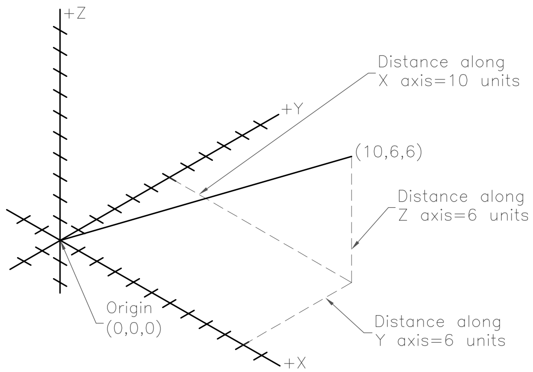


Figure 2-31 *Drawing a line from origin to 10,6,6*

Relative Coordinate System

The following are the three types of relative coordinate systems in 3D:

Relative Rectangular Coordinate System

This coordinate system is similar to the relative rectangular coordinate system of 2D, except that in 3D you also have to enter the *Z* coordinate along with the *X* and *Y* coordinates. The syntax of the relative rectangular system for the 3D is **@X coordinate, Y coordinate, Z coordinate**.

Relative Cylindrical Coordinate System

In this coordinate system, you can locate the point by specifying its distance from the reference point, the angle in the *XY* plane, and its distance from the *XY* plane. The syntax of the relative cylindrical coordinate system is **@Distance from the reference point in the XY plane < Angle in the XY plane from the X axis, Distance from the XY plane along the Z axis**. Figure 2-32 shows the components of the relative cylindrical coordinate system.

Relative Spherical Coordinate System

In this coordinate system, you can locate the point by specifying its distance from the reference point, the angle in the *XY* plane, and the angle from the *XY* plane. The syntax of the relative spherical coordinate system is **@Length of the line from the reference point < Angle in the XY plane from the X axis < Angle from the XY plane**. Figure 2-33 shows the components of the relative spherical coordinate system.

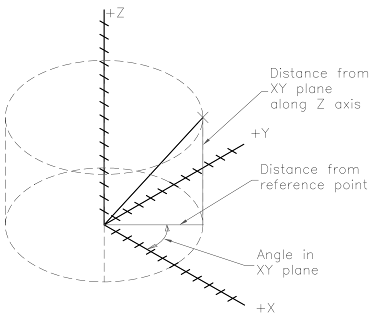


Figure 2-32 Various components of the relative cylindrical coordinate system

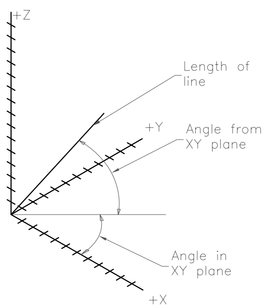


Figure 2-33 Various components of the relative spherical coordinate system

Tip

The major difference between the relative cylindrical and relative spherical coordinate systems is that in the relative cylindrical coordinate system, the specified distance is the distance from the reference point in the XY plane. On the other hand, in the relative spherical coordinate system, the specified distance is the total length of the line in the 3D space.

Example 1 — *Relative Coordinate System*

In this example, you will draw the 3D wireframe model shown in Figure 2-34. Its dimensions are given in Figure 2-35.

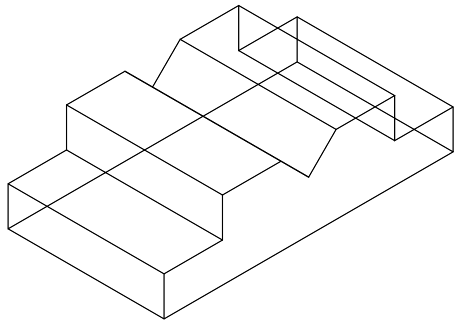


Figure 2-34 *The 3D Wireframe model for Example 1*

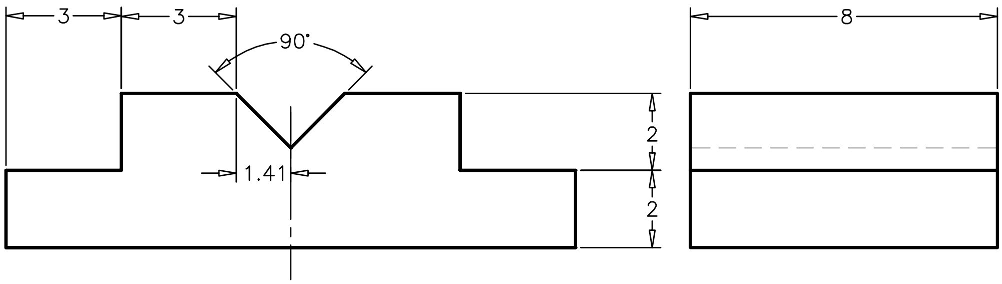


Figure 2-35 *Orthographic views of the Wireframe model*

1. Start a new *acad3D.dwt* file in the **3D Modeling** workspace and select the **SW Isometric** option from the **3D Navigation** drop-down list in the **View** panel of the **Home** tab. Alternatively, select the SouthWest top corner hotspot on the ViewCube, refer to the compass of the ViewCube to get the SouthWest view of the model.

2. As the start point of the model is not given, you can start the model from any point. Choose the **Line** tool from the **Draw** panel of the **Home** tab and follow the prompt sequence given next.

   Specify first point: **4,2,0**
   Specify next point or [Undo]: **@0,0,2**
   Specify next point or [Undo]: **@3,0,0**
   Specify next point or [Close/Undo]: **@0,0,2**
   Specify next point or [Close/Undo]: **@3,0,0**
   Specify next point or [Close/Undo]: **@2<0<315**
   Specify next point or [Close/Undo]: **@2<0<45**
   Specify next point or [Close/Undo]: **@3,0,0**

Specify next point or [Close/Undo]: **@0,0,-2**
Specify next point or [Close/Undo]: **@3,0,0**
Specify next point or [Close/Undo]: **@0,0,-2**
Specify next point or [Close/Undo]: **C**

3. Choose the **Line** tool again and follow the prompt sequence given next.

Specify first point: **4,2,0**
Specify next point or [Undo]: **@0,8,0**
Specify next point or [Undo]: **@0,0,2**
Specify next point or [Close/Undo]: **@3,0,0**
Specify next point or [Close/Undo]: **@0,0,2**
Specify next point or [Close/Undo]: **@3,0,0**
Specify next point or [Close/Undo]: **@2<0<315**
Specify next point or [Close/Undo]: **@2<0<45**
Specify next point or [Close/Undo]: **@3,0,0**
Specify next point or [Close/Undo]: **@0,0,-2**
Specify next point or [Close/Undo]: **@3,0,0**
Specify next point or [Close/Undo]: **@0,0,-2**
Specify next point or [Close/Undo]: **@0,-8,0**
Specify next point or [Close/Undo]: Enter

4. Complete the model by joining the remaining edges using the **Line** tool.

5. The final 3D model should look similar to the one shown in Figure 2-36.

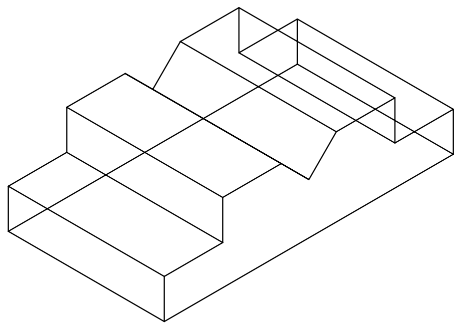


Figure 2-36 *The final 3D wireframe model for Example 1*

DIRECT DISTANCE ENTRY METHOD

In AutoCAD, you can directly create models in 3D space. This method is similar to that in 2D. The easiest way to draw a model by using the **Line** tool in 3D space of AutoCAD is by using the Direct Distance Entry method. Before drawing a model by using this method, ensure that the **Dynamic Input** button is chosen in the Status Bar. Next choose the **Line** tool; you will be prompted to specify the start point. Enter the coordinate values in the text box and press ENTER; you will be prompted to specify the next point. Move the cursor along the direction in which you need to draw the line and enter the absolute length of the line and its angle in the corresponding text boxes, with respect to the previous point. Note that you can use the TAB key to toggle between the text boxes.

To draw a line along the Z axis, move the cursor along the Z axis; you will notice that the tool tip displays **Ortho: (current length) < +Z**, if the ortho mode is on. Type the absolute distance and the angle, and press ENTER. If the ortho mode is not chosen, position the cursor at the desired angle, type the length at the Command prompt, and then press ENTER. If necessary change the UCS as discussed earlier.

Example 2 — *Direct Distance Entry*

In this example, you will draw the 3D wireframe model shown in Figure 2-37.

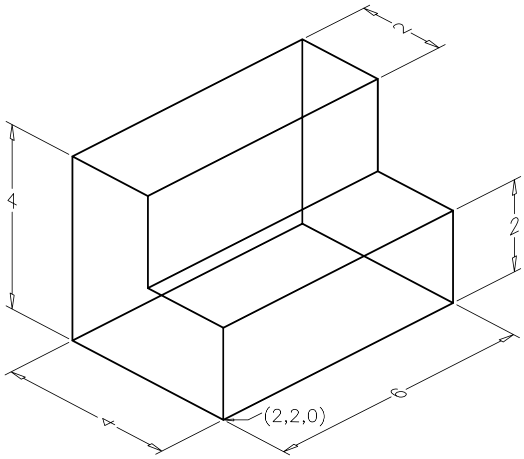


Figure 2-37 *3D wireframe model for Example 2*

1. Start a new *acad3D.dwt* file in the **3D Modeling** workspace and make sure the **Ortho Mode** and **Dynamic Input** buttons are chosen. Then, change the drawing view by using the ViewCube. The view needed is **SW Isometric**. The Southwest view of the ViewCube has three hotspots: bottom, edge, and top. You need to select the top hotspot to get the required view. Refer to Figure 2-38 for the SW Isometric hotspot. However, you can check different hotspots and view the model in different angles by using the ViewCube.

2. Choose the **Line** tool from the **Draw** panel and specify the start point at 2,2,0.

3. Move the cursor along the positive X axis. Then, type **6** and press ENTER; a line of length 6 units is drawn along the X axis.

4. Move the cursor along the positive Y axis, type **4**, and press ENTER; a line of length 4 units is drawn along the Y axis.

5. Draw the other two lines to create the closed profile at the bottom of the model.

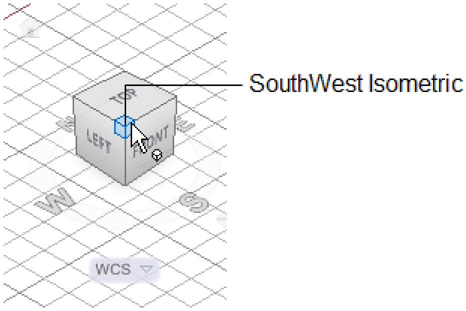


Figure 2-38 ViewCube showing the SW Isometric

6. From the start point of the model, move the cursor along the positive Z axis. You will notice that the tooltip displays **Ortho: (current length) < +Z**, as shown in Figure 2-39.

7. Type **2** at the Command prompt and press ENTER; a line of length 2 units is drawn.

8. Move the cursor along the positive X axis, type **6**, and press ENTER; a line of length 6 units is drawn.

9. Similarly, draw other lines to complete the model. The final model should look similar to the one shown in Figure 2-40.

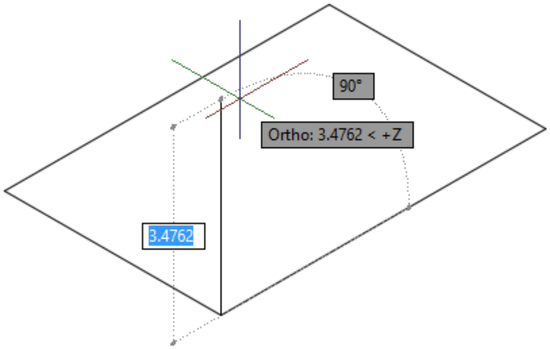


Figure 2-39 Tooltip displayed on moving the cursor along the positive Z axis

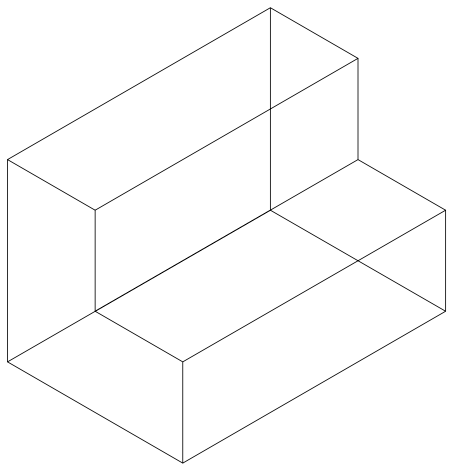


Figure 2-40 Final 3D model for Example 2

Exercise 1

In this exercise, you will draw the 3D wireframe model shown in Figure 2-41. Its dimensions are given in the same figure. You can start the model from any point.

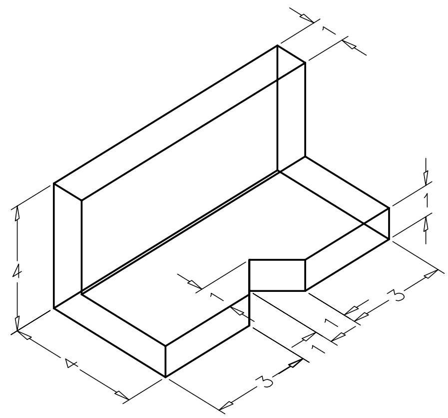


Figure 2-41 *Wireframe model for Exercise 1*

TRIM, EXTEND, AND FILLET TOOLS

In AutoCAD, you can use the trim, extend, and fillet tools in 3D workspace. You can also use the **PROJMODE** and **EDGEMODE** variables. The **PROJMODE** variable sets the projection mode for trimming and extending. The value of 0 implies **True 3D mode**, that is, no projection. In this case, the objects must intersect in 3D space to be trimmed or extended. It is similar to using the **None** option of the **Project** mode of the **Extend** and **Trim** tools. A value of 1 projects onto the *XY* plane of the current UCS and is similar to using the **UCS** option of the **Project** option of the **Trim** or **Extend** tools. A value of 2 projects onto the current view plane and is like using the **View** option of the **Project** option of the **Trim** or **Extend** tools. The value of the **EDGEMODE** system variable controls the cutting or trimming boundaries. A value of 0 considers the selected edge with no extension. You can also use the **No extend** option of the **Edge** option of the **Trim** or **Extend** tools. A value of 1 considers an imaginary extension of the selected edge. This is similar to using the **Extend** option of the **Edge** option of the **Trim** or **Extend** tools.

You can fillet coplanar objects whose extrusion directions are not parallel to the *Z* axis of the current UCS, by using the **Fillet** tool. The fillet arc exists on the same plane and has the same extrusion direction as the coplanar objects. If the coplanar objects to be filleted exist in opposite directions, the fillet arc will be on the same plane but will be inclined toward the positive *Z* axis.

SETTING THICKNESS AND ELEVATION FOR NEW OBJECTS

You can create the objects with a preset elevation or thickness using the **ELEV** command. This command is discussed next.

The ELEV Command

Command: ELEV

This is a transparent command and is used to set elevation and thickness for new objects. The following prompt sequence is displayed when you invoke this command:

Specify new default elevation <0.0000>: *Enter the new elevation value.*
Specify new default thickness <0.0000>: *Enter the new thickness value.*

Elevation

This option is used to specify the elevation value for new objects. Setting the elevation is nothing but moving the working plane from its default position. By default, the working plane is on the world *XY* plane. You can move this working plane by specifying the new value for elevation using the **ELEV** command. However, remember that the working plane can be moved only along the *Z* axis, refer to Figure 2-42. The default elevation value is 0.0 and you can set any positive or negative value. All objects that will be drawn hereafter will be with the specified elevation. The **ELEV** command sets the elevation only for the new objects and the existing objects are not modified using this option.

Thickness

This option is used to specify the thickness values for new objects. It can be considered as another method of creating surface models. Specifying the thickness creates extruded surface models. The thickness is always taken along the *Z* axis direction, refer to Figure 2-43. The 3D faces will be automatically applied on the vertical faces of the objects drawn with a thickness. The **Thickness** option sets the thickness only for the new objects and the existing objects are not modified.

Note

*By default, the model will be displayed in the **Realistic** visual style. To change it to wireframe, choose the **2D Wireframe** from the **Visual Style Controls** flyout in the **In-canvas Viewport Controls**. You can suppress the hidden edges in the model by using the **Hide** tool. The **Hide** tool is discussed in the next section.*

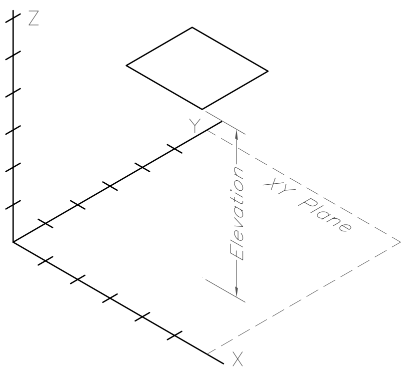


Figure 2-42 Object drawn with elevation

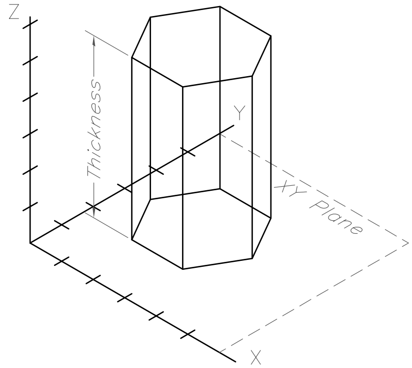


Figure 2-43 Object drawn with thickness

Tip

1. The elevation value will be reset to 0.0 when you change the coordinate system to WCS.

*2. The rectangles drawn using the **Rectangle** tool do not consider the thickness value set by using the **ELEV** command. To draw a rectangle with thickness, use the **Thickness** option of the **Rectangle** tool.*

3. Figure 2-44 shows a point, line, polygon, circle, and ellipse drawn with a thickness of 5 units.

4. To write text with thickness, first write a single line text and then change its thickness using any of the commands that modify its property, refer to Figure 2-45.

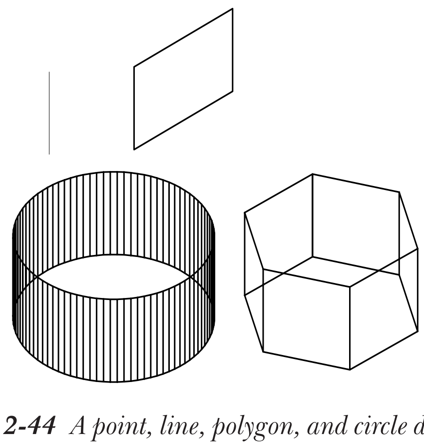


Figure 2-44 *A point, line, polygon, and circle drawn with a thickness of 5 units*

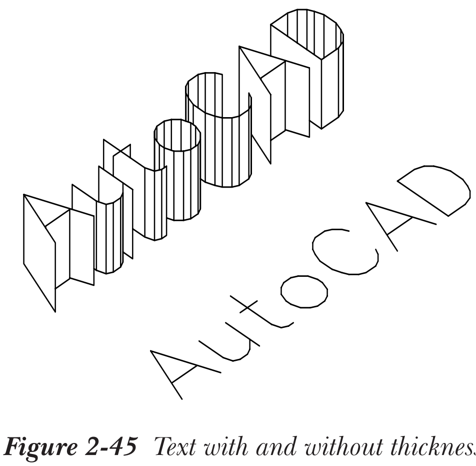


Figure 2-45 *Text with and without thickness*

SUPPRESSING THE HIDDEN EDGES

Menu Bar: View > Hide **Toolbar:** Render > Hide **Command:** HIDE

Whenever you create a surface or a solid model, the edges that lie at the back, called the hidden edges too, also become visible. As a result, the model appears like a wireframe model. You need to manually suppress the hidden lines in the 3D model using the **Hide** command. When you invoke this tool, all objects on the screen are regenerated. Also, the 3D models are displayed with the hidden edges suppressed. If the value of the **DISPSILH** system variable is set to **1**, the model is displayed only with the silhouette edges. The internal edges that have facets will not be displayed. The hidden lines are again included in the drawing when the next regeneration takes place. Figure 2-46 shows the surface models displaying the hidden edges. Figure 2-47 shows surface models without the hidden edges.

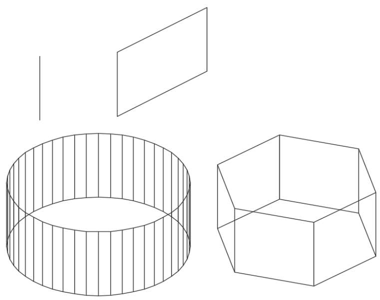


Figure 2-46 *3D models with hidden edges*

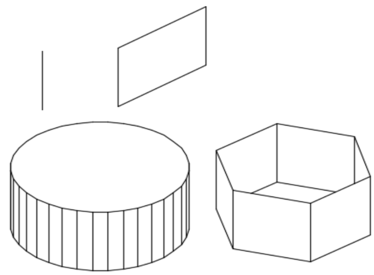


Figure 2-47 *Models without hidden edges*

> **Tip**
> *The **Hide** command considers Circles, Solids, Traces, Polylines with width, Regions, and 3D Faces as opaque surfaces that will hide objects.*

CREATING A 3D POLYLINE

Ribbon: Home > Draw > 3D Polyline **Menu Bar:** Draw > 3D Polyline
Command: 3DPOLY

The **3DPolyline** tool is used to draw straight polylines in a 2D plane or a 3D space. This tool is similar to the **Polyline** tool except the fact that you can draw polylines in a plane other than the *XY* plane also by using this tool. However, this tool does not provide the **Width** option or the **Arc** option; and therefore you cannot create a 3D polyline with a width or an arc. The **Close** and **Undo** options of this tool are similar to those of the **Polyline** tool.

> **Tip**
> *You can fit a spline curve about the 3D polyline using the **Edit Polyline** tool. Use the **Spline curve** option to fit the spline curve and the **Decurve** option to remove it.*

CONVERTING WIREFRAME MODELS INTO SURFACE MODELS

You can convert a wireframe model into a surface model using the **3DFACE** command and the **PFACE** command. These tools are discussed next.

Creating 3D Faces

Menu Bar: Draw > Modeling > Meshes > 3D Face	**Command:** 3DFACE

The **3D Face** tool is used to create 3D faces in space. You can create three-sided or four-sided faces using this tool. You can specify the same or a different Z coordinate value for each point of the face. On invoking this tool, you will be prompted to specify the first, second, third, and fourth points of the 3D face. Once you have specified the first four points, it will again prompt you to specify the third and fourth points. In this case, it will take the previous third and fourth points as the first and second points, respectively. This process will continue until you press ENTER at the **Specify third point or [Invisible] <exit>** prompt. Keep in mind that the points must be specified in the natural clockwise or the counterclockwise direction to create a proper 3D face. Figure 2-48 shows 3D faces created on the 3D wireframe model.

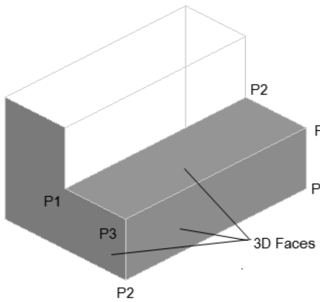


Figure 2-48 *3D faces created on the 3D wireframe model*

However, in case of complex wireframe models, refer to Figure 2-49, it is not easy to apply 3D faces. This is because when you apply a 3D face, it generates edges about all points that you specify. These edges will be displayed on the wireframe model, as shown in Figure 2-49. You can avoid these unwanted visible edges by applying the 3D faces with invisible edges using the **Invisible** option. It is very important to note here that the edge that will be created after you enter **I** at the **Specify next point or [Invisible] <exit>** prompt will not be the invisible edge. The edge that will be created after this edge will be the invisible edge. Therefore, to make the P3 edge and P4 edge invisible, specify P1 as the first point and P2 as the second point. Before specifying P3 as the third point, enter **I** at the **Specify third point or [Invisible] <exit>** prompt. Similarly, follow the same procedure for creating the other invisible edges, refer to Figure 2-50.

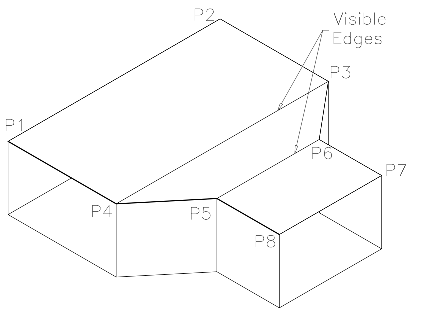


***Figure 2-49** Wireframe model with edges*

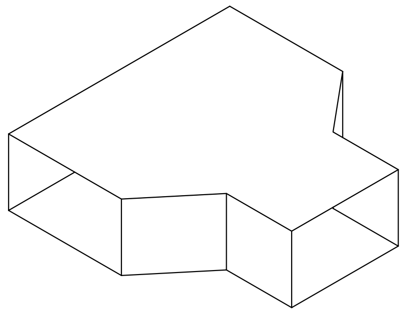


***Figure 2-50** Wireframe model without edges*

Creating Polyface Meshes

Command: PFACE

The **PFACE** command is similar to the **3D Face** tool. This command also allows you to create a mesh of any desired surface shape by specifying the coordinates of the vertices and assigning the vertices to the faces in the mesh. The difference between them is that on using the **3D Face** tool, you do not select the vertices that join another face twice. With the **PFACE** command, you need to select all the vertices of a face, even if they are coincident with the vertices of another face. In this way, you can avoid generating unrelated 3D faces that have coincident vertices. Also, with this command, there is no restriction on the number of faces and vertices the mesh can have.

Command: **PFACE**
Specify location for vertex 1: *Specify the location of the first vertex*.
Specify location for vertex 2 or <define faces>: *Specify the location of the second vertex*.
Specify location for vertex 3 or <define faces>: *Specify the location of the third vertex*.
Specify location for vertex 4 or <define faces>: *Specify the location of the fourth vertex*.
Specify location for vertex 5 or <define faces>: *Specify the location of the fifth vertex*.
Specify location for vertex 6 or <define faces>: *Specify the location of the sixth vertex*.
Specify location for vertex 7 or <define faces>: *Specify the location of the seventh vertex*.
Specify location for vertex 8 or <define faces>: *Specify the location of the eighth vertex*.
Specify location for vertex 9 or <define faces>: Enter

After defining the locations of all the vertices, press ENTER and assign vertices to the first face.

Face 1, vertex 1: *Specify the location of the first vertex*.
Enter a vertex number or [Color/Layer]: **1**
Face 1, vertex 2:
Enter a vertex number or [Color/Layer] <next face>: **2**
Face 1, vertex 3:
Enter a vertex number or [Color/Layer] <next face>: **3**
Face 1, vertex 4:
Enter a vertex number or [Color/Layer] <next face>: **4**

Once you have assigned the vertices to the first face, give a null response at the next prompt. Similarly, assign vertices to all faces, refer to Figure 2-51. To view the faces, select the **Realistic** option from the **Visual Styles** drop-down list in the **View** panel of the **Home** tab. To make an edge invisible, specify a negative number for its first vertex. The display of the invisible edges of 3D solid surfaces is governed by the **SPLFRAME** system variable. If **SPLFRAME** is set to 0, invisible edges are not displayed. If this variable is set to a number other than 0, all invisible edges are displayed after regeneration using the **REGENALL** command.

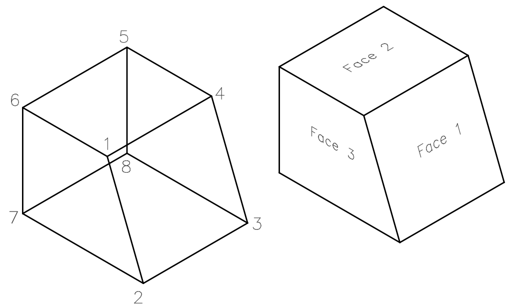


Figure 2-51 *Faces of block and assigned vertices*

Controlling the Visibility of the 3D Face Edges

Command: EDGE

The **EDGE** command is used to control the visibility of the edges created by using the **3D Face** tool. You can hide the edges of the 3D faces or display them using this command, refer to Figures 2-52 and 2-53. On invoking this command, you will be prompted to specify the 3D face edge to toggle its visibility or display. To hide an edge, select it. To display the edges, enter **D** at the **Specify edge of 3dface to toggle visibility or [Display]** prompt. You can display all invisible 3D face edges or the selected edges using this option.

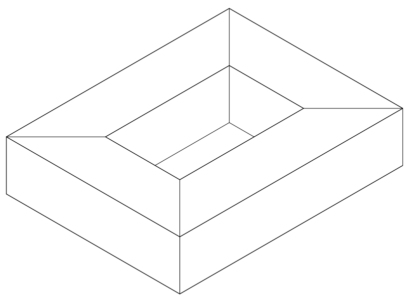


Figure 2-52 *Model with the visible 3D face edges*

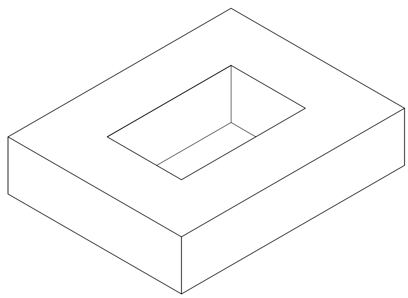


Figure 2-53 *Model after hiding the edges*

Exercise 2 — 3D Face

In this exercise, you will apply the 3D faces to the 3D wireframe model created in Example 2.

CREATING PLANAR SURFACES

Ribbon: Surface > Create > Planar **Toolbar:** Modeling > Planar Surface
Command: PLANESURF

The **Planar** tool is used to generate 2D surfaces on the working plane by specifying two diagonally opposite points. These two points specify the rectangular shape area to be covered by this planar surface, refer to Figure 2-54. The prompt sequence displayed on choosing the **Planar** tool from the **Create** panel in the **Surface** tab is discussed next.

Specify first corner or [Object] <Object>: *Specify the first corner point of the planar surface.*
Specify other corner: *Specify the diagonally opposite point of the planar surface.*

You can also convert an existing object to a surface by entering **O** at the **Specify first corner or [Object] <Object>** prompt. While selecting the object, you can directly select a closed boundary or a number of individual objects that result in a closed boundary, refer to Figure 2-55.

The number of lines displayed in the surface is controlled by the **SURFU** and **SURFV** system variables. The **SURFU** variable controls the number of lines to be displayed in the M direction and the **SURFV** variable controls the number of lines to be displayed in the N direction. The M and N directions are taken parallel to X and Y axis of the UCS respectively.

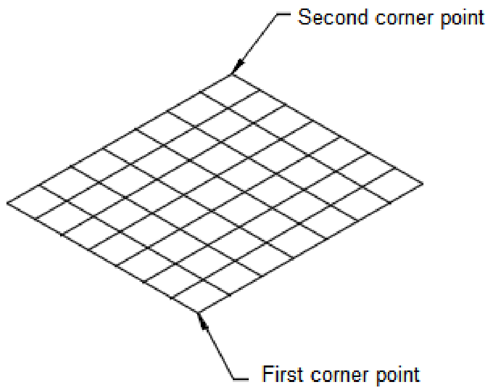


Figure 2-54 Planar surface created by specifying corner points

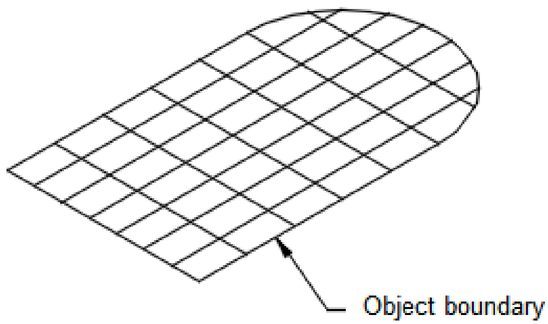


Figure 2-55 Planar surface created by selecting an object

THE 3DMESH COMMAND

Command: 3DMESH

The **3DMESH** command is used to create a free-form polygon mesh by using a matrix of M X N size. When you invoke this command, you will be prompted to specify the size of the mesh in the M direction and the N direction, where M X N is the total number of vertices in the mesh. The values of M and N can be 2 to 256. Once you have specified the size, you need to specify the coordinates of each and every vertex individually. For example, if the size of the mesh in the M and N directions is 12 X 12, then you will have to specify the coordinates of all the 144 vertices. You can specify any 2D or 3D coordinates for the vertices. Figure 2-54 shows a freeform mesh. This command is mostly used for programming. The 3D polygon mesh is always open in the M and N directions. You can edit this mesh or the mesh created using the **3D** command or the **Edit Polyline** tool.

EDITING THE SURFACE MESH

The polyface or polygon meshes can be edited using the **Edit Polyline** tool as discussed next.

The Edit Polyline Tool

Ribbon: Home > Modify > Edit Polyline	**Menu Bar:** Modify > Object > Polyline
Toolbar: Modify II > Edit Polyline	**Command:** PEDIT

To edit a polygon mesh, invoke the **Edit Polyline** tool from the **Modify** panel. You can also select the **Polyline Edit** option from the shortcut menu displayed on selecting the surface mesh and right-clicking on it, to edit a polygon face. The prompt sequence that will be followed on choosing the **Edit Polyline** tool and selecting the mesh is given next.
Select polyline or [Multiple]: *Select the 3D mesh.*

Enter an option [Edit vertex/Smooth surface/Desmooth/Mclose/Nclose/Undo]: *Select an option.*

Edit Vertex

This option is used for the individual editing of vertices of the mesh in the M direction or the N direction. On invoking this option, a cross will appear on the first point of the mesh and the following sub-options will be provided. Change the visual style to Wireframe by selecting the **Wireframe** option from the **Visual Styles** drop-down list in the **View** panel of the **Home** tab, so that you can view the vertices clearly.

Next. This sub-option is used to move the cross to the next vertex.

Previous. This sub-option is used to move the cross to the previous vertex.

Left. This sub-option is used to move the cross to the previous vertex in the N direction of the mesh.

Right. This sub-option is used to move the cross to the next vertex in the N direction of the mesh.

Up. This sub-option is used to move the cross to the next vertex in the M direction of the mesh.

Down. This sub-option is used to move the cross to the previous vertex in the M direction of the mesh.

Move. This sub-option is used to redefine the location of the current vertex. You can define a new location by using any of the coordinate systems or can directly select the location on the screen, refer to Figures 2-56 and 2-57.

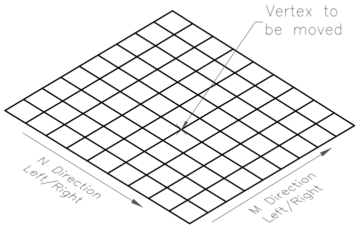


Figure 2-56 Vertex before moving

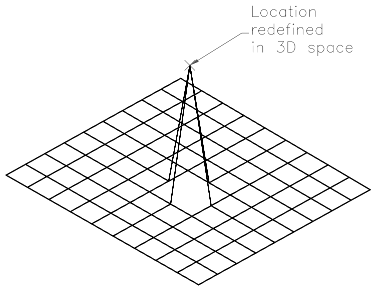


Figure 2-57 Vertex after moving

REgen. This sub-option is used to regenerate the 3D mesh.

eXit. This sub-option is used to exit the **Edit Vertex** option.

Smooth surface

This option is used to smoothen the surface of the 3D mesh by fitting a smooth surface, as shown in Figures 2-58 and 2-59. The smoothness of the surface will depend upon the **SURFU** and **SURFV** system variables. The value of these system variables can vary between 0 and 200.

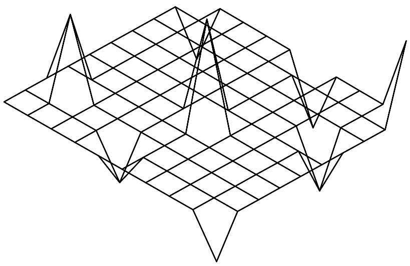


Figure 2-58 3D mesh before fitting the smooth surface

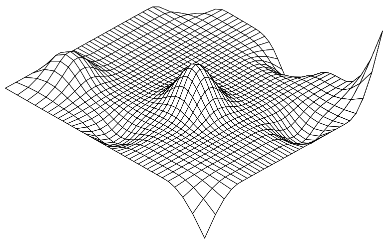


Figure 2-59 3D mesh after fitting the smooth surface

The type of the smooth surface to be fitted is controlled by the **SURFTYPE** system variable. If you set the value of the **SURFTYPE** system variable to 5, it will fit a Quadratic B-spline surface. If you set the value to 6, it will fit a Cubic B-spline surface. If you set the value to 8, it will fit a Bezier surface.

Desmooth

This option is used to remove the smooth surface that was fitted on the 3D mesh using the **Smooth surface** option and restore to the original mesh.

Mclose

This option is used to close the 3D mesh in the M direction, as shown in Figures 2-60 and 2-61.

Nclose

This option is used to close the 3D mesh in the N direction, as shown in Figures 2-60 and 2-61.

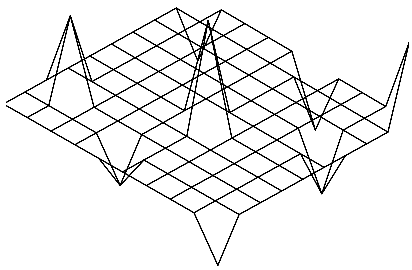


Figure 2-60 3D mesh open in the M and N directions

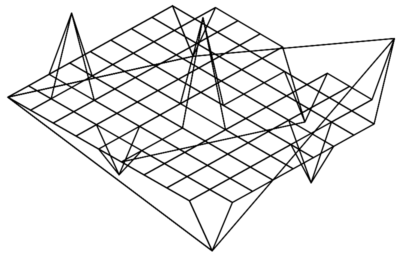


Figure 2-61 3D mesh closed in the M and N directions

Mopen
This option will be available if the 3D mesh is closed in the M direction. It is used to open the 3D mesh in the M direction, refer to Figures 2-60 and 2-61.

Nopen
This option will be available if the 3D mesh is closed in the N direction. It is used to open the 3D mesh in the N direction, refer to Figures 2-60 and 2-61.

Undo
This option is used to undo all the operations done by the **Edit Polyline** tool.

DYNAMIC VIEWING OF 3D OBJECTS

Ribbon: View > Navigate > SteeringWheels (Customize to add)
Navigator Toolbar: Full Navigation Wheel **Command:** NAVSWHEEL

The options available in the **Full Navigation Wheel** make the entire process of the solid modeling very interesting. You can dynamically rotate the solid models on the screen to view them from different angles. You can also define clipping planes and then rotate the solid models such that whenever the models pass through the cutting planes, they are sectioned just to be viewed. The original model can be restored as soon as you exit these commands or tools. The tools and their options that are used to perform all these functions are discussed next.

Using the SteeringWheels
The SteeringWheels enables you to easily view and navigate through your 3D models. It is an ideal tool for navigating through your models. It includes not only the common navigating tools such as **Zoom**, **Pan**, and **Orbit** but also an option to move the user specified center of the model to the center of the display window. Using this tool, you can also rewind and get back your previous views. It is a time saving tool, since it combines many common navigation tools into a single tool.

The SteeringWheel is divided into different sections known as Wedges. Each wedge on a wheel represents a command. You can activate a particular navigation option by clicking on the respective wedges and then modify the view of the model by dragging the cursor, refer to Figure 2-62. The navigation tools available in the SteeringWheels are discussed next.

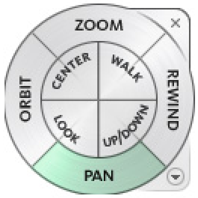


Figure 2-62 The SteeringWheel

ZOOM

This wedge is used in the same way as you use camera's zoom lens. It makes the object appear closer or far away, without changing the position of the camera. To invoke the **ZOOM** tool, move the cursor to the **ZOOM** wedge, press and hold left mouse button; a magnifier will be attached to the cursor that allows you to zoom in or out with reference to a base point. Now, drag to zoom in and zoom out. However, you can change the base point or pivot point to zoom in or zoom out of the model by choosing the **CENTER** wedge. You can even enable the incremental zoom of 25 percent on a single click from the **Zoom** area of the **SteeringWheels Settings** dialog box. To invoke this dialog box, right-click on the SteeringWheels; a shortcut menu will be displayed. Choose the **SteeringWheels Settings** option from the shortcut menu; the **SteeringWheels Settings** dialog box will be displayed. You can even choose the **Fit to Window** option from the shortcut menu of the SteeringWheels to view the entire drawing in the viewport.

REWIND

This wedge enables you to quickly restore the previous views. To invoke this tool, move the cursor to the **REWIND** wedge and click; a series of frames from the previous view orientation are displayed. You can now quickly pick the required view and control the thumbnail previews generated on changing various views using the **Rewind thumbnail** area of the **SteeringWheels Settings** dialog box.

PAN

This wedge allows you to drag the view to a new location in the drawing window. To invoke this tool, move the cursor to the **PAN** wedge, press and hold left mouse button; a plus arrow is attached to the cursor, enabling you to drag the view in any direction. Now, drag your mouse to pan in model space. You can choose the **Fit to Window** option from the shortcut menu of the SteeringWheels to view the entire drawing in the viewport.

ORBIT

This wedge allows you to visually maneuver around the 3D objects to obtain different views. When you move your cursor to the **ORBIT** wedge press and hold left mouse button; a circular arrow is attached to the cursor. Now, you can rotate the model with respect to a base point. Similar to the **ZOOM** option, in this case also, you can specify the required center point of the view and rotate the model using the **CENTER** wedge option. You can also set the viewpoint upside down using the options in the **SteeringWheels Settings** dialog box. Moreover, you can choose the **Fit to Window** option to view the full drawing view.

CENTER

This wedge helps you specify the pivot or center point of the view to orbit or zoom a model. When you move your cursor to this wedge and drag, a pivot ball is attached to the cursor. You can now adjust your center or target point by placing the cursor at the required point. However, you can restore the center of the viewport by choosing the **Restore Original Center** option from the shortcut menu of the SteeringWheels. You can also restore the home view of the model by choosing the **Go Home** option from the shortcut menu.

LOOK

This tool works similar to turning the camera on a tripod without changing the distance between the camera and the target. This means the source remains the same, but the target changes. To invoke this tool, move the cursor to the **LOOK** wedge; an arc is attached to the cursor which enables you to change your view direction. You can also invert the vertical direction of the **Look** tool by selecting the **Invert vertical axis for Look tool** check box from the **SteeringWheels Settings** dialog box. You can choose the **Fit to Window** option to view the entire drawing in the viewport.

WALK

This tool moves the camera around the model as if you are walking away or towards the focus direction of the camera. In the walk 3D navigation, the camera moves parallel to the XY plane and gives the effect as if you are walking. To invoke this tool, move the cursor to the **WALK** wedge. An arrow is attached to the cursor showing the direction of the camera. You can drag the cursor to move around the model and also control the walk speed using the options in the **Walk Tool** area of the **SteeringWheels Settings** dialog box. Using this dialog box, you can also constraint the walk angle to the ground plane.

UP/DOWN

This option slides the view along the Y-axis of the screen, which is like being in an elevator. When you move your cursor to the **UP/DOWN** wedge, a vertical slider is attached to the cursor. You can slide from up to down in the slider to view the model from up to down. Using the options in the shortcut menu of the SteeringWheels, you can go back to the original view, restore the original center, or fit the entire drawing in the viewport.

Modes of SteeringWheels

There are four working modes of SteeringWheels, refer to Figure 2-63. These working modes are discussed next.

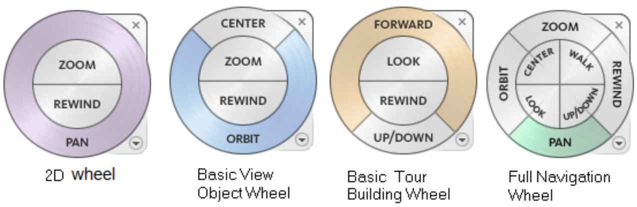


***Figure 2-63** Different modes of the SteeringWheels*

2D Wheel

You can invoke the SteeringWheel that has only the 2D navigation tools. To do so, choose the **2D** tool from **View > Navigate > SteeringWheels** drop-down; the 2D SteeringWheel with the **PAN**, **ZOOM**, and **REWIND** tools will be displayed.

Basic View Object Wheel

This mode contains only the viewing tools such as **CENTER**, **ZOOM**, **REWIND**, and **ORBIT**. You can invoke this mode by choosing **Basic Wheels > View Object Wheel** from the shortcut menu of the SteeringWheels or by choosing **Basic View Object Wheel** from the flyout that is displayed on clicking the down-arrow in the **SteeringWheels** group in the **Navigation bar**.

Basic Tour Building Wheel

This mode is ideal for navigating inside a building. It contains the **FORWARD**, **REWIND**, **LOOK**, and **UP/DOWN** tools. You can invoke this mode by choosing **Basic Wheels > Tour Building Wheel** from the shortcut menu of the SteeringWheels or by choosing **Mini Tour Building Wheel** from the flyout that is displayed on clicking on the down-arrow in the **SteeringWheels** group in the **Navigation bar**.

Full Navigation Wheel

This mode contains both the view and the navigating tools. It enables you to select a variety of tools from a single wheel. You can invoke this mode by choosing the **Full Navigation Wheel** option from the shortcut menu of the SteeringWheels or from the **Navigation bar**.

The SteeringWheels are of two types: Big Wheels and Mini Wheels. Both the wheels have the same functioning modes. The Big Wheels mode is displayed by default, whereas the Mini Wheels mode is invoked by choosing the **Mini View Object Wheel**, **Mini Tour Building Wheel**, or **Mini Full Navigation Wheel** option from the shortcut menu of the SteeringWheels or by choosing option from the **SteeringWheels** drop-down in the **Navigation bar** or by choosing option from the **SteeringWheels** drop-down in the **Navigate** panel of the **View** tab. You can customize the size, appearance, and behavior of the Steering Wheel using the **SteeringWheels Settings** dialog box, refer to Figure 2-64.

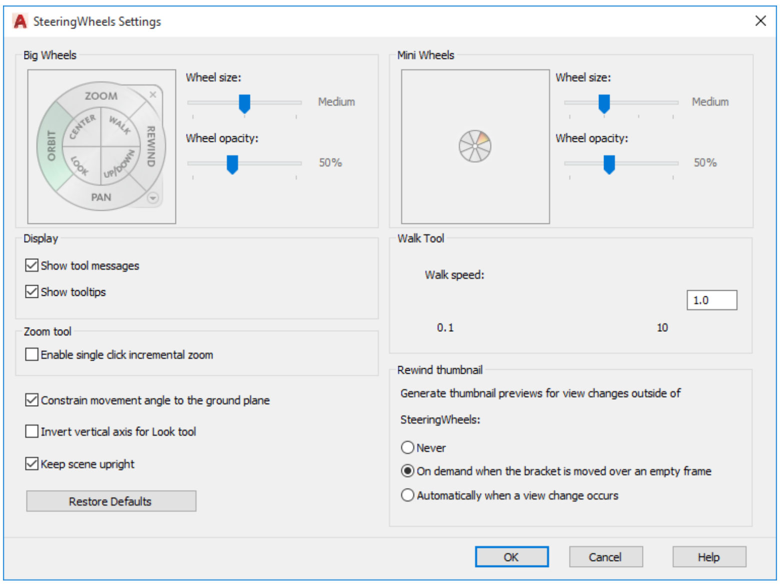


*Figure 2-64 The **SteeringWheels Settings** dialog box*

You can control the size and opacity of the Big Wheels and Mini Wheels with the help of a slider available next to the wheels. Also, using this dialog box you can control the tooltips and messages that are displayed when you move the cursor over any wedge.

Dynamically Rotating the View of a Model

Ribbon: View > Navigate > Orbit **Navigation Bar:** Orbit
Toolbar: 3D Navigation > Constrained Orbit **Command:** 3DO(3DORBIT)

The **Orbit** tool allows you to visually maneuver around 3D objects to obtain different views. This is one of the most important tools for the advanced 3D viewing options. All other advanced 3D viewing options can be invoked inside this tool or using the **3D Navigation** toolbar. This tool activates a 3D Orbit view in the current viewport. You can click and drag your pointing device to view the 3D object from different angles. In 3D orbit viewing, the target is considered stationary and the camera location is considered to be moving around it. The cursor looks like a sphere encircled by two arc shaped arrows. This is known as **Orbit mode** cursor, and clicking and dragging the pointing device allows you to rotate the view freely. You can move the **Orbit mode** cursor horizontally, vertically, and diagonally. If you drag the pointing device horizontally, the camera will move parallel to the XY plane of the WCS. If you move your pointing device vertically, then the camera will move along the Z axis. This tool is a transparent tool and can be invoked inside any other tool. You can select individual objects or the entire drawing to view before you invoke the **Orbit** tool.

Tip
Press and hold the SHIFT key and the middle mouse button to temporarily enter the constrained orbit mode.

You can invoke the other advanced viewing options using the shortcut menu that is displayed on right-clicking in the drawing window when you are inside the **Orbit** tool. The shortcut menu with the other advanced viewing options is shown in Figure 2-65. The options of the shortcut menu are discussed next.

Exit

This option is used to exit the **Orbit** tool. You can also exit this tool by pressing ESC.

Current Mode

This option displays the currently active navigation mode in which you obtained this shortcut menu.

Other Navigation Modes

This option enables you to choose the other navigation modes of the **3D Navigation** toolbar. In this way, you can toggle between the navigation modes without exiting the tool. The sub-options provided by the **Other Navigation Modes** are as follows:

Free Orbit. This sub-option can be invoked by choosing the **Free Orbit** tool from **View > Navigate > Orbit** drop-down in the **View** panel of the **Ribbon**, refer to Figure 2-66; 3D orbit view will be activated in the current viewport. When you invoke the **Free Orbit** tool, an arcball appears. This arcball is a circle with four smaller circles placed such that they divide the bigger circle into quadrants. The UCS icon is replaced by a shaded 3D UCS icon. In the 3D orbit viewing, the target is considered stationary and the camera location is considered to be moving around it. The target is the center of the arcball. The **Free Orbit** tool is a transparent tool and can be invoked inside any other tool.

The cursor icon changes as you move the cursor over the 3D Orbit view. The different icons indicate the different directions in which the view is being rotated. They are as follows.

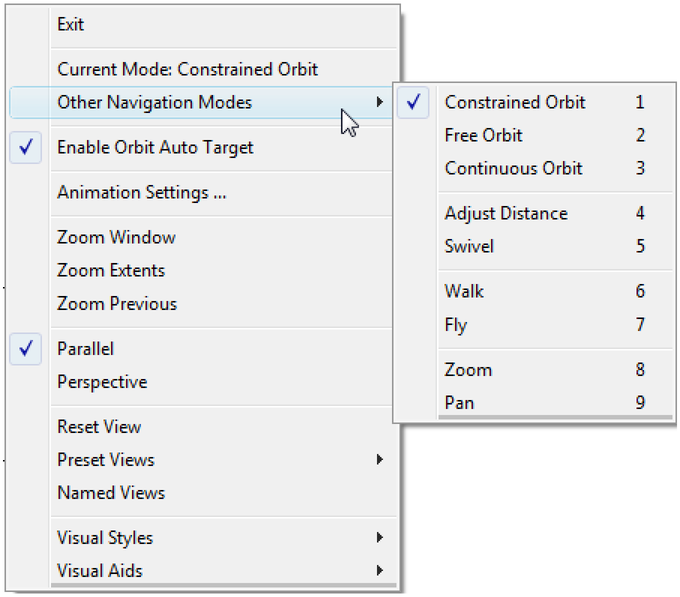


Figure 2-65 The shortcut menu displayed when the ***3DORBIT*** *command is active*

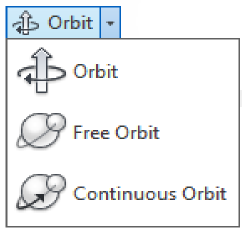
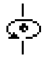


Figure 2-66 The tools in the ***Orbit*** *drop-down*

Orbit mode. When you move the cursor within the arcball, the cursor looks like a sphere encircled by two lines. This icon is the **Orbit mode** cursor, and clicking and dragging the pointing device allows you to rotate the view freely as if the cursor was grabbing a sphere surrounding the objects and moving it around the target point. You can move the **Orbit mode** cursor horizontally, vertically, and diagonally.

Roll mode. When you move the cursor outside the arcball, it changes to look like a sphere encircled by a circular arrow. When you click and drag, the view is rotated around an axis that extends through the center of the arcball, perpendicular to the screen.



Orbit Left-Right. When you move the cursor to the small circles placed on the left and right of the arcball, it changes into a sphere surrounded by a horizontal ellipse. When you click and drag, the view is rotated around the *Y* axis of the screen.

Orbit Up-down. When you move the cursor to the circles placed in the top or bottom of the arcball, it changes into a sphere surrounded by a vertical ellipse. Clicking and dragging the cursor rotates the view around the horizontal axis or the *X* axis of the screen, and passes it through the middle of the arcball.

Figure 2-67 shows a model being rotated using the **Free Orbit** tool.

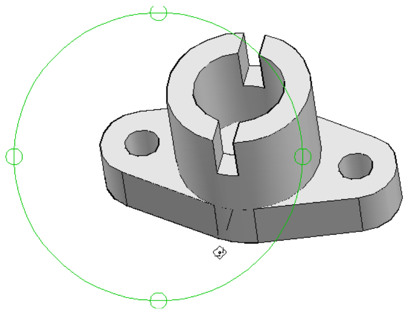


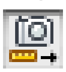
Figure 2-67 *Model being rotated using the* ***Free Orbit*** *tool*

Tip

1 . Press and hold the SHIFT+CTRL keys and the middle button of the mouse to temporarily enter the ***Free Orbit*** *navigation mode.*

2 . If you are in the ***3D Orbit*** *navigation mode, press and hold the SHIFT key to temporarily enter the* ***Free Orbit*** *navigation mode.*

Continuous Orbit. This tool allows you to set the objects you select in a 3D view into continuous motion in a free-form orbit. You can invoke this tool by choosing the **Continuous Orbit** tool from **View > Navigate > Orbit** drop-down. While using this tool, the cursor icon changes to the continuous orbit cursor. Clicking in the drawing area and dragging the pointing device in any direction starts to move the object(s) in the direction you have dragged it. When you release the pointing device button, that is, stop clicking and dragging, the object continues to move in its orbit in the direction you specified. The speed at which the cursor is moved determines the speed at which the objects spin. You can click and drag again to specify another direction for rotation in the continuous orbit. While using this sub-option, you can right-click in the drawing area to display the shortcut menu and choose the other sub-options without exiting the tool. Choosing **Pan**, **Zoom**, **Orbit**, **Adjust Clipping Plane** ends the continuous orbit.



Adjust Distance. This sub-option can be invoked by choosing **Other Navigation Modes > Adjust Distance** from the shortcut menu displayed on right-clicking in the drawing area. Alternatively, choose **View > Camera > Adjust Distance** from the menu bar. This sub-option is similar to taking the camera closer to or farther away from the target object. Therefore, it makes the object appear closer or farther away. You can press the left button and drag the mouse to adjust the distance between the target and the camera. This option is used to reduce the distance between the camera and the object when the object rotates by a large angle.

Swivel Camera. This sub-option can be invoked by choosing **View > Camera > Swivel** from the menu bar. It works similar to turning the camera on a tripod without changing the distance between the camera and the target. You can press the left mouse button and drag the mouse to swivel the camera.

Walk. This sub-option can be invoked by choosing the **Walk** tool from **Visualize > Animations> Walk** drop-down after customizing. It is similar to the **Walk** option in the SteeringWheel. This sub-option is used to move the camera around a model with the help of the keyboard, as if you are walking away or toward the focus direction of the camera. In walk 3D navigation, the camera moves parallel to the XY plane and this seems as if you are walking. Choose the **Walk** button; the **POSITION LOCATOR** window, showing the top view of the model with the location of the camera indicated by a red spot, will be displayed. The **POSITION LOCATOR** window shows a dynamic preview of the location and movement of the camera with respect to the object, refer to Figure 2-68. The following keys are used to control the location of the camera with respect to the object.

Motion Type	**Key**
Move the camera in the forward direction	UP ARROW or W key
Move the camera in the backward direction	DOWN ARROW or S key
Move the camera in the left direction	LEFT ARROW or A key
Move the camera in the right direction	RIGHT ARROW or D key
Change the focus direction of the camera	Press and hold the left mouse button in the drawing area and drag the mouse in the desired direction
Change the orientation of the XY plane	Press and hold the middle mouse button in the drawing area and drag the mouse in the desired direction
Change the distance of the camera	Move the cursor to the green triangular area in the **POSITION LOCATOR** window and pan to the desired location
Change the coverage area of the camera	Move the cursor onto the camera in the **POSITION LOCATOR** window and pan the camera

Fly. This sub-option can be invoked by choosing the **Fly** tool from **Visualize > Animations > Walk** drop-down after customizing. This sub-option is used to move the camera through a model with the help of the keyboard as if you are flying away or toward the focus direction, refer to Figure 2-68 of the camera. In the fly 3D navigation, the camera can even move at an inclination with the XY plane and this seems as if you are flying around the object. The **POSITION LOCATOR** window will be displayed on the screen while using the **Fly** tool also. The position of the camera with respect to the object can be controlled by using the same procedure as discussed in the **Walk** tool.

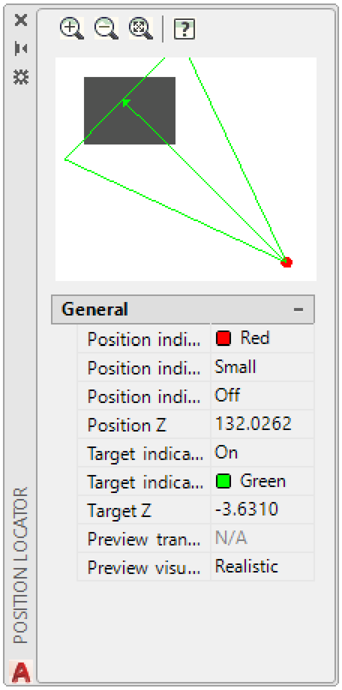


Figure 2-68 *The **POSITION LOCATOR** window*

Note

*1. To control the settings related to walk and fly, invoke the tool and right-click in the drawing area. Next, choose the **Walk and Fly Settings** option; the **Walk and Fly Settings** dialog box will be displayed. You can specify the settings in this dialog box.*

2. The Walk and Fly 3D navigation modes can be activated only in the perspective view. Activating the Walk or Fly 3D navigation mode in the parallel projection will display a warning window, as shown in Figure 2-69.

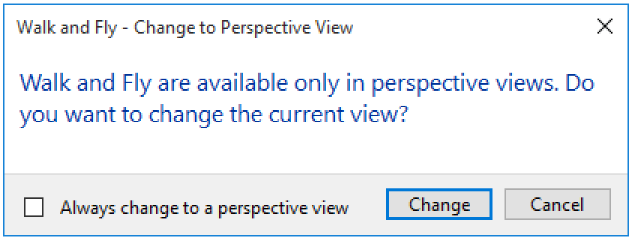


***Figure 2-69** The Warning displayed on activating the Walk and Fly 3D navigation modes in the parallel projection*

Tip
You can toggle between the 3D navigation modes from the shortcut menu or by entering the number displayed in front of each option in the shortcut menu at the Command prompt.

Zoom. This sub-option can be invoked by choosing **View > Zoom** from the menu bar. It is similar to the **ZOOM** option in the SteeringWheel and functions like the camera's zoom lens. You can invoke the other options of the Zoom by entering **3DZOOM** at the Command prompt. Alternatively, the options can also be displayed by right-clicking in the drawing area when you are inside this tool.

Pan. The **Pan** sub-option is invoked by choosing **View > Pan** from the menu bar. It is similar to the **PAN** option in the SteeringWheel. This tool also starts the 3D interactive viewing. You can right-click in the drawing area, when you are inside this tool, to display a shortcut menu that displays all the 3D Orbit options.

Enable Orbit Auto Target

This option, if chosen, ensures that the target point is focused on the object while rotating the view. If this option is cleared, the target point is set at the center of the viewport, in which the **Orbit** tool is invoked.

Animation Settings

This option, if chosen, displays the **Animation Settings** dialog box, refer to Figure 2-70. Using this dialog box, you can change the settings for visual style, resolution, number of frames to be captured in one second, and the format in which you want to save the file while creating the animation of 3D navigation. Creating animations will be discussed in the next chapter.

Zoom Window/Extents/Previous

These options are used to zoom the solid model using a window, zoom the objects to the extents, or retrieve the previous zoomed views of the model.

Parallel

This option is chosen to display the selected model using the parallel projection method. It is the default method when a new AutoCAD file is started with 2D templates. In this method, no parallel lines in the model converge at any point.

Perspective

This option is chosen to use the one point perspective method to display a model. It is the default method when a new AutoCAD file is started with 3D templates. In this method, all parallel lines in the model converge at a single point, thus providing a realistic view of the model when viewed with the naked eye, refer to Figure 2-71.

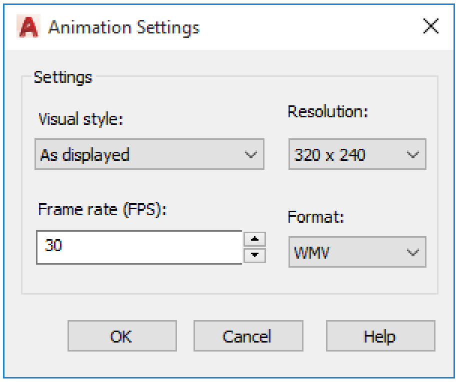


***Figure 2-70** The **Animation Settings** dialog box*

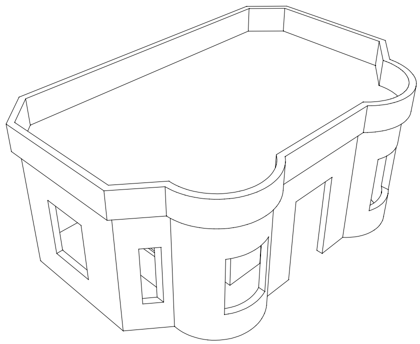


***Figure 2-71** Viewing the model using the perspective projection*

Reset View

This option, when chosen automatically, restores the view that was current when you started rotating the solid model in the 3D orbit. Note that you will not exit the **Orbit** tool if you choose this option.

Preset Views

This option is used to select any of the six standard orthogonal views or four isometric views.

Named Views

If you have created some views using the **VIEW** command, then the existing named views will be displayed as the sub-options under the **Named Views** option in the **3D Orbit** shortcut menu. Use this option to activate the saved view.

Visual Aids

The **Orbit** tool provides you with three visual aids for the ease of visualizing the solid model in the 3D orbit view. These visual aids are **Compass**, **Grid**, and **UCS Icon**. A check mark is displayed in front of the options you have selected. You can select none, one, two, or all three of the visual aids. Choosing **Compass** displays a sphere drawn within the arcball with three lines. These lines indicate the *X*, *Y*, and *Z* axis directions. Choosing the **Grid** displays a grid at the current *XY* plane. You can specify the height by using the **ELEVATION** system variable. The **GRID** system variable controls the display options of the **Grid** before using the **Orbit** tool. The size of the grid will depend upon the limits of the drawing. Choosing the **UCS Icon** displays the UCS icon. If this option is not selected, then AutoCAD turns off the display of the UCS icon. While using any of the 3D Orbit options, interactive 3D viewing is on and a shaded 3D view UCS icon is displayed. The *X* axis is red, the *Y* axis is green, and the *Z* axis is blue or cyan. Figure 2-72 shows a realistic shaded solid model displaying the compass, grid, and UCS icon.

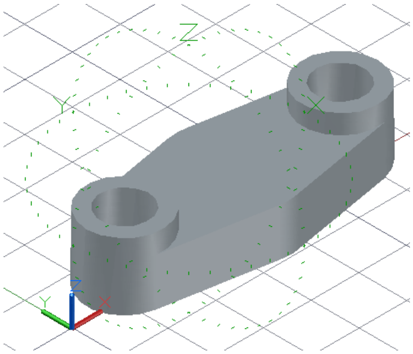


Figure 2-72 *Realistic shaded solid model showing the compass, grid, and UCS icon*

Clipping the View of a Model Dynamically

Command: 3DCLIP

The **3DCLIP** command is used to adjust the clipping planes for sectioning the selected solid model. Note that the actual solid model is not modified by this command. This command is used only for the purpose of viewing. The original solid model will be restored when you exit this command. With this option, you can adjust the location of the clipping planes. When you choose this option, the **Adjust Clipping Planes** window will be displayed, as shown in Figure 2-73. In this window, the object appears rotated at 90-degree to the top of the current view in the window. This facilitates the display of cutting planes. Setting the location of the front and back clipping planes in this window is reflected in the result in the current view. The various options of the **3DCLIP** command are displayed in the **Adjust Clipping Planes** toolbar in the window. There are seven buttons provided in this window. The functions of these buttons are discussed next.

Adjust Front Clipping. This button is chosen to locate the front clipping plane, which is defined by the line located in the lower end of the **Adjust Clipping Planes** window. You can see the result in the 3D Orbit view if the front clipping is on.

Adjust Back Clipping. This button is chosen to locate the back clipping plane, which is defined by the line located in the upper end of the **Adjust Clipping Planes** window. You can see the result in the 3D Orbit view if the back clipping is on.

Create Slice. Choosing this button causes both the front and back clipping planes to move together. The slice is created between the two clipping planes and is displayed in the current 3D view. You can first adjust the front and the back clipping planes individually by choosing the respective options. Next, by choosing the **Create Slice** button, activate both the clipping planes simultaneously and display the result in the current view.

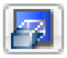
Pan. Displays the pan cursor that you can use to pan the object in the **Adjust Clipping Planes** window. Hold down the left mouse button and drag it in any direction. The pan cursor stays active until you click another button.

Zoom. Displays a magnifying-glass cursor that you can use to enlarge or reduce the clipping

plane. To do so, choose the **Zoom** tool and hold the left mouse button and drag it in up or down direction.

Front Clipping On/Off. Turns on or off the sectioning of a solid model using the front clipping

plane, refer to Figure 2-74. You can toggle between the front clipping on and off by choosing the **Front Clip On/Off** button available in the window.

Back Clipping On/Off. Turns on or off the sectioning of the solid model using the back clipping

plane, refer to Figure 2-74. You can toggle between back clipping on and off by choosing the **Back Clip On/Off** button available in the window.

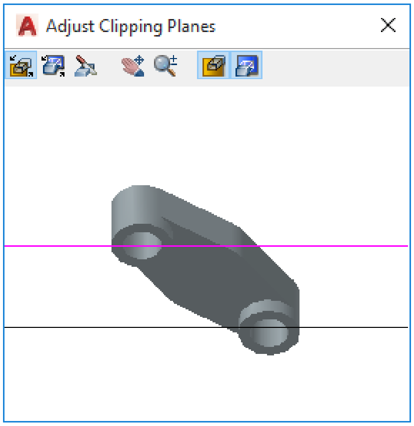


*Figure 2-73 The **Adjust Clipping Planes** window*

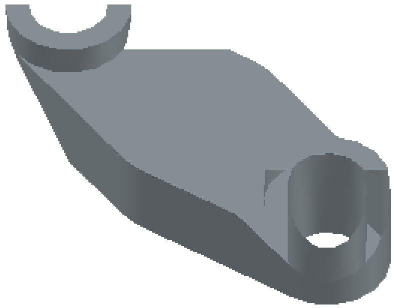


Figure 2-74 Model with the front and back clipping on and then hiding the hidden lines

NUDGE FUNCTIONALITY

Nudge functionality gives you the freedom to move any kind of object by 2 pixels in any direction orthogonal to screen. When the snap mode is ON, the object moves to the distance specified in the **Snap spacing** edit box in the **Drafting Settings** dialog box. To nudge any object, select the object and press CTRL + arrow key corresponding to the direction in which you want to move the object.

Self-Evaluation Test

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

1. The __________ and __________ commands can be used to change the viewpoint for viewing models in the 3D space.

2. Using the **Viewpoint Presets** dialog box, you can set the viewpoint with respect to __________ and __________.

3. The two types of 3D coordinate systems are __________ and __________.

4. To view 3D clipping on an object, the __________ command is used.

5. The **VPOINT** command is used to change the viewpoint to view a solid model. (T/F)

6. The wireframe models can be converted into surface models as well as solid models. (T/F)

7. In AutoCAD, the right-hand thumb rule is followed to identify the direction of the X, Y, and Z axes. (T/F)

8. In AutoCAD, the right-hand thumb rule is followed to find the direction of rotation in the 3D space. (T/F)

9. Once you exit the 3DCLIP command, the original solid model is restored. (T/F)

10. When you change the viewpoint, the 3D model moves from its default position. (T/F)

Review Questions

Answer the following questions:

1. In which of the following views a model is displayed by default in 3D Modeling workspace?

   (a) **Perspective View** (b) **Parallel View**
   (c) **Top View** (d) None of these

2. Which of the following tools is used to create a 3D polyline?

   (a) **Polyline** (b) **3D Poly**
   (c) **3D Polyline** (d) **Polyline 3D**

3. Which of the following commands is used to set the elevation and thickness for new objects?

   (a) **ELEVATION** (b) **THICKNESS**
   (c) **ELEV** (d) **THICK**

4. Which of the following commands is used to suppress the hidden edges in a 3D model?

   (a) **SUPPRESS** (b) **HIDE**
   (c) **3DHIDE** (d) **EDGE**

5. Which of the following views is set by default when you open a new drawing?

   (a) SE Isometric View (b) SW Isometric View
   (c) Custom View (d) Bottom View

6. For the **Sphere** and **Torus** meshes, if the **MESHTYPE** system variable is set to 0, the resulting surface will be a __________ mesh.

7. The highlighted area in a ViewCube that is used to change a view is called __________.

8. A 3D model can be rotated continuously using the **3D Orbit** tool. (T/F)

9. The **ELEV** command is a transparent command. (T/F)

10. You can directly write a text with thickness. (T/F)

Exercises 3 and 4

In these exercises, you will create the models shown in Figures 2-75 and 2-76. Assume the missing dimensions.

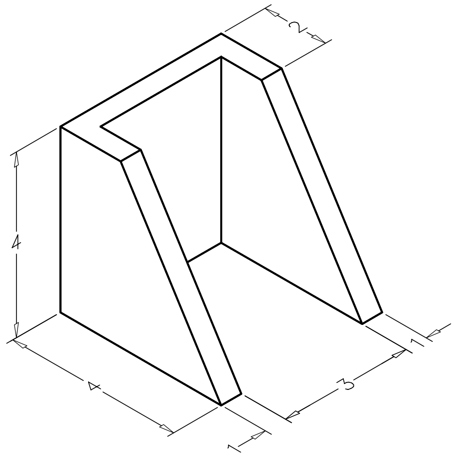


Figure 2-75 *Model for Exercise 3*

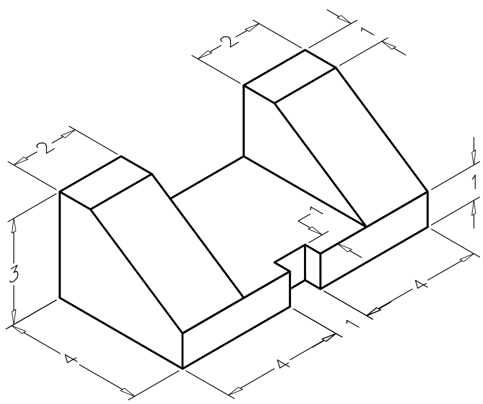


Figure 2-76 *Model for Exercise 4*

Answers to Self-Evaluation Test

1. **VPOINT, -VPOINT**, **2**. the angle in the *XY* plane from the *X* axis, angle from the *XY* plane, **3**. absolute, relative coordinate systems, **4**. **3DCLIP**, **5**. T, **6**. F, **7**. T, **8**. T, **9**. T, **10**. F