Learning Objectives

After completing this chapter, you will be able to:

✓ Change properties on solids.
✓ Align objects.
✓ Rotate objects in three dimensions.
✓ Mirror objects in three dimensions.
✓ Create 3D arrays.
✓ Fillet solid objects.
✓ Chamfer solid objects.
✓ Thicken a surface into a solid.
✓ Convert planar objects into surfaces.
✓ Slice a solid using various methods.
✓ Construct details on solid models.
✓ Remove features from solid models.

Changing Properties

Properties of 3D objects can be modified using the Properties palette, which is thoroughly discussed in AutoCAD and Its Applications—Basics. This palette is displayed using the PROPERTIES command. You can also double-click on a solid object or select the solid, right-click, and pick Properties from the shortcut menu.

The Properties palette lists the properties of the currently selected object. For example, Figure 11-1 lists the properties of a selected solid sphere. AutoCAD offers some parametric solid modeling options. A parametric solid modeling program allows you to change the parameters, such as a sphere’s diameter, in the Properties palette. You can also change the sphere’s position, linetype, linetype scale, color, layer, lineweight, and visual settings. The categories and properties available in the Properties palette depend on the selected object.

To modify an object property, select the property. Then, enter a new value in the right-hand column. The drawing is updated to reflect the changes. You can leave the Properties palette open as you continue with your work.
AutoCAD can automatically record a history of a composite solid model’s construction. The control of the history setting is found in the **Solid History** category of the **Properties** palette. By default, the History property in the **Properties** palette is set to **Record**, which means that the history will be saved. See **Figure 11-1**. It is generally a good idea to have the history recorded. Then, at any time, you can graphically display all of the geometry that was used to create the model.

If the **SOLIDHIST** system variable is set to a value of 1, all new solids have their **History** property set to **Record**. This is the default. If the system variable is set to 0, all new solids have their **History** property set to **None** (no recording). With either setting of the system variable, the **Properties** palette can be used to change the setting for individual solids.

To view the graphic history of the composite solid, set the **Show History** property in the **Properties** palette to **Yes**. All of the geometry used to construct the model is displayed. If the **SHOWHIST** system variable is set to 0, the **Show History** property is set to **No** for all solids and cannot be changed. If this system variable is set to 2, the **Show History** property is set to **Yes** for all solids and cannot be changed. A **SHOWHIST** setting of 1 allows the **Show History** property to be individually set for each solid.

An example of showing the history on a composite solid is provided in **Figure 11-2**. The object appears in its current edited format in **Figure 11-2A**. The **Conceptual** visual style is set current and the **Show History** property is set to **No**. In **Figure 11-2B**, the **Show History** property is set to **Yes**. Isolines have also been turned on. You can see the geometry that was used in the Boolean subtraction operations. Using subobject editing techniques, the individual geometry can be selected and edited. Subobject editing is discussed in detail in Chapter 12.

**NOTE**

If the **Solid History** property is set to **Yes** to display the components of the composite solid, as seen in **Figure 11-2**, the components will appear when the drawing is plotted. Be sure to set the **Show History** property to **No** before you print or plot.
AutoCAD provides two different methods with which to move and rotate objects in a single command. This is called aligning objects. The simplest method is to align 3D objects by picking source points on the first object and then picking destination points on the object to which the first one is to be aligned. This is accomplished with the 3DALIGN command, which allows you to both relocate and rotate the object. The second and much more versatile method allows you to not only move and rotate an object, but also to scale the object being aligned. This is possible with the ALIGN command.

**Move and Rotate Objects in 3D Space**

The basic function of moving and rotating an object relative to a second object or set of points is done with the 3DALIGN command. It allows you to reorient an object in 3D space. Using this command, you can correct errors of 3D construction and quickly manipulate 3D objects. The 3DALIGN command requires existing points (source) and the new location of those existing points (destination).

For example, refer to Figure 11-3. The wedge in Figure 11-3A is aligned in its new position in Figure 11-3B as follows. Set the Intersection or Endpoint running object snap to make point selection easier. Refer to the figure for the pick points.

Select objects: *(pick the wedge)*
1 found
Select objects:  
Specify source plane and orientation…
Specify base point or [Copy]: *(pick P1)*
Specify second point or [Continue] <C>: *(pick P2)*
Specify third point or [Continue] <C>: *(pick P3)*
Specify destination plane and orientation…
Specify first destination point: *(pick P4)*
Specify second destination point or [eXit] <X>: *(pick P5)*
Specify third destination point or [eXit] <X>:  

---

**Figure 11-2.**
A—The object appears in its current edited format with Show History property turned off. B—The Show History property is set to Yes and the display of isolines has been turned on.
PROFESSIONAL TIP
You can also use the 3DALIGN command to copy, instead of move, an object and realign it at the same time. Just select the Copy option at the Specify base point or [Copy]: prompt. Then, continue selecting the points as shown above.

Exercise 11-1
Complete the exercise on the student website.
www.g-wlearning.com/CAD

Move, Rotate, and Scale Objects in 3D Space
The ALIGN command has the same functions of the 3DALIGN command, but adds the ability to scale an object. Refer to Figure 11-4. The 90° bend must be rotated and scaled to fit onto the end of the HVAC assembly. Two source points and two destination points are required, Figure 11-4A. Then, you can choose to scale the object.

Select objects: (pick the 90° bend)
1 found
Select objects: .
Specify first source point: (pick P1)
Specify first destination point: (pick P2; a line is drawn between the two points)
Specify second source point: (pick P3)
Specify second destination point: (pick P4; a line is drawn between the two points)
Specify third source point or <continue>: .
Scale objects based on alignment points? [Yes/No] <N>: Y.

The 90° bend is aligned and uniformly scaled to meet the existing ductwork object. See Figure 11-4B. You can also align using three source and three destination points. However, when doing so, you cannot scale the object.

PROFESSIONAL TIP
Before using 3D editing commands, set running object snaps to enhance your accuracy and speed.
Chapter 11  Creating and Working with Solid Model Details

Exercise 11-2
Complete the exercise on the student website.
www.g-wlearning.com/CAD

3D Moving

The 3DMOVE command allows you to quickly move an object along any axis or plane of the current UCS. When the command is initiated, you are prompted to select the objects to move. After selecting the objects, press [Enter]. The move grip tool is displayed in the center of the selection set. By default, the move grip tool is also displayed when a solid is selected with no command active.

The move grip tool is a tripod that appears similar to the shaded UCS icon. See Figure 11-5A. You can relocate the tool by right-clicking on the tool and selecting Relocate Gizmo from the shortcut menu. Then, move the tool to a new location and pick. You can also realign the tool using the shortcut menu.

If you move the pointer over the X, Y, or Z axis of the grip tool, the axis changes to yellow. To restrict movement along that axis, pick the axis. If you move the pointer over one of the right angles at the origin of the tool, the corresponding two axes turn yellow. Pick to restrict the movement to that plane. You can complete the movement by either picking a new point or by direct distance entry.
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NOTE
If the GTAUTO system variable is set to 1, the move grip tool is displayed when a solid is selected with no command active. If the GLOCATION system variable is set to 0, the grip tool is placed on the UCS icon (not necessarily the UCS origin). Both variables are set to 1 by default.

3D Rotating

As you have seen in earlier chapters, the ROTATE command can be used to rotate 3D objects. However, the command can only rotate objects in the XY plane of the current UCS. This is why you had to change UCSs to properly rotate objects. The 3DROTATE command, on the other hand, can rotate objects on any axis regardless of the current UCS. This is an extremely powerful editing and design tool.

When the command is initiated, you are prompted to select the objects to rotate. After selecting the objects, press [Enter]. The rotate grip tool is displayed in the center of the selection set. See Figure 11-5B. If the 2D Wireframe visual style is current, the visual style is temporarily changed to the 3D Wireframe because the grip tool is not displayed in 2D mode. The grip tool provides you with a dynamic, graphic representation of the three axes of rotation. After selecting the objects, you must specify a location for the grip tool, which is the base point for rotation.

Now, you can use the grip tool to rotate the objects about the tool’s local X, Y, or Z axis. As you hover the cursor over one of the three circles in the grip tool, a vector is displayed that represents the axis of rotation. To rotate about the tool’s X axis, pick the red circle on the grip tool. To rotate about the Y axis, pick the green circle. To rotate about the Z axis, pick the blue circle. Once you select a circle, it turns yellow and you are prompted for the start point of the rotation angle. You can enter a direct angle at this prompt or pick the first of two points defining the angle of rotation. When the rotation angle is defined, the object is rotated about the selected axis.

Figure 11-5.
A—The move grip tool is a tripod that appears similar to the shaded UCS icon. B—This is the rotate grip tool. The three axes of rotation are represented by the circles. The origin of the rotation is where you place the center grip.
The following example rotates the bend in the HVAC assembly shown in Figure 11-6A. Set the Midpoint object snap and turn on object tracking. Then, select the command and continue:

```
Current positive angle in UCS: ANGDIR=(current) ANGBASE=(current)
Select objects: (pick the bend)
1 found
Select objects: ↵
Specify base point: (acquire the midpoint of the vertical and horizontal edges, then pick to place the grip tool in the middle of the rectangular face)
Pick a rotation axis: (pick the green circle)
Specify angle start point or type an angle: 180. ↵
```

Note that the rotate grip tool remains visible through the base point and the angle of rotation selections. The rotated object is shown in Figure 11-6B.

If you need to rotate an object on an axis that is not parallel to the current X, Y, or Z axes, use a dynamic UCS with the 3DROTATE command. Chapter 5 discussed the benefits of using a dynamic UCS when creating objects that need to be parallel to a surface other than the XY plane. With the object selected for rotation and the dynamic UCS option active (pick the Allow/Disallow Dynamic UCS button on the status bar), move the rotate grip tool over a face of the object. The grip tool aligns itself with the surface so that the Z axis is perpendicular to the face. Carefully place the grip tool over the point of rotation using object snaps. Make sure that the tool is correctly positioned before picking to locate it. Then, enter an angle or use polar tracking to rotate the object about the appropriate axis on the tool.

Figure 11-6.
A—Use object tracking or object snaps to place the grip tool in the middle of the rectangular face. Then, select the axis of rotation.
B—The completed rotation.
PROFESSIONAL TIP

By default, the move grip tool is displayed when an object is selected with no command active. To toggle between the move grip tool and the rotate grip tool, select the grip at the tool’s origin and press the space bar. Then, pick a location for the tool’s origin. You can toggle back to the move grip tool using the same procedure.

Exercise 11-3

Complete the exercise on the student website.
www.g-wlearning.com/CAD

3D Mirroring

The MIRROR command can be used to rotate 3D objects. However, like the ROTATE command, the MIRROR command can only work in the XY plane of the current UCS. Often, to properly mirror objects with this command, you have to change UCSs. The MIRROR3D command, on the other hand, allows you to mirror objects about any plane regardless of the current UCS.

The default option of the command is to define a mirror plane by picking three points on that plane, Figure 11-7A. Object snap modes should be used to accurately define the mirror plane. To mirror the wedge in Figure 11-7A, set the Midpoint running object snap, select the command, and use the following sequence. The resulting drawing is shown in Figure 11-7B.

Select objects: (pick the wedge)
1 found
Select objects: \[ ]
Specify first point of mirror plane (3 points) or [Object/Last/Zaxis/View/XY/YZ/ZX/3points] <3points>: (pick P1, which is the midpoint of the box’s top edge)
Specify second point on mirror plane: (pick P2)
Specify third point on mirror plane: (pick P3)
Delete source objects? [Yes/No] <N>: \[ ]

Figure 11-7.

The MIRROR3D command allows you to mirror objects about any plane regardless of the current UCS. A—The mirror plane defined by the three pick points is shown here in color. Point P1 is the midpoint of the top edge of the base. B—A copy of the original is mirrored.
There are several different ways to define a mirror plane with the `MIRROR3D` command. These are:

- **Object.** The plane of the selected circle, arc, or 2D polyline segment is used as the mirror plane.
- **Last.** Uses the last mirror plane defined.
- **Zaxis.** Defines the plane with a pick point on the mirror plane and a point on the Z axis of the mirror plane.
- **View.** The viewing direction of the current viewpoint is aligned with a selected point to define the plane.
- **XY, YZ, ZX.** The mirror plane is placed parallel to one of the three basic planes of the current UCS and passes through a selected point.
- **3points.** Allows you to pick three points to define the mirror plane, as shown in the above example.

**Exercise 11-4**
Complete the exercise on the student website.
www.g-wlearning.com/CAD

**Creating 3D Arrays**

The `ARRAY` command can be used to create either a rectangular or polar array of a 3D object on the XY plane of the current UCS. You probably used this command to complete some of the problems in previous chapters. The `3DARRAY` command allows you to array an object in 3D space. There are two types of 3D arrays—rectangular and polar.

**Rectangular 3D Arrays**

In a rectangular 3D array, as with a rectangular 2D array, you must enter the number of rows and columns. However, you must also specify the number of *levels*, which represents the third (Z) dimension. The command sequence is similar to that used with the 2D array command, with two additional prompts.

An example of where a rectangular 3D array may be created is the layout of structural steel columns on multiple floors of a commercial building. In Figure 11-8A, you can see two concrete floor slabs of a building and a single steel column. It is now a simple matter of arraying the steel column in rows, columns, and levels.

To draw a rectangular 3D array, select the `3DARRAY` command. Pick the object to array and press `[Enter]`. Then, specify the **Rectangular** option:

- Enter the type of array [Rectangular/Polar] <R>: R
- Enter the number of rows ( - - ) <1>: 3
- Enter the number of columns (###) <1>: 5
- Enter the number of levels (…) <1>: 2
- Specify the distance between rows ( - - ): 10
- Specify the distance between columns (###): 10
- Specify the distance between levels (…): 12

The result is shown in Figure 11-8B. Constructions like this can be quickly assembled for multiple levels using the `3DARRAY` command only once.
A polar 3D array is similar to a polar 2D array. However, the axis of rotation in a 2D polar array is parallel to the Z axis of the current UCS. In a 3D polar array, you can define a centerline axis of rotation that is not parallel to the Z axis of the current UCS. In other words, you can array an object in a UCS different from the current one. Unlike a rectangular 3D array, a polar 3D array does not allow you to create levels of the object. The object is arrayed in a plane defined by the object and the selected centerline (Z) axis.

To draw a polar 3D array, select the 3DARRAY command. Pick the object to array and press [Enter]. Then, specify the Polar option:

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

For example, the four mounting flanges on the lower part of the duct in Figure 11-9A must be placed on the opposite end. However, notice the orientation of the UCS. First, copy one flange and rotate it to the proper orientation. Then, use the 3DARRAY command as follows. Make sure polar tracking is on.

Select objects: (select the copied flange)
1 found
Select objects: .
Enter the type of array [Rectangular/Polar] <R>: P
Enter the number of items in the array: 4
Specify the angle to fill (+=ccw, –=cw) <360>: .
Rotate arrayed objects? [Yes/No] <Y>: .
Specify center point of array: CEN.
of: (pick the center of the upper duct opening)
Specify second point on axis of rotation: (move the cursor so the ortho line projects out of the center of the duct opening and pick)

The completed 3D polar array is shown in Figure 11-9B. If additional levels of a polar array are needed, they can be created by copying the array just created.

Exercise 11-5
Complete the exercise on the student website.
www.g-wlearning.com/CAD
Filleting Solid Objects

A fillet is a rounded interior edge on an object, such as a box. A round is a rounded exterior edge. The FILLET command is used to create both fillets and rounds. Before a fillet or round is created at an intersection, the solid objects that intersect need to be joined using the UNION command. Then, use the FILLET command. See Figure 11-10. Since the object being filleted is actually a single solid and not two objects, only one edge is selected. In the following sequence, the fillet radius is set at .25, then the fillet is created. First, select the FILLET command and then continue as follows.

Current settings: Mode = current, Radius = current
Select first object or [Undo/Polyline/Radius/Trim/Multiple]: R
Specify fillet radius <current>: .25
Select first object or [Undo/Polyline/Radius/Trim/Multiple]:
Enter fillet radius <0.2500>: .25
Select an edge or [Chain/Radius]:
1 edge(s) selected for fillet.

Examples of fillets and rounds are shown in Figure 11-11.

Figure 11-9.
A—A ductwork elbow with four flanges in place. Copies of these flanges need to be located on the opposite end. Start by creating one copy as shown. B—By creating a 3D polar array, the flanges are properly oriented without changing the UCS.

Figure 11-10.
A—Pick the edge where two unioned solids intersect to create a fillet. B—The fillet after rendering.
PROFESSIONAL TIP

You can construct and edit solid models while the object is displayed in a shaded view. If your computer has sufficient speed and power, it is often much easier to visualize the model in a 3D view with the Conceptual or Realistic visual style set current. This allows you to realistically view the model. If an edit or construction does not look right, just undo and try again.

Chamfering Solid Objects

A chamfer is a small square edge on the edges of an object. To create a chamfer on a 3D solid, use the CHAMFER command. Just as when chamfering a 2D line, there are two chamfer distances. Therefore, you must specify which surfaces correspond to the first and second distances. The detail to which the chamfer is applied must be constructed before chamfering. For example, if you are chamfering a hole, the object (cylinder) must first be subtracted to create the hole. If you are chamfering an intersection, the two objects must first be unioned.

After you enter the command, you must pick the edge you want to chamfer. The edge is actually the intersection of two surfaces of the solid. One of the two surfaces is highlighted when you select the edge. The highlighted surface is associated with the first chamfer distance. This surface is called the base surface. If the highlighted surface is not the one you want as the base surface, enter N at the [Next/OK] prompt and press [Enter]. This highlights the next surface. An edge is created by two surfaces. Therefore, when you enter N for the next surface, AutoCAD cycles through only two surfaces. When the proper base surface is highlighted, press [Enter].

Chamfering a hole is shown in Figure 11-12A. The end of the cylinder in Figure 11-12B is chamfered by first picking one of the vertical isolines, then picking the top edge. The following command sequence is illustrated in Figure 11-12A.
Figure 11-12.
A—A hole is chamfered by picking the top surface, then the edge of the hole. B—The end of a cylinder is chamfered by first picking the side, then the end. Both ends can be chamfered at the same time, as shown here.

(Trim mode) Current chamfer Dist1 = current, Dist2 = current
Select first line or [Undo/Polyline/Distance/Angle/Trim/Method/Multiple]: (pick edge 1)
Base surface selection…
(if the side surface is highlighted, change to the top surface as follows)
Enter surface selection option [Next/OK (current)] <OK>: N.
(the top surface should be highlighted)
Enter surface selection option [Next/OK (current)] <OK>: ↓
Specify base surface chamfer distance <current>: .125..
Specify other surface chamfer distance <current>: .125..
Select an edge or [Loop]: (pick edge 2, the edge of the hole)
Select an edge or [Loop]: ↓

PROFESSIONAL TIP
If you improperly create a fillet or chamfer, it is best to undo and try again as opposed to trying to fix it with editing methods. Faces and edges can be edited using the SOLIDEDIT command. Grips can also be used to edit solids. This procedure is discussed in Chapter 12; the SOLIDEDIT command is discussed in Chapter 13.

Exercise 11-6
Complete the exercise on the student website.
www.g-wlearning.com/CAD
A surface has no thickness. The value of the **THICKNESS** command does not affect the thickness of a planar surface, unlike for entities such as lines, polylines, polygons, and circles. But, a surface can be quickly converted to a 3D solid using the **THICKEN** command.

To add thickness to a surface, enter the command. Then, pick the surface(s) to thicken and press `[Enter]`. You are then prompted for the thickness. Enter a thickness value or pick two points on screen to specify the thickness. See Figure 11-13.

By default, the original surface object is deleted when the 3D solid is created with **THICKEN**. This is controlled by the **DELOBJ** system variable. To preserve the original surface, change the **DELOBJ** value to 0.

### Converting to Surfaces

AutoCAD provides a great deal of flexibility in converting and transforming objects. For example, a simple line can be quickly turned into a 3D solid in just a few steps. Refer to Figure 11-14.

1. Use the **Properties** palette to give the line a thickness. Notice that the object is still a line object, as indicated in the drop-down list at the top of the **Properties** palette.
2. Select the **CONVTOSURFACE** command.
3. Pick the line. Its property type is now listed in the **Properties** palette as a surface extrusion.
4. Use the **THICKEN** command to give the surface a thickness. Its property type is now a 3D solid.

In this process, the **CONVTOSURFACE** and **THICKEN** commands were instrumental in creating a 3D solid from a line. Other objects that can be converted to surfaces using the **CONVTOSURFACE** command are 2D solids, arcs with thickness, open polylines with a thickness and no width, regions, and planar 3D faces.
Additional flexibility in creating solids is provided by the CONVTOSOLID command. This command allows you to directly convert certain closed objects into solids. You can convert:

- Circles with thickness.
- Wide, uniform-width polylines with thickness. This includes polygons and rectangles.
- Closed, zero-width polylines with thickness. This includes polygons, rectangles, and closed revision clouds.
- Mesh primitives. Keep in mind that the smoothness level applied to the mesh primitives appears on the object when it is converted to a solid.

First, select the command. Then, select the objects to convert and press [Enter]. The objects are instantly converted with no additional input required. Figure 11-15 shows the three different objects before and after conversion to a solid.

If an object that appears to be a closed polyline with a thickness does not convert to a solid and the command line displays the message Cannot convert an open curve, the polyline was not closed using the Close option of the PLINE command. Use the PEDIT or PROPERTIES command to close the polyline and use the CONVTOSOLID command again.
To quickly create a straight section of pipe, draw a donut with the correct ID and OD of the pipe. Then, use the Properties palette to give the donut a thickness equal to the length of the section you are creating. Finally, use the CONVTOSOLID command to turn the donut into a solid.

Slicing a Solid

A 3D solid can be sliced at any location by using existing objects such as circles, arcs, ellipses, 2D polylines, 2D splines, or surfaces. Additionally, you can specify a slicing line by picking two points or specify a slicing plane by picking three points. After slicing the solid, you can choose to retain either or both sides of the model. The slices can then be used for model construction or display and presentation purposes.

The SLICE command is used to slice solids. When the command is initiated, you are asked to select the solids to be sliced. Select the objects and press [Enter]. Next, you must define the slicing path. The default method of defining a path requires you to specify two points on a slicing plane. The plane passes through the two points and is perpendicular to the XY plane of the current UCS. Refer to Figure 11-16 as you follow this sequence:

1. Select the command and pick the object to be sliced.
2. Pick the start point of the slicing plane. See Figure 11-16A.
3. Pick the second point on the slicing plane.
4. You are prompted to specify a point on the desired side to keep. Select anywhere on the back half of the object. The point does not have to be on the object. It must simply be on the side of the cutting plane that you want to keep.
5. The object is sliced and the front half is deleted. See Figure 11-16B.
When prompted to select the side to keep, you can press [Enter] to keep both sides. If both sides are retained, two separate 3D solids are created. Each solid can then be manipulated for construction, design, presentation, or animation purposes.

There are several additional options for specifying a slicing path. These options are listed here and described in the following sections.

- **Planar Object**
- **Surface**
- **Zaxis**
- **View**
- **XY**
- **YZ**
- **ZX**
- **3points**

**NOTE**

Once the **SLICE** command has been used, the history of the solid to that point is removed. If a history of the work is important, then save a copy of the file or place a copy of the object on a frozen layer prior to performing the slice.

**Planar Object**

A second method to create a slice through a 3D solid is to use an existing planar object. Planar objects include circles, arcs, ellipses, 2D polylines, and 2D splines. See **Figure 11-17A**. The plane on which the planar object lies must intersect the object to be sliced. The current UCS has no effect on this option.

Be sure that the planar object has been moved to the location of the slice. Then, select the **SLICE** command, pick the object to slice, and press [Enter]. Next, select the **Planar Object** option and select the slicing path object (the circle, in this case). Finally, specify which side is to be retained. See **Figure 11-17B**. Again, if both sides are kept, they are separate objects and can be individually manipulated.
Surface

A surface object can be used as the slicing path. The surface can be planar or nonplanar (curved). This method can be used to quickly create a mating die. For example, refer to Figure 11-18. First, draw the required surface. The surface should exactly match the stamped part that will be manufactured, Figure 11-18A. Then, draw a box that encompasses the surface. Next, select the SLICE command, pick the box, and press [Enter]. Then, enter the Surface option and select the surface. You may need to do this in a wireframe display. Finally, when prompted to select the side to keep, press [Enter] to keep both sides. The two halves of the die can now be moved and rotated as needed, Figure 11-18B.

Z Axis

You can specify one point on the cutting plane and one point on the Z axis of the plane. See Figure 11-19. This allows you to have a cutting plane that is not parallel to the current UCS XY plane. First, select the SLICE command, pick the object to slice, and press [Enter]. Next, enter the Zaxis option. Then, pick a point on the XY plane of the cutting plane followed by a point on the Z axis of the cutting plane. Finally, pick the side of the object to keep.
Figure 11-18. Slicing a solid with a surface. A—Draw the surface and locate it within the solid to be sliced. The solid is represented here by the wireframe. B—The completed slice with both sides retained. The top can now be moved and rotated as shown here.

Figure 11-19. Slicing a solid using the Zaxis option. A—Pick one point on the cutting plane and a second point on the Z axis of the cutting plane. B—The resulting slice.
View

A cutting plane can be established that is aligned with the viewing plane of the current viewport. The cutting plane passes through a point you select, which sets the depth along the Z axis of the current viewing plane. First, select the SLICE command, pick the object to slice, and press [Enter]. Next, enter the View option. Then, pick a point in the viewport to define the location of the cutting plane on the Z axis of the viewing plane. Use object snaps to select a point on an object. The cutting plane passes through this point and is parallel to the viewing plane. Finally, pick the side of the object to keep.

XY, YZ, and ZX

You can slice an object using a cutting plane that is parallel to any of the three primary planes of the current UCS. See Figure 11-20. The cutting plane passes through the point you select and is aligned with the primary plane of the current UCS that you specify. First, select the SLICE command, pick the object to slice, and press [Enter]. Next, enter the XY, YZ, or ZX option, depending on the primary plane to which the cutting plane will be parallel. Then, pick a point on the cutting plane. Finally, pick the side of the object to keep.

Figure 11-20.
Slicing a solid using the XY, YZ, and ZX options. A—The object before slicing. The UCS origin is in the center of the first hole and at the midpoint of the height. B—The resulting slice using the XY option. C—The resulting slice using the YZ option. D—The resulting slice using the ZX option.

Note UCS orientation
Three Points

Three points can be used to define the cutting plane. This allows the cutting plane to be aligned at any angle, similar to the Zaxis option. See Figure 11-21. First, select the SLICE command, pick the object to be sliced, and press [Enter]. Then, enter the 3points option. Pick three points on the cutting plane and then select the side of the object to keep.

Exercise 11-8

Complete the exercise on the student website.
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Removing Details and Features

Sometimes, it may be necessary to remove a detail that has been constructed. For example, suppose you placed a R.5 fillet on an object based on an engineering sketch. Then, the design is changed to a R.25 fillet. The UNDO command can only be used in the current drawing session. Also, even if the command can be used, you may have to step back through several other commands to undo the fillet. In another example, suppose an object has a bolt hole that is no longer needed. You will need to remove this feature.
In Chapter 2, solid modeling is described as working with modeling clay. If you think in these terms, you can remove features by adding “clay” to the object. Then, the new “clay” can be molded as needed.

For example, look at the object in Figure 11-22A. There are R.5 rounds (fillets) on the top surface of the base. However, these should be R.25 rounds. You cannot simply place the new fillets on the object. You must first add material to create a square edge. Then, the new fillets can be added.

1. Draw a solid box with the same dimensions as the base without the rounds. Center the new box on the base.
2. Use the UNION command to add the new box to the object. This, in effect, removes the rounds.
3. Use the FILLET command to place the R.25 rounds on the top edge of the base, Figure 11-22B.

The rounds have now, in effect, been changed from R.5 to R.25. However, there is an unseen problem. Display the object in wireframe. Notice how the hole no longer passes through the object, Figure 11-22C. To correct this problem, draw a solid cylinder of the same dimensions as the hole and centered in the hole. Then, subtract the cylinder from the object. The hole now passes through the object, Figure 11-22D.

This technique of adding material can be used to remove any internal feature and some external features, such as fillets (rounds). Other external features, such as a boss, can be removed by drawing a solid over the top of the feature. The feature to be removed must be completely enclosed by the new solid. Then, subtract the new solid from the original object. Be sure to “redrill” holes and other internal features as needed.

PROFESSIONAL TIP

There are several other methods for editing solids. These are covered in detail in Chapters 12 and 13. The above procedure can be simplified with these editing methods.
Constructing Details and Features on Solid Models

A variety of machining, structural, and architectural details can be created using some basic solid modeling techniques. The features discussed in the next sections are just a few of the possibilities.

Counterbore and Spotface

A counterbore is a recess machined into a part, centered on a hole, that allows the head of a fastener to rest below the surface. Create a counterbore as follows.

1. Draw a cylinder representing the diameter of the hole, Figure 11-23A.
2. Draw a second cylinder that is the diameter of the counterbore and center it at the top of the first cylinder. Move the second cylinder so it extends below the surface of the object to the depth of the counterbore, Figure 11-23B.
3. Subtract the two cylinders from the base object, Figure 11-23C.

A spotface is similar to a counterbore, but is not as deep. See Figure 11-24. It provides a flat surface for full contact of a washer or underside of a bolt head. Construct it in the same way as a counterbore.

Figure 11-23.
Constructing a counterbore. A—Draw a cylinder to represent a hole. B—Draw a second cylinder to represent the counterbore. C—Subtract the two cylinders from the base object.

Figure 11-24.
Constructing a spotface. A—The bottom of the second, larger-diameter cylinder should be located at the exact depth of the spotface. However, the height may extend above the surface of the base. Then, subtract the two cylinders from the base. B—The finished solid.
Countersink

A countersink is like a counterbore with angled sides. The sides allow a flat-head machine screw or wood screw to sit flush with the surface of an object. A countersink can be drawn in one of two ways. You can draw an inverted cone centered on a hole and subtract it from the base or you can chamfer the top edge of a hole. Chamfering is the quickest method.

1. Draw a cylinder representing the diameter of the hole, Figure 11-25A.
2. Subtract the cylinder from the base object.
3. Select the CHAMFER command.
4. Select the top edge of the base object.
5. Enter the chamfer distance(s).
6. Pick the top edge of the hole, Figure 11-25B.

Boss

A boss serves the same function as a spotface. However, it is an area raised above the surface of an object. Draw a boss as follows.

1. Draw a cylinder representing the diameter of the hole. Extend it above the base object higher than the boss is to be, Figure 11-26A.
2. Draw a second cylinder the diameter of the boss. Place the base of this cylinder above the top surface of the base object a distance equal to the height of the boss. Give the cylinder a negative height value so that it extends inside of the base object, Figure 11-26B.
3. Union the base object and the second cylinder (boss). Subtract the hole from the unioned object, Figure 11-26C.
4. Fillet the intersection of the boss with the base object, Figure 11-26D.
O-Ring Groove

An *O-ring* is a circular seal that resembles a torus. It sits inside of a groove constructed so that part of the O-ring is above the surface. An *O-ring groove* can be constructed by placing the center of a circle on the outside surface of a cylinder. Then, revolve the circle around the cylinder. Finally, subtract the revolved solid from the cylinder.

1. Construct the cylinder to the required dimensions, Figure 11-27A.
2. Rotate the UCS on the X axis (or appropriate axis).
3. Draw a circle with a center point on the surface of the cylinder, Figure 11-27B.
4. Revolve the circle 360° about the center of the cylinder, Figure 11-27C.
5. Subtract the revolved object from the cylinder, Figure 11-27D.

Architectural Molding

*Architectural molding* details can be quickly constructed using extrusions. First, construct the profile of the molding as a closed shape, Figure 11-28A. Then, extrude the profile the desired length, Figure 11-28B.

*Corner intersections* of molding can be quickly created by extruding the same shape in two different directions and then joining the two objects. First, draw the molding profile. Then, copy and rotate the profile to orient the local Z axis in the desired direction, Figure 11-29A. Next, extrude the two profiles the desired lengths, Figure 11-29B. Finally, union the two extrusions to create the mitered corner molding, Figure 11-29C.

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**Figure 11-27.**
Constructing an O-ring groove. A—Construct a cylinder; this one has a round placed on one end. B—Draw a circle centered on the surface of the cylinder. C—Revolve the circle 360° about the center of the cylinder. D—Subtract the revolved object from the cylinder. E—The completed O-ring groove.

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**Figure 11-28.**
A—The molding profile. B—The profile extruded to the desired length.
Chapter Test

Answer the following questions. Write your answers on a separate sheet of paper or complete the electronic chapter test on the student website.

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1. Which properties of a solid can be changed in the Properties palette?
2. What does the History property control?
3. What is the purpose of the ALIGN command?
4. How does the 3DALIGN command differ from the ALIGN command?
5. How does the 3DROTATE command differ from the ROTATE command?
6. How does the MIRROR3D command differ from the MIRROR command?
7. Which command allows you to create a rectangular array by defining rows, columns, and levels?
8. How does a 3D polar array differ from a 2D polar array?
9. How many levels can a 3D polar array have?
10. Which command is used to fillet a solid object?
11. Which command is used to chamfer a solid object?
12. What is the purpose of the THICKEN command and which type of object does it create?
13. Which system variable allows you to preserve the original object when the THICKEN command is used?
14. List four objects that can be converted to surfaces using the CONVTOSURFACE command.
15. Briefly describe the function of the SLICE command.

Figure 11-29.
Constructing corner molding. A—Copy and rotate the molding profile. B—Extrude the profiles to the desired lengths. C—Union the two extrusions to create the mitered corner. Note: The view has been rotated. D—The completed corner.
Drawing Problems

1. Construct an 8” diameter tee pipe fitting using the dimensions shown below. Hint: Extrude and union two solid cylinders before subtracting the cylinders for the inside diameters.
   A. Use EXTRUDE to create two sections of pipe at 90° to each other, then UNION the two pieces together.
   B. Use FILLET and CHAMFER to finish the object. The chamfer distance is .25” × .25”.
   C. The outside diameter of all three openings is 8.63” and the pipe wall thickness is .322”.
   D. Save the drawing as P11_01.

![Tee Pipe Fitting Diagram]

2. Construct an 8” diameter, 90° elbow pipe fitting using the dimensions shown below.
   A. Use EXTRUDE or SWEEP to create the elbow.
   B. Chamfer the object. The chamfer distance is .25” × .25”. Note: You cannot use the CHAMFER command.
   C. The outside diameter is 8.63” and the pipe wall thickness is .322”.
   D. Save the drawing as P11_02.

![Elbow Pipe Fitting Diagram]
Problems 3–6. These problems require you to use a variety of solid modeling functions to construct the objects. Use all of the solid modeling and editing commands you have learned so far to assist in construction. Use a dynamic UCS when practical and create new UCSs as needed. Use SOLIDHIST and SHOWHIST to record and view the steps used to create the solid models. Create copies of the completed models and split them as required to show the internal features visible in the section views. Save each drawing as P11_(problem number).

3. Thrust Washer

4. Collar
5. Diagram of Diffuser with dimensions:
- Ø0.375, Ø0.312, Ø0.125, Ø0.175, Ø0.500, Ø0.625, Ø0.625, Ø0.250, Ø1.750, Ø0.125, Ø0.562, Ø2.375, Ø2.000, Ø2.500, Ø3.000.

6. Diagram of Bushing with dimensions:
- Ø6.250, Ø45°, Ø31.75, Ø1.5x1.5, Ø6.00, Ø10.00, Ø25.00, Ø48.00.
7. In this problem, you will add legs to the kitchen chair you started in Chapter 2. In Chapter 9 you refined the seat and in Chapter 10 you added the seatback. In this chapter, you complete the model. In Chapter 17, you will add materials to the model and render it.

A. Open P10_07 from Chapter 10.
B. Using the **LINE** command, create the framework for lofting the legs and crossbars as shown below. The crossbars are at the midpoints of the legs. Position the lines for the double crossbar about 4.5″ apart.
C. Using a combination of the **3DROTATE** and **3DMOVE** commands in conjunction with new UCSs, draw and position circles as shapes for lofting the legs and crossbars. The legs transition from Ø1.00 at the ends to Ø1.25 at the midpoints. The crossbars transition from Ø0.75 at the ends to Ø1.00 at a position 2″ from each end.
D. Create the legs and crossbars. The completed model is shown below.
F. Save the drawing as P11_07.