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3D Visualization and Solid Primitive Conceptual Design in AutoCAD®

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GD111-3P This class will help you understand the viewing techniques in 3D AutoCAD and how to create simple conceptual models. We'll cover the many viewing commands and how to access them, plus the creation and editing of solid primitives and simple Boolean operations.

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About the Speaker:

Craig both manages the Autodesk Authorized Training Center and teaches Mechanical Design at Fox Valley Technical College (FVTC) in Appleton, Wisconsin. He is a Certified Instructor in AutoCAD, and has previous certification in Mechanical Desktop (now AutoCAD Mechanical) and Architectural Desktop (now AutoCAD Architecture). He teaches at ATCs across the United States, and has been elected twice to the Autodesk Training Center Executive Committee, serving as chairperson in 2001. Craig is coauthor of the popular book "AutoCAD and its Applications – Advanced," and contributing author to "AutoCAD and its Applications – Basics," and Republic Research Training Center's "R14 AutoLISP" Autodesk Certified Courseware. Craig is no stranger to AU, having taught AutoCAD training classes at this event for several years.



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3D Visualization and Solid Primitive Conceptual Modeling in AutoCAD®



Learning Objectives

After completing this course, you will be able to:

- Understand the 3D Modeling workspace and using the **Dashboard**.
- Display 3D objects from preset isometric viewpoints.
- Display 3D objects from any desired viewpoint.
- Use the **3DORBIT** command to dynamically rotate the display of a 3D model.
- Set a visual style current.
- Construct 3D solid primitives.
- Modify solids using Grip-editing.
- Extrude the surface of a solid using the **PRESSPULL** command.
- “Hollow out” a solid using the **SHELL** command.
- Create complex solids using the **UNION**, **SUBTRACT**, and **INTERSECT** command.
- Describe the function of the world and user coordinate systems.
- Move the user coordinate system to any surface.
- Save and manage user coordinate systems.

The use of three-dimensional (3D) drawing and design as a tool is becoming more prevalent throughout industry. Companies are discovering the benefits of 3D modeling in design, visualization, testing, analysis, manufacturing, assembly, and marketing. Three-dimensional models also form the basis of computer animations, architectural walkthroughs, and virtual worlds used with virtual reality systems. Drafters who can design objects, buildings, and “worlds” in 3D are in demand for a wide variety of positions, both inside and outside of the traditional drafting and design disciplines.

This course provides an introduction to several aspects of 3D drawing and visualization. To be effective in creating and using 3D objects, you must first have good 3D visualization skills, including the ability to see an object in three dimensions and to visualize it rotating in space. Once these concepts are understood you can begin mastering the basic tools of solid modeling: AutoCAD’s solid primitive commands. These tools are truly the “building blocks” to being able to construct any object in 3D and prepare you for entry into an exciting aspect of graphic communication.



Basic Overview of the Interface

Workspaces

A **workspace** is a drawing environment in which dockable windows, toolbars, menus, and Dashboard control panels are displayed for a specific task. A workspace stores not only which of these tools are visible, but also their on-screen locations. You can quickly change workspaces using the Workspaces toolbar. By default, the Workspaces toolbar is docked to the left of the Layers toolbar in the **AutoCAD Classic** workspace and to the left of the Standard toolbar in the **3D Modeling** workspace. In the **2D Drafting & Annotation** workspace, it is displayed to the left of the Standard Annotation toolbar.

In order to use the default 3D “environment”, you must start a new drawing file based on the acad3D.dwt template and set the 3D Modeling workspace current. All discussions in the remainder of the course assume that AutoCAD is in the default 3D environment.

EXERCISE #1

1. Select the File pulldown menu, and pick **New...**
2. Select **acad3D.dwt** from the list of template files
3. Pick the **Open** button, or **double-click** on the template file name
4. Select the dropdown arrow on the **Workspaces** toolbar
5. Select **3D Modeling** from the list of workspaces
6. The rest of the course/handout assumes you will always have the 3D Modeling workspace active.



Dashboard

Working in the 3D modeling environment is efficient and intuitive with the use of the **Dashboard**. It is composed of **control panels** that enable you to work with 3D modeling, viewing, and presentation commands without using toolbars and menus. See **Figure 1**. Each control panel contains tools for working in 3D.

The Dashboard is a dockable window, or palette, that is, by default, **docked** on the right side of the screen. However, to increase the size of the drawing area, you may want to **float** the Dashboard by dragging it into the drawing area. Like all other palettes (dockable windows) and toolbars, the Dashboard can be docked on the left or right or moved to a floating position anywhere on the screen using the grab bar or title bar.

If you prefer to have the Dashboard floating, you may want to enable the **auto-hide** feature. When this feature is enabled, the Dashboard is reduced to just the title bar to allow more of the drawing area to be displayed. To display the full Dashboard, simply move the cursor over the title bar. Then, select the tool you want to use and when the cursor is moved back into the drawing area, the Dashboard is again reduced to the title bar. To enable the auto-hide feature, pick the Auto-hide button at the bottom of the Dashboard title bar.

When the dashboard is floating, it covers up part of the drawing, though the amount of area that is covered is lessened by using the auto-hide feature. When the dashboard is docked, the drawing area is completely visible, though its size is reduced based upon how much of the dashboard you have showing. The best of both worlds can be realized by **anchoring** the dashboard. Anchoring keeps the dashboard docked, but incorporates the auto-hide feature too. To anchor the dashboard, right click on the title bar of the dashboard (it must be in floating mode). Then select Anchor Right > or Anchor Left < from the shortcut menu. Allow Docking must also be checking in order to anchor the dashboard. To display the full Dashboard,

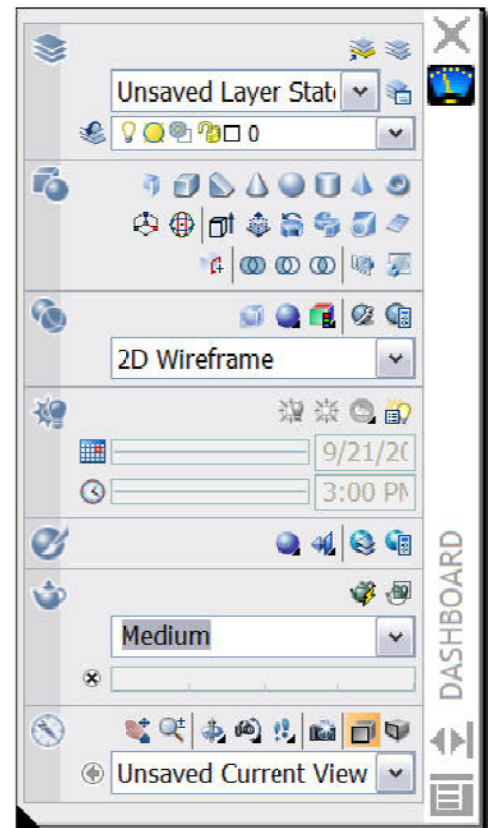


Figure 1. The Dashboard from the 3D Modeling workspace.



simply move the cursor over the anchored title bar. Then, select the tool you want to use and when the cursor is moved back into the drawing area, the Dashboard is again reduced to the anchored title bar.

You can close the Dashboard by picking the close control button (**X**) at the top of the title bar, or the right of the grab bar if docked. The DASHBOARD command can be used to display the Dashboard. The display of the Dashboard can also be toggled on and off using the Tools pull-down menu.

Control Panels

The seven default control panels that are displayed in the **Dashboard** in the 3D environment provide all of the functions you need to design, view, and render your 3D model. Each panel is identified by an icon in the upper-left corner of the panel. If you pause the cursor over the icon, the name of the panel is displayed. The panel can be expanded or condensed by picking this icon or by picking the maximize/minimize arrow in the lower-left corner of the panel. The maximize/minimize arrow is displayed when the cursor is over the vertical stripe on the left side of the panel. This stripe is dark gray when the panel can be expanded and orange when the panel is expanded.

EXERCISE #2

1. Experiment with the Dashboard and its possible locations: floating, docked or anchored.
2. Remember that in order to be able to anchor the Dashboard, Allow Docking must be checked.
3. Leave the Dashboard in the mode that is most comfortable.

Displaying 3D Views

It does not do much good to understand how to draw in 3D space if you cannot see what you draw in three dimensions. The default view in the 2D environment based on the acad.dwt template is a plan, or top, view of the XY plane. The default view in the 3D environment based on the acad3D.dwt template is a pictorial, or 3D, view. AutoCAD provides several methods of



changing your viewpoint to produce different pictorial views. The **viewpoint** is the location in space from which the object is viewed. The methods for changing your viewpoint covered in this course include preset isometric and orthographic viewpoints and the **3DORBIT** command.

Isometric and Orthographic Preset Views

A 2D isometric drawing is based on angles of 120° between the three axes. AutoCAD provides preset viewpoints that allow you to view a 3D object from one of four isometric locations. Each of these viewpoints produces an isometric view of the object. In addition, AutoCAD has presets for the six standard orthographic views of an object.

The four preset isometric views are southwest, southeast, northeast, and northwest. The six orthographic presets are top, bottom, left, right, front, and back. To switch your viewpoint to one of these presets, pick the drop-down list in the **3D Navigate** control panel of the **Dashboard** and select the view name. See **Figure 2**. You can also use the **View** pull-down menu or select the appropriate button on the **View** toolbar. This toolbar is not displayed by default.

Once you select a view, the viewpoint in the current viewport is automatically changed to display an appropriate isometric or orthographic view.

A view that looks straight down on the current drawing plane is called a **plan view**. An important aspect of the orthographic presets is that selecting one not only changes the viewpoint, but, by default, it also changes the UCS to be plan to the orthographic view. All new objects are created on that UCS instead of the WCS (or previous UCS). Working with UCSs is explained later in the course. However, to change the current UCS to the WCS type **UCS** to access the UCS command and then type **W** for the **World** option.

When an isometric or other 3D view is displayed, you can easily switch to a plan view of the current UCS using the **PLAN** command. Using the pull-down menus, select **View>3D Views>Plan View>World UCS**. There are also **Current UCS** and

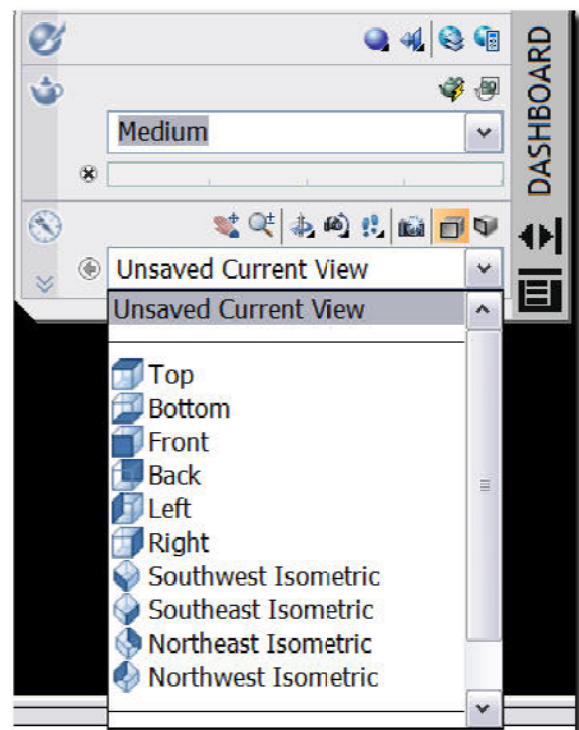


Figure 2. The Preset View Selections



Named UCS options in the cascading menu. The **PLAN** command is discussed in more detail later in the course.

EXERCISE #3

1. Open the **DICE.dwg** file
2. Experiment with isometric and orthographic preset views found in the **3D Navigate** control panel of the **Dashboard**.
3. To make the dots on the dice white, go to the **Options** dialog box, select the **Display** tab, pick the **Colors** button, highlight “**3D parallel projection**” in the **Context:** pane, highlight “**Uniform background**” in the **Interface element:** pane, then pick “**Black**” from the **Color:** dropdown list. Pick the **Apply & Close** button to dismiss the **Drawing Window Colors** dialog box, then pick **OK** to close the **Options** dialog box.

Dynamically Changing a 3D View

The **3DORBIT** command is a powerful and easy-to-use command for creating pictorial views of a 3D object. It allows you to dynamically rotate the view of a 3D object in real time. There are two basic modes for the **3DORBIT** command—constrained and free. These are discussed in the next sections.

Constrained Orbit

This command is accessed via the third button from the left in the top row of buttons in the 3D Navigate control panel of the Dashboard. See **Figure 3**.

A **constrained orbit** means that when you press and hold the pointer button, the view can be rotated up 90°, down 90°, and horizontally 360°. You cannot display an upside-down view of the opposite side of the model.

Within these constraints, the model can be freely rotated. This is the default mode when the command is typed. When the command is initiated, the constrained orbit cursor is displayed at the pointer position.

When in constrained orbit mode, a variety of options are available from the shortcut menu. In addition, you can select many of these options using the **3D Navigate** toolbar. These options are discussed in the following sections.



Figure 3. The Constrained Orbit Button

Professional Tip

The constrained mode of the **3DORBIT** command can be transparently accessed from within another command or with no command active. Press and hold the [Shift] key and simultaneously press and hold the mouse wheel. Then, move the pointer to achieve a new viewpoint. If you undo the command within which **3DORBIT** was accessed, the view is returned to the previous display.

Free Orbit

This command is also accessed via the third button from the left in the top row of buttons in the 3D Navigate control panel of the Dashboard. This button is a flyout – you must hold the pick button down to allow other buttons to fly out. The second button is Free Orbit. See **Figure 4**.



Figure 4. The Free Orbit Button

Without letting go of the pick button, move the cursor over the Free Orbit button, then let go of the pick button. This will activate the command. (Free Orbit then becomes the default command button for the third button in the top row.)

Free orbit mode allows the same movement as in constrained mode, but you can also vertically rotate the view 360°. When the command is initiated, a green circle is displayed in the middle of the current viewport. This is called the trackball, or arcball. By dragging inside or outside of the trackball, the view of the object is dynamically changed. The cursor shape indicates which action can be performed. Each shape is a visual cue to indicate which type of action will be performed if you pick, hold, and drag (move the pointing device). As in the constrained orbit mode, the shortcut menu shown of options is available.

Basic 3D Orbit Display Controls

Within the 3DORBIT command, you have the ability to toggle between constrained orbit, free orbit, realtime pan, and realtime zoom by using the shortcut menu. Display the shortcut menu by right-clicking and then pick Other Navigation Modes. Notice that a check mark appears next to the current mode. Also notice the numbers next to each item. The mode can be switched without displaying the shortcut menu by typing the number. See **Figure 5**.



Additionally, when in orbit mode and Enable Orbit Auto Target is on (checked) in the shortcut menu, as shown in Figure 5, the viewpoint is rotated about the center of the object display. When off (unchecked), the viewpoint is rotated about the center of the viewport. In this case, the objects may be rotated out of the viewport display.

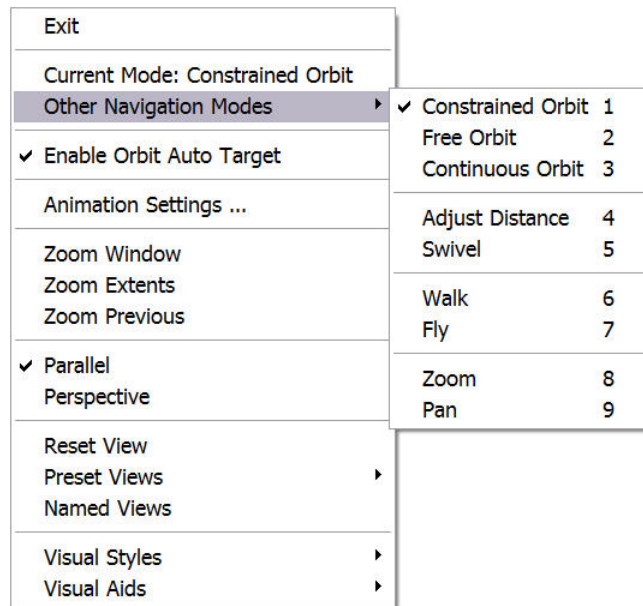


Figure 5. The 3DOrbit Shortcut Menu

Projection and Visual Aids

Using the **3DORBIT** command, a 3D model can be displayed using a wide variety of options. These include projection methods, such as parallel and perspective; access to visual styles; and visual aids, such as a spherical compass, grid, and shaded UCS icon. These options are selected in the shortcut menu available after the **3DORBIT** command has been entered, and are described in the following sections.

Projection

The projection of a pictorial view refers to how lines that recede into the background are treated. Projection of a 3D model can be either parallel or perspective.

In **parallel projection**, the sides of objects project parallel to each other. Axonometric views (isometric, dimetric, and trimetric) are all parallel projections.

In **perspective projection**, the sides of objects project toward one or more vanishing points. In 2D drafting, it is common to represent an object in pictorial as a one- or two-point



perspective, especially in architectural drafting.

Once the **3DORBIT** command is exited, the selected projection remains current. You can pan and zoom in both parallel and perspective projections. Additionally, objects can be edited in either type of projection once the **3DORBIT** command is exited.

Professional Tip

You can quickly change between projections at any time by picking the appropriate button in the **3D Navigate** control panel in the **Dashboard**. If the **3DORBIT** command is active, you may need to orbit the display slightly before the projection is changed.

Visual Aids

Visual aids help relate the view to the UCS. There are three visual aid options in the **3DORBIT** command.

- **Compass**
- **Grid**
- **3D UCS icon**

When the compass is on, a spherical 3D compass with the same diameter as the trackball appears. The compass has tick marks and labels indicating the X, Y, and Z axes. The compass can provide a reference for the current UCS. The **COMPASS** system variable controls this display, and is set to 0 if this item is not checked. If the compass is turned on, it remains on after the **3DORBIT** command is exited, except when the 2D Wireframe visual style is current.

The grid is displayed within the drawing limits when **Grid** is checked in the shortcut menu. If the current visual style is 2D Wireframe, the pictorial grid is displayed. The grid setting corresponds to the setting in the **Drafting Settings** dialog box. It is on by default. The setting is retained after the **3DORBIT** command is exited.

The shaded 3D UCS icon shows the orientation of the UCS. The X axis is red, Y axis is green, and Z axis is blue. When **UCS icon** is checked in the shortcut menu, the UCS icon is displayed. This is the default setting. When off (unchecked), the UCS icon is not displayed. This setting is *not* retained when the **3DORBIT** command is exited. The **UCSICON** command controls the visibility of this icon outside of the **3DORBIT** command. The 3D UCS icon is very useful and should be displayed at all times.



3D Orbit View Options

The **3DORBIT** command is extremely versatile because you can use a variety of established views to create a display. In addition, you can use the **3DORBIT** command to rotate the viewpoint and then reset the viewpoint to the view that was displayed prior to using the command. Three items in the 3D orbit view shortcut menu allow you to set views.

- **Reset View.** Resets the view that was displayed before **3DORBIT** was invoked. The command remains active.
- **Preset Views.** Displays a list of the orthographic and isometric presets. Note: Selecting a preset orthographic view here does *not* change the UCS.
- **Named Views.** The named views in the drawing are displayed in this cascading menu.

The drop-down list in the **3D Navigation** toolbar also displays all preset and named views. When the **3DORBIT** command is active, simply select a view from this list to display it. If you select one of the orthographic presets while the **3DORBIT** command is active, the UCS is not changed. However, if you select one of the orthographic presets from the drop-down list on the **3D Navigation** toolbar when the **3DORBIT** command is *not* active, the UCS *is* changed. Saved views can also be selected from the drop-down list in the toolbar when the **3DORBIT** command is not active.

3D Orbit Camera Settings

Two **3DORBIT** command options enable you to change the 3D display based on functions of a camera. The distance between camera and object can be adjusted, as well as the amount the camera is swiveled on a tripod. These options can be selected from the **3DORBIT** shortcut menu, the **3D Navigation** toolbar, or typed at the keyboard.

The distance between the viewer and the object can be set by adjusting the camera distance using the **3DDISTANCE** command. To access this command, type the command or use the **3D Navigation** toolbar, **View** pull-down menu, or **3D Navigate** control panel in the **Dashboard**. You can also select **Adjust Distance** from the **Other Navigation Modes** cascading menu in the **3DORBIT** shortcut menu. The cursor changes to arrows pointing up and down. Hold the pick button and move the cursor up to get closer to the object, or move it down to increase the camera distance from the object. Be warned, however, that this can distort the objects in the display.

In addition to adjusting the camera distance, you can also change the view by swiveling the camera. The **3DSWIVEL** command is similar to the **PAN** command except that the location from



which you are viewing the objects does not change, just the direction in which you are looking. To swivel the camera, use the **3DSWIVEL** command. To access this command, type the command or use the **3D Navigation** toolbar, **View** pull-down menu, or **3D Navigate** control panel in the **Dashboard**. You can also select **Swivel** from the **Other Navigation Modes** cascading menu in the **3DORBIT** shortcut menu. The cursor changes to a camera icon. Hold the pick button and move the cursor to swivel the camera in any direction.

EXERCISE #4

1. If it is not already so, open the **DICE.dwg** file
2. Experiment with the constrained and free orbit options of 3DORBIT. Be sure to try the visual aids such as, compass and grid. Also try the preset views, similar to the isometric and orthographic preset views found in the **3D Navigate** control panel of the **Dashboard**.
3. Try to closely match some of the views presented below in Figure 6. As you are doing so, be sure to pay attention to the UCS icon “tripod”. It can be a very handy tool, providing hints as to which way you might need to drag the cursor to obtain the desired view.

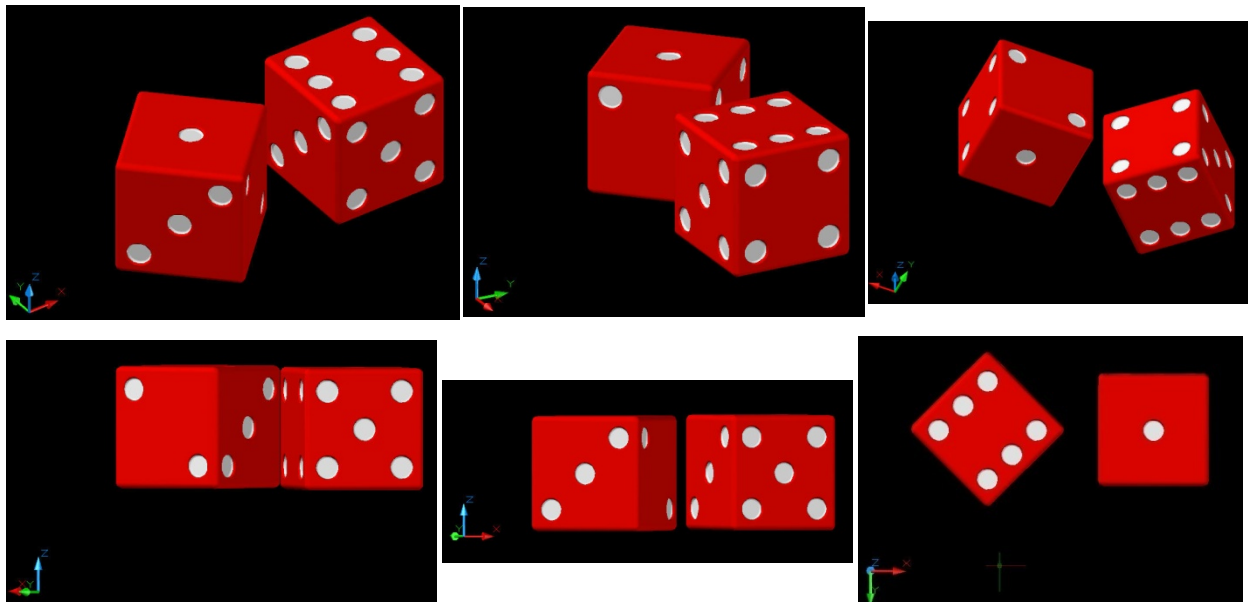


Figure 6. Try to Create the Views Above



Creating a Continuous 3D Orbit

The most dynamic aspect of the **3DORBIT** command is the ability to create a continuous orbit of a model. By moving your pointing device, you can set the model in motion in any direction and at any speed, depending on the power of your computer. An impressive display can be achieved using this command. To access continuous orbit mode, type the command or use the **3D Navigation** toolbar, **View** pull-down menu, or **3D Navigate** control panel in the **Dashboard**. You can also select **Continuous Orbit** from the **Other Navigation Modes** cascading menu in the **3DORBIT** shortcut menu. The continuous orbit cursor is displayed. This appears slightly different than the orbit cursor.

Press and hold the pick button and move the pointer in the direction that you want the model to rotate and at the desired speed of rotation. Release the button when the pointer is moving at the appropriate speed. The model will continue to rotate until you pick the left mouse button, press [Enter] or [Esc], or right-click and pick Exit or another option. At any time while the model is orbiting, you can left-click and adjust the rotation angle and speed by repeating the process for starting a continuous orbit.

EXERCISE #5

1. If not already the case, open the **DICE.dwg** file
2. Go ahead! You know you want to! Experiment with the continuous orbit capability. See how fast you can get it spinning. See how slow you can spin it. Make it spin vertically as well as horizontally. See if you can get it spinning “perfectly” horizontally, starting from an orthographic front view.

Introduction to Visual Styles

A 3D model can be displayed in a variety of visual styles. A **visual style** controls the display of edges and shading in a viewport. There are five basic visual styles—2D wireframe, 3D wireframe, 3D hidden, conceptual, and realistic. A **wireframe display** shows all lines on the object, including those representing back or internal features. A **hidden display** suppresses the display of lines that would normally be hidden by surfaces in the foreground.

To change styles, you can use the drop-down list in the **Visual Styles** control panel in the **Dashboard**. See **Figure 7**.

In the default 3D environment based on the acad3D.dwt template, the default display mode, or visual style, is Realistic. In this visual style, all objects appear as solids and are displayed in their assigned layer colors. Other display options are available, but are not discussed in this class.

- **2D Wireframe** This mode displays all lines of the model using assigned linetypes and lineweights. The 2D UCS icon and 2D grid are displayed, if turned on. If the **HIDE** command is used to display a hidden line view, use the **REGEN** command to redisplay the wireframe view.
- **3D Wireframe** Displays all lines of the model. The 3D grid and the 3D UCS icon are displayed, if turned on.
- **3D Hidden** Displays all visible lines of the model from the current viewpoint and hides all lines not visible. Objects are not shaded or colored.
- **Conceptual** The object is smoothed and shaded with transitional colors to help highlight details.
- **Realistic** Displays the shaded and smoothed model using assigned layer colors and materials.

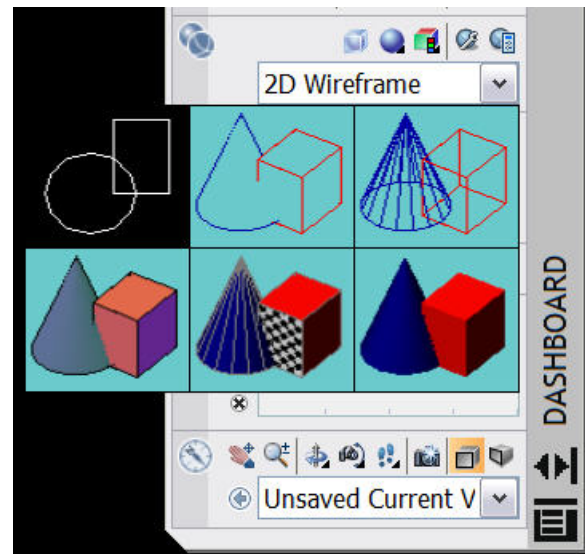


Figure 7. The Visual Styles from the Dashboard



Note

When the visual style is 2D Wireframe, you can quickly view the model with hidden lines removed by selecting **Hide** from the **View** pull-down menu or typing **HIDE**. The **HIDE** command can be used at any time to remove hidden lines from a wireframe display. If **HIDE** is used when the current visual style is 3D Wireframe, Conceptual, or Realistic, the 3D Hidden visual style is set current.

EXERCISE #6

1. It probably is open, but if not, open the **DICE.dwg** file
2. Try each of the various Visual Styles. Your results should look similar to the following images in Figures 8 - 13.

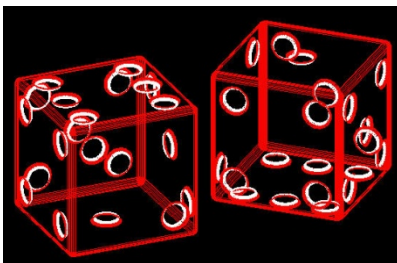


Figure 8. 2D Wireframe

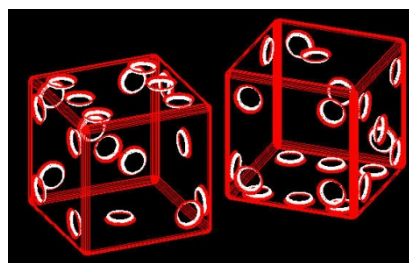


Figure 9. 3D Wireframe

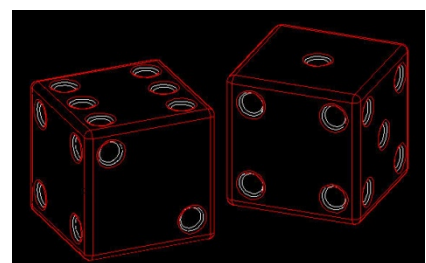


Figure 10. 3D Hidden

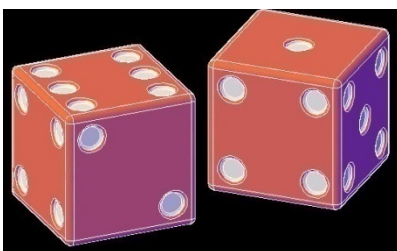


Figure 11. Conceptual

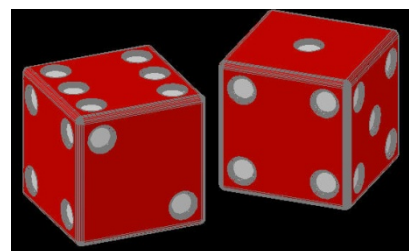


Figure 12. Realistic

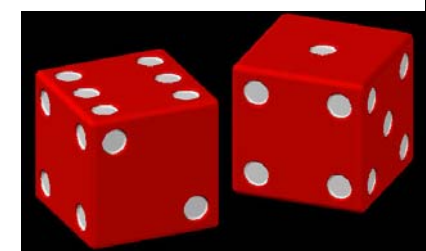


Figure 13. Visual Style 1 (custom)

Constructing Solid Primitives

As mentioned in the opening paragraph, a primitive is a basic building block. The eight **solid primitives** in AutoCAD are a box, wedge, pyramid, cylinder, cone, sphere, torus, and polysolid. These primitives can also be used as building blocks for complex solid models. This section provides detailed information on drawing all of the solid primitives. All of the 3D modeling primitive commands can be accessed using the **3D Make** control panel in the **Dashboard**, the **Modeling** toolbar, the **Draw** pull-down menu, or by typing the name of the 3D modeling primitive. See **Figure 14**.

The information required to draw a solid primitive depends on the type of primitive being drawn. For example, to draw a solid cylinder you must provide a center point for the base, a radius or diameter of the base, and the height of the cylinder. A variety of command options are available when creating primitives, but each primitive is constructed using just a few basic dimensions.

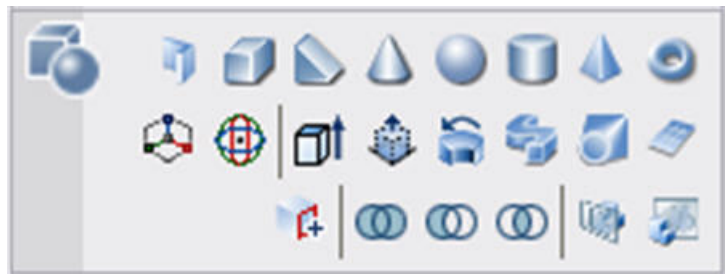


Figure 14. 3D Make Control Panel

Certain familiar editing commands can be used on solid primitives. For example, you can fillet or chamfer the edges of a solid primitive. In addition, there are other editing commands that are specifically for use on solids. You can also perform Boolean operations on solids. These operations allow you to add one solid to another, subtract one solid from another, or create a new solid based on how two solids overlap.

Using Dynamic Input and Dynamic Feedback

Dynamic input enables you to construct models in a “heads up” fashion with minimal eye movement around the screen. When a command is initiated, the command prompts are then displayed in the dynamic input area, which is at the lower-right corner of the crosshairs. As the pointer is moved, the dynamic input area follows it. The dynamic input area displays values of the cursor location, dimensions, command prompts, and command options (in a drop-down list). Coordinates and dimensions are displayed in boxes called *input fields*. When command options are available, a drop-down list arrow appears. Press the down arrow key on the keyboard to display the list. You can use your pointer to select the option, or press the down arrow key until



a dot appears by the desired option, then press [Enter].

For example, after selecting a modeling command such as **BOX**, the first item that appears in the dynamic input area is the prompt to specify the first corner and a display of the X and Y coordinate values of the crosshairs. At this point you can use the pointer to specify the first corner or type coordinate values. Type the X value and then a comma or the [Tab] key to move to the Y value input box. This locks the typed value and any movement of the pointer will not change it.

Caution

When using dynamic input to enter coordinate values from the keyboard, it is important that you avoid pressing [Enter] until you have completed the coordinate entry. When you press [Enter], all of the displayed coordinate values are accepted and the next command prompt appears.

In addition to entering coordinate values for sizes of solid primitives, you can provide direct distance dimensions. For example, the second prompt of the **BOX** command is for the second corner of the base. When you move the pointer, two dimensional input fields appear. Also, notice that a preview of the base is shown in the drawing area. This is the *dynamic feedback* that AutoCAD provides as you create a solid primitive. If you type a dimension at the keyboard and press the [Tab] key, the value is the length of the side. Then, type a dimension for the width of the base and press [enter], or press the left mouse button to set the base. But, if you enter a value followed by a comma, the dynamic input area changes to display X and Y coordinate boxes. In this case, the values entered are the X, Y, and Z coordinates of the opposite corner of the box base.

After establishing the location and size of the box base, the next prompt asks you to specify the height. Again, you can either enter a direct dimension value and press [Enter] or select the height with the pointer. AutoCAD provides dynamic feedback on the height of the box as the pointer is moved.

Professional Tip

When the **BOX** command prompts you to specify the other corner, and dynamic input is on, you can provide the coordinates for the opposite corner of the box to see the shape and size of the box before completing the command. To do this, type the X coordinate value and a comma. This changes the dynamic input area from direct distance input to coordinate input. Next, type the Y value and a comma, type the Z value, and press the [Tab] key. When all three values are entered, the preview is displayed, but the command is not complete. Press the [Tab] key to



cycle between the coordinate input boxes and type new values as needed. To complete the command, press the [Enter] key.

Note

The techniques described above can be used with any form of dynamic input. The current input field is always highlighted. You can always enter a value and use the [Tab] key to lock the input and move to the next field.

Box

A **box** has six flat sides and forms square corners. It can be constructed starting from an initial corner or the center. A cube can be constructed, as well as a box with unequal sides.

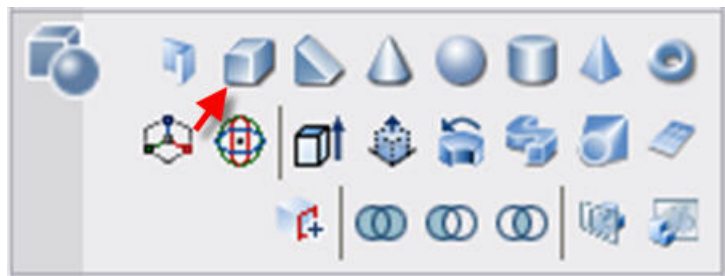


Figure 15. The BOX command on the 3D Make Control Panel

When the command is initiated, you are prompted to select the first corner or enter the **Center** option. The first corner is one corner on the base of the box. The center is the geometric center of the box, in each of the X, Y, and Z directions, not just the center of the base (*only* the X and Y directions). If you select the **Center** option, you are next prompted to select the center point.

After selecting the first corner or center, you are prompted to select the other corner or enter the **Cube** or **Length** option. The “other” corner is the opposite corner of the box base if you enter an XY coordinate, or the opposite corner of the box if you enter an XYZ coordinate. If the **Length** option is entered, you are first prompted for the length of one side. If dynamic input is on, you can also specify a rotation angle. After entering the length, you are prompted for the width of the box base. If the **Cube** option is selected, the length value is applied to all sides of the box.

Once the length and width of the base are established, you are prompted for the height, unless the **Cube** option was selected. Either enter the height or select the **2point** option. This option allows you to pick two points on screen to set the height. The box is created.

Professional Tip

If dynamic UCS is on, you can select a surface that is not parallel to the current UCS on which to locate the object. This feature is called a *dynamic UCS* and discussed in detail in Chapter 4.



EXERCISE #7

1. Begin a new drawing based on the acad3D.dwt template.
2. Select the BOX command.
3. Pick a point for the first corner.
4. Use dynamic input to enter dimensional values for the length and width of the box.
5. Use dynamic input to enter a dimensional value for the height of the box.
6. Start the BOX command again, and pick the first corner. Enter an X coordinate value and a comma.
7. Enter the Y and Z values without completing the command (do *not* hit [enter]). Use the [tab] key toggle between and re-enter the XYZ values to change the shape and size of the box. Complete the command.

Wedge

A **wedge** has five sides, four of which are at right angles and the fifth at an angle other than 90°. Once the command is initiated, you are prompted to select the first corner of

the base or to enter an option. By default, a wedge is constructed by

picking diagonal corners of the base and setting a height. To pick the center point, enter the **Center** option. The center point of a wedge is the middle of the angled surface. You must then pick a point to set the width and length before entering a height. If dynamic input is on, you can also set a rotation angle for the wedge.

After specifying the first corner or the center, you can enter the length, width, and height instead of picking a second corner. When prompted for the second corner, enter the **Length**

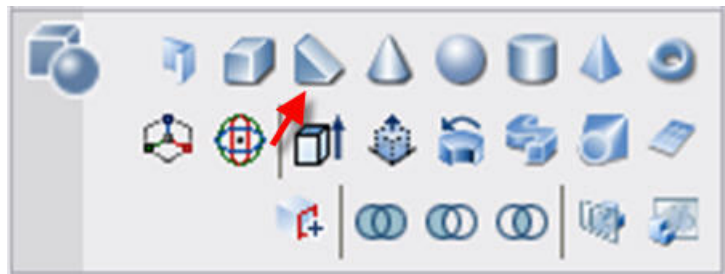


Figure 16. The WEDGE command on the 3D Make Control Panel



option and specify the length. You are then prompted for the width. After the width is entered, you are prompted for the height.

To create a wedge with equal length, width, and height, enter the **Cube** option when prompted for the second corner. Then, enter a length. The same value is automatically used for the width and height. Polar tracking is a good feature to use with this option.

Pyramid

A **pyramid** has a base composed of straight-line segments and edges that converge at a single point. The pyramid base can be composed of three to 32 sides, much like a 2D

polygon. A pyramid may be drawn with a pointed apex or as a **frustum pyramid**, which has a truncated, or flat, apex.

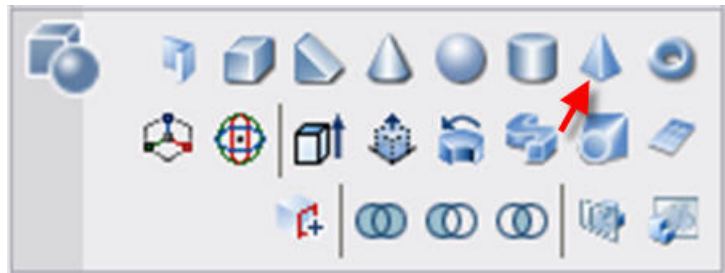


Figure 17. The PYRAMID command on the 3D Make Control Panel

Once the command is initiated, you are prompted for the center of the base or to enter an option. To set the number of sides on the base, enter the **Sides** option. Then, enter the number of sides. You are returned to the first prompt.

The base of the pyramid can be drawn by either picking the center and the radius of a base circle or by picking the endpoints of one side. The default method is to pick the center. Simply specify the center and then set the radius. To pick the endpoints of one side, enter the **Edge** option. Then, pick the first endpoint of one side followed by the second endpoint. If dynamic input is on, you can also set a rotation angle for the pyramid.

If drawing the base from the center point, the polygon is circumscribed about the base circle by default. To inscribe the polygon on the base circle, enter the **Inscribed** option before setting the radius. To change back to a circumscribed polygon, enter the **Circumscribed** option before setting the radius.

After locating and sizing the base, you are prompted for the height. To create a frustum pyramid, enter the **Top radius** option. Then, set the radius of the top circle. The top will be either inscribed or circumscribed based on the base circle. You are then returned to the height prompt.

The height value can be set by entering a direct distance. You can also use the **2point**

option to set the height. With this option, pick two points on screen. The distance between the two points is the height value. The **Axis endpoint** option can also be used to specify the center of the top in the same manner as a cone or cylinder.

Cylinder

A **cylinder** has a circular or elliptical base and edges that extend perpendicular to the base. When the command is initiated, you are prompted for the center point of the

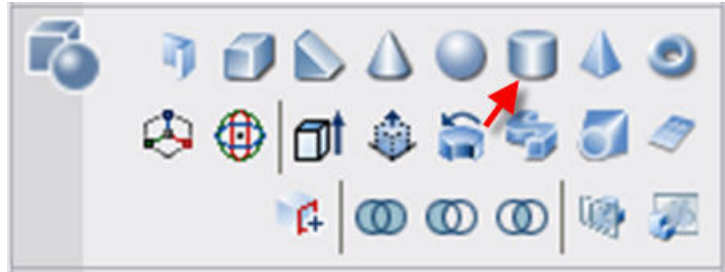


Figure 18. The **CYLINDER** command on the 3D Make Control Panel

cylinder base or to enter an option.

If you pick the center, you must

then set the radius of the base. To specify a diameter, enter the **Diameter** option after specifying the center.

The **3P**, **2P**, and **Ttr** options are used to define a circular base using either three points on the circle, two points on the circle, or two points of tangency on the circle and a radius. The **Elliptical** option is used to create an elliptical base.

If the **Elliptical** option is entered, you are prompted to pick both endpoints of one axis and then one endpoint of the other axis. If the **Center** option is entered, you are asked to select the center of the ellipse and then pick an endpoint on each of the axes.

After the base is defined, you are asked to specify a height or to enter the **2point** or **Axis endpoint** option. The **2point** option is used to set the height by picking two points on screen. The distance between the points is the height. The **Axis endpoint** option allows you to orient the cone at any angle, regardless of the current UCS, just as with a cone.

The **Axis endpoint** option is useful for placing a cylinder inside of another object to create a hole. The cylinder can then be subtracted from the other object to create a hole. If the axis endpoint does not have the same X and Y coordinates as the center of the base, the cylinder is tilted from the XY plane.

If polar tracking is on when using the **Axis endpoint** option, you can rotate the cylinder axis 90° from the current UCS Z axis, and then turn the cylinder to any preset polar increment. If the polar tracking vector is parallel to the Z axis of the current UCS, the tooltip displays a positive or negative Z value.

Cone

A **cone** has a circular or elliptical base with edges that converge at a single point. The cone may be *truncated* so the top is flat and the cone does not have an apex. When the command is initiated, you are prompted for the center point of the cone base or to enter an option. If you pick the center, you must then set the radius of the base. To specify a diameter, enter the **Diameter** option after specifying the center.

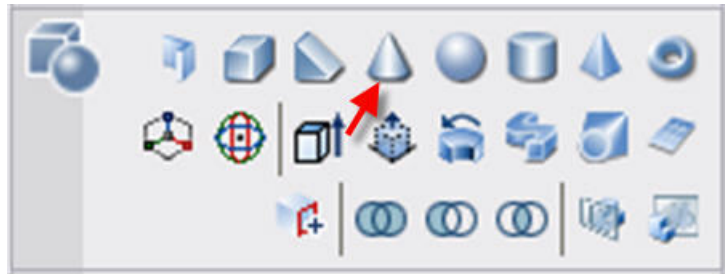


Figure 19. The CONE command on the 3D Make Control Panel

The **3P**, **2P**, and **Ttr** options are used to define a circular base using either three points on the circle, two points on the circle, or two points of tangency on the circle and a radius. The **Elliptical** option is used to create an elliptical base.

If the **Elliptical** option is entered, you are prompted to pick both endpoints of one axis and then one endpoint of the other axis. If the **Center** option is entered after the **Ellipse** option, you are asked to select the center of the ellipse and then pick an endpoint on each of the axes.

After the base is defined, you are asked to specify a height. You can enter a height or enter the **2point**, **Axis endpoint**, or **Top radius** option. The **2point** option is used to set the height by picking two points on screen. The distance between the points is the height.

The **Axis endpoint** option allows you to orient the cone at any angle, regardless of the current UCS. For example, to place a tapered cutout in the end of a block, first create a construction line. Then, locate the cone base and give a coordinate location of the apex, or axis endpoint. You can then use editing commands to subtract the cone from the box to create the tapered hole.

The **Top radius** option allows you to specify the radius of the top of the cone. If this option is not used, the radius is zero, which creates a pointed cone. Setting the radius to a value other than zero produces a **frustum cone**, or a cone where the top is truncated and does not come to a point.

Sphere

A **sphere** is a round, smooth object like a baseball or globe. Once the command is initiated, you are prompted for the center of the sphere or to enter an option. If you pick the center, you



must then set the radius of the sphere. To specify a diameter, enter the **Diameter** option after specifying the center. The **3P**, **2P**, and **Ttr** options are used to define the sphere using either three points on the surface of the sphere, two points on the surface of the sphere, or two points of tangency on the surface of the sphere and a radius.

Spheres and other curved objects can be displayed in a number of different ways. The manner in which you choose to

display these objects should be governed by the display requirements of your work. Notice the lines that define the shape of the spheres in a wireframe display. These lines are called **contour lines**, also known as *tessellation lines*. The **Visual Styles Manager** can be used to set the display of contour lines and silhouettes on spheres and other curved 3D surfaces for a given visual style.

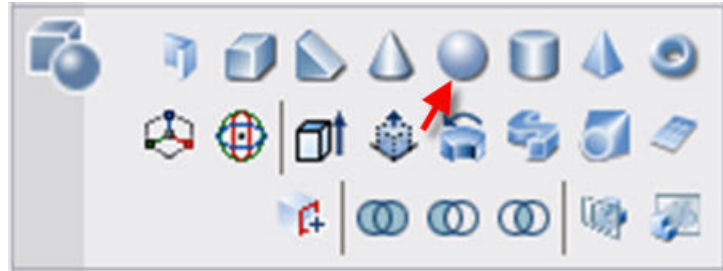


Figure 20. The SPHERE command on the 3D Make Control Panel

With the **Visual Styles Manager** displayed, select the 2D Wireframe image tile. The **Contour lines** setting in the **2D Wireframe options** area establishes the number of lines used to show the shape of curved objects. A similar setting appears in the 3D Wireframe, 3D Hidden, Conceptual, and Realistic visual styles if their **Edge mode** option is set to Isolines. The default value is four, but can be set to a value from zero to 2047. It is best to use a lower number during construction and preliminary displays of the model and, if needed, higher settings for more realistic visualization. The contour lines setting is also available in the **Display** tab of the **Options** dialog box or by typing ISOLINES.

The **Draw true silhouettes** setting in the **2D Wireframe options** area controls the display of silhouettes on 3D solid curved surfaces. The setting is either Yes or No. The **Draw true silhouettes** setting is stored in the **DISPSILH** system variable.

Professional Tip

The **Visual Style** control panel in the **Dashboard** can also be used to change the contour lines and silhouettes without changing the visual style definition. This is discussed in Chapter 3.

Torus

A basic **torus** is cylinder bent into a circle, similar to a doughnut or inner tube. There are

three types of tori. A torus with a tube diameter that touches itself has no center hole. This is the second type of torus and is called **self intersecting**. To create a self-intersecting torus, the tube radius must be greater than the torus radius. The third type of torus looks like a football. It is drawn by entering a negative torus radius and a positive tube diameter of greater value, i.e. -1 and 1.1.

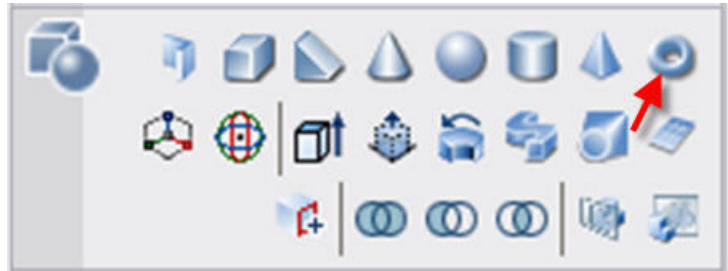


Figure 21. The TORUS command on the 3D Make Control Panel

Once the command is initiated, you are prompted for the center of the torus or to enter an option. If you pick the center, you must then set the radius of the torus. To specify a diameter, enter the **Diameter** option after specifying the center. This defines a base circle that is the centerline of the tube. The **3P**, **2P**, and **Ttr** options are used to define the base circle of the torus using either three points, two points, or two points of tangency and a radius.

Once the base circle of the torus is defined, you are prompted for the tube radius or to enter an option. The tube radius defines the cross-sectional circle of the tube. To specify a diameter of the cross sectional circle, enter the **Diameter** option. You can also use the **2point** option to pick two points on screen that define the diameter of the cross-sectional circle.

Polysolid

The **polysolid** primitive is simply a polyline that is constructed as a solid object by applying a width and height to the polyline. Many of the options used to create polylines are

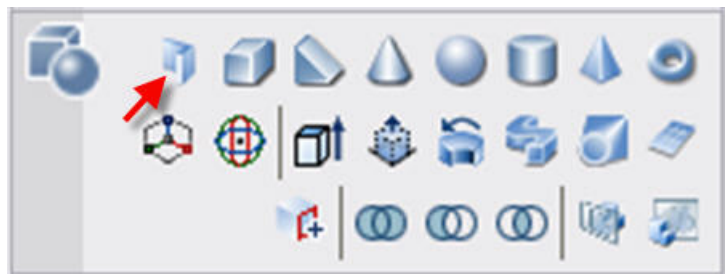


Figure 22. The POLYSOLID command on the 3D Make Control Panel

used with the **POLYSOLID** command. The principal difference is that a solid object is constructed using **POLYSOLID**.

When the command is initiated, you are prompted to select the first point or enter an option. By default, the width of the polysolid is equally applied to each side of the line you draw. This is center justification. Using the **Justify** option, you can set the justification to center, left, or right.



The justification applies to all segments created in this command session. If you select the wrong justification option, you must exit the command and begin again.

The default width is .25 units and height is four units. These values can be changed using the **Height** and **Width** options of the command. The height value is saved in the **PSOLHEIGHT** system variable. The width value is saved in the **PSOLWIDTH** system variable. Using these system variables, the default width and height can be set outside of the command.

The **Object** option allows you to convert an existing 2D object into a polysolid. AutoCAD entities such as lines, circles, arcs, polylines, polygons, and rectangles can be converted. The 2D object cannot be self intersecting. Some objects, such as 3D polylines and revision clouds, cannot be converted.

Once you have set the first point on the polysolid, pick the endpoint of the first segment. Continue adding segments as needed and press [Enter] to complete the command. After the first point is set, you can enter the **Arc** option. The current segment will then be created as an arc instead of a straight line. Arc segments will be created until you enter the **Line** option. The suboptions for the **Arc** option are:

- **Close.** If there are two or more segments, this option creates an arc segment between the active point and the first point of the polysolid.
- **Direction.** Specifies the tangent direction for the start of the arc.
- **Line.** Returns the command to creating straight line segments.
- **Second point.** Locates the second point of a two-point arc. This is not the endpoint of the segment.

Constructing a Planar Surface

A **planar surface** primitive is an object consisting of a single plane, and is created parallel to the current XY plane. The surface that is created has zero thickness and is composed

of a mesh of lines. It is created with the **PLANESURF** command. The

command prompts you to specify the first corner and then the second corner of a rectangle.

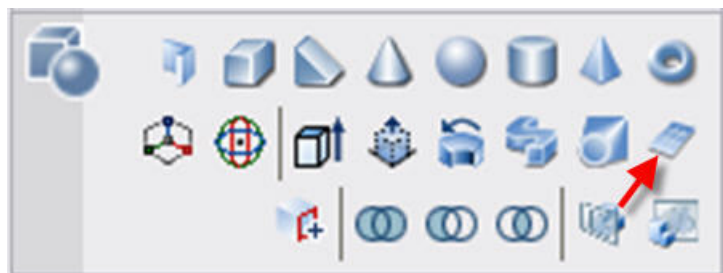


Figure 23. The **PLANESURF** command on the 3D Make Control Panel



Once drawn, the surface is displayed as a mesh with lines in the X and Y directions. These lines are called *isolines* and do not include the object's boundary. The **SURFU** (Y axis) and **SURFV** (X axis) system variables determine how many isolines are created when the planar surface is drawn. The isoline values can be changed later using the **Properties** window. The maximum number of isolines in either direction is 200.

The **Object** option of the **PLANESURF** command allows you to convert a 2D object into a planar surface. Any existing object or objects lying in a single plane and forming a closed area can be converted to a planar surface.

Although a planar surface is not a solid, it can be converted into a solid in a single step using the **THICKEN** command. The object that started as two arcs and two lines is now a solid model and can be manipulated and edited like any other solid. This capability enables you to create intricate planar shapes and quickly convert them to a solid for use in advanced modeling applications.

EXERCISE #8

1. Begin a new drawing based on the acad3D.dwt template.
2. Use the current 3D viewpoint, or choose one using the 3DORBIT command.
3. Construct the following solid primitives.
 - A sphere 1.5 in diameter.
 - A cone 2.5 high with a base diameter of 1.5.
 - A box that is $3 \times 2 \times 1$.
 - A wedge 4 long, 3 wide, and 2 high.
 - A cylinder 1.5 in diameter and 2.5 high.
 - A basic torus with a radius of 2 and a tube diameter of .75.
 - A pyramid with a base diameter of 2 and a height of 3.
 - A frustum pyramid with a base diameter of 2, an apex (Top) diameter of .5., and height of 2.5.
 - A polysolid 2 thick and 6 high using a variety of straight lines and curves.

Grip Editing Solid Primitives

There are three basic types of 3D solids in AutoCAD. The commands **BOX**, **WEDGE**, **CYLINDER**, **SPHERE**, etc., create 3D solid *primitives*. *Sweeps* are 2D profiles given thickness by the **EXTRUDE**, **REVOLVE**, **SWEEP**, and **LOFT** commands to create a 3D solid. Finally, 3D solid *composites* are created by a Boolean operation or by using the **SOLIDEDIT** command. There are two types of grips—base and parameter—that may be associated with a solid object. These grips provide an intuitive means of modifying solids. Base grips are square and parameter grips are typically arrows. The editing that can be performed with these grips are discussed in the next sections. The 3D solid primitives (box, wedge, pyramid, cylinder, cone, sphere, and torus) all have basically the same grips. However, not all grips are available on all primitives. All primitives have a base grip at the centroid of the base. This grip functions like a standard grip in 2D work. It can be used to stretch, move, rotate, scale, or mirror the solid. Boxes, wedges, and pyramids have square base grips at the corners that allow the size of the base to be changed. See **Figure 24**. Select one of these grips, move the cursor, and select

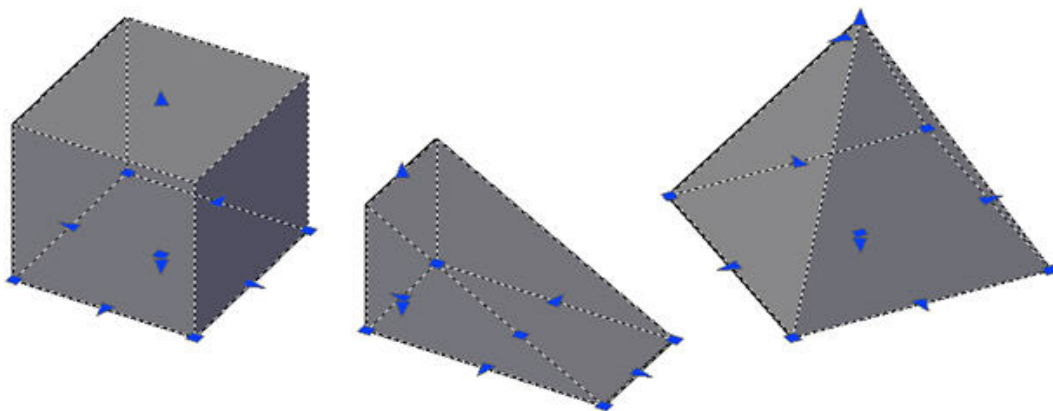


Figure 24. Grips as shown on a Box, Wedge, and Pyramid

a new point. The object dynamically changes in the viewport as you move the grip. You can also type the new coordinate location for the grip and press [Enter]. If ortho is off, the length and width can be changed at the same time by dragging the grip, except in the case of a pyramid. The parameter grips on the base allow the length or width to be changed. Additionally, the height of these objects can be changed using parameter grips. Each object has one parameter

grip for changing the height of the apex and one for changing the height of the plane on which the base sits. A pyramid also has a parameter grip at the apex for changing the radius of the top.

Cylinders, cones, and spheres have four parameter grips for changing the radius of the base, or the cross section in the case of a sphere. See **Figure 25**.

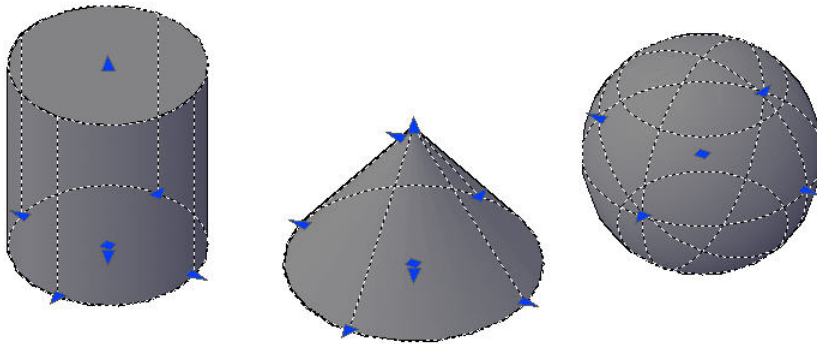
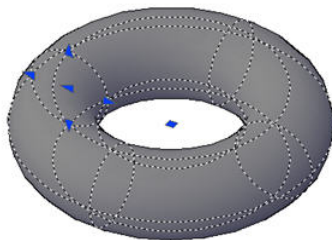


Figure 25. Grips as shown on a Cylinder, Cone, and Sphere

Cylinders and cones also have parameter grips for changing the height of the apex and the height of the plane on which the base sits. Additionally, a cone has a parameter grip at the apex for changing the radius of the top.

A torus has a parameter grip located at the center of the tube. See **Figure 26**. This grip is



used to change the radius of the torus. There is also a parameter grip at each quadrant of the tube. These are used to change the radius of the tube.

A polysolid does not have parameter grips. Instead, a base grip appears at

Figure 26. Grips as shown on a Torus each corner of the starting face of the solid. See **Figure 27**. Use these grips to change the cross sectional shape of the polysolid. The corners do not need to remain square. Base grips also appear at the endpoint of each segment centerline. Use these to change the location of each segment's endpoints.

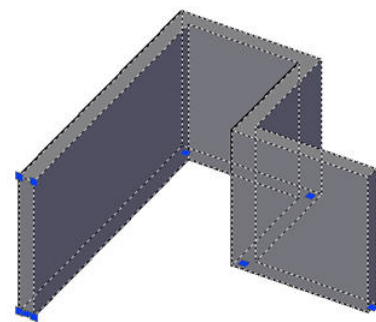


Figure 27. Grips as shown on a Polysolid



EXERCISE #9

1. Well, I am sure you are doing it already, but go ahead and use the grips and start modifying your solids primitives from the previous exercise.

Overview of Subobject Editing

AutoCAD solid primitives, such as cylinders, wedges, and boxes, are composed of three types of subobjects: faces, edges, and vertices. In addition, the objects that are used with Boolean commands to create a composite solid are considered subobjects, if the history is recorded. The primitive subobjects can be edited. Once selected, the primitive subobjects can even be deleted from the composite solid.

Subobjects can be easily edited using grips, which provide an intuitive and flexible method of solid model design. For example, suppose you need to rotate a face subobject in the current XY plane. You can select the subobject, pick its base grip, and then cycle through the editing functions to **ROTATE**. You can also use the **ROTATE** command on the selected subobject.

Selecting Subobjects

To select a subobject, press the [Ctrl] key and pick the subobject. You can select multiple subobjects and subobjects on multiple objects. To select a subobject that is hidden in the current view, first display the model as a wireframe. After creating a selection set, select a grip and edit the subobject as needed. Multiple objects can be selected in this manner. To deselect objects, press the [Shift]+[Ctrl] key combination and pick the objects to be removed from the selection set.

If objects or subobjects are overlapping, press the [Ctrl] key and the spacebar to turn on cycling and pick the subobject. Then, release the spacebar, continue holding the [Ctrl] key, and pick until the subobject you need is highlighted. Press [Enter] or the spacebar to select the highlighted subobject.

The [Ctrl] key method can be used to select subobjects for use with editing commands such as **MOVE**, **COPY**, **ROTATE**, **SCALE**, **ARRAY**, and **ERASE**. Some commands, like **ARRAY**, **STRETCH**, and **MIRROR**, are applied to the entire solid. Other operations may not be applied at all, depending on which type of subobject is selected. You can also use the **Properties** window



to change the color of edge and face subobjects or the material assigned to a face. The color of a subobject primitive can also be changed, but not the color of its subobjects.

Presspull

The **PRESSPULL** command allows any closed boundary to be extruded. The boundary can be a flat surface, a closed polyline, a circle, or a region. The extrusion is always applied perpendicular to the plane of the boundary, but can be in the positive or negative direction. When

applied to the face of a solid, it is very similar to the **Extrude Face** option of the **SOLIDEDIT** command, though dynamic feedback is provided for the extrusion with **PRESSPULL**.

Once the command is initiated, you are prompted to pick inside of the bounded areas to extrude. Move the pointer inside of a boundary and pick. Then, drag the boundary to a new location and pick or, if dynamic input is on, enter the distance to extrude the face.

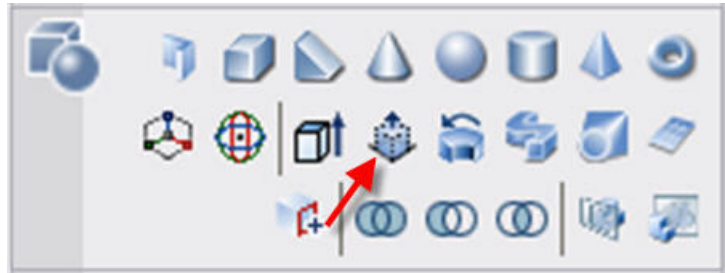


Figure 28. The **PRESSPULL** command on the 3D Make Control Panel

Note

The entire boundary must be visible on the screen, or the loop will not be found.

Shell

A **shell** is a solid that has been “hollowed out.” The **Shell** option of the **SOLIDEDIT** command creates a

shell of the selected object using a specified offset distance, or thickness.

To create a shell of a solid body, select the button shown in **Figure 29** from the Solid Editing toolbar. You are prompted to select the solid. Only one solid can be selected.

After selecting the solid, you have the opportunity to remove faces. If you do not remove

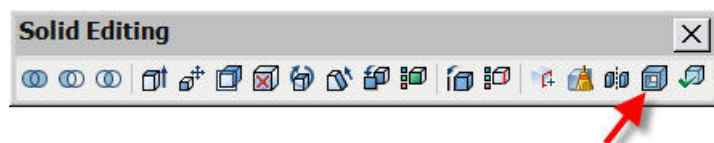


Figure 29. The **SHELL** button on the Solid Editing toolbar



any faces, the new solid object will appear identical to the old solid object when shaded or rendered. The thickness of the shell will not be visible. If you wish to create a hollow object with an opening, select the face to be removed (the opening).

After selecting the object and specifying any faces to be removed, you are prompted to enter the shell offset distance. This is the thickness of the shell. A positive shell offset distance creates a shell on the inside of the solid body. A negative shell offset distance creates a shell on the outside of the solid body. If you shell a solid that contains internal features, such as holes, grooves, and slots, a shell of the specified thickness is placed around those features.

Caution

Using the **SOLIDEDIT** command removes the history from a composite solid. Therefore, the original objects—the subobjects—are no longer available for subobject editing. However, you may still be able to perform some subobject edits, such as moving the original objects.

Creating Composite Solids

A **composite solid** is a solid model constructed of two or more solids, often primitives. Solids can be subtracted from each other, joined to form a new solid, or overlapped to create an intersection or interference. The commands used to create composite solids are found in the **3D Make** control panel in the **Dashboard**, in the **Solid Editing** cascading menu in the **Modify** pull-down menu, and on the **Solid Editing** toolbar.

Introduction to Booleans

There are three operations that form the basis of constructing many complex solid models. Joining two or more solids is called a **union** operation. Subtracting one solid from another is called a **subtraction** operation. Forming a solid based on the volume of overlapping solids is called an **intersection** operation. Unions, subtractions, and intersections as a group are called **Boolean operations**. George Boole (1815–1864) was an English mathematician who developed a system of mathematical logic where all variables have the value of either one or zero. Boole's two-value logic, or *binary algebra*, is the basis for the mathematical calculations used by computers, and specifically for those required in the construction of composite solids.

Joining Two or More Solid Objects

The **UNION** command is used to combine solid objects, **Figure 30**. The solids do not need to touch or intersect to form a union. Therefore, accurately locate the primitives when drawing them. After selecting the objects to join, just press [Enter] and the action is completed.

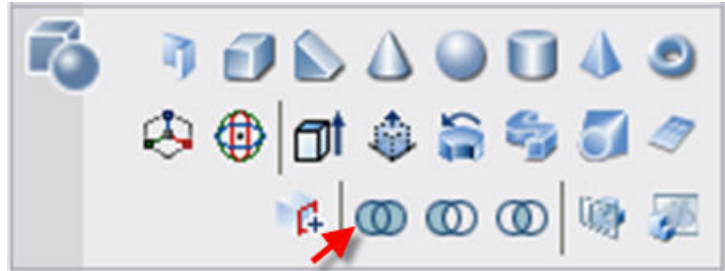


Figure 30. The UNION command on the 3D Make Control Panel

Subtracting Solids

The **SUBTRACT** command allows you to remove the volume of one or more solids from another solid. See **Figure 31**. The first object selected in the subtraction operation is the object *from* which volume is to be subtracted. The next object is the object to be subtracted from the first. The completed object will be a new solid. If the result is the opposite of what you intended, you may have selected the objects in the wrong order. Just undo the operation and try again.

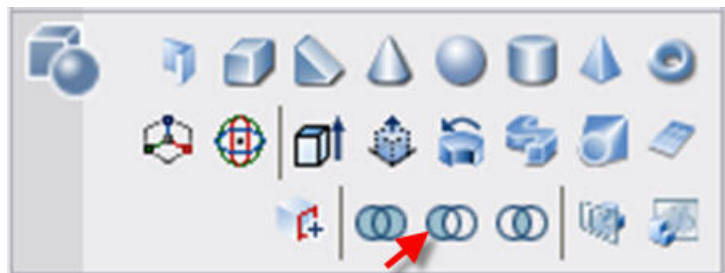


Figure 31. The SUBTRACT command on the 3D Make Control Panel

Creating New Solids from the Intersection of Solids

When solid objects intersect, the overlap forms a common volume, a space that both objects share. This shared space is called an **intersection**. An intersection can be made into a composite solid using the **INTERSECT** command. See **Figure 32**. A solid is formed from the common

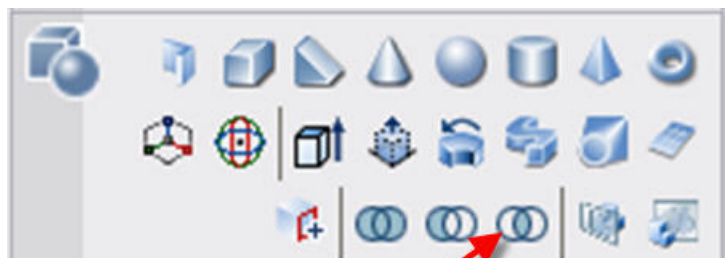


Figure 32. The INTERSECT command on the 3D Make Control Panel



volume. The original objects are removed.

Grip Editing Composite Solids

The Boolean commands (**UNION**, **SUBTRACT**, and **INTERSECT**) create composite solids. Solids that have been modified using any of the options of the **SOLIDEDIT** command also become composite solids. The solid may still look like a primitive, sweep, loft, etc., but it is a composite. The grips available with the previous objects are no longer available, unless performing subobject editing on a composite created with a Boolean command (discussed later in this chapter). Composite solids have a base grip located at the centroid of the base surface. This grip can be used to stretch, move, rotate, scale, or mirror the solid.

Note

When performing grip editing on a solid, AutoCAD must be able to “solve” the end result. If it cannot, the edit is not applied.



EXERCISE #9

1. Well, let's put these last few things together – and even do something we haven't quite covered yet.
2. Draw a BOX with dimensions of 6L x 3W x 2H
3. To the right of the BOX, draw a WEDGE with dimensions of 3L x 2W x 1H
4. Use grips on the WEDGE to “mate” the vertical face of the WEDGE with the end of the BOX. Do not worry about the length of the WEDGE changing while doing the grip edits.
5. We are now going to draw a cylinder on the inclined surface. This is the automatic part! Be sure the DUCS button is pushed in on the status bar. After initiating the CYLINDER command, hover the crosshairs over the inclined surface, while being prompted to select the center point. Notice the crosshairs change orientation to align with the surface. Use the “Midpoint between 2 Points” option from the object snap menu, to center the cylinder between the opposite midpoints or endpoints. Enter 1 for the radius, and “pull” the height of the cylinder to a point below the wedge.
6. We can now subtract the cylinder from the wedge **and** the box, forming a hole – and turning the result into one composite solid. Select the SUBTRACT command. When prompted to select objects to subtract from, select the box and the wedge, and hit [enter]. When prompted to select objects to subtract, select the cylinder, and hit [enter]. Notice that the solids are now one object, a composite solid.
7. Finally, we will hollow out our new solid from the bottom, with .15 thick “walls”. Use your viewing tools to create a view allowing you to see the bottom of the solid; we will need to select it soon. Select the SHELL command from the Solid Editing toolbar. Select the solid when prompted to do so. Select the bottom face when prompted to select faces to be removed, and hit [enter]. Enter .15 when prompted for the shell offset distance. Since “Shell” is actually a nested option to the “Body” option of the SOLIDEDIT command, you will need to hit [enter] to exit the “Body” option, then hit [enter] again to exit the SOLIDEDIT command.



8. Experiment with the various Visual Styles to view your new solid. Be sure to try the “Conceptual” style with the X-ray mode turned on – that is one of my favorite settings for working in a solid modeling environment.

Introduction to Working with User Coordinate Systems

All points in a drawing or on an object are defined with XYZ coordinate values (rectangular coordinates) measured from the 0,0,0 origin. Since this system of coordinates is fixed and universal, AutoCAD refers to it as the **world coordinate system (WCS)**. A **user coordinate system (UCS)**, on the other hand, can be defined with its origin at any location and with its three axes in any orientation desired, while remaining at 90° to each other. The **UCS** command is used to change the origin, position, and rotation of the coordinate system to match the surfaces and features of an object under construction. When set up to do so, the UCS icon reflects the changes in the orientation of the UCS and placement of the origin.

The available options for creating and managing a UCS are found in the **Tools** pull-down menu and related cascading menu, on the **UCS** and **UCS II** toolbars, or by typing the **UCS** command.

Displaying the UCS Icon

Before getting started with the various UCS command options it is best to become familiar with the symbol that identifies the orientation of the coordinate system called the **UCS icon**. When AutoCAD is first launched based on the acad3D.dwt template, the UCS icon is located at the WCS origin in the middle of viewport. The display of this symbol is controlled by the **UCSICON** command. If your drawing does not require viewports and altered coordinate systems, you may want to turn the icon off using the **OFF** option of the command. The icon disappears until you turn it on again using the **ON** option of the command. You can also turn the icon on or off and set the icon to display at the origin using the options in the **Settings** tab of the **UCS** dialog box. This dialog box is displayed by picking the **Named UCS...** button on the **UCS II** toolbar, selecting **Tools>Named UCS...**, or typing the **UCSMAN** command.

Professional Tip

It is recommended that you have the UCS icon turned on at all times when working in 3D



drawings. It provides a quick indication of the current UCS.

Changing the Coordinate System

To construct a three-dimensional object, you must draw shapes at many different angles. Different working planes are needed to draw features on angled surfaces. To construct these features, you must very often manipulate the UCS to match any surface on an object. The following options to the UCS command are some of the more common ways of manipulating the UCS.

Selecting Three Points to Create a New UCS

The **3 Point** option of the **UCS** command can be used to change the UCS to any flat surface. This option requires that you first locate a new origin, then a point on the positive X axis, and finally a point on the XY plane that has a positive Y value. Use object snaps to select points that are not on the current XY plane. After you pick the third point, the point on the XY plane, the UCS icon changes its orientation to align with the angled surface of the base.

Professional Tip

When typing the **UCS** command, enter 3 at the Specify origin of UCS or [Face/NAmed/OBject/Previous/View/World/X/Y/Z/ZAxis] <World>: prompt. Notice that the option is not listed in the prompt.

Naming and Saving a UCS

Once you have created a new UCS that may be used again, it is best to save it for future use. For example, you have created a UCS that you used to draw the wireframe of one side of the object. You can save this UCS using the **Save** option of the **UCS** command or the **UCS** dialog box.

If using the dialog box, right-click on the entry Unnamed and pick **Rename** in the shortcut menu. You can also pick once or double-click on the highlighted name. Then, type the new name in place of Unnamed and press [Enter]. A name can have up to 255 characters. Numbers, letters, spaces, dollar signs (\$), hyphens (–), and underscores (_) are valid. Use this method to save a new UCS or to rename an existing one. Now, the coordinate system is saved and can be easily recalled for future use.

Professional Tip

Most drawings can be created by rotating the UCS as needed without saving it. If the drawing is



complex with several planes, each containing a large amount of detail, you may wish to save a UCS for each detailed face. Then, restore the proper UCS as needed. For example, when working with architectural drawings, you may wish to establish a different UCS for each floor plan and elevation view, and for roofs and walls that require detail work.

Setting the UCS to an Existing Object

The **Object** option of the **UCS** command can be used to define a new UCS on an object. However, there are some objects on which this option cannot be used: 3D polylines, 3D meshes, and xlines. There are also certain rules that control the orientation of the UCS. For example, if you select a circle, the center point becomes the origin of the new UCS. The pick point on the circle determines the direction of the X axis. The Y axis is relative to X, and the UCS Z axis is the same as the Z axis of the selected object.

Setting the UCS to the Face of a 3D Solid

The **Face** option of the **UCS** command allows you to orient the UCS to any face on a 3D solid object. Select the command and then pick a face on the solid. After you have selected a face on a 3D solid, you have the options of moving the UCS to the adjacent face or flipping the UCS 180° on the X, Y, or both axes. Use the **Next**, **Xflip**, or **Yflip** options to move or rotate the UCS as needed. Once you achieve the UCS orientation you want, press [Enter] to accept. Be aware there are many different UCS orientations that can be selected for a single face.

Dynamic UCS

A powerful tool for 3D modeling is the **dynamic UCS function**. A dynamic UCS is a UCS temporarily located on any existing face of a 3D model. The function is activated by picking the **DUCS** button in the status bar, pressing the [Ctrl]+[D] key combination, or setting the **UCSDETECT** system variable to 1. When the pointer is moved over a model surface, the XY plane of the UCS is aligned with that surface. This is especially useful when adding primitives or shapes to model surfaces. In addition, dynamic UCSs are useful when inserting blocks and xrefs, locating text, editing 3D geometry, editing with grips, and area calculations.

An example of using dynamic UCS is to draw a cylinder on the angled face of an object, say a wedge. First, select the **CYLINDER** command. Make sure the dynamic UCS function is on. Then, move the pointer over one of the surfaces of the object. Notice that the 3D crosshairs change when they are moved over a new surface. The red (X) and green (Y) crosshairs are flat



on the face. For ease of visualizing the 3D crosshairs as they are moved across different surfaces, right-click on the **DUCS** button in the status bar and select **Display crosshair labels** to turn on the XYZ labels on the crosshairs.

As you move the pointer over the object faces, note that hidden faces are not highlighted, therefore you cannot work on those faces. If you wish to work on a hidden face you must first change the viewpoint to make that face visible.

The **CYLINDER** command is currently prompting to select a center point of the base. If you pick a point, this sets the center of the cylinder base *and* temporarily relocates the UCS so its XY plane lies on the selected face. Once the point is selected and the dynamic UCS created, the UCS icon moves to the temporary UCS. When the command is ended, the UCS and UCS icon revert to their previous locations. To locate a cylinder in the center of the angled face, use the following procedure.

1. At the “specify center point of base” prompt, [Shift] + right-click in the drawing area and pick **Mid Between 2 Points** from the object snap shortcut menu.
2. Use the **Endpoint** snap to pick two opposite corners of the angled face. You can also use the **Midpoint** snap and pick the midpoint of the two sides or top and bottom edges. The UCS is temporarily moved to the angled face at the pick point.
3. Specify the diameter for the base.
4. Specify the height for the cylinder.
5. The cylinder is located properly on the angled face and the UCS automatically returns to the previous location.

When setting a dynamic UCS, experiment with the behavior of the crosshairs as they are moved over different surfaces. The orientation of the crosshairs is related to the edge of the face that they are moved over. Can you determine the pattern by which the crosshairs are turned? The X axis of the crosshairs is always aligned with the edge that is crossed.

Professional Tip

If you want to temporarily turn off the dynamic UCS function while working in a command, press and hold the [Shift]+[Z] key combination while moving the pointer over a face. As soon as you release the key, dynamic UCS function is reinstated.

Preset UCS Orientations

AutoCAD has six preset orthographic UCSs that match the six standard orthographic views.



These orientations can be selected in the drop-down list that appears on the **UCS II** toolbar, in the **Tools** pull-down menu, or in the **Orthographic UCSs** tab of the **UCS** dialog box.

The **Relative to:** drop-down list at the bottom of the **Orthographic UCSs** tab of the **UCS** dialog box specifies whether the orthographic UCS is relative to a named UCS or absolute to the WCS. For example, suppose you have a saved UCS named Front Corner that is rotated 30° about the Y axis of the WCS. If you set the top UCS current relative to the WCS, the new UCS is perpendicular to the WCS. However, if the top UCS is set current relative to the named UCS Front Corner, the new UCS is also rotated from the WCS.

Professional Tip

Changing the **Relative to:** setting affects *all* preset UCSs and *all* preset viewpoints! Therefore, leave this set to **World** unless absolutely necessary to change it.

UCS Settings and Variables

As discussed in the previous section, the **UCSFOLLOW** system variable allows you to change how an object is displayed in relation to the UCS. There are also system variables that display a variety of information about the current UCS. These variables include:

- **UCSAXISANG.** (Stored value) The default rotation angle for the **X**, **Y**, or **Z** option of the **UCS** command.
- **UCSDETECT.** (On or off) Turns the dynamic UCS function on and off. The **DUCS** button on the status bar controls this variable, as does the [Ctrl]+[D] key combination.
- **UCSORTHO.** (On or off) If set to 1 (on), the related orthographic UCS setting is automatically restored when an orthographic view is restored. If turned off, the current UCS is retained when an orthographic view is restored.
- **UCSNAME.** (Read only) Displays the name of the current UCS.
- **UCSORG.** (Read only) Displays the XYZ origin value of the current UCS.
- **UCSVIEW.** (On or off) If this variable is set to 1 (on), the current UCS is saved with the view when a view is saved. Otherwise, the UCS is not saved with the view.
- **UCSXDIR.** (Read only) Displays the XYZ value of the X axis direction of the current UCS.
- **UCSYDIR.** (Read only) Displays the XYZ value of the Y axis direction of the current UCS.

UCS options and variables can also be managed in the **Settings** tab of the **UCS** dialog box.



The settings in this tab are:

- **Save UCS with viewport.** If checked, the current UCS settings are saved with the viewport and the **UCSVP** system variable is set to 1. This variable can be set for each viewport in the drawing. Viewports in which this setting is turned off, or unchecked, will always display the UCS settings of the current active viewport.
- **Update view to Plan when UCS is changed.** This setting controls the **UCSFOLLOW** variable. When checked, the variable is set to 1. When unchecked, the variable is set to 0.