

Chapter 11

Solid Problem Ten

In this chapter, you will learn the following to World Class standards:

- Sketch of Solid Problem Ten
- Starting a 3D part drawing
- Modifying UCS settings
- Creating a perimeter using the Polyline command
- Adding a beveled edge using the Chamfer command
- Extruding a solid from a closed polyline
- Rotating a 3D solid using the 3D Rotate command
- Drawing two solid cylinders
- Drawing a Solid Box
- Moving a solid into position on the master solid
- Subtracting 3D solids from the master solid
- Hiding the solid
- Moving the solid to the origin point
- Saving the solid problem

Starting a 3D Part Drawing

You will draw Problem Eight using the Mechanical template. First select the New tool on the Quick Access Toolbar. A list of templates is loaded in the “Select template” window. Scroll down the list and select Mechanical.dwt. A new drawing file will open that contains the system variables and layer definitions which you will use later in Paper Space to finish the drawing.

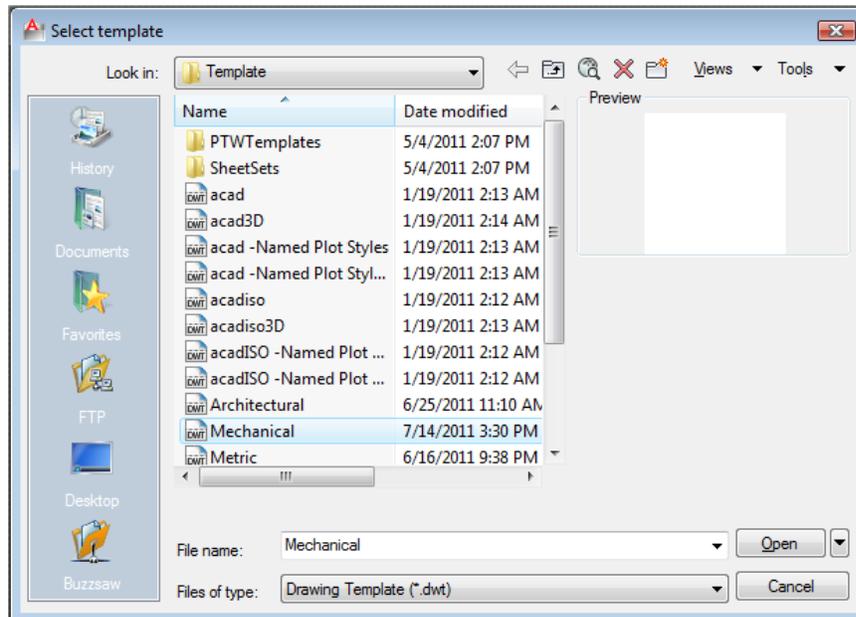


Figure 11.2 – Starting the drawing using the Mechanical template

Modifying UCS Settings

You may notice the UCS icon moving on the graphical display as you zoom or pan your view. If so, the UCS is currently representing the origin (0,0,0) of the drawing area. This can sometimes be distracting, especially if the origin happens to lie in the vicinity of your drawing. You can change the UCS settings so that the icon is always displayed in the bottom left-hand corner of the drawing area. Select the Show UCS Icon tool from the Coordinates panel under the Home tab (Figure 11.3) and select the “Show UCS Icon” option (Figure 11.4). You also have the choice of keeping the UCS displayed at the origin or turning it off completely.

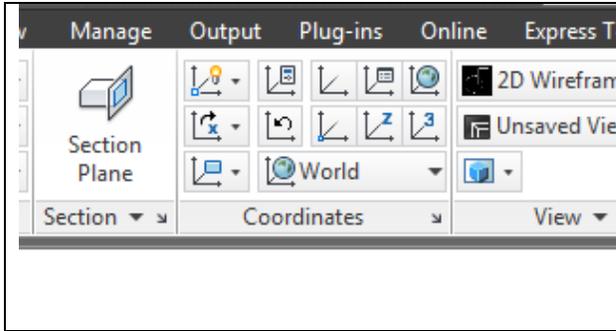


Figure 11.3 - The Coordinates panel

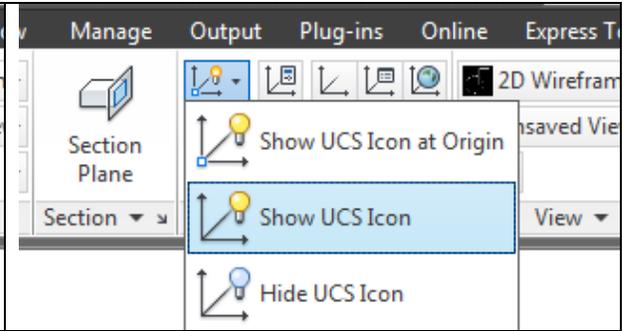
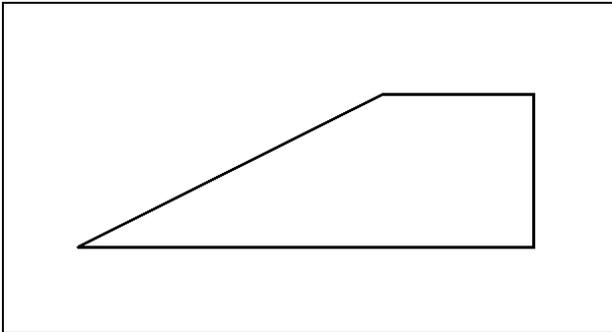


Figure 11.4 - Changing the UCS display settings

Creating a Perimeter Using the Polyline Command

You will use the Polyline command to draw a complex solid perimeter that includes an angled side. However, you will not have to calculate any of the angles.



With the Ortho mode “on”, draw a **6.0** unit line to the right, a **2.0** line up, a **2.0** line to the left, and then type “C” and ENTER to close the Polyline. AutoCAD computes the last line length and angle when you use the close option. A closed entity will appear in the graphical display as shown in Figure 11.5. Use the navigation bar to zoom or pan the drawing as necessary.

Figure 11.5 – The closed perimeter

Adding a Beveled Edge Using the Chamfer Command

Use the Chamfer command to add a beveled edge to a corner of the perimeter. In a previous problem, you placed the chamfer on a 3D solid edge, but in this exercise you will change the Polyline entity by placing the beveled edge in the perimeter before extruding.



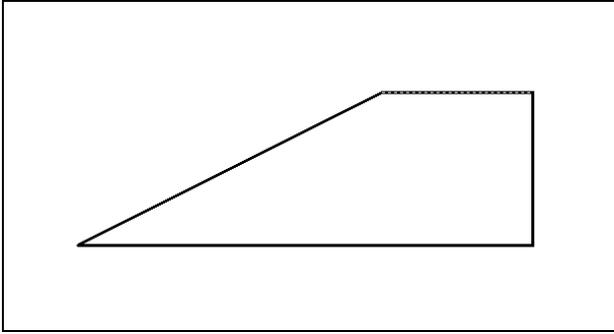


Figure 11.6 – Selecting the first line

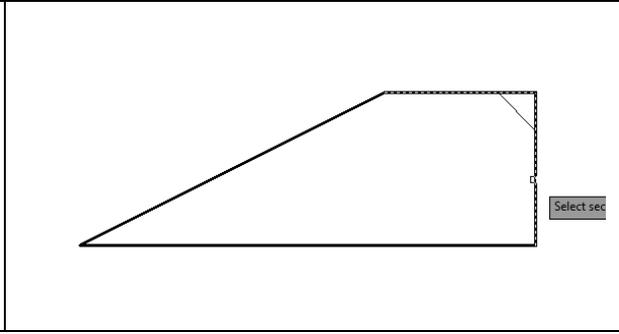


Figure 11.7 – Selecting the second line

Pick the Chamfer tool on the Modify panel by clicking on the drop-down arrow next to the Fillet tool and selecting the command. Type “**D**” for distance and hit ENTER. Both of the chamfer distances will be 0.5, so type “**0.5**” and ENTER for each distance. You will then be prompted to select the first line, so pick the top vertical line (Figure 11.6). Select the horizontal line as the second line (Figure 11.7). After selecting the second line, the corner will be replaced with a diagonal line connecting the edges as shown in Figure 11.8.

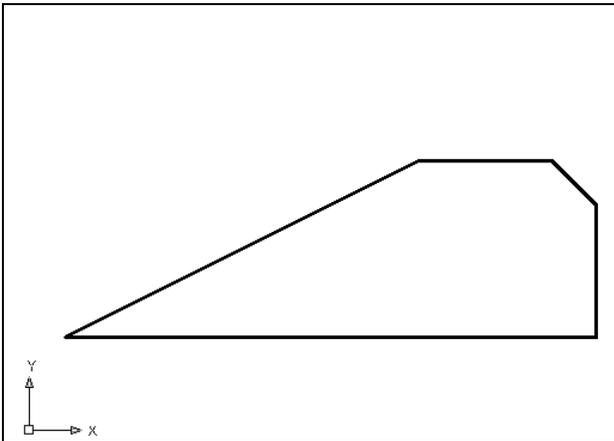


Figure 11.8 – The chamfered edge

Remember that you can use the Chamfer command in 2D or 3D mode. When you apply the tool in the 2D mode, you must select two lines to describe the corner that will be chamfered. When you are utilizing the tool in 3D, you only select the edge because its line represents the corner that will be chamfered. Both methods work equally well. You will find that many AutoCAD commands will allow you to accomplish the same tasks in different ways.

Extruding a Solid from Closed Polylines and Circles

Select the Southwest Isometric View tool to allow you to see the part on the XY plane with the Z axis rising vertically as shown in Figure 11.9. You can only extrude closed entities like Polylines and Circles using this very powerful tool on the Modeling panel.



Extrude

Select the Extrude tool on the Modeling panel and the command line will prompt you to “**select objects**”. Select the closed polyline and the command line will return with “**1 found**”. Hit **ENTER** to specify the height or path of the solid. Type “**2.5**” and **ENTER** for the height of the extrusion. The outline will expand 2.5 units into the Z axis as shown in Figure 11.10.

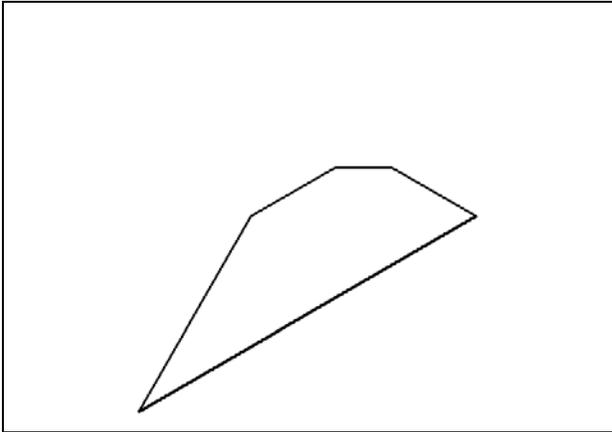


Figure 11.9 – SW Isometric View

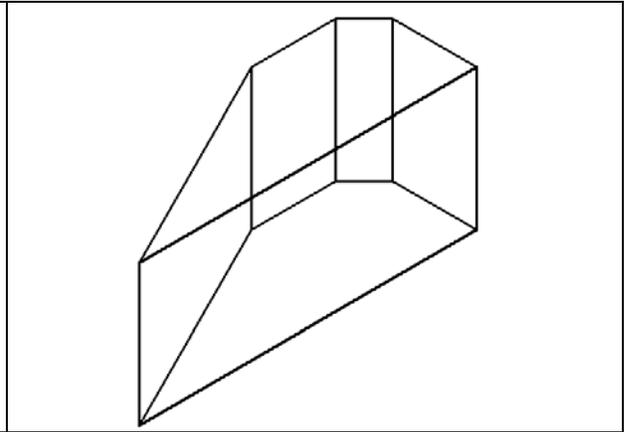


Figure 11.10 – The extruded solid

Rotating a 3D Solid Using the 3D Rotate and Rotate Commands

Select the 3D Rotate command on the Modify panel. At the prompt “select objects”, pick the solid on the graphical display and hit **ENTER**. Now select the midpoint Osnap of the bottom edge as the base point of rotation. Specify the X axis as the axis of rotation by selecting the circle that orbits the X axis (Figure 11.11). With the Ortho mode “on”, specify an angle start point and then use the cursor to move the solid 90 degrees clockwise (Figure 11.11). Finish the rotation by clicking with the left mouse button and the solid will appear rotated on its base as shown in Figure 11.12.

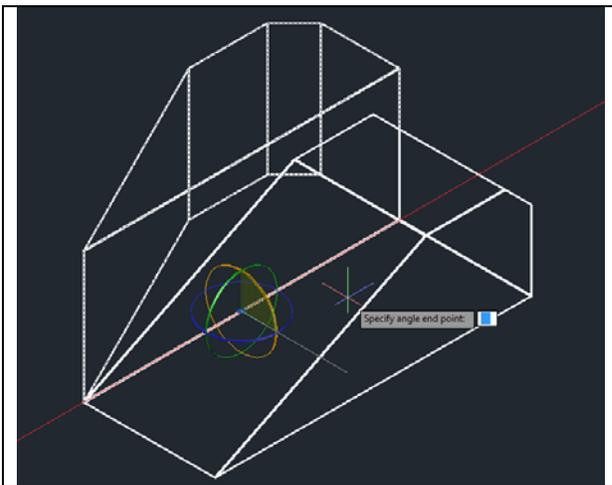


Figure 11.11 – Specifying angle end point

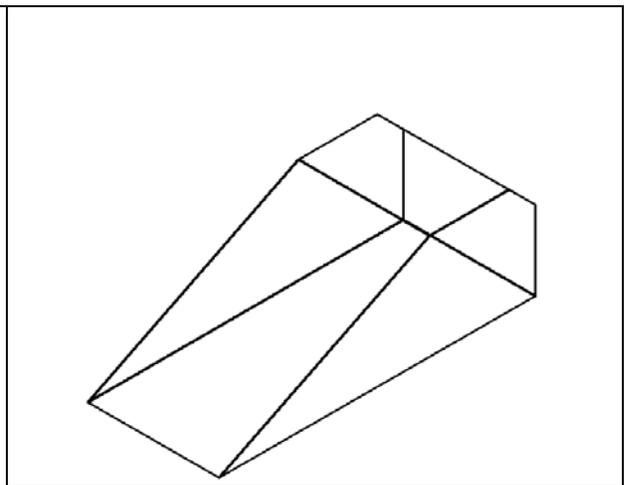


Figure 11.12 – The rotated solid

Drawing Two Solid Cylinders

You need to add two holes to the master solid: one 0.5 diameter hole going completely through the master solid, and a second 1.0 diameter hole called a counterbore. You will use the Cylinder command to create these features.

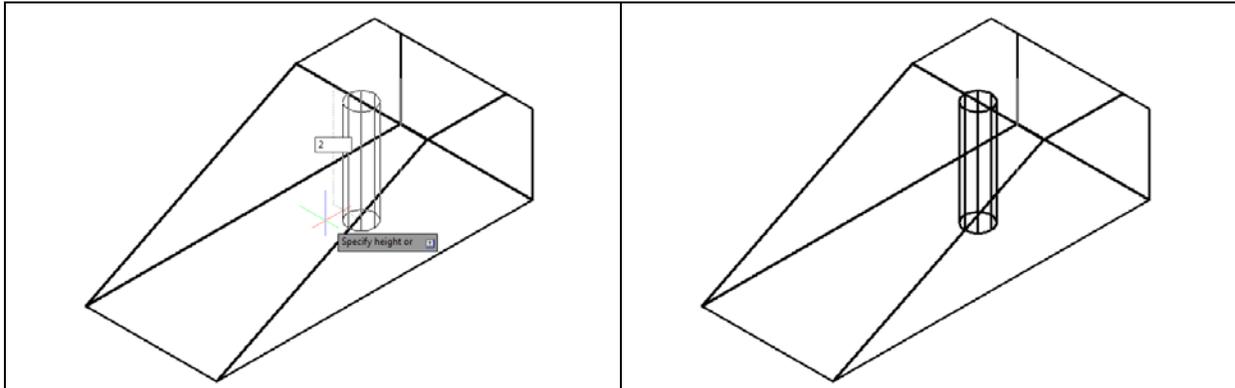


Figure 11.13 – Placing the cylinder

Figure 11.14 – The first cylinder

Select the Cylinder tool on the Modeling panel under the drop-down arrow beneath the Box command. Specify the center point for the base of the cylinder as the midpoint Osnap of the front edge of the top face of the master solid. Type “**D**” for diameter and hit **ENTER** and input the sketch’s measurement of **0.5**. Hit **ENTER** to go to the next prompt, which is to “specify the height of the cylinder”. With Dynamic Input toggled “on”, pull the cylinder down into the master solid, type **2** for the height of the cylinder and hit **ENTER** (Figure 11.13). The cylinder will appear as shown in Figure 11.14. If you do not have Dynamic Input on type -2 to pull the cylinder -2 units on the Z axis.

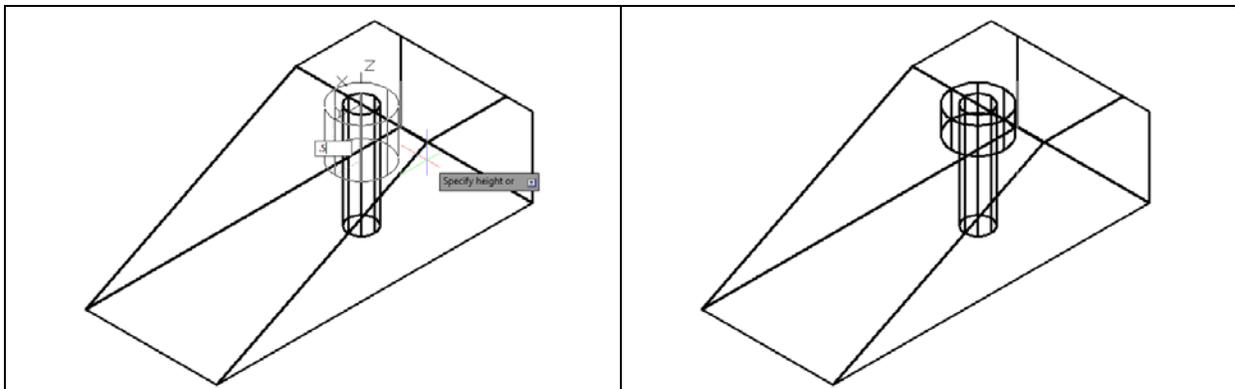


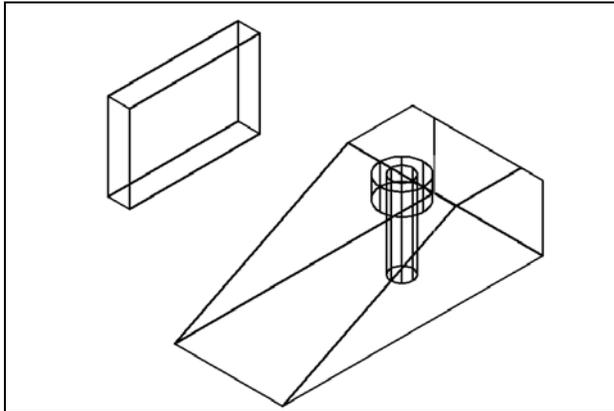
Figure 11.15 – Adding the second cylinder

Figure 11.16 – Two placed cylinders

Hit **ENTER** to repeat the Cylinder command and use the same midpoint Osnap for the base point. Type “**D**” for diameter and hit **ENTER**. Type **1.0** and **ENTER** to continue to the next prompt, which is to “specify the height of the cylinder”. With Dynamic Input toggled “on”, pull the cylinder down into the master solid, type **0.5** for the height of the cylinder and hit **ENTER** (Figure 11.15). The cylinder will appear as shown in Figure 11.16.

Drawing Solid Boxes

There is a slot in the front of the wedge which you will make using a solid box. You can create the rectangular solid anywhere on the graphical display without interfering with the master wedge.



Select the Box command from the Modeling panel and select a point to the right of the master solid as the first corner of the box. Type **L** and ENTER after specifying the corner on the graphical display to utilize the Length option of the Box command. Type **3.0** and ENTER as the length of the box. Type **0.5** and ENTER for the width, and **2.0** and ENTER for the height. The box will appear as shown in Figure 11.17.

Figure 11.17 – Adding a solid box

Moving a 3D Solid into Position on the Master Solid

Before you can subtract the solid box from the master solid, you must move it into the proper position. Select the Move command from the modify toolbar to begin.

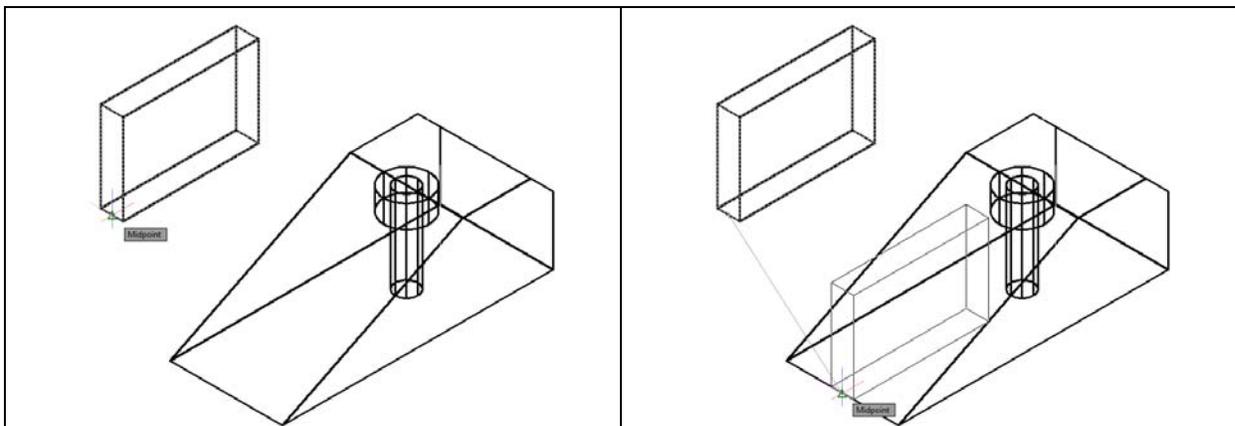


Figure 11.18 – Selecting the base point

Figure 11.19 – The displacement point

Select the box as the object you wish to move. The command line will respond with “1 found” so hit ENTER. Next, you need to “specify base point or displacement” by selecting the midpoint Osnap of the bottom left edge of the box (Figure 11.18). For the “second point of displacement”, select the midpoint Osnap of the front edge of the master solid as shown in Figure 11.19.

Subtracting 3D Solids from the Master Solids

To remove the cylinders and the box from the master solid, select the Subtract tool on the Solid Editing panel.

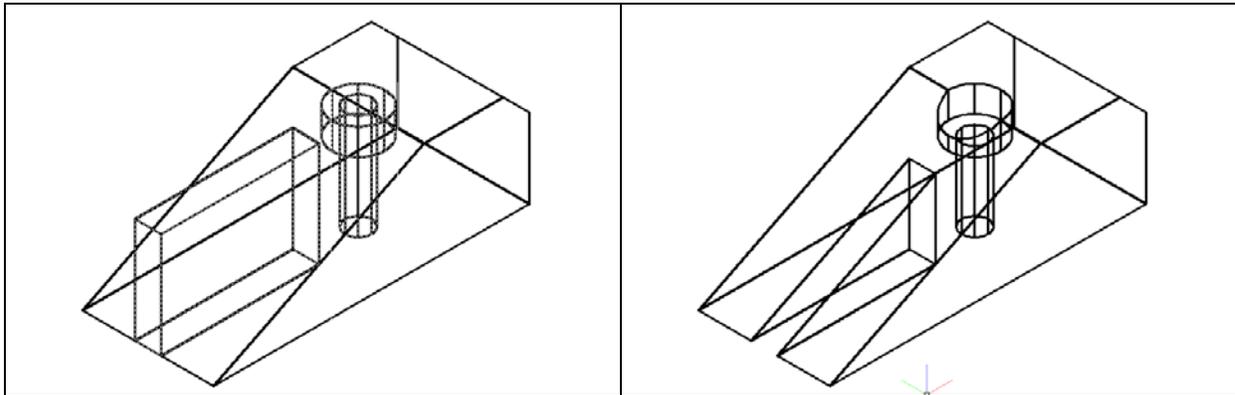
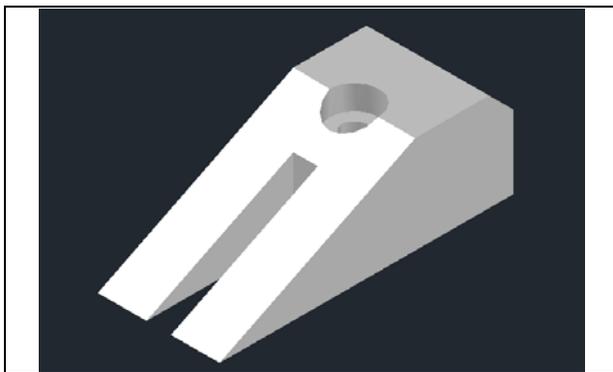


Figure 11.20 – Selecting objects to subtract **Figure 11.21 – The subtracted solids**

The command line will prompt you to “**Select solids and regions to subtract from**”. Select the master solid the system will respond with “**1 found**”. Hit **ENTER** to proceed to the next step, which is to “**Select solids and regions to subtract**”. Pick the two cylinders inside of the master solid as well as the box and the command line will respond with “**3 found**” (Figure 11.20). Hit **ENTER** and the shape of the two cylinders will disappear from the master solid as shown in Figure 11.21.

Hide the 3D Solid

In Figure 11.21, you will notice that the tessellation lines representing the solid’s outline on the far edge of the part are visible. In a real world solid part, you would not be able to see through the solid unless its material was transparent. You can use a different visual style to remove the hidden lines from your graphical display.



Select the Realistic visual style from the Visual Style panel under the View tab. The part will appear rendered with a metallic sheen as shown in Figure 11.22. If you wish to return the graphical display to the previous visual style, choose the visual style called 2D Wireframe. Neither command alters the mass of the solid or any of the dimensions, but both are available for the computer aided designer as a viewing option.

Figure 11.22 – The Realistic visual style

Moving the Solid to the Origin Point

To move Problem Ten to the drawing origin of 0,0,0 on the x, y and z axis, select the Move tool from the Modify panel under the Home tab, select the 3D solid, and hit ENTER. At the prompt “specify base point or displacement,” select the bottom left endpoint Osnap as shown in Figure 11.23, which will become its insertion point into an assembly drawing. At the “second point of displacement,” type 0,0,0 and ENTER, or #0,0,0 and ENTER if using Dynamic Input. Problem Ten will move to the origin point. Moving a part at the origin is important for when you wish to insert the solid into an External Referenced Assembly drawing.

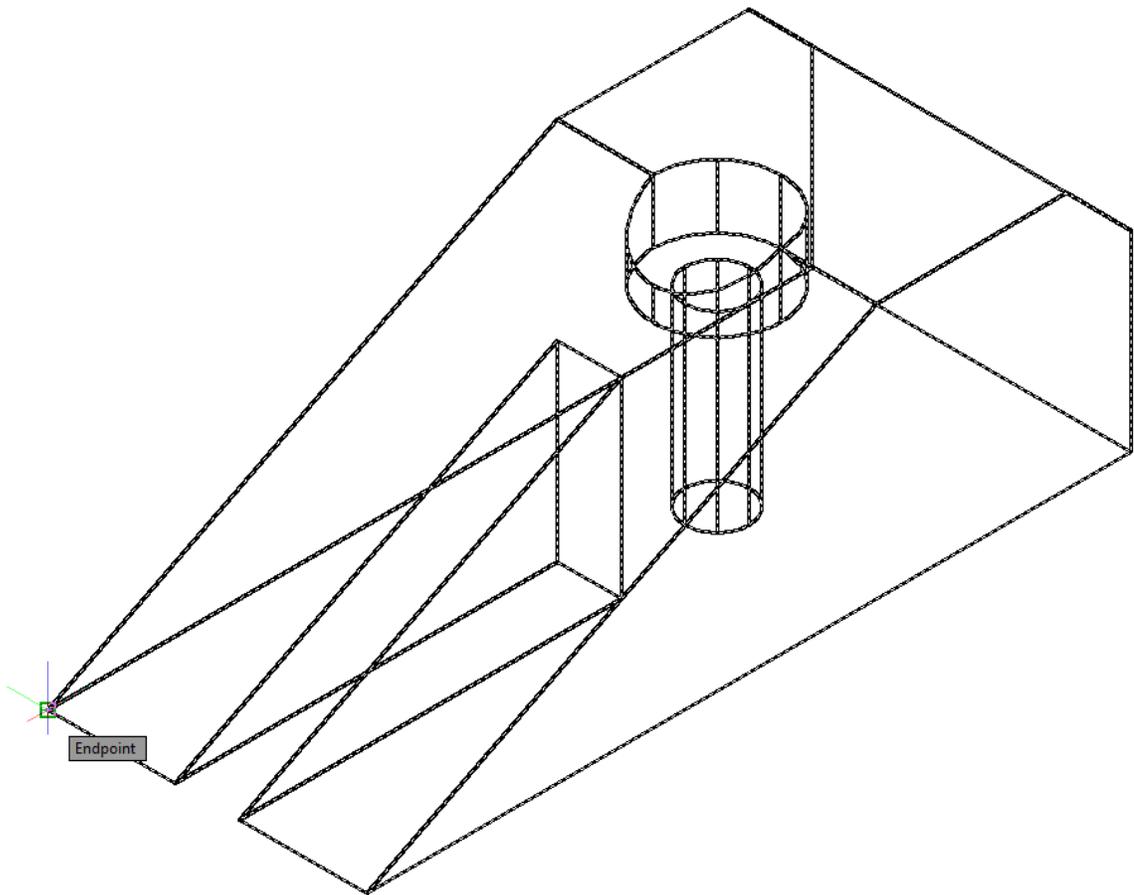


Figure 11.23 – Moving the finished solid to the origin

Saving the Solid Problem

To save Problem 10 in your Fundamentals of 3D Drawing Folder, select the Save As tool on the Quick Access toolbar. The Save Drawing As window will appear in your graphical display. In the Save In list box, select your drawing folder. Type “Problem 10” in the File Name textbox and hit the Save button to save the drawing (Figure 11.24).

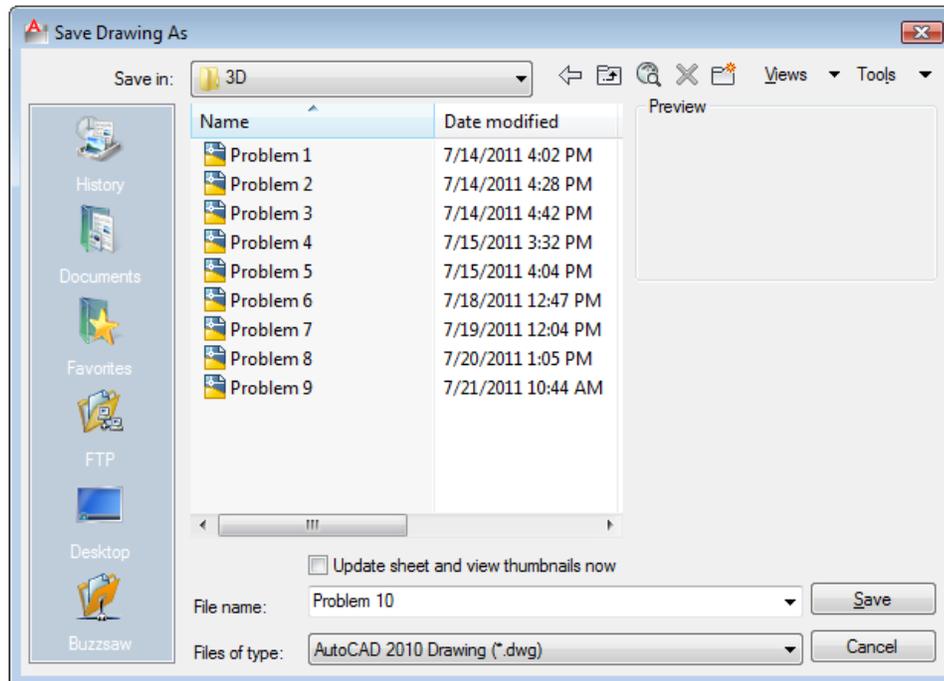


Figure 11.24 – Saving Problem Ten

*** World Class CAD Challenge 03-10 * - Close this drawing file. Create a New file and draw the solid using the techniques in this chapter. Move the finished solid to the origin of the drawing. Complete the task in less than 5 minutes. Continue this drill four times, each time completing the drawing in less than 5 minutes to maintain your World Class ranking.**

*** World Class CAD Challenge * - Report your best times to World Class CAD at www.worldclasscad.com to obtain your world class ranking.**