

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 How the UCS Icon is Displayed
- Creating a Perimeter Using the Polyline Command
- Add a Bevel Edge Using the Chamfer Command
- Extruding a Solids from a Closed Polyline
- Rotating a 3D Solid Using the Rotate3D and Rotate Commands
- Drawing Two Solid Cylinders
- Drawing a Solid Box
- Moving a Solid into Position on the Master Solid
- Subtracting 3D Solids from the Master Solid
- Hide the Solid
- Moving the Solid to the Origin Point
- Saving the Solid Problem

Starting a 3D Part Drawing

In problem ten, you will begin the 3D drawing by selecting the Mechanical template. In Figure 11.2, the window “Select template” is loaded by selecting the New tool on the Quick Access Toolbar. A list of templates is loaded in the Select a Template list box. 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.

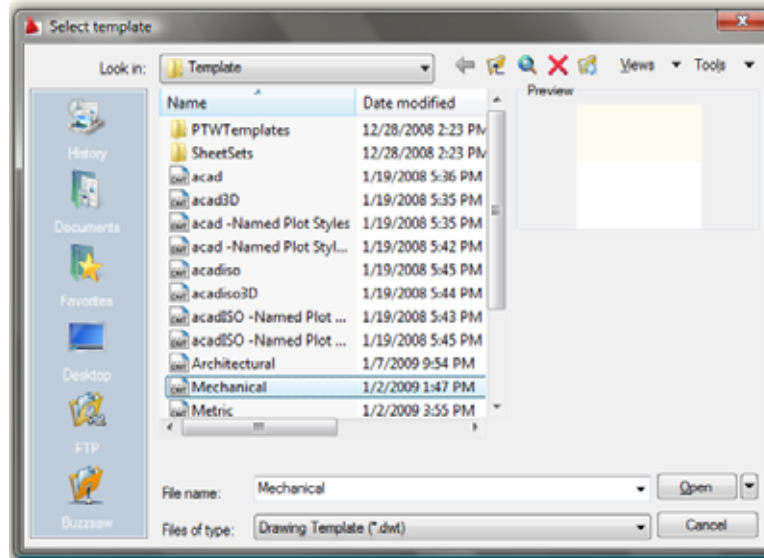


Figure 11.2 – Starting the Drawing Using the Metric Template

Modifying How the UCS Icon is Displayed

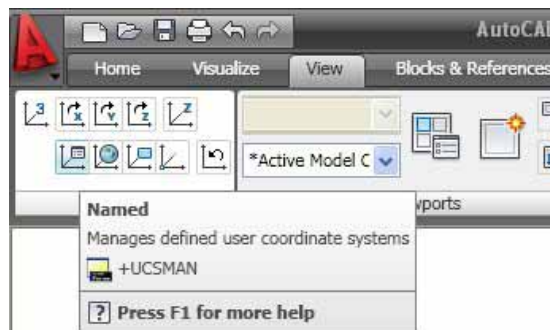


Figure 11.3 - The UCS Settings

When working in AutoCAD, you may notice the UCS icon moving on the graphical display showing the precise location of the origin. Many computer aided design operators do not appreciate having addition UCS lines in with their drawing or solid, so you can set the system to show the UCS in the lower left hand corner of the display.



Select the Named tool on the Coordinates panel under the View tab and the UCS window will appear on graphical display. Pick the Settings tab to change the UCS Icon settings. Remove the check from “Display at UCS origin point” as shown in Figure 11.4 and press the OK button to have the system variable change. Now you can start to create the solid.

Select the Named tool on the Coordinates panel of the View tab and the UCS window will appear on the graphical display. Pick the Settings tab to change the UCS Icon settings. Remove the check from “Display at UCS origin point” as shown in Figure 8.4 and press the OK button to have the system variable change. Now you can start to create the solid.

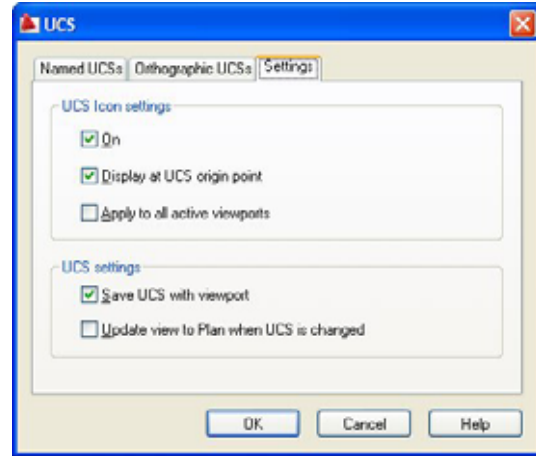


Figure 11.4 - The UCS Window, Settings Tab

Creating a Perimeter Using the Polyline Command

Drawing a closed Polyline is not new to you, and this shape will end with the sharp slope of the geometric silhouette. Pick the Polyline tool on the Draw panel to begin the process.

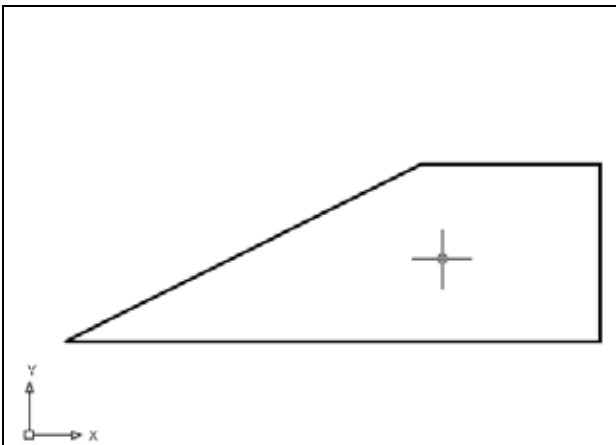


Figure 11.5 – Polyline Edit the Perimeter

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” 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. Select the Zoom tool on the Standard toolbar to have the box fill the monitor, so you can proceed to the next step in the drawing process.

Add a Bevel Edge Using the Chamfer Command

To add a beveled edge to a corner, you can use the Chamfer command to create the feature rather than drawing a line at an angle. In a previous problem, you place the chamfer on a 3D solid edge, but on this exercise, you will change the Polyline entity by placing the beveled edge in the perimeter.

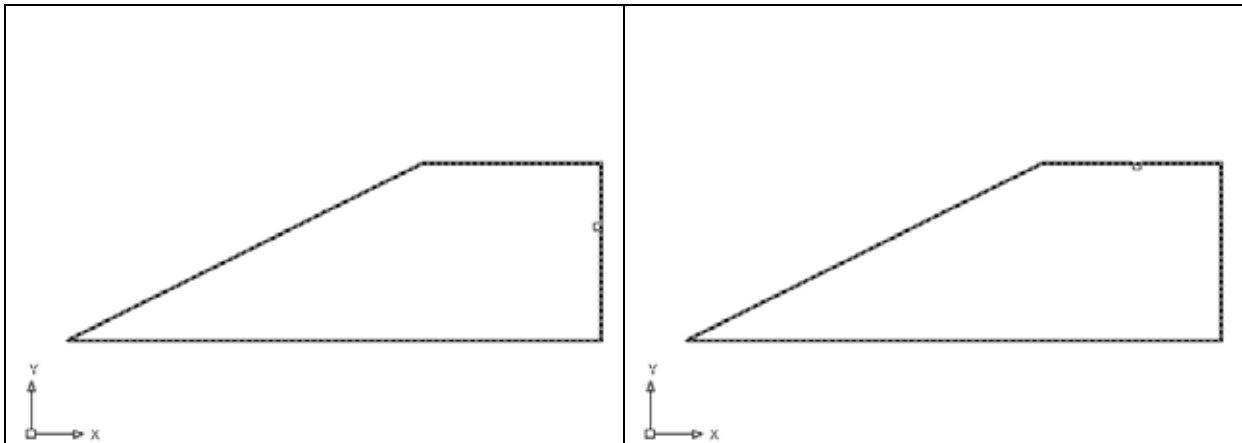


Figure 11.6 – Selecting the First Line

Figure 11.7 – Selecting the Second Line

Pick the Chamfer tool under the drop-down arrow next to the Fillet tool on the Modify panel. 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 vertical line on the right. Select the top horizontal line as the second line. After selecting the second line, the corner will be replaced with a diagonal line connecting the edges, as shown in Figure 11.8.

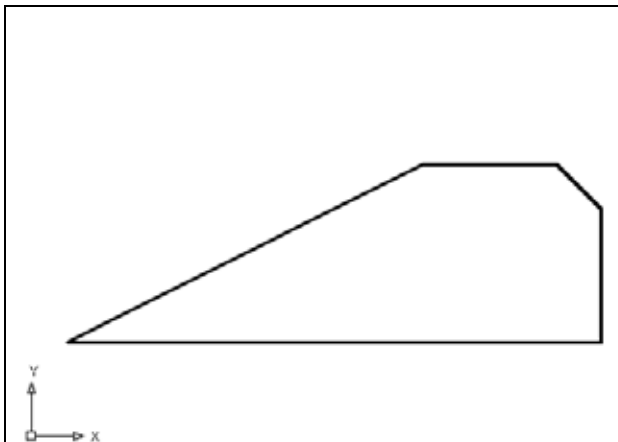


Figure 11.8 – Chamfer the Edge

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 bevel edge. When you are utilizing the tool in the 3D manner, you only select the edge, which is represented by a single tessellation line. Either method will result in the same result. Because of the many commands in the AutoCAD program, you can accomplish similar tasks using different approaches.

Extruding a Solid from Closed Polylines and Circles

On the View panel, select the Southwest Isometric View tool to allow you to see the part on the XY plane and the vertical Z-axis will rise upward 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.



Select the Extrude tool on the Modeling panel and the command line will prompt you to “select objects”. Pick the perimeter of the Polyline and the command line will return with “1 found”. Press **Enter** to specify the height or path of the solid. You will type “**2.50**” for the height of the extrusion and for the angle of taper; you will **Enter** to accept the zero degree default. The 3D solid will appear in the graphical display as shown in Figure 11.10.

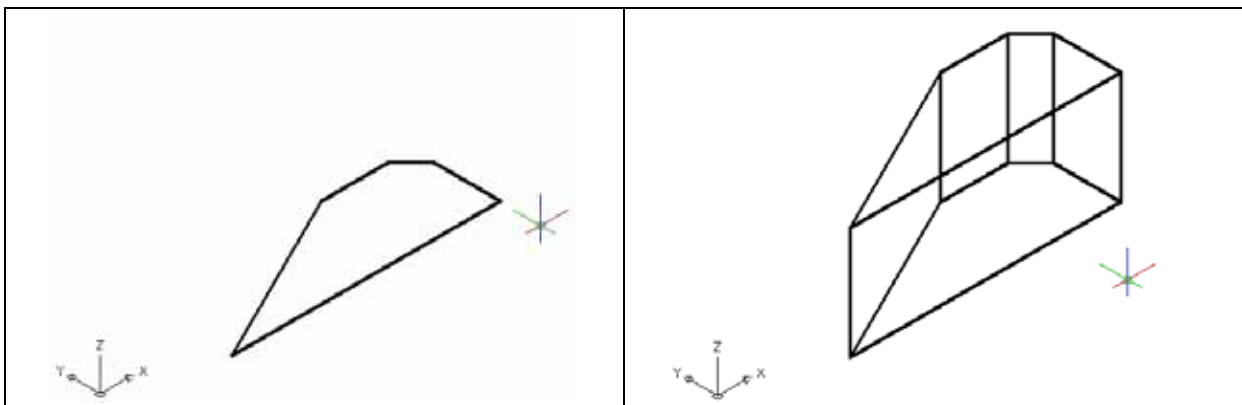


Figure 11.9 – SW View

Figure 11.10 – Extruded solid

Rotating a 3D Solid Using the Rotate3D 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** to proceed to the second part of the function. You are going to rotate the solid part on the x-axis, so pick a point lower edge of the solid, and then select the x-axis diameter. Type **-90** to rotate the solid as shown in Figure 11.12.

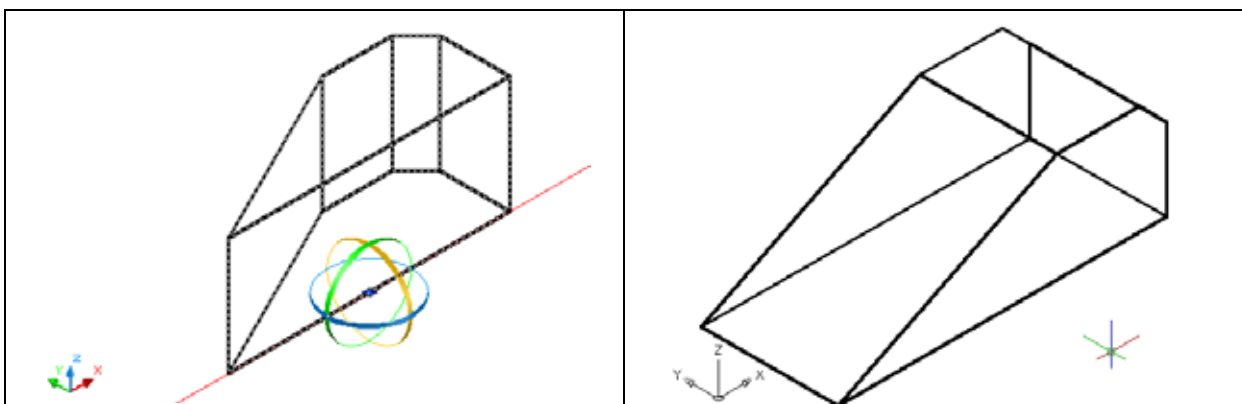


Figure 11.11 – Rotate3D the Solid

Figure 11.12 – Rotated Solid

Drawing Two Solid Cylinders

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

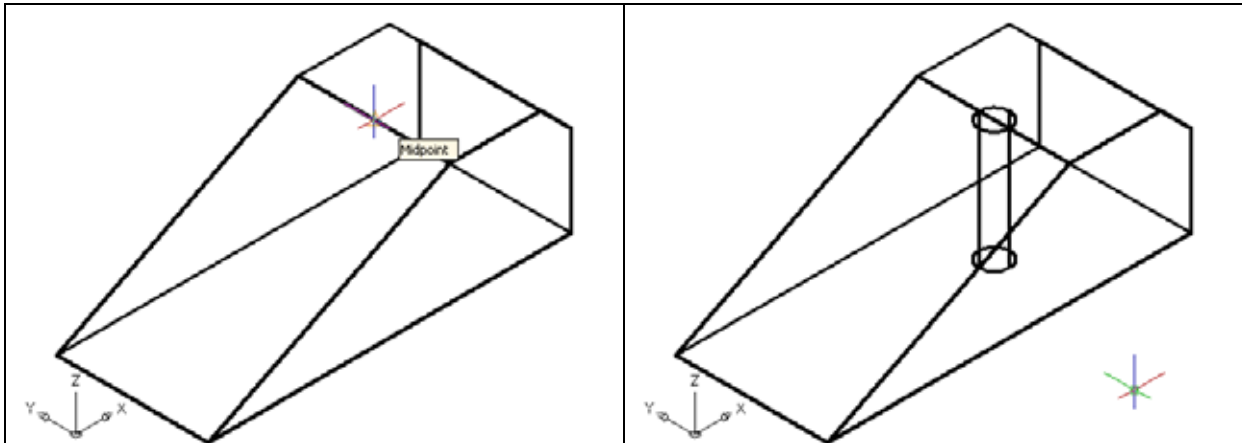


Figure 11.13 – Selecting the Midpoint

Figure 11.14 – The First Cylinder

Pick the Cylinder tool in the left drop-down box on the Modeling panel. Specify the center point for the top of the cylinder as the Midpoint on the solid wedge as shown in Figure 11.13. You need to type “**D**” for diameter and hit **Enter** to input the sketch’s measurement of **0.5**. Hit **Enter** to the next prompt, which is “specify the height of the cylinder”. Key in **-2.0** for the elevation and hit Enter, and the cylinder will appear in the graphical display growing in the negative direction as shown in Figure 11.14.

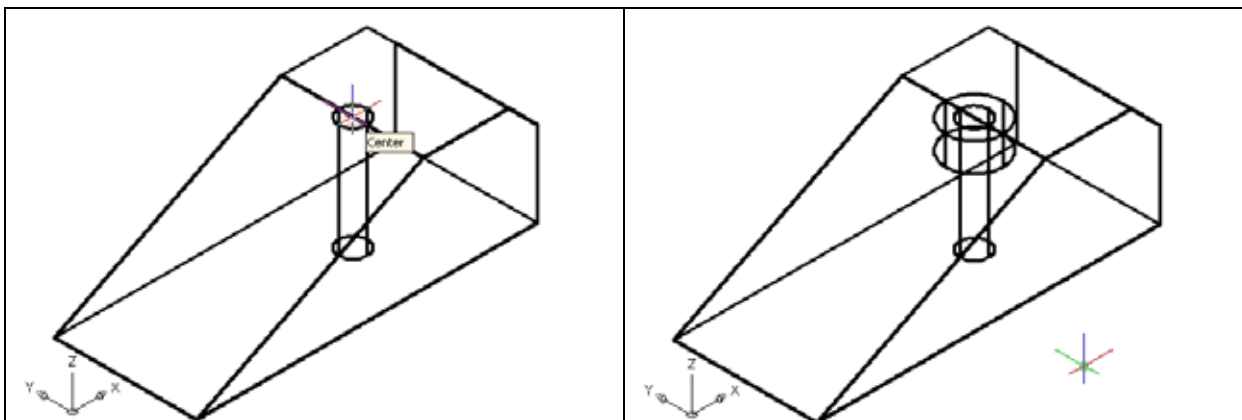


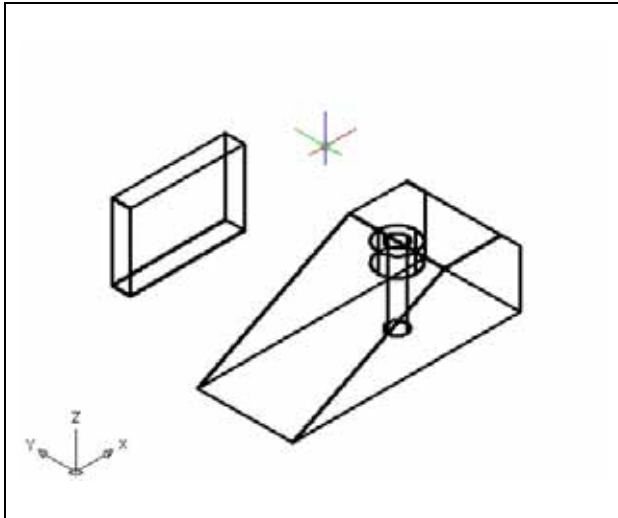
Figure 11.15 – Adding a Solid Cylinder

Figure 11.16 – Adding a Solid Cylinder

Press the **Enter** key to repeat the Cylinder command. Pick the Center object snap on the top of the cylinder as shown in Figure 11.15. You need to type “**D**” for diameter and hit **Enter** to input the sketch’s measurement of **1.0**. Hit **Enter** to the next prompt, which is “specify the height of the cylinder”. Key in **-0.5** for the elevation and hit **Enter**, and the cylinder will appear in the graphical display growing in the negative direction as shown in Figure 11.15.

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 tool on the Modeling panel. At the command line, a prompt asks you to specify the corner of the solid. For this box, specify any point to the left of the solid. Type **L** after specifying the starting point and the system will ask you three questions. The first prompt is “Specify Length” where you will type **3.0** and Enter for the X coordinate. The second prompt is “Specify Width” where you will type **0.5** for the y coordinate. The third prompt is “Specify Height” where when you type **2.0** and Enter for the Z coordinate. The second box will appear as shown in Figure 11.17.

Figure 11.17 – Adding a Solid Box

Moving 3D Solids into Position on the Master Solid

The Move command is actually a three-dimensional function in the modern AutoCAD application software program. The way you can determine whether a command is three dimensional in nature is to attempt to use the function outside the Z-plane, that is the Z coordinate is not 0. You can use move to relocate any entity to another point (X,Y,Z) in Model Space.

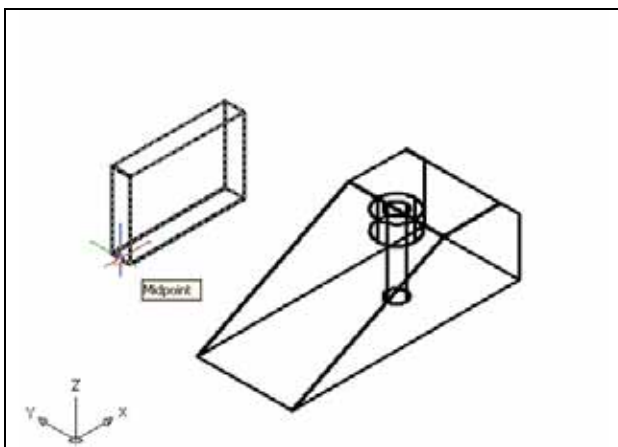


Figure 11.18 – Moving the Box

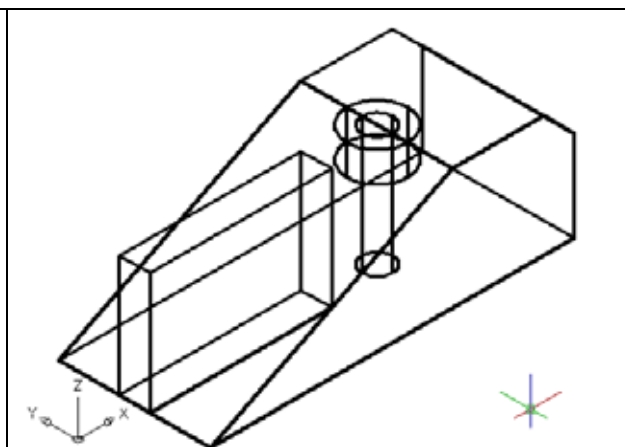


Figure 11.19 – Moving the Cylinder

To move the box, you select the Move tool on the Modify panel. Once you choose the Move command, the prompt will be “Select Objects”. Pick the solid box and the command line will respond with “1 found”, so just press **Enter** to go the second part of the command. Next, you need to “specify base point or displacement”, which will be the Midpoint of the front edge of the Box as shown in Figure 11.18. For the “second point of displacement”, you need to pick the Midpoint of the front edge of the solid wedge 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.

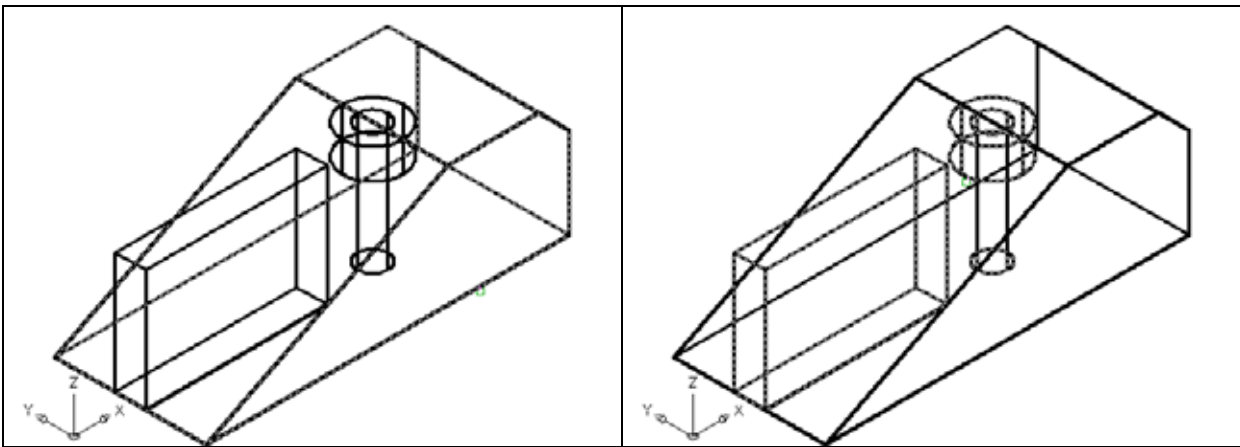


Figure 11.20 – Select Master Solid

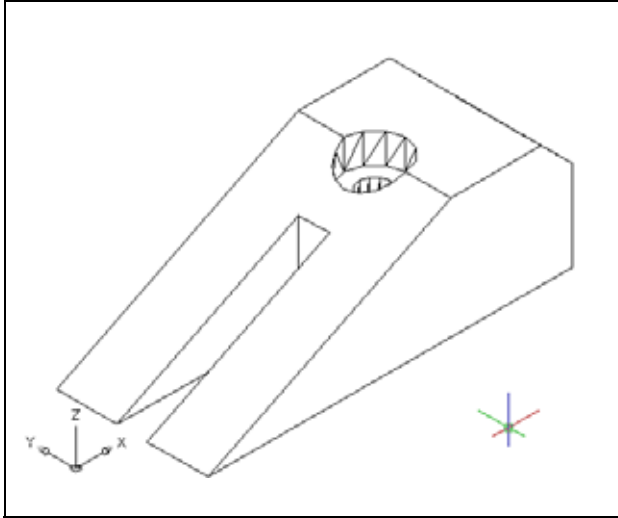
Figure 11.21 – Select Solids to Remove

AutoCAD will prompt you to “Select solids and regions to subtract from”. Select the master solid as shown in Figure 11.20 and 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 and the box as shown in Figure 11.21 and the software application will counter with “3 found”. Hit **Enter** and a region exactly the shape of the two cylinders and the box will disappear from the larger one as shown in Figure 11.24.

Hide the 3D Solid

In Figure 11.22, 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 solid unless the material was transparent. You can use the Hide command to remove the hidden lines from your graphical display.





Select the 3D Hidden tool on the View panel. This is a very simple command, because moments after picking the tool, the tessellation lines representing the back of the part are removed from view. If you wish to return the graphical display to the previous setup where all the tessellation lines are seen, then choose the tool on the View panel called 3D Wireframe. By choosing the 3D wireframe tool, the solid part will return to the previous appearance. This command does not alter the mass of the solid or any of the dimensions, but is available for the computer aided designer as a viewing option.

Figure 11.22 – Hide the Solid

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, pick the Move tool on the Modify panel, select the 3D solid, and hit Enter. At the prompt “**specify base point or displacement**”, select the endpoint shown in Figure 11.23 at the lower left hand side of the problem ten, which will become the problem ten’s insertion point into an assembly drawing. At the “second point of displacement”, type 0,0,0 and Enter. Problem ten will move to the new origin point. There is more than one way to do many actions in AutoCAD, but the result of having a part at the origin is important when you wish to insert the solid into an External Referenced Assembly drawing.

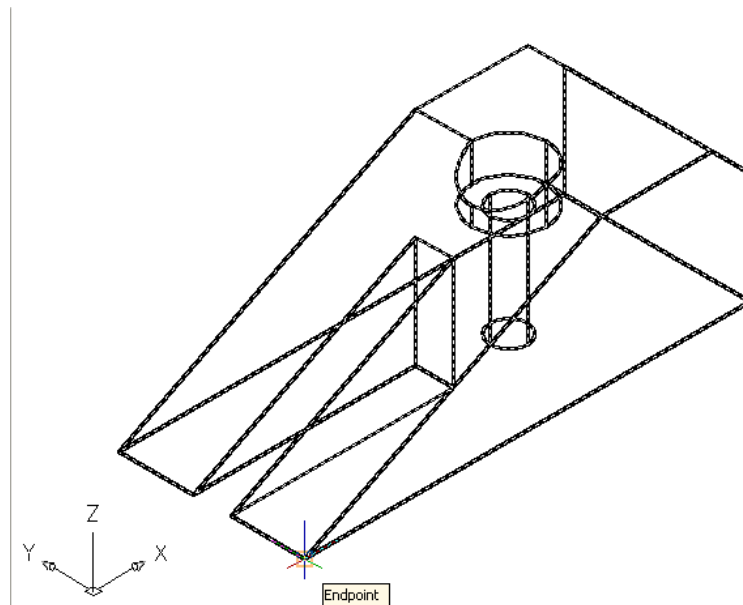


Figure 11.23 – Moving the Finished Solid to the Origin

Saving the Solid Problem

To save Problem 10 in your Fundamentals of 3D Drawing Folder, select the Save 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. At the File Name textbox, type “Problem 10” and press the Save button to save the drawing. (See Figure 11.24)

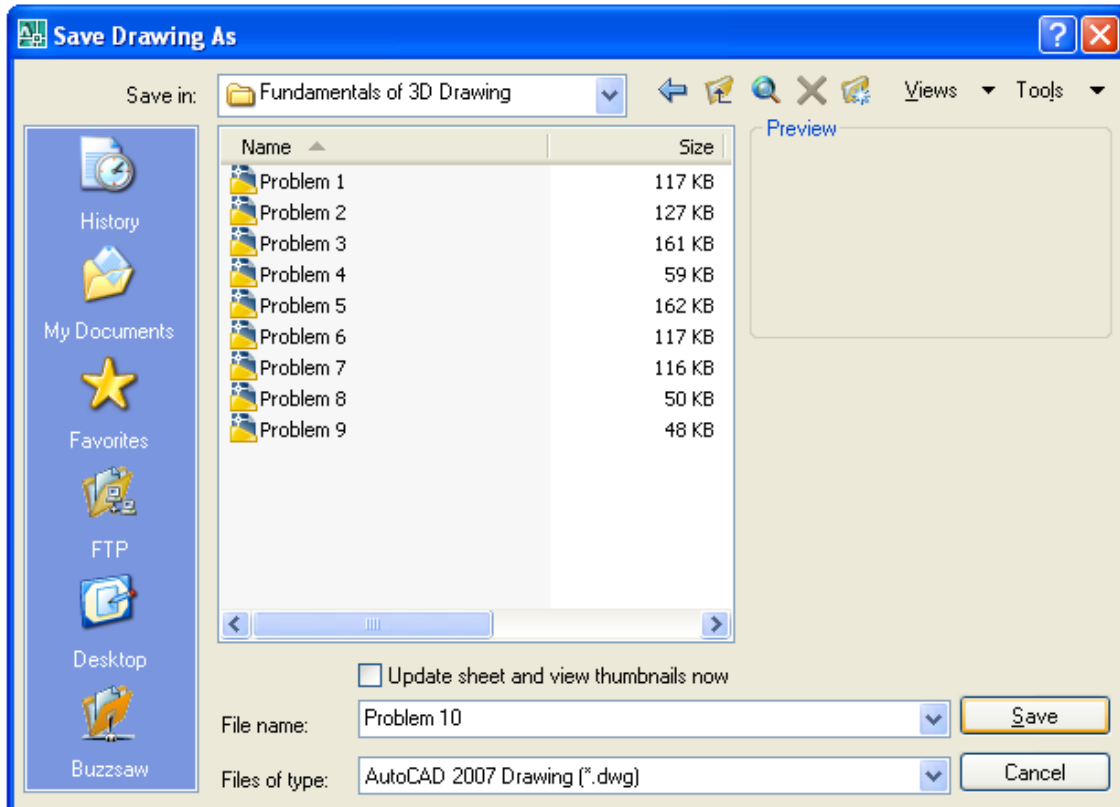


Figure 11.24 – Saving Problem Ten

*** 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 under 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.**

