

Solid Problem Seven

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

- Sketch of Solid Problem Seven
- Starting a 3D part drawing
- Modifying UCS settings
- Drawing a closed polyline shape to be extruded
- Adding circles to the detail
- Moving the circle inside the polyline
- Extruding solids from a closed polyline and circle
- Subtracting a 3D solid from another solid
- Rotating a 3D solid using the 3D Rotate command
- Drawing a solid box
- Moving a 3D solid
- Constructing a solid perimeter using the Line command
- Creating polylines using the Edit Polyline tool
- Extruding a solid from a closed polyline
- Aligning a solid to the master solid
- Subtracting 3D solids from the master solid
- Hiding the solid
- Moving solid to the origin point
- Saving the Solid Problem

Sketch of Solid Problem Seven

You start the seventh solids problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across the X axis. In this exercise, you will draw closed polylines that contain the most complex shape of the component and then use simple geometric solids to subtract detail from the extruded master solid. In problem seven, the master solid contains a chamfer drawn into the polyline shape. You will place a circle near the edge of the main shape. After extruding the polyline and circular solids, you will subtract the cylinder. You will finish by subtracting a box and a wedge shape from the master solid to end with the product as shown in Figure 8.1.

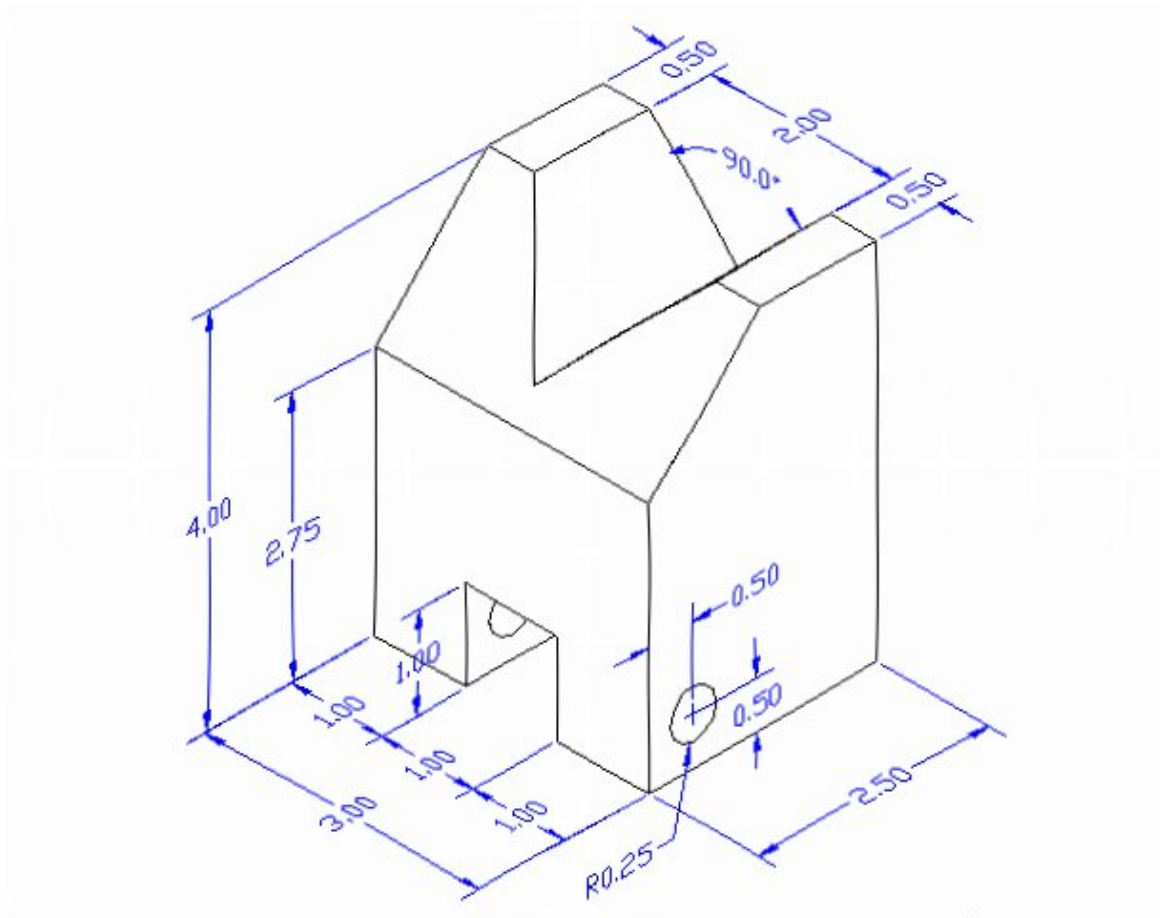


Figure 8.1 – Problem Seven Sketch

Remember, when you receive a sketch from a professional, quickly identify the different shapes and decide how to begin the drawing. Problem Seven has an angled cut on the top, so you will create a special prism that you will move and rotate using the Align command, and then subtract from the master solid. All of the drills in the Fundamentals of 3D Drawing textbook will walk you through such common repetitive maneuvers that you will use often in 3D modeling.

Starting a 3D Part Drawing

You will draw Problem Seven 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.

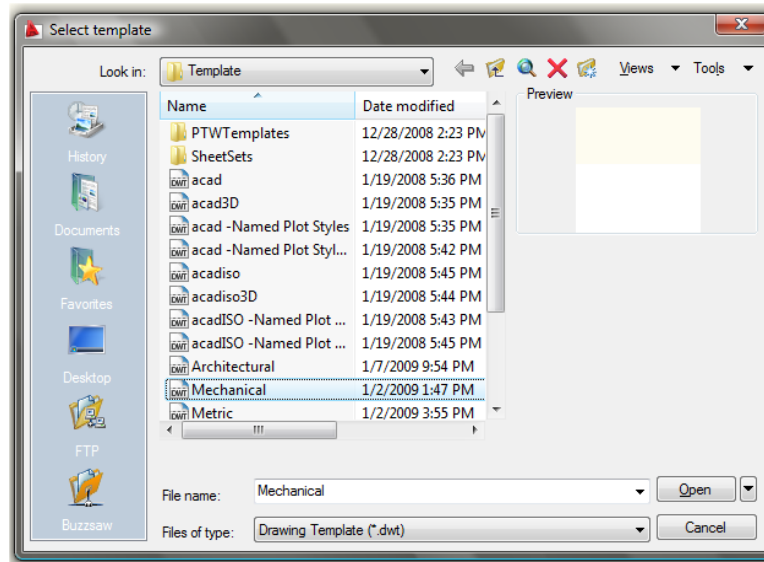


Figure 8.2 – Starting the drawing using the Mechanical template

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 8.3) and select the “Show UCS Icon” option (Figure 8.4). You also have the choice of keeping the UCS displayed at the origin or turning it off completely.

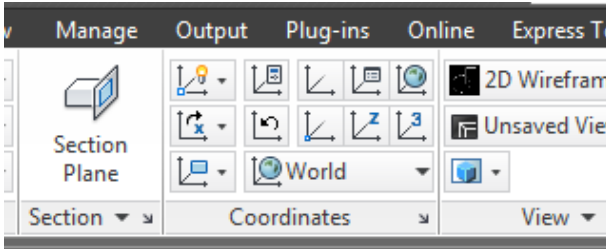


Figure 8.3 - The Coordinates panel

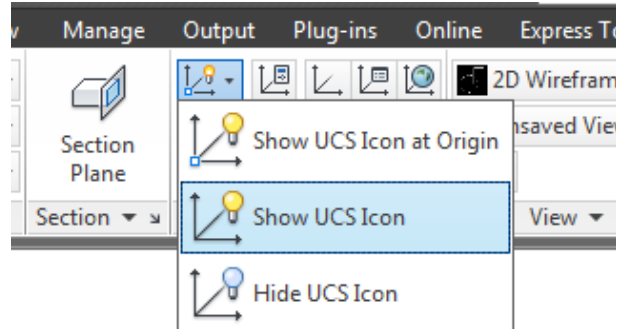


Figure 8.4 - Changing the UCS display settings

Drawing a Closed Polyline Shape to be Extruded

Select the Polyline tool from the Draw panel and specify a start point in the upper left-hand corner of the graphical display. You can see the actual continuous line of data you need to specify at the command line in Figure 8.5.



```
Command: _pline
Specify start point:
Current line-width is 0.00
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: <Ortho on> 2.75
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 2.5
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 4.0
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 1.25
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: C
```

Figure 8.5 – The command line text from the Polyline command

With the Ortho mode “on”, draw a **2.75** unit line downward, a **2.5** line to the right, a **4.0** line upward, a **1.25** line to the left, and then type “**C**” and **ENTER** to close the polyline. A closed shape will appear in the graphical display. Zoom and pan the drawing as necessary to maximize the shape on your display. In Figure 8.6 you can see the step-by-step progression of building the polyline with Figure 8.7 showing the final two-dimensional entity.

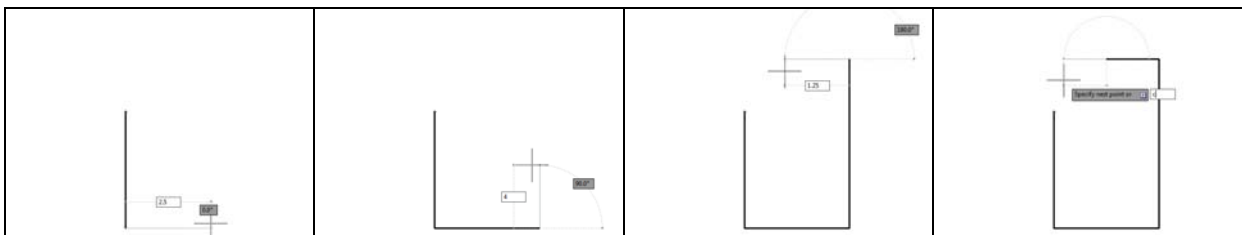
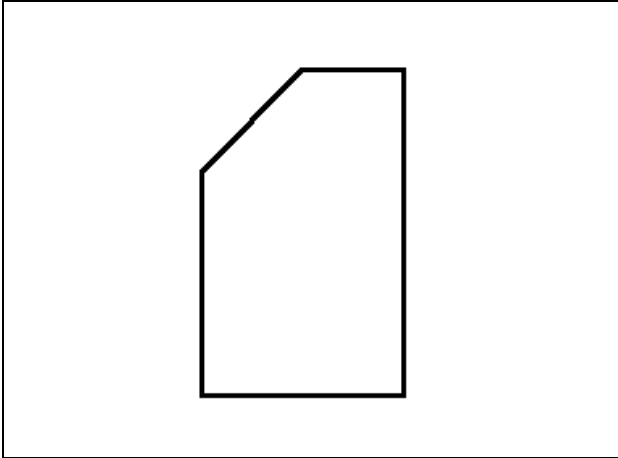


Figure 8.6 – Drawing a series of lines using the Polyline command



Notice how simple it was to add the angled side of the polyline. By knowing the measurements of all of the other sides of the shape and drawing them first, you were able to let the program finish the part and calculate the angles and distance of this edge when you used the Close option of the Polyline command. You will save time whenever you can forgo complex and time-consuming mathematical computations by letting the computer do the work.

Figure 8.7 – The finished polyline

Adding Circles to the Detail

Select the Circle command on the Draw panel. Select the endpoint Osnap of the lower left-hand corner of the polyline as shown in Figure 8.8. Type “D” and **ENTER** when the command line prompts you to “specify radius of circle”. Type **0.5** for the diameter and the circle will appear on the corner of the polyline as shown in in Figure 8.8.

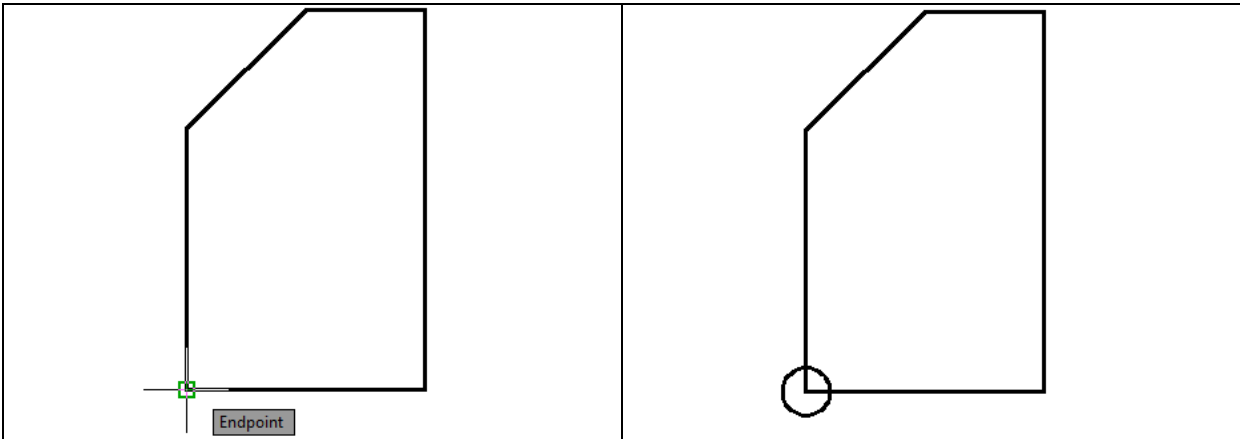


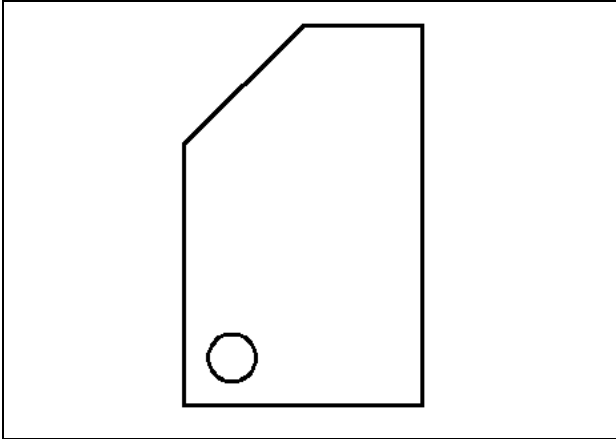
Figure 8.8 – Selecting the center point

Figure 8.9 – The 0.5 diameter circle

Moving a Circle Inside the Polyline

Before extruding both the Polyline and the Circle, you should position the circle at a more precise location. Use the Move tool on the Modify panel to accomplish the task.





Select the Move tool on the Modify panel. Pick the circle when prompted to “Select Objects” and the command line will respond with “**1 found**”, so hit ENTER to go the second part of the command. Next, you need to “specify base point or displacement” on the graphical display. Pick a base point anywhere on the display. Then type **0.5, 0.5** and hit ENTER. The circle will move 0.5 units to the right and 0.5 units up on the display as shown in Figure 8.10.

Figure 8.10 – The moved circle

Extruding Solid from a Closed Polyline and Circle

Only closed entities like Polylines and Circles can be extruded using this very powerful tool on the Modeling panel. You can extrude multiple entities at a time into solids with the Extrude command.

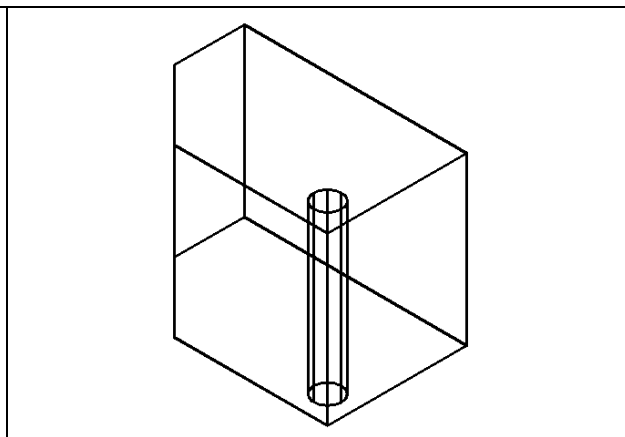
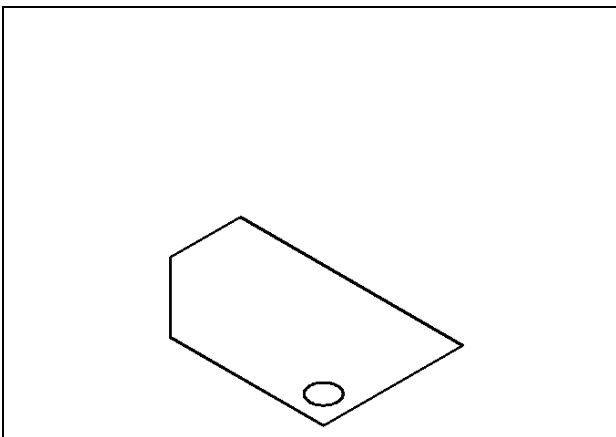
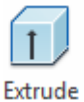


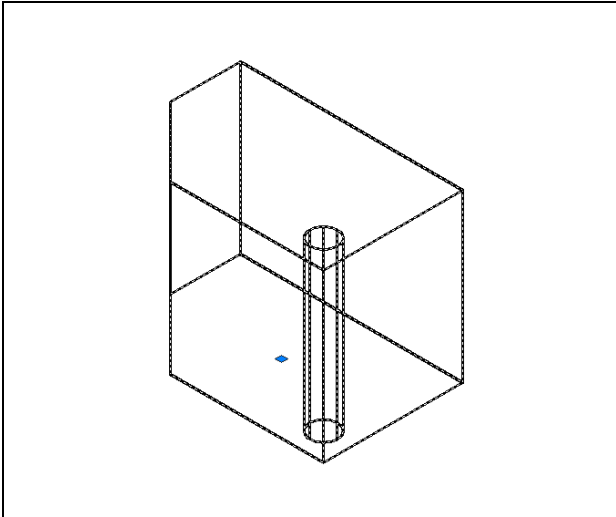
Figure 8.11 – Southwest view

Figure 8.12 – Two extruded entities

Select the Southwest Isometric View on the View panel to change the view as shown in Figure 8.11. Select the Extrude tool on the Modeling panel and the command line will prompt you to “**select objects**”. Select both closed shapes and the command line will return with “**2 found**”. Hit ENTER to specify the height or path of the solid. Type “**3**” and ENTER for the height of the extrusion. Two complex 3D solids will appear in the graphical display as shown in Figure 8.12.

Subtracting a 3D Solid from Another Solid

To remove the cylinder from the master solid, select the Subtract tool on the Solid Editing panel.



AutoCAD will prompt you to “Select solids and regions to subtract from”. Select the master solid and the system will respond with “1 found”. Hit **ENTER** and you will be prompted to “Select solids and regions to subtract”. Pick the cylinder and the command line will reply with “1 found”. Hit **ENTER** and the cylinder will disappear from the drawing leaving empty space in its place inside the master solid as shown in Figure 8.13. You can check the subtraction by selecting the master solid with the cursor; the entire solid will be dashed as shown in Figure 8.13.

Figure 8.13 – Subtracting the cylinder

Rotating a 3D Solid Using the 3D Rotate Command

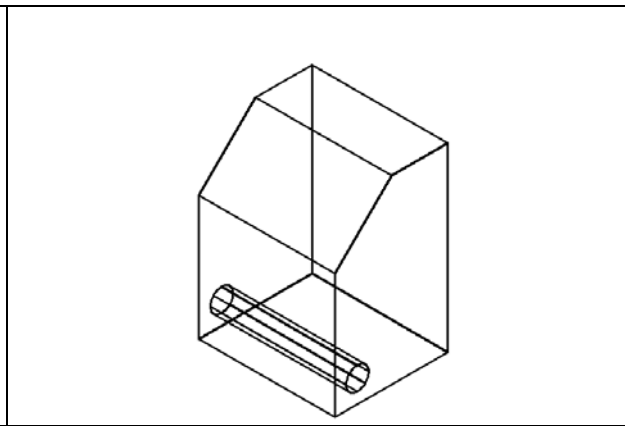
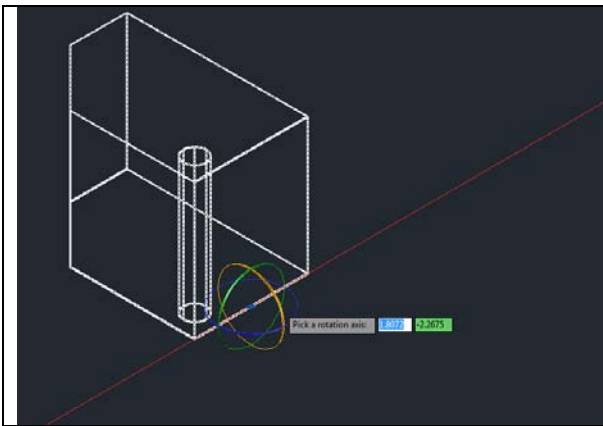


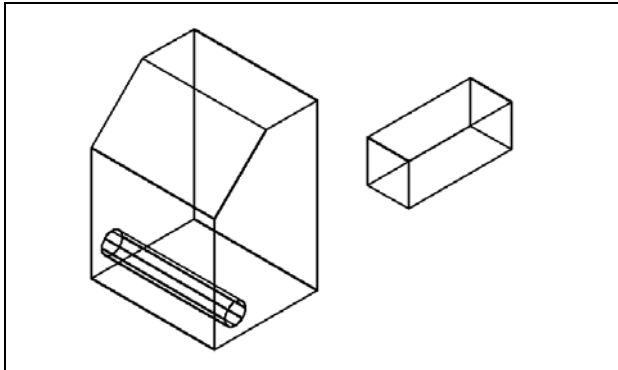
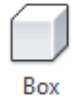
Figure 8.14 – Selecting the X axis

Figure 8.15 – The rotated solid

Select the 3D Rotate command on the Modify panel. When prompted to “select objects”, select the solid and hit **ENTER** and then select the midpoint Osnap of the bottom edge of the front face of the solid as shown in Figure 8.14. You are going to rotate the solid part on the X axis, so select the circle that orbits the X axis as shown in Figure 8.14. Specify an angle start point, and then with the Ortho mode move the cursor so that the preview outline moves 90 degrees as shown in Figure 8.15.

Drawing a Solid Box

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** after specifying the corner on the graphical display to utilize the Length option of the Box command.



Enter **2.5** for the length of the box, **1.0** for the width, and **1.0** also for the height. The solid box will appear in the display to the right of the master solid as shown in Figure 8.16

Figure 8.16 – A new box

Moving a 3D Solid

The Move command is a three-dimensional function in AutoCAD 2012.

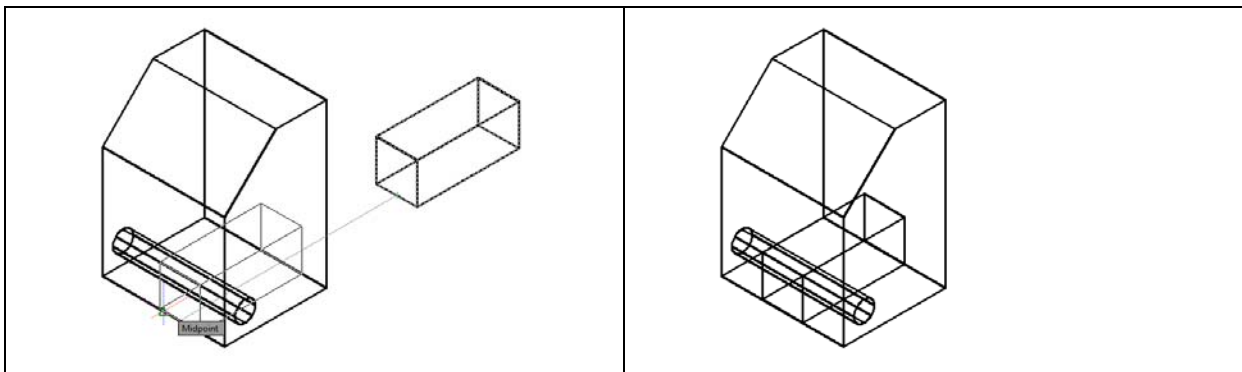


Figure 8.17 – The destination point

Figure 8.18 – The moved box

To move the box, select the Move tool on the Modify panel. Pick the new box when prompted to select objects and the command line will respond with “1 found”, so hit **ENTER**. Select the midpoint Osnap of the bottom edge of the front face of the new box as the base point and select the midpoint Osnap of the bottom edge of the front face of the master solid as the destination point as shown in Figure 8.17. The solid box will move as shown in Figure 8.18.

Constructing a Solid Perimeter Using the Line Command

Now you need to create a right triangle with a 2-inch hypotenuse that you will extrude into a solid and eventually subtract from the top of the master solid to create the angled detail shown in the sketch. First change to the Top view using the 3D Navigation tool on the View panel under the Home tab. Then select the Line command from the Draw panel.



Start a line to the left of the master solid and with the Ortho mode “On”, draw a line **2.0** units to the left. Now you will draw a line using a new technique called polar displacement. Type **2.0<-45** and a line will appear as shown in Figure 8.19.

Remember that if you are not using Dynamic Input (toggled on and off on the tray at the bottom of your screen) you will need to add the @ symbol before typing 2.0<-45. The formula for this particular entry has 2.0 specifying the length of the line. The < symbol signifies that you are going to enter an angular value, and the -45 is the degree measurement from 0 degrees, parallel with the X axis.

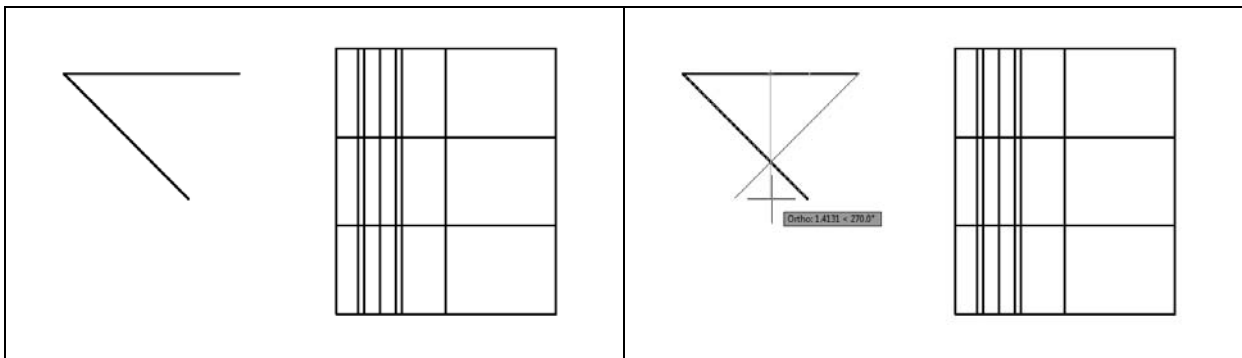


Figure 8.19 – Drawing the 45 degree line

Figure 8.20 – Mirroring the line

You will use the Mirror tool on the Modify panel to copy the 45 degree line across the midpoint of the 2.0 line. At the prompt to select objects, select the 45 degree line and hit **ENTER**. “Specify first point of mirror line:” by picking the midpoint Osnap of the horizontal 2.0 line and “Specify second point of mirror line:” by picking anywhere above or below the first point with the Ortho mode “On” as shown in Figure 8.20. The default to delete the source objects is no, so at the prompt “Delete source objects? [Yes/No] <N>:” hit **ENTER**. The 45 degree line will be copied and mirrored across the middle of the horizontal line as shown in Figure 8.21. You can either use the Fillet or Trim tool to trim the lines as shown in Figure 8.22.

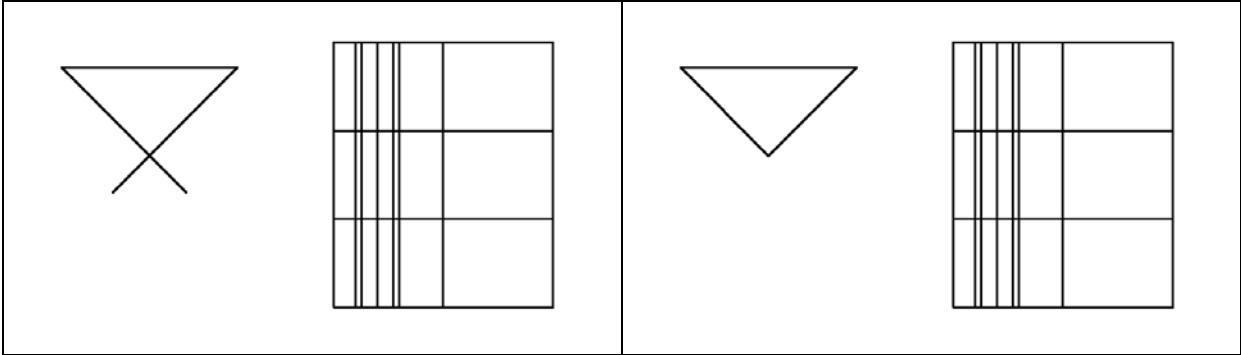


Figure 8.21 – The mirrored line

Figure 8.22 – The triangle after trimming

Creating Polylines Using the Edit Polyline Tool

Use the Edit Polyline tool to merge the three line segments of the triangle together into a closed polyline. Select the Edit Polyline tool from the drop-down arrow on the Modify panel and the command line will prompt you to “**Select polyline or [Multiple]:**”

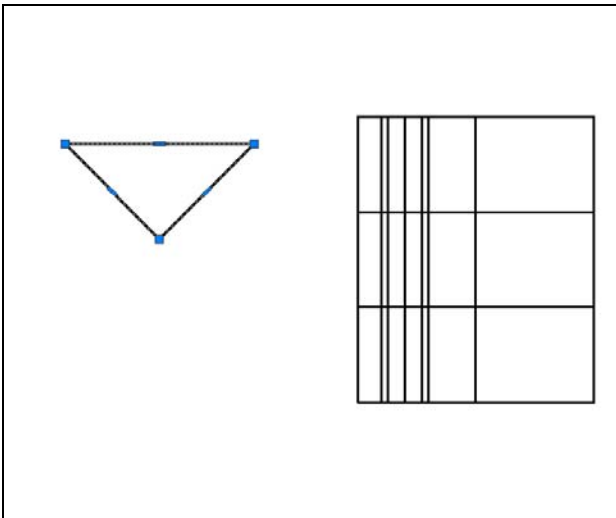
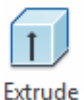


Figure 8.23 – The closed polyline, selected

Select one of the lines of the triangle and the command line will reply with “**Object selected is not a polyline Do you want to turn it into one? <Y>**”. Type **Y** and **ENTER**, and then the command line will prompt you with a number of editing options from which to choose. Type “**J**” and **ENTER** to choose the Join option. The command line will prompt you to “Select objects” so select all of the lines in the triangle. The command line should read “**3 total**”. Hit **ENTER** to join all of the entities into a closed shape, and then hit **ESCAPE** to exit the Edit Polyline tool.

Extruding a Solid from Closed Polyline

Only closed entities like Polylines and Circles can be extruded using this very powerful tool on the Modeling panel. You can extrude multiple entities at a time into solids with the Extrude command.



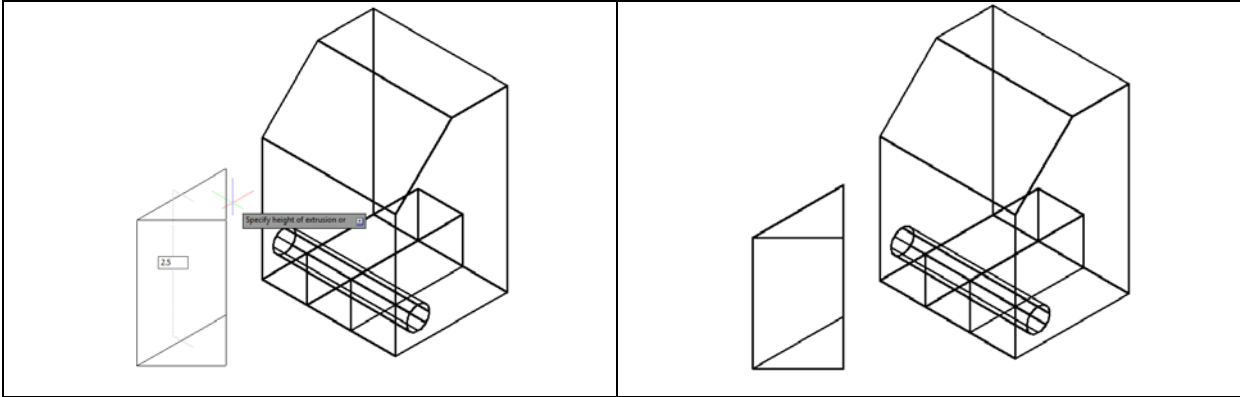


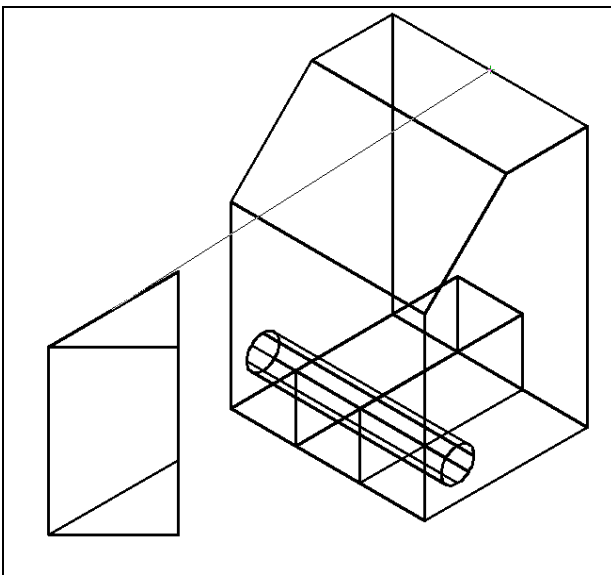
Figure 8.24– Extruding the triangle

Figure 8.25 – The new solid prism

Select the Southwest Isometric View on the View panel to change the view as shown in Figure 8.24. Select the Extrude tool on the Modeling panel and the command line will prompt you to “**select objects**”. Select the triangle 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 (Figure 8.24). The triangle will expand into a 3D prism as shown in Figure 8.25.

Aligning a Solid to the Master Solid

Select the Align command by expanding the Modify panel. At the command, “**select objects**”, pick the prism hit **ENTER** to proceed to the second part of the function. Notice that this is not the 3D Align command.



The first source and destination points will end up becoming the same point in the finished alignment. In this case, select the midpoint Osnap of the longest side of the top face of the prism as the first source point and then the midpoint Osnap of the back edge of the top face of the master solid as the destination point. These two points will join to become the same point when the alignment is finished. A temporary line will appear on the screen showing the relationship of the two points (Figure 8.26).

Figure 8.26 – First source and destination points

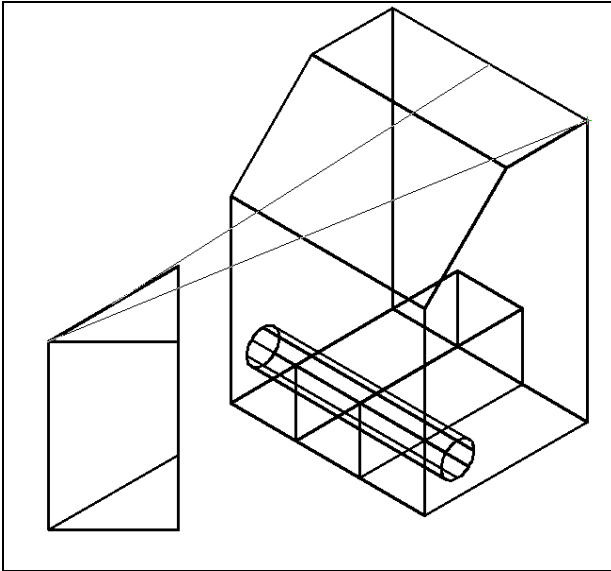


Figure 8.27 – Second source and destination points

The second and third set of points you select will not end up becoming the same point, but they will reside on the same respective axes. For the second source point, select the left corner of the top face of the prism. Select the top back right corner of the master solid as the destination point. Again a temporary line will appear connecting both points, showing their relationship in the alignment (Figure 8.27). When finished with the command these two points will reside on the same parallel line.

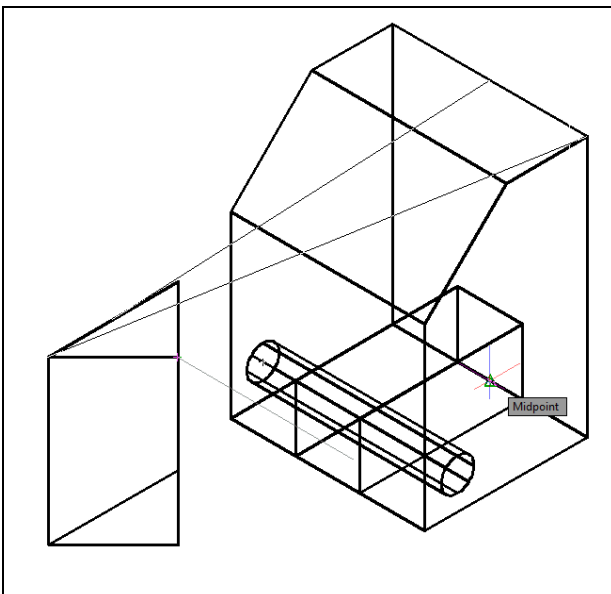


Figure 8.28 – Third source and destination points

The third set of points will finalize the definition of the planes of the two shapes that you are aligning. Select the 90 degree angle corner of the top face of the prism as the third source point. Select the midpoint Osnap of the back bottom edge of the master solid as the third destination point (Figure 8.28). Immediately the prism will snap into alignment with the master solid (Figure 8.29). Again, the difficulty of the Align command lies not in its complexity as a tool but in properly identifying the source and destinations points necessary to create the desired alignment. This complex technique will require some practice but it can be mastered.

Subtracting 3D Solids from the Master Solid

To remove the two solids from the master solid, select the Subtract tool on the Solid Editing panel.



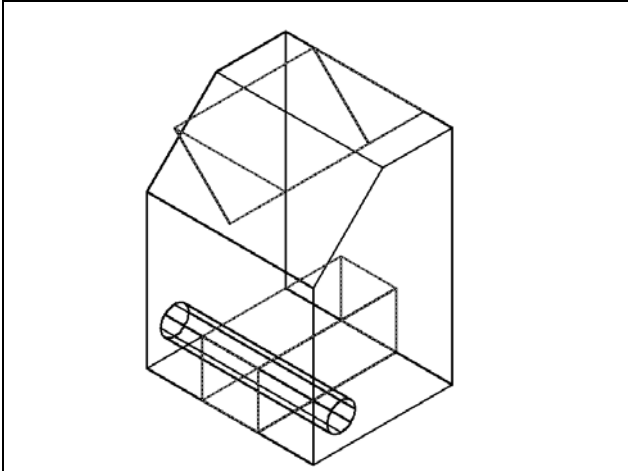


Figure 8.29 – Solids selected to be subtracted

The command line will prompt you to “**Select solids and regions to subtract from**”. Select the larger master solid 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 box and the prism and the command line will respond with “**2 found**” (Figure 8.29). Hit ENTER and the shape of the two solids will disappear from the master solid as shown in Figure 8.30.

Hide the 3D Solid

To view a more realistic image of Solid Problem Seven, select the Hide command from the Visual Styles panel under the View tab (Figure 8.30).

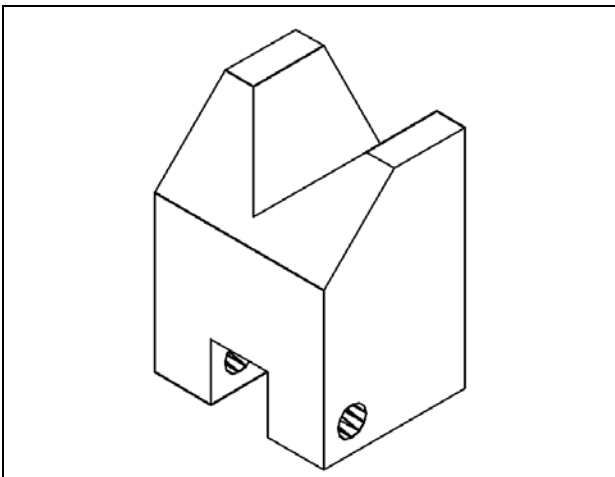


Figure 8.30 – The Hidden visual style

If you wish to return the graphical display to the previous, then choose the 2D Wireframe visual style on the View panel under the Home tab. 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.

Moving the Solid to the Origin Point

To move Problem Seven 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 8.31, 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 Seven 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.

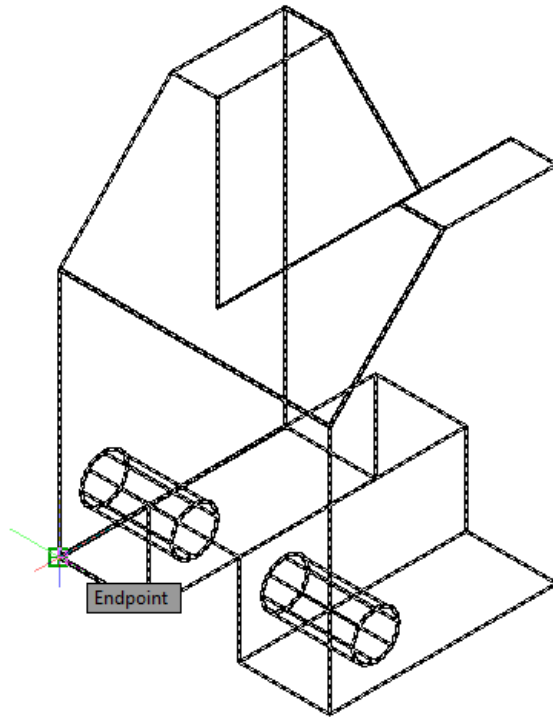


Figure 8.31 – Moving the finished solid to the origin

Saving the Solid Problem

To save Problem 7 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 7” in the File Name textbox and hit the Save button to save the drawing (Figure 8.32).



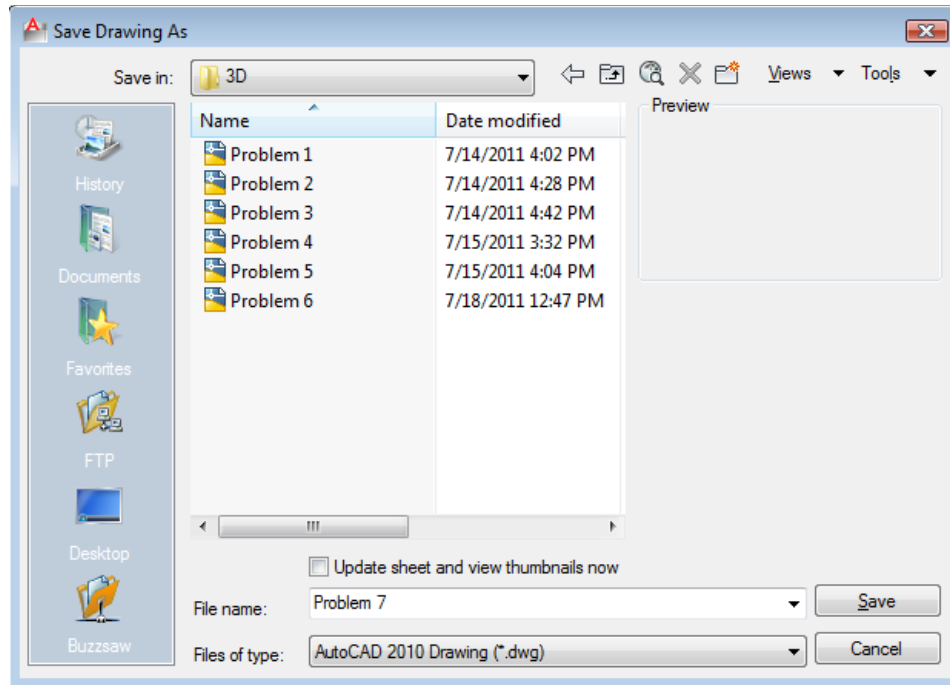


Figure 8.32 – Saving Problem Seven

*** World Class CAD Challenge 03-07 * - 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.**

