

Solid Problem Five

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

- Sketch of Solid Problem Five
- Starting a 3D part drawing
- Creating a new layer in an AutoCAD drawing
- Modifying UCS settings
- Drawing a closed polyline shape to be extruded
- Changing to the Line layer
- Constructing a solid perimeter using the Line command
- Adding circles
- Trimming the circles and removing extra lines
- Freezing an AutoCAD layer
- Creating polylines using the Edit Polyline tool
- Extruding a solid from closed polylines and circles
- Moving solids adjacent to the master solid
- Union of the solids
- Subtracting a 3D solid from another solid
- Shading the solids
- Rotating a 3D solid using 3D Rotate
- Moving the solid to the origin point
- Saving the solid problem

Starting a 3D Part Drawing

You will draw Problem Five using the Metric 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 Metric.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.

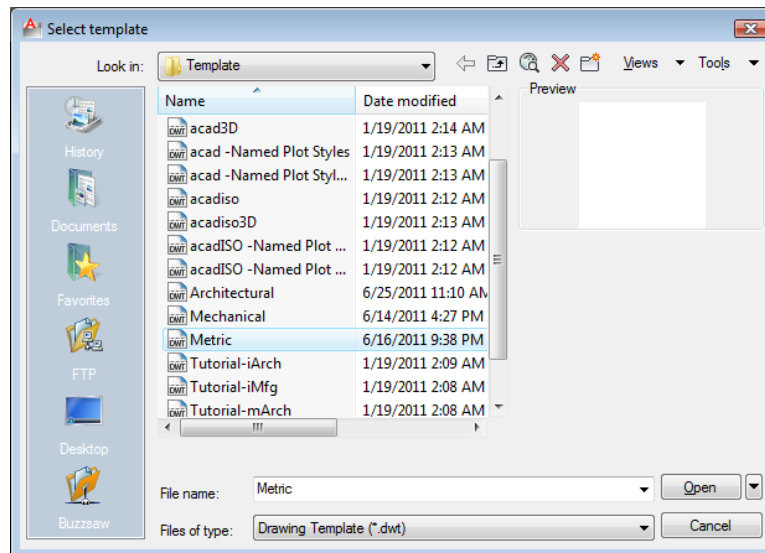


Figure 6.2 – Starting the Drawing Using the Metric Template

Creating a New Layer in an AutoCAD Drawing

Many times you may wish to add a special layer so that you can control the formation of the solid part by freezing or thawing the desired entity which is on a layer. In this exercise you need to create a new layer which you will call “Line” so you can have that extra measure of control.

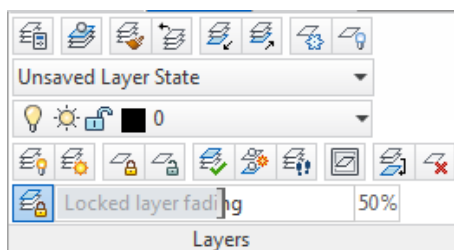


Figure 6.3 - The Layers panel

To create the “Line” layer using the Layers Properties Manager, select the Layers Properties Manager command button under the drop-down arrow on the Layers panel under the Home tab (Figure 6.3). Add the “Line” layers shown in Figure 6.4 using the New button in the Layer Properties Manager.

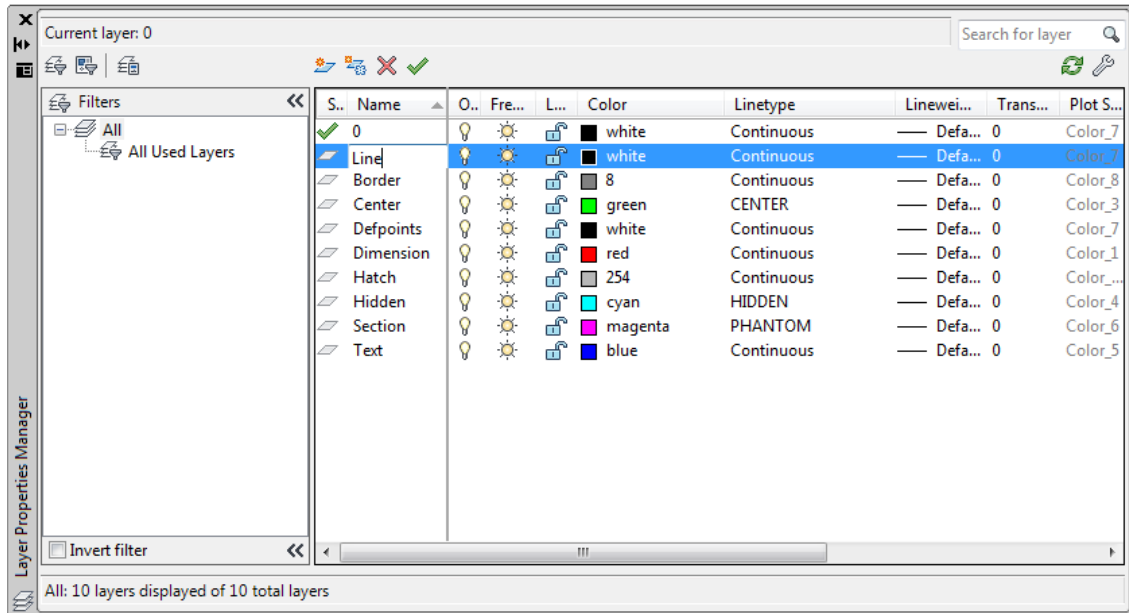


Figure 6.4 – Making a new “Line” layer

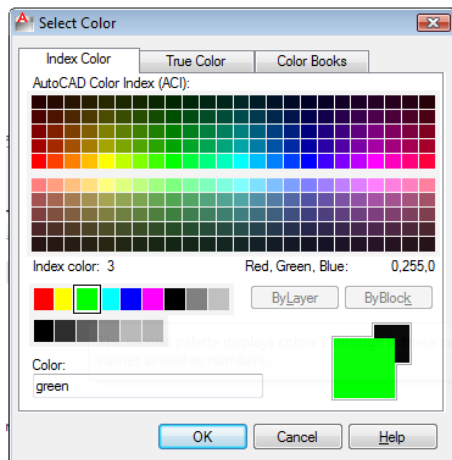


Figure 6.5 – Setting the “Line” layer’s color

Now right-click the color in the Line layer row. The Select Color window will appear in front of the Layers Properties Manager as shown in Figure 6.5. Select green from the Standard Colors Row and hit the OK button. Once you have set up the new layer and its color you can exit the Layer Properties Manager. You will use the Line layer to control different groups of entities as necessary.

Modifying How the UCS Icon is Displayed

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 6.6) and select the “Show UCS Icon” option (Figure 6.7). You also have the choice of keeping the UCS displayed at the origin or turning it off completely.

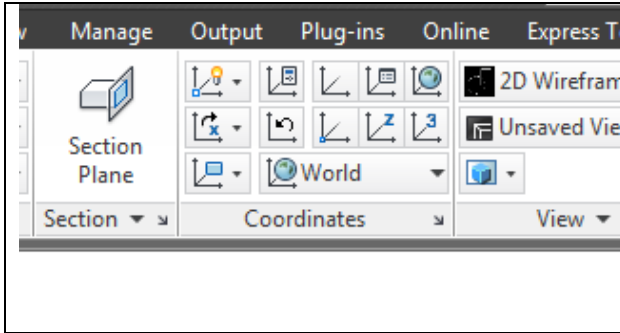


Figure 6.6 - The Coordinates panel

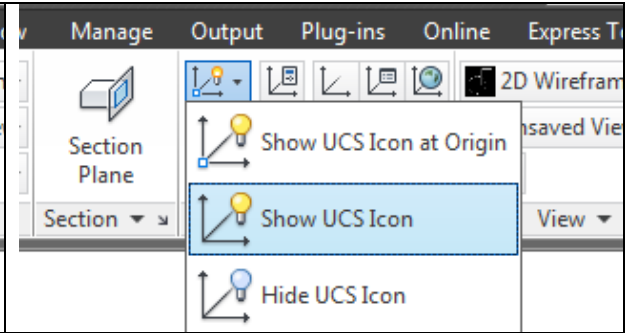


Figure 6.7 - Changing the UCS display settings

Drawing a Closed Polyline Shape to be Extruded

Select the Polyline tool from the Draw panel under the Home tab and specify a starting point in the lower left hand corner of the graphical display. You can see the actual continuous line of command line data for this action in Figure 6.8.



With the Ortho mode “on”, draw a **20** unit line to the right, a **20** line up, a **20** line to the left, and then type “**C**” to close the Polyline. A closed entity will appear in the graphical display as shown in Figure 6.9.

```
Command: _pline
Specify start point:
Current line-width is 0.00
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: <Ortho on> 20
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 20
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 20
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: c
```

Figure 6.8 – Command line prompts from the first polyline

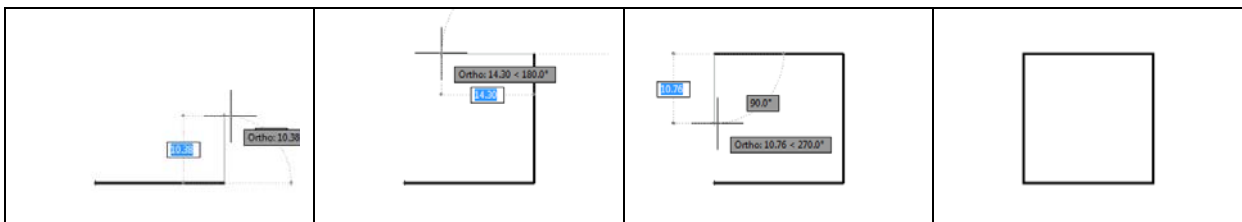
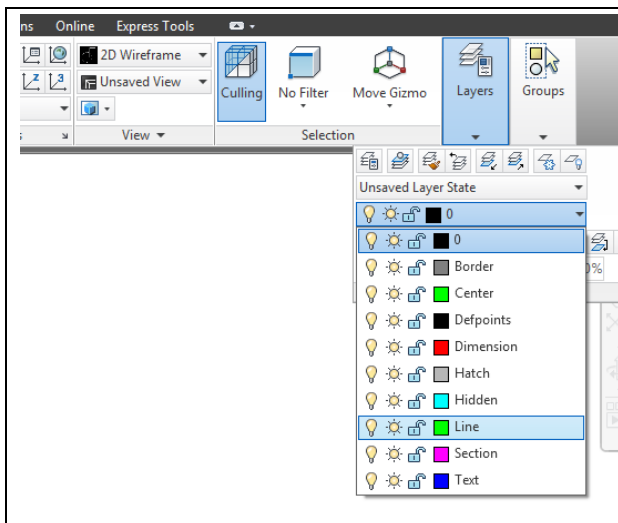


Figure 6.9 – The first closed polyline

Changing to the Line Layer



Select the Layer panel and then select the Line layer from the Layer Control list box as shown in Figure 6.10. The layer displayed on the Layer Panel is the active layer. To the left of the layer name is a light bulb representing the layer on – off function, the yellow sun representing the freeze – thaw function, and the padlock representing the lock – unlock function. These features will become increasingly useful as you gain more experience in viewing techniques by using more layers, deciding which layer will be visible or invisible, or locking the layer to prevent anyone from changing the drawing.

Figure 6.10 – Choosing the Line layer

Constructing a Solid Perimeter Using the Line Command

You need to assemble a 15 by 20 rectangle to the right of the first polyline. Select the Line tool from the Draw panel. Next select the endpoint Osnap at the lower right side of the Polyline box as shown in Figure 6.11. Keep the Ortho mode On to keep lines perfectly horizontal or vertical.



Pull your cursor to the right, type “15”, and hit ENTER. To draw the vertical line move the cursor upward type “20”, and hit ENTER. To draw the third line move the mouse back to the left, and type “15”, and hit ENTER. To close the rectangle, just type “C” and ENTER (Figure 6.11).

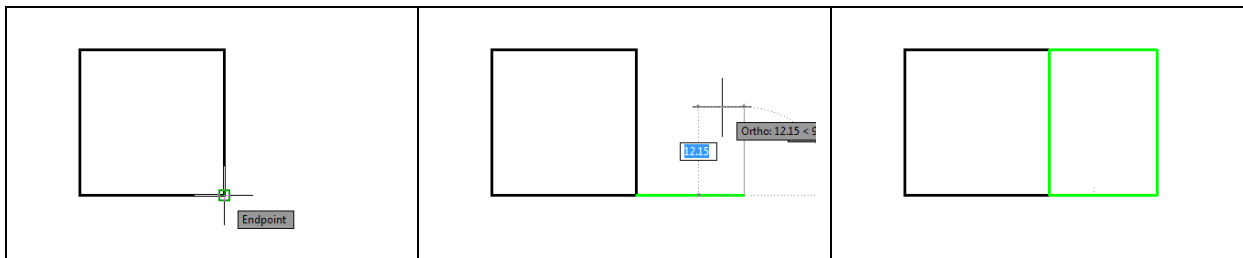
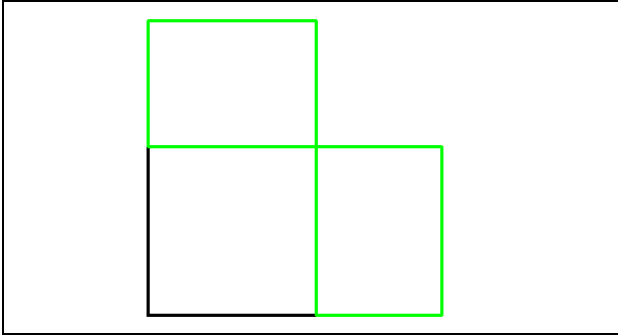


Figure 6.11 – Drawing an outline to the right of the first polyline



Hit ENTER to repeat the line command. Select the top left corner of the polyline as the first point and then move your cursor to the right and type “20” and ENTER. Next move the cursor up and type “15” and ENTER. To draw the third line move the cursor back to the left and type “20” and ENTER. To close the rectangle, just type “C” and ENTER (Figure 6.12).

Figure 6.12 –Another rectangular outline

Adding Circles to the Detail

Next, select the Center, Radius option of the Circle command on the Draw panel. Select the midpoint Osnap of the right side of the first rectangle as the center point of the circle. Type “10” and ENTER or use the endpoint Osnap of the midpoint line to specify the radius of the circle. Repeat the Circle command by hitting ENTER and place a 6 radius circle inside the 10-diameter circle using the same midpoint Osnap as the center point. You will have created two concentric circles as shown in Figure 6.13.

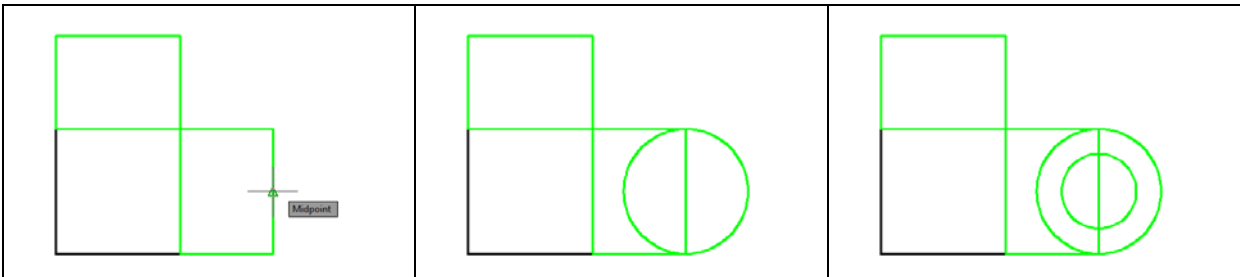
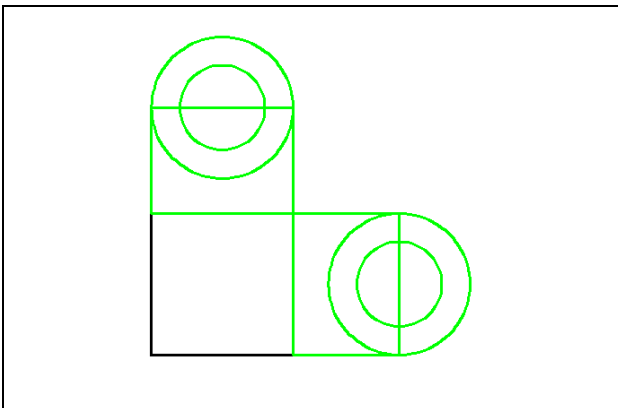


Figure 6.13 –Adding circles to the solid



Hit ENTER to repeat the Circle command. Select the midpoint osnap of the top side of the second rectangle as the center point of the circle. Type “10” and ENTER or use the endpoint Osnap of the midpoint line to specify the radius of the circle. Repeat the Circle command by hitting ENTER and place a 6 radius circle inside the 10-diameter circle using the same midpoint Osnap as the center point. Two concentric circles are shown in Figure 6.14.

Figure 6.14 – Adding additional circles

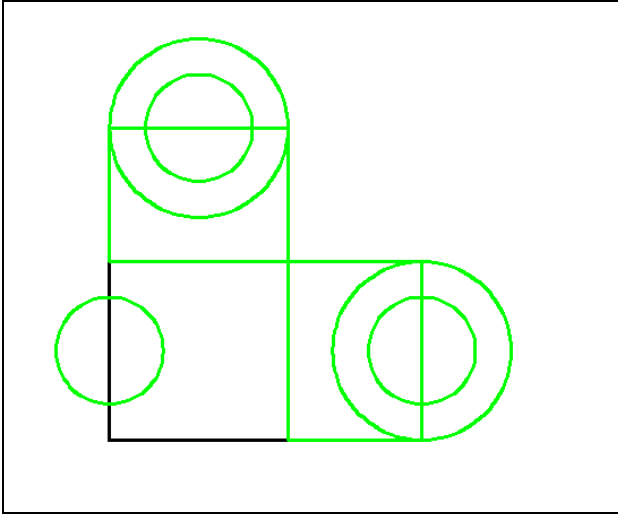


Figure 6.15 – Adding the last circle

Hit ENTER again to repeat the circle command. Select the midpoint Osnap of the left side of the polyline. Type “6” when the command line prompts you to “specify radius of circle”. The circle will appear on the midpoint of the line as shown in Figure 6.15. To get the circle into the correct position, you need to move the entity to the right half the distance of the square Polygon, or a distance of 10.

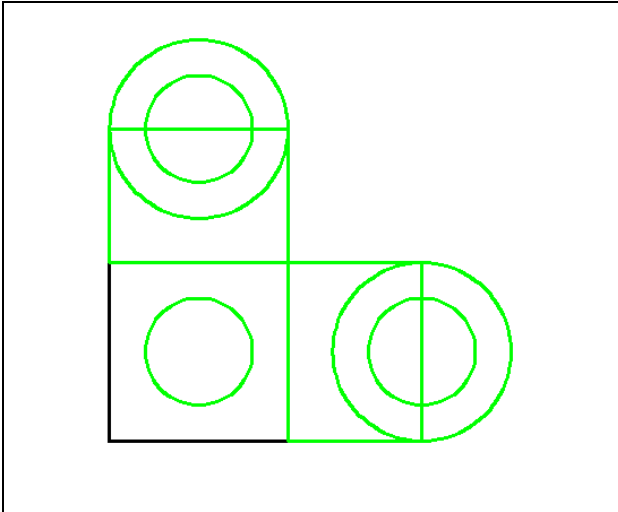


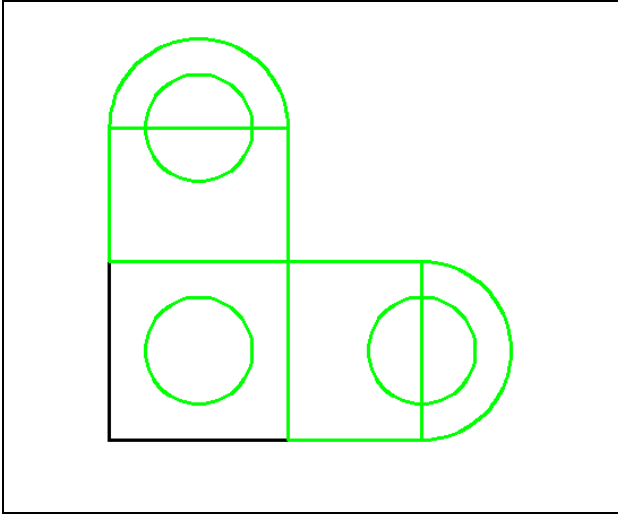
Figure 6.16 – Moving the last circle

Select the move circle from the Modify panel. Pick the last circle you created when prompted to “Select Objects” and the command line will respond with “1 found”; hit ENTER to go the second part of the command. Next you need to “specify base point or displacement” on the graphical display. Pick your base point anywhere on the display, pull the cursor to the right, type “10”, and hit ENTER. The circle will move to the center of the polyline as shown in Figure 6.16.

Trim the Circles and Erase Extra Lines

Now you will remove the inside arc segments of the larger circles to create a box with a semicircle attached, and then erase the lines passing through the smaller circles. These actions will allow you to join the remaining objects into closed polylines in order to extrude them.





Select the Trim command on the Modify panel and hit ENTER to automatically select all lines as cutting lines. Select the sides of each 10-unit circle that lies inside of the rectangles as shown in Figure 6.17. Next, select the lines that run through each 6-unit circle with your cursor so that their grips appear, and then hit DELETE. This will remove the two lines as shown in Figure 6.18.

Figure 6.17 –Trimming two circles

Select the small green circle inside the 20 by 20 polyline box. Select the 0 layer from the Layer Control list box in the Layer panel. You will notice the selected entities will turn black, the color that was assigned to the 0 layer (Figure 6.19). Press the “ESC” key to remove the circle from the selection set.

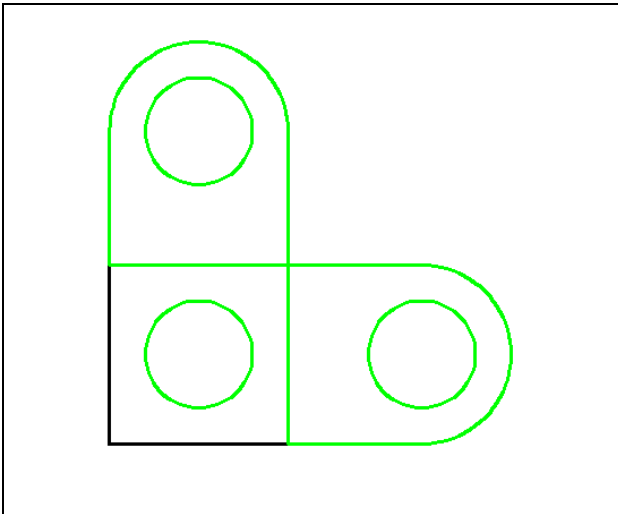


Figure 6.18 – Erasing two lines

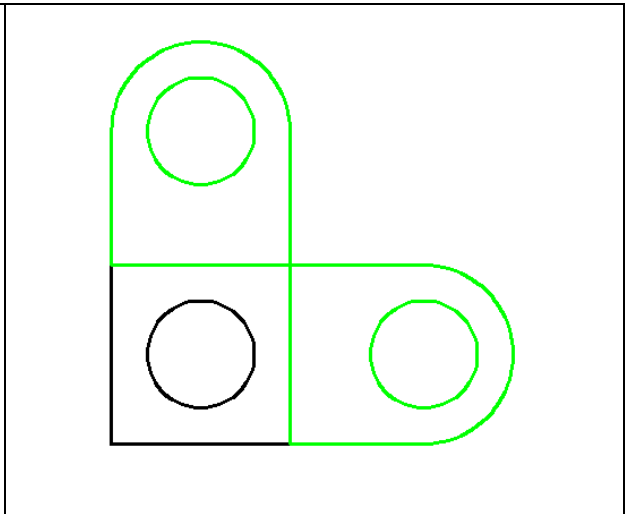
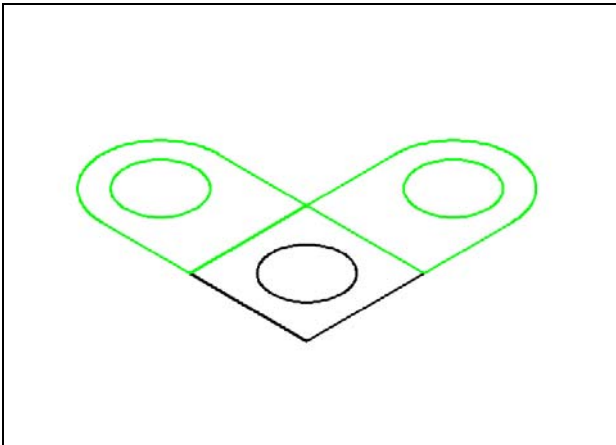


Figure 6.19 – Changing the circle’s layer

Freezing an AutoCAD Layer



Select the Southwest Isometric View from the View panel which allows you to see the part on the XY plane with the vertical Z-axis rising upward (Figure 6.20).

To make the Polyline box and circle on Layer 0 invisible, you need to use the Freeze Layer option. Select the sun icon next to the 0 layer listed in the Layer Control list box on the Layer panel as shown in Figure 6.21. The Polygon Box and circle will disappear as shown in Figure 6.22.

Figure 6.20 – SW isometric view

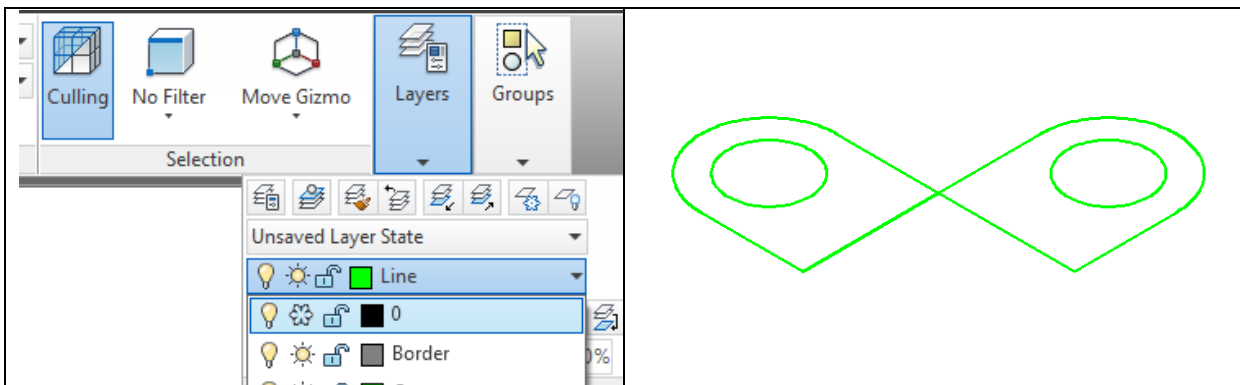


Figure 6.21 – Freezing the 0 layer

Figure 6.22 – The 0 layer frozen

Creating Polylines Using the Edit Polyline Tool

Before you can extrude your 2D shapes into 3D, you need to merge the lines and the arc segments together into two closed polylines. You will use a new tool called the Edit Polyline tool to accomplish this task.



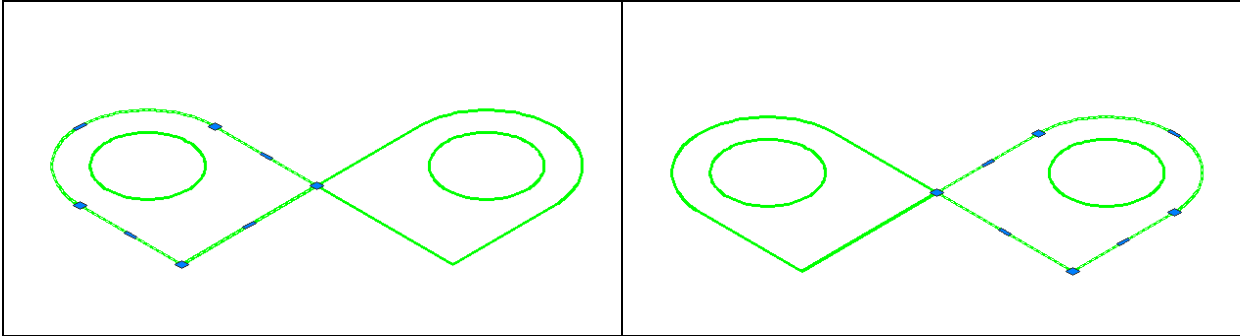


Figure 6.23 – Creating a closed polyline

Figure 6.24 – The other closed polyline

Select the Edit Polyline tool from the drop-down arrow on the Modify panel and the command line on the AutoCAD application will prompt you to “**Select polyline or [Multiple]:**” Starting with the left side, select one of the lines on the outline 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 to choose from. Type “**J**” and **ENTER** to choose the Join option. The command line will prompt you to “Select objects” so select all of the objects in the outline of the polyline you wish to create, including the object you originally selected. The command line should read “**4 total**”. Hit **ENTER** to join all of the entities into a closed shape, and then hit **ESCAPE** to exit the Edit Polyline tool. You should now be able to select the entire closed polyline with one selection of the mouse as shown in Figure 6.23. Hit **ENTER** to repeat the Edit Polyline command and use the same technique to join the entities on the right into a closed polyline as shown in Figure 6.24

Extruding a Solid from Closed Polylines and Circles

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

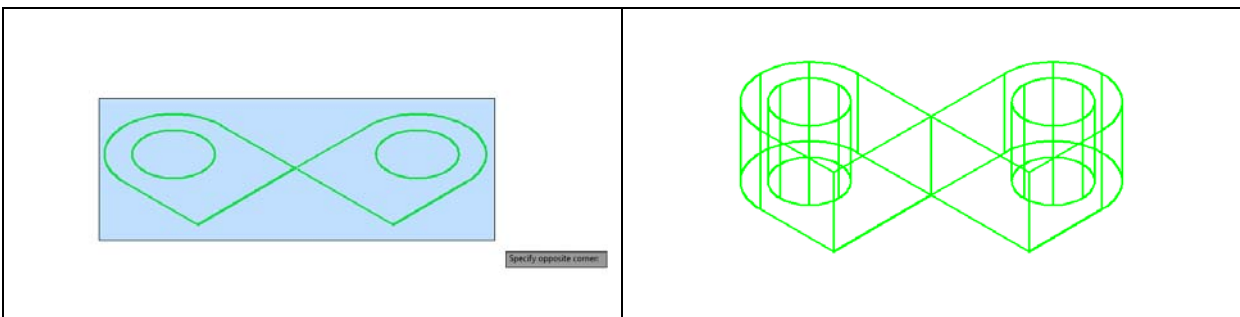
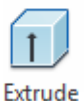
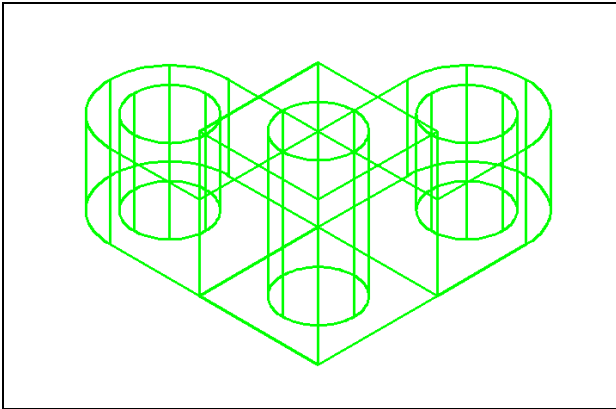


Figure 6.25 – Selecting entities to extrude

Figure 6.26 – The extruded entities

Select the Extrude tool on the Modeling panel and the command line will prompt you to “**select objects.**” Use a selection window to select both circles and both closed polylines. Hit **ENTER** to specify the height or path of the solid. Type “**14**” and **ENTER** for the height of the extrusion. The closed outline of the polyline will expand 14 units into the Z axis, creating a complex 3D solid as shown in Figure 6.26.

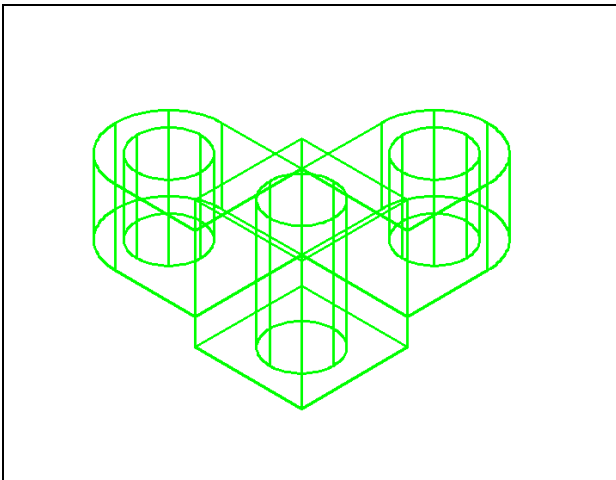


Unfreeze the 0 layer and then move the polyline square and 6-unit circle from the 0 layer to the Line layer using the Layer Control list box. The polyline square and 6-unit circle should now be green. Use the Extrude tool on the Modeling panel again to extrude these two objects 24 units into the Z axis, as shown in Figure 6.27. If the objects are already selected when you start the Extrude tool, the command line will only prompt you to enter the height of extrusion.

Figure 6.27 – Extruding the middle

Moving Solid Adjacent to the Master Solid

Remember, the Move command is actually a three dimensional function and you can use the tool to relocate any entity to another point (X,Y,Z) in Model Space.

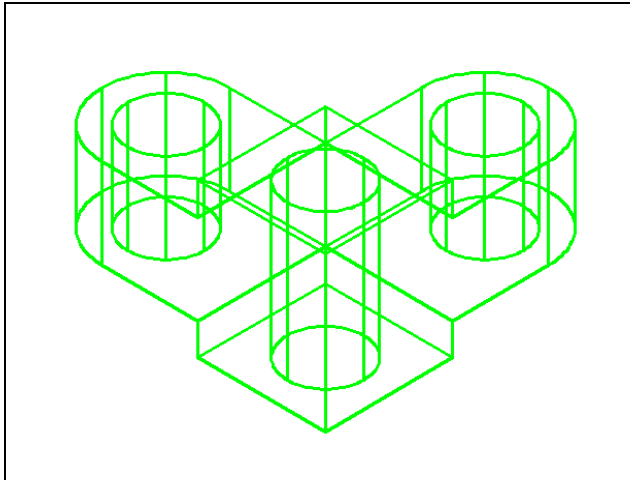


Select the Move tool on the Modify panel to move the wings to the center of the hub. Select the wings and the cylinders inside them as the entities you wish to move and the command line will respond with “4 found”, so just hit ENTER to go the second part of the command. Next you need to “specify base point or displacement” on the graphical display. Pick your base point anywhere on the display. Move your cursor up along the Z axis, type “5”, and hit ENTER. The wings will now reside center on the middle hub as shown in Figure 6.28.

Figure 6.28 – Moving the wings

Union the Solids

Now that you have all of the 3D solids in place, you are ready to join and subtract them as necessary to create the final product. You will use the Boolean command Union first to create the master solid.

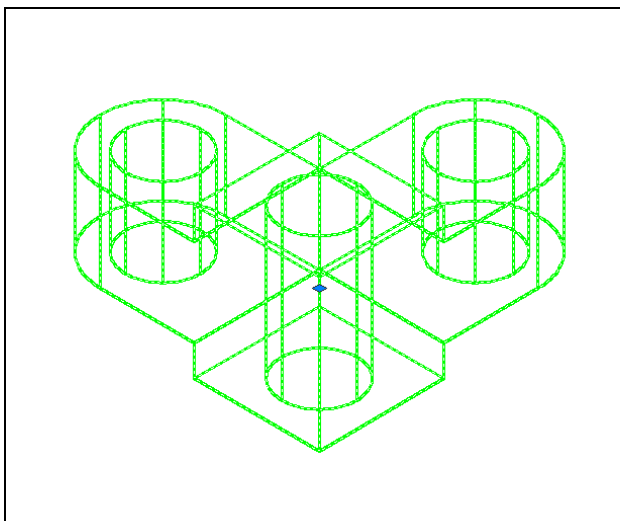


Select the Union tool on the Solid Editing panel under the Home tab. The command line will prompt you to “**Select objects**”. Select only the perimeter solids (none of the cylinders) and the command line will respond with “**3 found**”. Hit ENTER to combine all of the selected solids into one entity as shown in Figure 6.29. You can see that the tessellation lines outlining the connecting edges of the solids are now missing. If you select the solid with one pick of the mouse the entire solid will become highlighted.

Figure 6.29 – Union of the solids

Subtracting a 3D Solid from Another Solid

Select the Subtract tool on the Solid Editing panel to remove the cylinders from the master solid.

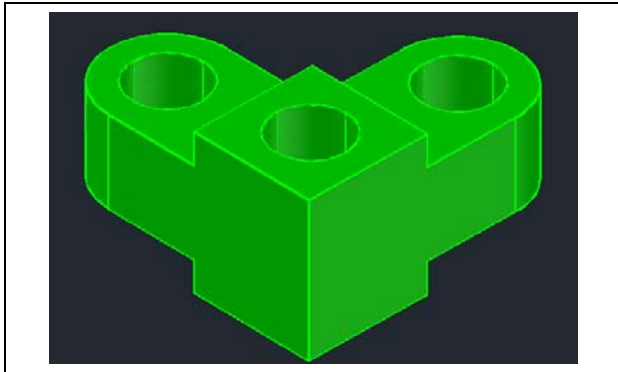


The command line will prompt you to “**Select solids and regions to subtract from**”. Select the larger master solid and the system will respond with “**1 found**”. Hit ENTER to proceed to the next step, which is to “**Select solids and regions to subtract**”. Pick the three cylinders and the command line will respond with “**3 found**”. Hit ENTER and the shape of the three cylinders will disappear from the master solid as shown in Figure 6.30. Notice that you cannot see any difference in the solid when you subtract the cylinders. You will learn the Shade command to see your results better.

Figure 6.30 – The single solid selected

Shade the 3D Solid

For a more realistic view of Solid Problem Five, select the Shaded with Edges visual style from the View panel. The solid will acquire a painted look matching the color of the layer, which the part resides.



The part is now a solid green color. The tessellation lines associated with the Hide command are no longer interfering with your ability to read the drawing. Your customers will most likely prefer this mode to the Hide command because they will be able to visualize the product clearly. If you ever desire to return to standard viewing, choose the 2D Wireframe tool to return to the previous visual style.

Figure 6.31 – Shade the Solid

Rotating a 3D Solid Using 3D Rotate

Select the 3D Rotate tool from the Modify panel. When prompted to “select objects”, pick the solid on the graphical display and hit **ENTER** to proceed to the second part of the command. You are going to rotate the solid part on the x-axis, so select the circle that corresponds with that axis (the gold circle as shown in Figure 6.32; in this case you do not need to select a base point). With the Ortho mode on, move the cursor so that the solid rotates 90 degrees as shown in Figure 6.32. Now select the circle that represents the Z axis (the gold circle as shown in Figure 6.33) and rotate the solid 90 degrees so that it appears as shown in Figure 6.33.

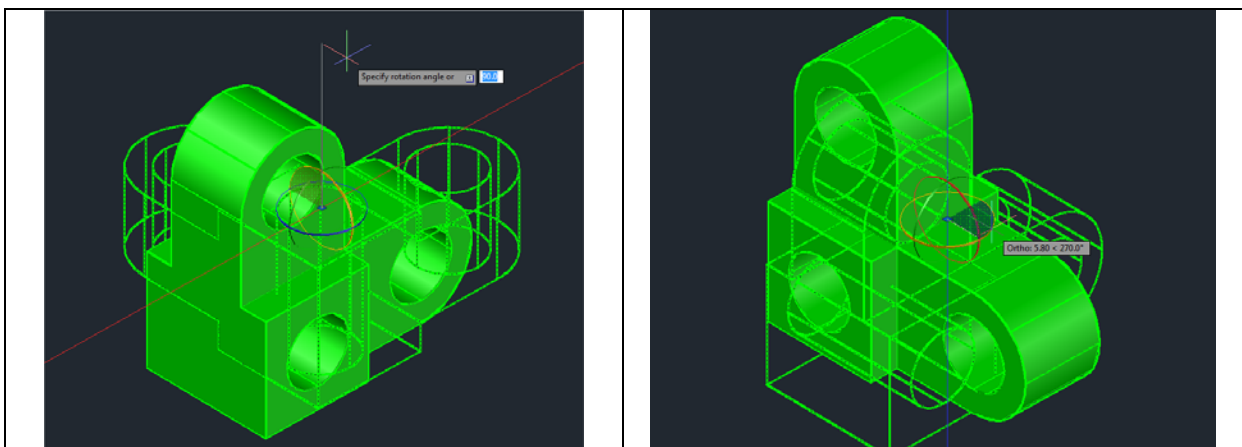


Figure 6.32 – Rotating on the X axis

Figure 6.33 – Rotating on the Z axis

Moving the Solid to the Origin Point

To move Problem Five 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 6.34, 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 Five 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.

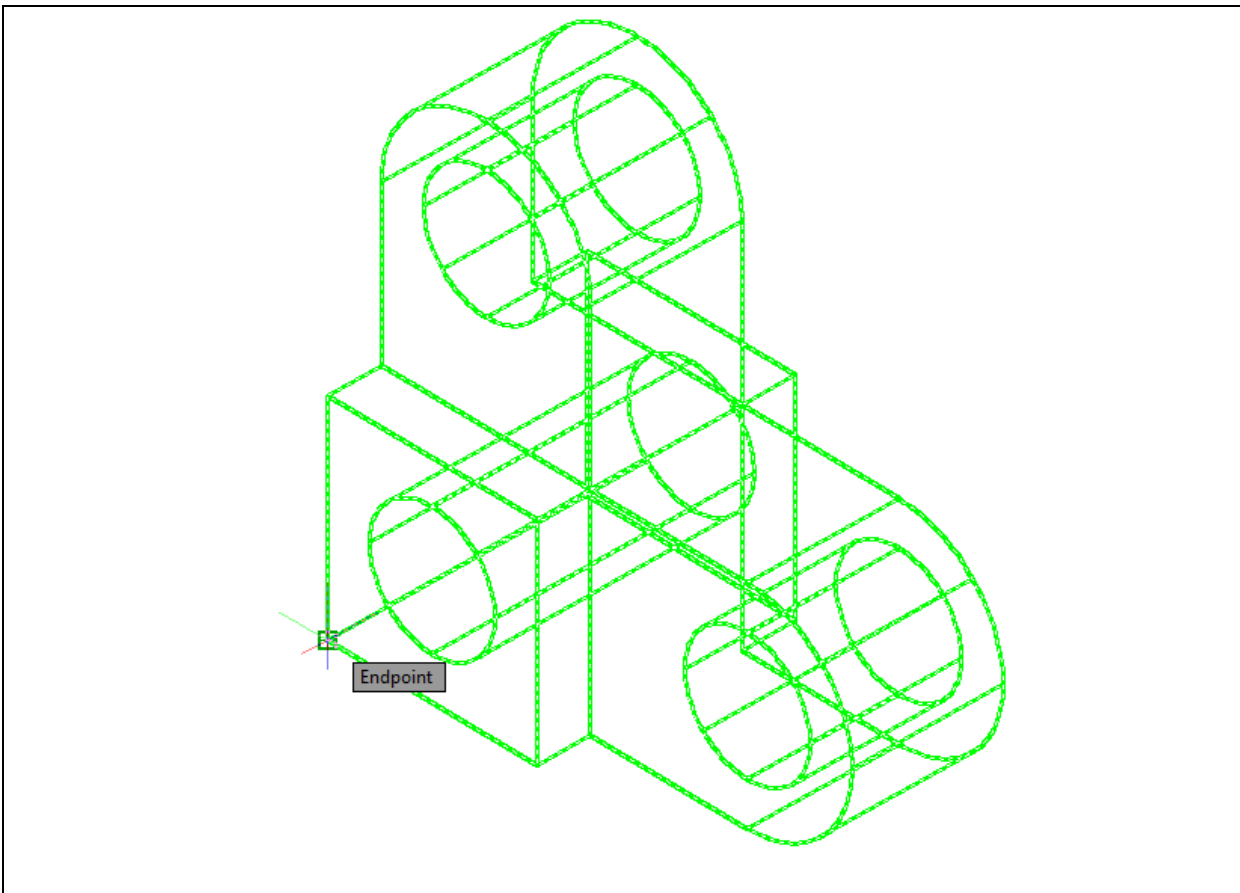


Figure 6.34 – Moving the finished solid to the origin

Saving the Solid Problem

To save Problem 5 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. At the File Name



textbox, type “Problem 5” and hit the Save button to save the drawing (Figure 6.35).

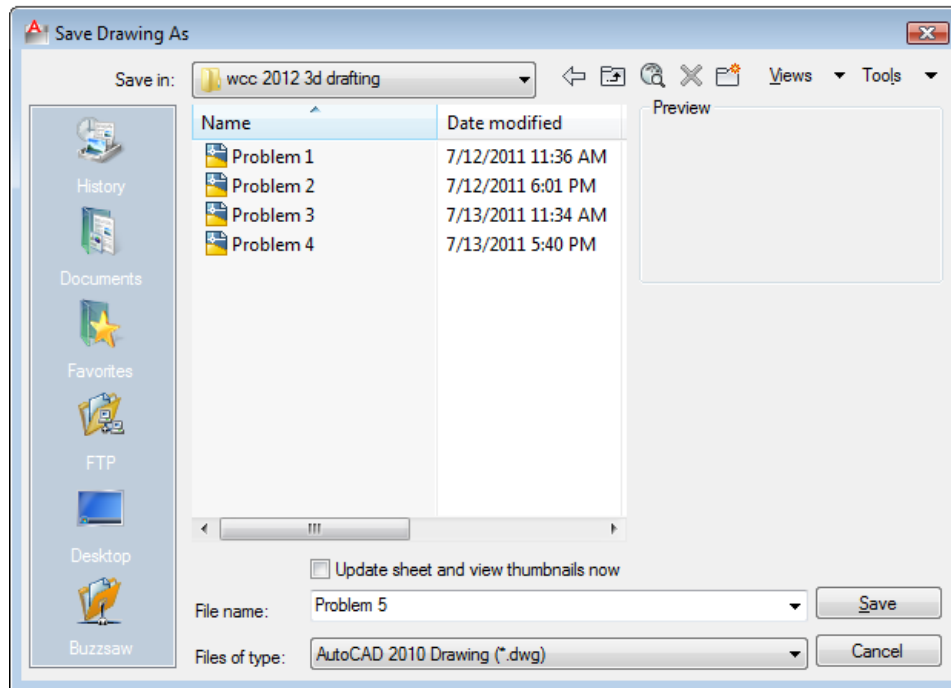


Figure 6.35 – Saving Problem Five

*** World Class CAD Challenge 03-05 * - Close this drawing file. Create a New file and draw the closed Polyline, create the two details adjoining the Polyline box. Modify and extrude the entities, union and subtract them. 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.**