

Solid Problem Three

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

- 1. Sketch of Solid Problem Three**
- 2. Starting a 3D Part Drawing**
- 3. Drawing a Closed Polyline Shape to be Extruded**
- 4. Adding Circles to be Extruded**
- 5. Rotating a 3D Solid Using the Rotate Command**
- 6. Extruding a Solid from Closed Polylines and Circles**
- 7. Moving a Solid within the Master Solid**
- 8. Union Two Solids**
- 9. Subtracting Two Solids from the Master Solid**
- 10. Placing an Curved Edge on a Solid Using the Fillet Tool Moving**
- 11. Saving the Solid Problem**

Starting a 3D Part Drawing

In Problem Three, you will begin the 3D drawing by selecting the Mechanical template. In Figure 4.2, the window “Select template” window is loaded by selecting the New tool on the Standard Toolbar. 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.

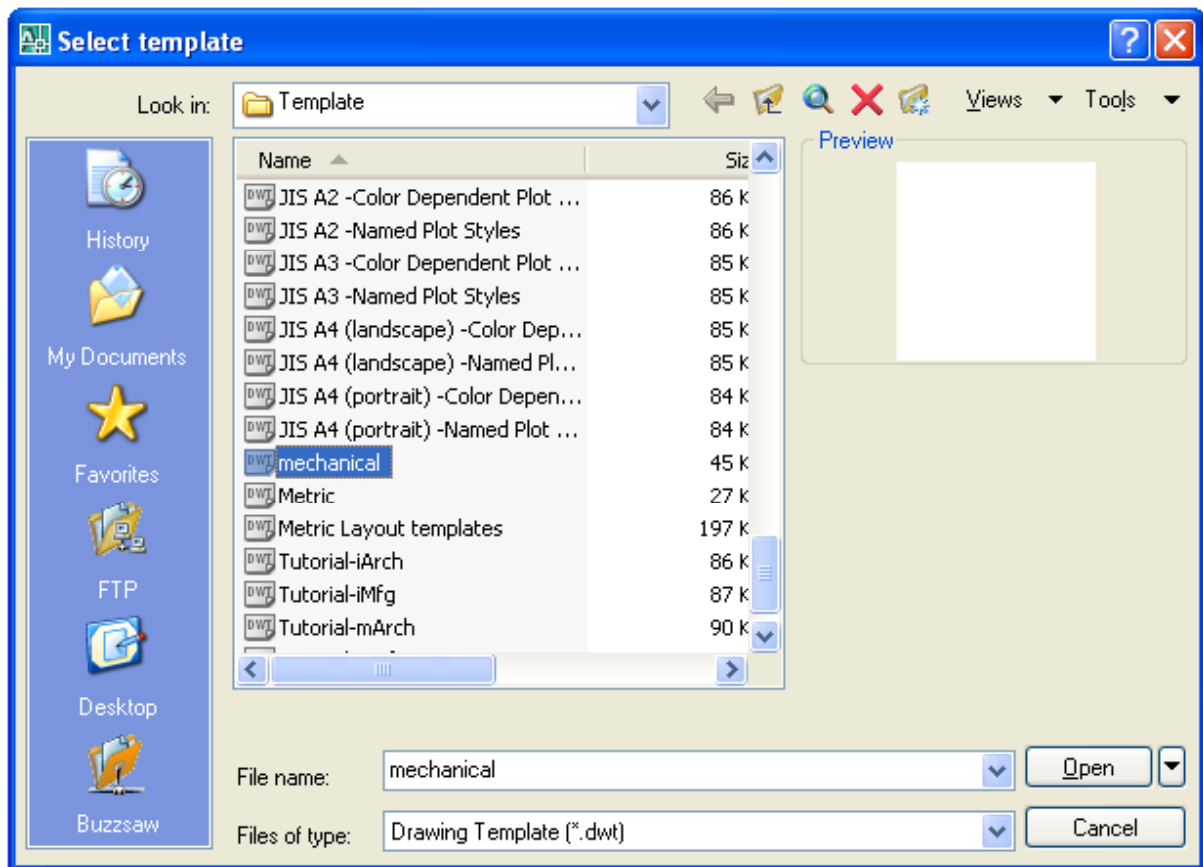


Figure 4.2 – Starting the Drawing Using the Mechanical Template

Drawing a Closed Polyline Shape to be Extruded

On the Draw Toolbar, select the Polyline tool and specify a start point in the lower left hand corner of the graphical display. In Figure 4.3, you can see the actual continuous line of data you need to specify at the command line.

```
Command: _pline  
Specify start point:  
Current line-width is 0.0000  
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: 2.75  
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 3  
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 2.75  
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 0.75
```

Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 2
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 1.5
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 2
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: C

Figure 4.3 – The Polyline Text from the Command Line

With the Ortho mode “on”, draw a **2.75** unit line to the right, a **3.0** line up, a **2.75** line to the left, a **0.75** line downward, a **2.0** line to the right, a **1.5** line downward, a **2.0** to the left and then type “C” to close the Polyline. A closed entity will appear in the graphical display as shown in Figure 4.4. Select the Zoom Extents tool on the Standard toolbar to have the part fill the monitor, so you can proceed to the next step in the drawing process.

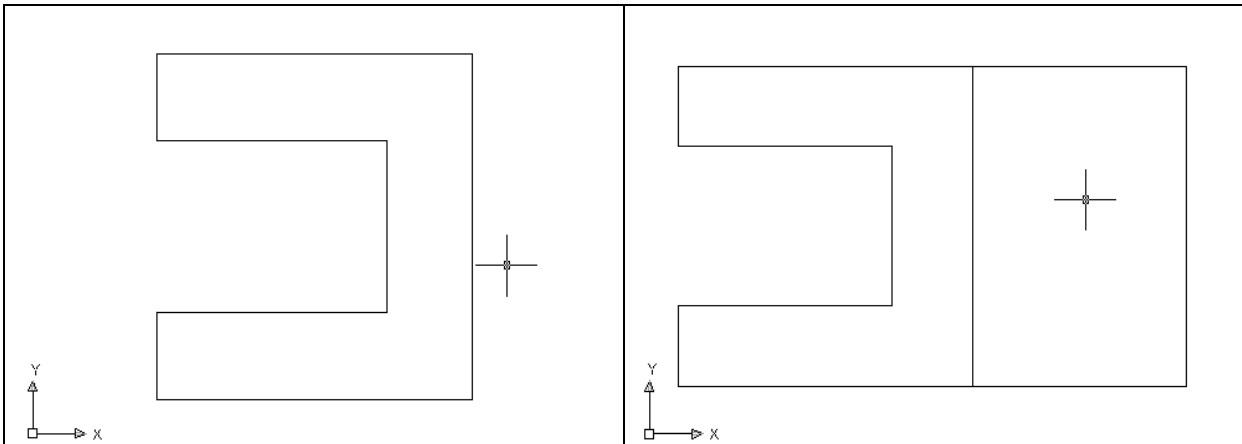


Figure 4.4 – The First Closed Polyline

Figure 4.5 – The Second Closed Polyline

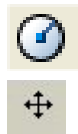
Command: **_pline**
Specify start point:
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: 2
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 3
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 2
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: C

Figure 4.6 – The Polyline Text from the Command Line

With the Ortho mode still “on”, press Enter to repeat the Polyline command and select the lower right end of the first Polyline as your starting point. Draw a **2.0** unit line to the right, a **3.0** line up, a **2.0** 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 4.5. Select the Zoom Extents tool on the Standard toolbar to have the box fill the monitor, so you can proceed to the next step in the drawing process.

Adding Circles to be Extruded

Next, select the Circle command on the Draw toolbar. Place the mouse aperture midway up the vertical line as shown in Figure 4.7 and a Midpoint Osnap symbol (triangle) will appear in the middle of the line. Select the center point of the circle by selecting with the left mouse button. You need to type “d” for diameter when the



command line prompts you to “**specify radius of circle or [Diameter]**”. Then type “**1**” when the command line prompts you to “specify diameter of circle”. Repeat the Circle command by pressing Enter, and place the **1.25** diameter circle outside the 1-diameter circle having them sharing the same center points. There will be two concentric circles as shown in Figure 4.7.

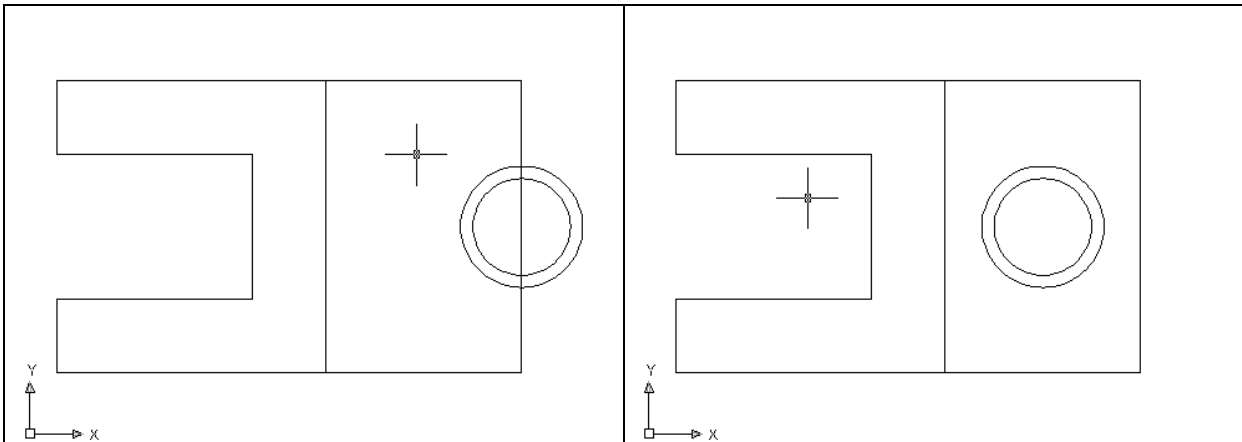


Figure 4.7 – Drawing Two Circles

Figure 4.8 – Moving Two Circles

Next, you need to move the two circles one inch to the left, so select the Move tool on the Modify toolbar. Once you choose the Move command, you will be prompted to “**Select Objects**”. You can do this by placing the pick box that now has replaced the aperture cursor on each circle and select both entities. The command line will respond with “**2 found**”, so just press 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 graphical display. Push the mouse to the left to direct the circles and type “**1**”. Both circles will be 1 inch to the left of the right vertical line segment as shown in Figure 4.8.

Rotating a 3D Solid Using the Rotate Command

On the View toolbar, select the Southwest Isometric View tool to allow you to see the part on the XY plane and the vertical Z-axis will rise from the level surface. (See Figure 4.9). To rotate the solid in the XY plane, select the regular Rotate command on the Modify toolbar. Select the Rotate command on the Modify toolbar and pick the solid. The selected entity will be dotted and to continue with the command, press **Enter** to select the base point of rotation. Select any point at the base of the solid with the left mouse button and then type **90** for the degrees of counter clockwise rotation to finish this detail (See Figure 4.10).



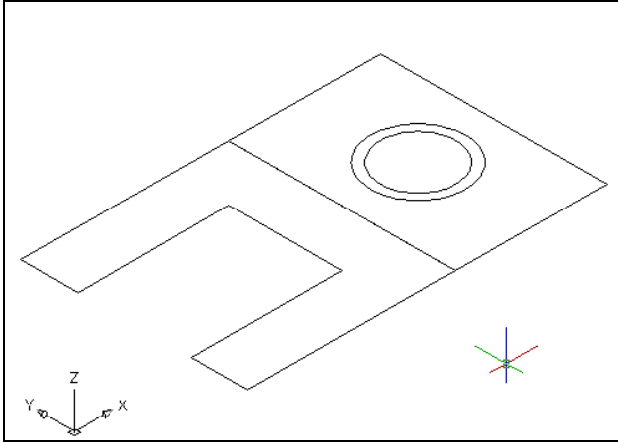


Figure 4.9 – Southwest Isometric View

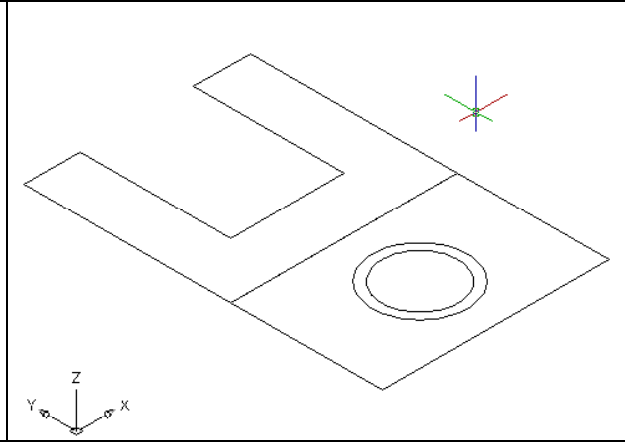


Figure 4.10 – Rotating 90 Degrees

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 Solids toolbar. If there is even a minute space in the entity a warning will appear on the command line that you are “**unable to extrude the selected object.**” If you use an attention to detail in your 3D computer aided design work, you can create the most complex shapes with the extrusion instrument.

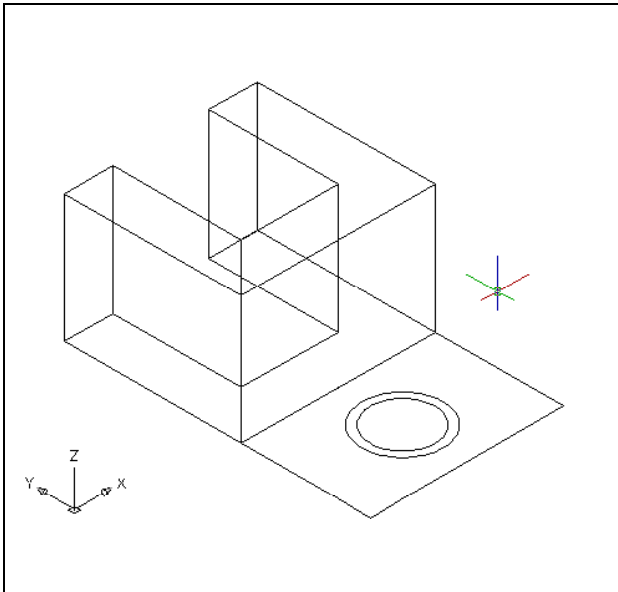


Figure 4.11 – Extrude the Inverted “C”

Select the Extrude tool on the Solids toolbar and the command line on the AutoCAD application will prompt you to “**select objects**”. Pick the Closed Polyline that looks like an inverted “C” anywhere on the peripheral line and the command line will return with “**1 found**”. Press **Enter** to specify the height or path of the solid. You can type “P” for Path and choose an entity to extrude the shape along, but you will not want that option for problem two. You will type “2” for the height of the extrusion and for the angle of taper you will **Enter** to accept the zero degree default. A more complex 3D solid will appear in the graphical display as shown in Figure 4.11.

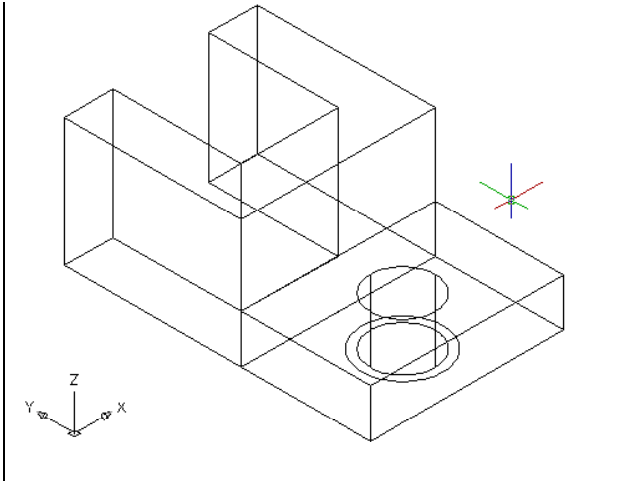


Figure 4.12 – Extrude Two Solids

For the second set of solids, choose the Extrude tool on the Solids toolbar and the command line will prompt you to “**select objects**”. Pick the Closed Polyline that looks like a rectangle anywhere on the peripheral line and the smaller circle and the command line will return with “**2 found**”. Press **Enter** to specify the height or path of the solid. You will type “**0.75**” for the height of the extrusion and for the angle of taper you will **Enter** to accept the zero degree default. A solid box will appear with a solid cylinder in the graphical display as shown in Figure 4.12.

Notice that you are able to extrude more than one entity at a time to create separate solids. This procedure comes in handy when making holes in plates when there are already circles available for extrusion.

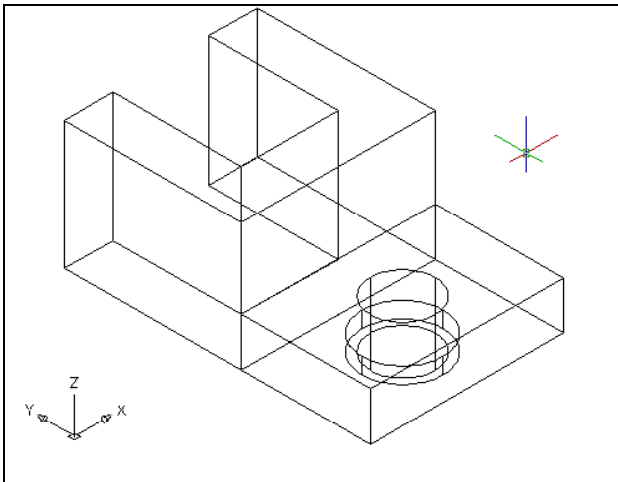


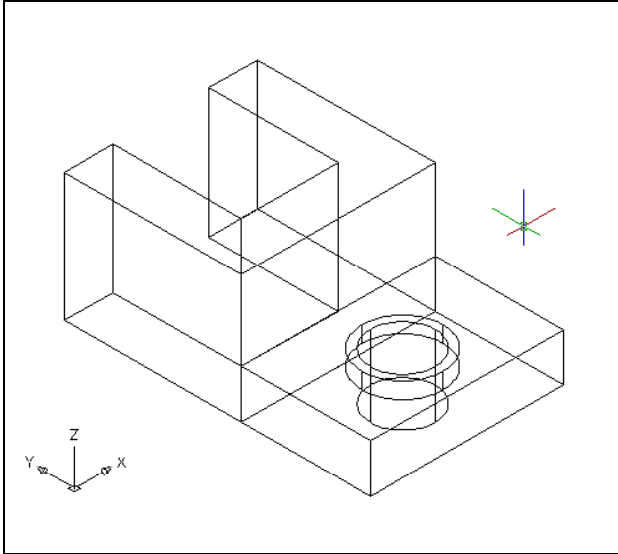
Figure 4.13 – Extrude the Second Cylinder

For the third solid, pick the Extrude tool on the Solids toolbar and the command line will prompt you to “**select objects**”. Pick the larger circle anywhere on the peripheral line and the command line will return with “**1 found**”. Press **Enter** to specify the height or path of the solid. You will type “**0.25**” for the height of the extrusion and for the angle of taper you will **Enter** to accept the zero degree default. A solid cylinder in the graphical display as shown in Figure 4.13. Now you need to move the 1.25 diameter cylinder into place as shown in your sketch.

Moving a Solid onto 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.



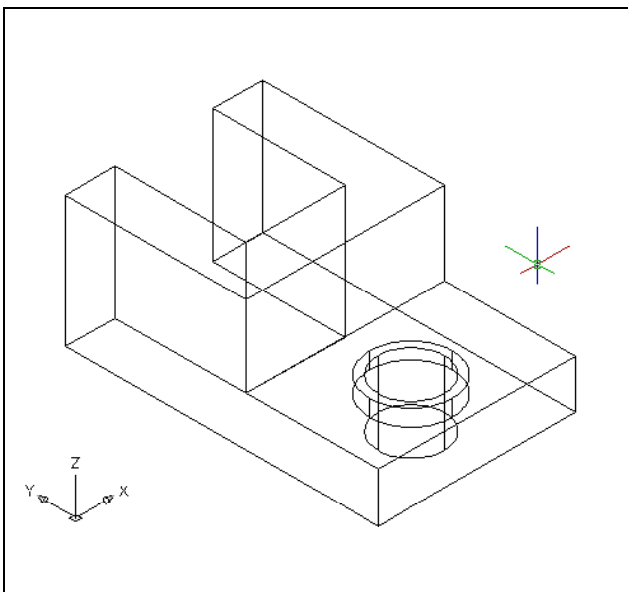


To move the 1.25, you select the Move tool on the Modify toolbar. Once you choose the Move command, the prompt will be “**Select Objects**”. Pick the 1.25 cylinder and the command line will respond with “**1 found**”, so just press Enter to go the second part of the command. Next, you need to “**specify base point or displacement**” on the graphical display. Pick your base point with the Center Osnap selecting the top of the 1.25 cylinder. The “**second point of displacement**” is the center point of the 1.0 cylinder. Pick the second point using the Center Osnap and the 1.25 cylinder will move as shown in Figure 4.14.

Figure 4.14 – Moving a Cylinder

Union Two Solids

For the next modification of a 3D solid shape in AutoCAD, you need to use the Boolean command, Union. With the Union command, you will join one shape or more together.

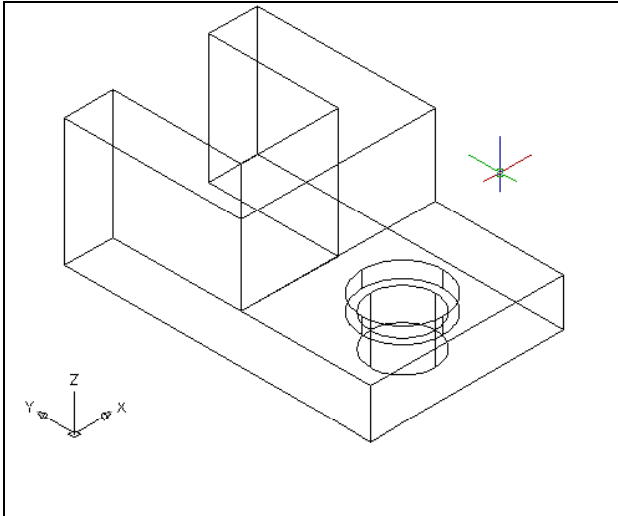


To add two solid to another one, select the Union tool on the Solid Editing toolbar. Then the AutoCAD program will prompt you with “**Select objects**”. Select the inverted “C” solid and the system will respond with “**1 found**”. Pick the rectangular solid and the software application will counter with “**1 found**”. Hit Enter and now both solids become as one entity, as shown in Figure 4.15. You can see in the graphical display that the tessellation lines outlining both boxes and were crossing each other in the corner of the “L” shape are now missing. Select the solid with one pick of the mouse and the entire solid will become highlighted.

Figure 4.15 – Union Two Solids

Subtracting a 3D Solid from Another Solid

To remove the smaller solid boxes from the master solid, select the Subtract tool on the Solid Editing toolbar.



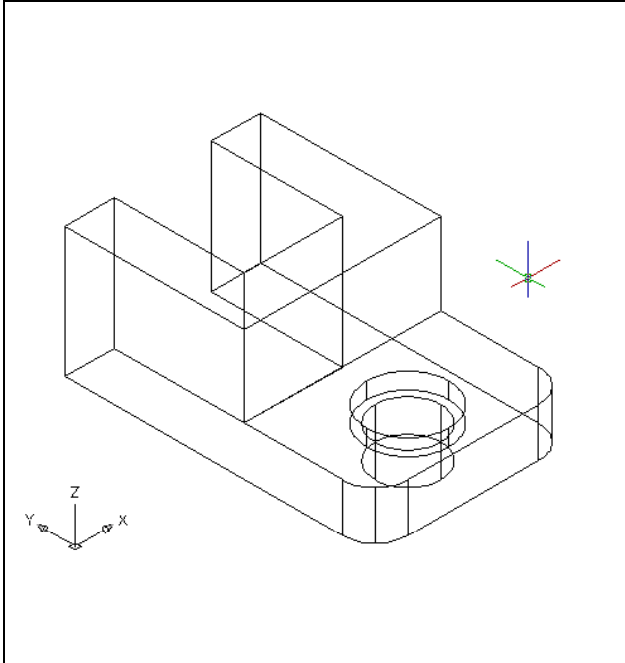
The AutoCAD program will prompt you with **“Select solids and regions to subtract from”**. Select the larger master solid and the system will respond with **“1 found”**. You need to hit Enter to proceed to the next step, which is **“Select solids and regions to subtract”**. Pick the two cylinders and the software application will counter with **“2 found”**. Hit Enter and a region exactly the shape of the two cylinders is missing from the larger one as shown in Figure 4.16. This technique to enhance the detail of a solid entity by removing a part of the feature is commonplace in the world of 3D design.

Figure 4.16 – Subtract the Two Cylinders

Placing a Curved Edge on a Solid Using the Fillet Tool

To place a fillet or curve on two converging straight lines in the Fundamentals of 2D Drawing, you need to select both lines to identify the position of the radius. When placing a three dimensional curve on a solid, you pick the straight line on the corner. In this case, the curvature is on the edge of a box in the z-axis.





Select the Fillet tool on the Modify toolbar. Again, this is a very simple command, because of the movements after picking the tool. Type an “**R**” for radius and then 0.5 for the measurement. Now pick the vertical line on the straight corner and Enter and the line will become dotted. At the prompt, “**Select an edge or [Chain/Radius]:**” just hit Enter again since you will not be linking the curved surface with any other edge causing a continuous rounded periphery. On the second Enter, the sharp corner becomes a 0.5 curvature. Repeat the process for the other side of problem three and you will have symmetrical fillets. You can actually pick more than one edge to fillet at a time with the same measurement, but on this first attempt, stay with the basics.

Figure 4.18 – Placing Curves on a Solid

Moving the Solid to the Origin Point

To move Problem Three to the drawing origin of 0,0,0 on the x, y and z axis, pick the Move tool on the Modify toolbar, select the 3D solid, and hit Enter. At the prompt, “**specify base point or displacement**”, select the endpoint shown in Figure 4.18 at the upper left hand side of the Problem Three, which will become the Problem Three insertion point into an assembly drawing. At the “second point of displacement”, type 0,0,0 and Enter. Problem Three will move to the new origin point as shown in Figure 4.19. There is more than one way to do many actions in AutoCAD, but the end result of having a part at the origin is important when you wish to insert the solid into an External Referenced Assembly drawing.

*** World Class CAD Challenge 03-03 * - Close this drawing file. Create a New file and draw the twp closed Polylines, extrude and union them. Draw and extrude the two circles, move and subtract them. Add two curve edges on the solid. 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.**

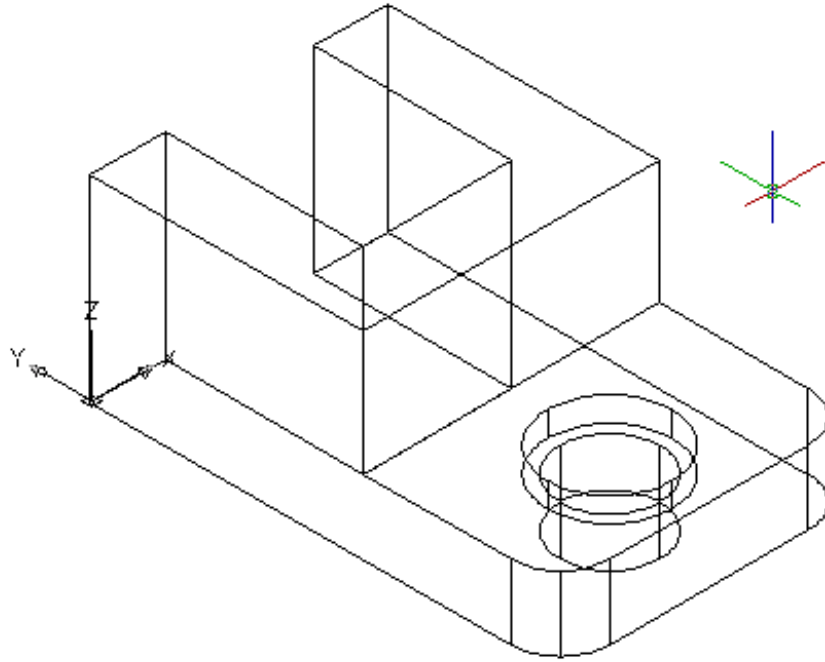


Figure 4.19 – Moving the Finished Solid to the Origin

Saving the Solid Problem

To save Problem 3 in your Fundamentals of 3D Drawing Folder, select the Save tool on the Standard 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 3” and press the Save button to save the drawing. (See Figure 4.20)



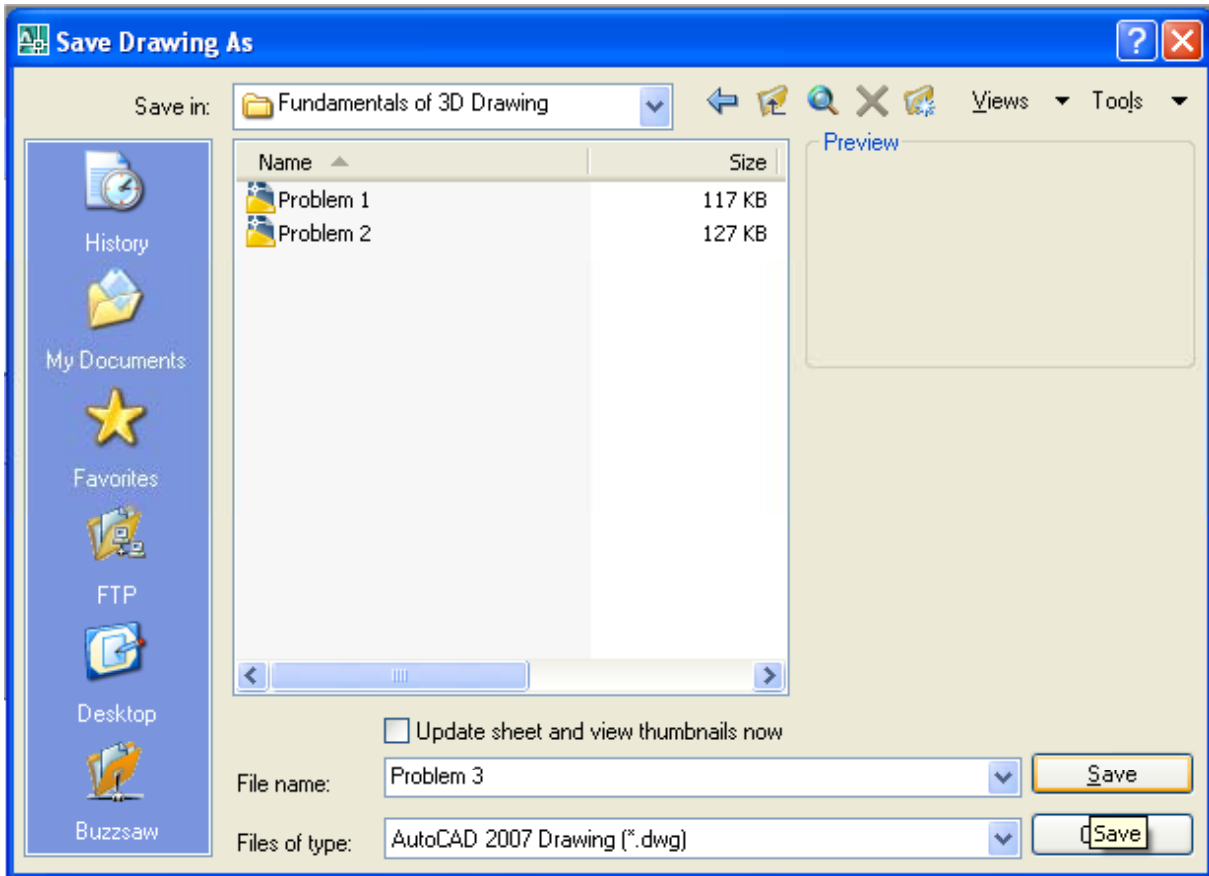


Figure 4.20 – Saving Problem Three

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