

Chapter

4

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. Combining Two Solids with Union
9. Subtracting Two Solids from the Master Solid
10. Placing an Curved Edge on a Solid Using the Fillet Tool
11. Saving the Solid Problem

Sketch of Solid Problem Three

You start the third solids problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across one line that you can call the y-axis. In this exercise, a strategy you will continue to use is to draw closed Polylines that contains the most complex shape of the component and then use simple geometric solids to add or subtract detail. In problem three the solid contains an inverted “c,” that will be formed using a Polyline shape, along with a second rectangular Polyline contour. You will place two circles in the middle of the second shape. After extruding the Polylines and Circles, you will union the two shapes together and then after subtracting the two circles you will fillet a curve in two places as shown in your sketch (Figure 4.1).

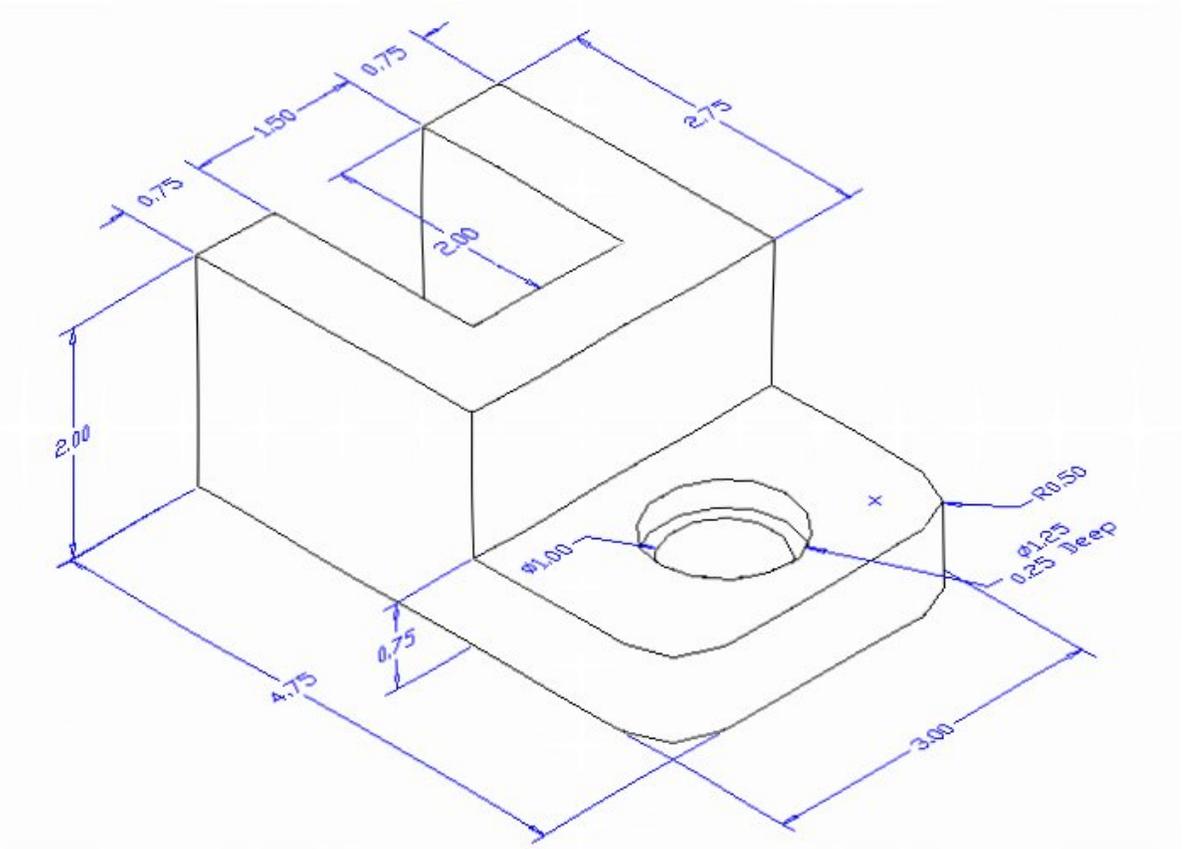


Figure 4.1 – Problem Three Sketch

Remember, when you receive a sketch from a professional, you need to practice to quickly identify the different shapes and decide how to proceed with the drawing. You can see rather quickly that problem three has different heights, so separate each extrusion or part and then bring them together. This problem will help you address solid parts containing different complexities and thicknesses. All ten drills in the Fundamentals of 3D Drawing textbook will give you those repetitive maneuvers, which will make you successful in 3D modeling.

Starting a 3D Part Drawing

In Problem Three, you will begin the 3D drawing by selecting the New tool on the Standard Toolbar. Select the “Start from Scratch” (first button) at the top of the Start window. Choose to use English units as your Default Settings and hit OK. 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.

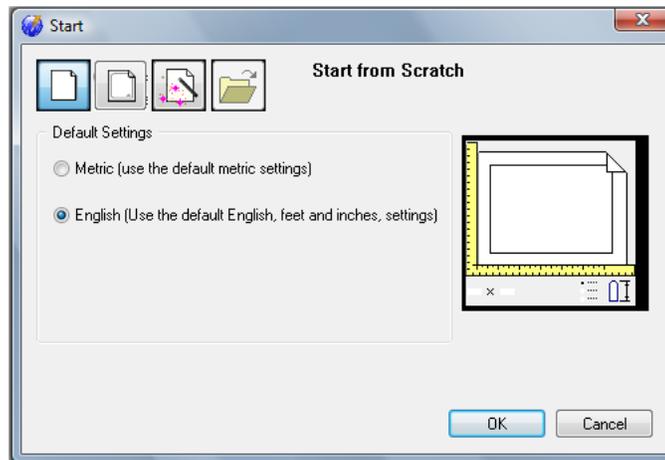


Figure 4.2 – Starting the Drawing from Scratch

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 : _POLYLINE  
Start of polyline:  
Arc/Distance/Halfwidth/Width/<Next point>: 2.75  
Arc/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 3  
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 2.75  
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 0.75  
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 2  
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 1.5  
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 2  
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 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. Zoom Extents to have the part fill the monitor so you can proceed to the next step in the drawing process.

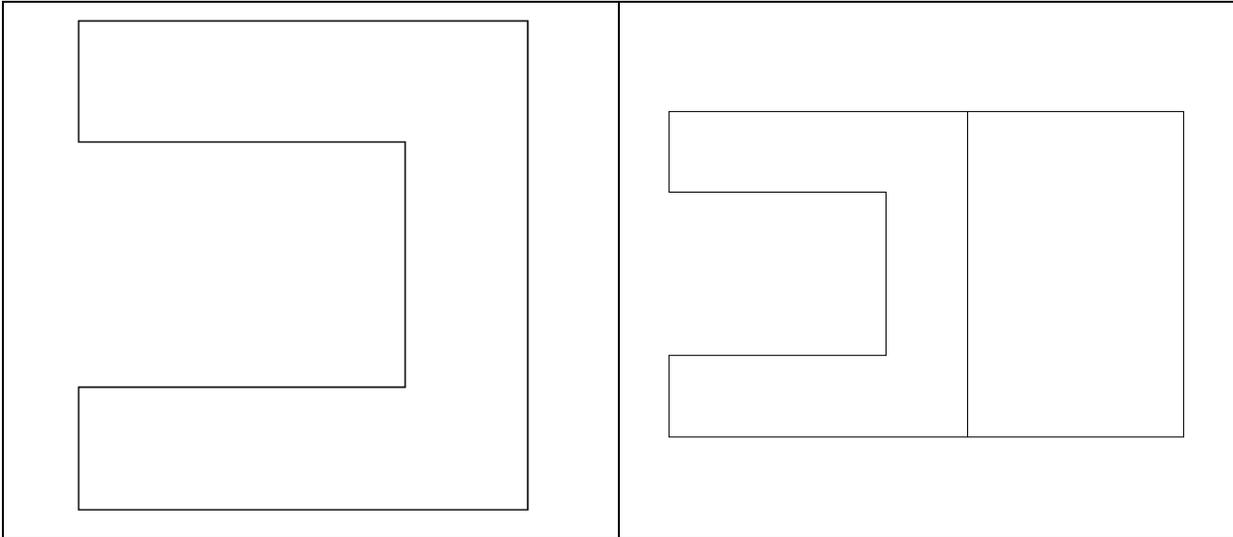


Figure 4.4 – The First Closed Polyline

Figure 4.5 – The Second Closed Polyline

```
Command : _POLYLINE
ENTER to use last point/Follow/<Start of polyline>:
Arc/Distance/Follow/Halfwidth/Width/<Next point>: 2
Arc/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 3
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 2
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: C
```

Figure 4.6 – The Polyline Text from the Command Line

With the Ortho mode still “on,” repeat the Polyline command and select the bottom right corner 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. Zoom extents again if necessary.

Adding Circles to be Extruded

Next, select the Circle command on the Draw toolbar. Place the cursor midway up the left side of the drawing (Figure 4.7) and a Midpoint Esnap symbol (triangle) will appear in the middle of the line. Make the midpoint the center point of the circle by clicking with the left mouse button. You need to type “**d**” for diameter when the command line prompts you with “**Diameter/<Radius>:**.” Then type “**1**” when the command line prompts you with “**Diameter of circle:**.” Repeat the Circle command by hitting ENTER and place a **1.25** diameter circle outside the 1-diameter circle so that both circles share the same center point. Your drawing should now show two concentric circles as shown in Figure 4.7.



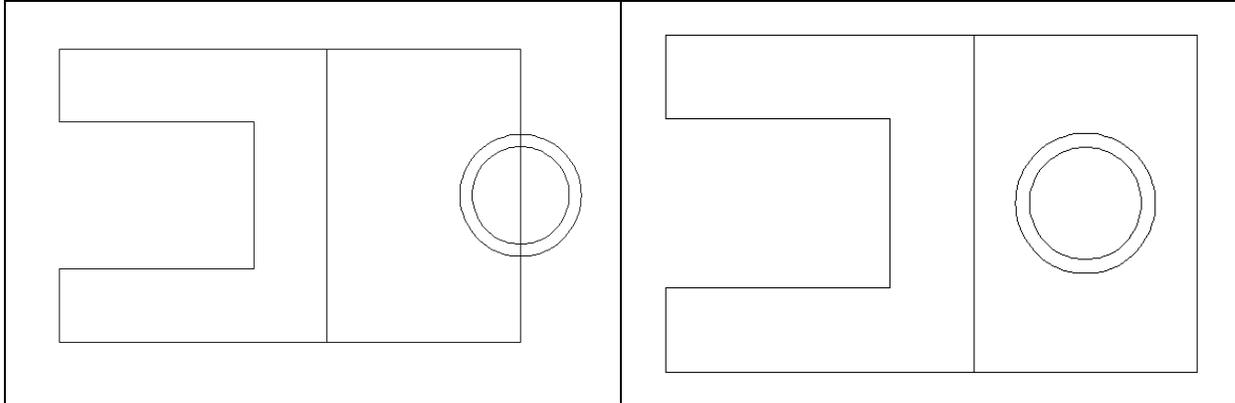


Figure 4.7 – Drawing Two Circles

Figure 4.8 – Moving Two Circles

Now 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 entities to move:**” You can do this by placing the pick box that now has replaced the aperture cursor on each circle and clicking to select. The command line will respond with “**Entities in set: 2,**” so just hit ENTER to go the second part of the command. Next you need to choose a “**Vector/<Base point>:**” somewhere on the graphical display. Push the mouse to the left and type **1**. Both circles will move 1 inch to the left 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 with the vertical Z-axis rising from the level surface (Figure 4.9). To rotate the solid in the XY plane, select the Rotate command on the Modify toolbar. Use a selection window to select all of the circles and polylines. The selected entity will be dotted; to continue with the command, hit **ENTER** to select the base point of rotation. Select any point at the base of the solid with the left mouse button. With the Ortho on, you should be able to easily move the objects 90 degrees clockwise, using the preview as your guide (Figure 4.10).

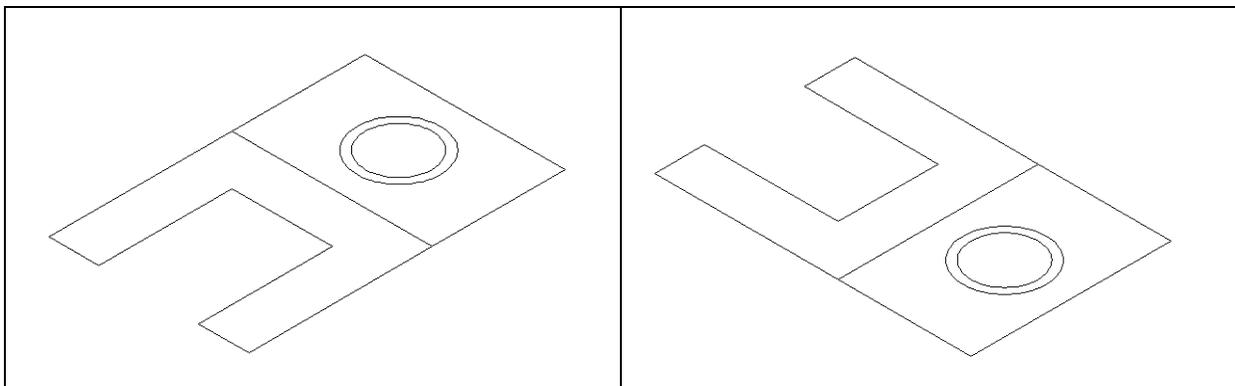
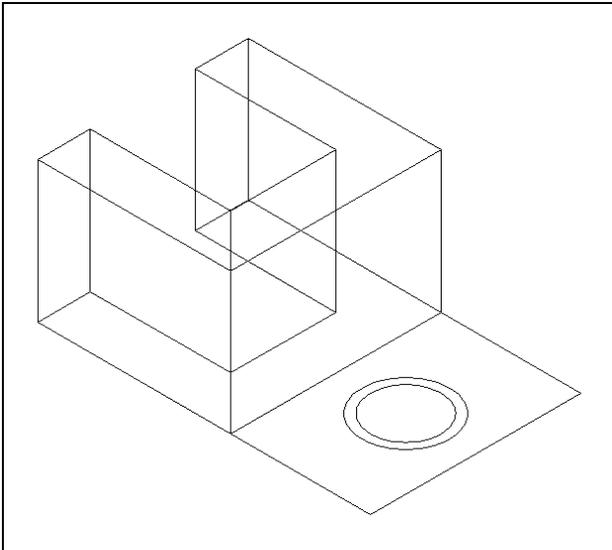


Figure 4.9 – Southwest Isometric View

Figure 4.10 – Rotating 90 Degrees

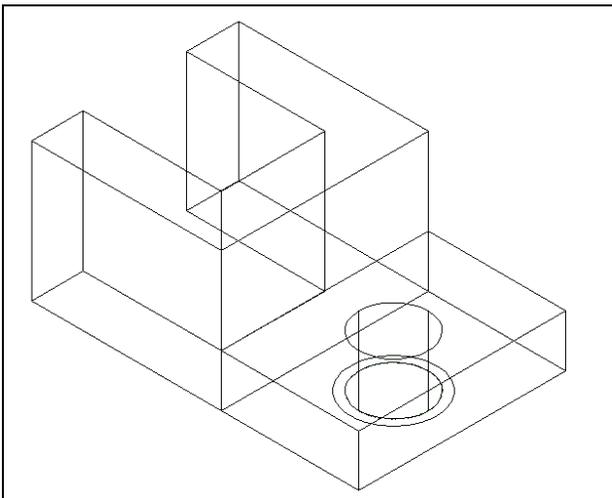
Extruding a Solid from Closed Polylines and Circles

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 you will not be able to select it to extrude or the tool may reject the selection. If you use an attention detail in your 3D computer aided design work, you can create quite complex shapes with this tool.



Select the Extrude tool on the Solids toolbar and the command line will prompt you to “**Select entities:**.” Pick the closed Polyline that looks like an inverted “C” anywhere on the peripheral line and the command line will return with “**Entities in set: 1.**” Hit **ENTER** to specify the height or path of the solid. You can type “P” for Path and choose an entity along which to extrude the shape, but you will not want that option for this problem. Type “2” for the height of the extrusion, and for the angle of taper hit **ENTER** to accept the zero degree default. The polyline will expand 2 units into the z-axis as shown in Figure 4.11.

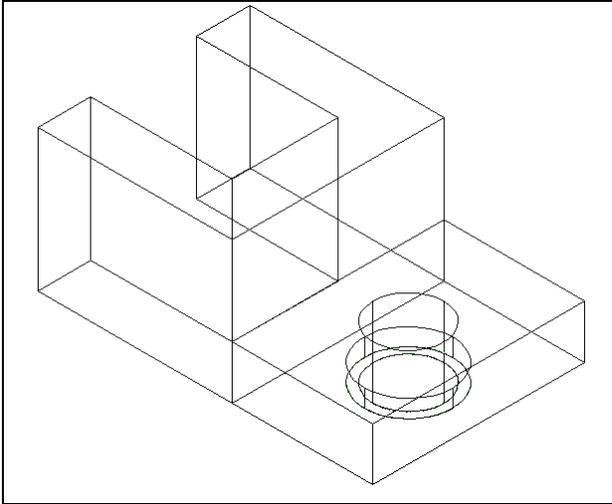
Figure 4.11 – Extrude the Inverted “C”



For the second set of solids, again choose the Extrude tool on the Solids toolbar and the command line will prompt you to **Select entities:**.” Pick the closed Polyline that looks like a rectangle and the smaller circle and the command line will respond with “**Entities in set: 2.**” Hit **ENTER** to specify the height or path of the solid. Type “0.75” for the height of the extrusion; for the angle of taper just hit **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.

Figure 4.12 – Extrude Two Solids

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 with circles already available for extrusion.

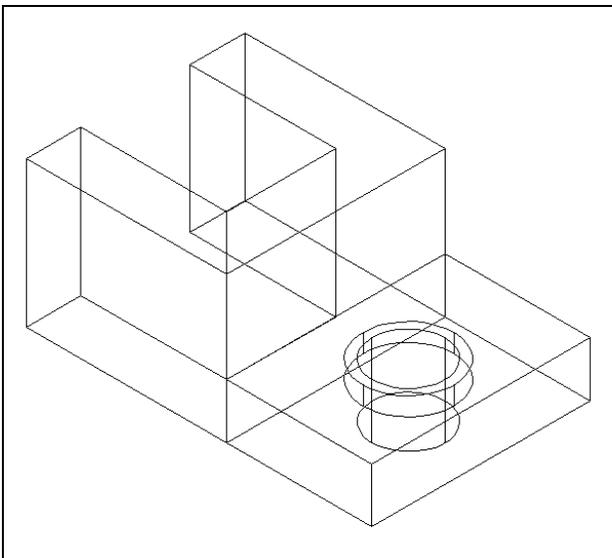


For the third extrusion, again choose the Extrude tool on the Solids toolbar and the command line will prompt you to **Select entities:**” Pick larger circle and the command line will respond with “**Entities in set: 1.**” Hit **ENTER** to specify the height or path of the solid. Type “**0.25**” for the height of the extrusion; for the angle of taper just hit **ENTER** to accept the zero degree default. A shorter by wider cylinder will appear 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.

Figure 4.13 – Extrude the Second Cylinder

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.



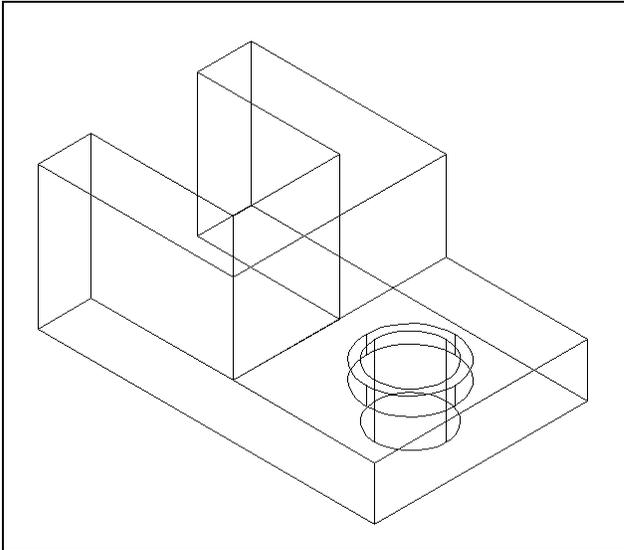
To move the 1.25-diameter circle, select the Move tool on the Modify toolbar and pick the 1.25 cylinder. The command line will respond with “**Entities in set: 1,**” so just hit **ENTER** to go the second part of the command. Next you need to select a “**Vector/<Base point>:**” on the graphical display. Pick your base point with the Center Esnap representing the top of the 1.25 cylinder. The “**Displacement point:**” is the center point of the 1.0 cylinder. Pick the second point using the Center Esnap and the 1.25 cylinder will move as shown in Figure 4.14.

Figure 4.14 – Moving a Cylinder

Combining Two Solids with Union

For the next step in this problem, you need to use the Union Boolean command. With the Union command, you will join multiple solids into one.



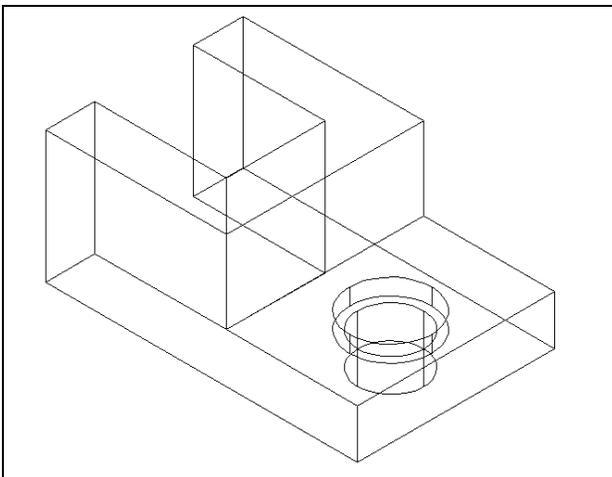


To combine more than one solid, select the Union tool on the Solids Editing toolbar. You will be prompted to “**Select ACIS objects to union:**.” Select the inverted “C” solid and the system will respond with “**Entities in set: 1.**” Also select the rectangular solid so that there are two entities in the set. Hit ENTER to combine both solids into one entity as shown in Figure 4.15. You can see in the graphical display that the tessellation lines outlining both boxes that 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 cylinders from the master solid, select the Subtract tool on the Solids Editing toolbar.



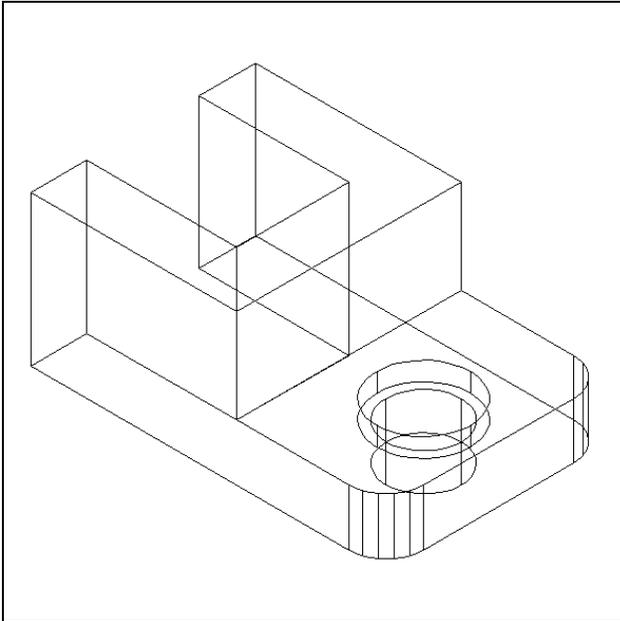
You will be prompted to “**Select ACIS object to subtract from:**.” Select the larger master solid and the system will respond with “**Entities in set: 1.**” Hit Enter to proceed to the next step, which is to “**Select ACIS objects to subtract:**.” Pick the two cylinders and the software command line will respond with “**Entities in set: 1.**” Hit ENTER and a region exactly the shape of the two cylinders will be removed from the larger one as shown in Figure 4.16.

Figure 4.16 – Subtract the Two Cylinders

Placing an Curved Edge on a Solid Using the Fillet Tool

To place a fillet or curve on two converging straight lines in 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 of the edge you wish to round. In this case, the curvature is on the edge of a box along the z-axis.





Select the Fillet tool on the Modify toolbar., Type an **R** for radius and then **0.5** for the measurement. You will see what a very simple command this is by picking the vertical line on the straight corner and hitting ENTER; the line will become dotted. At the prompt, “**Select an edge: Chain/Radius:**” just hit ENTER again since you will not be linking the curved surface with any other edge. Now the sharp corner becomes a 0.5 curvature. Repeat the process for the other side of problem three and you will have symmetrical fillets.

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. When prompted to select a “**Vector/<Base point>:**,” select the endpoint shown in Figure 4.18 at the very left corner of Problem Three, which will become Problem Three’s insertion point if it were inserted into an assembly drawing. For the prompt to select a “**Displacement point:**,” type **0,0,0** and hit ENTER. Problem Three will move to the new origin point as shown in Figure 4.19. Try using Zoom Extents if the part completely escapes your viewing area.

*** World Class CAD Challenge 101-03 * - 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.**

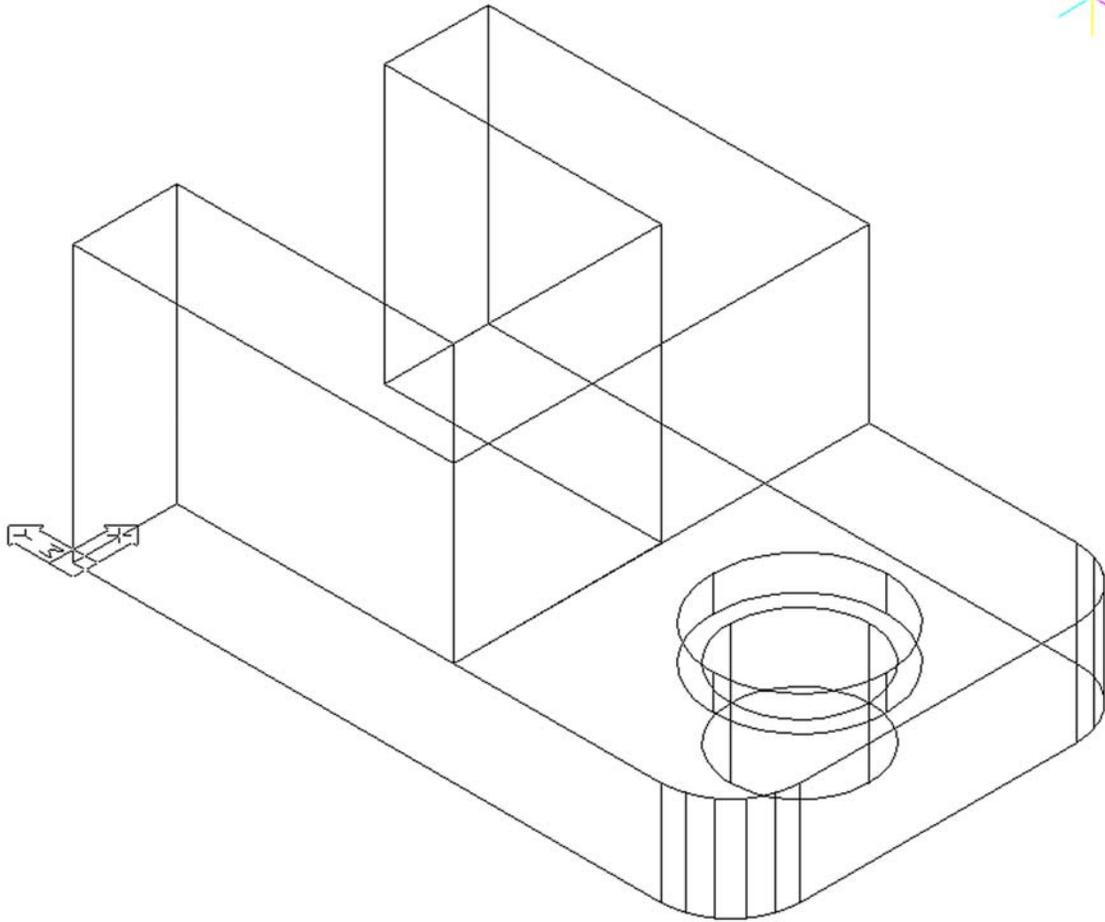


Figure 4.19 – Moving the Finished Solid to the Origin

Saving the Solid Problem

To save Problem 3, 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 (Figure 4.20).



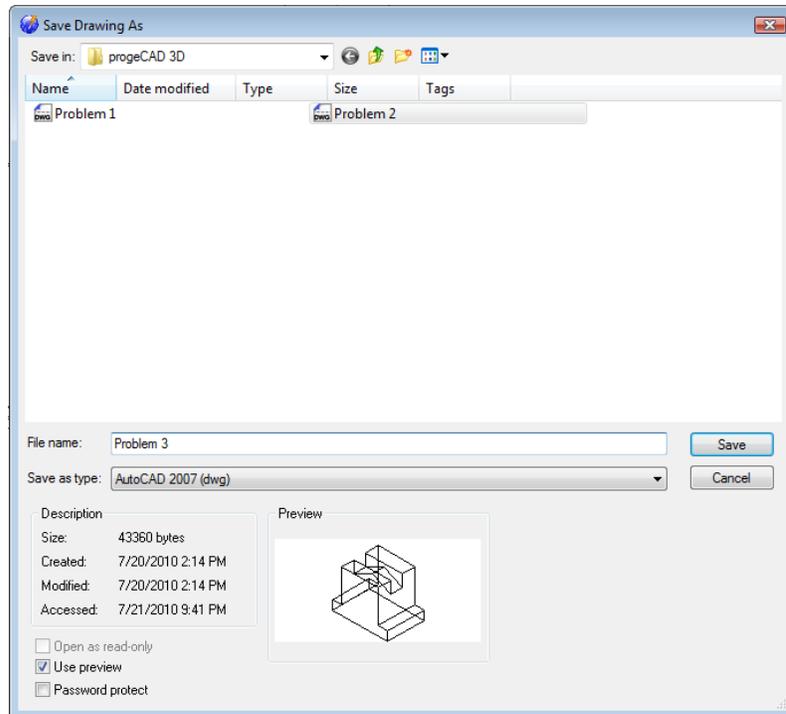


Figure 4.20 – Saving Problem Three

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