C h a p t e r

4

Solid Problem Three

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

- **Sketch of Solid Problem Three**
- **Starting a 3D part drawing**
- **Drawing a closed polyline shape to be extruded**
- **Adding circles to be extruded**
- **Rotating a 3D solid using the Rotate command**
- **Extruding a solid from closed polylines and circles**
- **Moving a solid within the master solid**
- **Union of two solids**
- **Subtracting two solids from the master solid**
- **Placing an curved edge on a solid using the Fillet tool**
- **Moving the solid to the origin point**
- **Saving the solid problem**

Sketch of Solid Problem Three

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You will start the third solids problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across one the y-axis. In this exercise, you will draw closed Polylines that contain the most complex shapes of the component and then use simple geometric solids to add or subtract additional details. Solid problem three contains an inverted "c" that will be formed into a Polyline shape along with a second rectangular Polyline contour with two circles in its center. After extruding the Polylines and Circles, we will use the Union command to join the two shapes together. Then you will subtract the two circles to make the hole and counterbore. You will fillet a curve in two places as shown in the sketch in Figure 4.1.
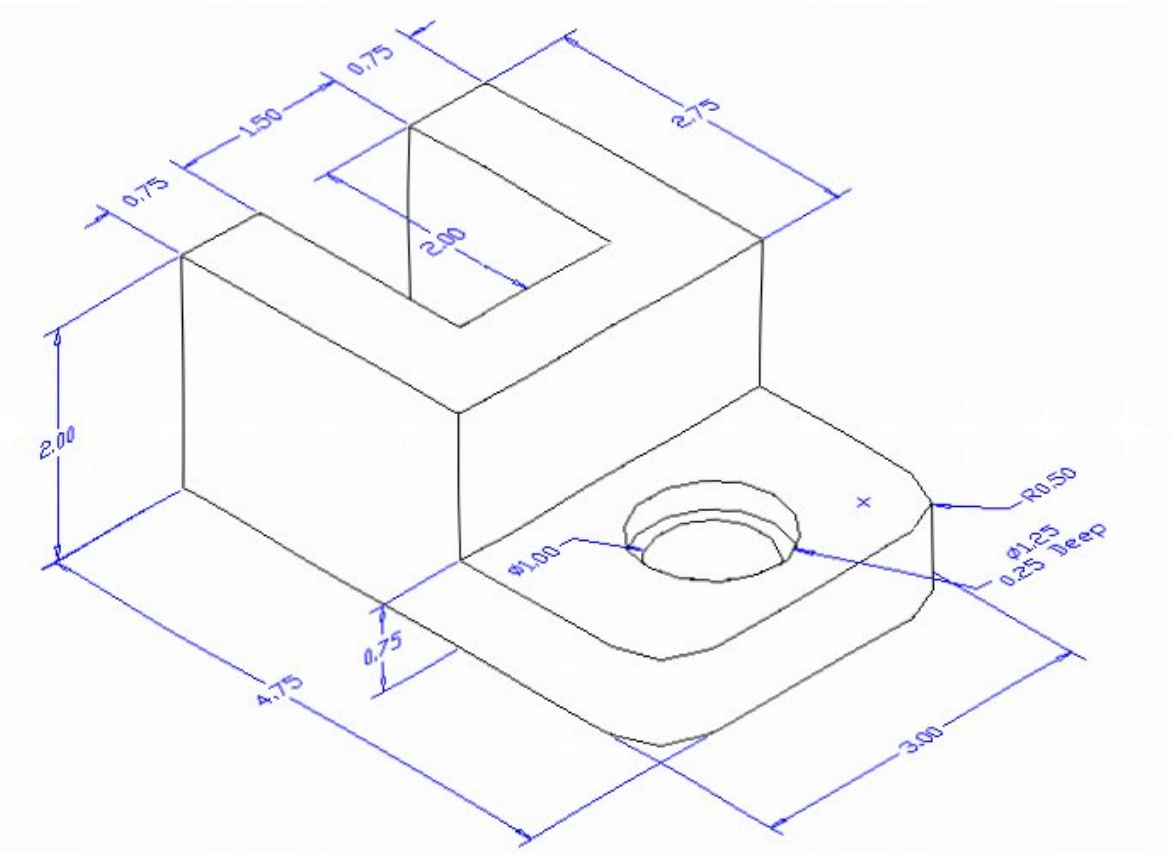
Figure 4.1 – Problem Three Sketch

When you receive a sketch from a professional, quickly identify the complex shapes and decide how to begin your drawing. Problem three has different heights, so you will create individual extrusions for each and then bring the solids together as one. This problem will help you address solid parts containing different complexities and thicknesses. 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

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You will draw Problem Three 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.
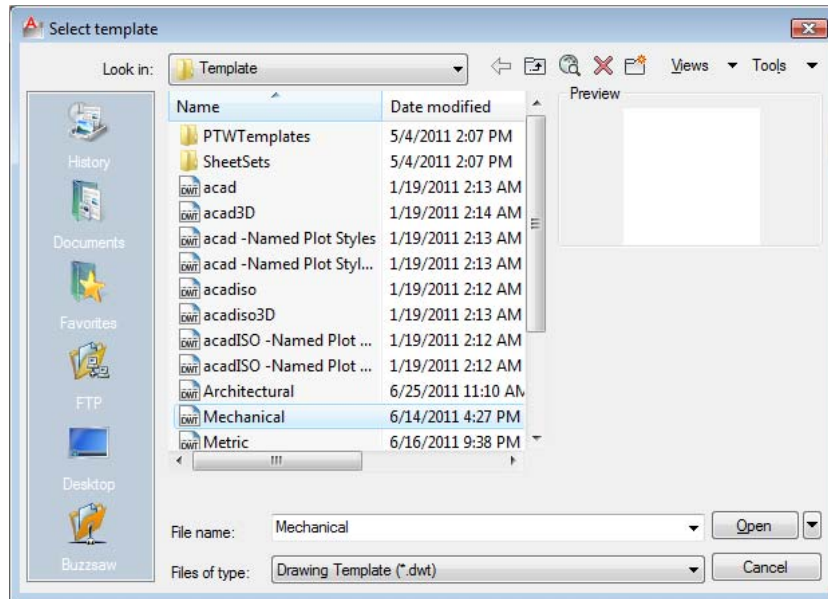

Figure 4.2 – Starting the drawing using the Mechanical template

# Drawing a Closed Polyline Shape to be Extruded

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Select the Polyline tool from the Draw panel under the Home tab and specify a start point in the lower 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 4.3.

```
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. Use the zoom tools on the navigation bar to enlarge the shape on your screen if necessary.
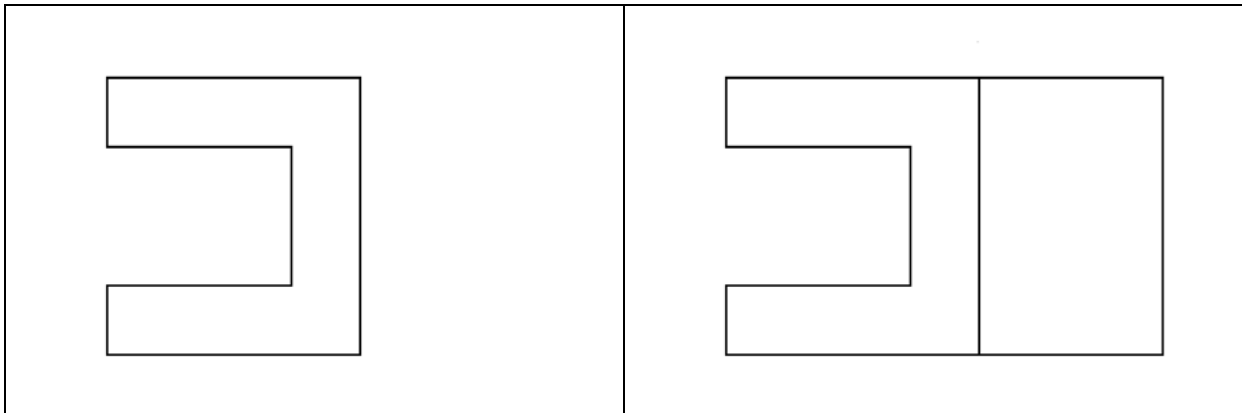
Figure 4.4 – The first closed polyline   **Figure 4.5 – The second closed polyline**

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", hit ENTER to repeat the Polyline command and select the lower 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.

Adding Circles to be Extruded

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Select the Center, Diameter command from the Circles tool on the Draw panel of the Home tab as shown in Figure 4.7. Select the midpoint Osnap on the far right side of the second polyline as the center of the circle. Type "**1**" when the command line prompts you to "specify diameter of circle". Repeat the Center, Radius command by hitting ENTER and place a **1.25** diameter circle outside the 1-diameter circle using the same centerpoint. You should have two concentric circles as shown in Figure 4.9.
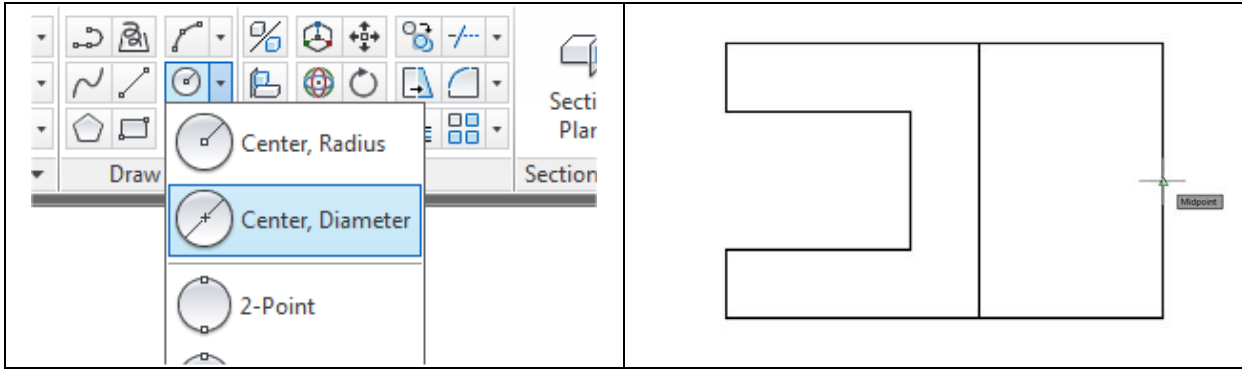
Figure 4.7 – Circle tool on the Draw panel   **Figure 4.8 – Midpoint on the Polyline**
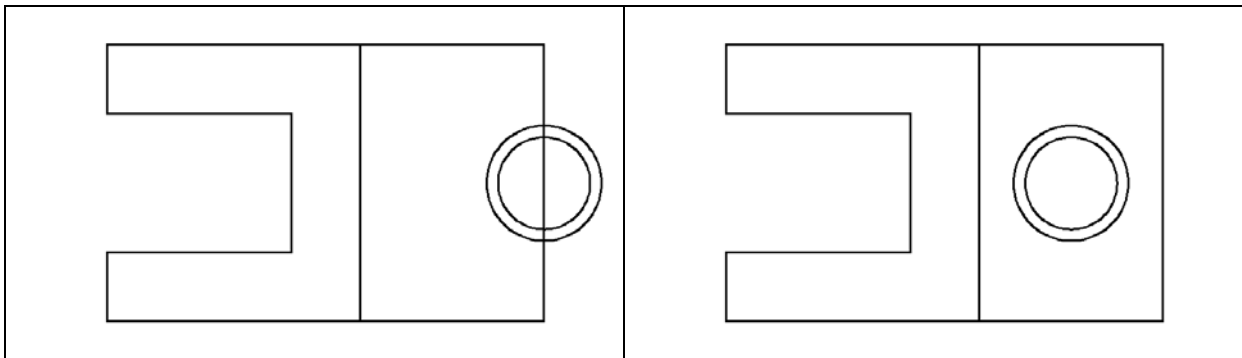

Figure 4.9 – Drawing two circles   **Figure 4.10 – Moving two circles**

Now you need to move the two circles one inch to the left, so select the Move tool on the Modify panel. When prompted to "**Select Objects**" place the pick box that now has replaced the aperture cursor on each circle and select both circles. The command line will respond with "**2 found**", so just hit ENTER to go the second part of the command. Pick your base point anywhere on the graphical display. Push the mouse to the left to direct the circles and type "**1**" and ENTER. Both circles will move 1 inch to the left as shown in Figure 4.10.

Rotating a 3D Solid Using the Rotate Command
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Select the Southwest Isometric View tool from the View panel under the Home tab to view the part from a 3D perspective (Figure 4.11). To rotate the solid on the Z axis, select the Rotate command on the Modify panel and select all entities on the screen. Hit ENTER to select the base point of rotation. Select any point at the base of the solid with the left mouse button and use the cursor with the Ortho mode on to rotate the objects, or type **-90** and ENTER for the degrees of counter clockwise rotation (Figure 4.12).
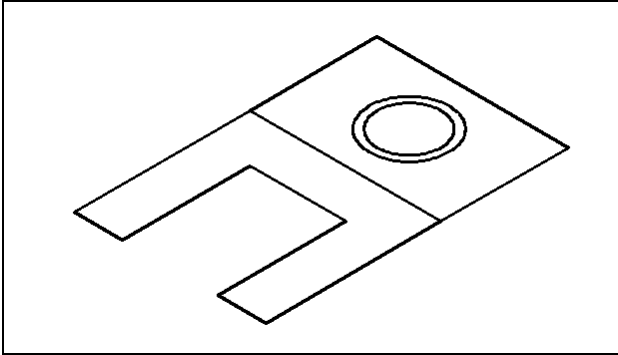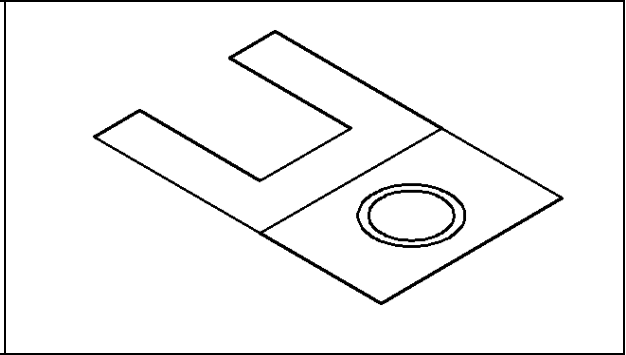
Figure 4.11 – Southwest Isometric View   **Figure 4.12 – Rotating 90 degrees**

Extruding a Solid from Closed Polylines and Circles

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Only closed entities like Polylines and Circles can be extruded into solids using this very powerful tool on the Modeling panel. If a there is even a minute space in outline of the entity, the program will consider it a line and then extrude the outline to a surface rather than a solid.
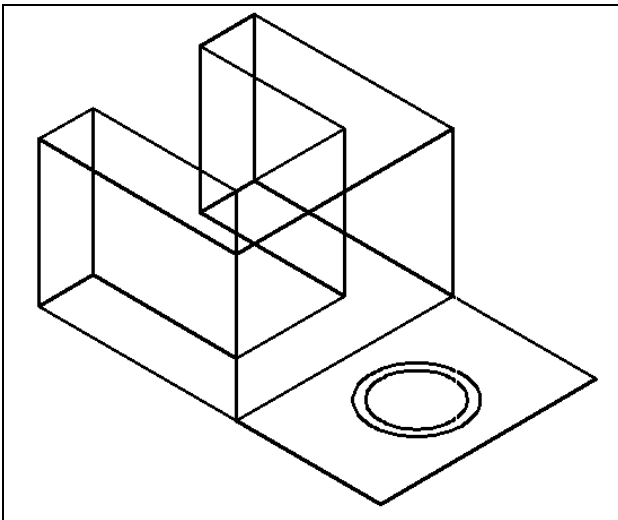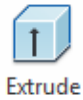

Select the Extrude tool on the Modeling panel and the command line will prompt you to "**select objects.**" Pick the closed polyline that looks like an inverted "C" anywhere on its outline and the command line will return with "**1 found**". Hit **ENTER** to specify the height or path of the solid. You can type "P" for Path and choose an entity to along which to extrude the shape, but you will not want to choose that option for problem three. Type "**2**" and **ENTER** for the height of the extrusion. The closed outline of the polyline will expand 2 units into the Z axis, creating a complex 3D solid as shown in Figure 4.13.

Figure 4.13 – Extruding the inverted "C"

Figure 4.14 – Extruding two solids
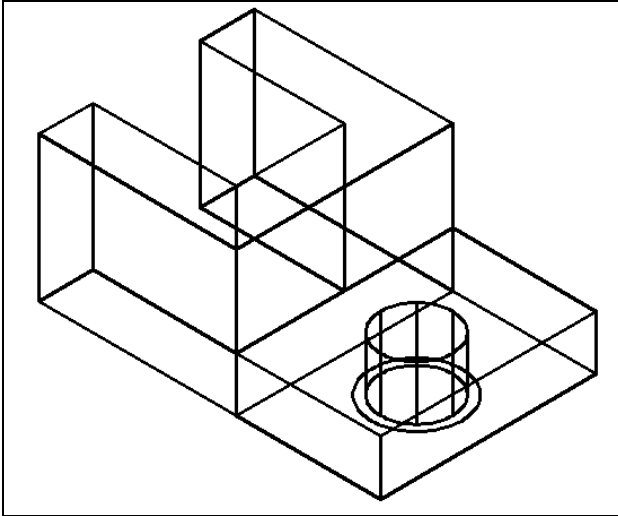
Hit ENTER to repeat the Extrude command. Pick the rectangular polyline and the smaller 1" diameter circle and the command line will return with "**2 found**". Hit **ENTER** to specify the height or path of the solid. Type "**0.75**" for the height of the extrusion and hit **ENTER**. A solid box and a solid cylinder will expand from the outlines of the two 2D entities as shown in Figure 4.14.

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 multiple solids with the same width.


Figure 4.15 – Extruding the second cylinder
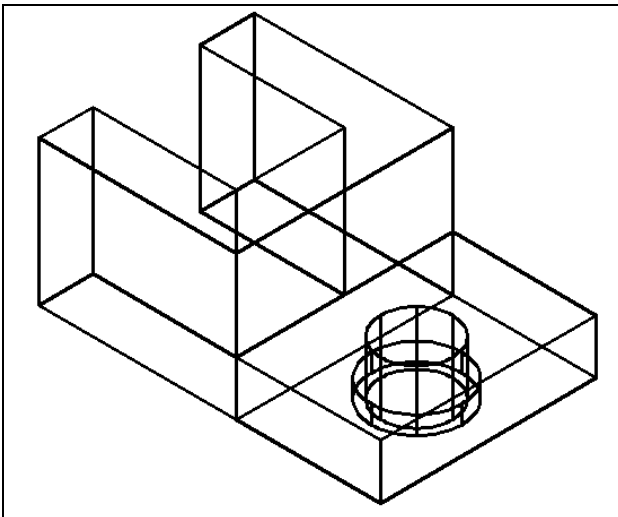
Hit ENTER once more to repeat the Extrude command. Pick the larger 1.25" diameter circle and the command line will return with "1 **found**". Hit **ENTER** to specify the height or path of the solid. Type "0**.25**" for the height of the extrusion and hit **ENTER**. A solid cylinder will rise from the original circle as shown in Figure 4.15. Now you need to move the 1.25 diameter cylinder into place as shown in your sketch.

Moving a Solid onto the Master Solid

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Remember that 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.

Figure 4.16 – Moving a cylinder
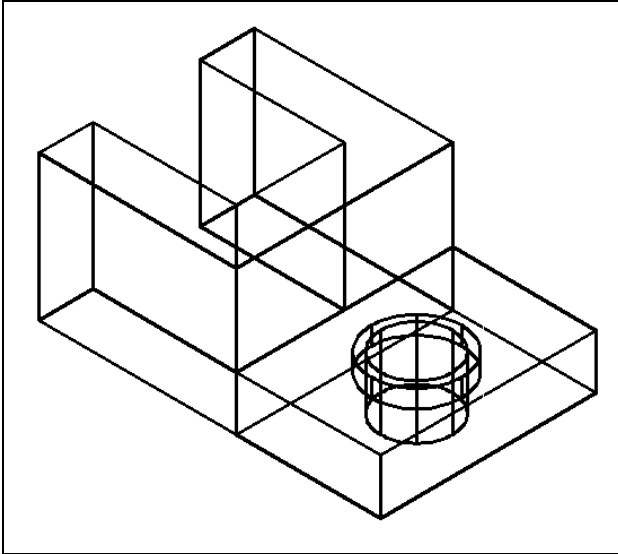
To move the 1.25 diameter cylinder, select the Move tool on the Modify panel. The command line will prompt you to "**Select Objects**". Pick the 1.25 cylinder and the command line will respond with "**1 found**", so hit ENTER to proceed to the second part of the command. Next, you need to "**specify base point or displacement**" on the graphical display. Pick the center Osnap of the top of the 1.25 cylinder as your base point. The "**second point of displacement**" is the center Osnap of the top of the 1.0 cylinder. Pick the second point and the 1.25 cylinder will move into place as shown in Figure 4.16.

Union of Two Solids

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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.

Figure 4.17 – Union two solids
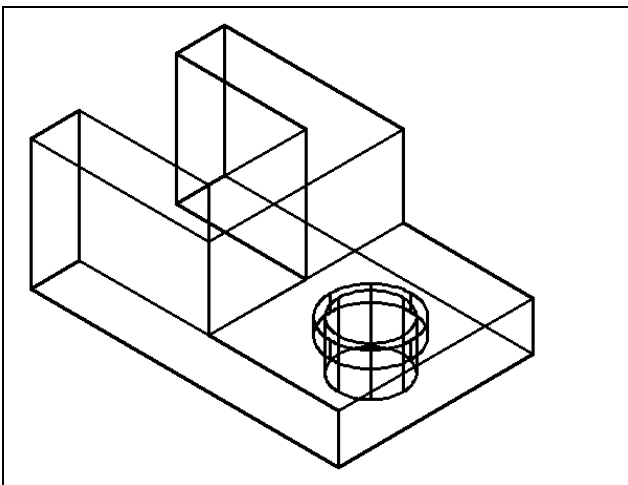
Select the Union tool on the Solid Editing panel under the Home tab. The command line will prompt you to "**Select objects**". Select the inverted "C" solid and the system will respond with "**1 found**". Also pick the rectangular solid so that the system responds with "**2 found**". Hit ENTER to combine both solids into one entity as shown in Figure 4.17. You can see that the tessellation lines outlining the connecting edges of both boxes are now missing. If you select the solid with one pick of the mouse the entire solid will become highlighted.
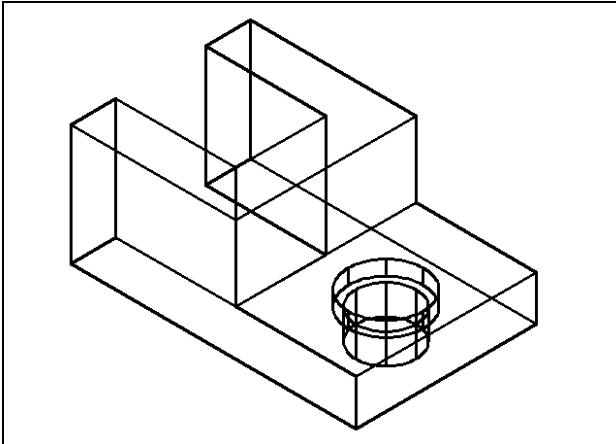
Subtracting a 3D Solid from Another Solid

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

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 two cylinders and the command line will respond with "**2 found**". Hit ENTER and the shape of the two cylinders will disappear from the master solid as shown in Figure 4.18.

Figure 4.18 – 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 selected both lines to identify the position of the radius. When placing a three dimensional curve on a solid, you just pick the edge that represents the corner where the two lines meet. In this case, the curvature is on the edge of a box in the Z-axis.
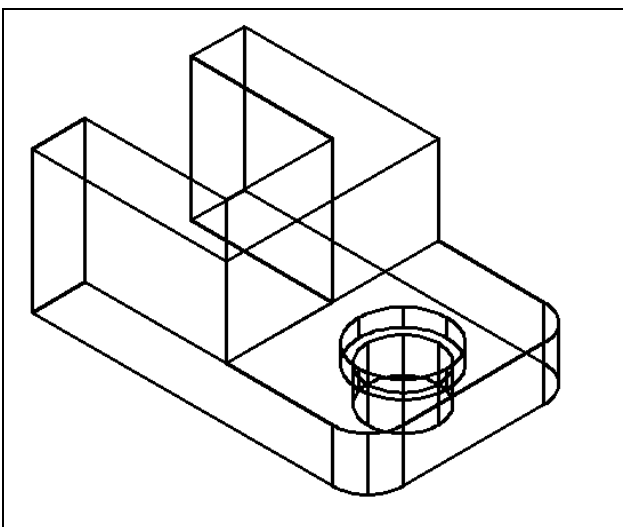
Select the Fillet tool on the Modify panel. Type "**R**" and **ENTER** for radius and then "**0.5**" and **ENTER** for the measurement, as well as "**M**" and **ENTER** to place multiple fillets. Pick the vertical line on the closest corner and hit **ENTER**; the line will become dashed. At the prompt, "**Enter fillet radius or [Expression] <0.5000>:**" just hit **ENTER** again to keep the default radius value you already specified. Now select the far right corner of the part so that both corner edges are dashed. Hit **ENTER** once more to place both fillets (Figure 4.19).

**Figure 4.19 – 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, 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 corner of solid problem three, 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 Three 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 4.21).
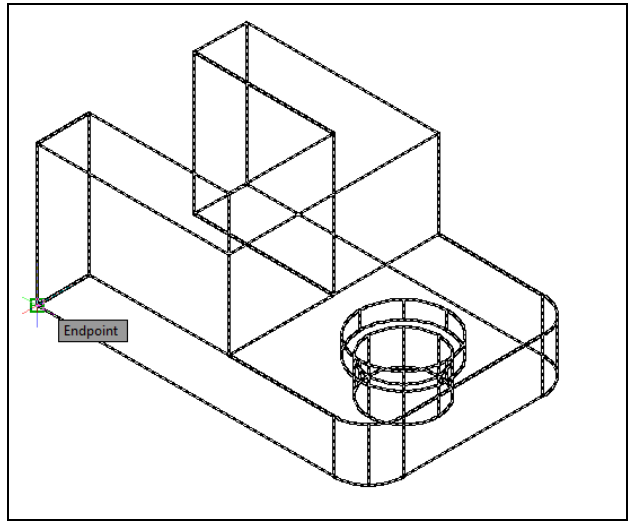
Figure 4.20 – The base point for moving the solid
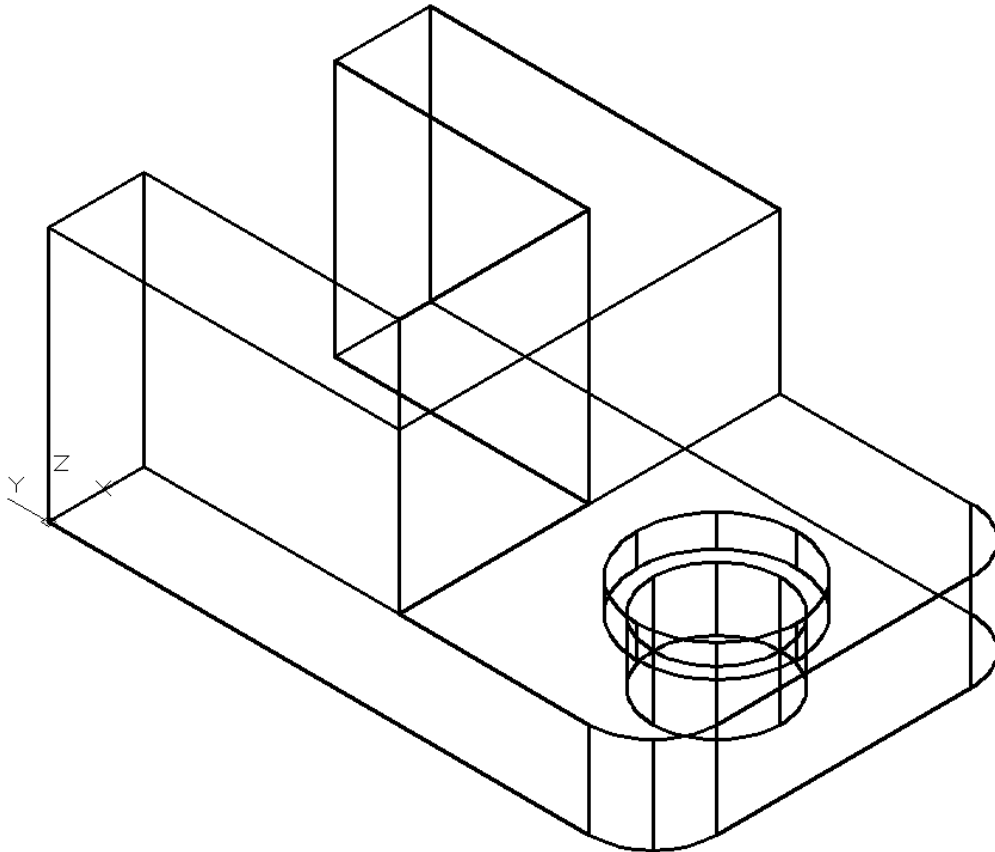
Figure 4.21 – Moving the finished solid to the origin

Saving the Solid Problem

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To save Problem 3 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 3" and hit the Save button to save the drawing (Figure 4.22).
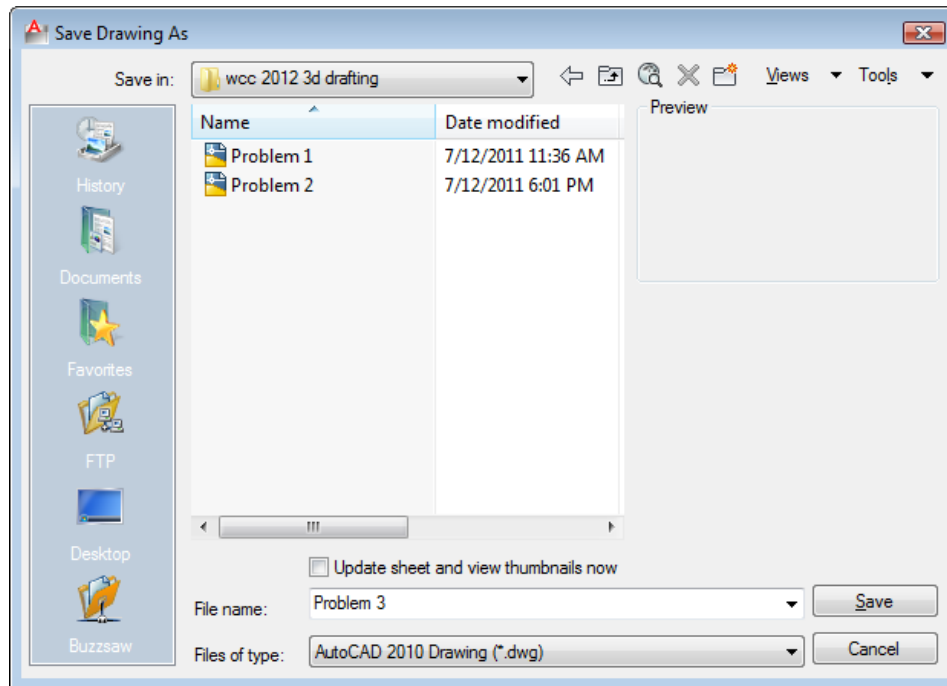

Figure 4.22 – Saving Problem Three

* 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 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.