

## Solid Problem Five

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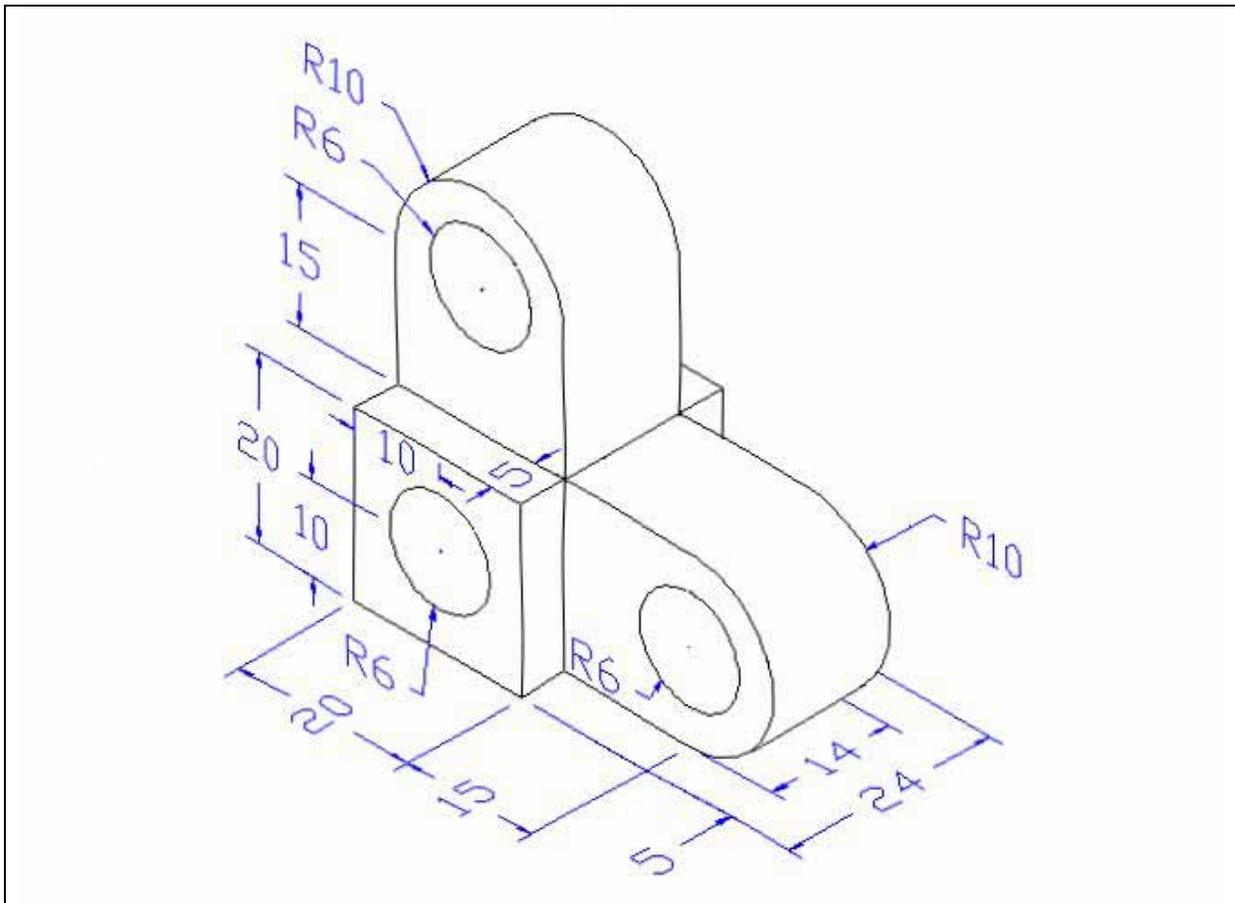
In this chapter, you will learn the following to World Class standards:

- Sketch of Solid Problem Five
- Starting a 3D Part Drawing
- Creating a New Layer in an AutoCAD Drawing
- Modifying How the UCS Icon is Displayed
- Drawing a Closed Polyline Shape to be Extruded
- Changing Layer to the Line Layer
- Constructing a Solid Perimeter Using the Line Command
- Adding Circles to the Detail
- Trim the Circles and Erase Extra Lines
- Freezing an AutoCAD Layer
- Creating Polylines Using the Edit Polyline Tool
- Extruding a Solid from Closed Polylines and Circles
- Moving Solid Adjacent to the Master Solid
- Union the Solids
- Subtracting a 3D Solid from Another Solid
- Shade the Solids
- Rotating a 3D Solid Using the Rotate3D and Rotate Commands
- Moving the Solid to the Origin Point
- Saving the Solid Problem

## Sketch of Solid Problem Five

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You start the fifth solids problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across one line which you can call the y-axis. In this exercise, a strategy you will continue to use is to draw closed Polylines that contain the most complex shape of the component and then use simple geometric solids to add or subtract details. In Problem Five, the solid has an “L” shape which is formed by three extruded Polylines with three holes. After extruding the Polylines and Circles, you will union the exterior shapes together and subtract the cylinders, as shown in your sketch in Figure 6.1.



**Figure 6.1 – Problem Five 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 solid exercise. You can see rather quickly that Problem Five has different thicknesses, so separate each extrusion or part that is made and bring them together. This problem will help you address solid parts where there are different complexities and thicknesses. All ten drills in the Fundamentals of 3D Drawing textbook will give you those repetitive maneuvers, which will make you a successful in the 3D modeling.

## Starting a 3D Part Drawing

In Problem Five, you will begin the 3D drawing by selecting the Metric template. In Figure 6.2, the window “Select Template” is loaded by selecting the New command on the AutoCAD Main Menu. 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.

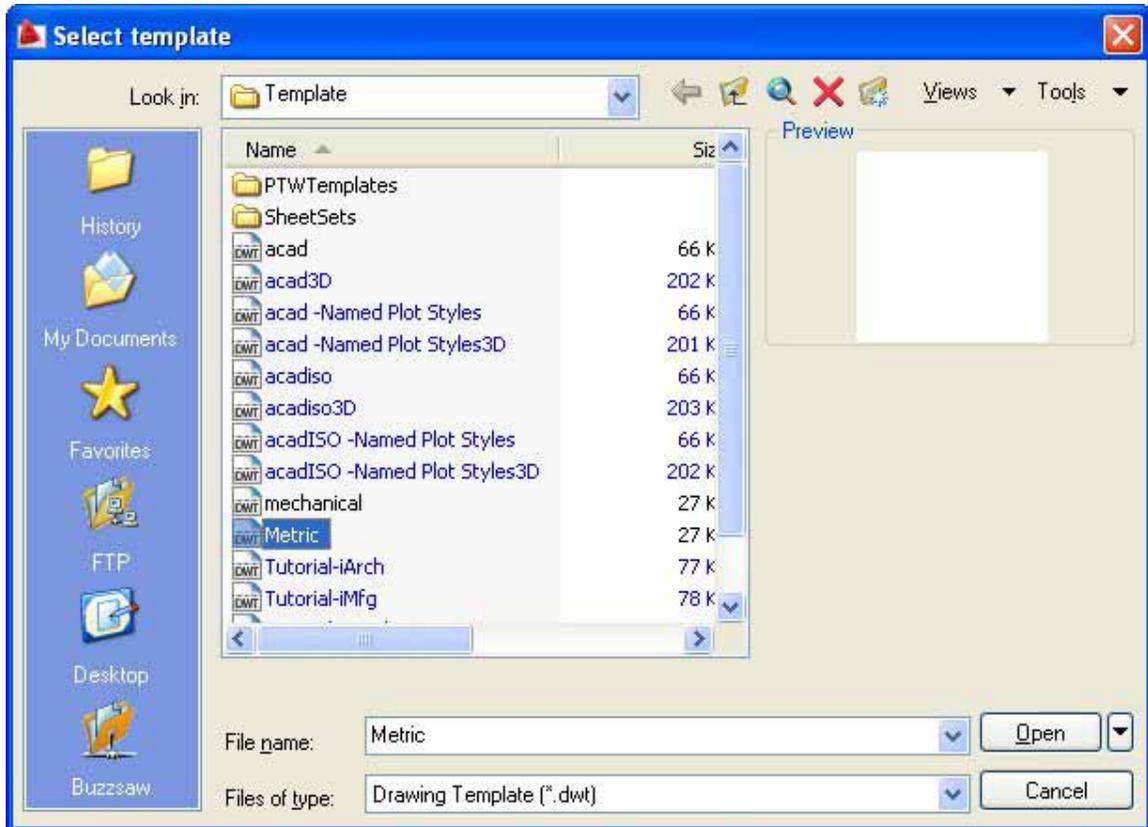


Figure 6.2 – Starting the Drawing Using the Metric Template

## 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.

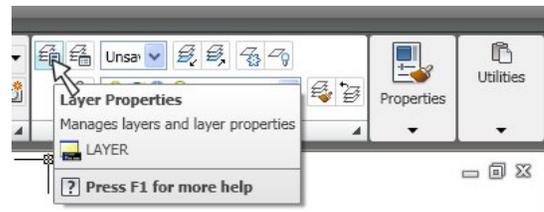


Figure 6.3 - The Layers Panel

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

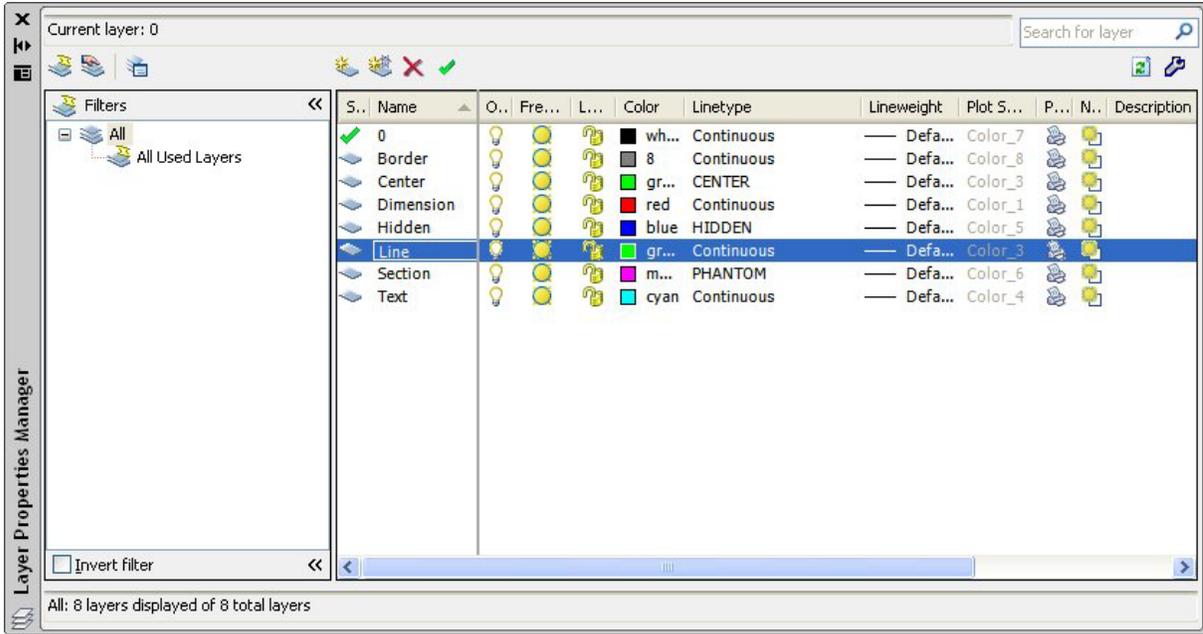
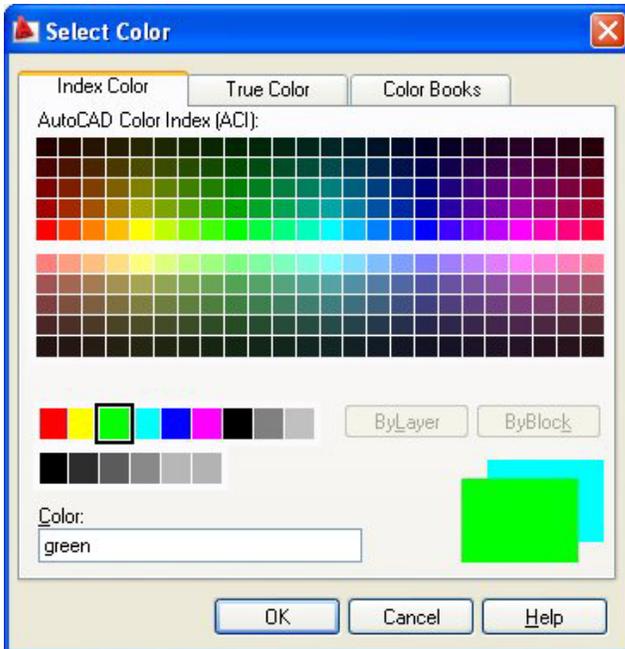


Figure 6.4 – Making a New “Line” Layer

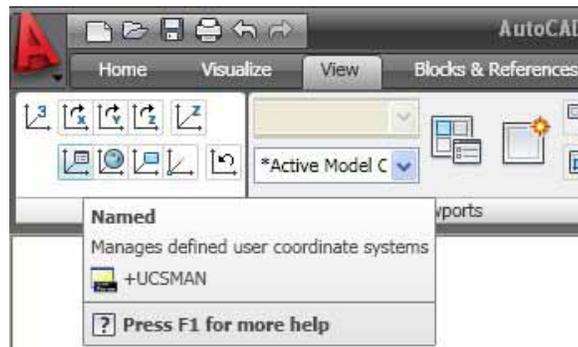


To change colors for the “Line” layer, place your mouse arrow over the color box in the Line layer row and single left click with your mouse. The Select Color window will appear in front of the Layers Properties Manager as shown in Figure 6.5. Select the green color on the Standard Colors Row from the Standards Colors row and select the OK button. You will now see that Line layer is green. You will draw some of the solids in this exercise on layer “0” and some will be on the layer “Line”. The important step in making separate layers to assist in creating a solid is using color to distinguish between them. For those who cannot see color, linetypes serve the same purpose.

Figure 6.5 – Making the “Line” Layer, Green

## Modifying How the UCS Icon is Displayed

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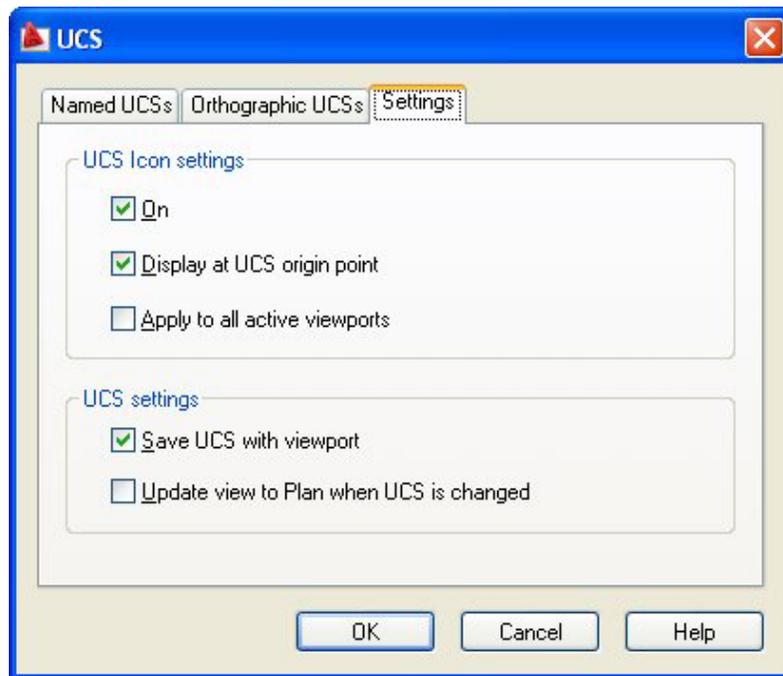


**Figure 6.6 - The UCS Settings**

When working in AutoCAD, you may notice the UCS icon moving on the graphical display showing the precise location of the origin. Many computer aided design operators do not appreciate having additional UCS lines in with their drawing or solid, so you can set the system to show the UCS in the lower left hand corner of the display.



Select the View tab and click on the Named tool on the Coordinates panel and the UCS window will appear on the graphical display. Pick the Settings tab to make the UCS Icon settings change. Remove the check from “Display at UCS origin point” as shown in Figure 6.7 and press the OK button to have the system variable change. Now you can start to create the solid.



**Figure 6.7 - The UCS Window, Settings Tab**

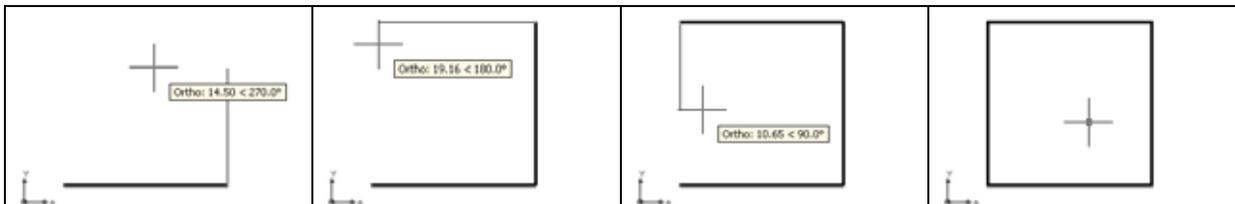
## Drawing a Closed Polyline Shape to be Extruded

On the Draw panel of the Home tab, select the Polyline tool and specify a start point in the lower left hand corner of the graphical display. In Figure 6.8, you can see the actual continuous line of data you need to specify at the command line. 

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.

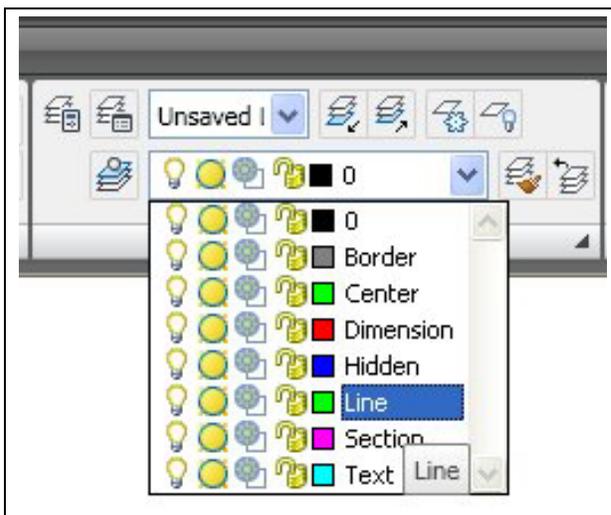
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Command: _pline  
Specify start point:  
Current line-width is 0.00  
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: <Ortho on> 20  
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 20  
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 20  
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: c
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**Figure 6.8 – The Polyline Text from the Command Line**



**Figure 6.9 – The First Closed Polyline**

## Changing Layer to the Line Layer



To change to the Line layer pick in the Layer Selection panel and select Line as shown in Figure 6.10. Whatever layer is shown in the window is the current layer. To the left of the layer name is a light bulb representing the layer on – off function, the yellow sun depicting the freeze – thaw function and the padlock portraying the lock – unlock function. 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, this window will become increasingly useful.

**Figure 6.10 – Choosing the Line Layer**

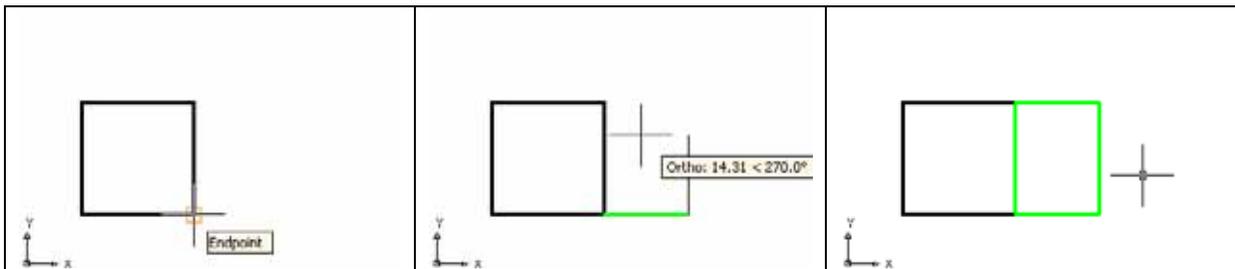
## Constructing a Solid Perimeter Using the Line Command

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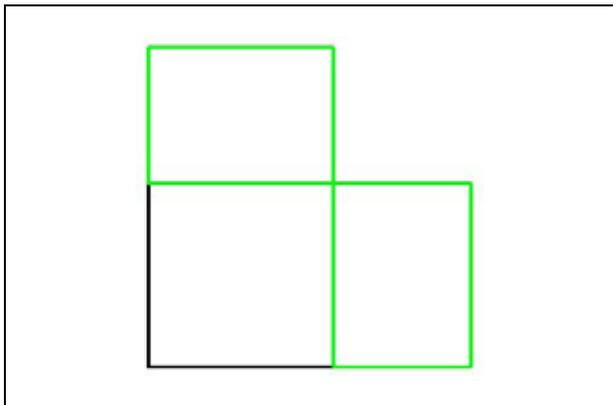
You need to assemble a 15 by 20 rectangle off the Polyline entity. Select the Line tool from the Draw panel. With your mouse, select the Endpoint at the lower right side of the Polyline box as shown in Figure 6.11. Even while in the Line command, continue with the Ortho mode On to keep the lines perfectly horizontal or vertical when drawing.



Move the line being drawn on the graphic screen to the right and type “15” and Enter at the keyboard. To draw the vertical line, move the mouse upward, dragging the line and type “20” and Enter. Drawing the third line, move the mouse upward, dragging the line and type “15” and Enter. To close the rectangle, just type “C” and Enter.



**Figure 6.11 – Drawing a Series of Lines Off the Polyline Box**



While in the Line command, continue with the Ortho mode (F8) on to keep the lines perfectly horizontal or vertical when drawing. Move the line being drawn on the graphic screen to the right and type “15” and Enter at the keyboard. To draw the vertical line, move the mouse upward, dragging the line and type “20” and Enter. Drawing the third line, move the mouse upward, dragging the line and type “15” and Enter. To close the rectangle, just type “C” and Enter.

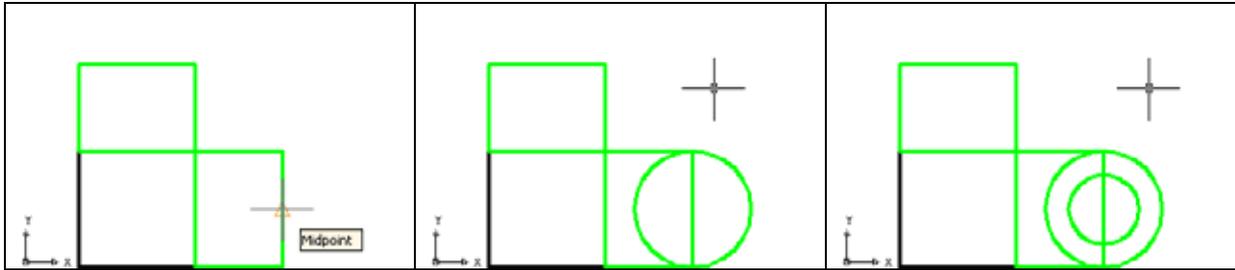
**Figure 6.12 –An Additional Series of Lines**

## Adding Circles to the Detail

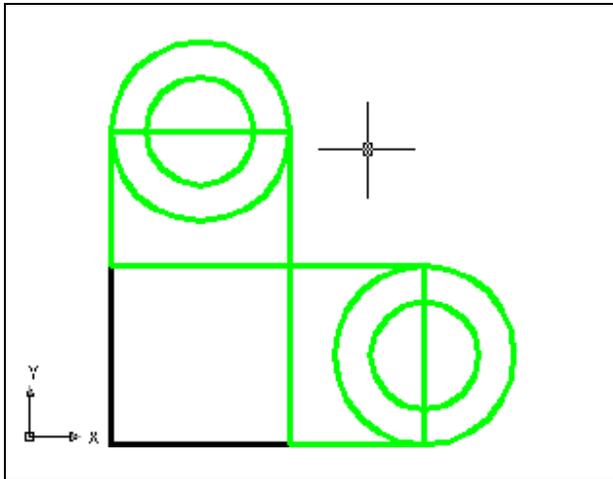
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Next, select the Center, Radius command on the Draw panel. Place the mouse aperture midway up the vertical line as shown in Figure 6.13 and a Midpoint Osnap symbol (triangle) will appear in the middle of the line. Select the center point of the circle by selecting with the left mouse button. Then type “10” when the command line prompts you to “specify radius of circle”. Repeat the Circle command by pressing Enter, and place the 6 radius circle inside the 10-diameter circle having them sharing the same center points. There will be two concentric circles as shown in Figure 6.13.



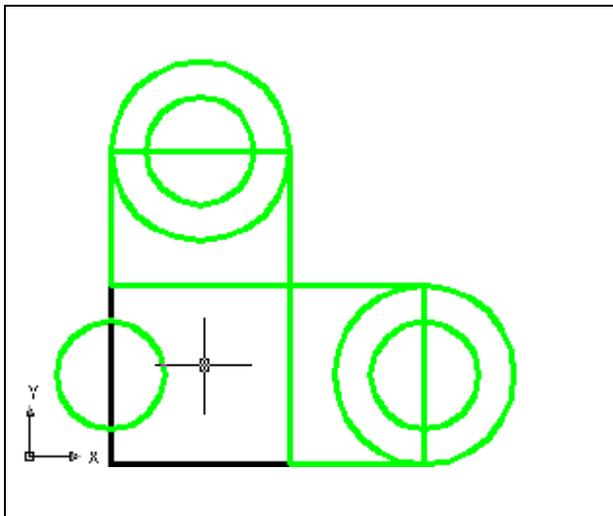


**Figure 6.13 –Adding Circles For The Solid**



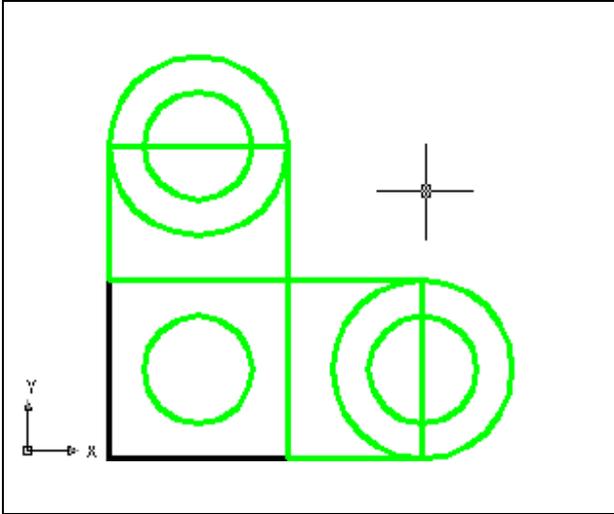
Next, press Enter to repeat the Circle, Radius command. Place the mouse aperture on the center of the horizontal line and a Midpoint Osnap symbol (triangle) will appear in the middle of the line. Select the center point of the circle by selecting with the left mouse button. Then type “10” when the command line prompts you to “specify radius of circle”. Repeat the Center, Radius command by pressing Enter, and place the 6 radius circle inside the 10-diameter circle having them sharing the same center points. Two concentric circles are shown in Figure 6.14.

**Figure 6.14 –Adding Circles For The Solid**



Again, push Enter to repeat the Center, Radius command. Place the mouse aperture midway up the black vertical line 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. Then type “6” when the command line prompts you to “specify radius of circle”. The radius 6 circle will appear on the midpoint of the line as shown in Figure 6.15. Now 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 6.15 –Adding Circles For The Solid**

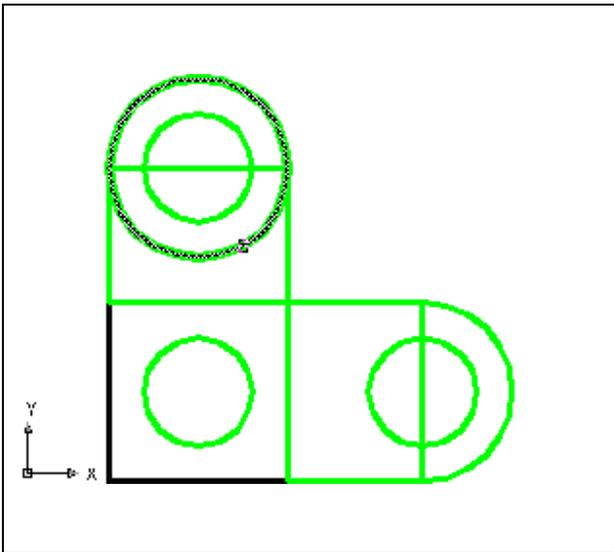


To move the circle, you select the Move tool on the Modify panel. Once you choose the Move command, the prompt will be to “Select Objects”. Pick the circle and the command line will respond with “1 found”, so just press Enter to go the second part of the command. Next, you need to “specify base point or displacement” on the graphical display. Pick your base point anywhere on the display. Type Enter to proceed to the “second point of displacement” as input @10,0 which will move the entity a distance of 10 in the X-axis, 0 in the Y-axis and when leaving the Z coordinate off, the default is 0 in the Z-axis.

**Figure 6.16 –Adding Circles For The Solid**

## Trim the Circles and Erase Extra Lines

The next step is to remove the inside arc segment of the larger circle to create a box with a half circle attached, then erase the line passing through the smaller circle. By accomplishing both of these sequences, you will be ready to join the individual entities into a special Polyline.



Select the Trim command on the Modify panel and Enter to automatically select all lines on the graphical screen as cutting lines. Then proceed to pick the radius 10 circle segment on the right as shown in Figure 6.17. You will notice the system removes the arc slice that was between the two parallel lines. Continue to remove the other arc segment on the radius 10 circle to the top as shown in Figure 6.17. Select the Erase tool on the Modify panel, and select the line entities passing through the small holes and hit Enter. This will remove the two lines as shown in Figure 6.18. Now you need to move a single entity to a different layer.

**Figure 6.17 –Trim Two Circles**

Select the small green circle inside the 20 by 20 Polygon box. In the Layer Selection window on the Object Properties toolbar that is presently displaying layer “Line”, select the 0 layer. You will notice the selected entities will turn black, which the color that was assigned to the 0 layer. Press the “ESC” key to remove the entities from the selection set.

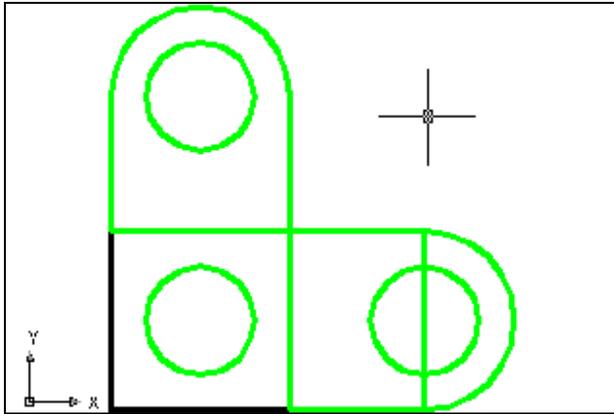


Figure 6.18 –Erase The First Line

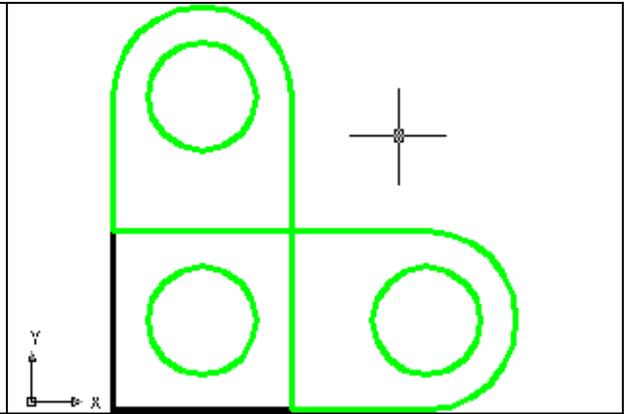


Figure 6.19 –Erase Two Lines

## Freezing an AutoCAD Layer

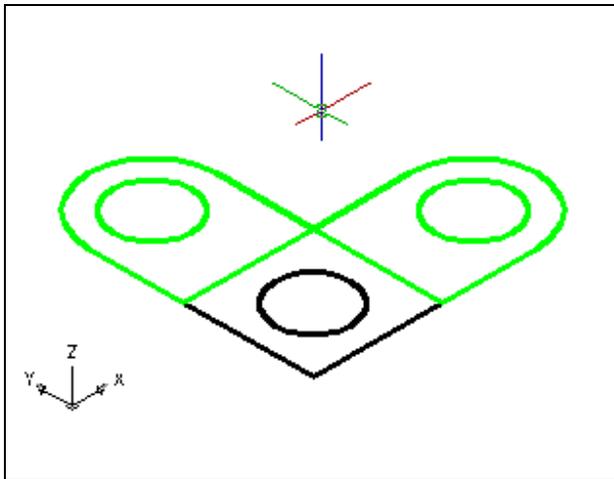


Figure 6.20 – View in SW Isometric View

On the View panel, 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 6.20)

To make the Polyline box and circle on Layer 0 invisible, you need to use the Freeze Layer option. Pick in the Layer panel the Layer Selection drop-down arrow to freeze Layer 0 as shown in Figure 6.20. The Polygon Box and circle will disappear as shown in Figure 6.22.

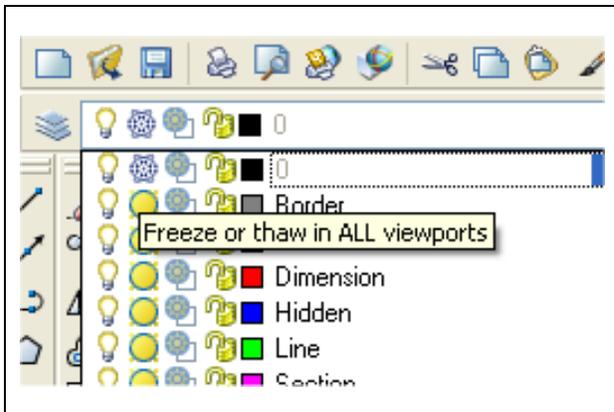


Figure 6.21 – Freeze the 0 Layer

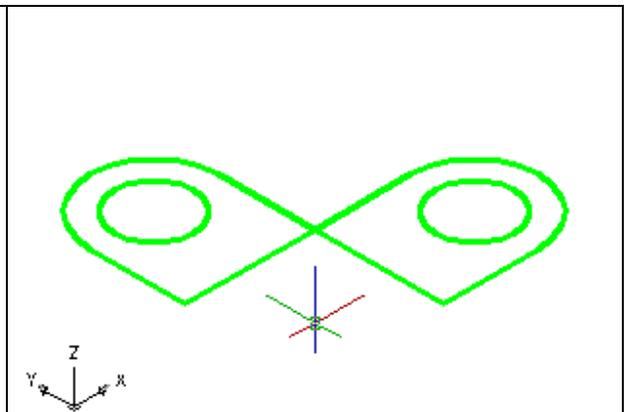
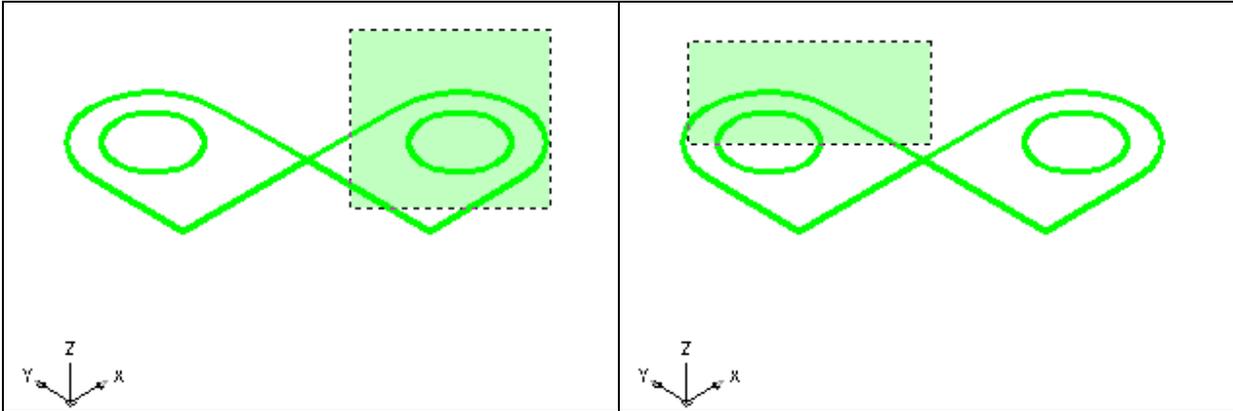


Figure 6.22 –The Line Layer

## Creating Polylines Using the Edit Polyline Tool

To merge the three lines and the arc segment together you need to use a new command, which is the Edit Polyline tool. 



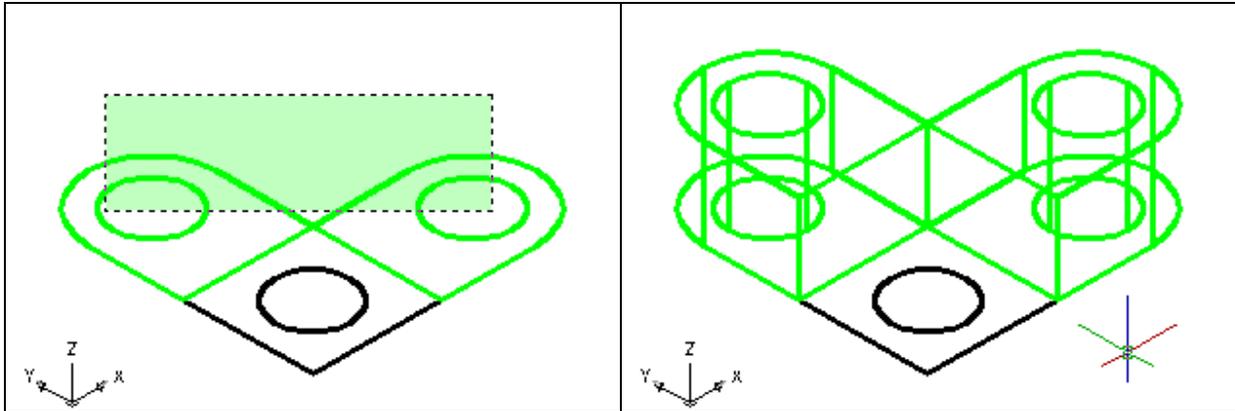
**Figure 6.23 – Polyline Edit the Right Side**      **Figure 6.24 – Polyline Edit the Left Side**

Select the Edit Polyline tool on the drop-down arrow of the Modify panel and the command line on the AutoCAD application will prompt you to “**Select polyline or [Multiple]:**” Type “M” for multiple, and then pick to the upper right of the graphical display as shown in Figure 6.23. Going to the left, you will see the selection window signifying the “crossing” option in selecting entities. Pick again to make the crossing selection. The command line will return with “Do you wish to turn the line into a Polyline <y>”. Hit Enter to convert the line to a Polyline and at the next prompt, type “J” for join. Hit Enter again and all adjoining lines and arcs will become a Closed Polyline. The small radius 6 circle will remain a separate entity.

Again, select the Edit Polyline tool on the Modify panel and the command line on the AutoCAD application will prompt you to “**Select polyline or [Multiple]:**” Type “M” for multiple, and then pick the upper middle of the graphical display as shown in Figure 6.24 and going to the left, you will again see the selection window signifying the “crossing” option in selecting entities. Pick again to make the crossing selection. The command line will return with “Do you wish to turn the line into a Polyline <y>”. Hit Enter to convert a line to a Polyline and at the next prompt, type “J” and Enter for join. All adjoining lines and arcs will become a Closed Polyline. You can now unfreeze, or thaw, the 0 layer.

## Extruding a Solid from Closed Polylines and Circles

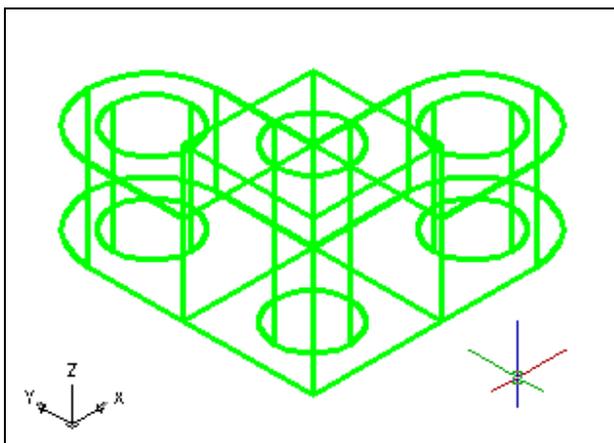
Only closed entities like Polylines and Circles can be extruded using this very powerful tool on the Modeling panel. You can extrude multiple Polylines and Circles at a time with the Extrude command. 



**Figure 6.25 – Selecting to Extrude**

**Figure 6.26 – Extrude the Entity**

Select the Extrude tool on the Modeling panel and the command line on the AutoCAD application will prompt you to “**select objects**”. Pick the upper right of the graphical display as shown in Figure 6.25 and going to the left, you will see the selection window signifying the “crossing” option in selecting entities. The command line will return with “**4 found**”. Press **Enter** to specify the height or path of the solid. Type “**14**” for the height of the extrusion and for the angle of taper, hit **Enter** to accept the zero degree default. A more complex 3D solid will appear in the graphical display as shown in Figure 6.26.



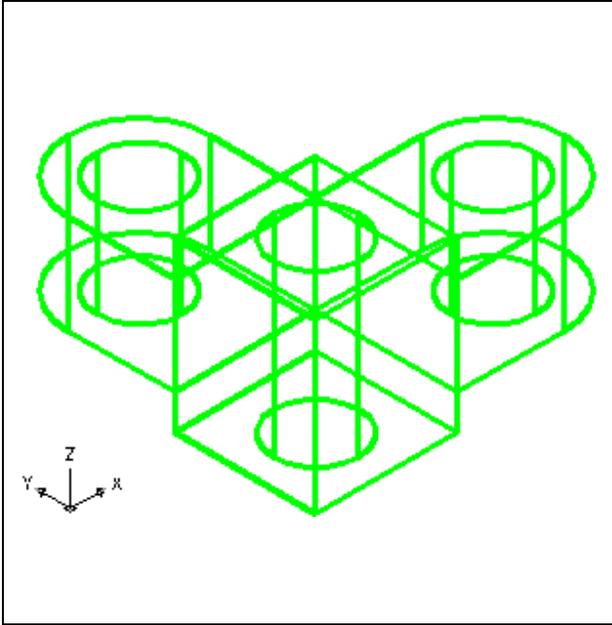
For the next set of solids, choose the Extrude tool on the Modeling panel and the command line will prompt you to “**select objects**”. Pick the Closed Polyline and the circle inside the box. The command line will return with “**2 found**”. Press **Enter** to specify the height or path of the solid. Type “**24**” 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 6.27.

**Figure 6.27 – Extrude 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.



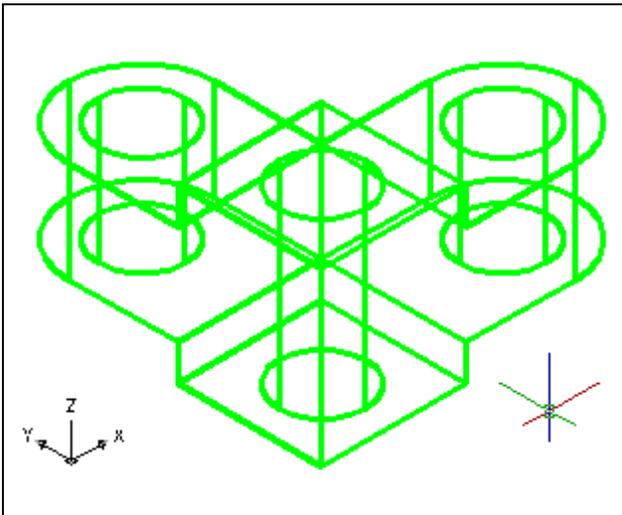


To move the outside solids, you select the Move tool on the Modify panel. Once you choose the Move command, the prompt will be “Select Objects”. Pick the wings, both the exterior solids and cylinders and the command line will respond with “**4 found**”, so just press Enter to go the second part of the command. Next, you need to “specify base point or displacement” on the graphical display. Pick your base point anywhere on the display. Type the “second point of displacement” as **@0,0,5** moving the entities zero distance in the X and Y-axis and 5 in the Z-axis cylinder will move as shown in Figure 6.28. You can check your work by viewing the “step up” appearance of the solids adjacent to the middle.

**Figure 6.28 – Moving the Outside Wings**

## Union the Solids

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



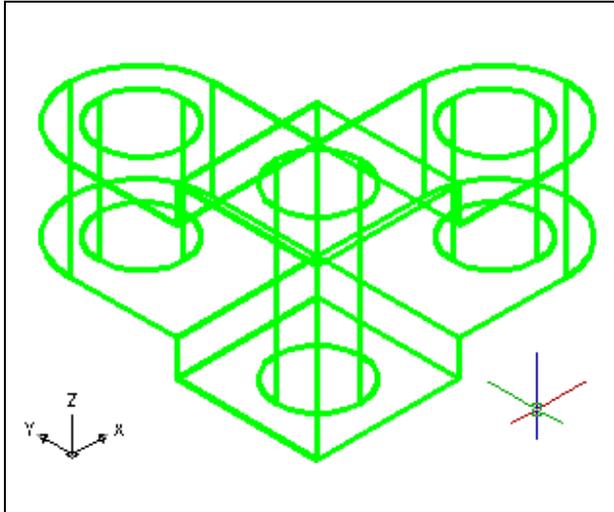
To add three solid to another one, select the Union tool on the Solid Editing panel. Then the AutoCAD program will prompt you with “Select objects”. Select all the perimeter solids and none of the cylinders and the system will respond with “**3 found**”. Hit Enter and now all the exterior solids become as one entity as shown in Figure 6.29. You can see in the graphical display that the tessellation lines outlining all the adjoining entities that were crossing each other are now missing. Select the solid with one pick of the mouse and the entire solid, minus the cylinders, will become highlighted.

**Figure 6.29 – Union the Solids**

## Subtracting a 3D Solid from Another Solid

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



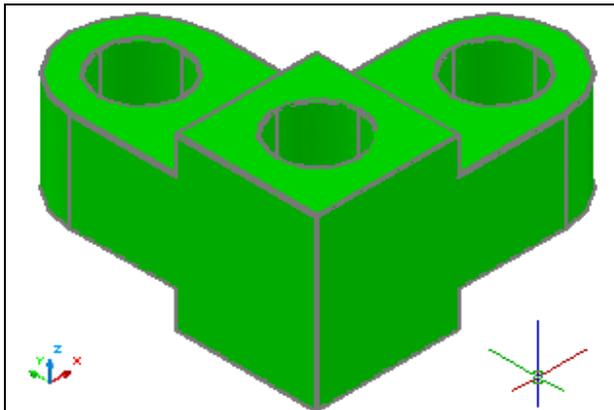
You will be prompted to “**Select solids and regions to subtract from**”. Select the larger master solid and the system will respond with “**1 found**”. Hit Enter to proceed to the next step, which is “**Select solids and regions to subtract**”. Pick the three cylinders and the software application will counter with “**3 found**”. Hit Enter and a region exactly the shape of the two cylinders will disappear from the larger one 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 – Subtract the Three Cylinders

## Shade the 3D Solid

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To view the 3D solid with holes in a better means, select Realistic Visual Style on 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 since they can visualize the product clearly. If you ever desire to return to standard viewing, choose the 2D Wireframe tool on the Visual Styles toolbar to return the drawing to the previous settings.

Figure 6.31 – Shade the Solid

## Rotating a 3D Solid Using the Rotate3D and Rotate Commands

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Select 3D Rotate tool from the Modify panel. At the command, “select objects”, pick the solid on the graphical display and **Enter** to proceed to the second part of the function. You are going to rotate the solid part on the x-axis, so pick any point on the display and then, with the Ortho mode still “on”, pick a second point drawing a line on the x-axis. Type **-90** to rotate the solid as shown in Figure 6.32.

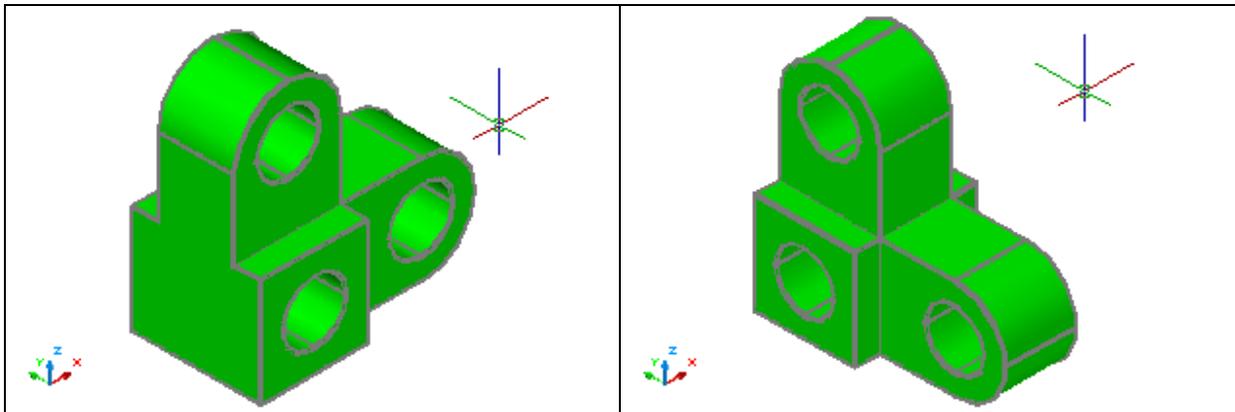


Figure 6.32 – Rotate3D the Solid

Figure 6.33 – Rotate the Solid

To rotate the solid in the XY plane, select the regular Rotate command on the Modify panel. Select the Rotate command on the Modify panel 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 6.33).

## Moving the Solid to the Origin Point

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To move Problem Five to the drawing origin of 0,0,0 on the x, y and z axis, pick the Move tool on the Modify panel and select the 3D solid and Enter. At the prompt, “**specify base point or displacement**”, select the endpoint shown in Figure 6.34 at the upper left hand side of the model, which will become the Problem Five insertion point into an assembly drawing. At the “second point of displacement”, type 0,0,0 and Enter. Problem Five will move to the new origin point.

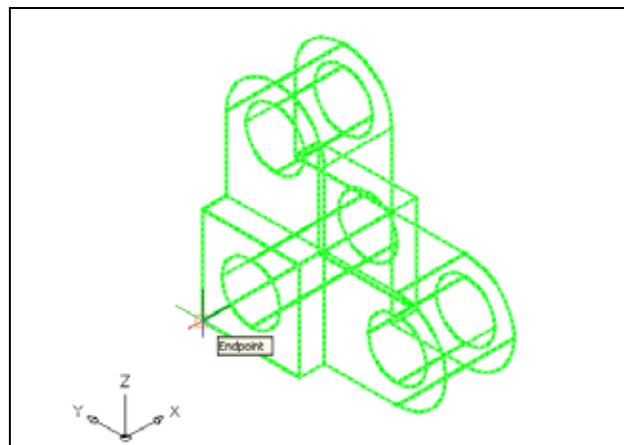


Figure 6.34 –Finished Solid at the Origin

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.

## Saving the Solid Problem

To save Problem 5 in your Fundamentals of 3D Drawing Folder, select the Save 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 press the Save button to save the drawing. (See Figure 6.35)

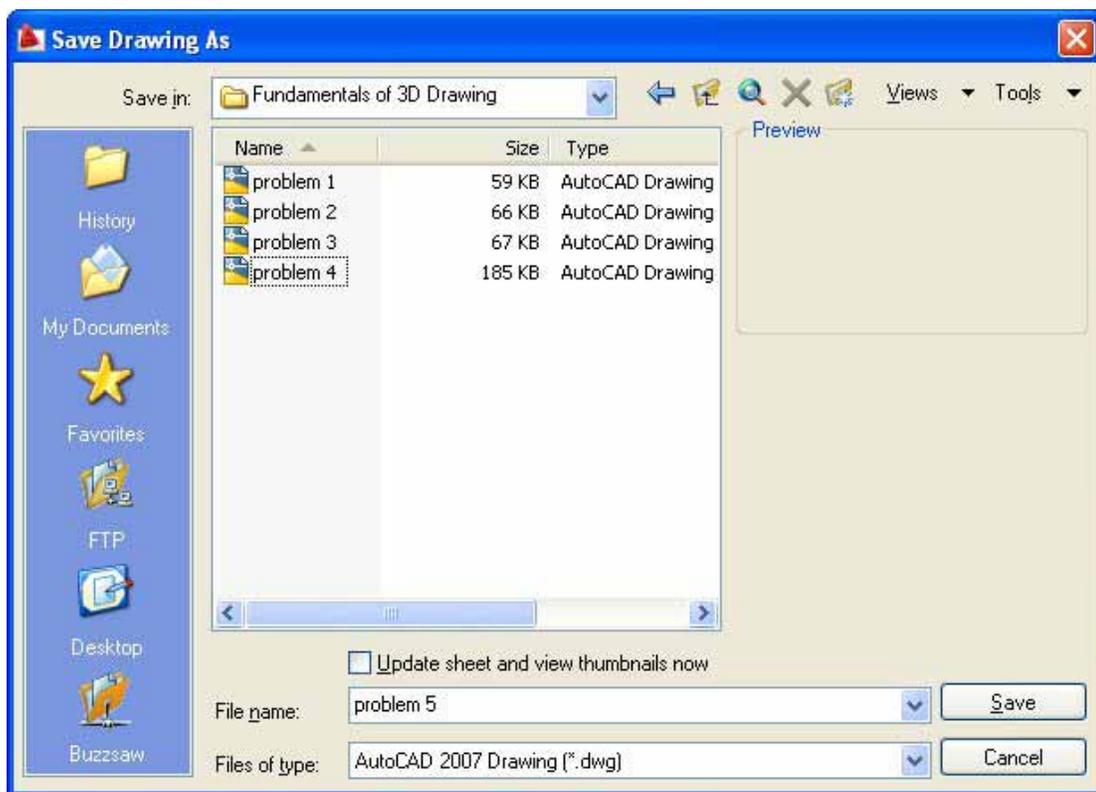


Figure 6.35 – Saving Problem Five

**\* 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 under 5 minutes to maintain your World Class ranking.**

**\* World Class CAD Challenge \* - Report your best times to World Class CAD at [www.worldclasscad.com](http://www.worldclasscad.com) to obtain your world class ranking.**

