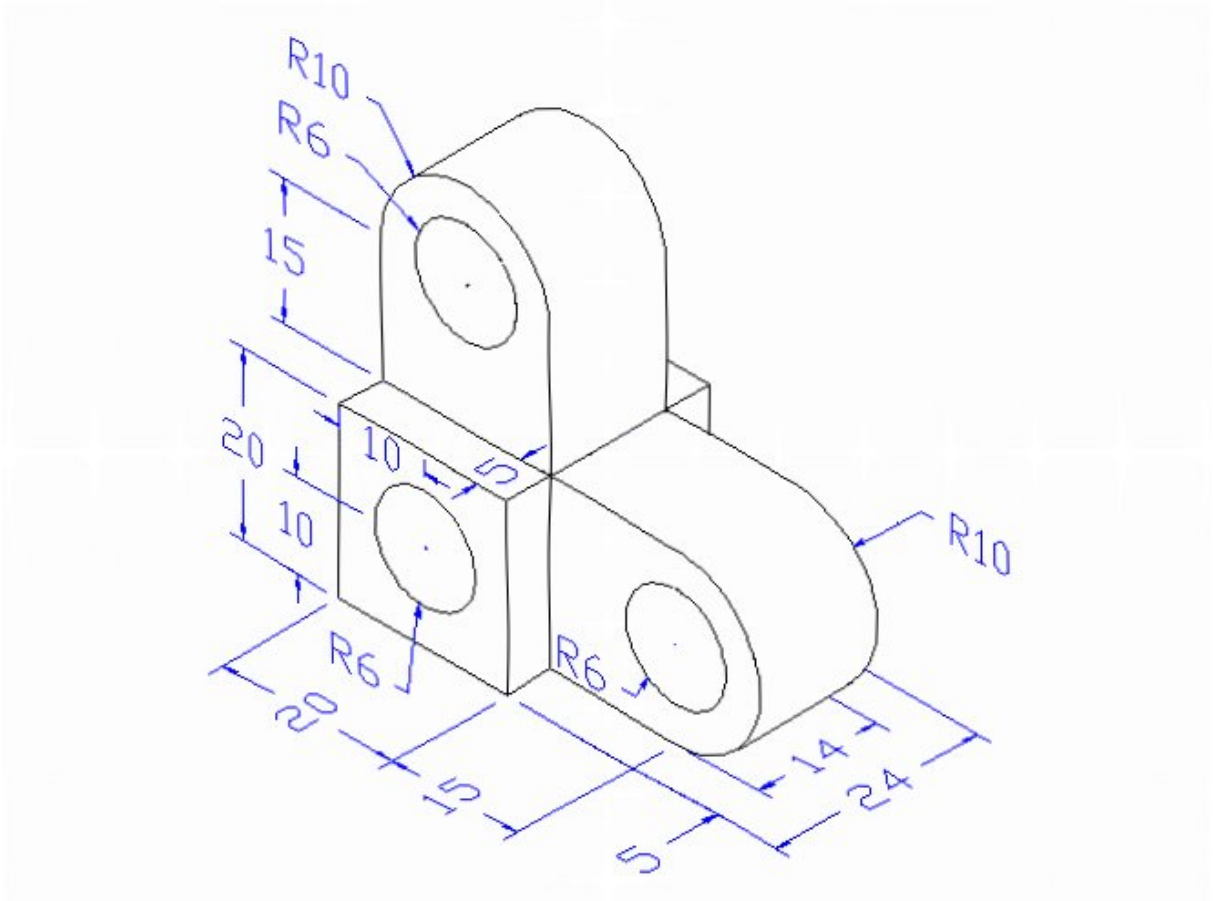# C h a p t e r

6

Solid Problem Five

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

1. Sketch of Solid Problem Five
2. Starting a 3D Part Drawing
3. Creating a New Layer in progeCAD
4. Modifying How the UCS Icon is Displayed
5. Drawing a Closed Polyline Shape to be Extruded
6. Changing to the Line Layer
7. Constructing a Solid Perimeter Using the Line Command
8. Adding Circles to the Detail
9. Trimming Circles and Erasing Extra Lines
10. Freezing Layers
11. Creating Polylines Using the Edit Polyline Tool
12. Extruding a Solid from Closed Polylines and Circles
13. Moving Solids Adjacent to the Master Solid
14. Combining Solids using the Union tool
15. Subtracting a 3D Solid from Another Solid
16. Shading Solids
17. Rotating a 3D Solid Using the Rotate3D and Rotate Commands
18. Moving the Solid to the Origin Point
19. Saving the Solid Problem

Sketch of Solid Problem Five

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You will start the fifth solid 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 contains the most complex shape of the component and then use simple geometric solids to add or subtract detail. In Problem Five, the solid contains a rectangular detail that will be turned into 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, combining the two shapes together using Union, and subtracting the two circles, you will fillet a curve in two places 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 heights, so separate each extrusion or part and 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 teach you those repetitive maneuvers, which will make you a successful in the 3D modeling.

Starting a 3D Part Drawing

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In Problem Four, 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 (Figure 6.2). 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.
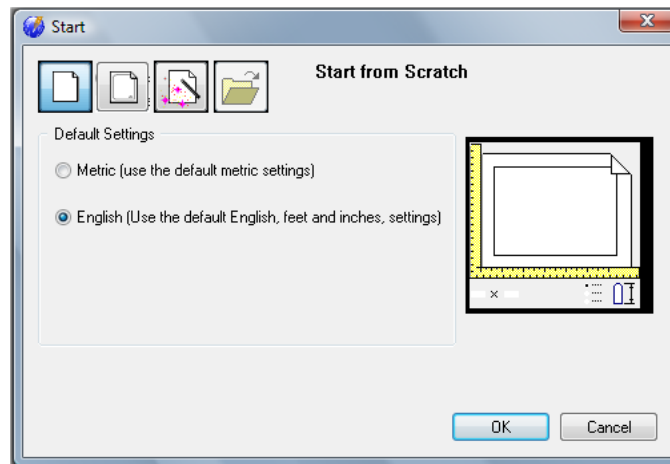

Figure 6.2 – Starting the Drawing Using the Metric Template

Creating a New Layer in a progeCAD Drawing

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Figure 6.3 - The Entity Properties Toolbar

Unfortunately, the templates that you created in your 2D training with progeCAD Smart! are not compatible with progeCAD professional. Until now, you have been able to draw four solid problems without the aid of extra layers. In Solid 5, however, you will need to use an extra layer in order to perform a step in designing the problem. Before you begin drawing, you will need to create a set of layers; you can save the file as a template, that way you can retrieve the levels at any time.

Select the Explore Layers tool from the Entity Properties toolbar and the Layer Properties Manager will appear. Use the New Layer button to add the following layers: Border, Center, Dimension, Hidden, Line, Section, and Text. Change the colors and linetypes of each layer so that the Layer Properties Manager appears as shown in Figure 6.4. Hit and Apply and Ok. Now you can save the file as a template, using the Save As tool from File on the Menu Bar. Change the "Save as type" to Drawing Template (Figure 6.6). You can change the settings and defaults of the template at any time.
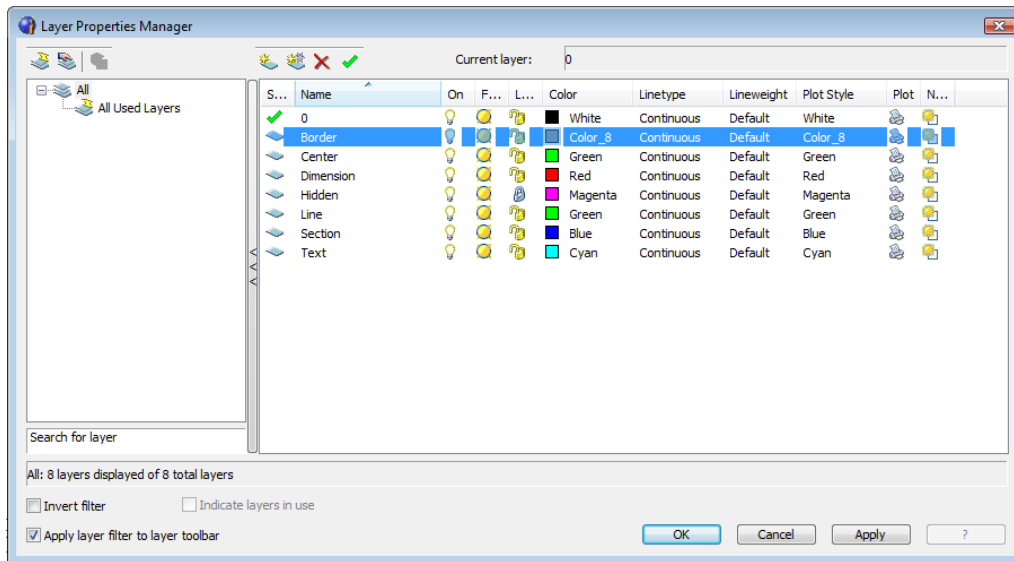
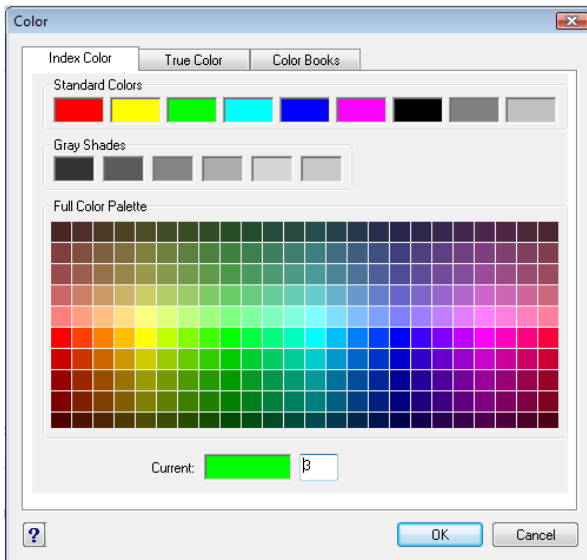**Figure 6.4 - Adding New Layers**
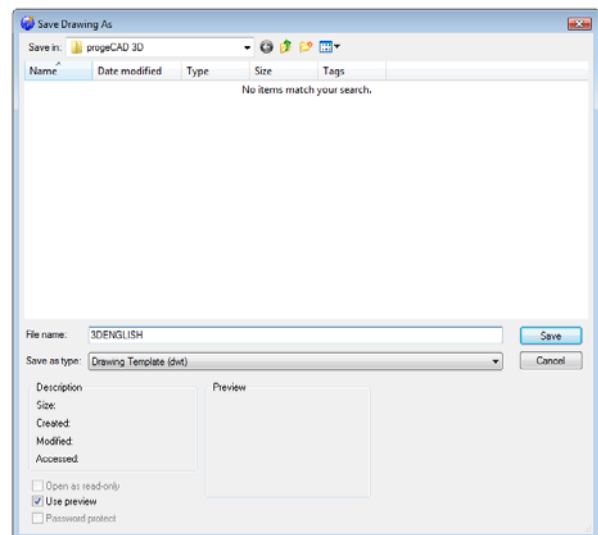

Figure 6.5 - Changing a Layer Color

Figure 6.6 - Saving the Drawing Template

Modifying How the UCS Icon is Displayed
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When working in progeCAD, you may notice that the UCS icon has left its position in the corner of the display. This is because the default setting places the UCS at the origin of model space if the origin is in view. Many computer aided design operators do not appreciate having addition UCS lines in with their drawing or solid, so you can change the system to show the UCS in the lower left hand corner of the display at all times.

Select Tools from the Menu Bar and then select Drawing Settings to pull up the Drawing Settings window. Under the Display tab, choose to Change settings for Display. In the UCS Icon section select the On radial button and then hit OK. Now the UCS icon will always appear in the corner of the display.
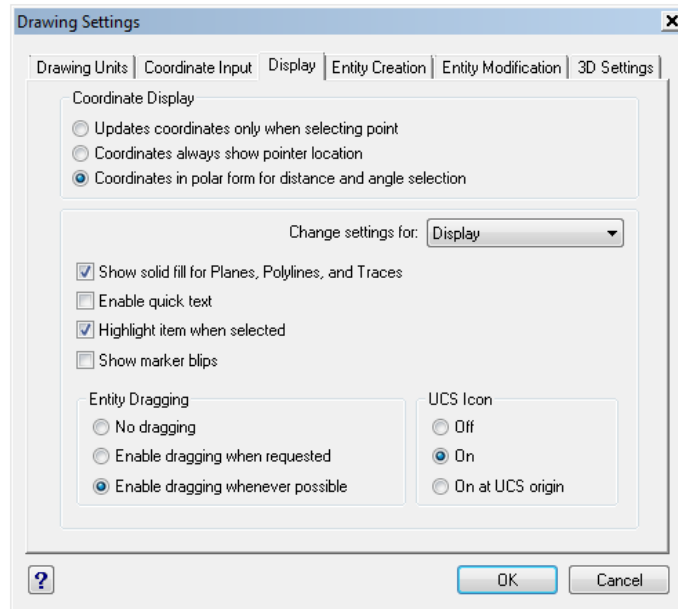

Figure 6.7 - The UCS Window, Settings Tab

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 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. If progeCAD won't let you move a line segment the way you want to, try opening the Esnap Settings window and closing it without making any changes; this seems to fix this issue.

```
Command : _POLYLINE
ENTER to use last point/Follow/<Start of polyline>:
Arc/Distance/Follow/Halfwidth/Width/<Next point>: 20
Arc/Distance/Follow/Halfwidth/Width/Undo/<Next point>: '._SETESNAP
Arc/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 20
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 20
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: C
```

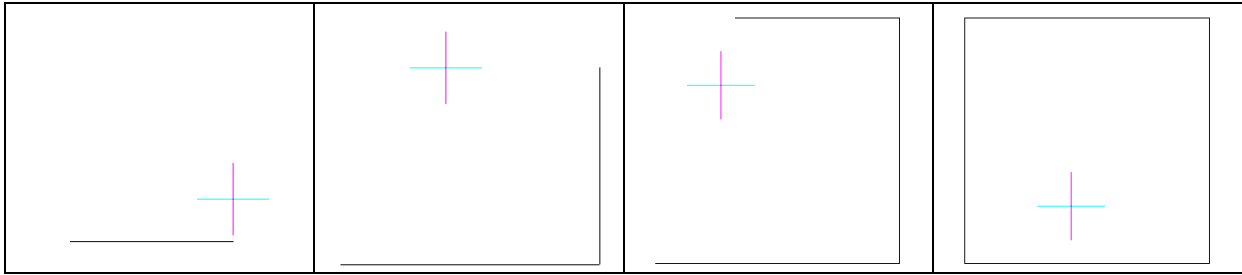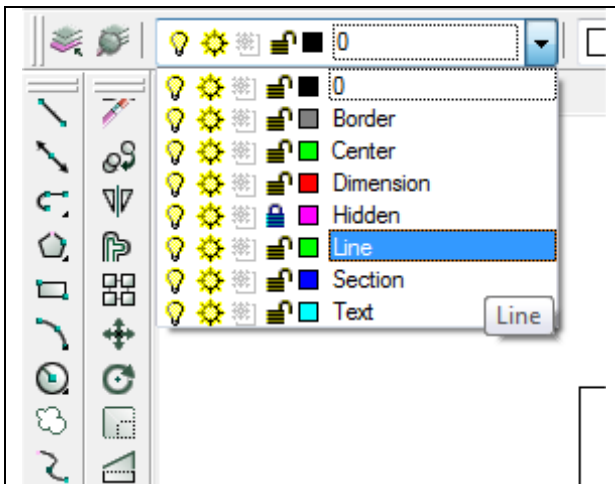**Figure 6.8 – The Polyline Text from the Command Line**

Figure 6.9 – The First Closed Polyline

Changing to the Line Layer

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To change to the Line layer select the Line layer from the Layer Drop Menu on the Entity Properties toolbar as shown in Figure 6.10. 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 menu will become increasingly useful.
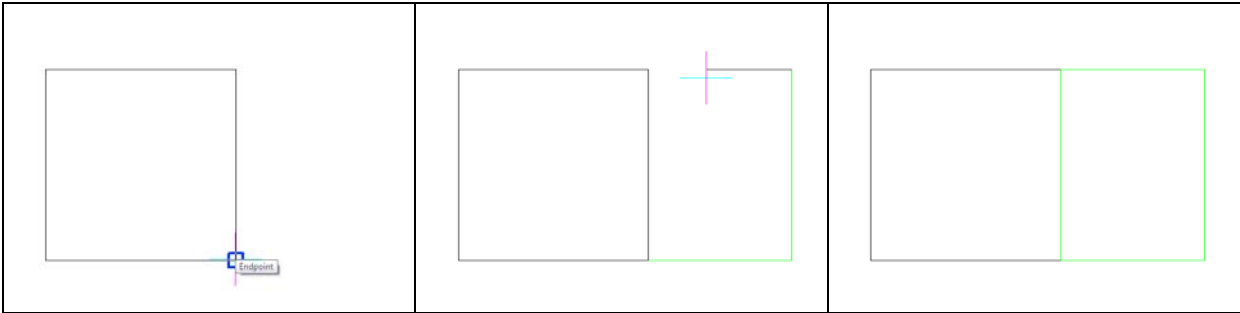
Figure 6.10 – Choosing the Line Layer

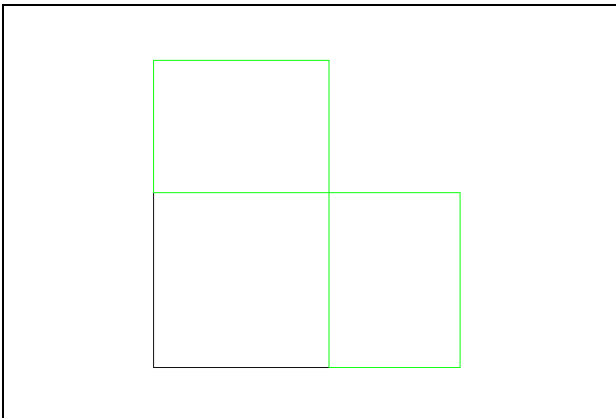## Constructing a Solid Perimeter Using the Line Command

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You need to assemble a 15 by 20 rectangle off the right side of the square Polyline entity. Select the Line tool from the Draw toolbar.

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.

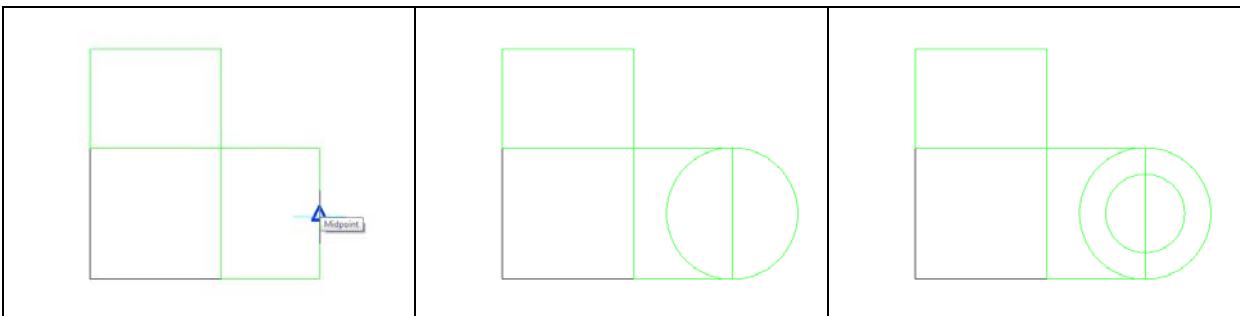**Figure 6.11 – Drawing a Series of Lines Off the Polyline Box**

Hit ENTER to restart the Line command and select the top left corner of the square polyline as the starting point of the command. Move the line being drawn on the graphic screen to the right and type "20" and ENTER. Bring the line up 15 units, to the left 20 units, and then to close the rectangle, just type "C" and ENTER.
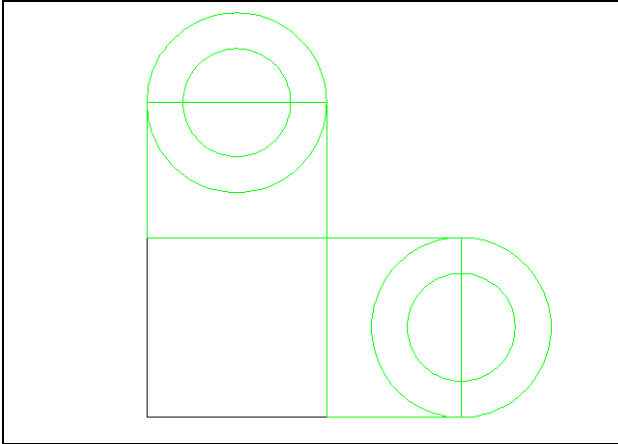
Figure 6.12 –An Additional Series of Lines

Adding Circles to the Detail

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Next, select the Circle command on the Draw toolbar. Place the mouse aperture midway up the vertical line as shown in Figure 6.13 and a Midpoint Esnap symbol (triangle) will appear in the middle of the line. Select the center point of the circle by selecting with the left mouse button. Type "**10**" when the command line prompts you to specify the "Diameter/<Radius>", or you can use the endpoint Esnap as the radius selection. Repeat the Circle command by hitting ENTER and place the **6** radius circle inside the 10-diameter circle so that they share the same center points. There will be two concentric circles as shown in Figure 6.13.
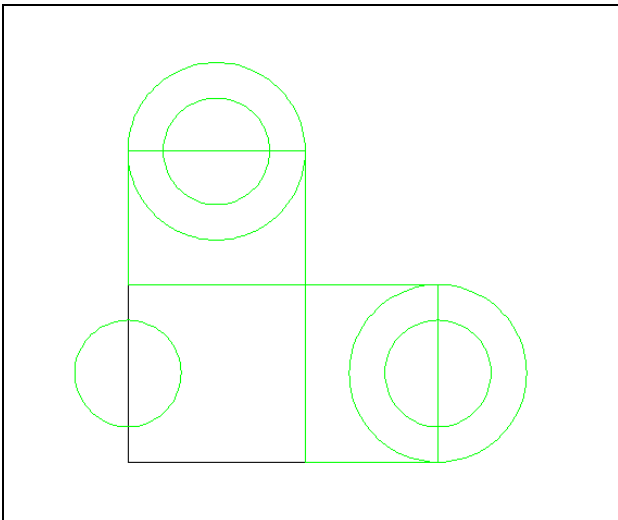

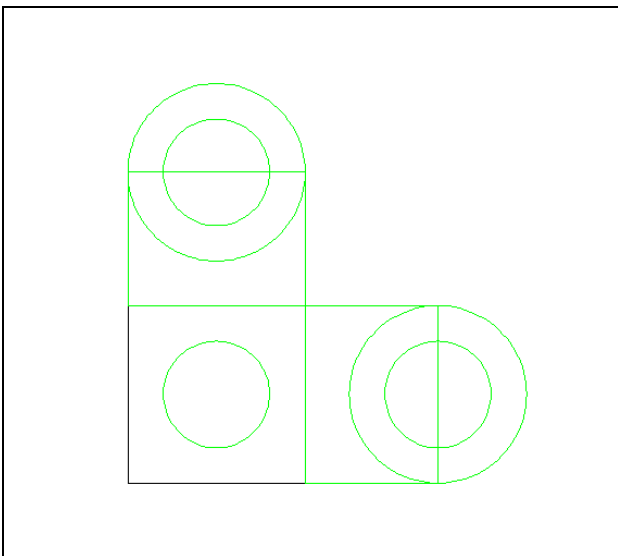**Figure 6.13 –Adding Circles For The Solid**

Figure 6.14 –Adding Circles For The Solid

Next, hit ENTER to repeat the Circle command. Place the mouse on the horizontal line as shown in Figure 6.14 and a Midpoint Esnap symbol (triangle) will appear in the middle of the line. Select it as the center of the circle and use the same technique as you just did to place a 10-unit circle. Repeat the Circle command by hitting ENTER and place a **6** radius circle inside the 10-diameter circle so that they share the same center points. Two concentric circles are shown in Figure 6.14.

Figure 6.15 –Adding Circles For The Solid

Again, hit ENTER to repeat the Circle command. Place the mouse aperture midway up the vertical line as shown in Figure 6.15 and a Midpoint Esnap 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 the "Diameter/<Radius>". The 6-unit 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, which is a distance of 10 units.
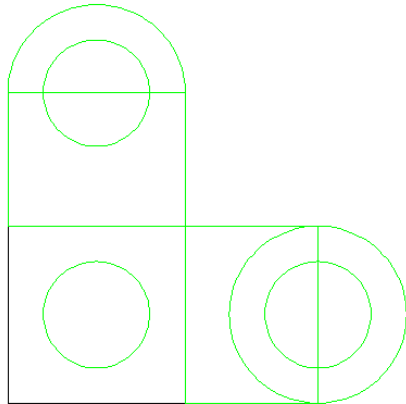

Figure 6.16 –Adding Circles For The Solid

To move the circle, select the Move tool on the Modify toolbar. Once you choose the Move command, you will be prompted to "Select entities to move:". Pick the circle and 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 choose a "Vector/<Base point>:" on the graphical display. Pick your base point anywhere on the display. Hit ENTER to move on to the "Displacement point:" and input **@10,0** which will move the entity a distance of 10 in the X-axis and 0 in the Y-axis. When leaving the Z coordinate off, the default movement is 0 on the Z-axis.

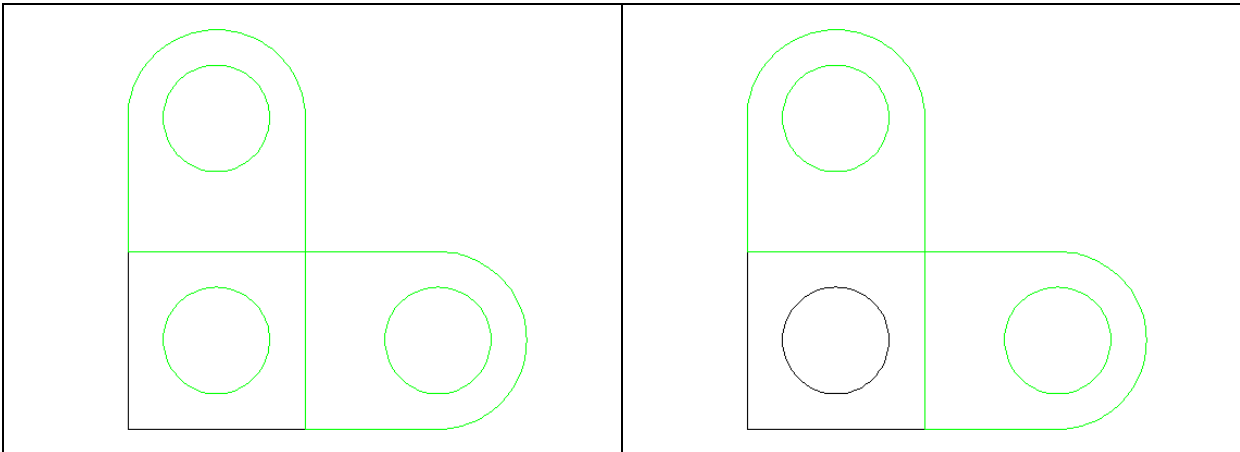# Trimming the Circles and Erasing Extra Lines

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

Select the Trim command on the Modify toolbar and hit ENTER to automatically select all lines on the graphical screen as cutting lines. Then proceed to pick the 10-unit 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 toolbar 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.
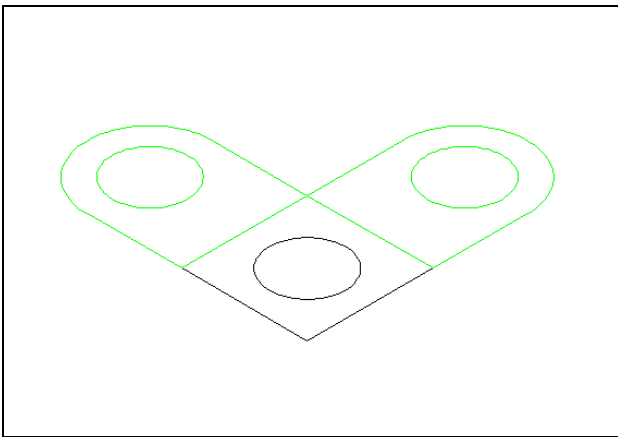
Figure 6.17 –Trim Two Circles

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

Figure 6.18 –Erase Two Lines   **Figure 6.19 –Erase Two Lines**

Freezing a Layer

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Figure 6.20 – View in SW Isometric View

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 (Figure 6.20).

To make the Polyline box and circle on Layer 0 invisible, you need to Freeze the Layer option. First select the entites to include them in the selection set. Expand the Layer Selection window and Select Layer 0 as shown in Figure 6.21. The Polygon Box and circle will disappear as shown in Figure 6.22.
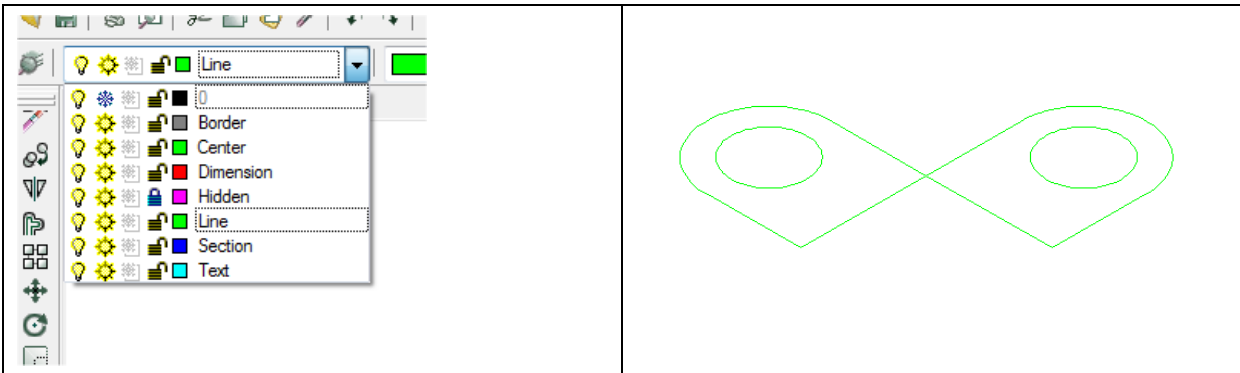

Figure 6.21 – Freezing the Line Layer

Figure 6.22 –Freezing the Line Layer

Creating Polylines Using the Edit Polyline Tool

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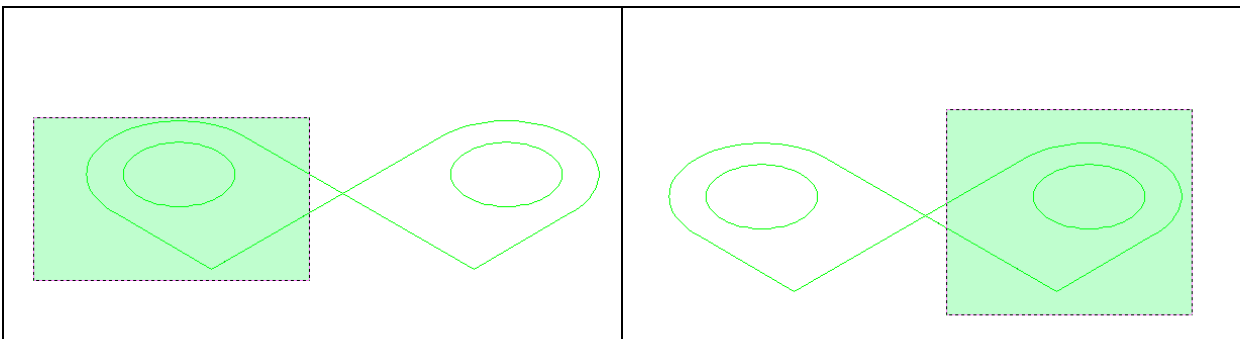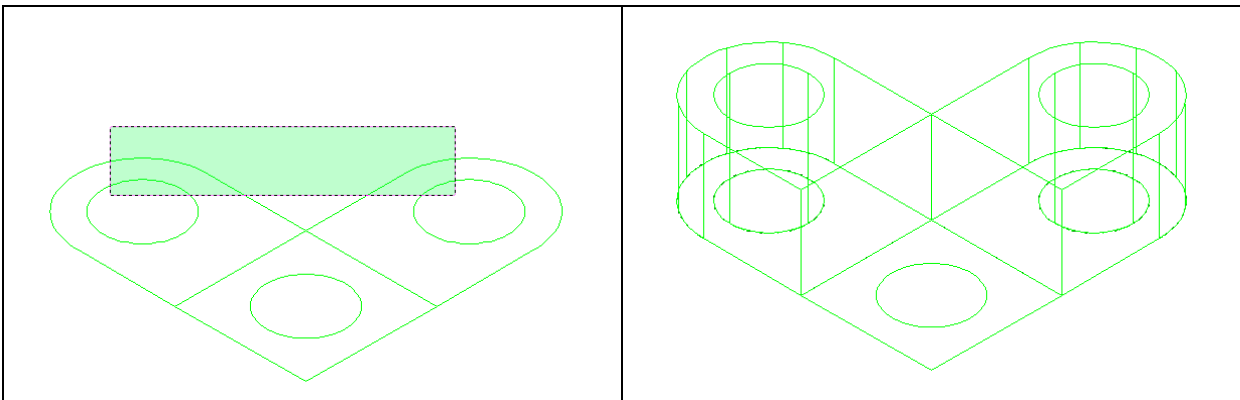

Figure 6.23 – Polyline Edit the Right Side

Figure 6.24 – Polyline Edit the Left Side

6-10

Select the Edit Polyline tool on the Modify II toolbar and the command line prompt you to "**Select polyline to edit:**". Select a side of either box, and then type "J" and ENTER. Then select to the upper right of the graphical display as shown in Figure 6.23 and going to the left, you will see the dotted selection window signifying the "crossing" option in selecting entities. Pick again to make the crossing selection. Hit ENTER to convert the entities to a Polyline; the system will respond with "Polyline is now closed. 2 vertices added to polyline." Repeat the Edit Polyline command and perform the same action with the other rectangular entity. The small radius 6 circles will remain a separate entity. You should now have four closed objects in the display. Unfreeze the 0 layer and the other two objects in the drawing will appear. You can move the 0 layer objects to the Line layer now, or you can let the Union tool take care of this later.
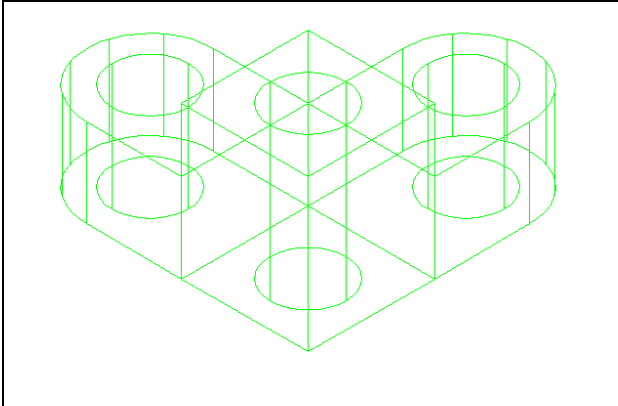
Extruding a Solid from Closed Polylines and Circles
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Only closed entities like Polylines and Circles can be extruded using this very powerful tool on the Solids toolbar.

Figure 6.25 – Selecting to Extrude	Figure 6.26 – Extrude the Entity

Select the Extrude tool on the Solids toolbar and the command line will prompt you to "**Select entities:**". Pick to the upper right of the graphical display as shown in Figure 6.25 and going to the left, you will see the dotted selection window signifying the "crossing" option in selecting entities. The command line will return with "**Entities in set: 4**". Hit **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. The polylines will expand 14 units into the z axis as shown in Figure 6.26.
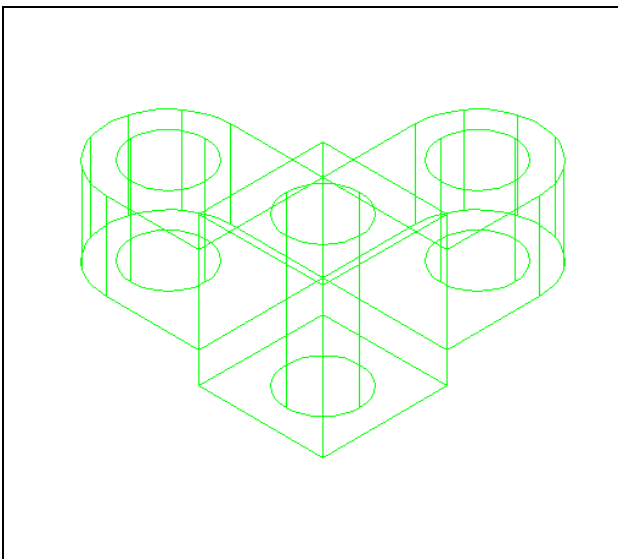
Figure 6.27 – Extrude the Middle

For the next set of solids, choose the Extrude tool on the Solids toolbar and the command line will prompt you to "**Select entities:**". Pick the square Polyline and the circle inside it. The command line will return with "**2 Entities in set: 4**". Hit **ENTER** to specify the height or path of the solid. Type "**24**" for the height of the extrusion and just hit **ENTER** to accept the zero degree angle default. A solid box will appear with a solid cylinder in the graphical display as shown in Figure 6.27.

Moving Solid Adjacent to the Master Solid

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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.
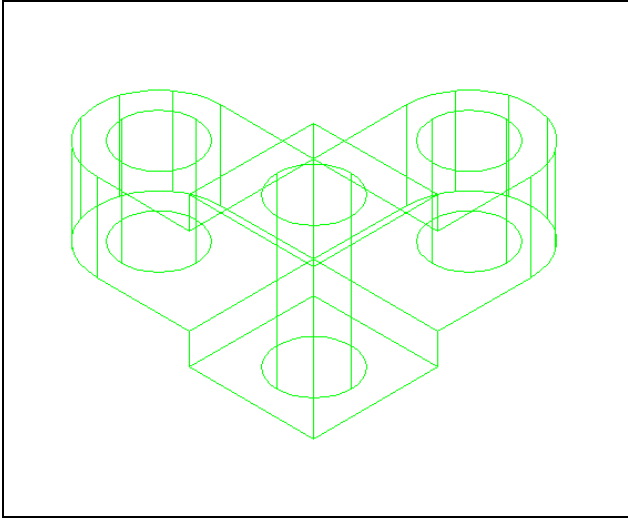

To move the solid loops to the center of the middle block, select the Move tool on the Modify toolbar. Once you choose the Move command the system will prompt you to "Select entities to move:". Pick the wings, both the exterior solids and cylinders and the command line will respond with "**Entities in set: 4**", so just hit ENTER to go the second part of the command. Next, you need to specify a "Vector/<Base point>: on the graphical display. Pick your base point anywhere on the display. Type **@0,0,5** for the "Displacement point:" moving the entities zero units in the X and Y-axis and 5 in the Z-axis as shown in Figure 6.28.

Figure 6.28 – Moving the Outside Loops

Combining Solids using Union

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For the next modification of a 3D solid shape in progeCAD, you need to use the Boolean command Union. With Union, you will join all the outside shapes together.
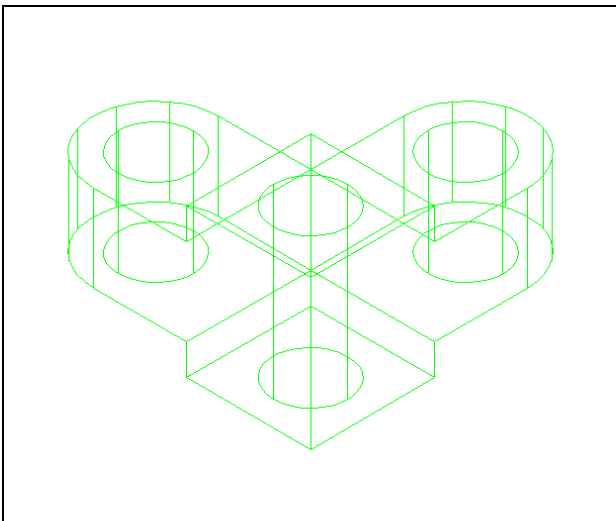
Figure 6.29 – Union the Solids

To add solids together, select the Union tool on the Solids Editing toolbar. You will be prompted to "Select ACIS objects to union:". Select all the perimeter solids but none of the cylinders. The system will respond with "**Entities in set: 3**". Hit ENTER to combine the solids into 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 object with one pick of the mouse and the entire solid (minus the cylinders) will become highlighted.

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 Solids Editing toolbar.
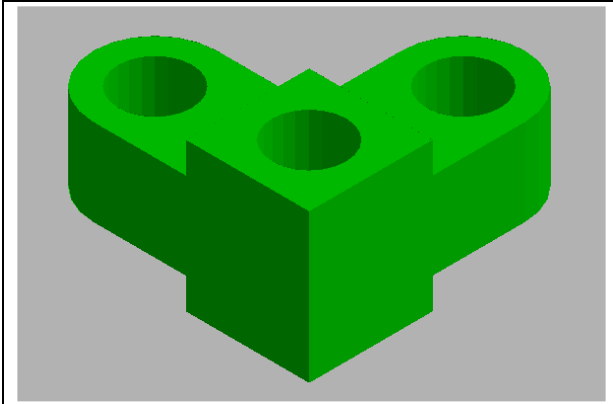

Figure 6.30 – Subtract the Three Cylinders

progeCAD will prompt you 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 three cylinders and the command line will counter with "**Entities in set: 1**". Hit ENTER and regions exactly the shape of the cylinders will disappear from the larger solid as shown in Figure 6.30. Notice that you cannot see any difference in the solid when you subtract the cylinders. You will use the Shade command to see your results better.

Shading the 3D Solid

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To view the 3D solid with holes in a better means, select the Shade tool on the Shade toolbar. The solid will acquire a filled, painted look matching the color of the layer on which the part resides.

Select the Shade tool on the Shade toolbar. 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 on the Shade 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 Modify on the Menu Bar, then 3D Operations, and pick Rotate 3D from the list of commands. At the command, "Select entities to rotate:", pick the solid on the graphical display and hit **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.
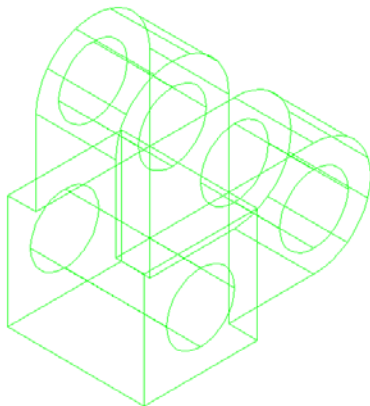
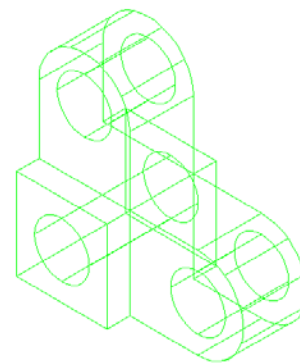**Figure 6.32 – Rotate3D the Solid**   **Figure 6.33 – Rotating the Solid**
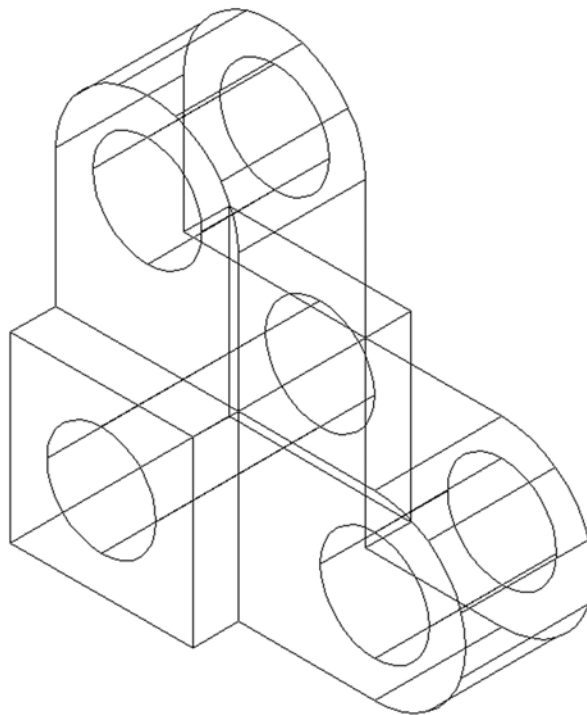
To rotate the solid in the XY plane, select the regular Rotate command on the Modify toolbar and pick the solid. The selected entity will be dotted; to continue with the command, hit **ENTER** to select the base point of rotation. Select any point at the base of the solid and type -90 for the degrees of clockwise rotation to finish this detail (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 toolbar, select the 3D solid, and hit ENTER. When prompted to select a "**Vector/<Base point>:**," select the endpoint shown in Figure 6.34 at the lower left-hand side of the Problem Five, which will become its 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 Five will move to the new origin point as shown in Figure 5.17. Try using Zoom Extents if the part completely escapes your viewing area. Change the layer of the solid to the 0 layer before saving the file.
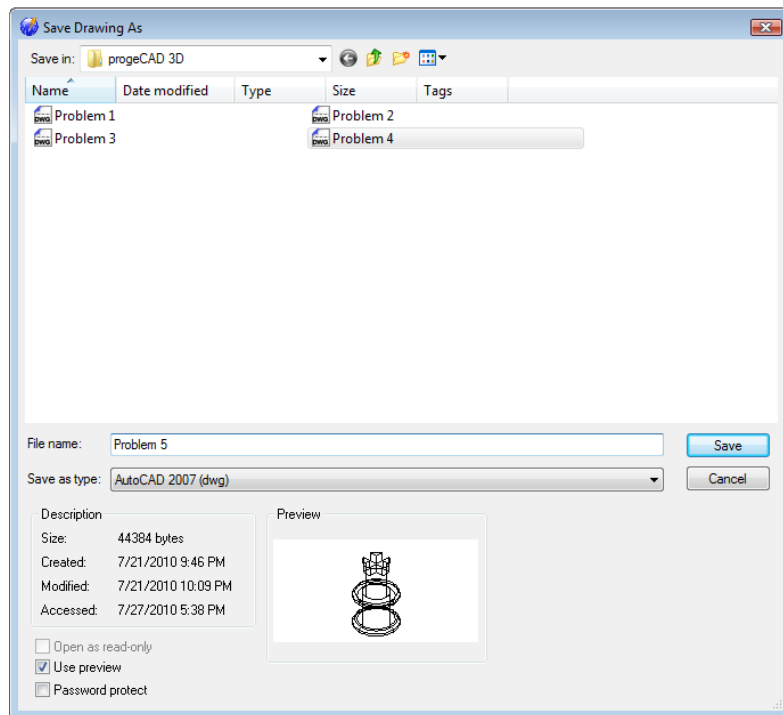
* World Class CAD Challenge 101-05 * - Close this drawing file. Create a New file and draw the closed Polyline, create the two details adjoining the Polyline box. Modify and extrude the entities, union and subtract them. Move the finished solid to the origin of the drawing. Complete the task in less than 5 minutes. Continue this drill four times, each time completing the drawing under 5 minutes to maintain your World Class ranking.

Figure 6.34 – Moving the Finished Solid to the Origin

Saving the Solid Problem

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To save Problem Five, 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 5" and press the Save button to save the drawing (Figure 6.35).

Figure 6.35 – Saving Problem Five

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