C h a p t e r

# 9

# Solid Problem Eight

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

1. Sketch of Solid Problem Eight
2. Starting a 3D Part Drawing
3. Modifying How the UCS Icon is Displayed
4. Constructing a Solid Perimeter Using the Line Command
5. Adding Circles to the Detail
6. Moving a Circle Inside the Perimeter
7. Using the Copy Command to Create Another Circle
8. Using the Offset Command to Create a New Line
9. Using the Arc Command to Add a Circular Segment
10. Creating Polylines Using the Edit Polyline Tool
11. Drawing Solid Boxes
12. Extruding a Solid from a Closed Polyline and Circles
13. Drawing a Solid Cylinder
14. Moving Solids into Position on the Master Solid
15. Subtracting 3D Solids from the Master Solid
16. Aligning Two Solids
17. Combining Solids with Union
18. Adding a Bevel Edge Using the Chamfer Command
19. Hiding the Solid
20. Moving the Solid to the Origin Point
21. Saving the Solid Problem

# Sketch of Solid Problem Eight

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You will start the eighth solids problem with a rough sketch showing the dimensions of each detail. This part is not symmetrical across any of the axis. In this exercise, a strategy you will continue to use is to draw and extrude closed Polylines that contain the most complex shape of the component and then use simple geometric solids to subtract detail. In problem Eight, the vertical solid contains an arc drawn into the shape. You will place two circles into that shape. After extruding the Polyline and Circles, you will subtract the cylinders. The second horizontal shape will have a Box and Cylinder detail subtracted. You will learn another new 3D tool, the Chamfer command, which you will use to place a 45 degree broken edge on the solid, finishing the problem as shown in Figure 9.1.
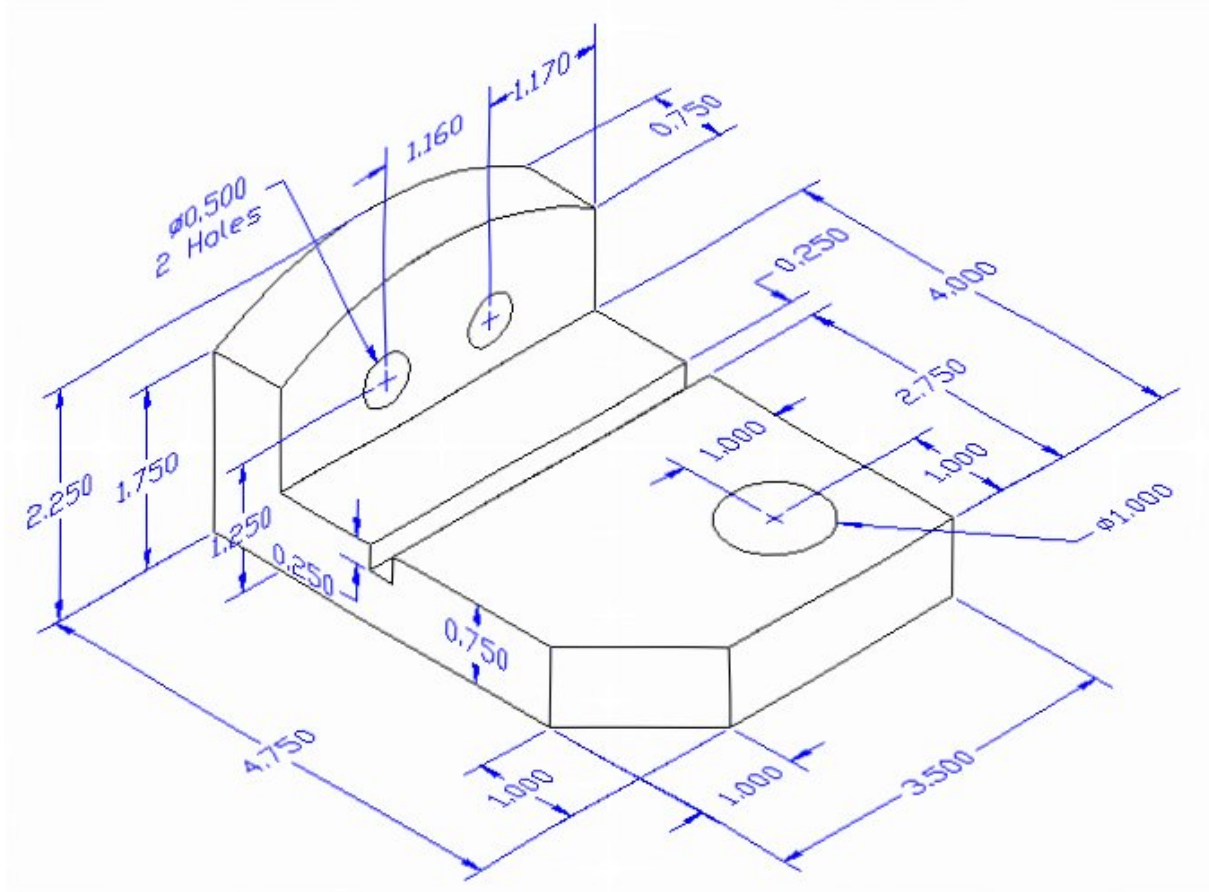
Remember, when you receive a sketch from a professional, you need to practice to quickly identify the different shapes and decide how to proceed with the solid exercise. You can see rather quickly that problem has different contours, so separate each extrusion or solid that and subtract them from the master solid. This problem will help you address solid parts where there are different complexities and sizes. 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

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In Problem Eight, 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 8.2). 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.
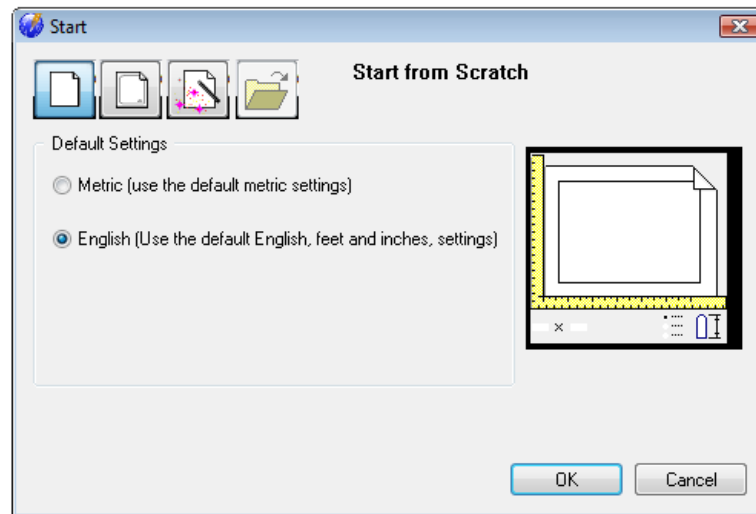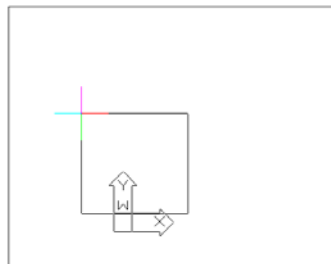


**Figure 9.2 – Starting the Drawing**

# Modifying How the UCS Icon is Displayed

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**Figure 9.3  -  The UCS in the way**

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.
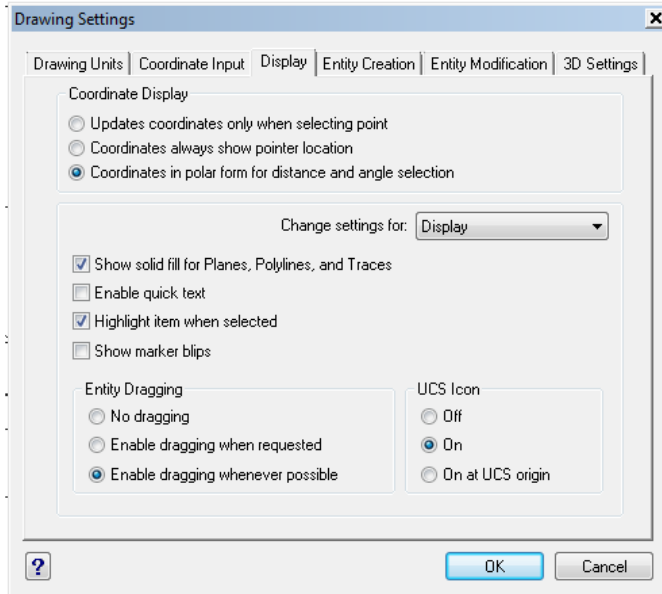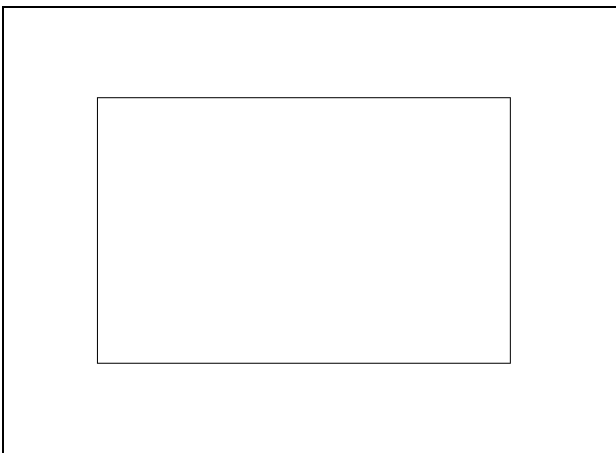
Figure 9.4 - Drawing Settings window, UCS Icon

# Constructing a Solid Perimeter Using the Line Command

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First, you need to construct a 3.5 by 2.25 rectangle by selecting the Line tool from the Draw toolbar. With your mouse, select any point at the lower right side of the graphical display.



With the Ortho mode On to keep the lines perfectly horizontal or vertical when drawing, move the line on the graphic screen to the right and type "**3.5**" and hit **ENTER**. To draw the vertical line, move the mouse upward, dragging the line and type "**2.25**" and **ENTER**. To draw the third line, move the mouse to the left, dragging the line and type "**3.5**" and **ENTER**. To close the rectangle, just type "**C**" and **ENTER**.

Figure 9.5 – Drawing a Perimeter

Notice that you did not draw a Polyline, because you will need to add some additional details to the perimeter. In the next few steps, you will add and move a circle into position. Then you will copy the circle. You will also add a construction line, which will allow you to place a 3-point arc in on the perimeter. After two lines are erased, you can edit the lines and turn the circular segment and three lines into a Polyline.

# Adding Circles to the Detail

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Next select the Circle command on the Draw toolbar. Place the mouse aperture at the upper left Endpoint as shown in Figure 9.6 and an Endpoint Esnap symbol (square) will appear on the end of the line. Select the center point of the circle by selecting with the left mouse button. Type "**D**" when the command line prompts you to specify the "Diameter/<Radius>:" and hit **ENTER**. Type **0.5** for the diameter and the circle will appear on the corner as shown in Figure 9.7
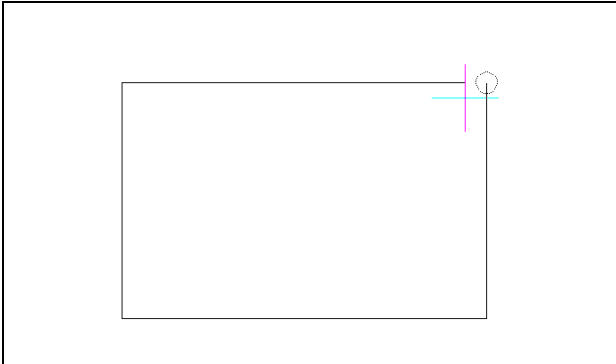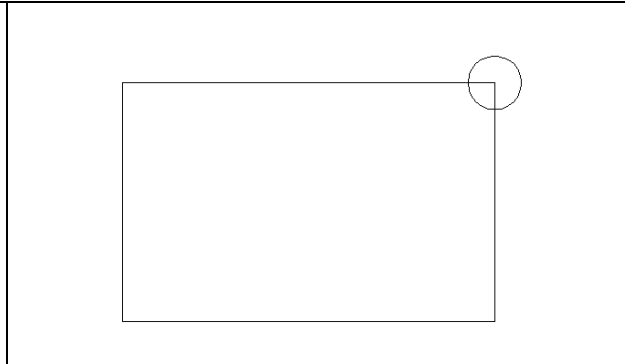


**Figure 9.6 – Selecting the Center**          **Figure 9.7 – Adding a Circle to the Solid**

# Moving a Circle Inside the Perimeter

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Next, you will need to position the circle to a precise location. Use the Move tool on the Modify toolbar to accomplish the task.
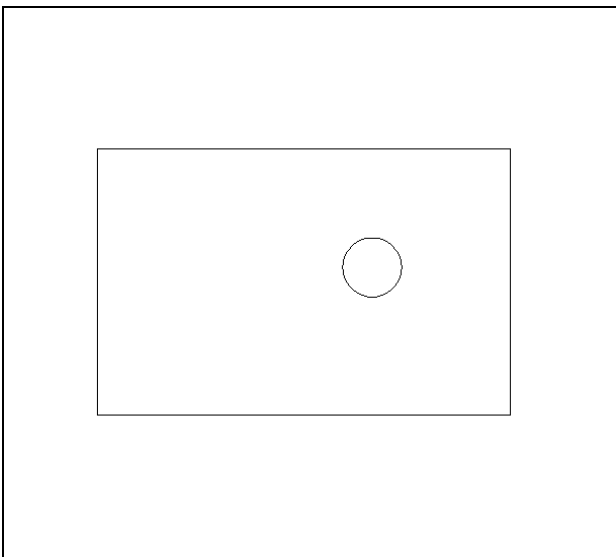


Once you choose the Move command, you will be prompted to "Select entities to move:". Pick the circle and the command line will respond with "**Entities in set: 1**", so hit ENTER to go the second part of the command. Next, you need to choose a "Vector/<Base point>:" on the graphical display. Pick your base point anywhere on the display. Hit **ENTER** and then type the "Displacement point:" as **@-1.17,-1.0** which will move the entity a negative distance of 1.17 on the X-axis, a negative 1.0 in the Y-axis, and when leaving the Z coordinate off, the default is 0 in the Z-axis. The circle will be as shown in Figure 9.8.

**Figure 9.8 – Positioning the Circle**

# Use the Copy Command to Create Another Circle
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The Copy command is actually a three-dimensional function in progeCAD. Like the Move tool, you can use the Copy function to relocate any entity to another point (X,Y,Z) in Model Space.
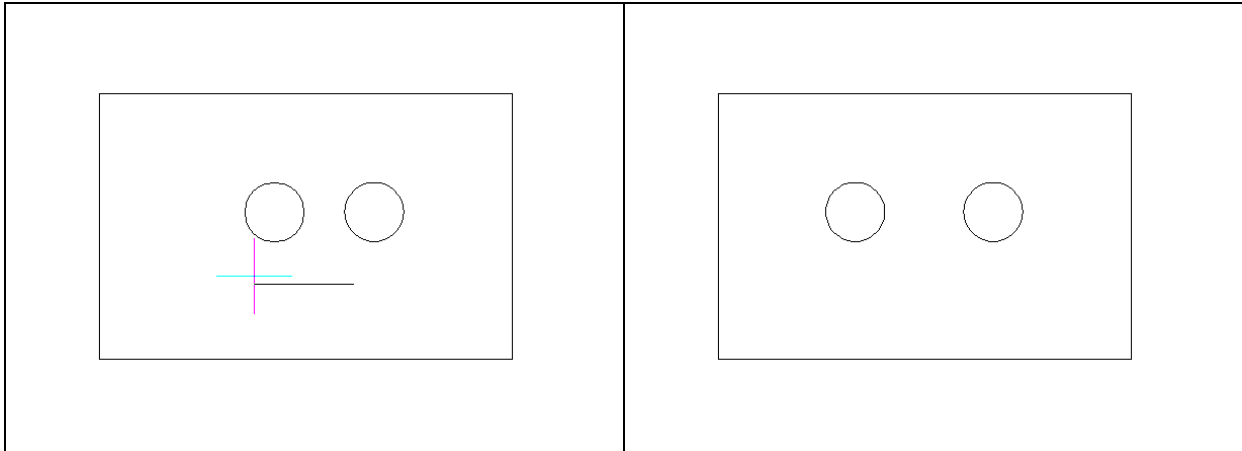


**Figure 9.9 – Copying the First Circle**          **Figure 9.10 – Positioning the Circle**

The Copy command is separated into two parts, the first being the selection process and the second being the function of the command itself.  To copy an entity such as the circle, select the Copy tool on the Modify toolbar.  Once you choose the Copy command, the prompt will be "Select entities to copy:".  You can do this by placing the pick box that now has replaced the aperture cursor on the 0.5 diameter circle and selecting the entity.  The command line will respond with "Entities in set:  1", so just hit **ENTER** to go the second part of the command.

Now you need to "Vector/mOde/<Base point>:" on the graphical display.  Pick your base point on any blank space on the graphical display but not on top of any entity because Entity Snaps can interfere with the process. For the "Displacement point:", push the circle to the left as shown in Figure 9.9. Type **1.16** and the second circle will appear 1.16 to the left as shown in Figure 9.10.

# Using the Offset Command to Create a New Line

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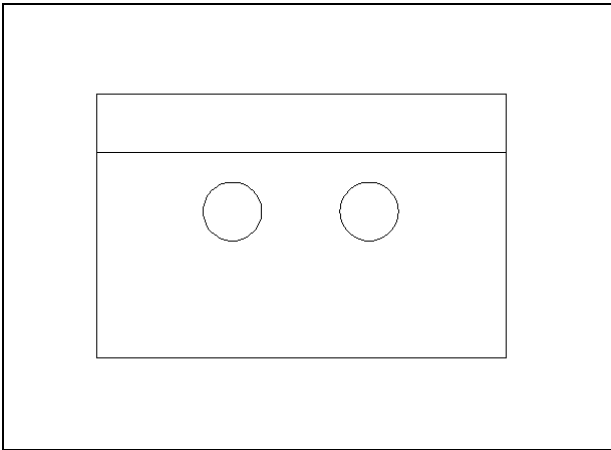


To create a new line 1.75 above the bottom line of the box, you will use the Offset command which will allow you to create a line without selecting points to define the start and endpoints. Choose the Offset tool on the Modify toolbar. The command line will prompt you for the offset distance; type **1.75** and **ENTER.**

**Figure 9.11 – Offset the Bottom Line, 1.75**

Next, select the line you wish to offset: the bottom horizontal line. After selecting the line, pick the direction of movement, which is above the bottom horizontal line. A new line will appear 1.75 above the bottom line as shown in Figure 9.11. Hit **ENTER** to leave the Offset command after placing the new line.

# Using the Arc Command to Add a Circular Segment

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This problem has a few new functions to learn. You will need to become skilled at creating a circular segment or arc as occasionally there will be no other way to place one. The Fillet command will join two lines and place a circular segment at the precise radius without many construction problems. Computer-aided designers will usually select the Fillet tool rather than the Arc command. You can spend hours learning the many variations of the Arc command, but today you will learn how to draw a simple three point circular segment.
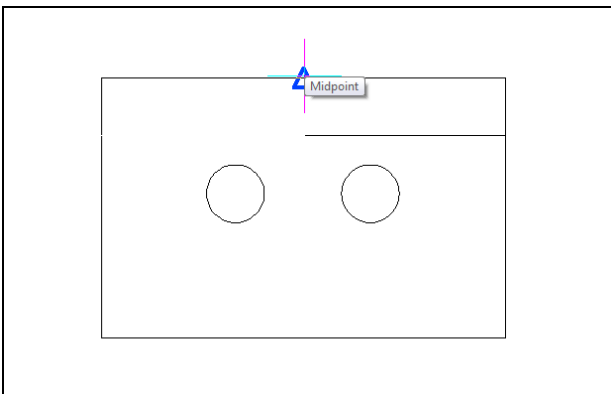
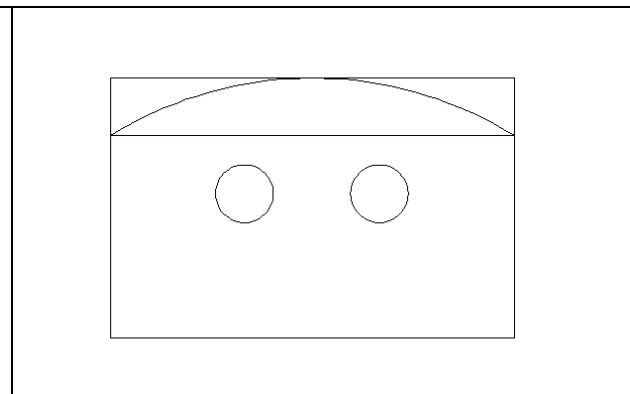

**Figure 9.12 – The First Point of the Arc**       **Figure 9.13 – The Second Point of the Arc**

9-7

Pick the Three Point Arc tool on the Draw toolbar. As shown in Figure 9.12, pick the Endpoint on the offset line just drawn. Next, using the Midpoint entity snap, select the center of the top line as shown in Figure 9.13. After the second selection, you can see a light tracing of the arc on the graphical display; the last pick will be the Endpoint on the right side of the offset line as shown in Figure 9.14. Use the Erase tool on the Modify toolbar to remove the offset and top lines as shown in Figure 9.15.
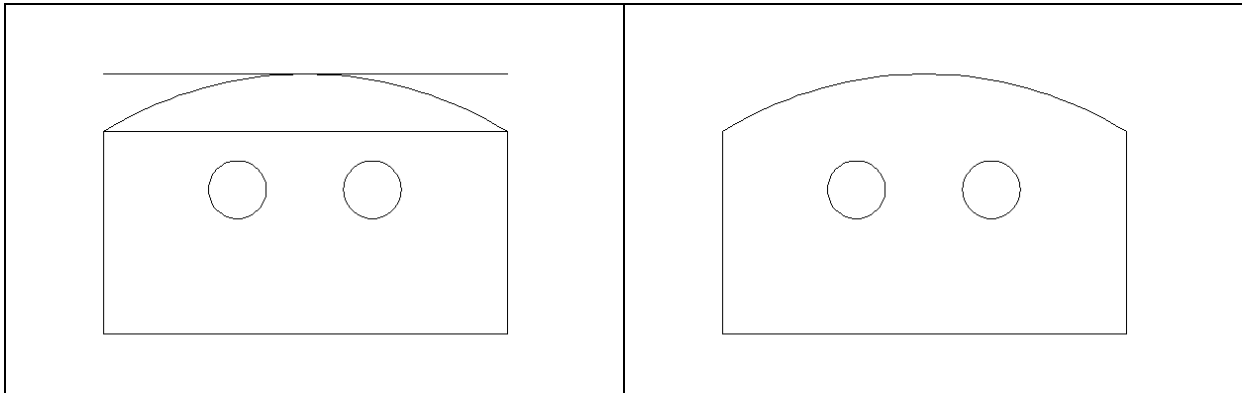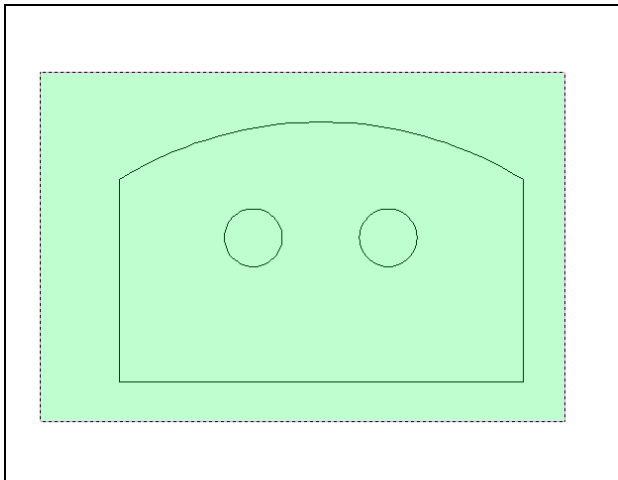


**Figure 9.14 – The Third Point of the Arc**       **Figure 9.15 – Erase Two Lines**


# Creating Polylines Using the Edit Polyline Tool
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To merge the three lines and arc segments together you need to use the Edit Polyline tool.



Select the Edit Polyline tool on the Modify II toolbar and prompt you to "**Select polyline to edit:**". Select a segment of the part outline and the command line will prompt you to enter an Edit Polyline function. Type J and ENTER to go to the Join feature of the Edit Polyline tool. You will be prompted to "Select entities to join." Use a selection window to select all of the lines in the outline and hit ENTER. The command line will respond with "Polyline is now closed. 2 vertices added to polyline." Hit Escape to exit the Edit Polyline command

**Figure 9.16 – Polyline Edit the Perimeter**

On the View toolbar, select the Southwest Isometric View tool to allow you to see the part on the XY plane with the Z-axis rising vertically upward.

# Drawing Solid Boxes

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The technique you will use in this exercise will utilize the Length option of the Box command.

Select the Box tool on the Solids toolbar. At the command line, a prompt asks you to specify the corner of the solid.  For this box, specify the point anywhere to the right of the solid. Then type **L** after specifying the starting point and the system will ask you three questions. Specify the "Length of side of box:" as **3.5** for the X coordinate.  Enter **4.75** as the value for the width and type **0.75** as the height of the box.  The rectangular solid will appear as shown in Figure 9.17.

Hit ENTER to repeat the Box command. Pick the closest corner to you of the box you just drew. Then type **L** after specifying the starting point and the system will ask you three questions. Specify the "Length of side of box:" as **3.5** for the X coordinate.  Enter **0.25** as the value for the width and type -**0.25** as the height of the box.  The rectangular solid will appear as shown in Figure 9.18.
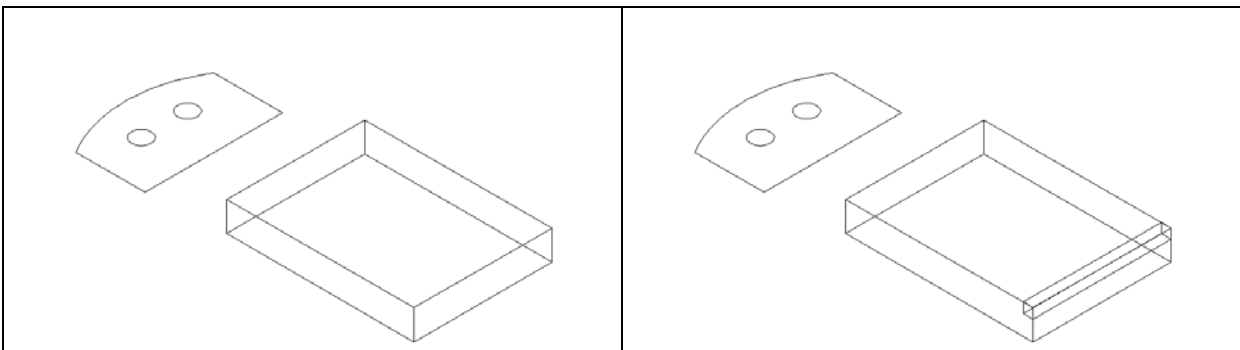
| | |
|---|---|
|  |  |
| **Figure 9.17 – The Solid Box** | **Figure 9.18 – The Second Solid Box** |

# 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. You now have three shapes that are single entities.

Select the Extrude tool on the Solids toolbar and the command line will prompt you to "**Select entities:**". Use a selection window to select the two circles and the solid outline. The command line will return with "**Entities in set: 3**". Hit **ENTER** to specify the height or path of the solid. Type "**0.75**" for the height of the extrusion, hitting ENTER again to keep the angle of taper at 0 degrees. The shapes will extend 0.75 units up into the z axis as shown in Figure 9.20.
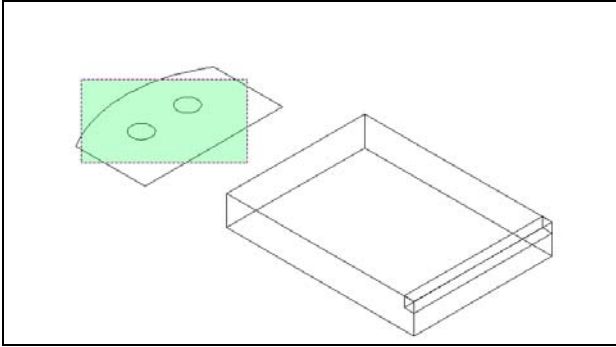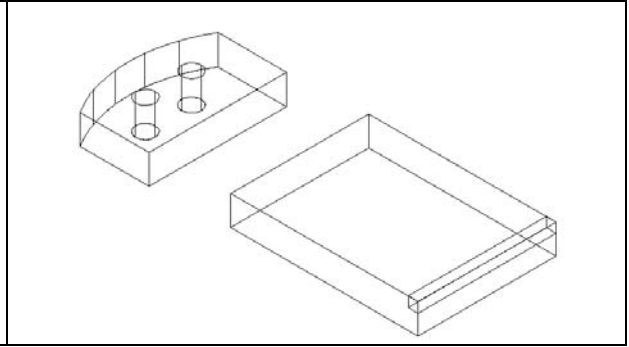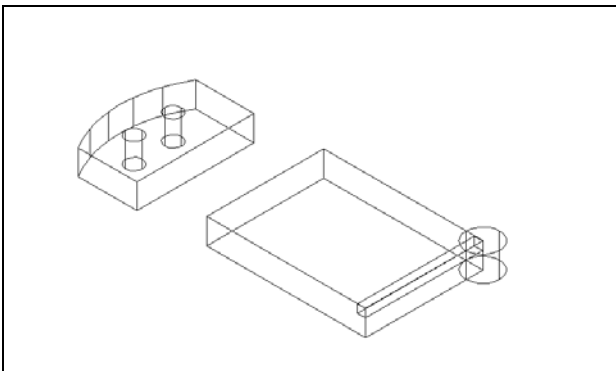
**Figure 9.19 – Select Polyline and Circles**   **Figure 9.20 – Extruding Two Entities**

# Drawing a Solid Cylinder
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You need to add a 1.0 diameter hole in the master solid, so use the Cylinder command
to create the feature.



Pick the Cylinder tool on the Solids toolbar.
Specify the center point for the base of the
cylinder as the lower right Endpoint on the
solid box. Type "**D"** for diameter and hit
**ENTER** to input the sketch's measurement of
**1.0**. for the next prompt, "Center of second
end/<Height of cylinder>:". Key in **0.75** for
the elevation of the can shaped solid and hit
ENTER. The cylinder will appear in the
graphical display as shown in Figure 9.21.

**Figure 9.21 – Adding a Solid Cylinder**

# Moving 3D Solids into Position on the Master Solid
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The Move command is actually a three-dimensional function. The way you can
determine whether a command is three dimensional in nature is to attempt to use the
function outside the Z-plane. You can use move to relocate any entity to another point
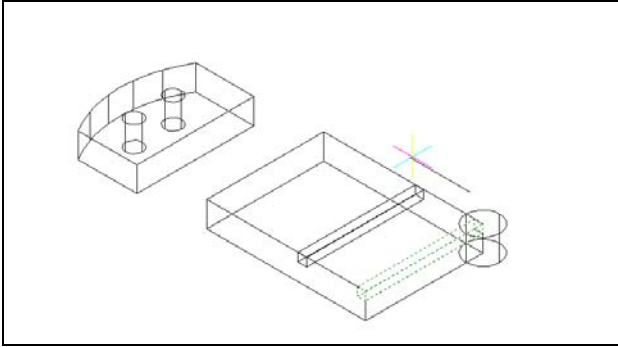(X,Y,Z) in Model Space.
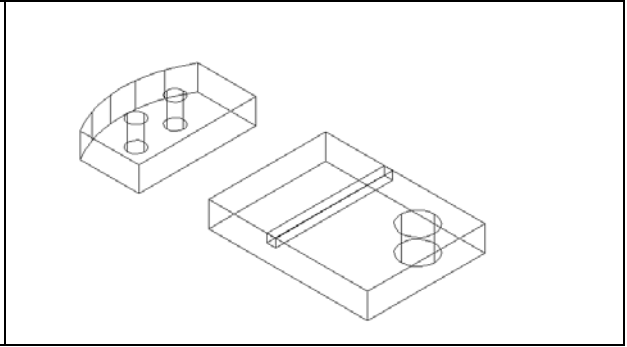
**Figure 9.22 – Moving the Box**          **Figure 9.23 – Moving the Cylinder**

Once you choose the Move command, you will be prompted to "Select entities to move:". Pick the smaller solid box and the command line will respond with "**Entities in set: 1**", so 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** and then type **2.75** as the "Displacement point" while moving the box to the left as shown in Figure 9.22.

Hit ENTER to repeat the Move command. You will be prompted to "Select entities to move:". Pick the cyliner and the command line will respond with "**Entities in set: 1**", so 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** and then type the "Displacement point:" as **@-1.0,1.0** which will move the entity a negative distance of 1 on the X-axis, a negative 1 in the Y-axis, and when leaving the Z coordinate off, the default is 0 in the Z-axis. The cylinder will be as shown in Figure 9.23.

## Subtracting 3D Solids from the Master Solids
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To remove the cylinders and box from the master solids, select the Subtract tool on the Solids Editing toolbar.
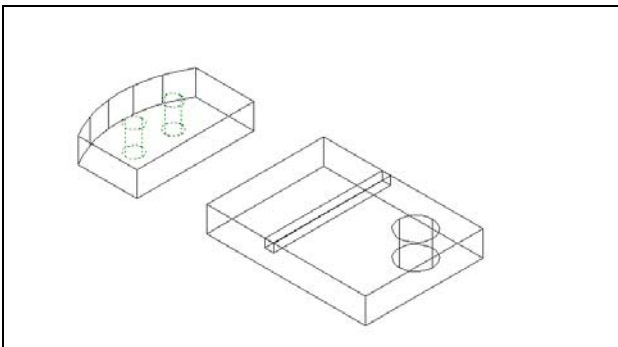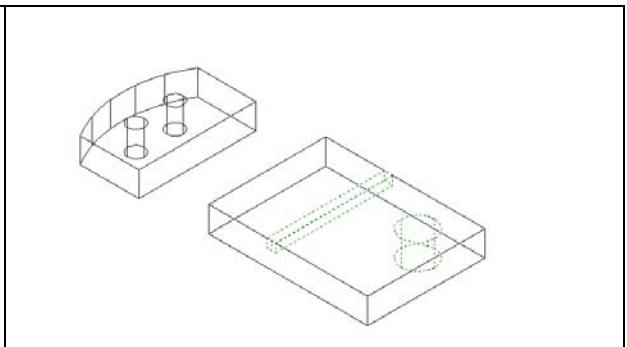


**Figure 9.24 – Subtract Solids**          **Figure 9.25 – Subtracting Solids**

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The command line will prompt you to "Select ACIS object to subtract from:". Select the larger master solid to the left 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 cylinders inside of the master solid and the command line will reply with "Entites in set:  2". Hit **ENTER** and a region exactly the shape of the cylinders will be removed from the main solid as shown in Figure 9.24. Notice that you cannot see any difference in the solid after subtracting the cylinders.

Hit ENTER to repeat the Subtract command.  The command line will prompt you to "Select ACIS object to subtract from:". Select the larger master solid to the right 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 cylinder and the small thin box and the command line will reply "Entites in set:  2". Hit **ENTER** and a region exactly the shape of the cylinder and box will be removed from the main solid as shown in Figure 9.25.
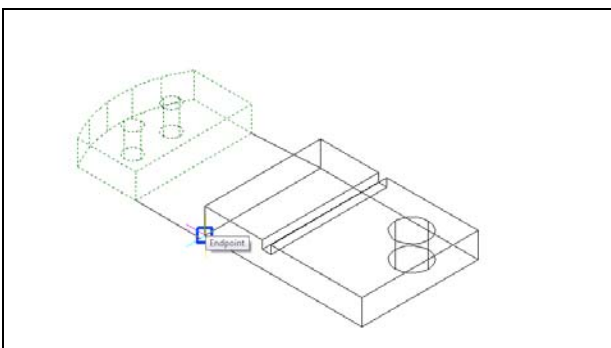
# Aligning Two Solids

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Select Modify on the Menu Bar, then 3D Operations, and pick Align from the list of commands. At the command, "Select entities:", pick the master solid to the left and hit **ENTER** to proceed to the second part of the function.



**Figure 9.26 – Align First Points**
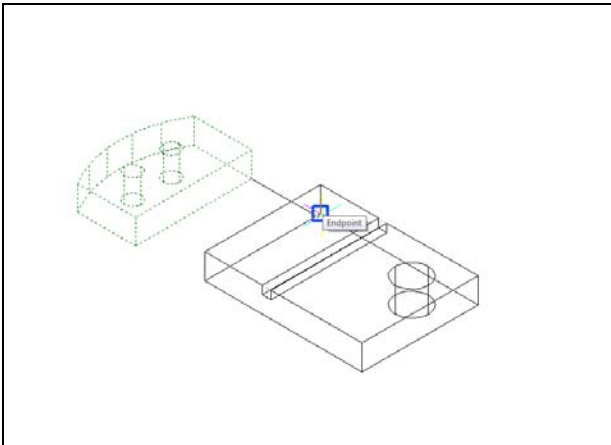
The first alignment point brings the two points selected together. Of the three choices for alignment, you can only guarantee that the first group will unite exactly. This makes the first selection the most important, since you will only want to run the command once. In Figure 9.26, the first source point is the Endpoint on the bottom of the arced solid. The destination is the Endpoint on the bottom left tessellation line of the master solid. A temporary line appears on the display showing the connection.



**Figure 9.27 – Align Second Points**

The second set of points will signify the two dimensional line of alignment. You can see in Figure 9.27 that the second source point is the lower right corner of the arced solid. The second destination point is the lower far corner of the master solid. The two exact points defining this line will be the first and second point selections.
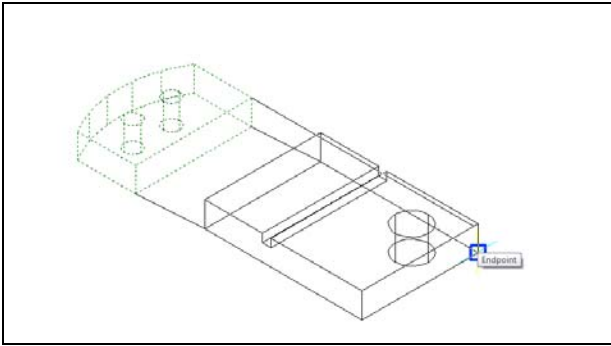
The last group of points will align the solid to the defined plane. You can see in Figure 9.28 that the third source point selection is the upper right corner of the arced solid. The destination point selection is the lower right corner Endpoint of the solid box. The arced solid will move into its new position vertically aligned with the second solid.

**Figure 9.28 – Align Third Points**

# Combining 3D Solids with Union

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Two bring the two master solids together as one, you will use the Union command. Select the Union tool from the Solids Editing toolbar.
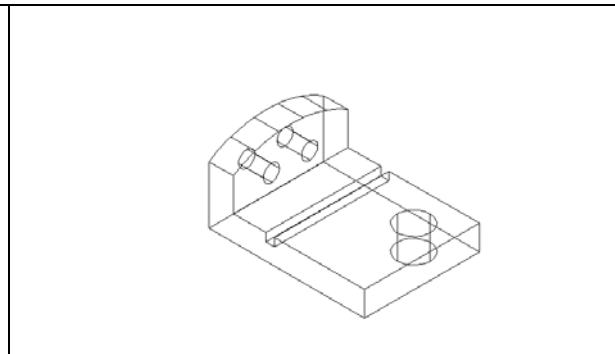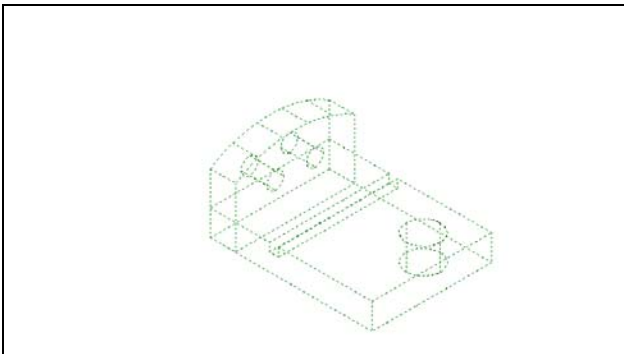


**Figure 9.29 – Before Joining Two Solids**      **Figure 9.30 – After Joining Two Solids**

The command line will prompt you to "Select ACIS objects to union:". Select both solids and hit ENTER. Now all the solids have become one entity as shown in Figure 9.30. You can see in the graphical display the difference in that the tessellation lines outlining and overlapping each other are gone. To check your work, select the entity with one pick of the mouse and the entire solid will become highlighted.

## Add a Bevel Edge Using the Chamfer Command

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To add a beveled edge to a corner, you can use the Chamfer command to create the feature rather than adding a triangle and subtracting the detail.

9-13
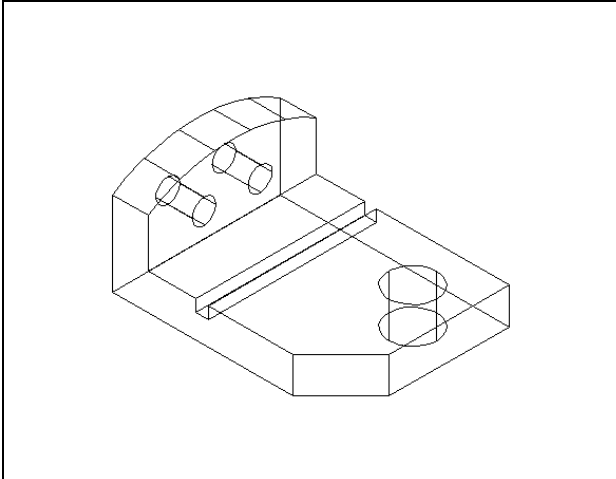
Pick the Chamfer tool on the Modify toolbar and select the lower vertical corner on the edge of the solid. Hit ENTER to OK the Surface Selection option. Type "**1.00**" for the first offset distance. Hit ENTER and type "**1.00**" for the second offset distance. Hit ENTER and select the vertical tessellation line closest to you once again. Hit ENTER and the 1.0 by 45 degree chamfer will appear on the edge of the master solid as shown in Figure 9.31. You will commonly use this technique to improve the process of fitting closely dimensioned parts together.

**Figure 9.31 – Chamfer the Edge**

# Hide the 3D Solid

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In Figure 9.31, you will notice that the tessellation lines representing the solid's outline on the far edge of the part are visible. In a real world solid part, you would not be able to see through solid unless its material was transparent. You can use the Hide command to remove the hidden lines from your graphical display.
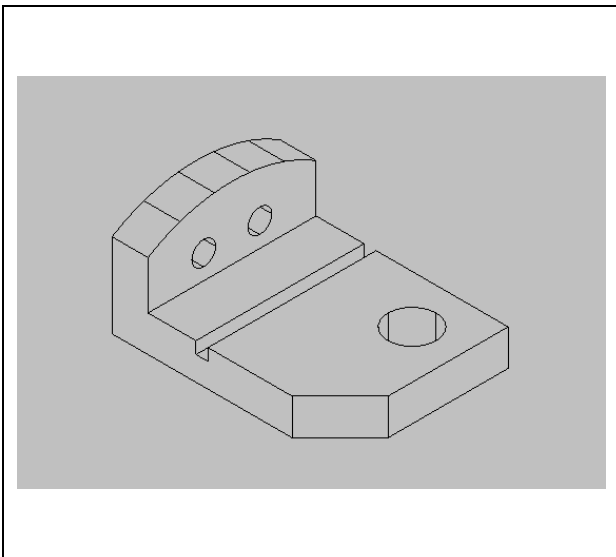


Select the Hide tool on the Shade toolbar. This is a very simple command because after picking the tool the tessellation lines representing the back of the part are removed from view as shown in Figure 8.30. If you wish to return the graphical display to the previous setup in which all the tessellation lines can be seen, choose the first tool on the Shade toolbar called 2D Wireframe. By choosing the 2D Wireframe tool, the solid part and the UCS icon will return to the previous appearance. This command does not alter the mass of the solid or any of the dimensions, but is available for the computer aided designer as a viewing option.

**Figure 9.32 – Hide the Solid**

# Moving the Solid to the Origin Point

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To move Problem Eight 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 8.31 at the lower left-hand side of the Problem Seven, which will become its insertion point if it were inserted into an assembly drawing (Figure 9.33). For the prompt to select a "Displacement point:," type **0,0,0** and hit ENTER. Problem Seven will move to the new origin point. Try using Zoom Extents if the part completely escapes your viewing area .

**\* World Class CAD Challenge 101-08 \* - Close this drawing file. Create a New file and draw the solid using the techniques in this chapter. 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.**
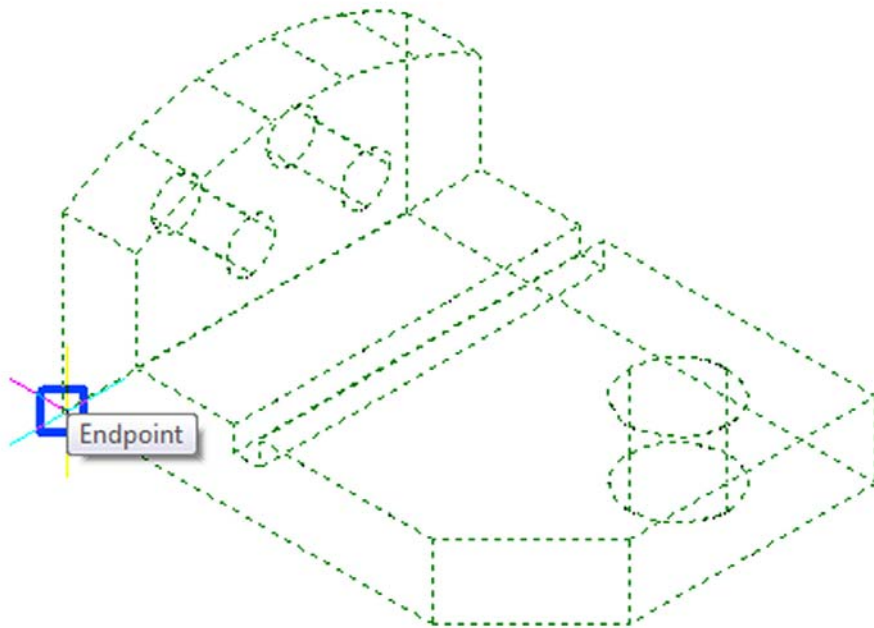


**Figure 9.33 – Moving the Finished Solid to the Origin**

# Saving the Solid Problem

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To save Problem Eight, 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 7" and press the Save button to save the drawing (Figure 9.34).
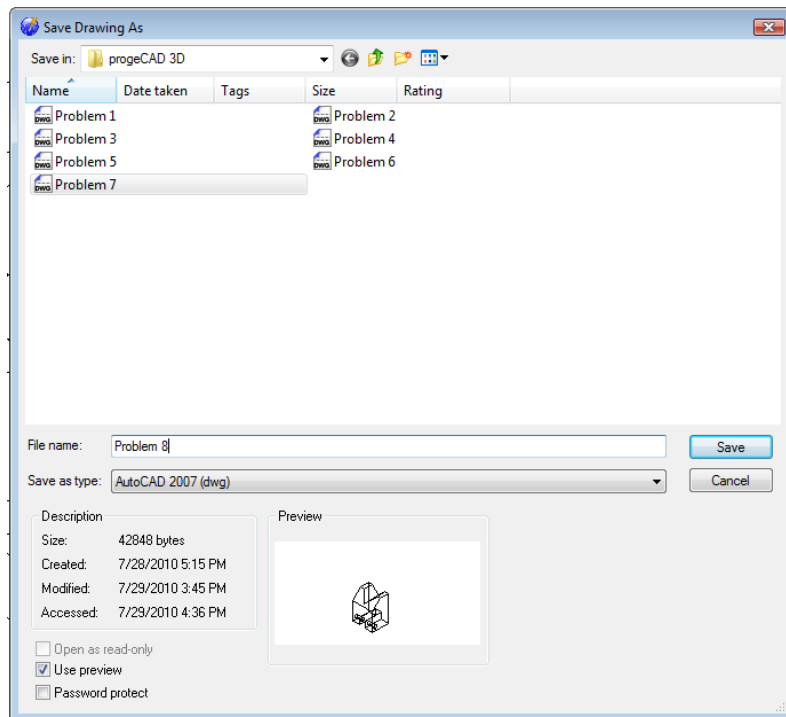
**Figure 9.34 – Saving Problem Eight**

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