C h a p t e r

7

Solid Problem Six

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

1. Sketch of Solid Problem Six
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 a Circle to the Detail
6. Trimming the Circle and Extra Lines
7. Creating Polylines Using the Edit Polyline Tool
8. Drawing a Closed Polyline Shape to be Extruded
9. Extruding a Solid from Closed Polylines
10. Rotating a 3D Solid Using the Rotate3D Command
11. Aligning a 3D Solid Using the Align Command
12. Combining Solids Using Union
13. Hiding the Solid
14. Moving the Solid to the Origin Point
15. Saving the Solid Problem

Sketch of Solid Problem Six

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You start the sixth solids problem with a rough sketch showing the dimensions of each detail. This part is not symmetrical. In this exercise, a strategy you will use is to draw lines, add a circle and trim to create a complex shape. You will convert the intricate silhouette to a closed Polyline. Next, you will draw a closed Polyline containing just line segments. After extruding the two Polylines, you will align the two shapes together and union them. Your three dimensional drawing will appear as shown in your sketch in Figure 7.1.
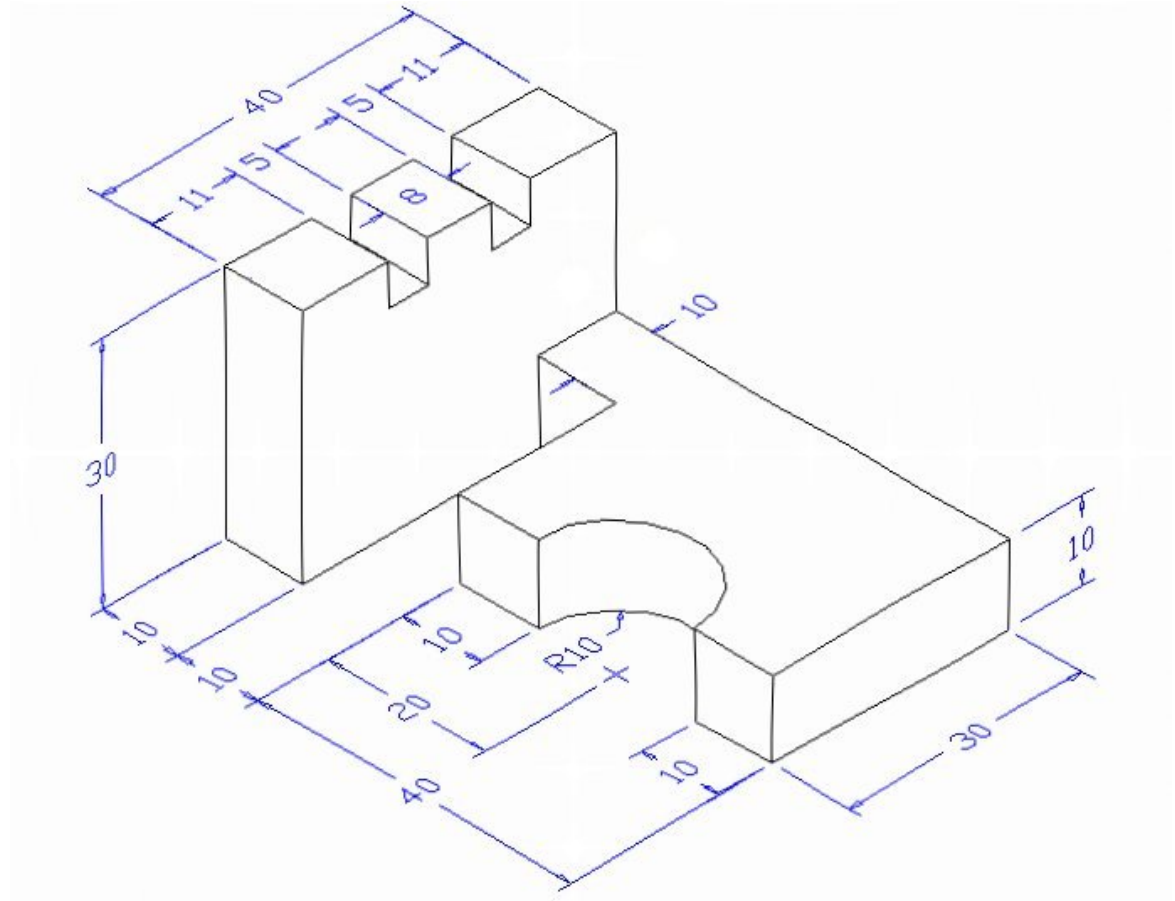
Figure 7.1 – Problem Six Sketch

By this point in your three-dimensional training, reading an isometric sketch should be getting easier. 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 six has two different complex shapes to be drawn and brought together. Those two recognizable silhouettes are 90 degrees apart, but your ability to rotate solids is getting better. Also, you will be learning about the Align command which will both move and rotate the solid simultaneously. So, this problem will help you address solid parts containing different complexities, and all ten drills in the Fundamentals of 3D Drawing textbook will give you those repetitive maneuvers that will make you successful in 3D modeling.

Starting a 3D Part Drawing

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In Problem Six, 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 7.2). 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.
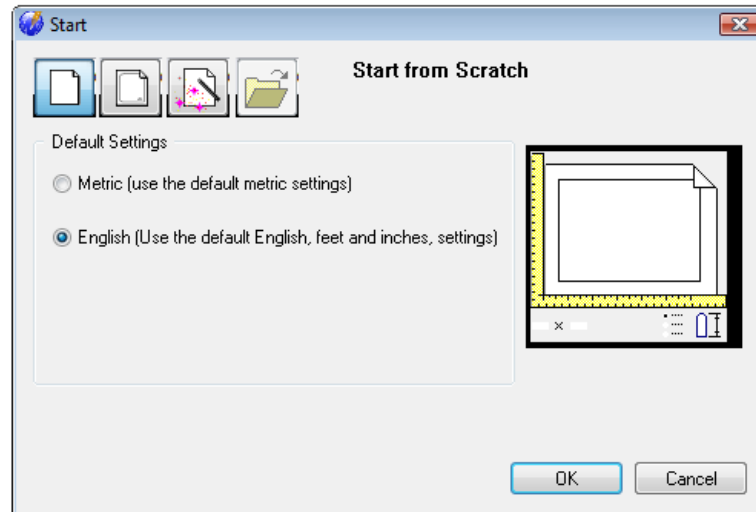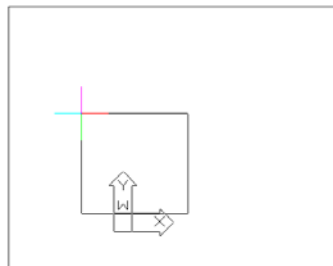

Figure 7.2 – Starting the Drawing

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.

Figure 7.3 - The UCS in the way

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.
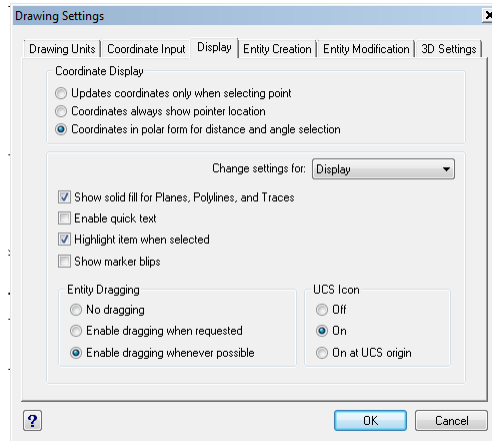
Figure 7.4 - Drawing Settings: UCS Icon

Constructing a Solid Perimeter Using the Line Command
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You need to assemble a special shape using the Line command. Select the Line tool from the Draw toolbar. With your mouse, select a point at the lower right side of the graphical display. Even while in the Line command, set the Ortho mode to On to keep the lines perfectly horizontal or vertical when drawing.
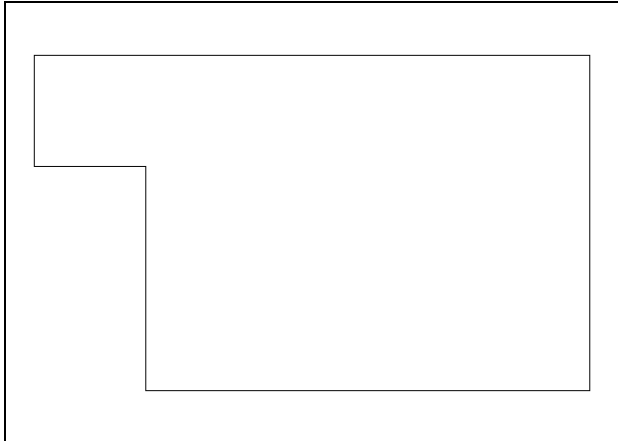

Move the line being drawn on the screen to the right and type "40" and ENTER at the keyboard. To draw the vertical line, move the mouse upward, dragging the line and type "30" and ENTER. Drawing the third line, move the mouse to the left, dragging the line and type "50" and ENTER. On the fourth line, move the mouse down and type "10" and ENTER. Next, move the line being drawn on the graphic screen to the right and type "10" and ENTER at the keyboard. To close this shape, just type "C" and ENTER.

Figure 7.5 – Drawing a Series of Lines

The purpose of drawing lines and not using the Polyline function is to enable you to add more detail before making the shape a single entity. You need to proceed to adding a circle.

Adding Circles to the Detail

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Adding a circle and not an arc makes the problem easier for you because arc segments have a degree of complexity that is more difficult to achieve; designers are not as well trained in the placing of arcs. Over the years, computer aided designers have learned to draw circles and trim out unwanted sections.
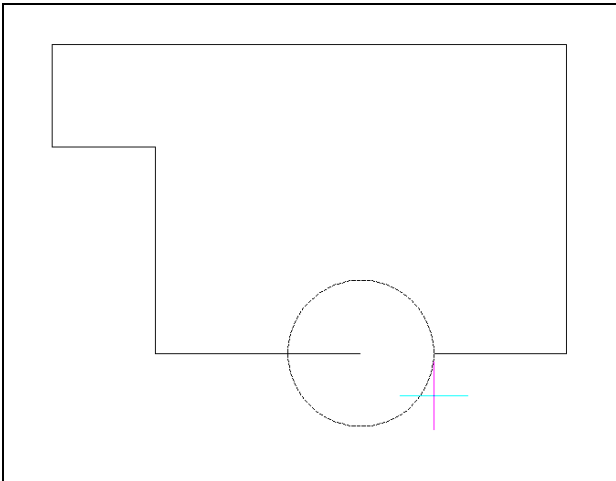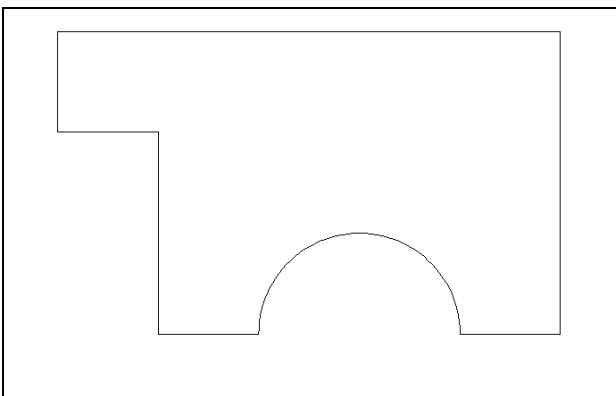

Select the Circle command on the Draw toolbar. Place the mouse aperture in the middle of the horizontal line as shown in Figure 7.6 and a Midpoint Esnap symbol (triangle) will appear in the middle of the line. Select the center point of the circle by selecting with the left mouse button. Then type "**10**" when the command line prompts you to specify the "Diameter/<Radius>:". Now the circle will appear; the subsequent procedure is to remove the outside hemisphere of the circle by using the Trim command.

Figure 7.6 –Adding a Circle to the Solid

Trimming the Circle and the Extra Line

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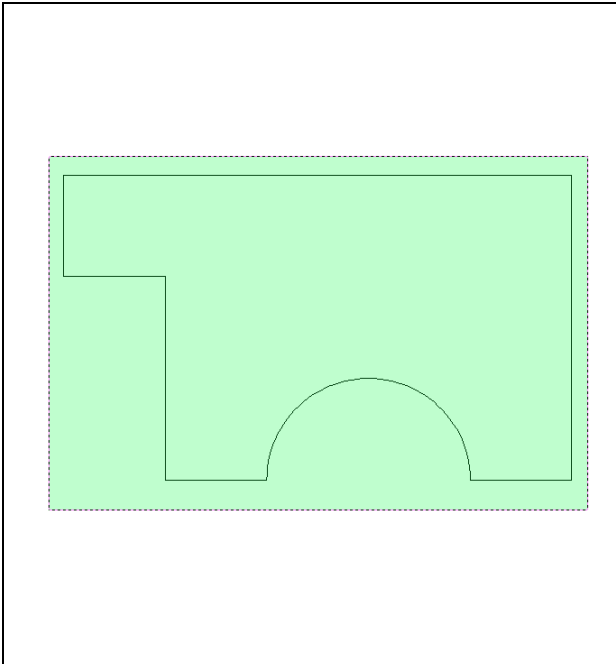

Select the Trim command on the Modify toolbar and hit ENTER to automatically select all lines on the graphical screen as cutting lines. Then proceed to pick the segment of the circle outside of the box as shown in Figure 7.7. You will notice the system removes the arc slice that was outside the line. You will also want to trim the line extending across the diameter of the arc so that your part appears as shown in Figure 7.7.

Figure 7.7 –Trimming the Circle

This shape contains eight entities: seven line segments and one arc segment. You can only extrude the shape after all eight components are brought together using the Edit Polyline tool.

Creating Polylines Using the Edit Polyline Tool

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To merge the seven lines and the single arc segment together, you need to use the Edit Polyline tool. Remember, the tool allows you to merge or join separate entities into one.

Select the Edit Polyline tool on the Modify II toolbar and the command line will prompt you to "**Select polyline to edit:**". Select any of the line segments and the command line will continue with "**Edit polyline:**" followed by a number of different possible actions. Type **J** and hit ENTER, and you will be asked to "Select entities to join." Use a selection window to select all of the entities of the shape and the command line will respond with "Entities in set: 8". Hit ENTER and the command line will display "Polyline is now closed. 6 vertices added to polyline." Your collection of line segments is now a complete polyline; hit Escape to exit the Polyline Edit tool, and then to check your work select the new polyline. It should become highlighted as a single entity.

Figure 7.8 – Creating a Polyline

Drawing a Closed Polyline Shape to be Extruded

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On the Draw Toolbar, select the Polyline tool and specify a start point in the lower left hand corner of the graphical display, to the left of the first shape. In Figure 7.9, you can see the actual continuous line of data you need to specify at the command line. If progeCAD won't let you move your next line in the direction you want, try opening the Esnap Settings window and closing it again without changing anything; this seems to fix the problem.

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Command : _POLYLINE
ENTER to use last point/Follow/<Start of polyline>:
Arc/Distance/Follow/Halfwidth/Width/<Next point>: 30
Arc/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 11
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 5
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 5
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 5
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 8
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 5
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 5
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Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **5**
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **11**
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **30**
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: **C**

Figure 7.9 – Command Line Prompts

There are twelve line segments in the Polyline and the dimensions that are shown in the sketch in Figure 7.1. Little by little you should be reading from the sketch more and relying less on the step by step outline in these chapters. Through practice, you will improve in your ability to "read" the engineering isometric drawing.
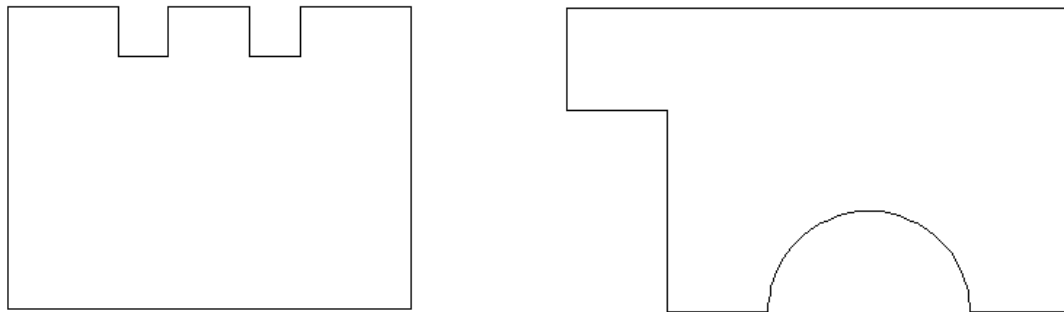

Figure 7.10 – The Finished Polyline

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 two shapes that are single entities.
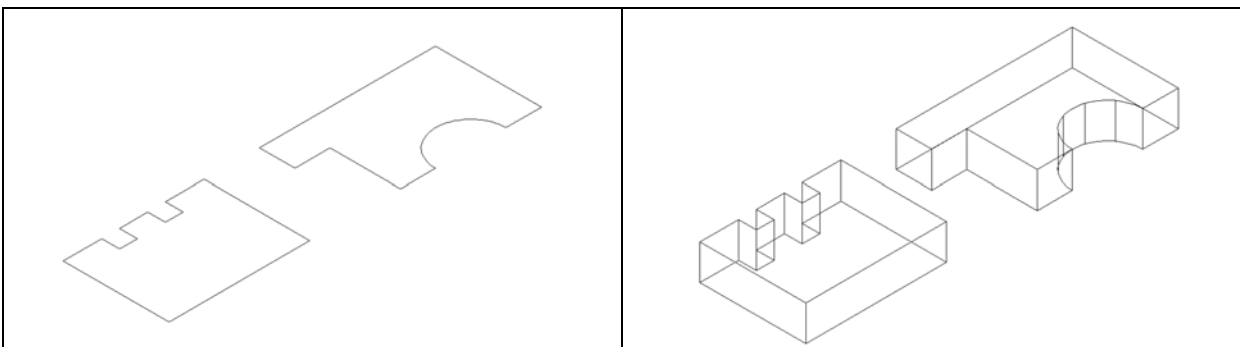

Figure 7.11 – Selecting to Extrude   **Figure 7.12 – Extrude the Entity**

On the View toolbar, select the Southwest Isometric View tool to allow you to see the part on the XY plane with the vertical Z-axis rising vertically as shown in Figure 7.11. 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 and going to the left you will see the dotted selection window signifying the "crossing" option in selecting entities. Select both

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shapes and the command line will return with "**Entities in set: 2**". Hit **ENTER** to specify the height or path of the solid. Type "**10**" for the height of the extrusion and hit **ENTER** to keep the angle of taper at the zero degree default. The shapes will extend 10 units into the z-axis as shown in Figure 7.12.

Rotating a 3D Solid Using the Rotate3D and Rotate Commands

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You can rotate the solid to the left on the X-axis using the Rotate3D command. Select Modify from the Menu Bar, then 3D Operations, and pick Rotate 3D from the list of commands. At the command, "Select entities to rotate:", pick the left 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 to draw a line on the x-axis. Type **90** to rotate the solid as shown in Figure 7.32.
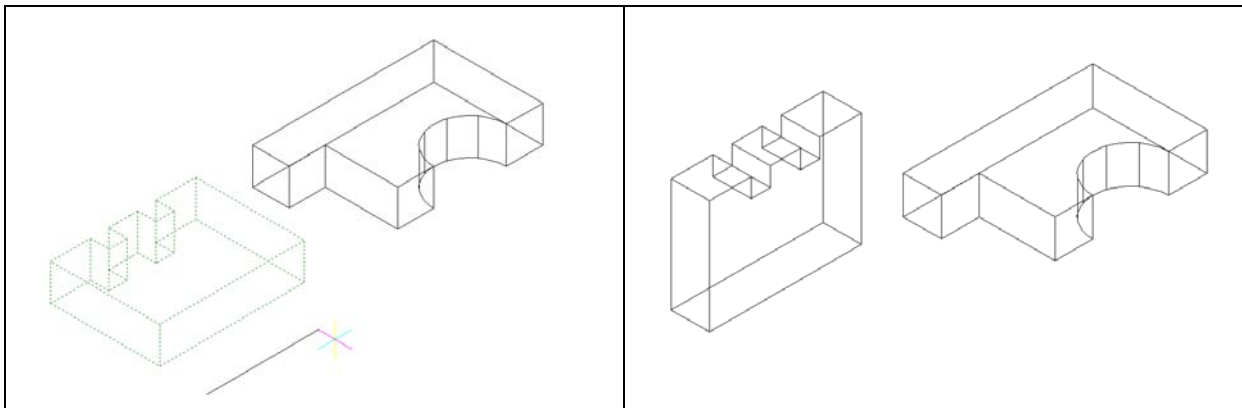

Figure 7.14 – Setting the Rotation Axis   Figure 7.15 – The Rotated Solid
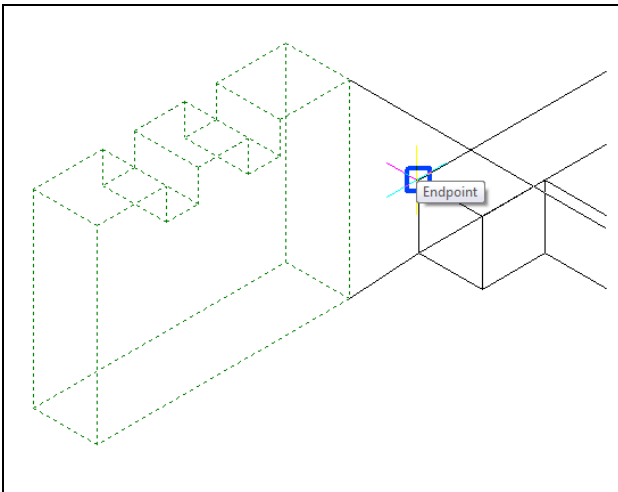
Align Solid Adjacent to the Master Solid

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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 solid on the left side graphical display and **ENTER** to proceed to the second part of the function.
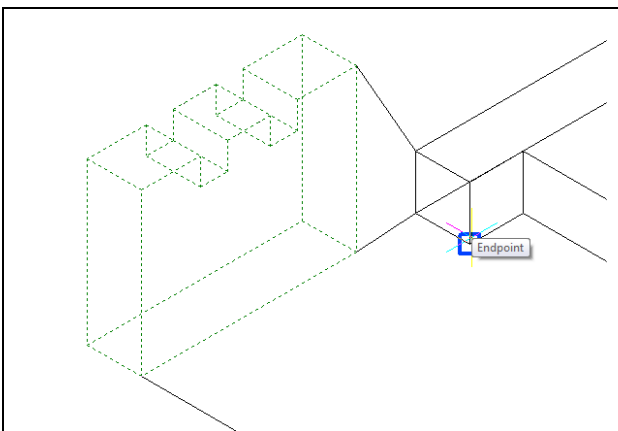
Figure 7.16 – The First Alignment Points
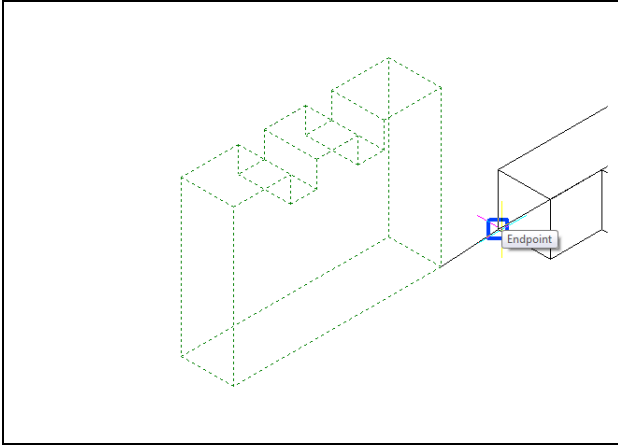
The command line will ask you to "Specify first source point:". Select the bottom right-hand corner of the left solid. When the command line prompts you to "Specify first destination point:" select the bottom left-hand corner of the small square face of the solid on the right. In the Align command, the first source point will always join with the first destination point so that the two will share the same coordinate. A temporary line will appear to show the relationship between the two points (Figure 7.16).

Figure 7.17 – Second Alignment Points

Next the command line will prompt you to select the second source and destination points. With the second and third sets of point selections you will designate the plane on which your two solids will become aligned. Select the second source point as the upper right-hand corner of the part on the left, and then the second destination point as the upper left-hand corner of the small square face of the part on the right. A temporary line will appear to designate the relationship between the two points (Figure 7.17).

Figure 7.18 – Third Alignment Points

Finally you are prompted to select the third source and destination points. Choose the bottom left-hand corner of the left solid as the source point and the bottom right-hand corner of the square face of the right solid as the destination point (Figure 7.18). Immediately the source part will move to the chosen alignment as shown in Figure 7.19.
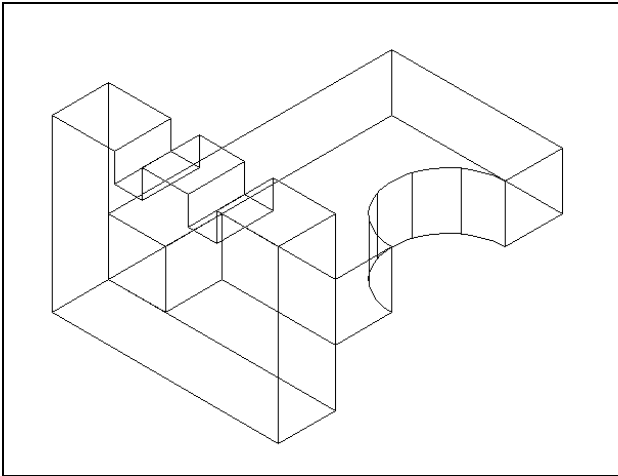
In your first exercise using the Align command you rotated the solid on the left using the Rotate3D command so you could easily see the function of the Align command. In the future, you can unite two solids, both moving and rotating them, without doing any previous setups. The challenge is to spot the relationships between the source and destination points, the two dimensional line of bonding, and the common plane. As you may suspect, this more advanced progeCAD command requires some practice.

Figure 7.19 – The Aligned Solids

Combining Solids with Union

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For the next modification of a 3D solid,, you need to use the Boolean command Union. With Union, you will join both shapes together.
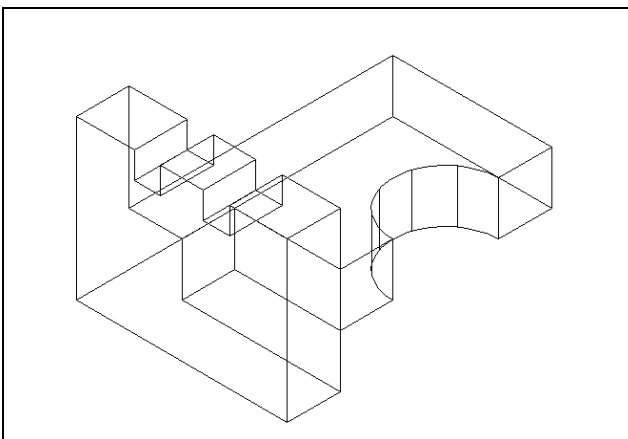

To add the two solids into one, select the Union tool on the Solids Editing toolbar. The program will prompt you to "Select ACIS objects to union:". Select both solids and the command line will respond with "**Entities in set: 2**". Hit ENTER and now the two solids will combine into one entity as shown in Figure 7.20. Select the solid with one pick of the mouse and the entire entity will become highlighted.

Figure 7.20 – The Combined Solids

Hide the 3D Solid

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In Figure 7.20, 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 material unless it was transparent. You can use the Hide command to remove the hidden lines from your graphical display.
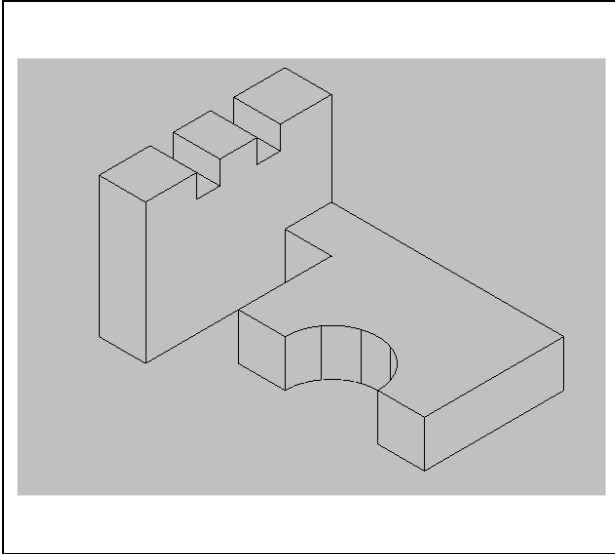
Select the Hide tool on the Shade toolbar. This is a very simple command because after picking the tool the tessellation lines representing the back of the part are removed from view as shown in Figure 7.21. If you wish to return the graphical display to the previous setup in which all the tessellation 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 7.21 – The Shaded Solid

Moving the Solid to the Origin Point

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To move Problem Six 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 7.22 at the lower left-hand side of the Problem Six, 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 SIX will move to the new origin point. Try using Zoom Extents if the part completely escapes your viewing area.

* World Class CAD Challenge 101-06 * - 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.
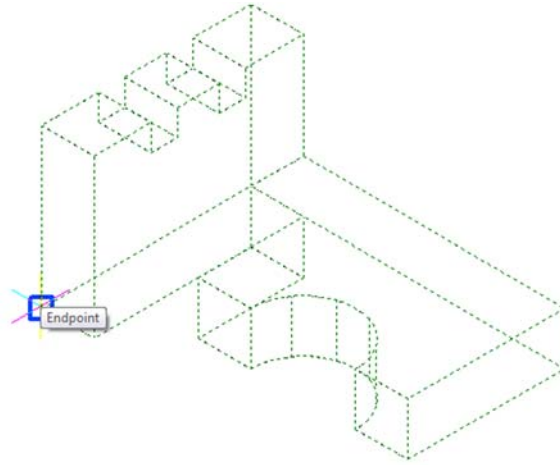
Figure 7.22 – Moving the Finished Solid to the Origin

Saving the Solid Problem

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To save Problem Six, 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 6" and press the Save button to save the drawing (Figure 7.23).
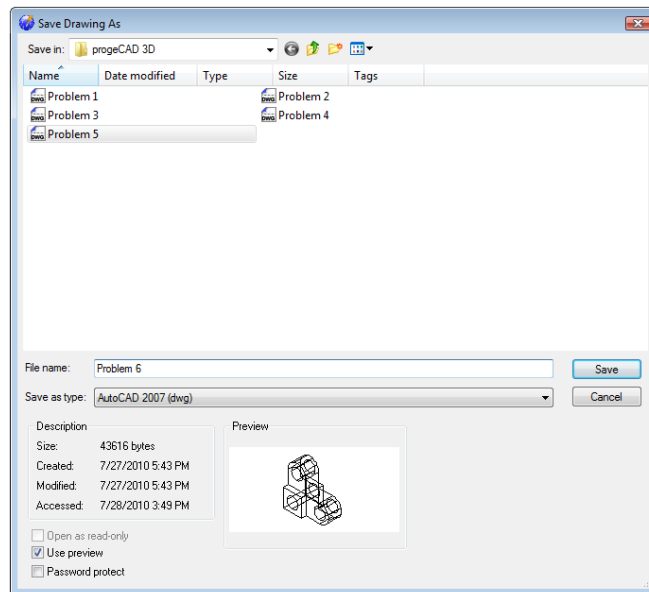

Figure 7.23 – Saving Problem Six

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