Solid Problem Six

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

- Sketch of Solid Problem Six
- Starting a 3D Part Drawing
- Modifying How the UCS Icon is Displayed
- Constructing a Solid Perimeter Using the Line Command
- Adding a Circle to the Detail
- Trim the Circle and Erase Extra Line
- Creating Polylines Using the Edit Polyline Tool
- Drawing a Closed Polyline Shape to be Extruded
- Extruding a Solid from Closed Polylines
- Rotating a 3D Solid Using the Rotate3D Command
- Align a 3D Solid Using the Align Command
- Union the Solids
- Hide the Solids
- Moving the Solid to the Origin Point
- 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. Convert the intricate silhouette to a closed Polyline. Next, you will draw a closed Polyline containing just line segments. After extruding the two Polylines, align the two shapes together and union them. Your three dimensional drawing will appear as shown in your sketch in Figure 7.1.
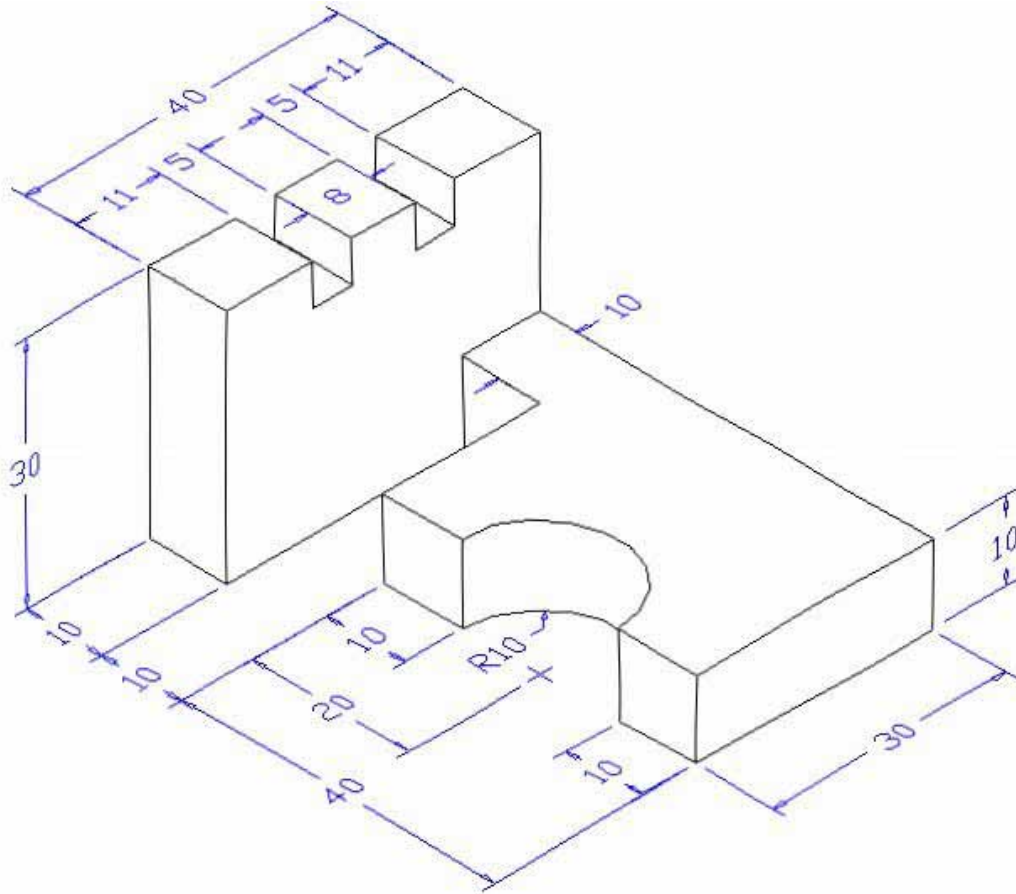

Figure 7.1 – Problem Six Sketch

By this point in your three dimension 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 where there are different complexities and 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

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In Problem Six, 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.
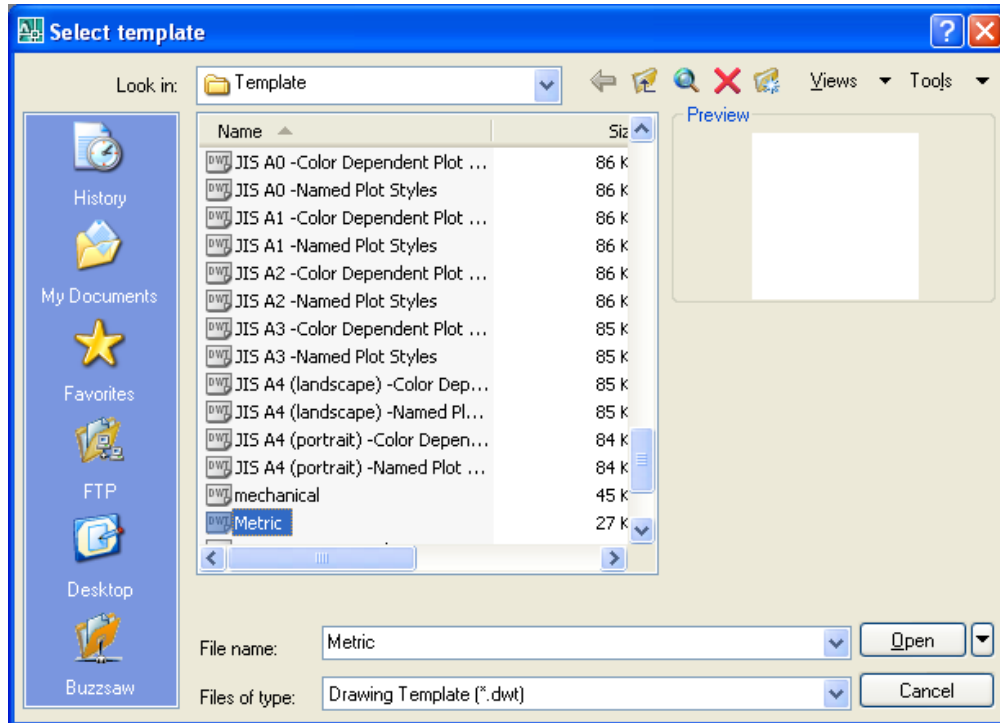

Figure 7.2 – Starting the Drawing Using the Metric Template

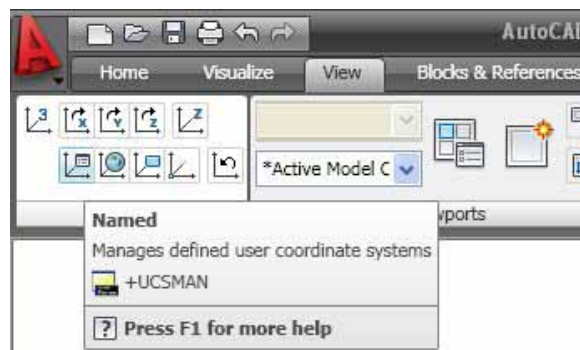# Modifying How the UCS Icon is Displayed

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Figure 7.3 - 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 addition 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 Named tool on the Coordinates panel under the View tab and the UCS window will appear in the graphical display. Pick the Settings tab to change the UCS Icon settings. Remove the check from "Display at UCS origin point" as shown in Figure 7.4 and press the OK button to have the system variable change. Now you can start to create the solid.
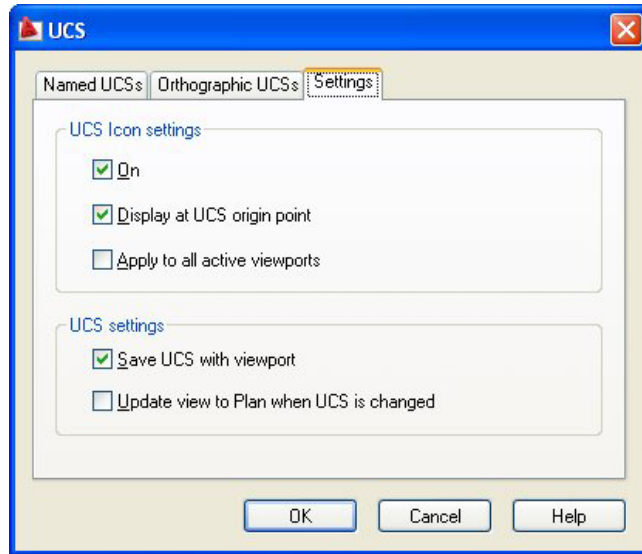

Figure 7.4 - The UCS Window, Settings Tab

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 panel. 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.
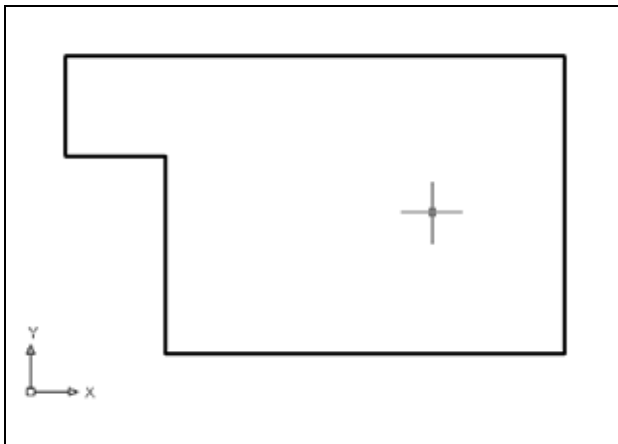

Move the line being drawn on the graphic screen to the right and type "40" and Enter at the keyboard. To draw the vertical line, move the mouse upward, dragging the line and type "30" and Enter. Drawing the third line, move the mouse to the left, dragging the line and type "50" and Enter. On the fourth 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 the lines and not using the Polyline function is enabling 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 since arc segments have a degree of complexity that is more difficult to achieve and designers are not as well trained in the placing of arcs. Over the years, computer aided designers learn to draw circles and trim out unwanted sections.
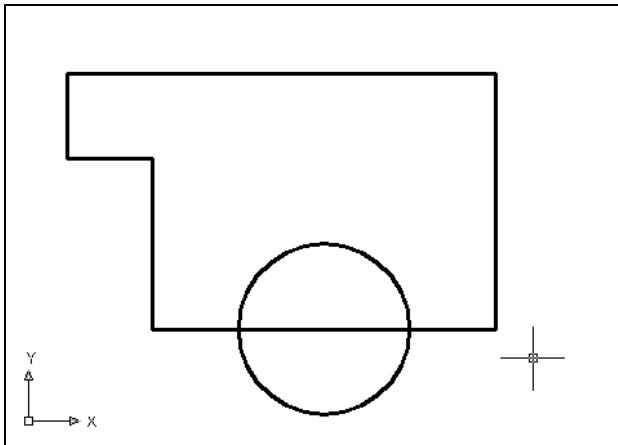
Select the Center, Radius command on the Draw panel. Place the mouse aperture midway on the horizontal line as shown in Figure 7.6 and a Midpoint Osnap symbol (triangle) will appear in the middle of the line. Select the center point of the circle by selecting with the left mouse button. Type "**10**" when the command line prompts you to "specify radius of circle". 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

Trim the Circle and the Extra Line

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The next step is to remove the outside arc segment of the circle to create a box with a half circle inscribed, then to erase the line passing through the circle. By accomplishing both of these sequences, you will be ready to join the individual entities into a special Polyline.
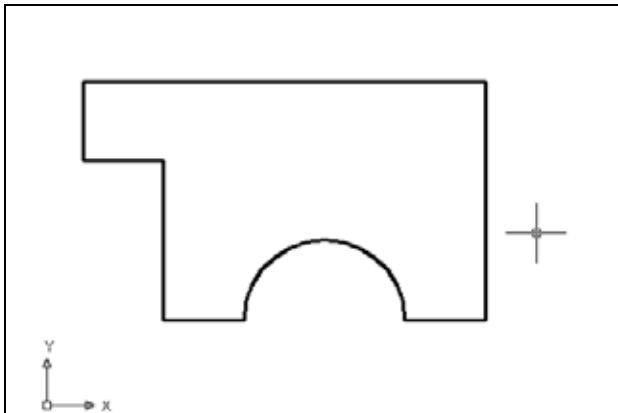
Select the Trim command on the Modify panel and hit Enter to automatically select all lines on the graphical screen as cutting lines. Proceed to pick the radius 10 circle segment on the right as shown in Figure 7.7. You will notice the system removes the arc slice that was outside the line. Hit Enter to reselect the trim command and click on the line hit Enter. This will remove the line as shown in Figure 7.7.

Figure 7.7 –The Trimmed 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 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. Pick to the upper right of the graphical display as shown in Figure 7.8 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.
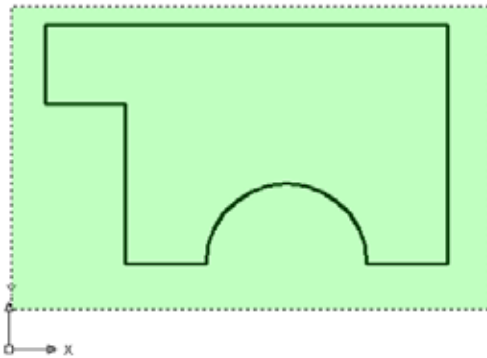
The command line will return with "Do you wish to turn the line into a Polyline <y>". Press Enter to convert a line to a Polyline and at the next prompt, type "J" and Enter for join. Reselect the lines and all adjoining lines and arcs will become a Closed Polyline. When asked for "fuzz distance or [Jointype]," just right click and select cancel or press Enter until the command is complete and the lines will still be joined.

Figure 7.8 – Polyline Edit the Right Side

Drawing a Closed Polyline Shape to be Extruded

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On the Draw panel, 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.

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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> 40
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 30
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 11
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 5
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 5
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 5
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 8
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 5
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 5
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 5
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 11
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: C
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Figure 7.9 – The Polyline Text from the Command Line

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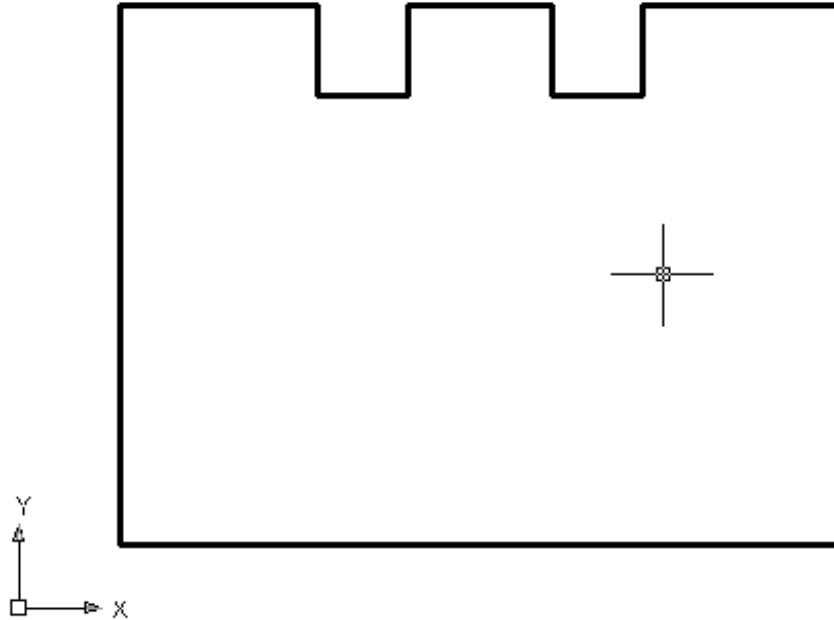

Figure 7.10 – The Completed Polyline

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.
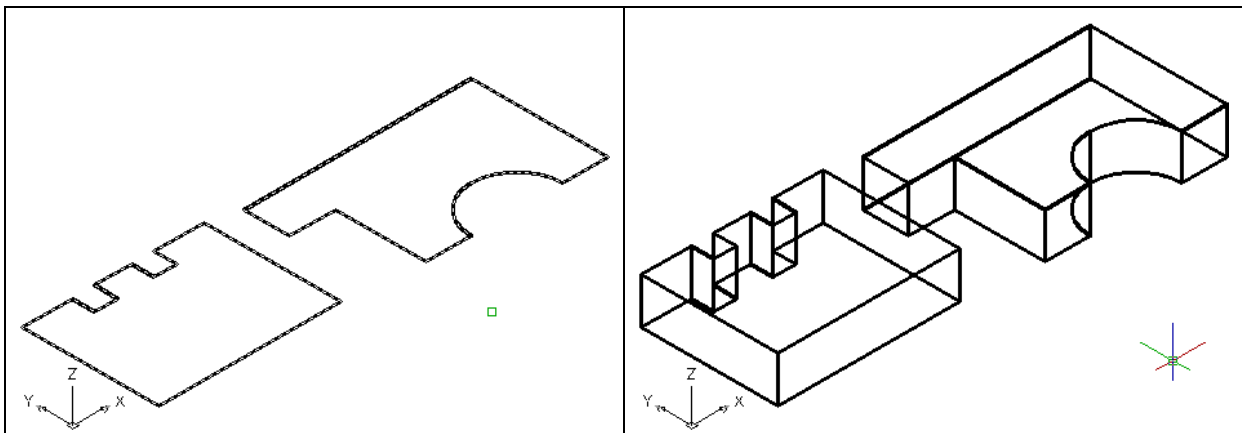

Figure 7.11 – Selecting to Extrude   **Figure 7.12 – Extrude the Entity**

On the View 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 upward as shown in Figure 7.11. Select the Extrude tool on the Modeling panel and the command line will prompt you to "**select objects**". Pick 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. The command line will return with "**2 found**". Press **Enter** to specify the height or path of the solid. You will type "**10**" for the height of the extrusion and for the angle of taper, hit **Enter** to accept the zero degree default. Two complex 3D solid will appear in the graphical display as shown in Figure 7.12.

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

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You can rotate the solid to the left on the X-axis using the 3D Rotate command. The 3D Rotate command is found on 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 perpendicular to the x-axis. Type **90** to rotate the solid as shown in Figure 7.32. You may want to move the part back along the y-axis before continuing to the Align tool.
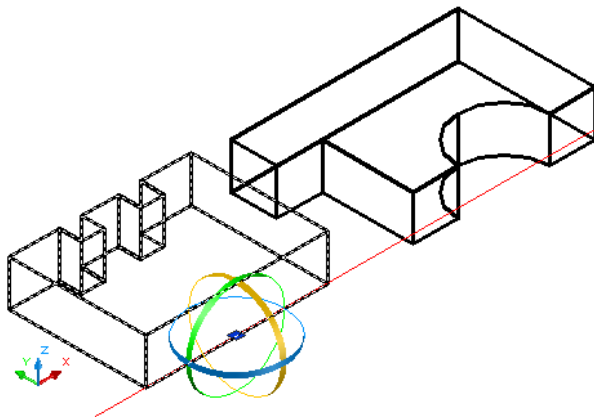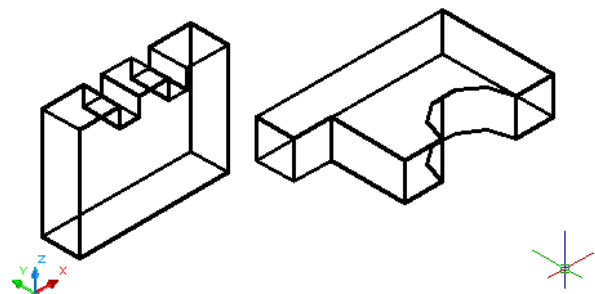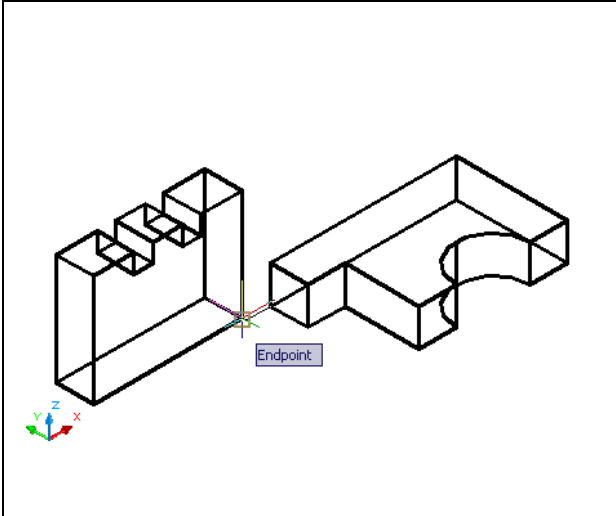

Figure 7.14 – Rotate3D the Solid   **Figure 7.15 – The Rotated Solid, moved back**
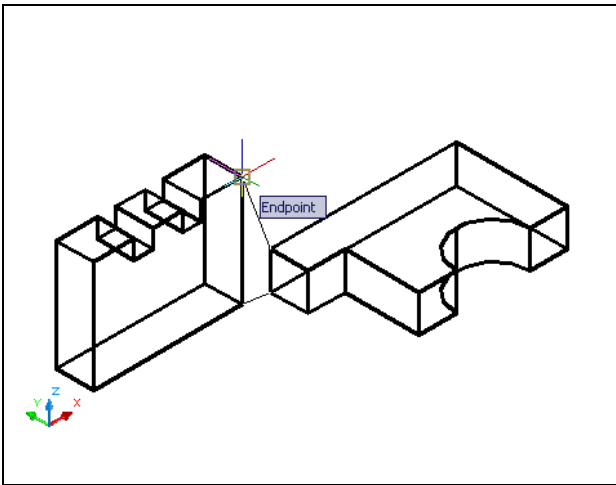
Align Solid Adjacent to the Master Solid

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Select Align on the drop-down arrow of the Modify panel. At the command, "**select objects**", pick the right solid on the graphical display and hit Enter to proceed to the second part of the function.
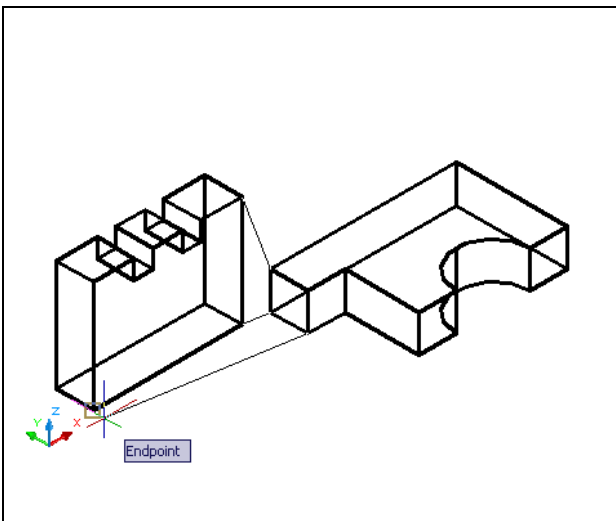
Figure 7.16 – The First Alignment Points

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. Unless the solids are exactly the same size this will always be true. This makes the first selection the most important, since you will only want to run the command once. In Figure 7.16, the first point of alignment is the endpoint on the right lower corner of the solid on the right. The destination is the endpoint on the lower right corner of the solid on the left. A temporary line appears on the display showing the connection.

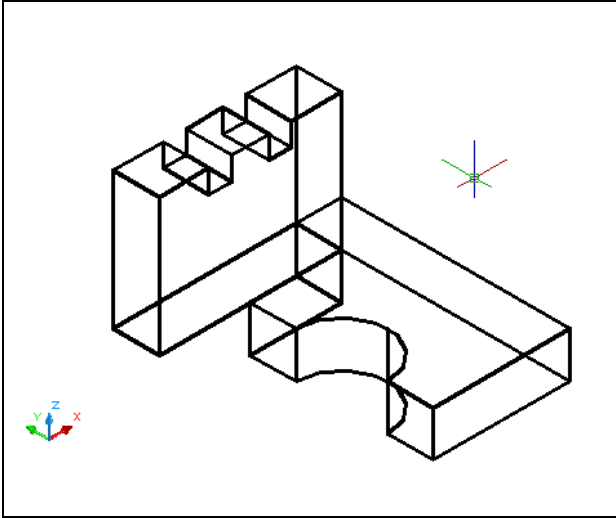**Figure 7.17 – Second Alignment Points**

The second set of points will signify the two dimensional line of alignment. You can see in Figure 7.17 that the first pick is the upper right corner of the solid on the right. The second selection is the upper right corner of the solid on the left. The sides are different in height but these two tessellation lines will unite. The exact points along this line will be the bottom selections from group one. If you were doing a two-dimension drawing, this would be your last set of picks, and by Entering through the last prompts, you would make a 2D alignment.

Figure 7.18 – Third Alignment Points

The last group of points you will select will align the planes of both solids. You can see in Figure 7.18 that the first pick is the lower right corner of the solid on the right. The second selection is the lower left corner of the solid on the left. As in each set of associations, a temporary line appears on the graphical display showing the relationship. These last two picks will force an arrangement between the two planes that you are showing by the selection of three points. You may remember from previous math classes that any three points will represent a plane.

Figure 7.19 – Rotate3D the Solid

After the last selection, the first solid will pull to the second destination solid as shown in Figure 7.19. In your first exercise using the Align command, you rotated the solid to the left using the 3D Rotate command so you could easily see the arrangement using 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 exact points of unity, the two dimensional line of bonding and the common plane. As you may suspect, this more advanced AutoCAD command requires some practice.

Union the Solids

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


Figure 7.20 – Union the Solids
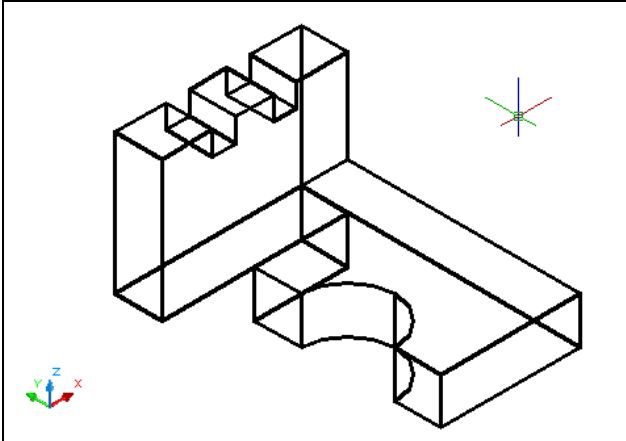
To add the two solids to another one, select the Union tool on the Solid Editing panel. AutoCAD will prompt you to "**Select objects**". Select both solids and the system will respond with "**2 found**". Hit Enter and all the exterior solids will become one entity as shown in Figure 7.20. You can see 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 will be highlighted.

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 unless the material was transparent. You can use the Hide command to remove the hidden lines from your graphical display.
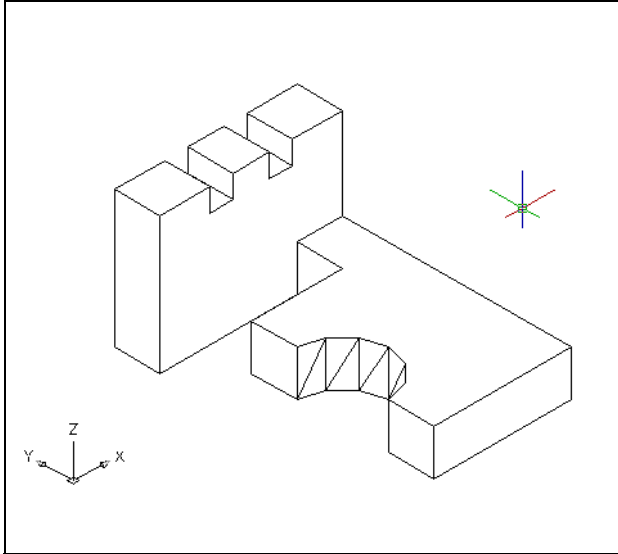
Select the 3D Hidden tool on the View panel. This is a very simple command, because moments after picking the tool, the tessellation lines representing the back of the part are removed from view as shown in Figure 7.21. If you wish to return the graphical display to the previous setup where all the tessellation lines are seen, then choose the 3D Wireframe Visual Style on the View panel. 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 – Shade the 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 panel, select the 3D solid, and hit Enter. At the prompt, "**specify base point or displacement**", select the endpoint shown in Figure 7.22 at the left hand side of the problem six, which will become the problem six-insertion point into an assembly drawing. At the "second point of displacement", type 0,0,0 and Enter. Problem six will move to the new origin point. 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.
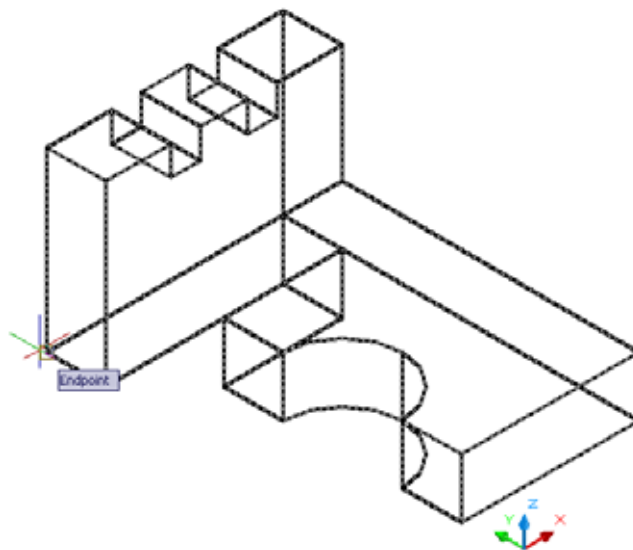

Figure 7.22 – Moving the Finished Solid to the Origin

Saving the Solid Problem

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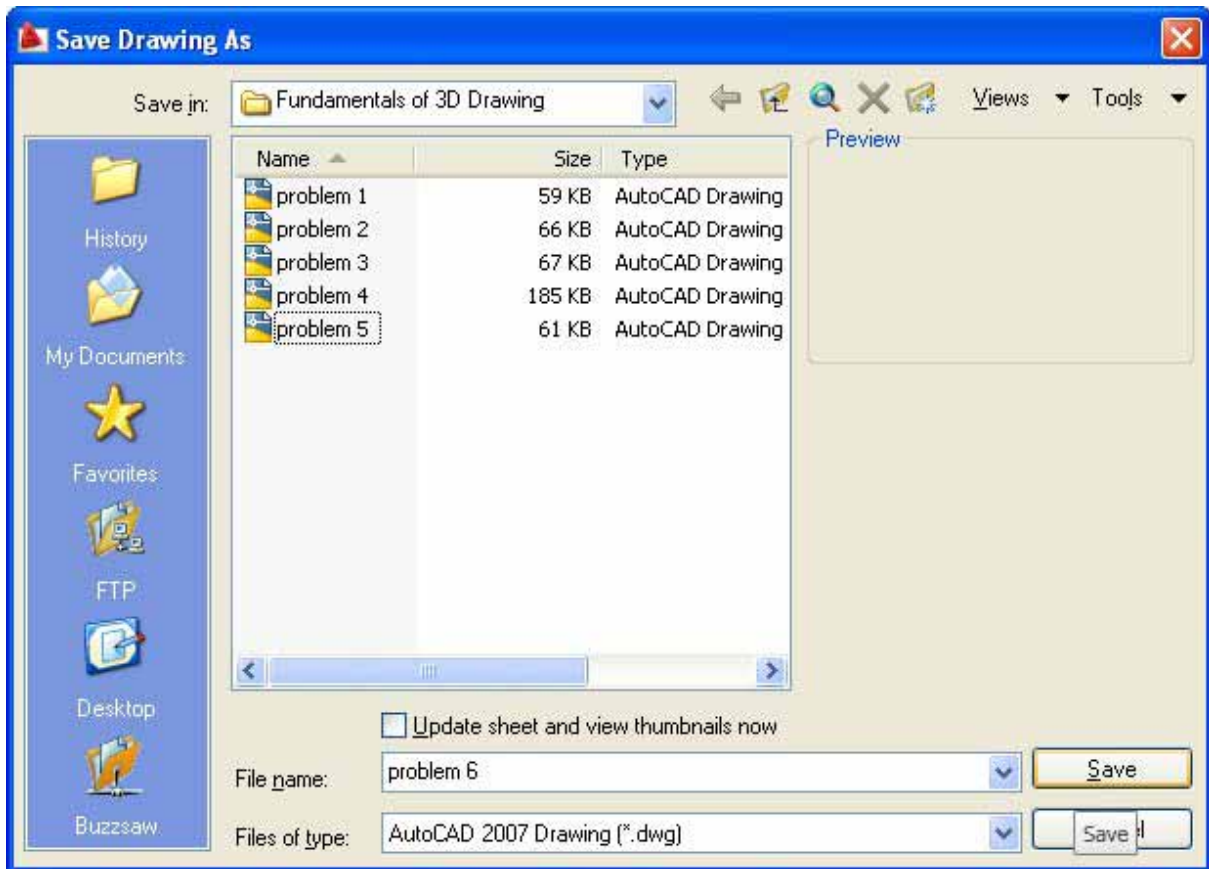

Figure 7.23 – Saving Problem Six

* 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 to obtain your world class ranking.