

## Solid Problem Six

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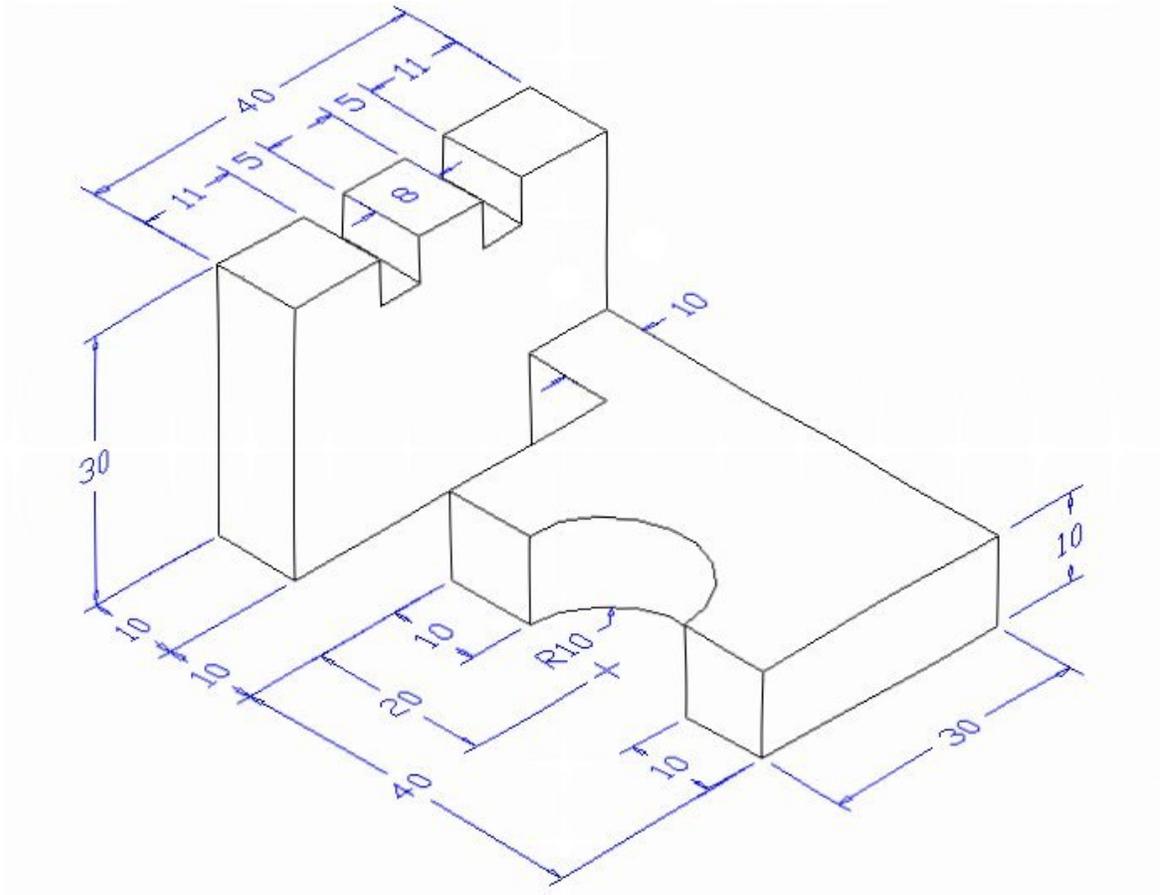
In this chapter, you will learn the following to World Class standards:

- Sketch of Solid Problem Six
- Starting a 3D part drawing
- Modifying UCS settings
- Constructing a solid perimeter using the Line command
- Adding a circle to the detail
- Trimming the circle and erasing 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 3D Rotate command
- Aligning 3D solids using the Align command
- Union of the solids
- Hiding the solid
- 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, you will draw lines, add a circle, and trim away construction lines to create a complex shape. Then you will convert the outline 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 combine them into one solid using Union. Your three dimensional drawing will appear as shown in Figure 7.1.



**Figure 7.1 – Problem Six sketch**

By this point in your 3D training, you should be comfortable with interpreting an isometric sketch. When you receive a sketch from a professional, quickly identify the different shapes and decide how to begin your drawing. Problem six has two different complex shapes to be drawn and brought together. You will be learning about the Align command in this problem, which will both move and rotate a solid simultaneously. All of the drills in the Fundamentals of 3D Drawing textbook will walk you through such common repetitive maneuvers that you will use often in 3D modeling.

## Starting a 3D Part Drawing

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You will draw Problem Six using the Metric template. First select the New tool on the Quick Access Toolbar. 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.

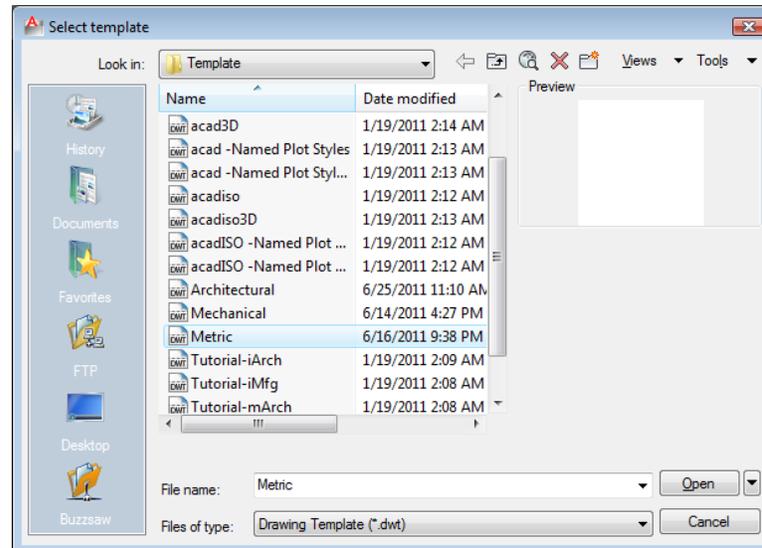
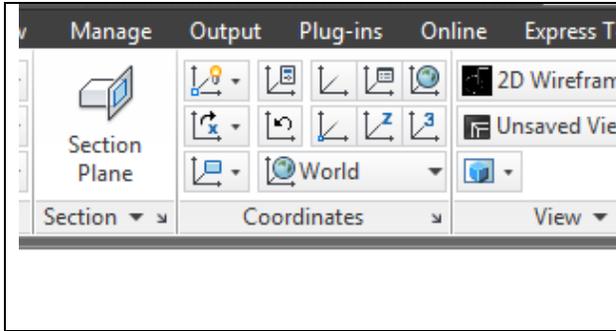


Figure 7.2 – Starting the drawing using the Metric template

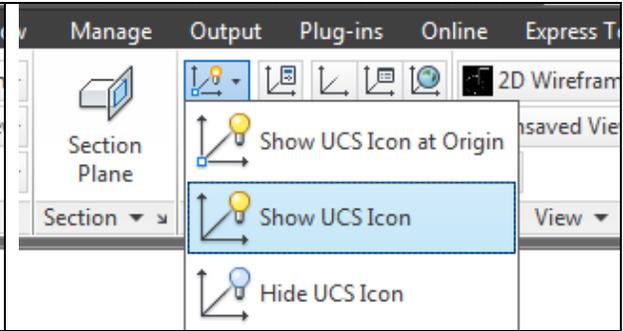
## Modifying UCS Settings

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You may notice the UCS icon moving on the graphical display as you zoom or pan your view. If so, the UCS is currently representing the origin (0,0,0) of the drawing area. This can sometimes be distracting, especially if the origin happens to lie in the vicinity of your drawing. You can change the UCS settings so that the icon is always displayed in the bottom left-hand corner of the drawing area. Select the Show UCS Icon tool from the Coordinates panel under the Home tab (Figure 7.3) and select the “Show UCS Icon” option (Figure 7.4). You also have the choice of keeping the UCS displayed at the origin or turning it off completely.



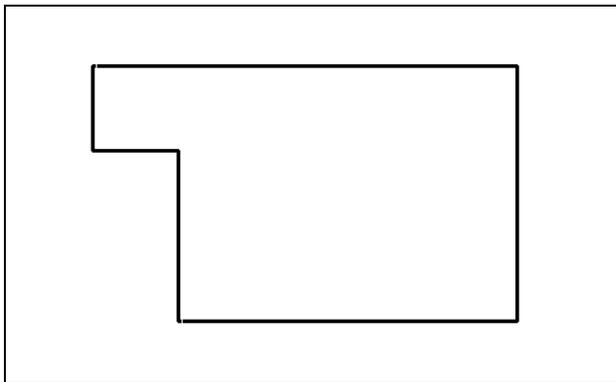
**Figure 7.3 - The Coordinates panel**



**Figure 7.4 - Changing the UCS display settings**

## Constructing a Solid Perimeter Using the Line Command

To begin you need to assemble a shape using the Line command. Select the Line tool from the Draw panel. Select a point in the lower right side of the graphical display. Toggle the Ortho mode to On to keep lines perfectly horizontal or vertical.



Move the cursor to the right and type “40” and ENTER at the keyboard. To draw the vertical line next, move the cursor upward and type “30” and ENTER. Now move the cursor to the left and type “50” and ENTER. For the fourth line, move the cursor down and type “10” and ENTER. Next, move the cursor back to the right and type “10” and ENTER. To close this shape, just type “C” and ENTER.

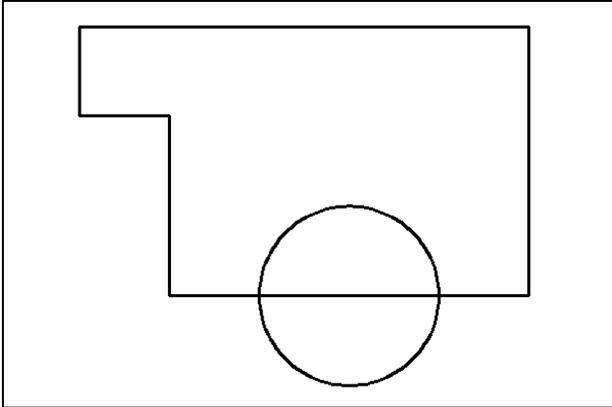
**Figure 7.5 – Drawing a series of lines**

Here you used the line command as opposed to the polyline command so that you can add an additional detail to the perimeter using the Circle tool. You will create the finished outline into a polyline using the Edit Polyline tool.

## Adding Circles to the Detail

Adding a circle as opposed to an arc makes the problem easier for you because the arc tool is rather complex. Many computer aided designers learn to draw circles and trim out unwanted sections to create arcs.



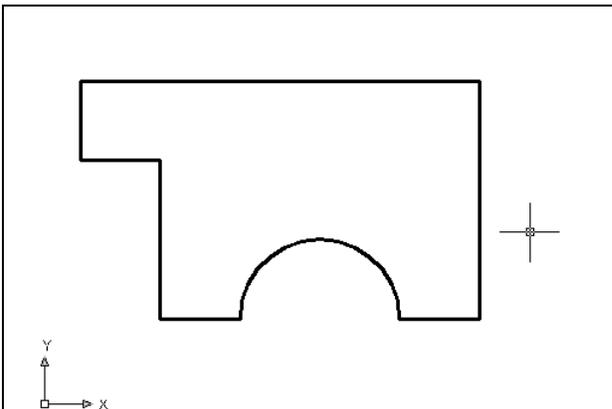


Select the Circle command from the Draw panel under the Home tab. Select the midpoint Osnap of the bottom side of the shape as the center point of the circle. Type “10” and **ENTER** when the command line prompts you with “specify radius of circle”. You have placed the arc as shown in the sketch, and now you only need to trim away the other half of the circle and the horizontal line away to finish the detail of this perimeter.

**Figure 7.6 –Adding a circle to the solid**

## Trim the Circle and the Extra Line

Now you need to remove the line segment passing through the circle and the outside arc segment of the circle to create a box with a half circle inscribed.



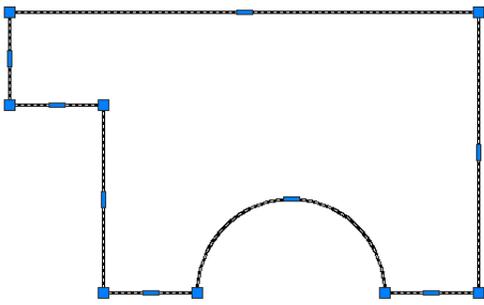
Select the Trim command on the Modify panel and hit **ENTER** to automatically select all lines on the screen as cutting lines. Select the line segment passing through the circle and the outside arc segment of the circle so that the drawing appears as shown in Figure 7.7. This shape now contains eight entities: seven line segments and one arc segment. You can only extrude the shape after bringing all eight components together using the Edit Polyline tool.

**Figure 7.7 –The trimmed circle**

## Creating Polylines Using the Edit Polyline Tool

Select the Edit Polyline tool from the drop-down arrow on the Modify panel and the command line will prompt you to “**Select polyline or [Multiple]:**” Select one of the lines of the perimeter and the command line will reply with “**Object selected is not a polyline Do you want to turn it into one? <Y>**”. Type **Y** and **ENTER**, and then the command line will prompt you with a number of editing options from which to choose. Type “**J**” and **ENTER** to choose the Join option.





The command line will prompt you to “Select objects” so select all of the objects in the outline of the polyline you wish to create, including the object you originally selected when you began the Edit Polyline command. The command line should read “**8 total**”. Hit **ENTER** to join all of the entities into a closed shape, and then hit ESCAPE to exit the Edit Polyline tool. You should now be able to select the entire closed polyline with one selection of the mouse as shown in Figure 7.8.

**Figure 7.8 – The closed polyline selected**

## Drawing a Closed Polyline Shape to be Extruded

Select the Polyline tool from the Draw panel and specify a starting 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 for this shape.



```
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
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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. At this point you should be able to read directly from the sketch and therefore rely less on the step by step outline in these chapters. Such practice will improve your ability to read isometric drawings.

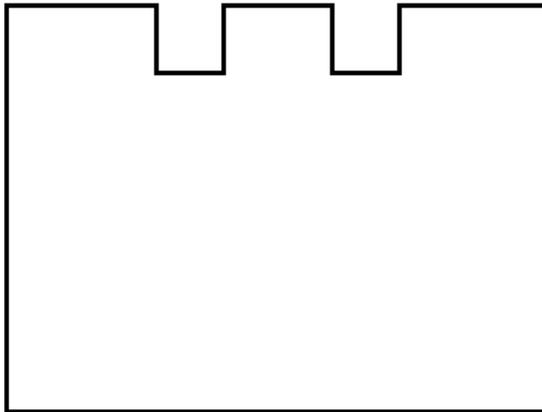


Figure 7.10 – The completed polyline

## Extruding Solids from Closed Polylines

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

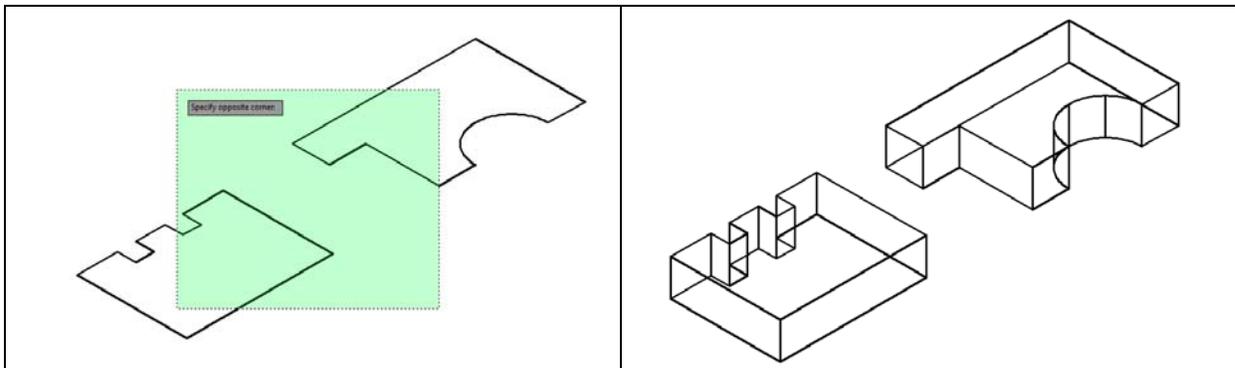


Figure 7.11 – Selecting to extrude

Figure 7.12 – Two extruded polylines

Select the Southwest Isometric View on the View panel 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 create dotted selection window signifying the “crossing” option, which will select any entities that come in contact with the selection window. The command line will return with “**2 found**”. Hit **ENTER** to specify the height or path of the solid. Type “**10**” and **ENTER** for the height of the extrusion. Two complex 3D solids will appear in the graphical display as shown in Figure 7.12.

## Rotating a 3D Solid Using the 3D Rotate Command

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You can rotate the second solid on the X-axis using the 3D Rotate command. Select the 3D Rotate tool from the Modify panel. When prompted to “**select objects**”, select the second solid and hit **ENTER** and then select the midpoint Osnap of the bottom edge of the front face of the solid as shown in Figure 7.14. You are going to rotate the solid part on the X axis, so select the circle that orbits the X axis as shown in Figure 7.14. Specify an angle start point, and then with the Ortho mode move the cursor so that the preview outline moves 90 degrees as shown in Figure 7.15. You may want to move the part back along the y-axis before continuing to the Align tool (Figure 7.15).

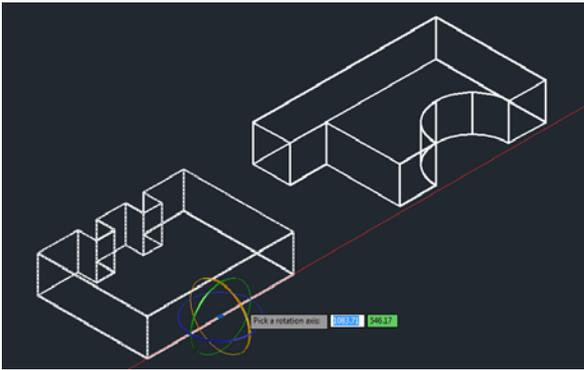


Figure 7.14 – Selecting the axis of rotation

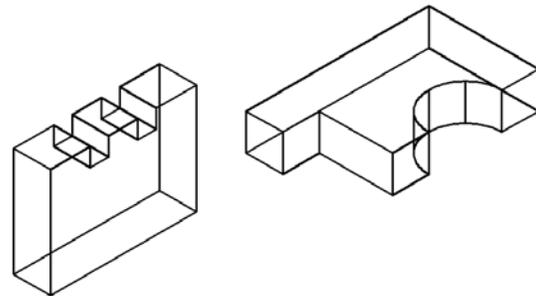


Figure 7.15 – The rotated solid, moved back

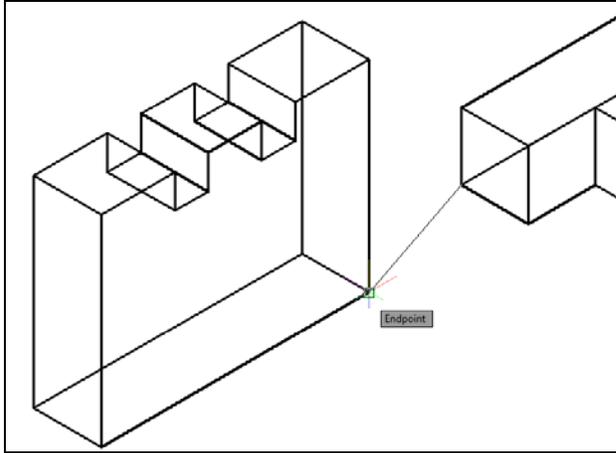
## Align Solid Adjacent to the Master Solid

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Select the Align command by expanding the Modify panel. At the command, “**select objects**”, pick the solid on the right and hit **ENTER** to proceed to the second part of the function. Notice that this is not the 3D Align command.

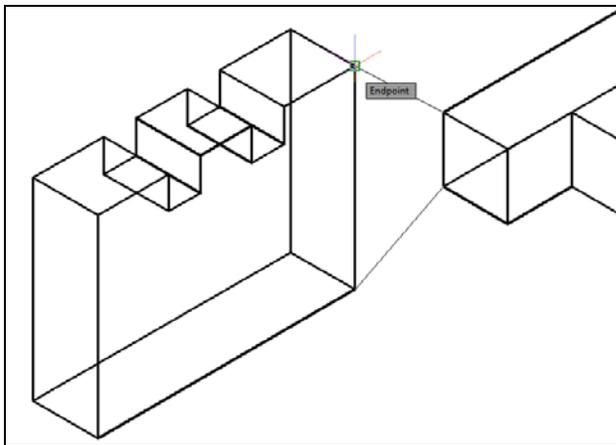


The Align command allows you to join the faces of two 2D or 3D objects. In this case you will be joining the small 10x10 face of the right solid to the bottom corner of the front face of the left solid, as is evident in the sketch of this problem. To accomplish the command you select six points: three source points and three destination points. These six points will define the two faces to be aligned with each other and the orientation of the source object.



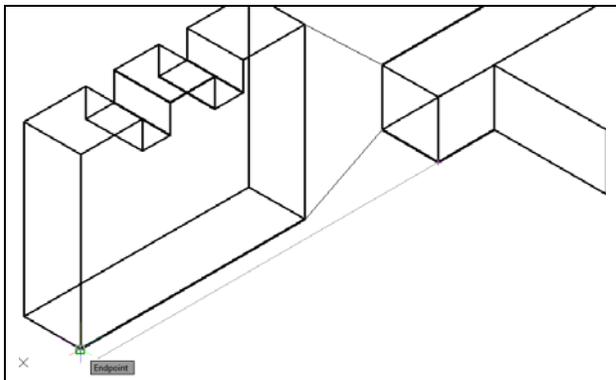
The first source and destination points will end up becoming one point in the finished alignment. In this case, select the bottom left-hand corner of the 10x10 face on the right solid as the first source point and then the bottom right-hand corner of the left solid as the first destination point. These two points will join to become the same point when the alignment is finished. Use endpoint Osnaps to select both, and a temporary line will appear on the screen showing the relationship of the two points (Figure 7.16).

**Figure 7.16 – The first alignment points**



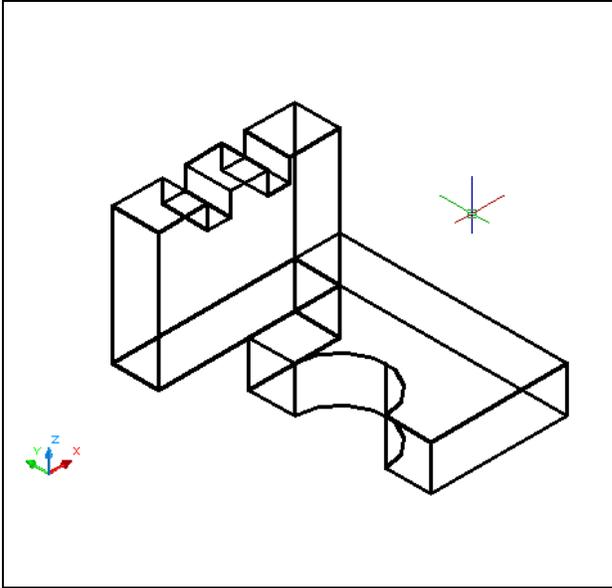
The second and third set of points you select will not end up becoming the same point, but they will reside on the same respective axes. For the second source point, select the upper left-hand corner of the 10x10 face of the right solid. Select the upper right-hand corner of the left solid as the destination point. Again a temporary line will appear connecting both points, showing their relationship in the alignment (Figure 7.17). When finished with the command these two points will reside on the same parallel line.

**Figure 7.17 – Second alignment points**



The third set of points will finalize the definition of the planes of the two shapes that you are aligning. Select the bottom right-hand corner of the 10x10 face of the right solid as the third source point. Select the bottom left-hand corner of the front face of the left solid as the third destination point (Figure 7.18).

**Figure 7.18 – Third alignment points**

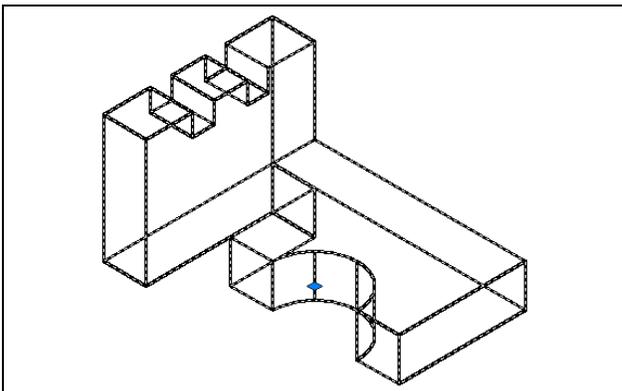


Immediately the right solid will move and rotate 90 degrees, aligning its 10x10 face with the front face of the left solid in the bottom right-hand corner (Figure 7.19). You may recognize the powerful nature of the Align command by its ability to both move and rotate objects with a few simple selections. However, you most likely also discovered that the difficulty of the Align command lies not in its complexity as a tool but in properly identifying the source and destination points necessary to create the desired alignment. This complex technique will require some practice but it can be mastered.

**Figure 7.19 – The completed alignment**

## Union the Solids

Now that you have both 3D solids in place, you are ready to join to create the final product. You will use the Boolean command Union to create the master solid.



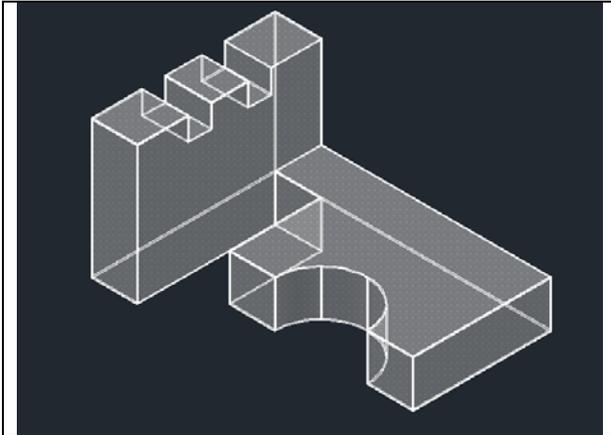
Select the Union tool on the Solid Editing panel under the Home tab. The command line will prompt you to “**Select objects**”. Select both solids and the command line will respond with “**2 found**”. Hit ENTER to combine both selected solids into one entity as shown in Figure 7.20. No apparent change will occur to the solid, but if you select the object with one pick of the cursor the entire solid will become highlighted.

**Figure 7.20 – The single master solid selected**

## Hide the 3D Solid

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To view a more realistic image of Solid Problem Six, select the Hide tool and the X-Ray Effect tool from the Visual Styles panel under the View tab.



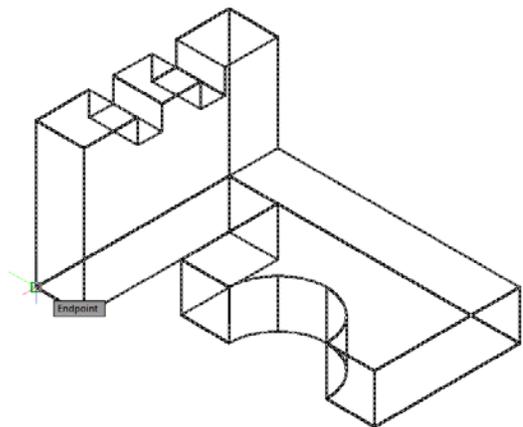
You will see the part as shown in Figure 7.21, with the visible faces and edges highlighted, but also the non-visible faces and edges still visible behind the part giving it a slightly transparent look. This combination of visual styles allows you to see the part in a more lifelike manner but still see the details not normally apparent in an opaque solid.

**Figure 7.21 – The X-Ray Effect**

## Moving the Solid to the Origin Point

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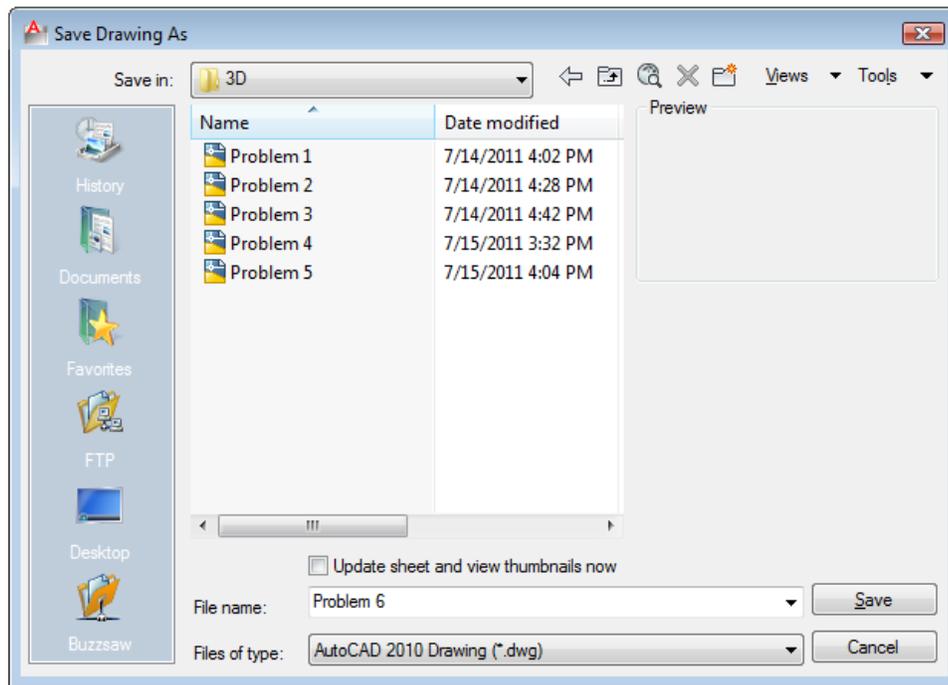
To move Problem Six to the drawing origin of 0,0,0 on the x, y and z axis, select the Move tool from the Modify panel under the Home tab, select the 3D solid, and hit ENTER. At the prompt “specify base point or displacement,” select the bottom left endpoint Osnap as shown in Figure 7.22, which will become its insertion point into an assembly drawing. At the “second point of displacement,” type 0,0,0 and ENTER, or #0,0,0 and ENTER if using Dynamic Input. Problem Six will move to the origin point. Moving a part at the origin is important for 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

To save Problem 6 in your Fundamentals of 3D Drawing Folder, select the Save As 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. Type “Problem 6” in the File Name textbox and hit the Save button to save the drawing (Figure 7.23).



**Figure 7.23 – Saving Problem Six**

**\* World Class CAD Challenge 03-06 \* - 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 in less than 5 minutes to maintain your World Class ranking.**

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