

Solid Problem Two

This chapter will cover the following to World Class standards:

- **Sketch of Solid Problem Two**
- **Starting a 3D part drawing**
- **Drawing a closed polyline shape to be extruded**
- **Extruding a solid from a closed polyline**
- **Rotating a 3D solid using the Rotate3D and Rotate commands**
- **Drawing a solid box**
- **Rotating and moving a solid onto the master solid**
- **Mirroring a 3D solid**
- **Subtracting a solid from another solid**
- **Improving the view by hiding tessellation lines**
- **Moving the solid to the origin point**
- **Saving the solid problem**

Sketch of Solid Problem Two

You will start the second solid problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across two axes, which you can call the x-axis and the y-axis, leaving the z-axis as the vertical rise of the piece. In this exercise, you will learn to draw a closed Polyline that contains a complex shape of the component and then use simple geometric solids to add or subtract detail. In problem two, you will form the upside-down “T” that contains the intricate contour into a Polyline shape, and then subtract the 63 degree wedge in the middle of the part.

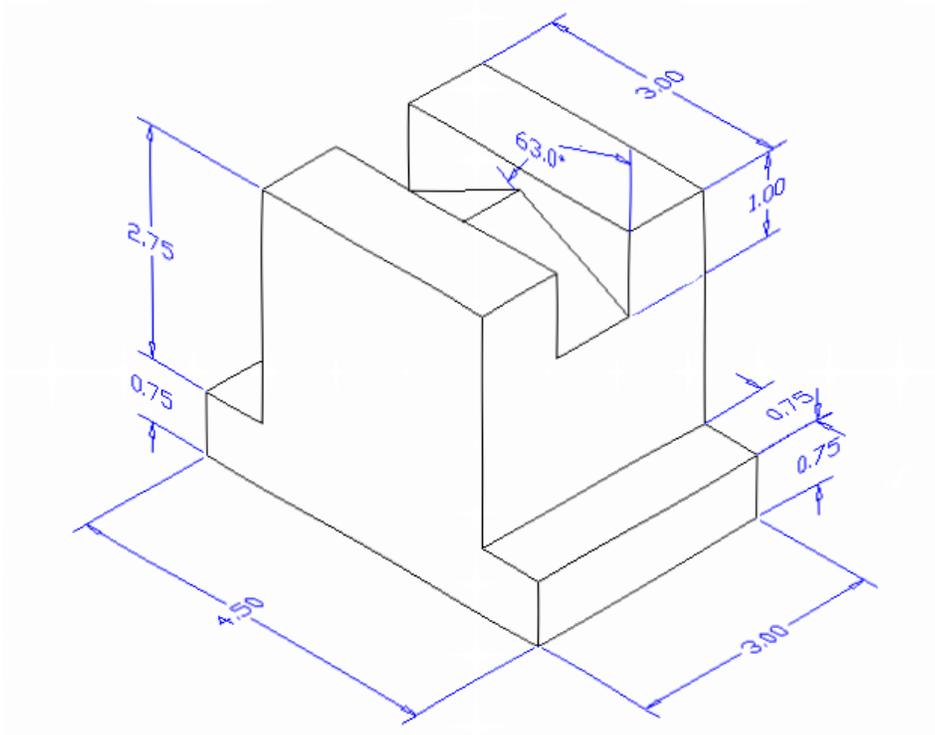


Figure 3.1 – Problem Two sketch

When you receive a sketch from a professional, quickly identify the complex shapes and decide how to begin your drawing. The first ten drills in the Fundamentals of 3D Drawing textbook will walk you through these common repetitive maneuvers that you will use often in 3D modeling.

Starting a 3D Part Drawing

You will draw Problem Two using the Mechanical 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 Mechanical.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.



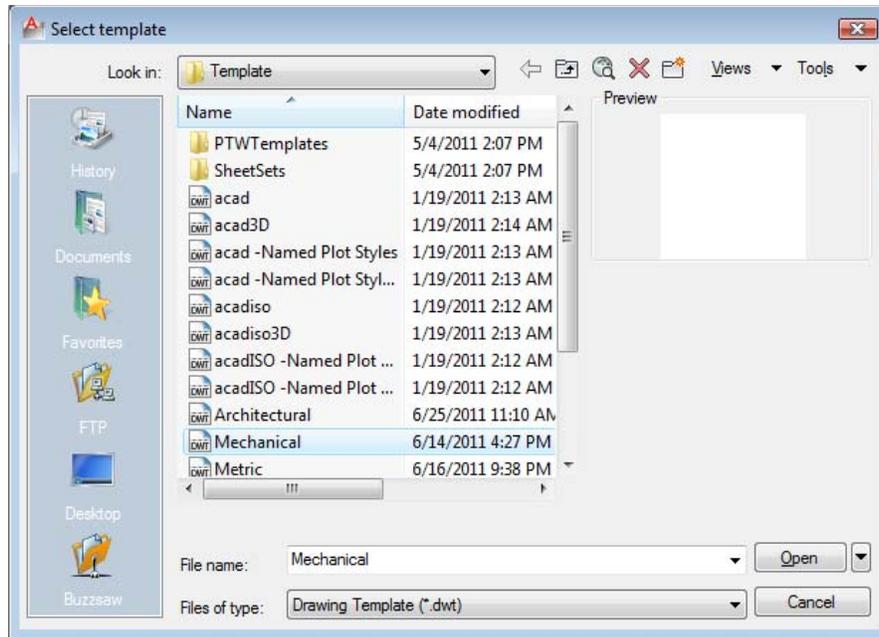


Figure 3.2 – Starting the drawing using the Mechanical template

Drawing a Closed Polyline Shape to be Extruded

You will draw the contour of this solid similarly to the way you created the outside wall of the architectural floor plan in the Fundamentals of 2D Drawing. A Polyline is not like a typical set of lines because the entity can contain multiple vertices identifying each line segment in the shape. Polylines can also contain arcs that form intricate smooth curves that will be useful in developing castings in advanced parts drawn in your future.



Select the Polyline tool from the Draw panel under the Home tab and specify a start point in the lower left hand corner of the graphical display. In Figure 3.3, you can see the actual continuous line of data you need to specify at the command line.

```

Command: _pline
Specify start point:
Current line-width is 0.0000
Specify next point or [Arc/Halfwidth/Length/Undo/Width]: 4.5
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 0.75
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 0.75
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 2.75
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 3
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 2.75
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: 0.75
Specify next point or [Arc/Close/Halfwidth/Length/Undo/Width]: C

```

Figure 3.3 – Starting the drawing using the mechanical template

With the Ortho mode “on,” draw a **4.5** unit line to the right, a **0.75** line up, a **0.75** line to the left, a **2.75** line upward, a **3.0** line to the left, a **2.75** line downward, a **0.75** to the left and then type “**C**” to close the Polyline. A closed entity will appear in the graphical display (Figure 3.4).

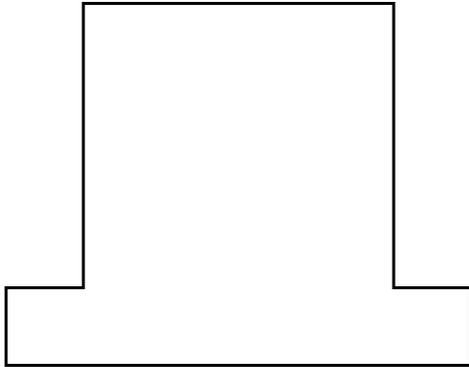


Figure 3.4 – Drawing a closed polyline

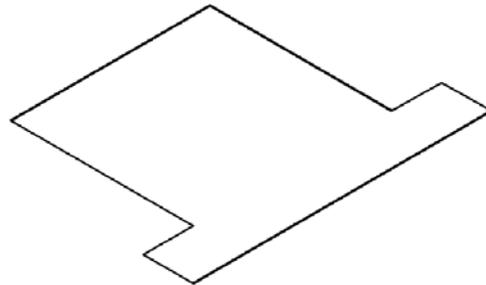


Figure 3.5 – Isometric view of the shape

On the View panel, select the Southwest Isometric view to see the part by looking at it from slightly above the XY plane. The Z-axis will rise vertically from the level surface (Figure 3.5).

Extruding a Solid from a Closed Polyline

Only closed entities like Polylines and Circles can be extruded into solids using this very powerful tool on the Modeling panel. If there is even a minute space in outline of the entity, the program will consider it a line and then extrude the outline to a surface rather than a solid.



Select the Extrude tool on the Modeling panel and the command line will prompt you to “**select objects.**” Pick the Closed Polyline anywhere on its outline and the command line will return with “**1 found**” as shown in Figure 3.6. Hit **ENTER** to specify the height or path of the solid. You can type “**P**” for Path and choose an entity to along which to extrude the shape, but you will not want to choose that option for problem two. Type “**3**” and **ENTER** for the height of the extrusion. The closed outline of the polyline will expand into the Z axis 3 units, creating a complex 3D solid as shown in Figure 3.7.

```
Command: _extrude
Current wire frame density: ISOLINES=8, Closed profiles creation mode = Solid
Select objects to extrude or [MOde]: _MO Closed profiles creation mode
[SOLid/SURface] <Solid>: _SO
Select objects to extrude or [MOde]: 1 found
Select objects to extrude or [MOde]:
Specify height of extrusion or [Direction/Path/Taper angle/Expression]
<0.6079>: 3
```

Figure 3.6 – The extrude command

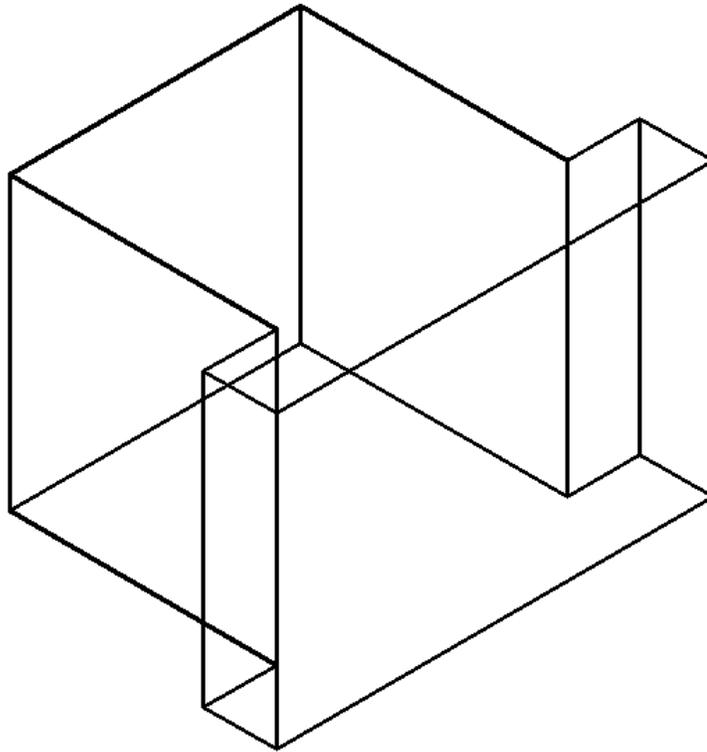


Figure 3.7 – Extruding the polyline shape

Rotating a 3D Solid Using the Rotate3D and Rotate Commands

There are four modifying commands used specifically for handling 3D components: 3D Array, 3D Mirror, 3D Rotate and 3D Align.



The 3D Array function opens the third tier of creating an array using rows and columns, but adding the third layer to copy new entities into the Z axis as well. The 3D Mirror tool allows any entity capable of 3D movement to cross any plane defined by three points. The 3D Rotate tool will rotate any solid or even two dimensional units around any axis defined by two points in Model Space. The most powerful tool that you can use is the 3D Align utility. This last function will move and rotate any object from the original position to the destination by drawing three paths from the starting point to their ending point. You will learn the Align command in a later exercise. It is probably the most conceptually difficult to grasp of the four functions. You can find the 3D Mirror, 3D Rotate, and 3D Align icons on the Modify panel.

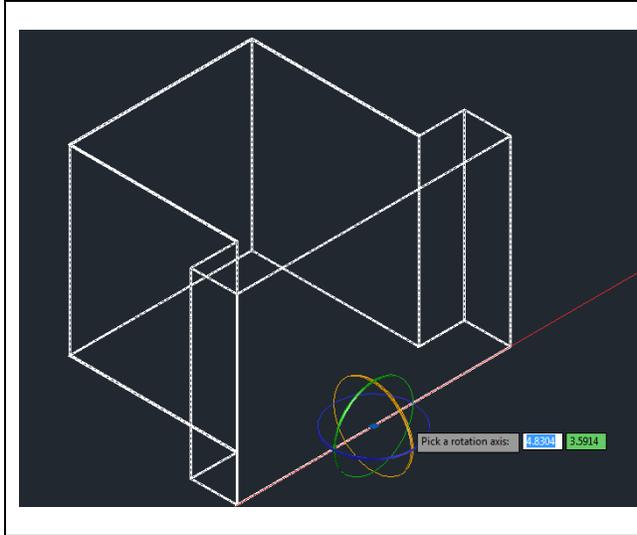


Figure 3.8 – Picking the Rotate3D axis

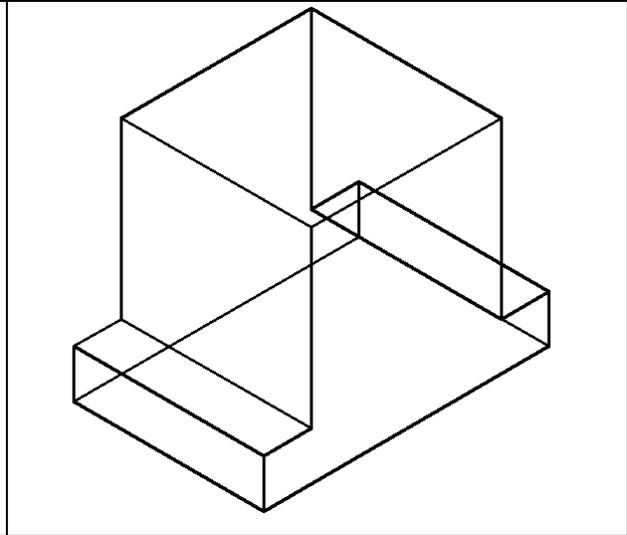


Figure 3.9 – The part rotated on the x axis

Select the 3D Rotate command from the Modify panel under the Home tab. The background will turn navy blue and the outline of the solid will turn white. At the prompt to “select objects,” pick the solid on the graphical display and hit ENTER to continue. Select the midpoint of the bottom right edge of the solid as the rotation point. You are going to rotate the solid part on the x-axis, so pick the diameter of the sphere that surrounds the x-axis. A red line will appear that runs through the bottom edge of the solid as shown in Figure 3.8. With the Ortho mode still “on” specify an angle start point by selecting a point in the bottom right hand corner of the screen. Type **90** to rotate the solid as shown in Figure 3.9.

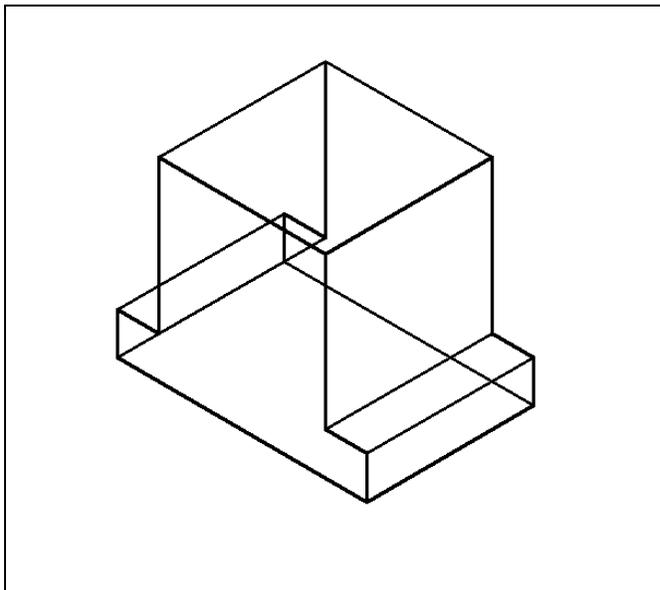
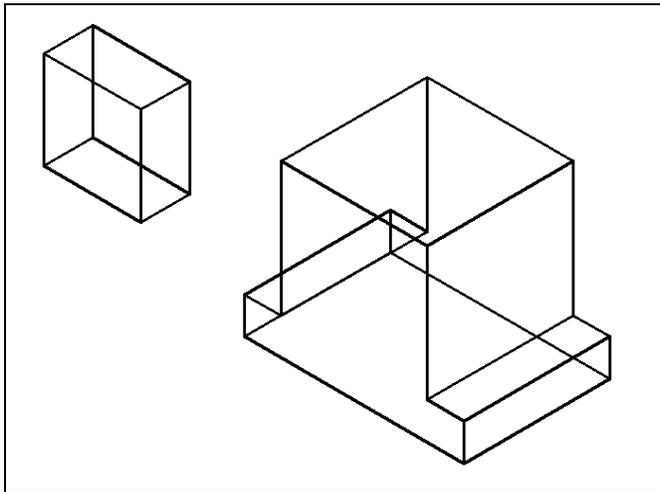


Figure 3.10 – Rotated on the Z axis

To rotate the solid in the XY plane, select the regular 2D Rotate command from the Modify panel. Select the solid and hit **ENTER** to select the base point of rotation. Select any point at the base of the solid with the left mouse button and then type 90 for the degrees of counter clockwise rotation to finish this detail (Figure 3.10).

Drawing a Solid Box

Now you will draw a box that you will move, mirror, and then subtract to create the detail on the top of solid problem two. Select the Box command from the Modeling panel under the Home tab. Type **L** after specifying the corner of the box to the left of the solid as shown in Figure 3.11.



Type **L** after specifying the starting point and the system will ask you three questions. The first prompt is to “Specify Length” so type **1** and **ENTER** for the X coordinate. The second prompt is “Specify Width” so type **2** for the Y coordinate and hit **ENTER**. The third prompt is “Specify Height” so type **2** and **ENTER** for the Z coordinate. The second box will appear as shown in Figure 3.11.

Figure 3.11 – Making a solid box

Rotating and Moving a Solid onto the Master Solid

Pick Rotate 3D from the Modify panel. At the prompt, “select objects,” pick the box you just drew on the graphical display and hit **ENTER** to proceed to the next part of the function. Select the midpoint of the bottom right edge of the box as the base point. You are going to rotate the solid part on the x-axis, so pick the diameter of the sphere that surrounds the x-axis, and then with the Ortho mode still “on,” pick a second point pulling the line from the base point up on the drawing screen to specify the angle start point. Type **63** to rotate the solid as shown in Figure 3.12.



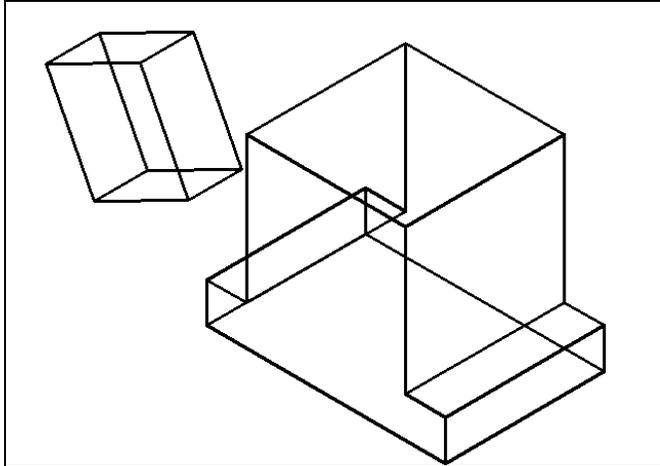
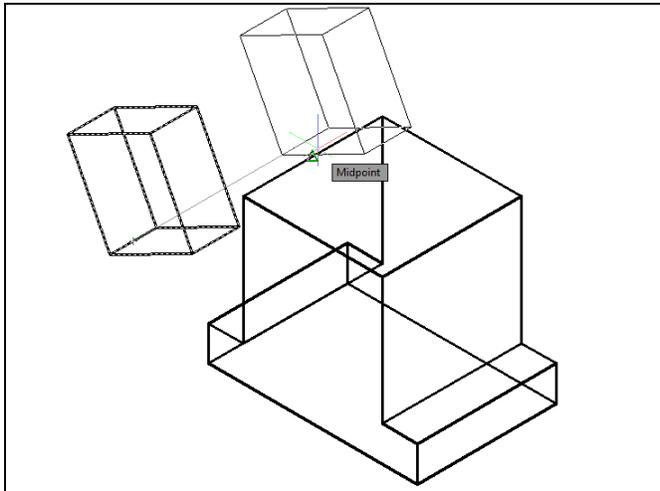


Figure 3.12 – Rotating the solid box

The Move command is actually a three dimensional function in AutoCAD. You can determine whether a command is three dimensional by attempting to use the function outside of the XY-plane where the Z coordinate is not 0. You can use Move to relocate any entity to another point (X,Y,Z) in Model Space.



3.13 – Moving the Box to the Part

To move the box, select the Move tool on the Modify panel. Once you choose the Move command, the prompt will be to “Select Objects.” You can do this by placing the pick box that now has replaced the aperture cursor on the small box and selecting the entity. The command line will respond with “1 found,” so hit ENTER to go the second part of the command.

Next, you need to “specify base point or displacement” on the graphical display. Pick the midpoint of the bottom back edge of the box. The “second point of displacement” is the midpoint of the top back edge of the 2 inch box. Use the Midpoint Osnap for both the base point and point of displacement as shown in Figure 3.13. Notice that the 2 x 2 x 1 box is now centered on the 2 inch box.

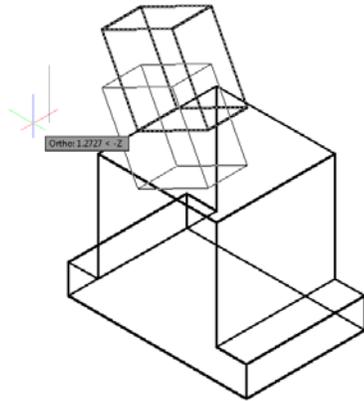


Figure 3.14 – Moving the box down

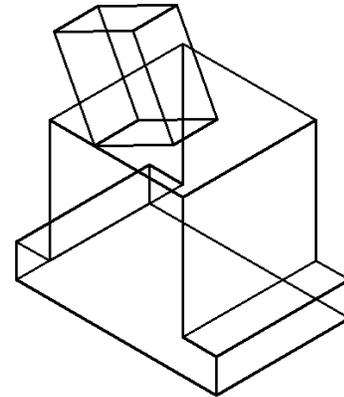


Figure 3.15 – Moving the box down

You can use the Move command to relocate any entity as simply as you did in the Fundamentals of 2D Drawing textbook. The Copy command will work similarly, except the original item will remain in place and a new solid will appear in the desired location.

Move the box again, this time 1 unit down the Z-axis as shown in Figure 3.14. Pick any base point on the box. When prompted for Displacement, type **1** and **ENTER**. Your finished product should match Figure 3.15

Mirroring a 3D Solid

You will use the 3D Mirror tool on the Modify panel to copy the box across the midpoint of the master solid. At the prompt to select objects, pick the small box and then hit **ENTER** to proceed to the second half of the Mirror command. “Specify first point of mirror plane:” by picking the midpoint of the near left edge of the large box. “Specify second point of mirror plane:” by picking the midpoint of the far right edge of the large box.

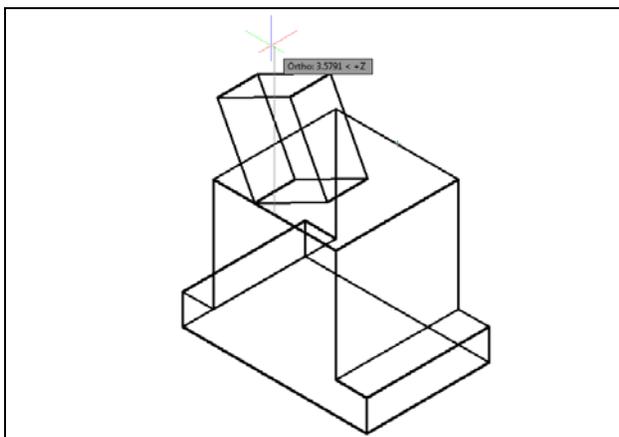
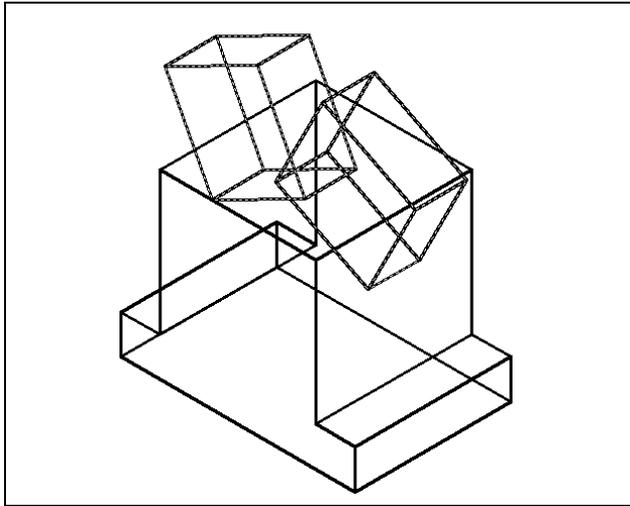


Figure 3.16 – Mirroring the box

Finally, move your cursor straight up and select to define the horizontal z axis of the mirror plane (Figure 3.17). The default to delete the source objects is no, so at the prompt “Delete source objects? [Yes/No] <N>:,” just hit **ENTER** and the mirrored and duplicated entities from one side of the mirror line will appear on the other side as well (Figure 3.17).

Subtracting 3D Solids from Another Solid

To remove the smaller solid boxes from the master solid, select the Subtract tool on the Solids Editing toolbar.



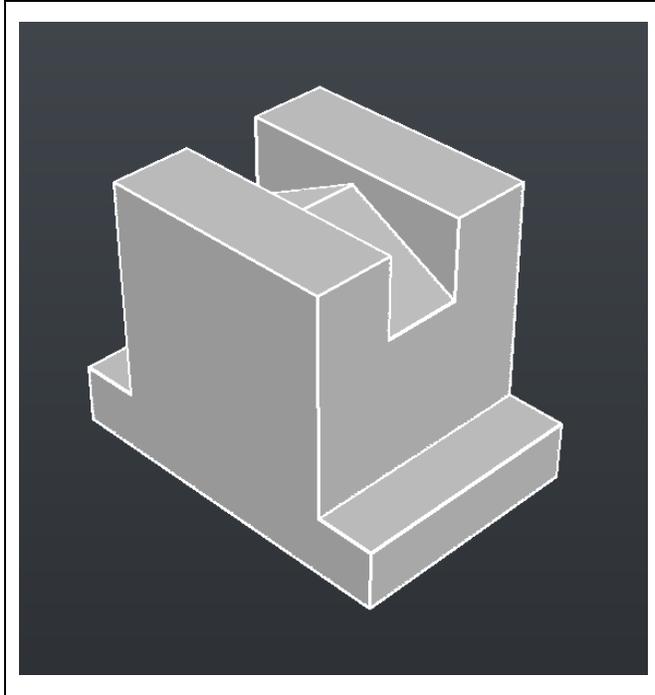
The command line will prompt you to “Select solids and regions to subtract from.” Select the larger solid box and the system will respond with “1 found.” Hit ENTER to proceed to the next step which is to “Select solids and regions to subtract.” Pick both small 2 x 2 x 1 boxes and the software application will counter with “2 found.” Hit ENTER and a region exactly the shape of the small boxes will disappear from the larger one.

Figure 3.17 – Subtracting the two boxes

Improve the View by Hiding Tessellation Lines

In Figure 3.18, you will notice that the tessellation lines representing the solid’s outline on the far edge of the part are visible. In an actual model, you would not be able to see through solid unless the material was transparent. You can use the Shaded with edges Visual Style command to remove the hidden lines from your graphical display.





Select the Shaded with edges Visual Style from the drop menu on the View panel under the Home tab. 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. The background color will change to navy blue, and then edges will turn white while the faces of the solid will turn grey. If you wish to return the graphical display to the previous setup showing all the tessellation lines, choose the first tool on the Visual Styles 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 3.18 – Hiding tessellation lines

Moving the Solid to the Origin Point

To move Problem Two 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 endpoint at the bottom left-hand corner of solid problem two, 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 Two 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.

*** World Class CAD Challenge 03-02 * - Close this drawing file. Create a New file, draw the Polyline, and extrude the entity to a solid; draw, rotate and mirror the solid box 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 to obtain your world class ranking.**

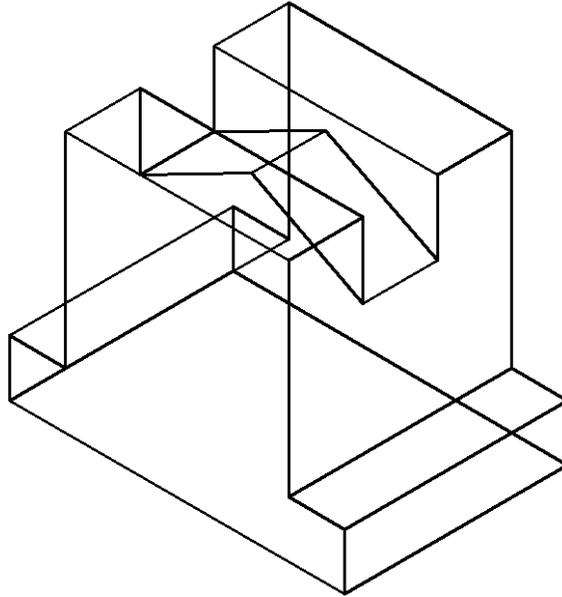


Figure 3.19 – Moving the finished solid

Saving the Solid Problem

To save Problem 2 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. At the File Name textbox, type “Problem 2” and hit the Save button to save the drawing (Figure 3.20).

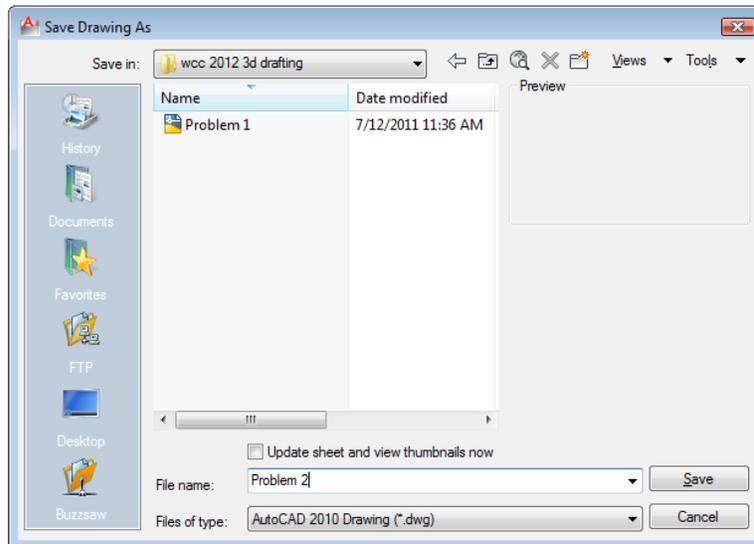


Figure 3.20 – Saving Problem Two

