Chapter

3

Solid Problem Two

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

1. **Sketch of Solid Problem Two**
2. **Starting a 3D Part Drawing**
3. **Drawing a Closed Polyline Shape to be Extruded**
4. **Extruding a Solid from a Closed Polyline**
5. **Rotating a 3D Solid Using the Rotate3D and Rotate Commands**
6. **Drawing a Solid Box**
7. **Rotating and Moving a Solid onto the Master Solid**
8. **Mirroring a 3D Solid**
9. **Subtracting a Solid from another Solid**
10. **Improve the View by Hiding Tessellation Lines**
11. **Moving the Solid to the Origin Point**
12. **Saving the Solid Problem**

Sketch of Solid Problem Two

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You start the second solids problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across two lines 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, a strategy you will learn is to draw a closed Polyline that contains the most complex shape of the component and then use simple geometric solids to add or subtract detail. In problem two, the inverted tee contains the intricate contour that will be formed into the Polyline shape, and the 63 degree wedge in the middle of the part will be the smaller aspect that will be addressed secondly.
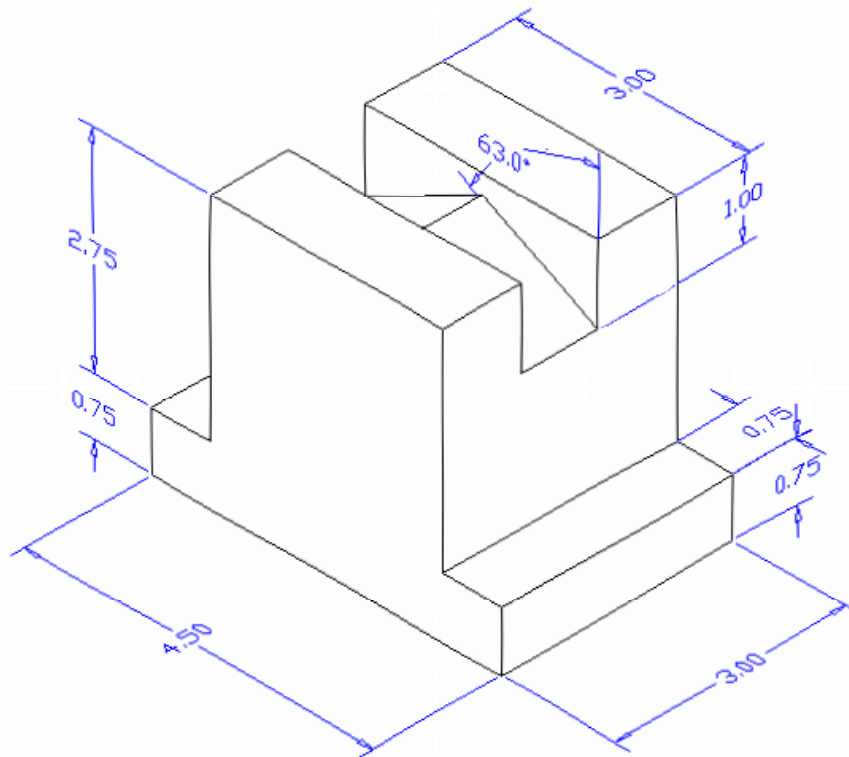

Figure 3.1 – Problem Two Sketch

When you receive a sketch from a professional, you need to practice to quickly identify the complex shapes and decide how to proceed with the solid exercise. These 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 Two, you will begin the 3D drawing by selecting the Mechanical template. In Figure 3.2, the window "Create New Drawing" is loaded by selecting the New tool on the Standard Toolbar. Select the "Use the Template" (third button) at the top of the Create New Drawing window. A list of templates is loaded in the Select a Template list

box. 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.
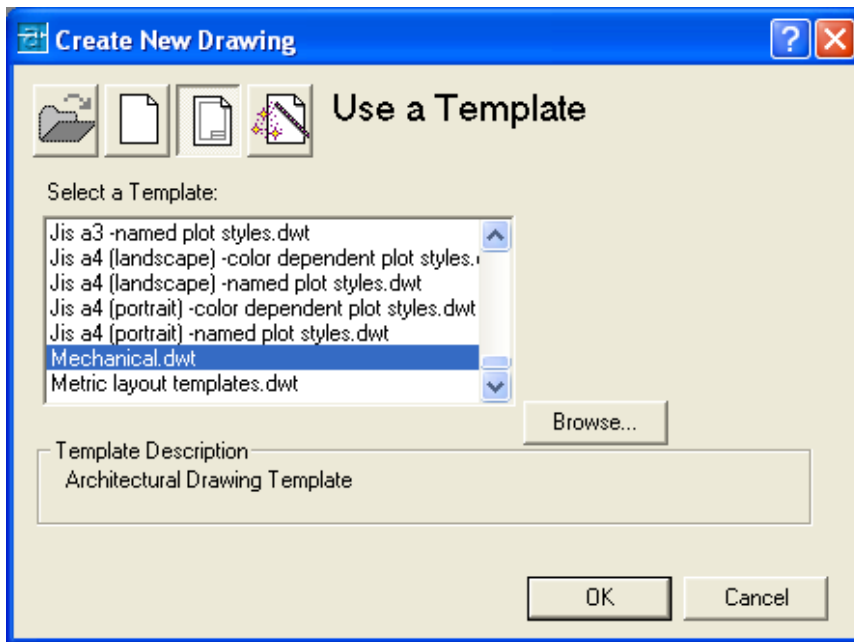

Figure 3.2 – Starting the Drawing Using the Mechanical Template

Drawing a Closed Polyline Shape to be Extruded

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Drawing a closed Polyline is not new to you, because you will draw the contour of this solid similarly to the outside wall drawn in the Fundamentals of 2D Drawing, Architectural Floor Plan. A Polyline is not like a typical set of lines since the entity can contain multiple start and ending points 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.

On the Draw Toolbar, select the Polyline tool 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.

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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 – The Polyline Text from the Command Line

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. Select the Zoom Extents tool on the Standard toolbar to have the box fill the monitor, so you can proceed to the next step in the drawing process.
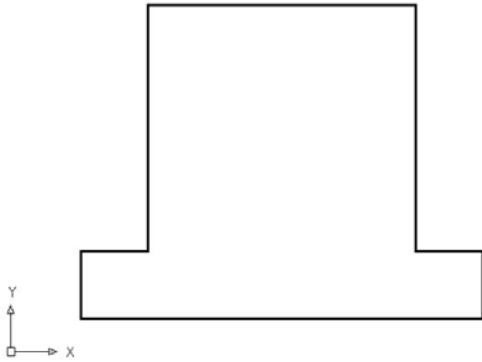

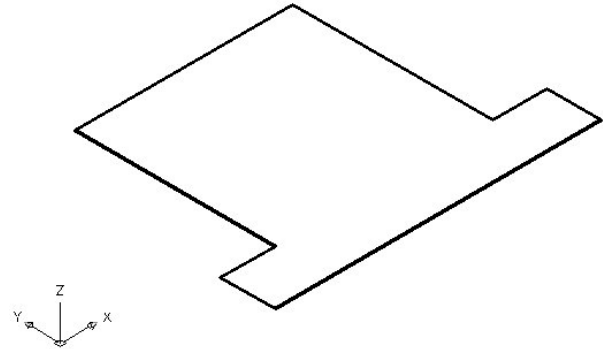**Figure 3.4 – Drawing a Closed Polyline**   **Figure 3.5 – Isometric View of the Shape**

On the View toolbar, select the Southwest Isometric View tool to allow you to see the part on the XY plane and the vertical Z-axis will rise from the level surface. (See Figure 3.5).

Extruding a Solid from a Closed Polyline

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Only closed entities like Polylines and Circles can be extruded using this very powerful tool on the Solids toolbar. If a there is even a minute space in the entity a warning will appear on the command line that you are "**unable to extrude the selected object**". If you use an attention detail in your 3D computer aided design work, you can create the most complex shapes with the extrusion instrument.

Select the Extrude tool on the Solids toolbar and the command line on the AutoCAD application will prompt you to "**select objects**". Pick the Closed Polyline anywhere on the peripheral line and the command line will return with "**1 found**" as shown in Figure 3.6. Press **Enter** to specify the height or path of the solid. You can type "P" for Path and choose an entity to extrude the shape along, but you will not want that option for problem two. You will type "**3**" for the height of the extrusion and for the angle of taper you will **Enter** to accept the zero degree default. A more complex 3D solid will appear in the graphical display as shown in Figure 3.7.

Command: _extrude
Current wire frame density: ISOLINES=4
Select objects: **1 found**
Select objects:
Specify height of extrusion or [Path]: **3**
Specify angle of taper for extrusion <**0**>:
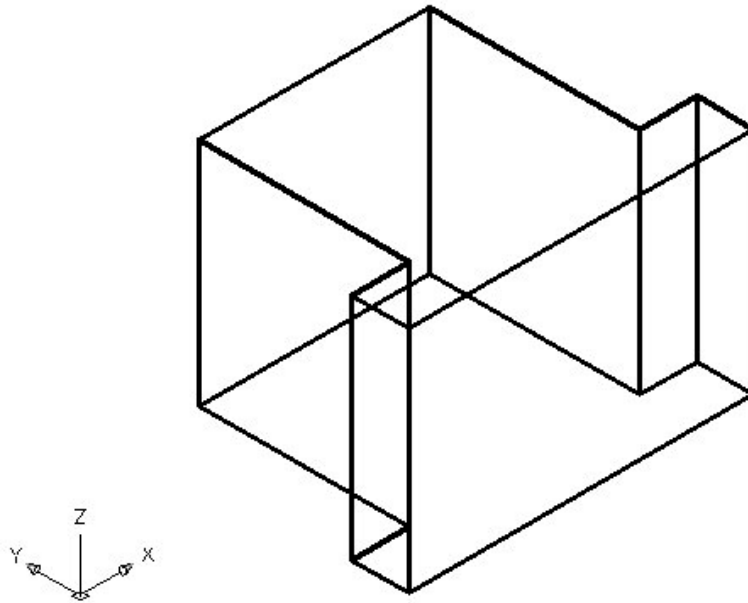
Figure 3.6 – The Extrude Command

Figure 3.7 – Extruding the Polyline Shape
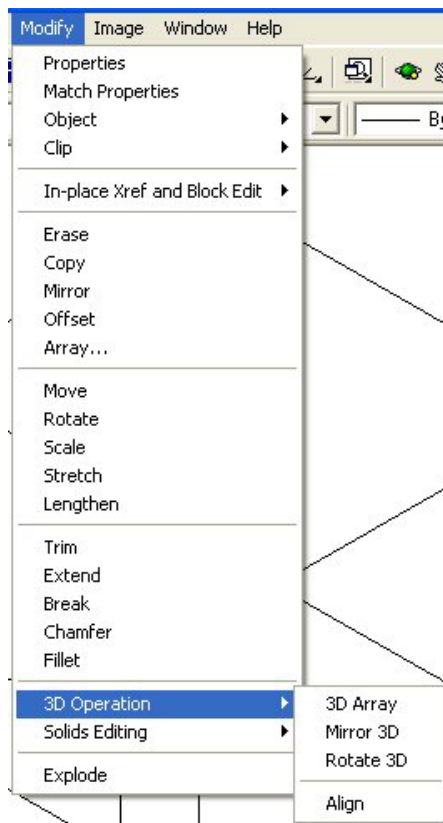
Rotating a 3D Solid Using the Rotate3D and Rotate Commands

There are four modifying commands that are useful in handling 3D components and they are 3DArray, Mirror3D, Rotate3D and 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 direction. The 3D mirror tool will allow any entity capable of 3D movement to cross any plane defined by three points. The 3D rotation device will move 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 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.
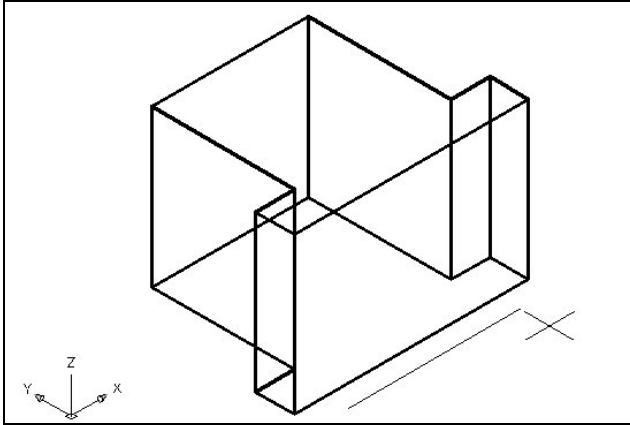
Figure 3.8 – Selecting the Rotate3D Command

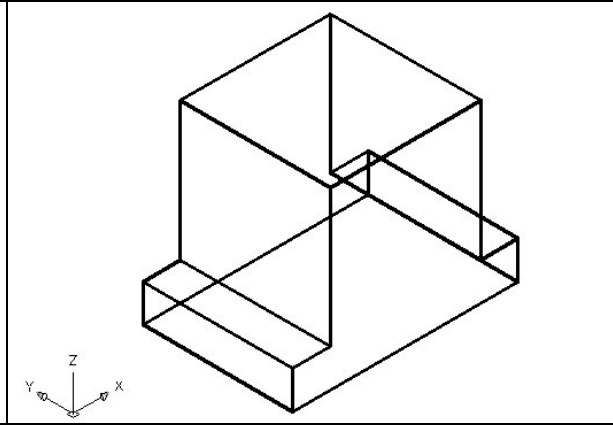**Figure 3.9 – Picking the Rotate3D Axis**   **Figure 3.10 – The Part Rotated on an Axis**

Select the Modify on the Menu Bar, then 3D Operations and pick Rotate 3D from the list of four commands. 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 as shown in Figure 3.9 and then with the Ortho mode still "on" pick a second point drawing a line on the x-axis. Type **-90** to rotate the solid as shown in Figure 3.10.
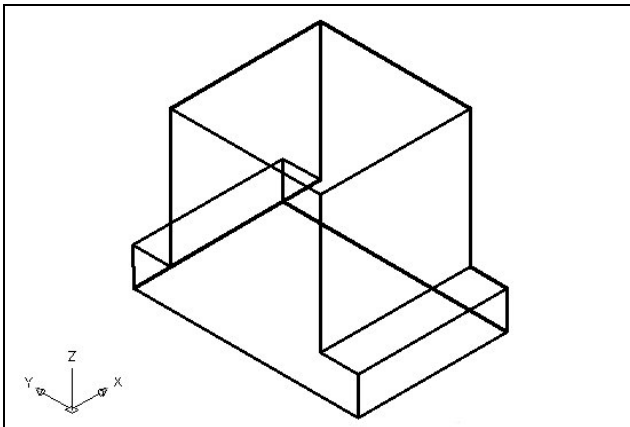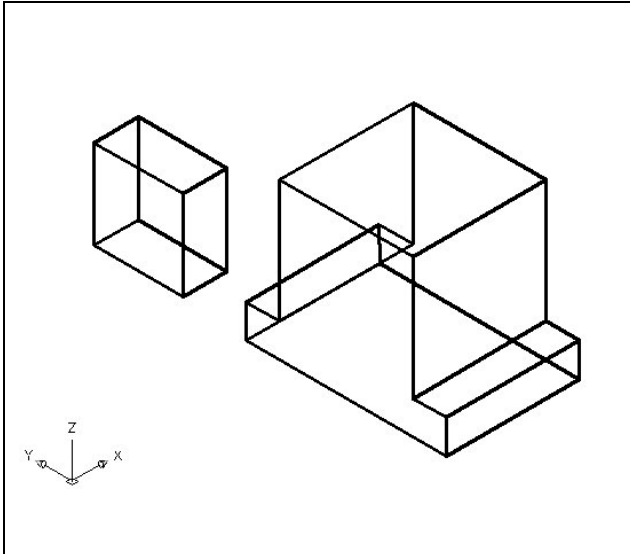

To rotate the solid in the XY plane, select the regular Rotate command on the Modify toolbar. Select the Rotate command on the Modify toolbar and pick the solid. The selected entity will be dotted and to continue with the command, press **Enter** to select the base point of rotation. Select any point 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 (See Figure 3.11).

Figure 3.11 – Rotated on the Second Axis

Drawing a Solid Box

The technique you will use in this exercise will utilize the Length option of the Box command. Type **L** after specifying the corner on the graphical display and the system will ask three questions. The first prompt is "Specify Length" where when you type a number and Enter will designate the **X** coordinate. The second prompt is "Specify Width" where when you type a number and Enter will designate the **Y** coordinate. The third prompt is "Specify Height" where when you type a number and Enter will designate the **Z** coordinate.
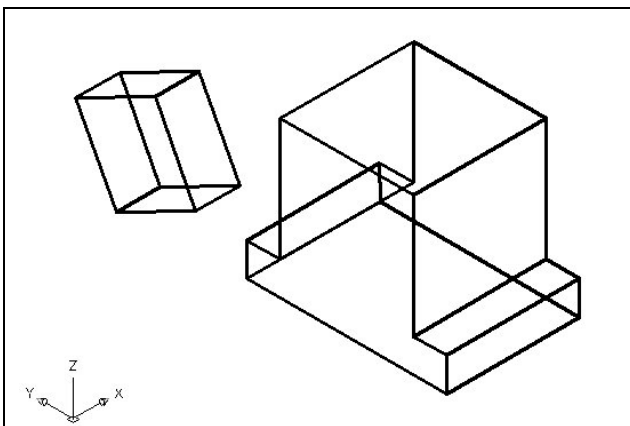
Select the Box tool on the Solids toolbar. At the command line, a prompt asks you to specify the corner of the solid and for this box, specify any point to the left of the solid as shown in Figure 3.11. Then, type **L** after specifying the starting point and the system will ask you three questions. The first prompt is "**Specify Length**" where then you will type **1** and **Enter** for the X coordinate. The second prompt is "Specify Width" where you will type a **2** for the Y coordinate and **Enter**. The third prompt is "Specify Height" where when you type **2** and **Enter** for the Z coordinate. The second box will appear as shown in Figure 3.11.
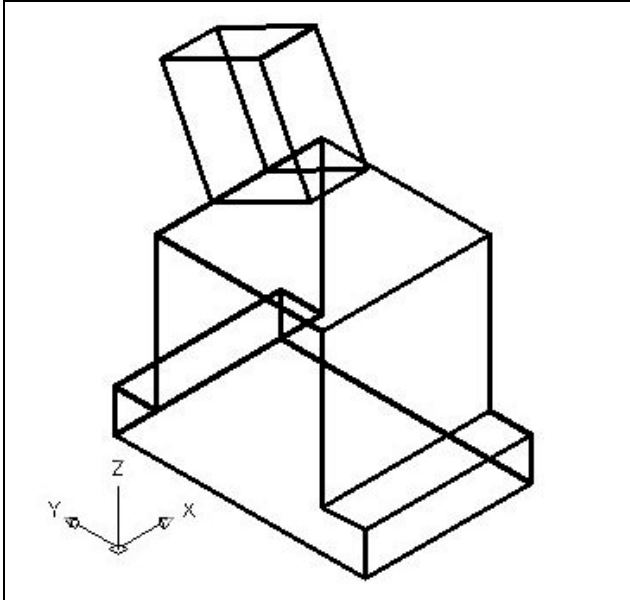
Figure 3.12 – Making a Solid Box

Rotating and Moving a Solid onto the Master Solid

Select the Modify on the Menu Bar, then 3D Operations and pick Rotate 3D from the list of four commands. 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 as shown in Figure 3.9 and then with the Ortho mode still "on" pick a second point drawing a line on the x-axis. Type **-90** to rotate the solid as shown in Figure 3.10.
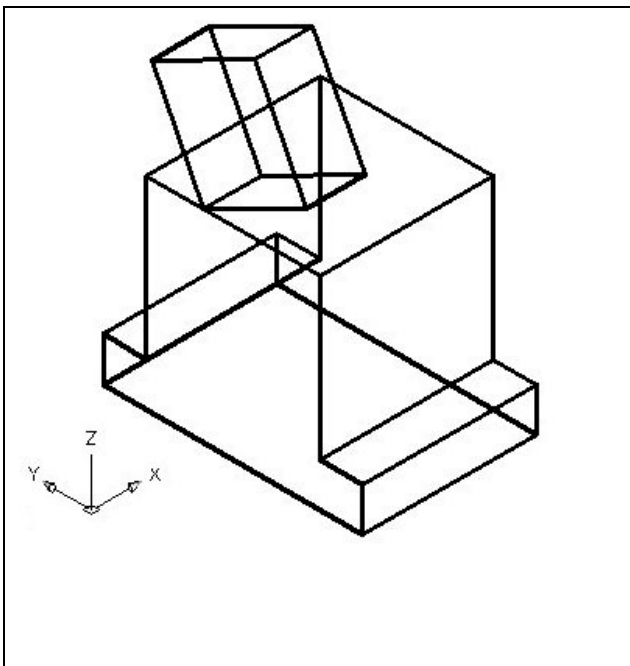

The Move command is actually a three dimensional function in the modern AutoCAD application software program. The way you can determine whether a command is three dimensional in nature is to attempt to use the function outside the Z-plane, that is the Z coordinate is not 0. You can use move to relocate any entity to another point (X,Y,Z) in Model Space.

Figure 3.13 – Rotating the Solid Box

Figure 3.15 – Moving the Box to the Part

The Move command is separated into two parts, the first being the selection process and the second being the function of the command itself. To move an AutoCAD entity such as a solid box, you select the Move tool on the Modify toolbar. Once you choose the Move command, the prompt will be "Select Objects". You can do this by placing the pick box that now has replaced the aperture cursor on the small box and select the entity. The command line will respond with "1 found", so just press ENTER to go the second part of the command.
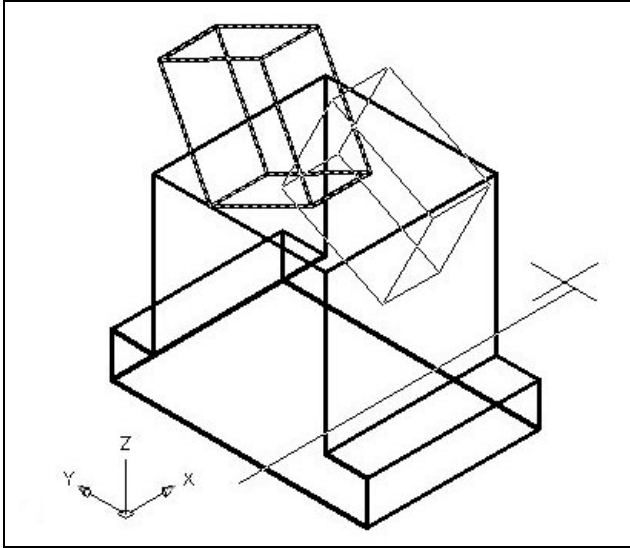

Figure 3.16 – Moving the Box Down

Next, you need to "specify base point or displacement" on the graphical display. Pick your base point with the Midpoint Osnap as shown in Figure 2.9. The "second point of displacement" is the center point of the 2 inch box. Pick the second point using the Midpoint Osnap of the larger box, notice that the 1 x 1 x 0.5 box is now centered on the 2 inch box (Figure 2.10). Again, you have accomplished another milestone when moving an AutoCAD entity by selecting two known points. 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.

Mirroring a 3D Solid

You will use the Mirror tool on the Modify toolbar to copy the solid object across the midpoint of the master solid.
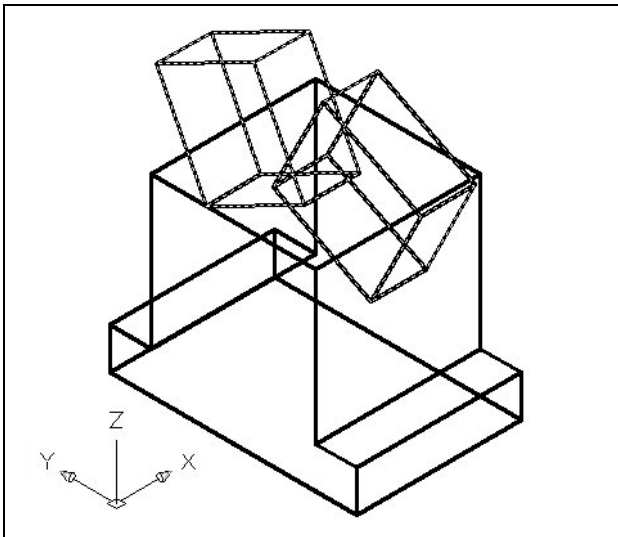
At the prompt to select objects, pick the box and then **Enter** proceed to the second half of the Mirror command. "Specify first point of mirror line:" by picking the left endpoint of the centerline and "Specify second point of mirror line:" by picking the right endpoint of the centerline. The default to delete the source objects is no, so at the prompt "Delete source objects? [Yes/No] <N>:", just Enter and the mirrored and duplicated entities from the top half will appear on the bottom half of the top orthographic view. (See Figure 10.12) Zoom extents if necessary to view the entire drawing.

Figure 3.17 – Moving the Box Down

Subtracting a 3D Solid from Another Solid

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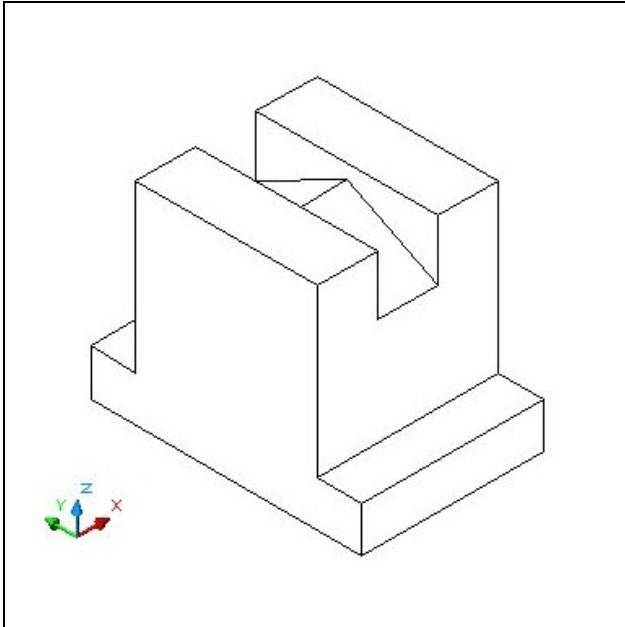

Then the AutoCAD program will prompt you with "Select solids and regions to subtract from". Select the larger solid box and the system will respond with "1 found". You need to Enter to proceed to the next step which is "Select solids and regions to subtract". Pick the small 1 x 1 x 0.5 box and the software application will counter with "1 found". Enter and a region exactly the shape of the small box is missing from the larger one as shown in Figure 2.11. This technique to enhance the detail of a solid entity by removing a part of the feature is commonplace in the world of 3D design.

Figure 3.18 – Subtracting the Two Boxes

Improve the View by Hiding Tessellation Lines

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In Figure 3.14, 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.

Figure 3.18 – Hiding the Tessellation Lines

Select the Hidden tool on the Shade toolbar. 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. Notice in Figure 3.14 that the UCS icon changes from the thin lined and arrowed icon to a colorful image. If you wish to return the graphical display to the previous setup where all the tessellation lines are seen, then 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.

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, pick the Move tool on the Modify toolbar and select the 3D solid and Enter. At the prompt, "specify base point or displacement", select the endpoint shown in Figure 3.15 at the lower right hand side of the Problem Two, which will become the Problem Two insertion point into an assembly drawing. At the "second point of displacement", type 0,0,0 and Enter. Problem Two will move to the new origin point as shown in Figure 3.16. 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.

* World Class CAD Challenge 03-02 * - Close this drawing file. Create a New file and 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 under 5 minutes to maintain your World Class ranking.
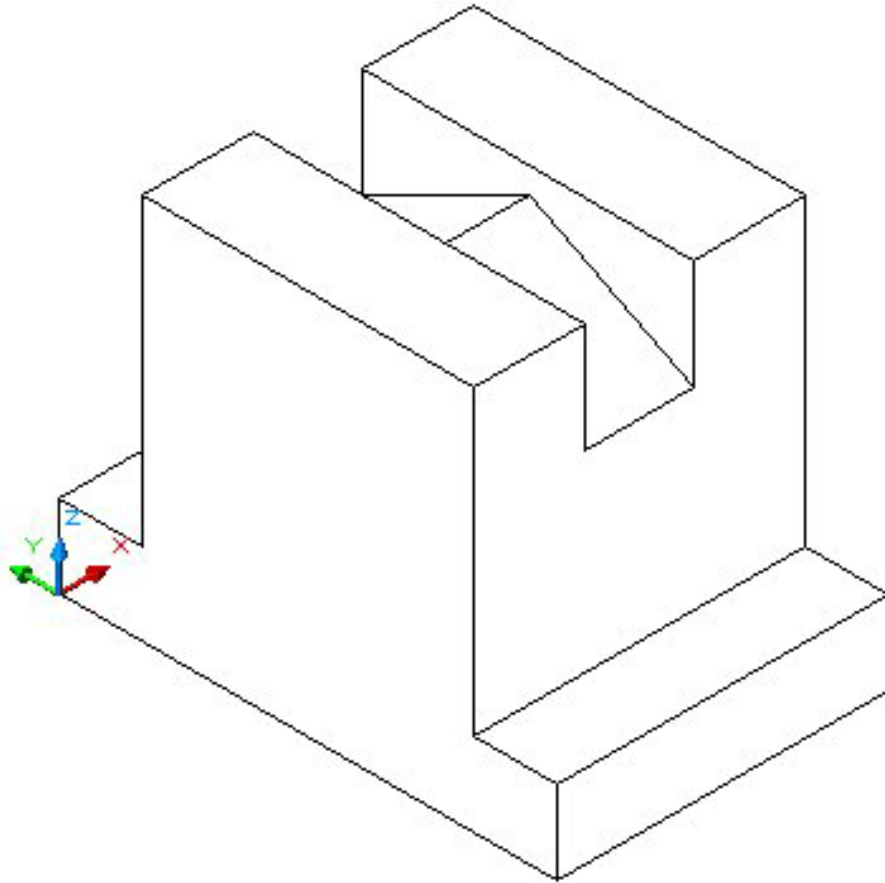
Figure 3.20 – Moving the Finished Solid

Saving the Solid Problem

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To save Problem 2 in your Fundamentals of 3D Drawing Folder, 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 2" and press the Save button to save the drawing. (See Figure 3.21)
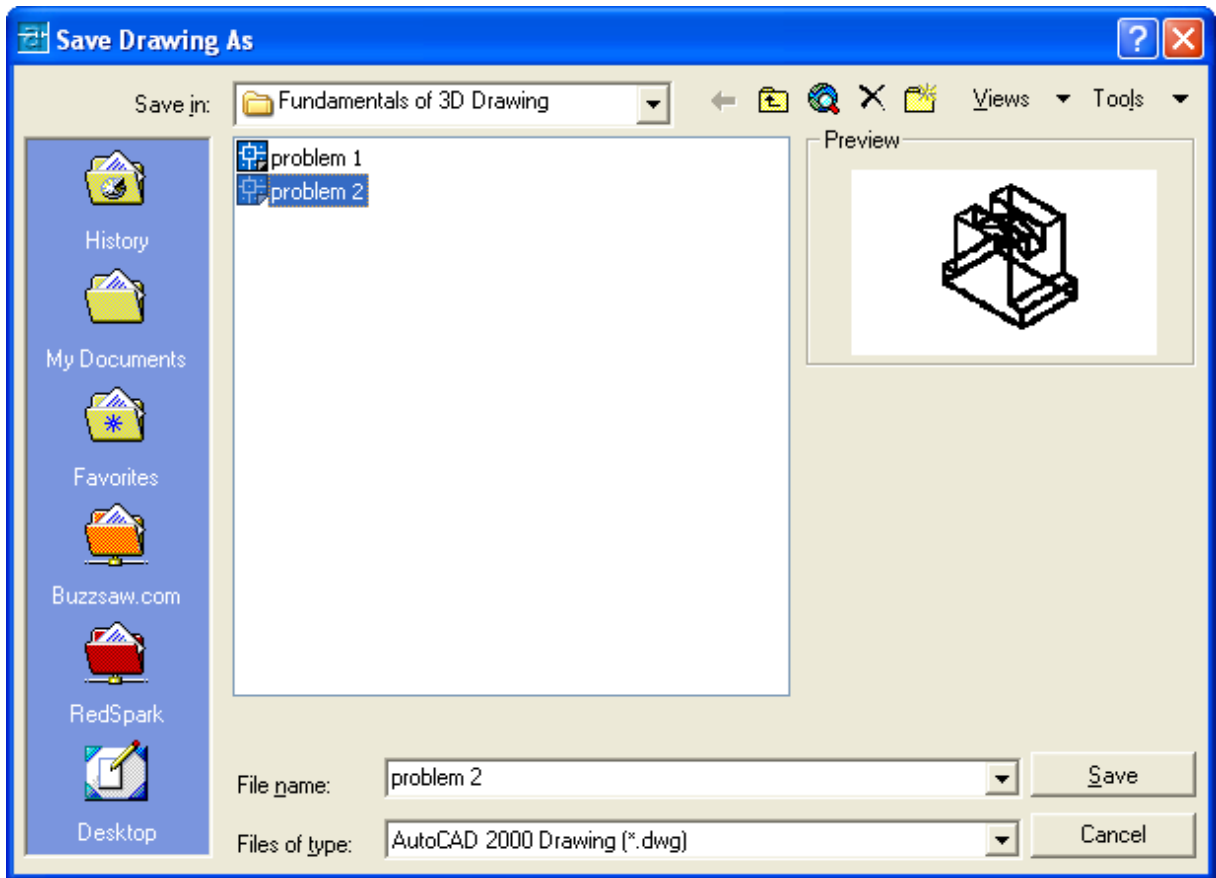
Figure 3.21 – Saving Problem Two

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