

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. Improving the View by Hiding Tessellation Lines
11. Moving the Solid to the Origin Point
12. Saving the Solid Problem

Sketch of Solid Problem Two

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 w-axis as the vertical rise of the piece. In this exercise, the 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 a Polyline shape, and the 63 degree wedge in the middle of the part will be the smaller aspect that will be addressed secondly.

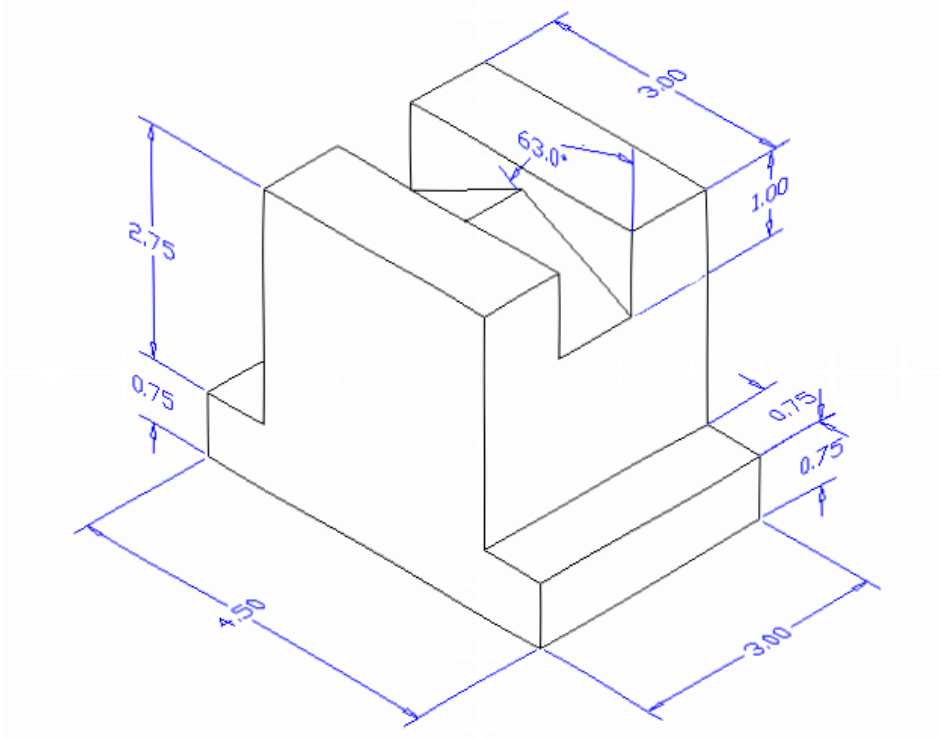


Figure 3.1 – Problem Two Sketch

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. The ten drills in the of 3D Drawing with progeCAD Professional textbook will give you those repetitive maneuvers which will make you a successful in the 3D modeling.

Starting a 3D Part Drawing

In Problem Two, you will begin the 3D drawing by selecting the New tool on the Standard toolbar. In Figure 3.2 you can see the Start window that appears. Select the “Start from Scratch” (first button) at the top of the Start window. Select English as the Default Setting and hit OK. A new drawing file will open, and now you are ready to begin.



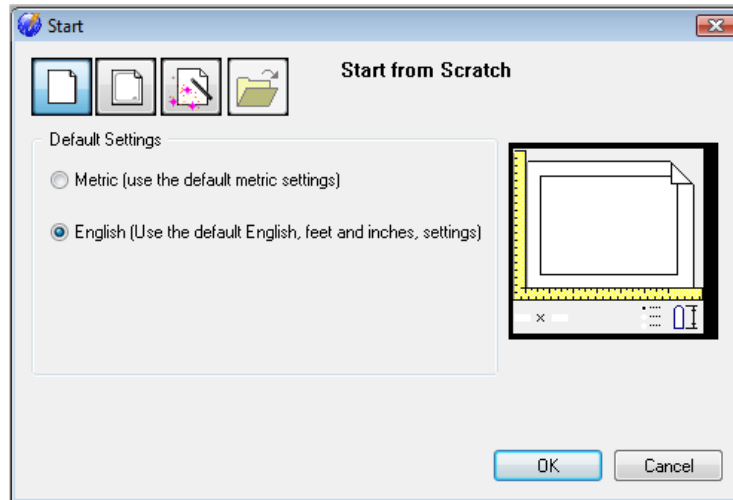


Figure 3.2 – Starting the Drawing Using the Mechanical Template

Drawing a Closed Polyline Shape to be Extruded

Drawing a closed Polyline should not new to you because you used the tool to draw outside wall drawn of the Architectural Floor Plan in 2D Drawing with progeCAD Smart!. A Polyline is not like a typical set of lines because 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 when drawing advanced parts 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.

```
Command : '._ORTHOgonal
ORTHOMODE is currently off: ON/Toggle/<Off>: _T
Command : _POLYLINE
Start of polyline:
Arc/Distance/Halfwidth/Width/<Next point>: 4.5
Arc/Distance/Follow/Halfwidth/Width/Undo/<Next point>: .75
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: .75
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 3
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 2.75
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: .75
Arc/Close/Distance/Follow/Halfwidth/Width/Undo/<Next point>: 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. Zoom Extents to zoom in on the box so you can proceed to the next step in the drawing process.

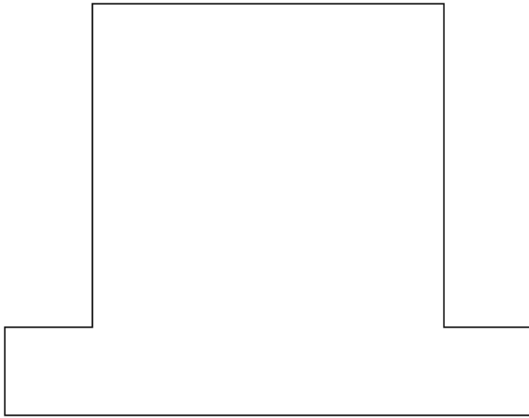


Figure 3.4 – Drawing a Closed Polyline

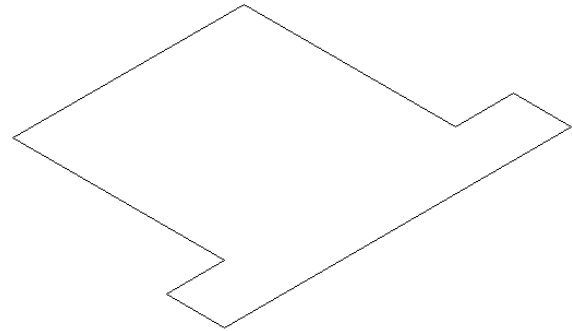


Figure 3.5 – Isometric View of the Shape

On the View toolbar, select the Southwest Isometric View tool to allow you to see the part on the XY plane with the vertical z-axis rising from the level surface (Figure 3.5).

Extruding a Solid from a Closed Polyline

Only closed entities like Polylines and Circles can be extruded using this very powerful tool on the Solids toolbar. If there is even a minute space in the entity you will not be able to select the entity when prompted to “Select Entities.” If you use an attention to detail in your computer aided design work, you can create very complex shapes with the Extrusion tool.



Select the Extrude tool on the Solids toolbar and the command line will prompt you to “**Select entities:**” Pick the Closed Polyline anywhere on the peripheral line and the command line will return with “**Entities in set: 1**” 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 along which to extrude the shape, but you will not want that option for problem two. Rather, type “**3**” for the height of the extrusion, and for the angle of taper just hit **ENTER** to accept the zero degree default. The polyline will expand 3 units up into the z-axis as shown in Figure 3.7.

```
Command : _extrude
Select entities:
Entities in set: 1
Select entities:
Specify height of extrusion or Path: 3
Specify angle of taper for extrusion <0>:
```

Figure 3.6 – The Extrude Command

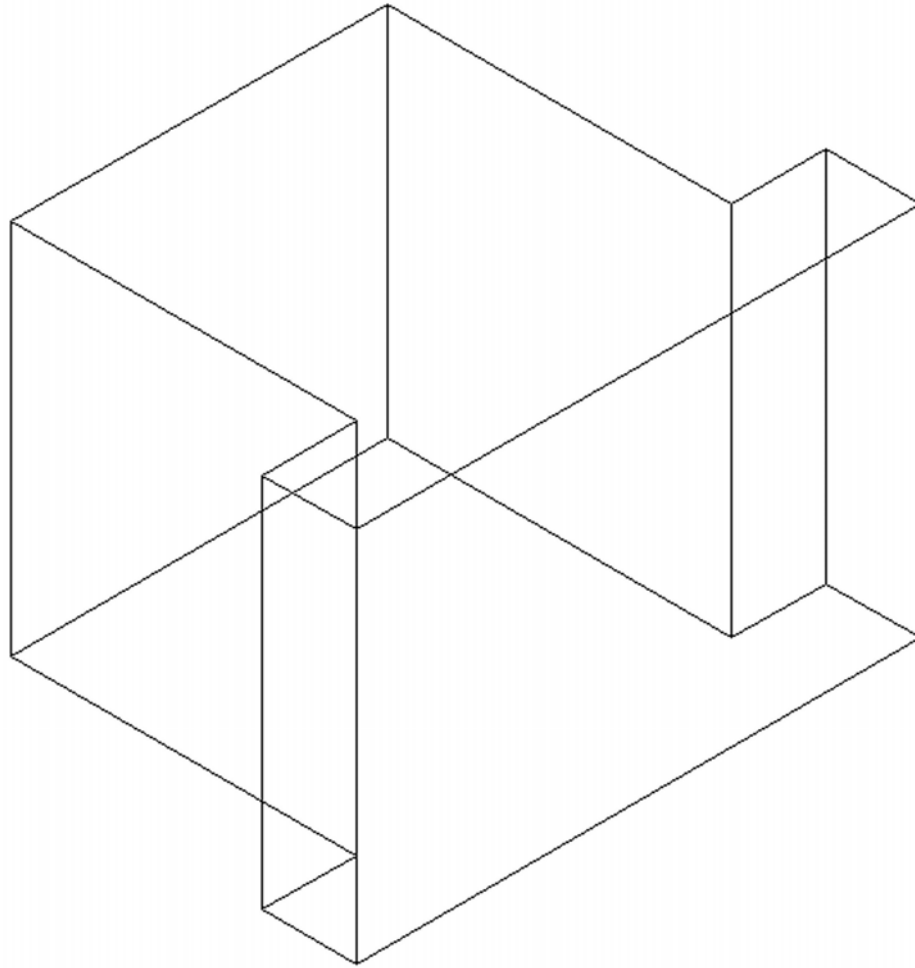
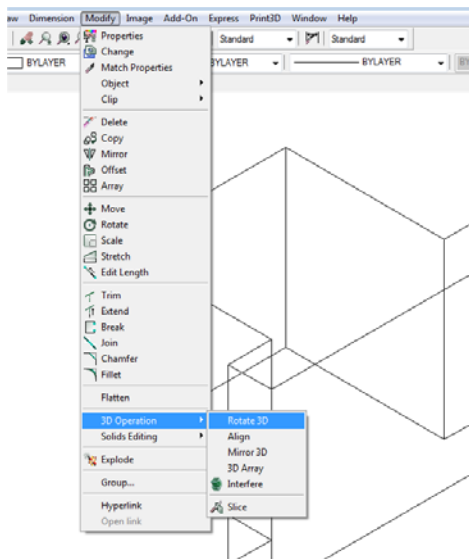


Figure 3.7 – Extruding the Polyline Shape

Rotating a 3D Solid Using the Rotate3D and Rotate Commands



There are five modifying commands that are useful in handling 3D components: 3DArray, Mirror3D, Rotate3D, Align, and Interference. 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 w direction. The 3D mirror tool will allow any entity capable of 3D movement to cross any plane defined by three points. The 3D rotation tool will move any solid or even two dimensional units around any axis defined by two points in Model Space. Interfere works much like the Intersection command. 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

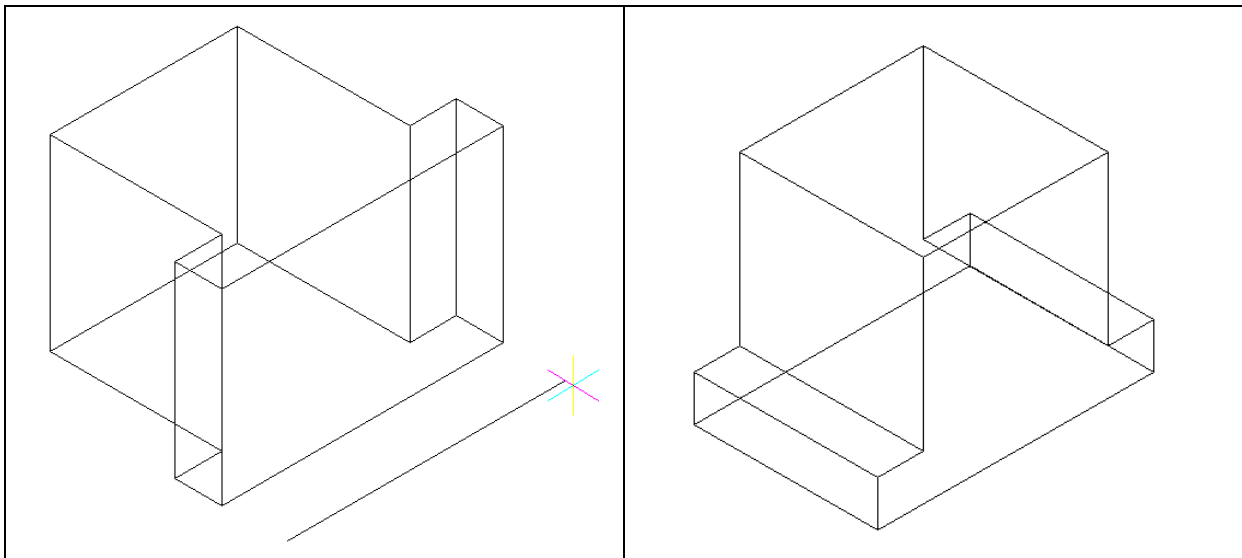
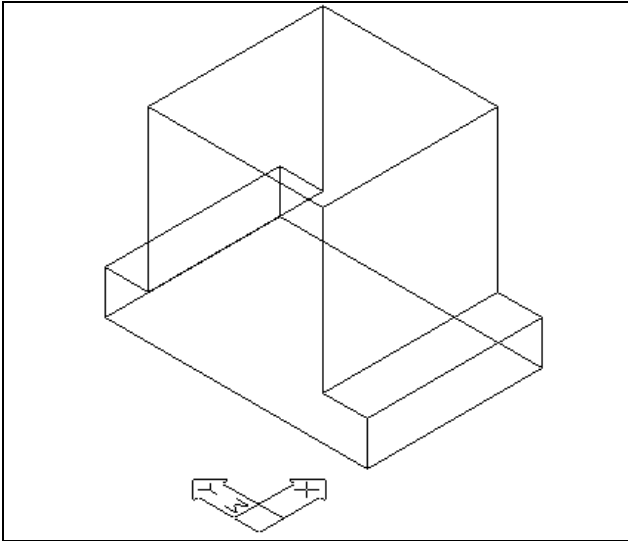


Figure 3.9 – Picking the Rotate3D Axis

Figure 3.10 – The Part Rotated on an Axis

Select Modify from the Menu Bar, then 3D Operations, and pick Rotate 3D from the list of commands. At the command line prompt, “Select entities to rotate:,” pick the 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 as shown in Figure 3.9, 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 3.10.

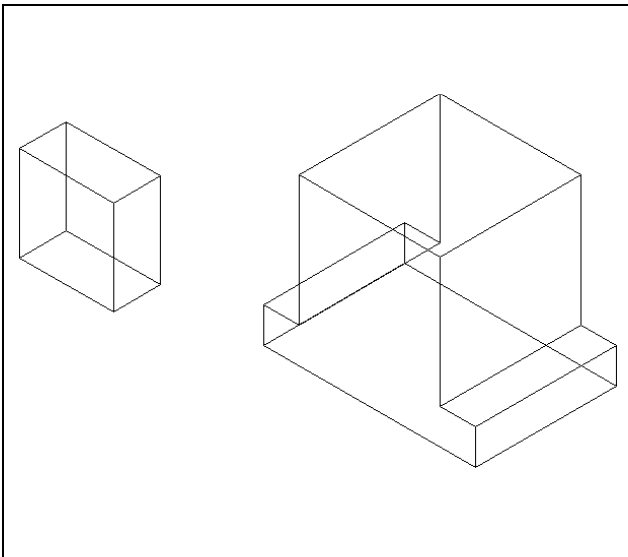


To rotate the solid in the XY plane, select the regular Rotate command on the Modify toolbar. Select the part and to continue with the command, press **ENTER** to select the base point of rotation. Select any point at the base of the solid with the left mouse button. With the Ortho on, you should be able to easily rotate the part 90 degrees so that it appears as shown in 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 to enter the “Length of side of box:”. When you type a number and hit **ENTER**, that number will designate the **X** coordinate. The second prompt is to specify the “Width of box:”. Type a number and hit **ENTER** to designate the **Y** coordinate. The third prompt is “Height of box or [2Points]:”. Type a number and hit **ENTER** to designate the **Z** coordinate.

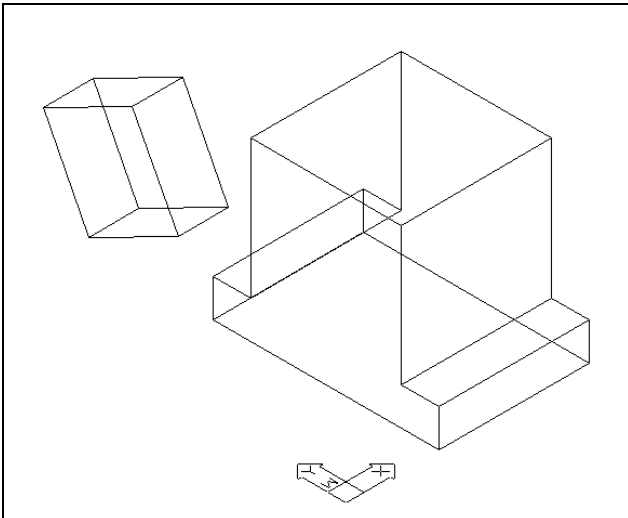
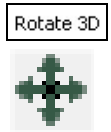


Select the Box tool on the Solids toolbar. At the command line, a prompt asks you to specify the corner of the solid. For this box, specify any point to the left of the solid. Type **L** after specifying the starting point and the system will ask you three questions. The first prompt asks for the “**Length of side of box:**” Type **1** and hit **ENTER** for the X coordinate. The second prompt asks for the “Width of box:” Type a **2** for the Y coordinate and hit **ENTER**. The third prompt asks for the “Height of box or [2Points]:” Type **2** and **Enter** for the Z coordinate. The second box will appear as shown in Figure 3.12.

Figure 3.12 – Making a Solid Box

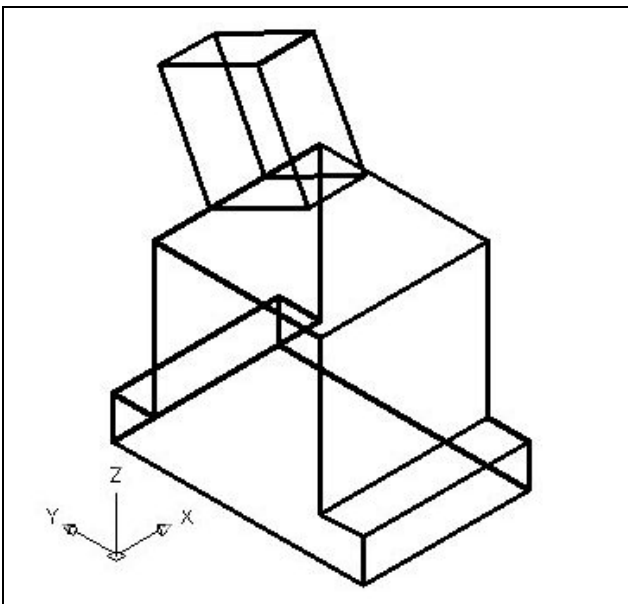
Rotating and Moving a Solid onto the Master Solid

Select Modify from the Menu Bar, then 3D Operations, and pick Rotate 3D from the list of commands. At the command line prompt, “Select entities to rotate:” pick the new 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 as shown in Figure 3.9, and then with the Ortho mode still “on” pick a second point to draw a line on the x-axis. Type **63** to rotate the solid as shown in Figure 3.13.



The Move command is separated into two parts: the first is the selection process and the second is the function of the command itself. To move an entity such as a solid box, you select the Move tool on the Modify toolbar. Once you choose the Move command, you will be prompted to “Select entities to move:.” You can do this by placing the pick box that has now replaced the aperture cursor on the small box and selecting the entity. The command line will respond with “Entities in set: 1,” so just hit ENTER to go the second part of the command.

Figure 3.13 – Rotating the Solid Box



Next you need to select a “Vector/<Base point>:” on the graphical display. Pick your base point as the midpoint of the bottom edge of the small box. Select the midpoint of the top left edge of the larger solid as the “Displacement point:.” Now the 1 x 1 x 0.5 box is now centered on top of the 2 inch box (Figure 3.14). Again, you have accomplished another milestone by moving a progeCAD entity using two known points. This time you’ve done it across more than one axis.

Figure 3.15 – Moving the Box to the Part

You need to move the smaller box once more, exactly 1" down on the Z axis. To do this easily, switch to the Right View or Left View from the View toolbar. Select the Move command again from the Modify toolbar and select the smaller box as the entity to move. For the base point, select anywhere on the display. Move the cursor below the basepoint, type 1, and hit ENTER. Move back to the Southwest View on the View toolbar so that the drawing appears as shown in Figure 3.17.

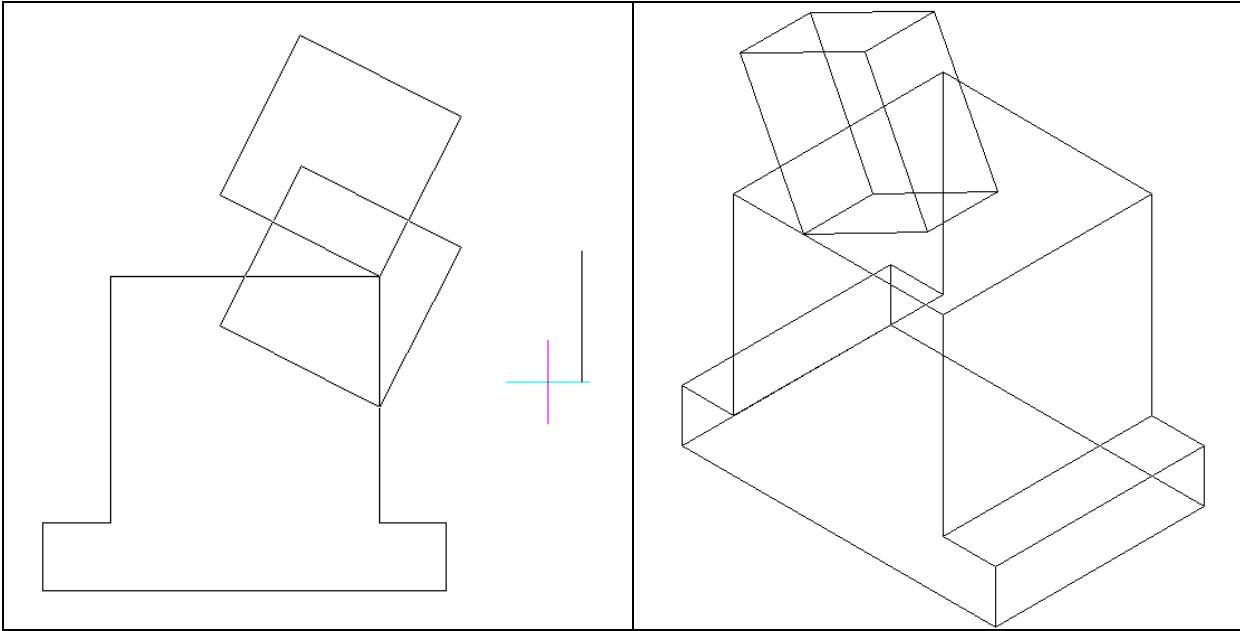
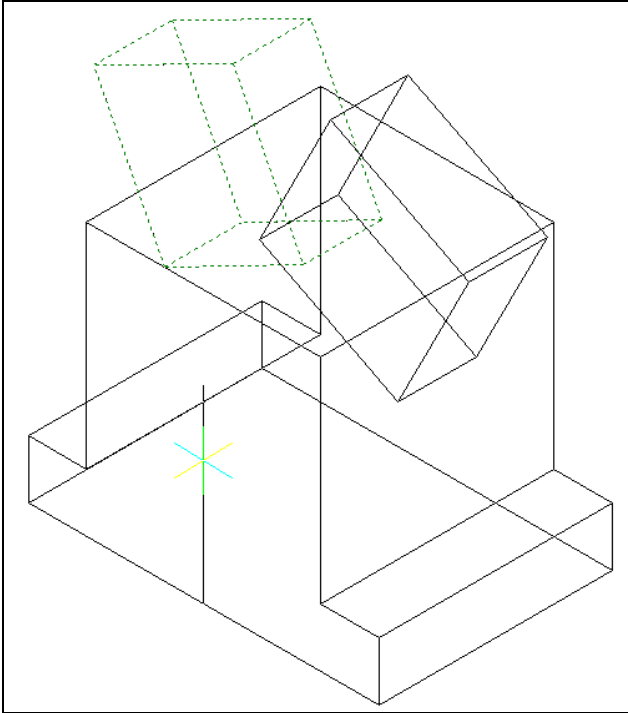


Figure 3.16 – Moving the Box Down

Figure 3.17 – The Smaller Box Moved

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.



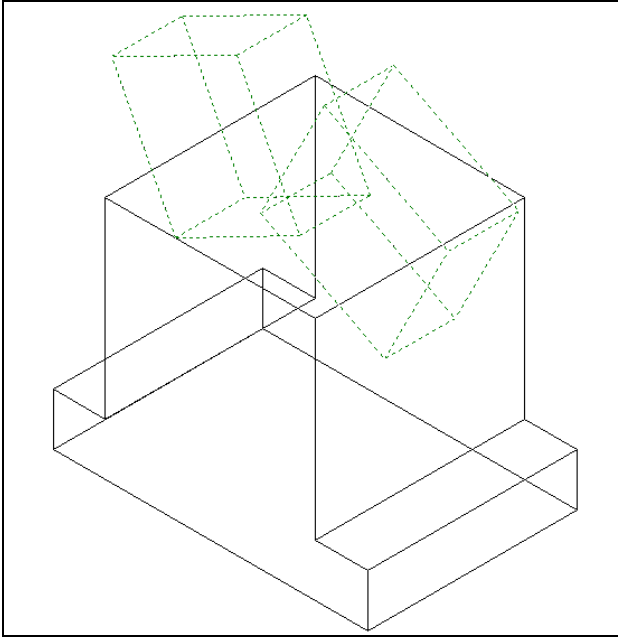
At the prompt to “Select entities to mirror:” pick the small box and then hit **ENTER** to proceed to the second half of the Mirror command. Select the “Start of mirror line:” by picking the midpoint of the bottom left edge of the large part. Drag the line straight up (with the Ortho on) and select anywhere so that the preview of the mirrored part appears as shown in Figure 3.18. The default to delete the source objects is no, so at the prompt “Delete the original entities? <N>” just hit ENTER and the mirrored and duplicated box will appear opposite its original. Zoom extents if necessary to view the entire drawing.

Figure 3.18 – Mirroring the Small Box

Subtracting a 3D Solid from Another Solid

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



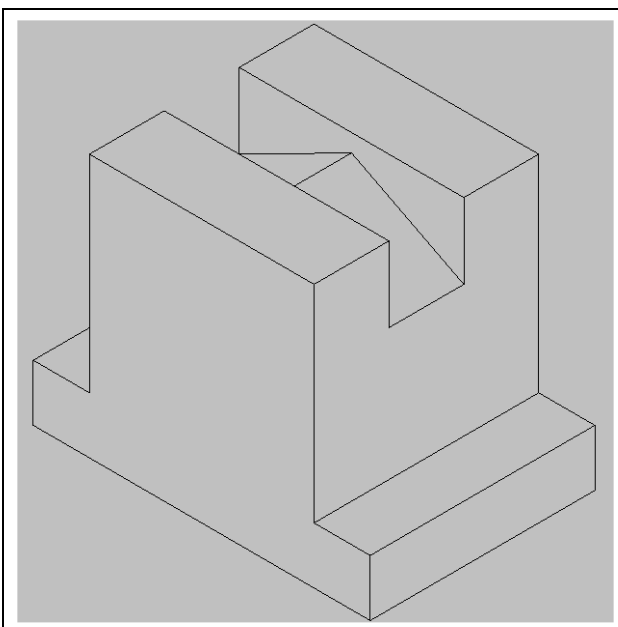


progeCAD will prompt you to “Select ACIS object to subtract from:.” Select the larger solid box and the system will respond with “Entities in set: 1.” Hit ENTER to proceed to the next step, which is to “Select ACIS objects to subtract:.” Select both small boxes and the command line will respond with “Entities in set: 2.” Hit Enter and a region exactly the shape of the small boxes will have disappeared from the larger one as shown in Figure 3.19.

Figure 3.19 – Subtracting the Two Boxes

Improve the View by Hiding Tessellation Lines

In Figure 3.19, 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 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. Notice in Figure 3.20 that the UCS icon has disappeared. If you wish to return the graphical display to the previous setup where all the tessellation lines are 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 3.20 – Hiding the Tessellation Lines

Moving the Solid to the Origin Point

To move the Problem Two solid 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. At the prompt, “Vector/<Base point>:,” select the endpoint at the lower right hand side of the Problem Two, which would become its insertion point if the part was inserted into an assembly drawing. For the “Displacement point:,” type 0,0,0 and ENTER. Problem Two will move to the new origin point as shown in Figure 3.21. There is more than one way to do many actions in progeCAD, 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 101-02 * - 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.**

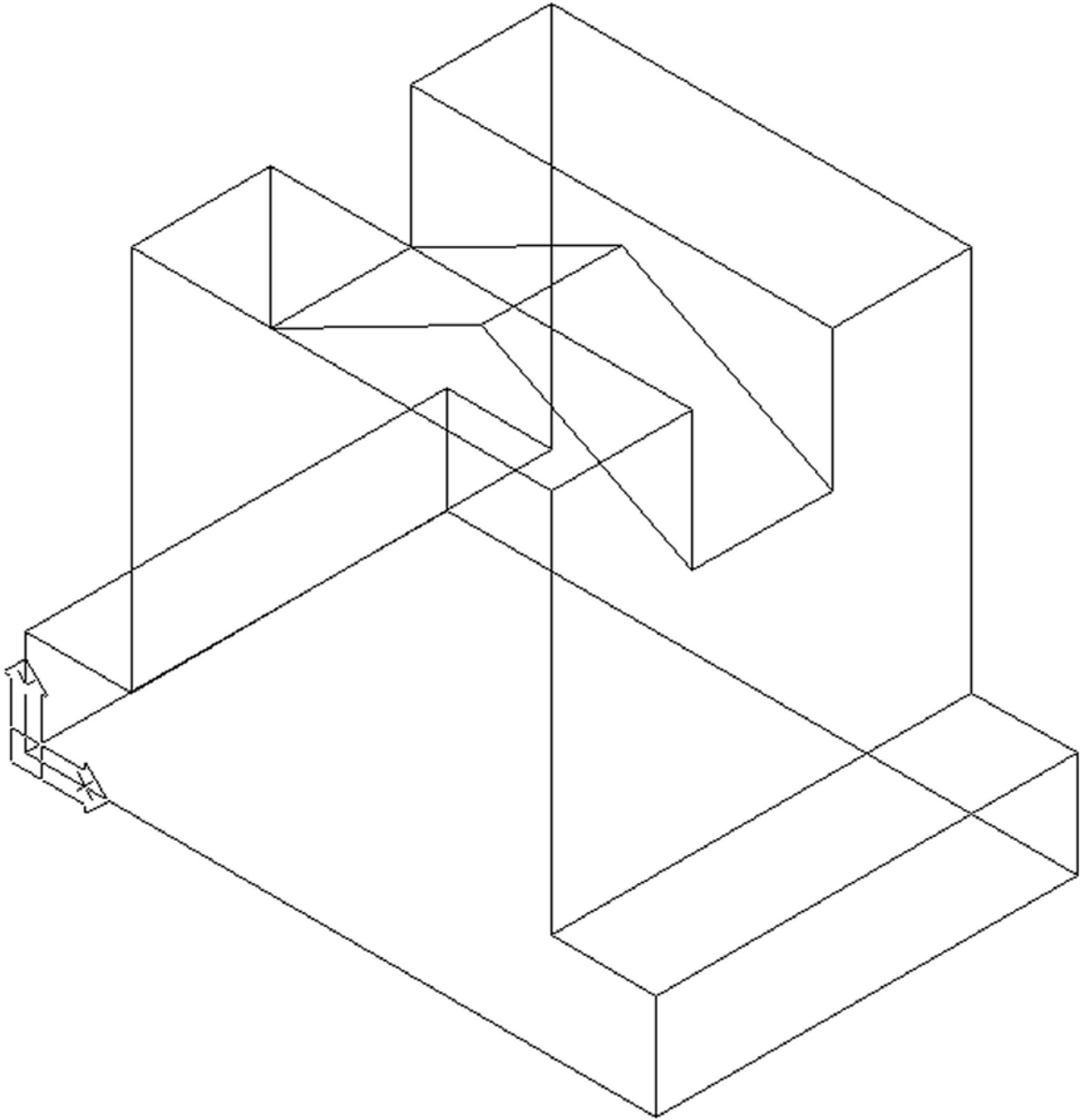


Figure 3.21 – Moving the Finished Solid

Saving the Solid Problem

To save Problem 2, 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 (Figure 3.22).



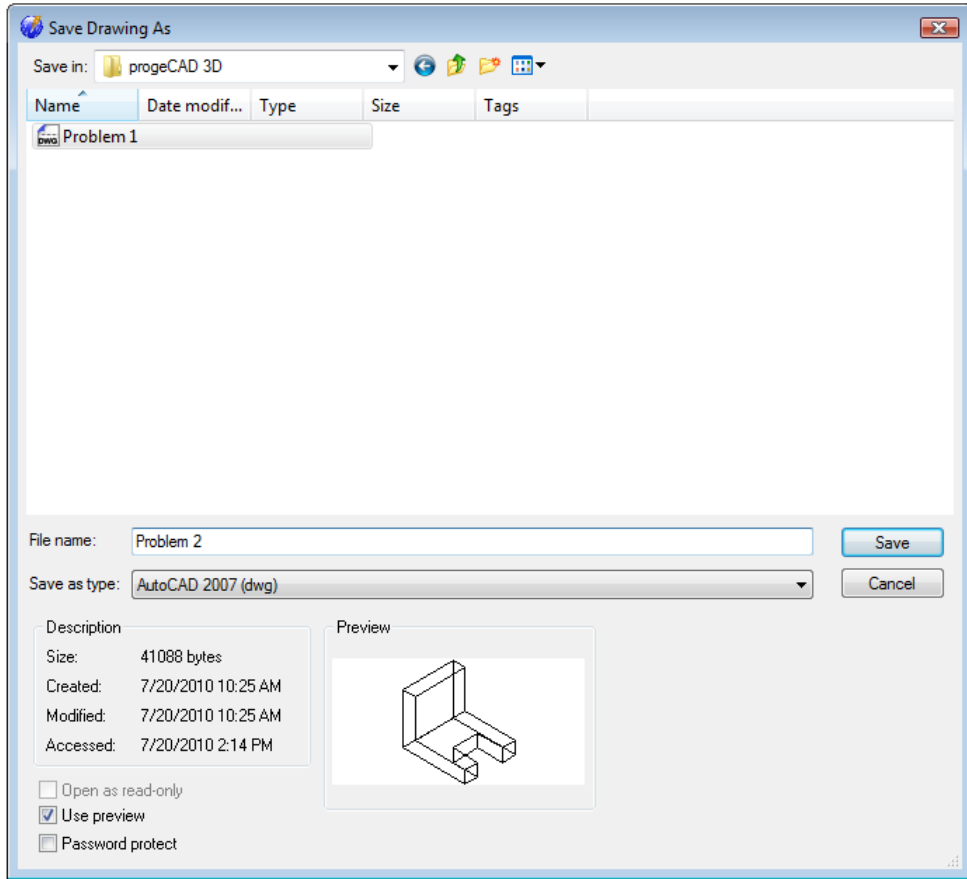


Figure 3.22 – Saving Problem Two

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