Chapter 2

Solid Problem One

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

1. **The Sketch of Solid Problem One**
2. **Starting a 3D Part Drawing**
3. **Drawing your First Solid Boxes**
4. **Moving a 3D Solid**
5. **Subtracting a Solid from another Solid**
6. **Using Union to Join Two Solids**
7. **Improve the View by Hiding Tessellation Lines**
8. **Moving the Solid to the Origin Point**
9. **Saving the Solid Problem**

Sketch of Solid Problem One

As in two dimensional design, three dimensional work starts with a problem, allowing you to work through the design process until a sketch captures all the features that will solve it. In the world of orthographic detailing, an architect, engineer, or designer will layout the information in an alignment of views, typically on paper, that they wish the Computer Aided Design (CAD) professional to load into the progeCAD software application. Many experts in the graphics trade know the isometric technique, showing three sides of the component. The sketch in Figure 2.1 reflects the isometric method of specifying the shape and dimensions you will need to draw your first 3D solid part.

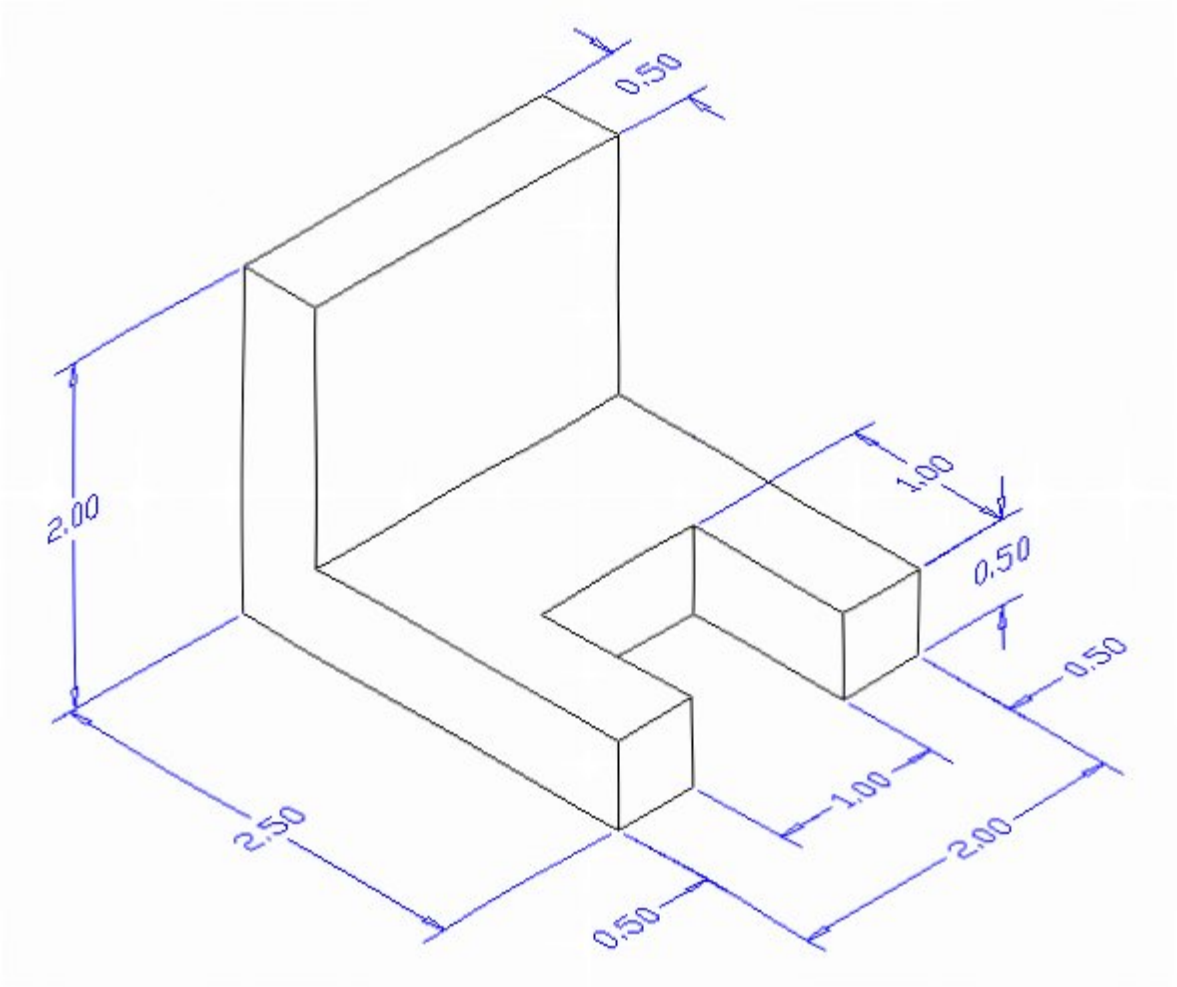


Figure 2.1 – Problem One Sketch

After receiving a sketch from a professional, the CAD technician will try to identify what fundamental features are hidden in the composite solid. The basic building shapes in three-dimensional design are the rectangular or cubic box, sphere, cone, cylinder, and triangular wedge. In your first 3D solid, you might already observe that there are three rectangular boxes. The first 2 inch by 2.5 inch by 0.5 inch box is lying horizontal and is missing a

segment represented by a second box that is a 1 inch by 1 inch by 0.5 inch. The third rectangular box is a vertical 2 inch by 2 inch by 0.5 inch solid. Other specialists in our field may see a few other methods to finish the object. Whatever procedure you use to complete the exercise, most beginners find that the Box tool on the Solids toolbar provides one of the easiest methods to draw the component.

Starting a 3D Part Drawing

In Problem One, you will begin the 3D drawing by selecting the New tool on the Standard toolbar. In Figure 2.2 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.

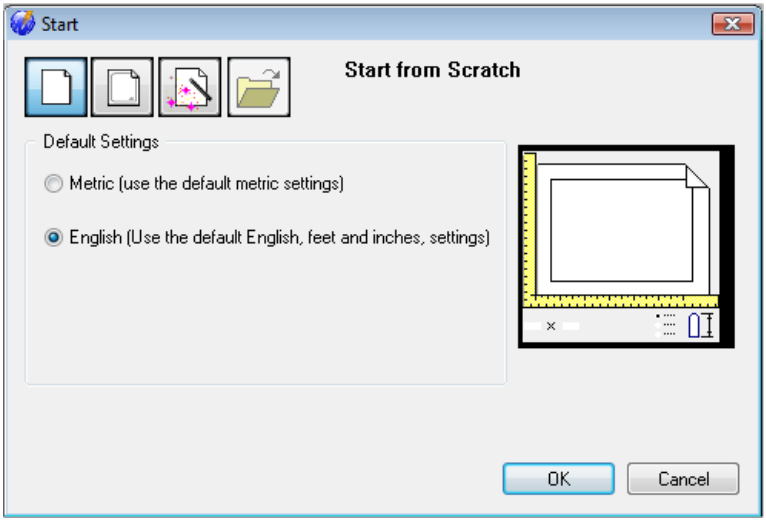


Figure 2.2 – Starting the Drawing from Scratch

Drawing your First Solid Boxes

To draw your first solid rectangular box, select the Box tool on the Solids toolbar. At the command line a prompt asks you to specify the corner of the box, or to type CE to specify the center of the object. You will need to identify the corner of the box by picking anywhere in the middle of the graphical display. After picking the starting point for the box, again the system asks you to denote a corner, type **C** for cube, or **L** for length. You can choose to pick another corner by using the mouse or to type a relative point from the first point selection (like **@2,2.5,0.5,** or specific coordinate like **2,3,1)**; either way will work to draw the box. If you type **C** for cube, the computer will ask you for the length. When you type a number and hit ENTER the cube will be drawn.

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 **W** coordinate. In Problem One, type L, then **2** for the X coordinate, **2.5** for the **Y** coordinate and **0.5** for the **Z** coordinate. The actual input from the command line is shown in Figure 2.3.

```
Command :_BOX
Center/<Corner of box> <90227209139.1890,-115954823827.1443,0.0000>:
Cube/Length/<Opposite corner>: L
Length of side of box: 2
Width of box: 2.5
Height of box or [2Points]: 0.5
```

Figure 2.3 – Command Line Prompts for the Box Command

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 (Figure 2.4).

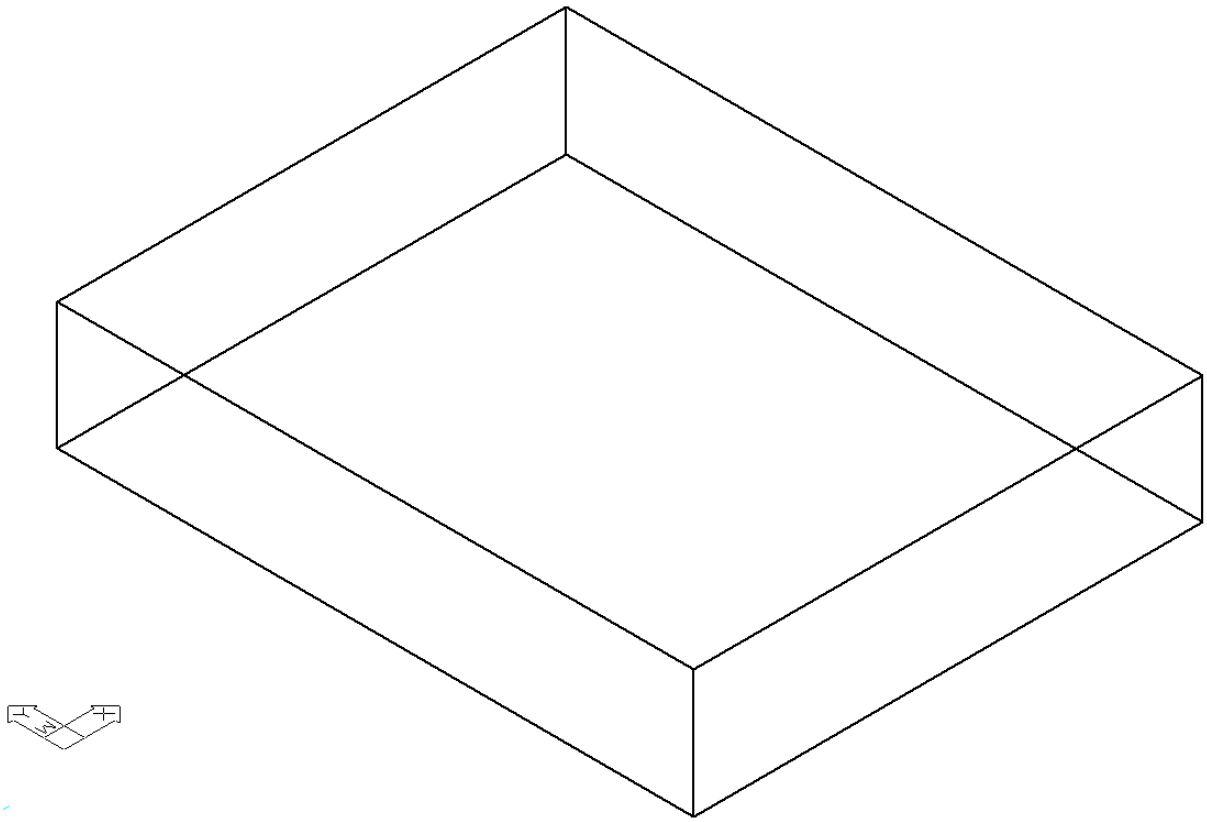


Figure 2.4 – Drawing your First 3D Box

For the second solid box, 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 the endpoint as shown in Figure 2.5. Type **L** after specifying the starting point and the system will again ask you three questions. The first prompt asks for the "Length of side of box:" so type **2** and ENTER for the X coordinate. The second prompt asks for the "Width of box:" so type **-0.5** for

the **y** coordinate. The third prompt asks for the "Height of box or [2Points]:" so type **2** and ENTER for the **Z** coordinate. The second box will appear as shown in Figure 2.6.

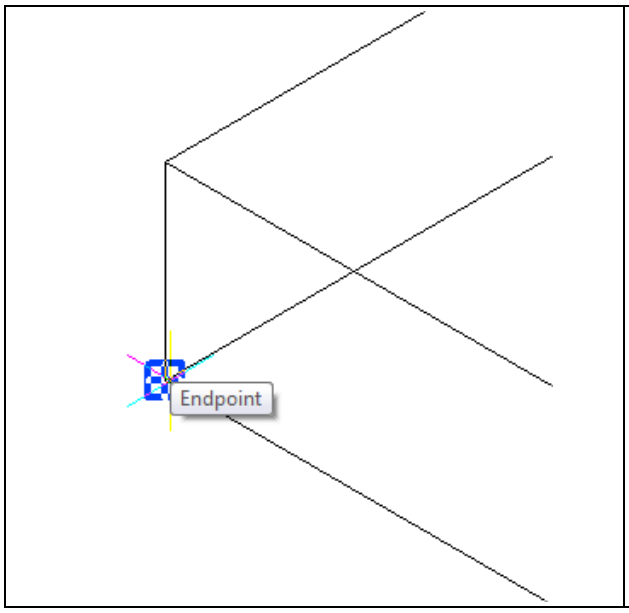


Figure 2.5 – Starting Point for Next Box

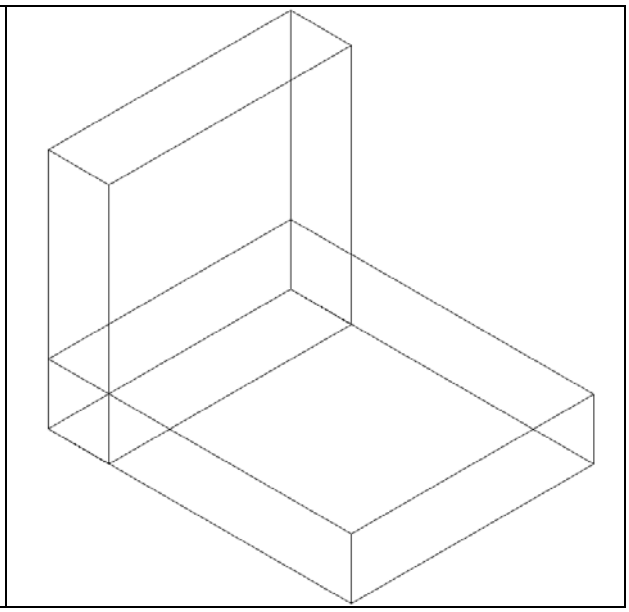


Figure 2.6 – Drawing the Next Box

You need to draw a third box that you will subtract from the larger box, resulting in the appearance of two prongs.

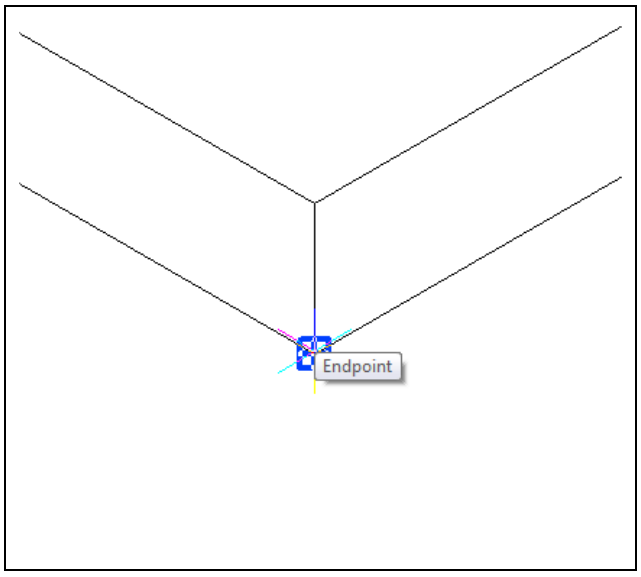
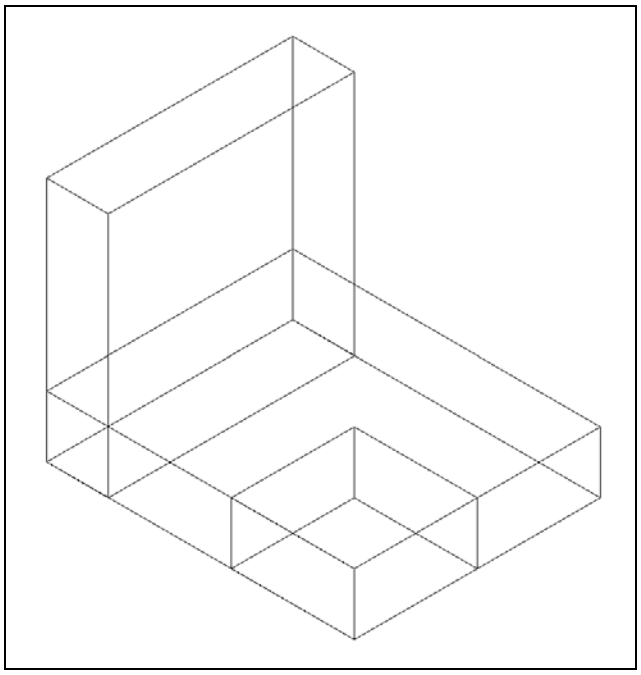


Figure 2.7 – Starting Point for Next Box

Again, select the Box tool on the Solids toolbar. When the command line prompts you to specify the corner of the solid for this box, specify the endpoint as shown in Figure 2.7. Type **L** after specifying the starting point and the system will ask you three questions. For the first prompt "Length of side of box:" type **1** and ENTER for the X coordinate. For the second prompt "Width of box:" type **1** for the **y** coordinate. When the third prompt "Height of box or [2Points]: appears, type **0.5** and ENTER for the **Z** coordinate. The second box will appear as shown in Figure 2.8.

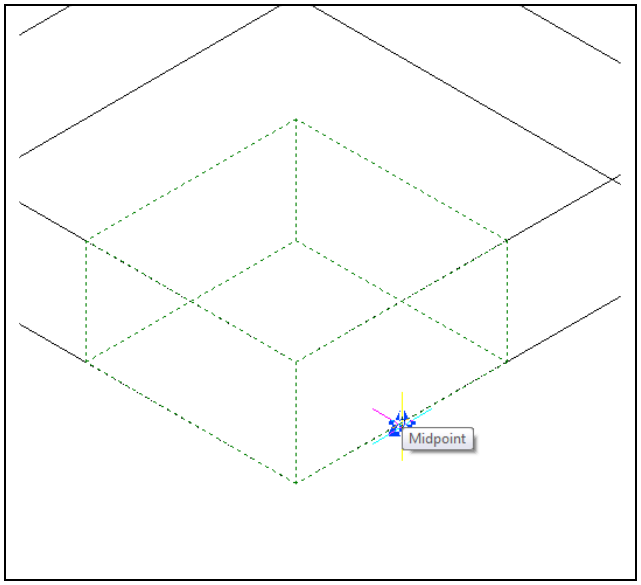


Many simple drawings involving plain shapes, such as those designs made from finished wood, involve placing a rectangular box at some point in the drawing. So far in Problem One, you have created a base solid and then placed other solids on the initial component by strategically picking a starting point. Observe the UCS icon displaying the X, Y and Z coordinates which match the length, width and height prompts in the Box command. Depending on where you pick the starting location, the numbers representing the different coordinates can be positive or negative, which will extend the solid in that particular direction. Now that you have all three solid boxes in the drawing file, you need to adjust the position of the smallest box before removing it.

Figure 2.8 – Inserting the Third Box

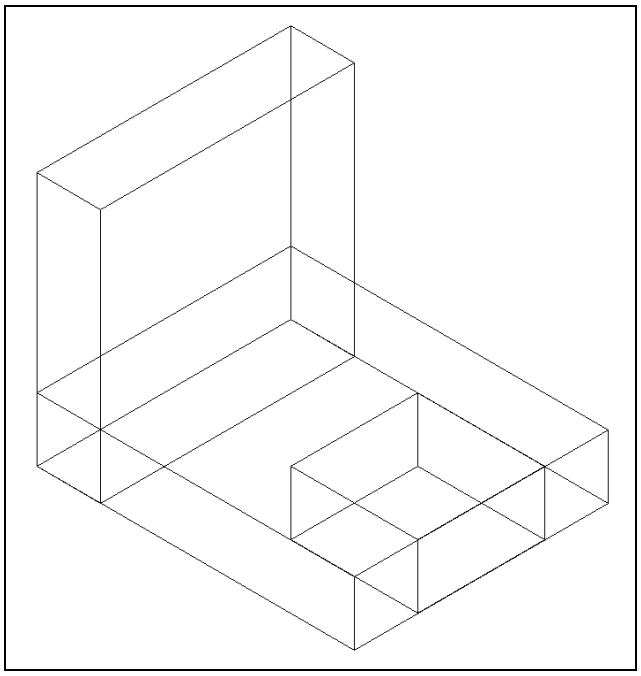
Moving a 3D Solid

The Move command is actually a three-dimensional function in progeCAD. 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, where the Z coordinate is not 0. You can use Move to relocate any entity to another point (X,Y,Z) in Model Space.



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 entity such as a solid box, select the Move tool on the Modify toolbar. Once you choose the Move command, the command line will prompt you to "Select entities to move:". 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 "Entities in set: 1," so hit ENTER to go the second part of the command.

Figure 2.9 – Base Point to Move the Box

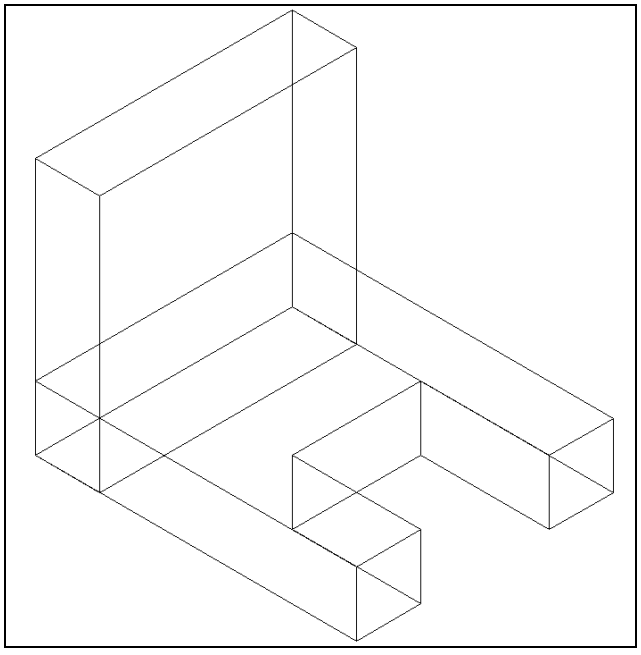


Now you need to specify a "Vector/<Base point>:" on the graphical display. Pick your base point with the Midpoint Esnap as shown in Figure 2.9. The "Displacement point:" is the center point of the 2 inch box. Pick the second point using the Midpoint Esnap of the larger box, taking notice that the 1 x 1 x 0.5 box is now centered on the 2 inch box (Figure 2.10). Again, you have accomplished another milestone by moving a progeCAD entity using two known points. You can use the Move command to relocate any entity as simply as you did in the 2D Drawing with progeCAD Smart! 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.

Figure 2.10 – Centering the Third Box

Subtracting a Solid from another Solid

To modify a 3D solid shape in progeCAD, you need to learn a series of Boolean commands such as Union and Subtract. The first editing tool you will learn will be the Subtract command, which will allow you to remove one shape from another.



To remove the smaller solid box from the larger one, select the Subtract tool on the Solids Editing toolbar. The progeCAD program will prompt you with "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:." Pick the small 1 x 1 x 0.5 box and the command line will reply with "Entities in set: 1." Hit ENTER and a region exactly the shape of the small box will be removed from the larger one as shown in Figure 2.11. This technique to create complex details by removing simple solids from the main design is commonplace in the world of 3D design.

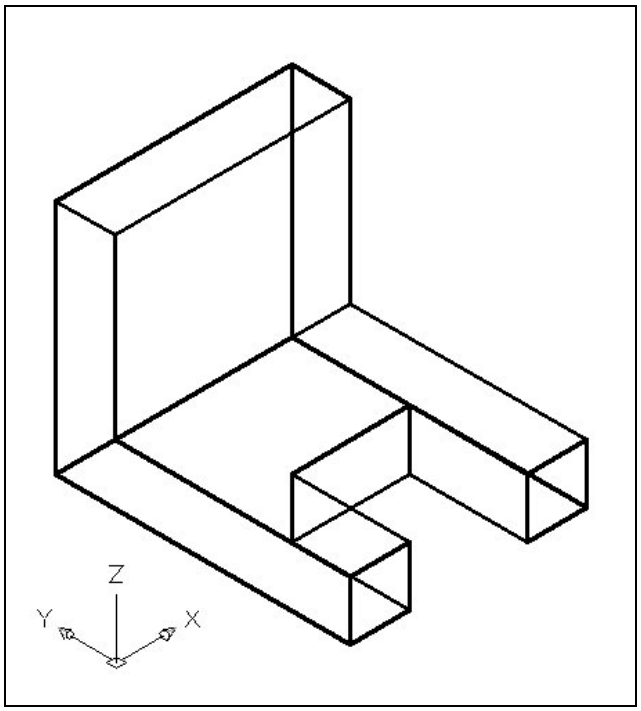
Figure 2.11 – Subtracting the Third Box

```
Command : _subtract
Select ACIS object to subtract from:
Entities in set: 1
Select ACIS object to subtract from:
Select ACIS objects to subtract:
Entities in set: 1
Select ACIS objects to subtract:
```

Figure 2.12 – Command Line for Subtracting the Third Box

Union Two Solids

For the next modification of the 3D solid, you need to learn the second Boolean command, Union. With the Union command, you will learn how to join one or more solids together.

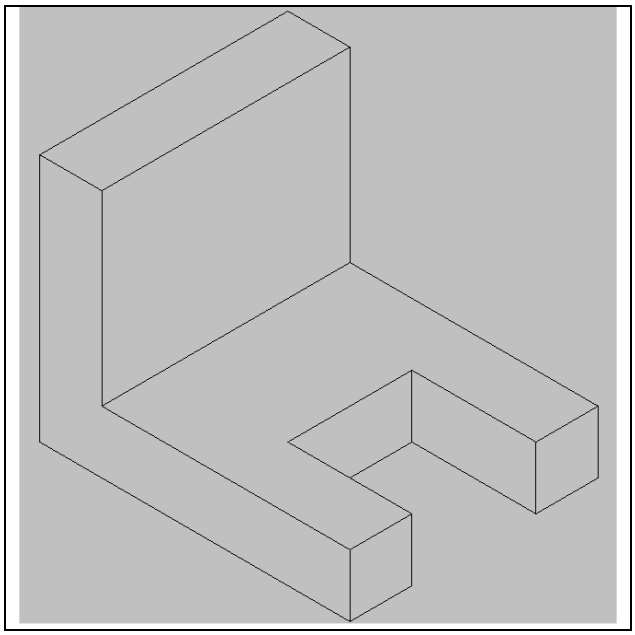


To add the two solid boxes together, select the Union tool on the Solids Editing toolbar. The command line will prompt you with "Select ACIS objects to union:." Select the horizontal solid box and the system will respond with "Entities in set: 1." Pick the vertical solid box and the command line will counter with "Entities in set: 2." Hit ENTER to combine both solids into one as shown in Figure 2.13. You can see in the graphical display, the lines outlining both boxes that were crossing each other where the two boxes joined are now missing. Select the solid with one pick of the mouse and the entire solid will become highlighted. In this case the two solids were touching each other. You will find out in future projects that the Union command can also connect separated solids.

Figure 2.13 – Union the Two Boxes

Improve the View by Hiding Tessellation Lines

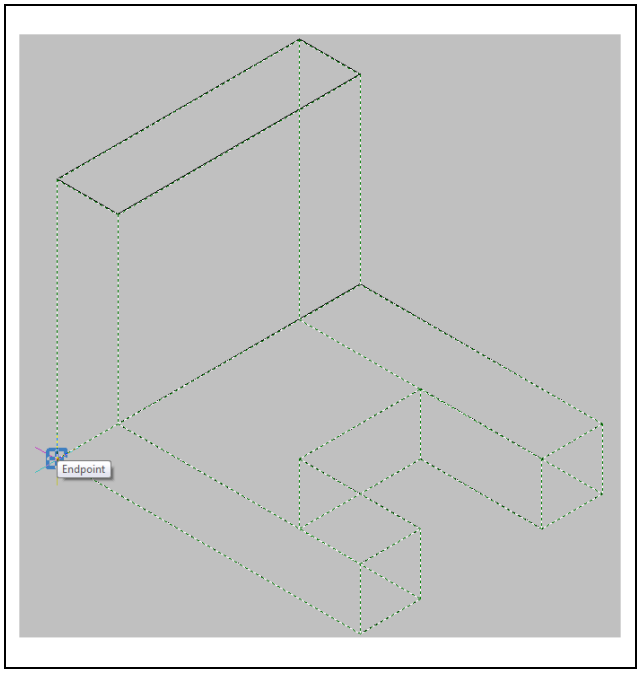
In Figure 2.14, you will notice that the tessellation lines representing the solid's outline on the far edge of the part are visible. In a real world solid part, you would not be able to see through solid unless the material was transparent. You can use the Hide command to remove the hidden lines from your graphical display.



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 2.14 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 2.14 – Hiding the Tessellation Lines

Moving the Solid to the Origin Point



To move the Problem One 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 shown in Figure 2.15 at the lower right hand side of the Problem One, which would become the Problem One insertion point if the part was inserted into an assembly drawing. For the "Displacement point:," type 0,0,0 and ENTER. Problem One will move to the new origin point as shown in Figure 2.16. 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.

Figure 2.15 – Moving the Finished Solid

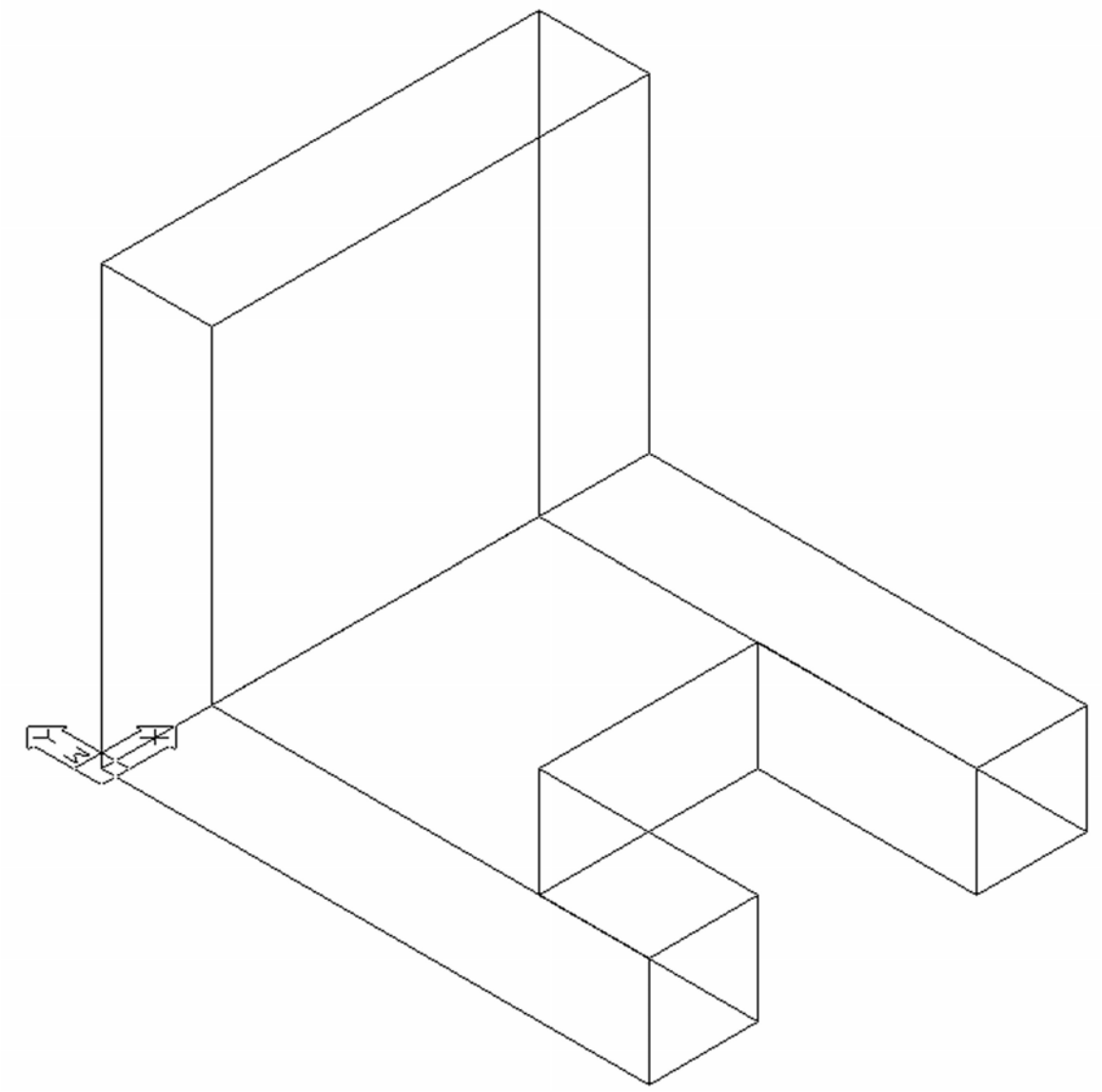


Figure 2.15 – Moving the Finished Solid

Saving the Solid Problem

To save Problem 1, 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 1” and press the Save button to save the drawing (Figure 2.16).

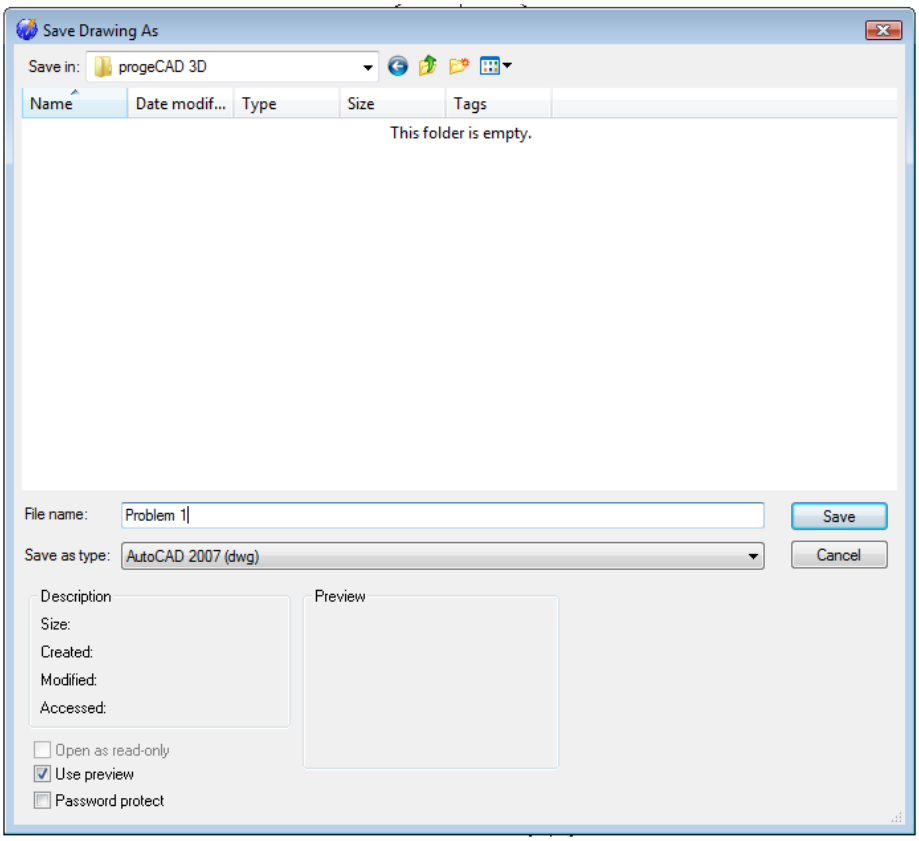


Figure 2.16 – Saving Problem One

*** World Class CAD Challenge 101-01 * - Close this drawing file. Create a New file and draw the three solid boxes, move and subtract the third box and union the two remaining solids. 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.**