

Chapter

2

Solid Problem One

This chapter will cover the following to World Class standards:

- **Sketch of Solid Problem One**
- **Starting a 3D Part Drawing**
- **Drawing your First Solid Boxes**
- **Moving a 3D Solid**
- **Subtracting a Solid from another Solid**
- **Union Two Solids**
- **Improve the View by Hiding Tessellation Lines**
- **Moving the Solid to the Origin Point**
- **Saving the Solid Problem**

Sketch of Solid Problem One

As in two-dimensional design, three-dimensional work starts with a problem, and then involves working through the design process until a sketch captures all the features that will solve the dilemma. In the world of orthographic detailing, an architect, engineer or designer will layout the information in an alignment of views, typically on paper, which they wish the Computer Aided Design (CAD) professional to load into the AutoCAD software application. Many experts in the graphics trade know the isometric technique, where three sides of the component are shown. 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.

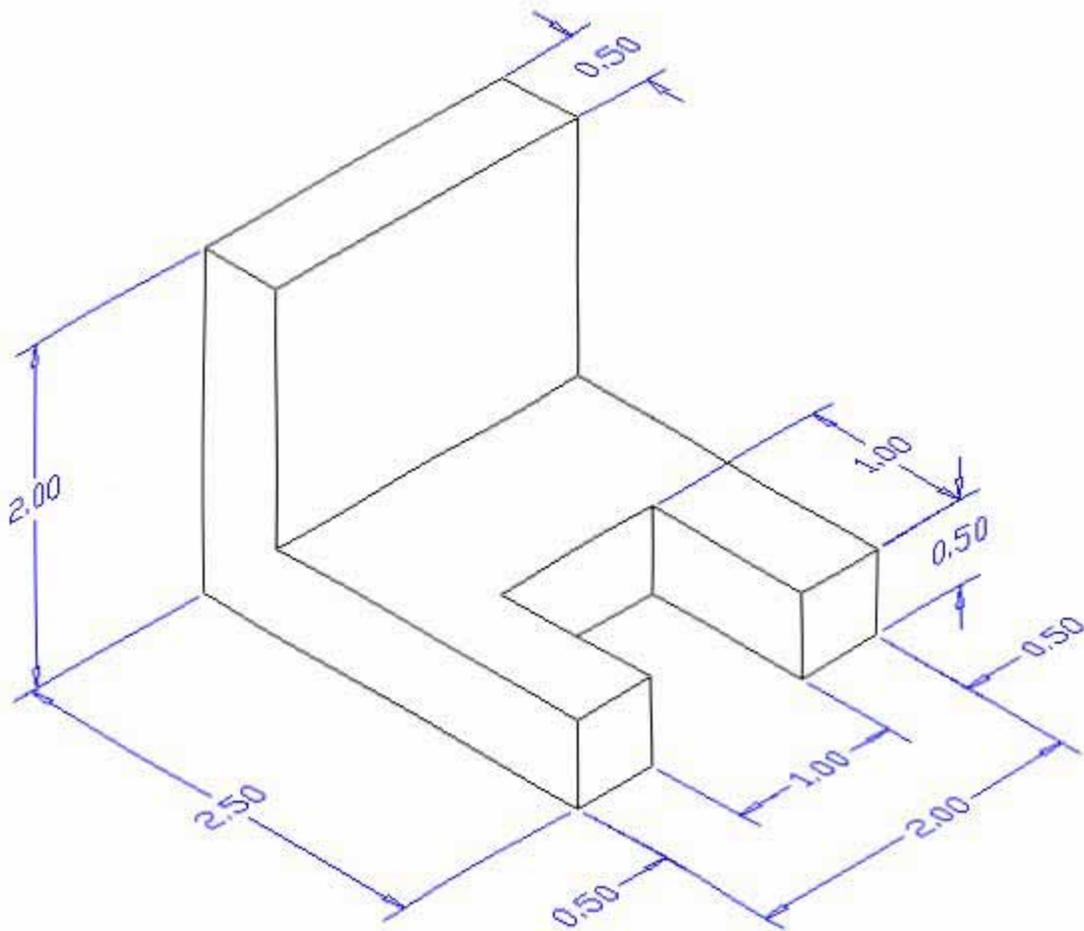


Figure 2.1 – Problem One sketch

When 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 the three dimensional design are rectangular or cubic box, spherical shapes, conical shapes, cylindrical shapes and the 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.

As you have already seen from your 2d drafting training, AutoCAD 2009 has a different interface than previous versions. The interface for 3d modeling is also different. To access the 3d modeling workspace, select workspace switching icon in the bottom right-hand corner of the screen. Select 3D Modeling from the menu that appears. Your workspace will change to more efficiently accommodate 3D modeling techniques. At this point, you will not need the Tool Pallets located on the right side of the screen, so move your mouse to the top right-hand corner of the pallet, and when an X and a Minimize bar appear, select the X to close. You are now ready to begin the first solid problem.



Starting a 3D Part Drawing

In Problem One, you will begin the 3D drawing by selecting the template you created in the previous 2D textbook. In Figure 2.2, the window “Create New Drawing” is loaded by selecting the New tool on the Quick Access 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.

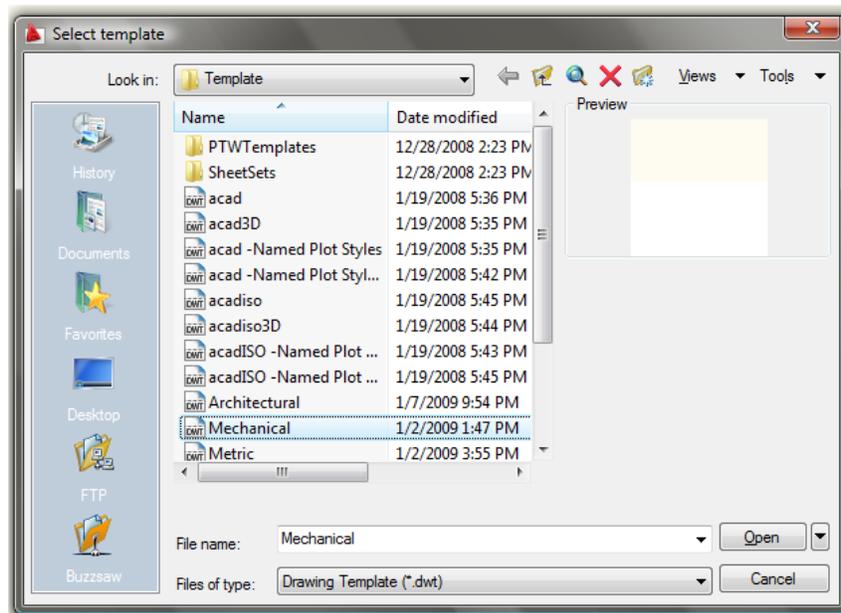


Figure 2.2 – Starting the drawing using the mechanical template

Drawing your First Solid Boxes

To draw your first solid rectangular box, select the Box tool from the 3D modeling panel under the Home tab. At the command line, a prompt asks you to specify the corner of the box or 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. If you choose to pick another corner by using the mouse, or type a relative point from the first point selection like **@2,2.5,0.5**, or a specific coordinate like **2,3,1**, the box will be drawn on the monitor. If you type **C** for cube, and the computer will ask you for the length and when you type a number and ENTER, the cube is 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 “Specify Length”: type a number and hit ENTER to designate the **X** length. The second prompt is “Specify Width”: type a number and hit ENTER to designate the **Y** length. The third prompt is “Specify Height”: type a number and hit ENTER to designate the **Z** length. For 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
Specify corner of box or [Center] <0,0,0>:
Specify corner or [Cube/Length]: L
Specify length: 2
Specify width: 2.5
Specify height: 0.5
```

Figure 2.3 – Starting the drawing using the mechanical template

On the View panel under the Home tab, select the Southwest Isometric from the drop-down menu to see the part on the XY plane. The vertical Z-axis will rise from the level surface (Figure 2.4).

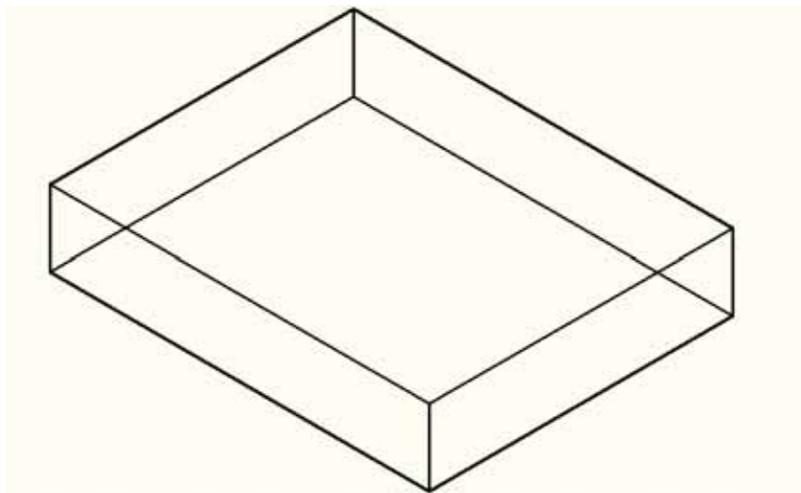


Figure 2.4 – Drawing your first 3d box

For the second solid box, select the Box tool again from the 3D Modeling toolbar. At the command line, when the prompt asks you to specify the corner of the solid and for this box, specify the endpoint as shown in Figure 2.5. Then, type **L** after specifying the starting point and the system will again ask you three questions. At the first prompt, “Specify Length,” type **2** and ENTER for the X coordinate. At the second prompt, “Specify Width,” type **0.5** for the Y coordinate. At the third prompt, “Specify Height,” type **2** and ENTER for the Z coordinate. The second box will appear as shown in Figure 2.6.

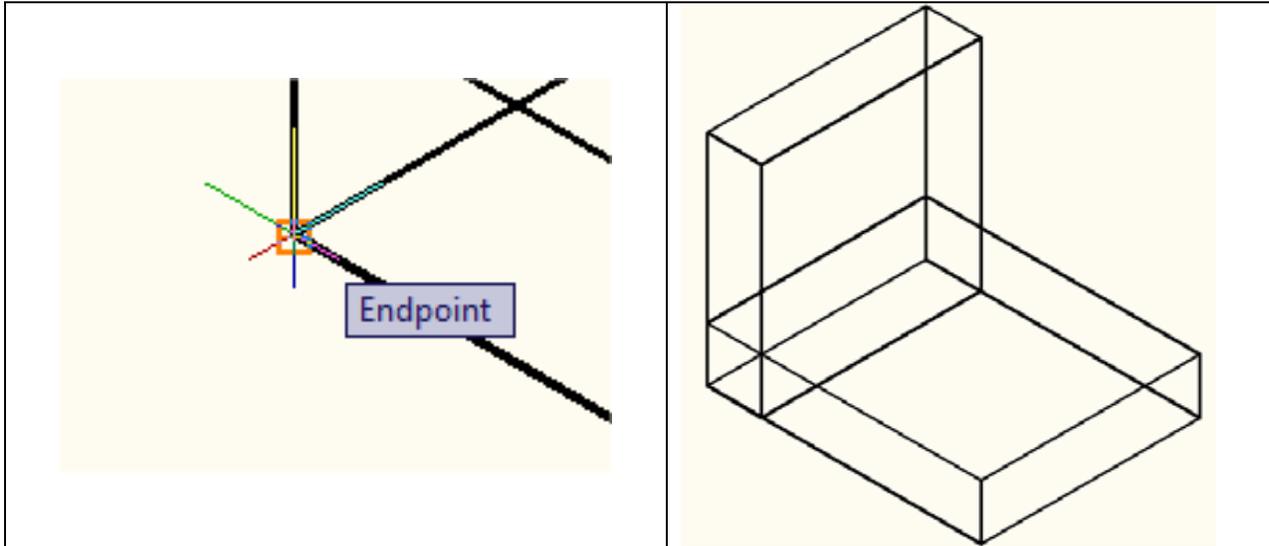


Figure 2.5 – Starting point for next box

Figure 2.6 – Drawing the next box

You need to draw a third box. When you subtract the shape from the larger box, it will create a two-prong fork appearance.

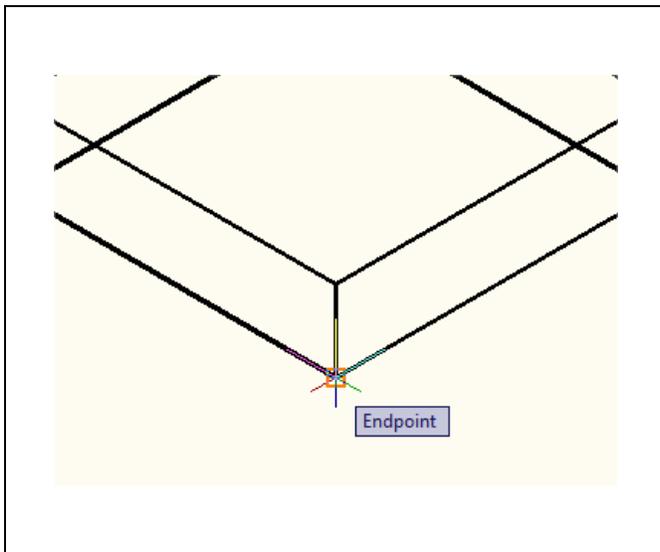
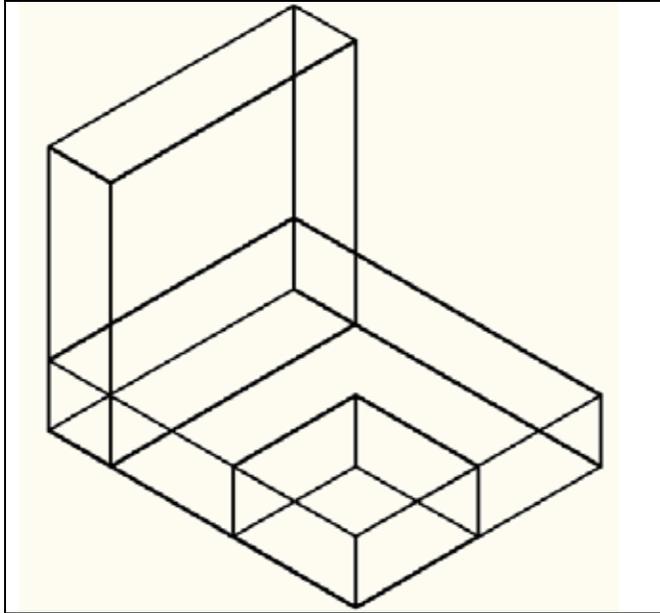


Figure 2.7 – Starting point for next box

Again, select the Box tool from the 3D Modeling panel. At the command line, a prompt asks you to specify the corner of the solid and for this box, so specify the endpoint as shown in Figure 2.7. 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 **1** for the y coordinate. The third prompt is “Specify Height” where when you type **0.5** and ENTER for the Z coordinate. The second box will appear as shown in Figure 2.8.

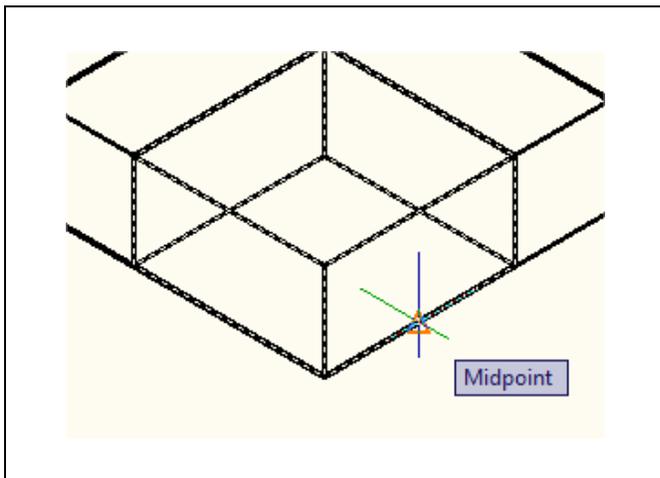


Unlike your training in 2D drafting, 3D modeling requires more attention to detail in regards to the model space. Many actions in 2D required little or no attention to the model space, as the keyboard and command line commands resulted in the same exact actions every time you executed them. In 3D model space, however, the position of your cursor on the screen can change many commands; the addition of a Z plane adds a bit more complexity to the equation. If you seem to be having trouble with a part of a problem, try the command again but with the cursor in a different position. Eventually you will learn where and where not to place your cursor during certain commands.

Figure 2.8 – Inserting the third box

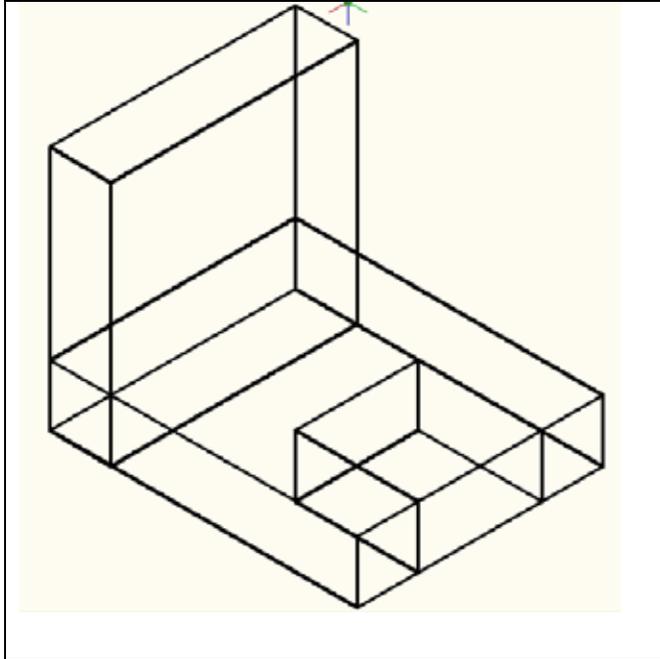
Moving a 3D Solid

Now that you have all three solid boxes in the drawing file, you need to adjust the position of the smallest box before removing the shape using the Move command. 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, when 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 AutoCAD entity such as a solid box, you select the Move tool on the Modify panel. 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 hit ENTER to go the second part of the command.

Figure 2.9 – Base point to move the box

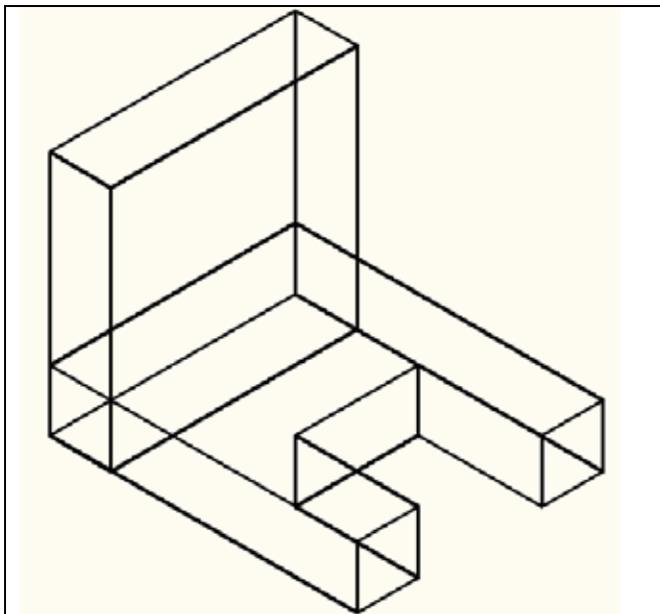


Next, you need to “specify base point or displacement” on the graphical display. Pick your base point with the Midpoint Osnap as shown in Figure 2.9. The “second point of displacement” is the midpoint on the edge of the 2-inch box. Pick the second point using the Midpoint Osnap of the larger box. Notice 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 by moving an AutoCAD entity with 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.

Figure 2.10 – Centering the third box

Subtracting a Solid from another Solid

To modify a 3D solid shape in AutoCAD, 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 Solid Editing panel. AutoCAD will prompt you with “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 the small 1 x 1 x 0.5 box and the prompt will read “1 found.” Hit ENTER and a region exactly the shape of the small box will disappear 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.

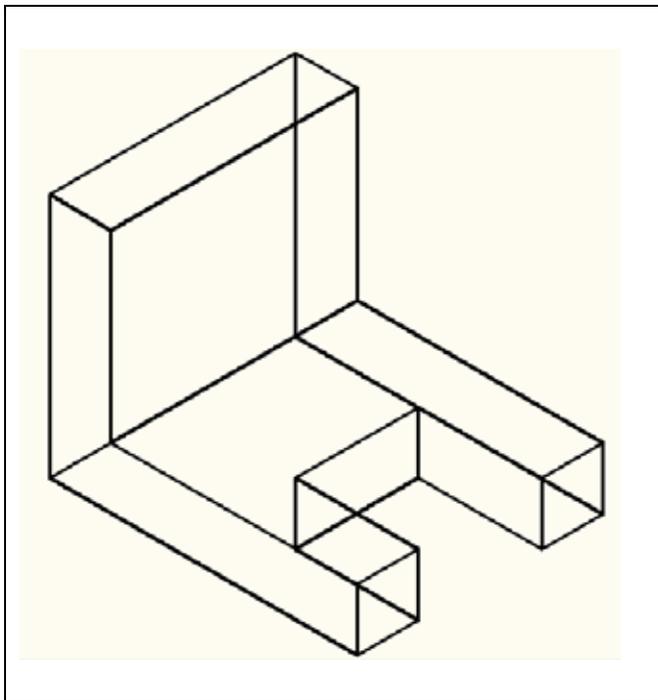
Figure 2.11 – Subtracting the third box

```
Command: _Subtract
Select solids and regions to subtract from..
Select objects: 1 found
Select objects: Select solids and regions to subtract..
Select objects: 1 found
```

Figure 2.12 – Command line for subtracting the third box

Union Two Solids

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



To add a solid box to another one, select the Union tool on the Solid Editing panel. Then the AutoCAD program will prompt you with “Select objects.” Select the horizontal solid box and the system will respond with “1 found.” Pick the vertical solid box and you will be prompted with “1 found, 2 total.” Now hit ENTER and both solids will become one entity, as shown in Figure 2.13. You can see in the graphical display that the tessellation lines outlining both boxes and crossing each other in the corner of the “L” shape 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 works also with solids not in contact.

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 the solid unless the material was transparent. You can use the 3D Hidden Visual Style command to remove the hidden lines from your graphical display. 

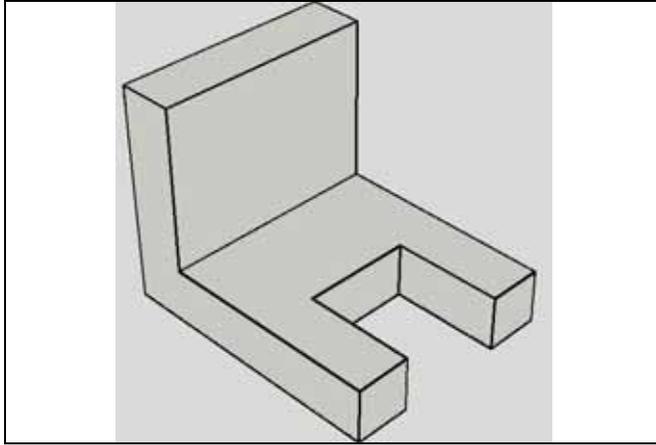


Figure 2.14 – Hiding the tessellation lines

Select the 3D Hidden Visual Style tool from the drop-down menu on the View panel. This is a very simple command, because after picking the tool, the tessellation lines representing the back of the part are removed from view. Rolling over the part in this visual style makes the tessellation lines reappear. Notice in Figure 2.14 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 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. You will also notice a box in the upper right-hand corner of the screen that is on a compass. This box, called the ViewCube, lets you control the view perspective of the model space in relation to the origin. Experiment with it to learn how to control the views while in different viewing modes. Also, right-click and select Perspective (rather than Parallel) for a more realistic view of the part. Perspective view is shown in Figure 2.14. After you have moved the part to the origin point, go back to the Hidden visual style and experiment with the ViewCube again.

Moving the Solid to the Origin Point

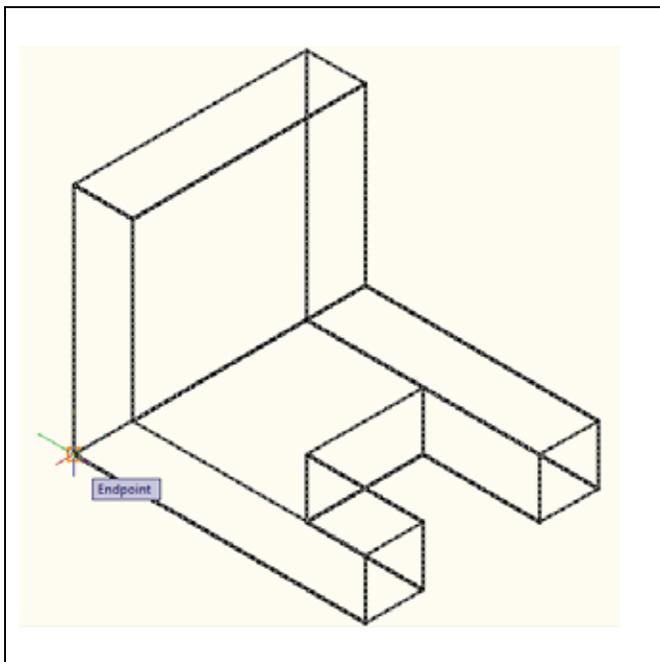


Figure 2.15 – Moving the finished solid

To move Problem One to the drawing origin of 0,0,0 on the X, Y, and Z axis, pick the Move tool on the Modify panel, select the 3D solid, and hit ENTER. At the prompt, “specify base point or displacement,” select the endpoint shown in Figure 2.15 at the lower right hand side of the Problem One, which will become the Problem One insertion point into an assembly drawing. At the “second point of displacement,” 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 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.

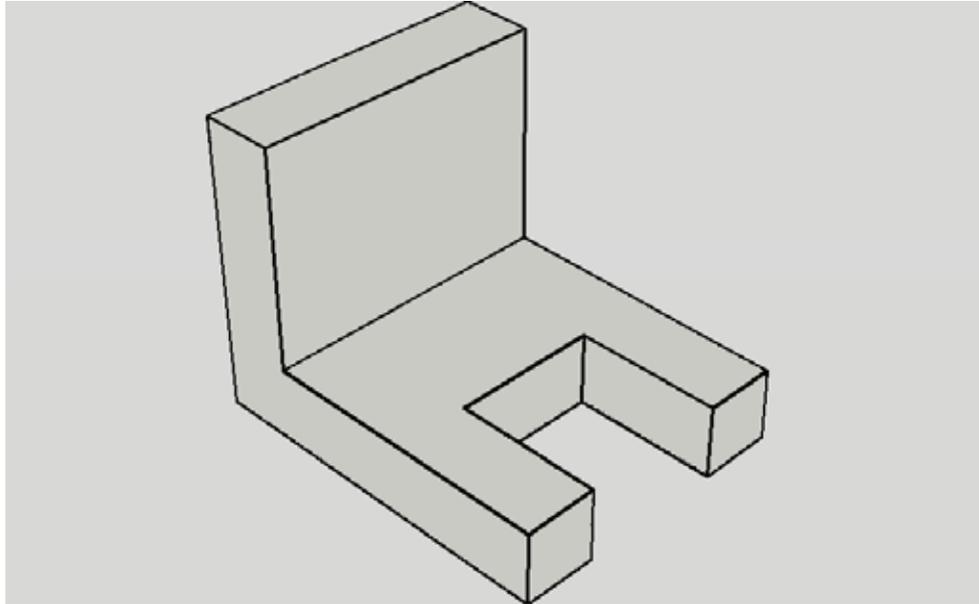


Figure 2.15 – The finished solid

Saving the Solid Problem

To save Problem 1 in your Fundamentals of 3D Drawing Folder, select the Save 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 1” and press the Save button to save the drawing (Figure 2.16).

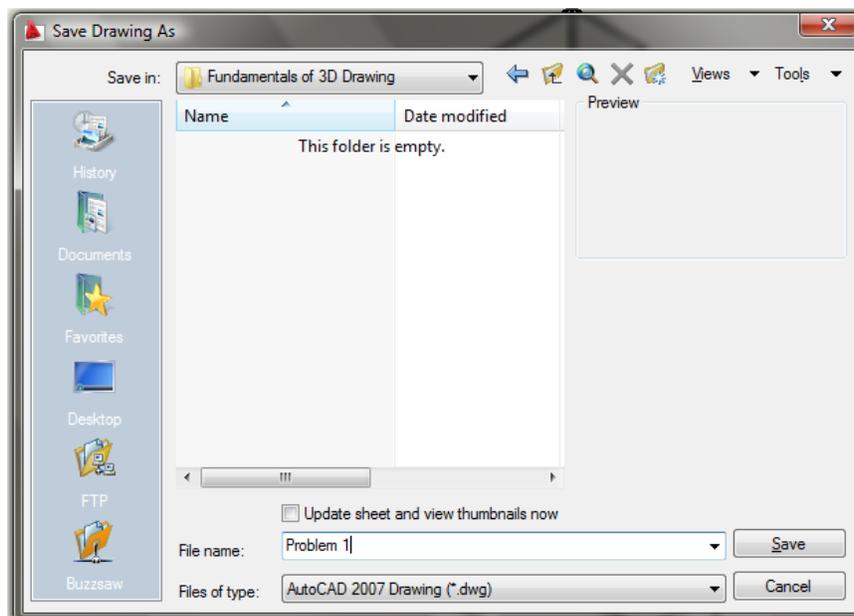


Figure 2.16 – Saving problem one

*** World Class CAD Challenge 03-01 * - 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.**

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

