

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 of two solids**
- **Improving the view by Hiding tessellation lines**
- **Moving the solid to the origin point**
- **Saving the solid problem**

Sketch of Solid Problem One

Like two-dimensional design, a three-dimensional designer starts with a problem, and then works through the design process, eventually creating a sketch that includes all the features of the solution. In the world of orthographic detailing, an architect, engineer, or designer will lay out the information in an alignment of views, typically on paper, that they wish the Computer Aided Design (CAD) professional to enter into the AutoCAD software application. Many experts in the graphics trade know the isometric technique, which shows three sides of the component in one view. The sketch in Figure 2-1 shows solid problem one in an isometric view along with the dimensions you will need to draw your first 3D solid part.

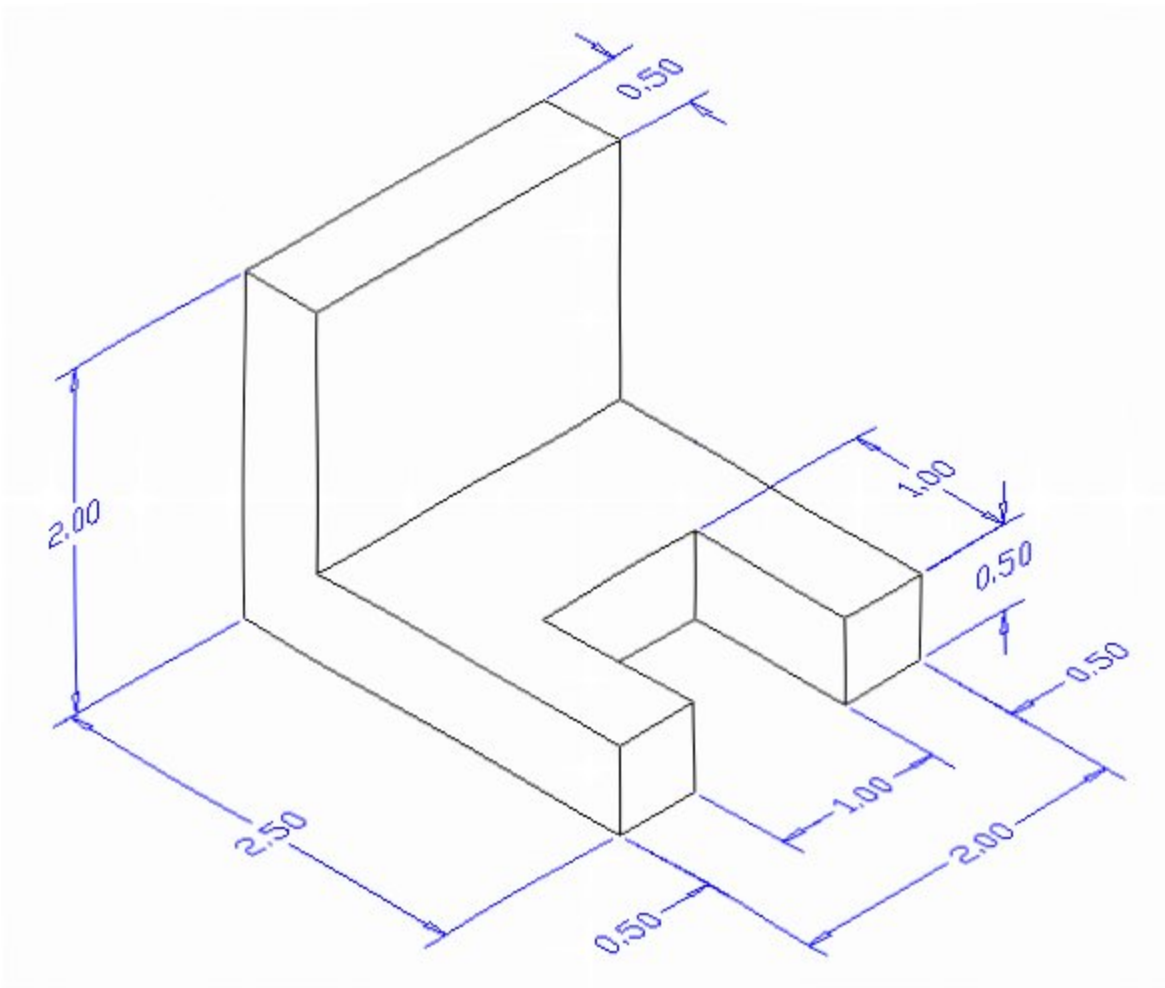


Figure 2.1 – Problem One sketch

After receiving a sketch from a designer, the CAD technician should try to identify what fundamental features are hidden in the composite solid. The basic building shapes in three dimensional designs are the rectangular or cubic box, spherical shapes, conical shapes, cylindrical shapes, and the triangular wedge. You may have already observed that there are three rectangular boxes in this particular 3D solid. 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 this component.

As you have already seen from your 2D drafting training, AutoCAD 2012 has a different interface than previous versions. The interface for 3D modeling is also different. To access the 3D modeling workspace, select the workspace switching drop menu next to the Quick Access toolbar in the top left area of the screen. Select 3D Modeling from the menu that appears. Your workspace will change to more efficiently accommodate 3D modeling techniques. You are now ready to begin the first solid problem.

Starting a 3D Part Drawing

You will draw Problem One 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.

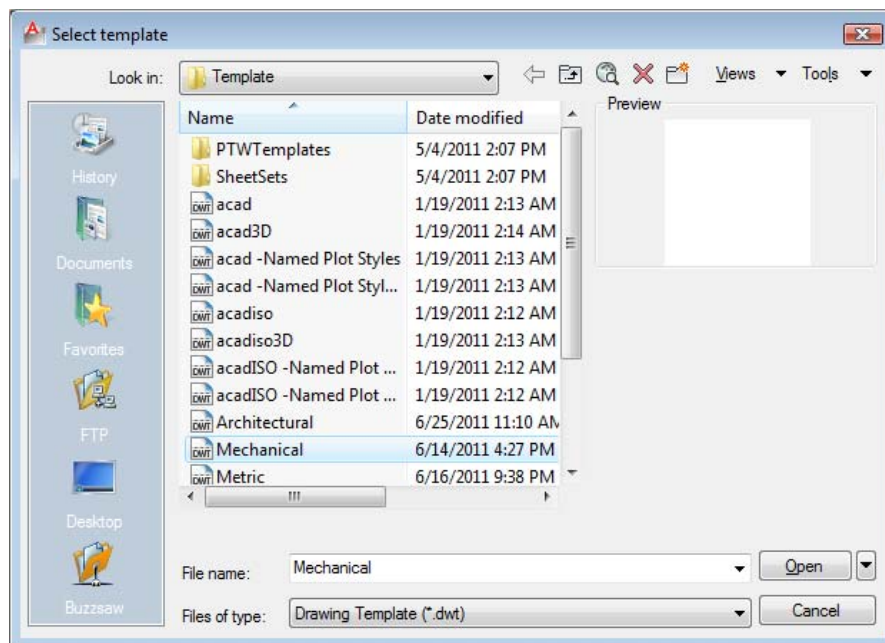
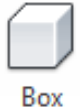


Figure 2.2 – Starting the drawing using the Mechanical template

Drawing your First Solid Boxes

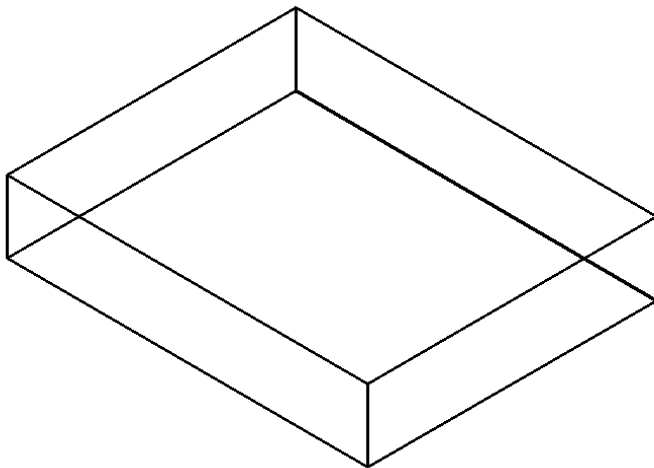
To draw your first solid rectangular box, select the Box tool from the Modeling panel under the Home tab. At the command line, a prompt asks you to specify the corner of the box or type CE to specify the center of the object. You will need to identify the corner of the box by picking anywhere in the 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**, the box will be drawn on the monitor. If you type **C** for cube, the computer will ask you for the length; you would enter a value for the length and hit ENTER to finish the cube using this method.



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 length on the **X axis**. The second prompt is “Specify Width”: type a number and hit ENTER to designate the length on the **Y axis**. The third prompt is “Specify Height”: type a number and hit ENTER to designate the length on the **Z axis**. For Problem One, type **L**, then **2** for the X coordinate, **2.5** for the y coordinate, and **0.5** for the Z coordinate. Be aware that if you have Dynamic Input toggled on, the position of your cursor will affect the direction that each measurement extends on the screen. The actual input from the command line is shown in Figure 2.3.

```
Command: _box  
Specify first corner or [Center]:  
Specify other corner or [Cube/Length]: L  
Specify length: 2  
Specify width: 2.5  
Specify height or [2Point]: 0.5
```

Figure 2.3 – The Box command line prompts



On the View panel under the Home tab, select the Southwest Isometric from the drop-down menu 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 2.4).

Figure 2.4 – Drawing your first 3d box

Select the Box tool again from the Modeling panel to draw your second solid box. When the prompt asks you to specify the corner of the solid and for this box at the command line, specify the endpoint Osnap as shown in Figure 2.5. You may need to toggle the 3D Object Snap mode off so that the 3D Osnap modes do not interfere with your usual 2D Osnap selections. To do this just click the 3D Osnap button on the tray. Type **L** after specifying the starting point of the box 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. Remember that if you have Dynamic Input toggled on, the position of your cursor will affect the direction that each measurement extends on the screen. The second box should appear as shown in Figure 2.6.

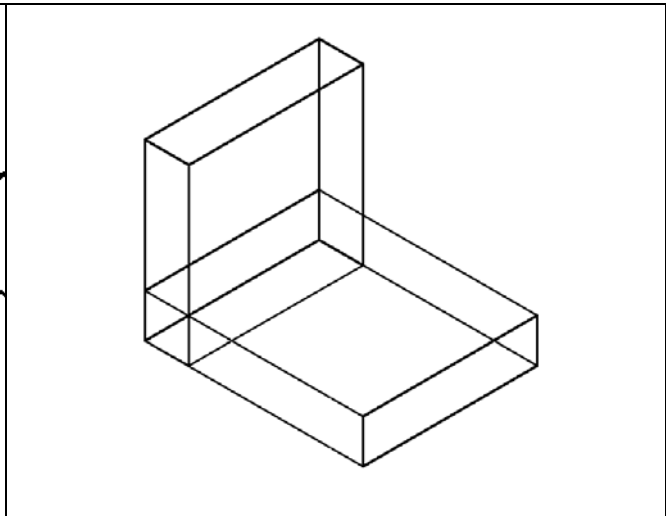
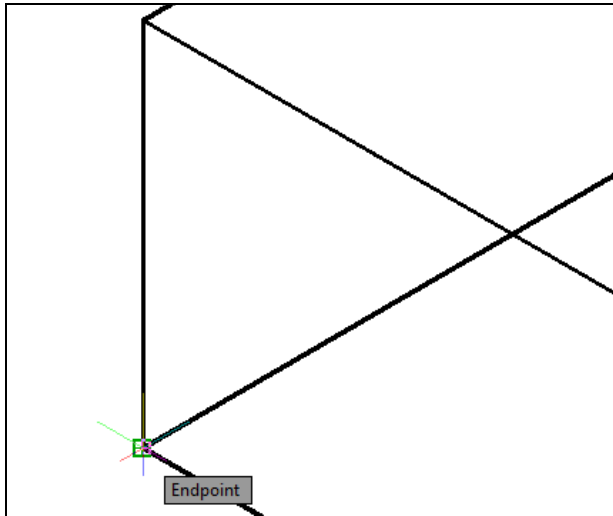
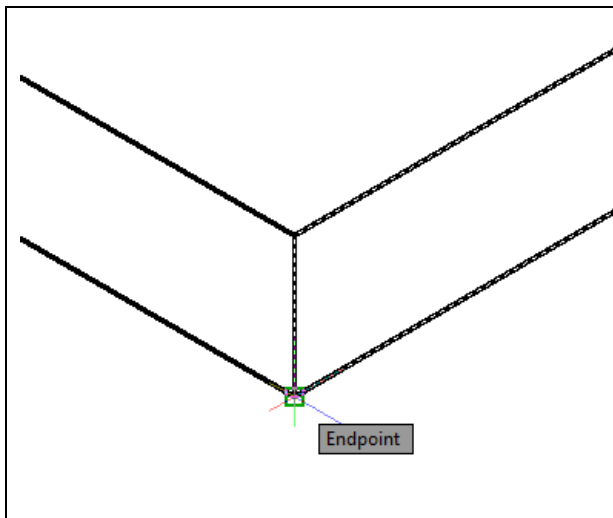


Figure 2.5 – Starting point for next box

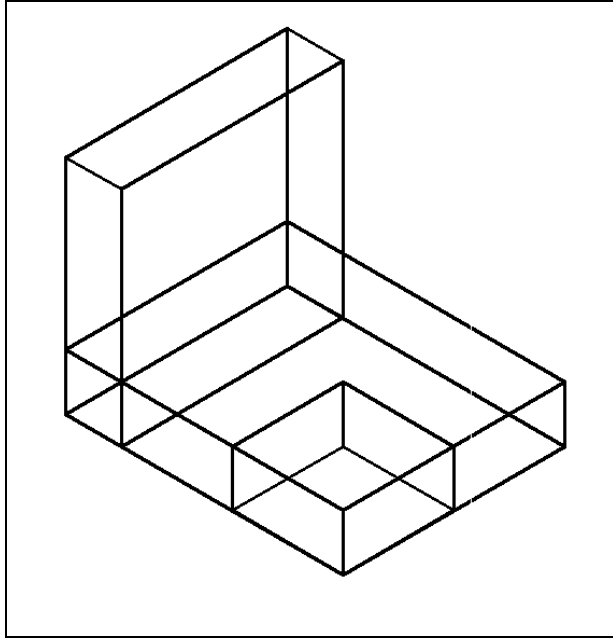
Figure 2.6 – Drawing the next box

You need to draw a third box. When you subtract this last shape from the larger box, it will create the pronged-fork appearance shown in the sketch.



Again, select the Box tool from the 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. Type **L** after specifying the starting point and the system will ask you three questions. The first prompt is “Specify Length,” so will type **1** and ENTER for the X coordinate. The second prompt is “Specify Width,” so type **1** and ENTER for the y coordinate. The third prompt is “Specify Height,” so type **0.5** and ENTER for the Z coordinate. The second box will appear as shown in Figure 2.8.

Figure 2.7 – Starting point for final box

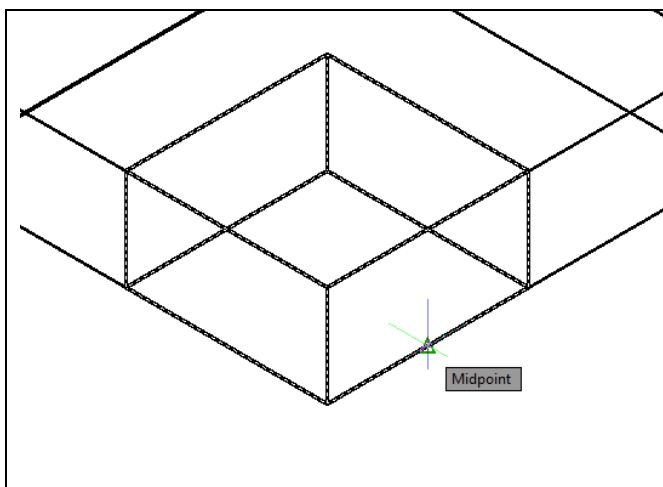


3D modeling requires more attention to detail in regards to the model space. Many actions in 2D required little or no attention to the model space, as the keyboard and command line commands resulted in the same exact actions every time you executed them. In 3D model space, however, the position of your 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

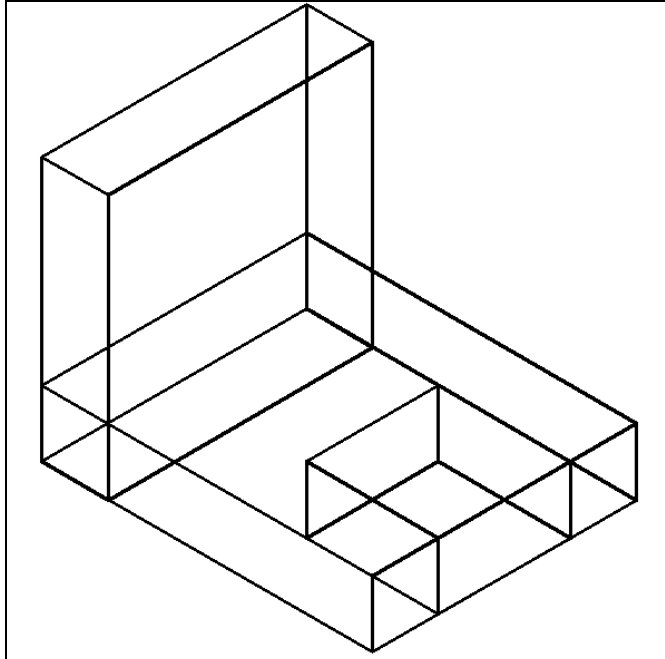
Moving a 3D Solid

Now that you have all three solid boxes in the drawing file, you need to adjust the position of the smallest box using the Move command before removing the shape. The Move command is actually a three-dimensional function in AutoCAD. You can determine whether a command is three dimensional in nature by attempting to use the function anywhere in the Z axis; 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, select the Move tool on the Modify panel under the Home tab. You will be prompted 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.” Hit ENTER to go the second part of the command.

Figure 2.9 – Base point to move the box

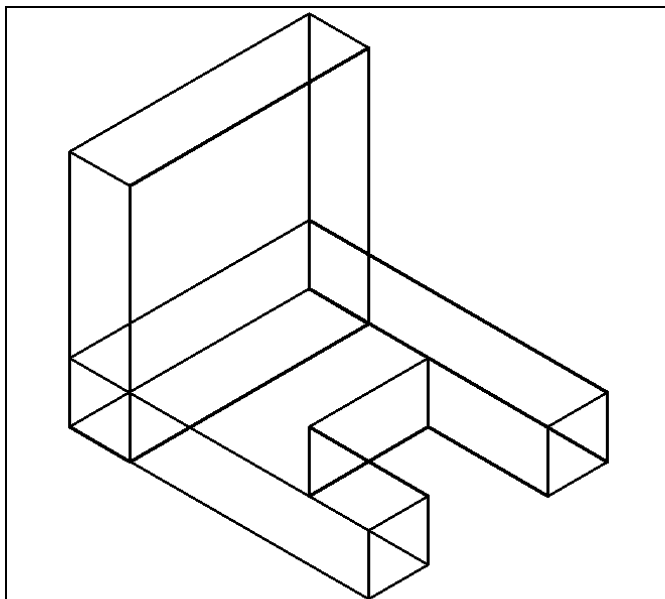


Next, you will be prompted “specify base point or displacement” on the graphical display. Pick your base point as the Midpoint Osnap shown in Figure 2.9. The “second point of displacement” is the midpoint on the edge of the 2-inch box. Pick the second point using the Midpoint Osnap on the edge of the larger box. Notice that the 1 x 1 x 0.5 box is now centered on the 2-inch box (Figure 2.10). 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 also functions in 3D the same way it does in 2D.

Figure 2.10 – Centering the third box

Subtracting a Solid from another Solid

AutoCAD features a series of Boolean commands such as Union and Subtract that are extremely valuable for 3D modeling. First you will learn will the Subtract command, which will allow you to remove one or more solids from another master solid.



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. Subtracting solids from a master solid is a very common technique used in 3D modeling.

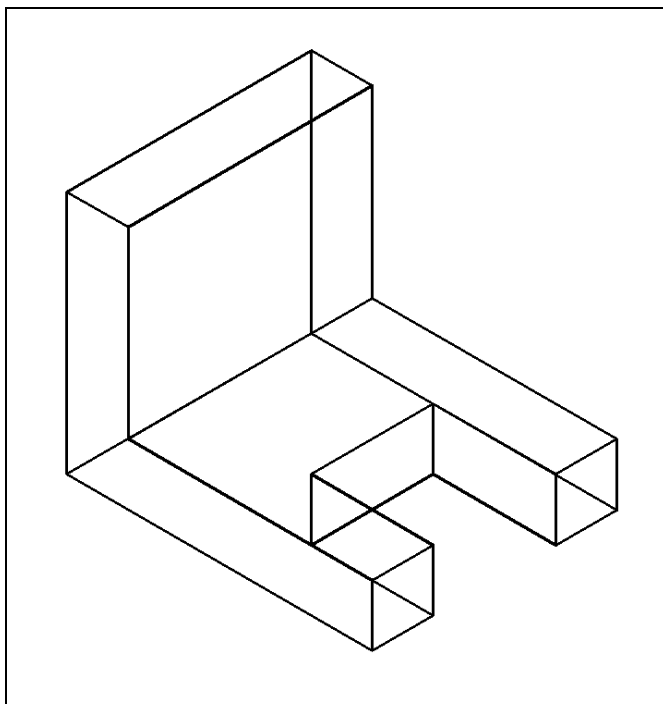
Figure 2.11 – Subtracting the third box

Command: **_subtract** Select solids, surfaces, and regions to subtract from ..
Select objects: **1 found**
Select objects:
Select solids, surfaces, and regions to subtract ..
Select objects: **1 found**

Figure 2.12 – Command line prompts for subtracting the third box

Union Two Solids

Now you will learn a second Boolean command: Union. With the Union command, you can join one or more shapes into one complete solid.



To create a single solid from the two solid boxes, select the Union tool on the Solid Editing panel. The command line will prompt you to “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.” Hit ENTER to join both solids into one as shown in Figure 2.13. You can see in the graphical display that the tessellation lines outlining the intersecting edges of both boxes in the corner of the “L” shape are now missing. If you select the solid with one pick of the mouse the entire solid will become highlighted. In this case the two solids were intersecting each other. You will find out in future projects that the Union command also works 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. You would not be able to see these edges on the opposite sides of an actual physical model unless its material was transparent. You can use the 3D Hidden Visual Style to remove the hidden lines from your graphical display.



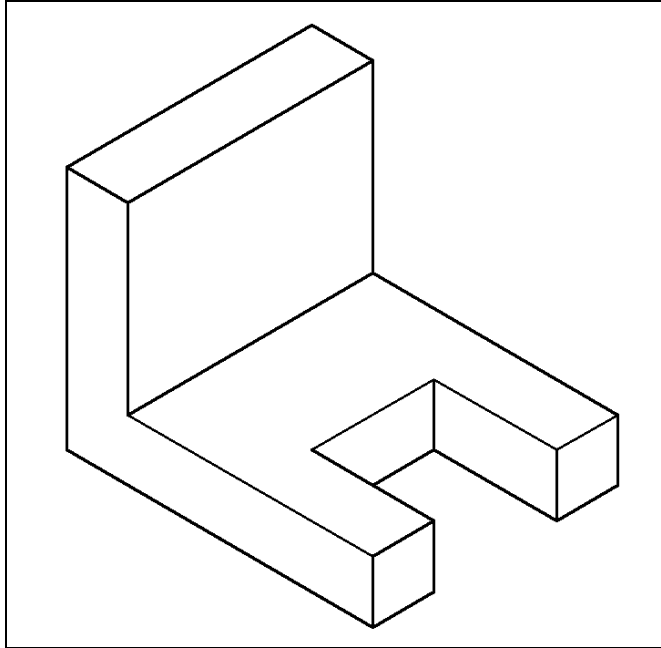


Figure 2.14 – Hiding the tessellation lines

Select the Hide tool from the Visual Styles panel from the Visual Styles drop-down menu under the View tab. This is a very simple command; after picking the tool, the tessellation lines representing the back of the part are removed from view. You can also experiment with the various visual styles available in the Visual Styles drop menu on the Visual Styles panel. You should still be in the 2D Wireframe style, but feel free to change the style to fit your preferences. You can also use the Viewcube in the top right-hand corner of the drawing area to rotate the solid or view it from different angles. Toggle the Viewcube on using the navigation bar. You can right-click on the Viewcube and Select the Perspective option to give the part a more life-like and less boxy appearance. Return to the 2D Wireframe visual style to finish solid problem one.

Moving the Solid to the Origin Point

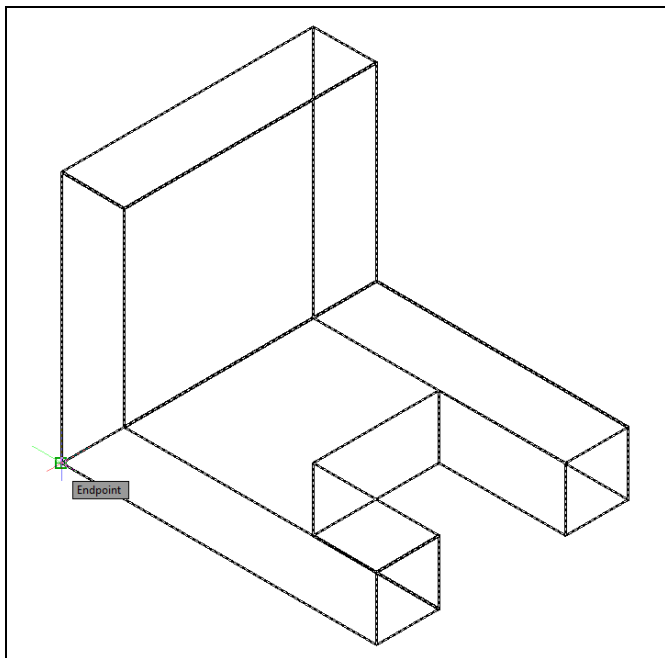


Figure 2.15 – Moving the finished solid

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 bottom left side of the Problem One, 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 One will move to the origin point as shown in Figure 2.16. Moving a part at the origin is important for when you wish to insert the solid into an External Referenced Assembly drawing.

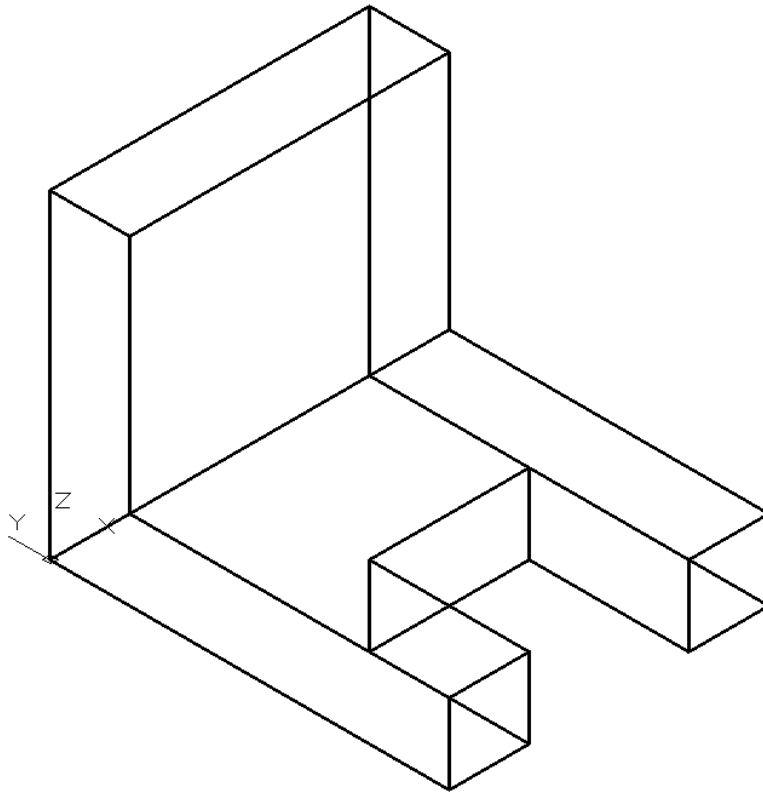


Figure 2.15 – The finished solid

Saving the Solid Problem

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



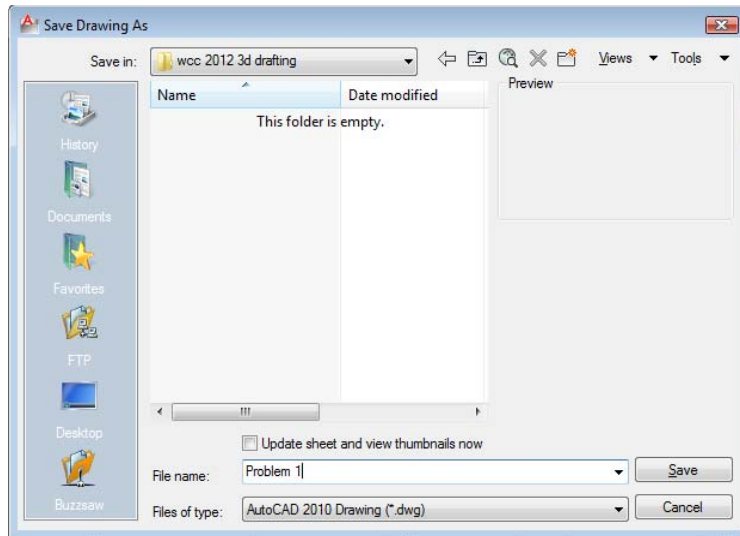


Figure 2.16 – Saving problem one

*** 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 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.**

