Chapter 11

Solid Problem Ten

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

1. **Sketch of Solid Problem Ten**
2. **Starting a 3D Part Drawing**
3. **Modifying How the UCS Icon is Displayed**
4. **Creating a Perimeter Using the Polyline Command**
5. **Adding a Bevel Edge Using the Chamfer Command**
6. **Extruding a Solid from a Closed Polyline**
7. **Rotating a 3D Solid Using the Rotate3D and Rotate Commands**
8. **Drawing Two Solid Cylinders**
9. **Drawing a Solid Box**
10. **Moving Solids into Position on the Master Solid**
11. **Subtracting 3D Solids from the Master Solid**
12. **Hiding the Solid**
13. **Moving the Solid to the Origin Point**
14. **Saving the Solid Problem**

Sketch of Solid Problem Ten

You will start the tenth solids problem with a rough sketch showing the dimensions of each detail. This part is symmetrical across the X-axis. In this exercise, a strategy you will continue to use is to draw and extrude a closed Polyline that contains the most complex shape of the component and then use simple geometric solids to subtract detail. In problem ten, the master solid contains a beveled or chamfered edge in the main shape, so you get a chance to use the Chamfer tool again. After extruding the Polyline perimeter, you will place two cylinders and a box onto that main solid and subtract them.

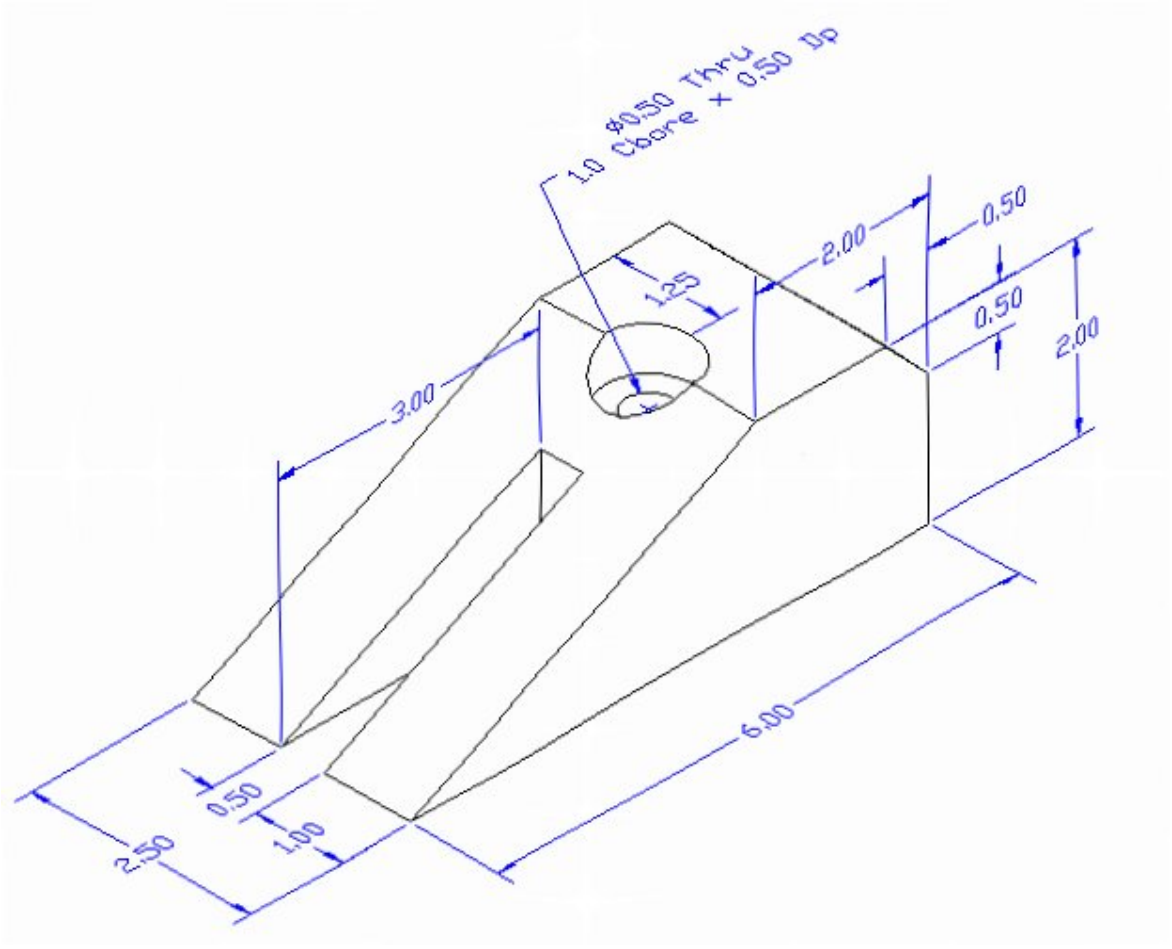


Figure 11.1 – Problem Ten Sketch

Remember, when you receive a sketch from a professional, you need to practice to quickly identify the different shapes and decide how to proceed with the solid exercise. You can see rather quickly that problem has different contours, so separate each extrusion or solid and subtract them from the master. This problem will help you address solid parts containing different complexities and sizes. All ten drills in the Fundamentals of 3D Drawing textbook will give you those repetitive maneuvers which will make you successful in 3D modeling.

Starting a 3D Part Drawing

In Problem Ten, you will begin the 3D drawing by selecting the New tool on the Standard Toolbar. Select the “Start from Scratch” (first button) at the top of the Start window. Choose to use English units as your Default Settings and hit OK (Figure 11.2). 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.

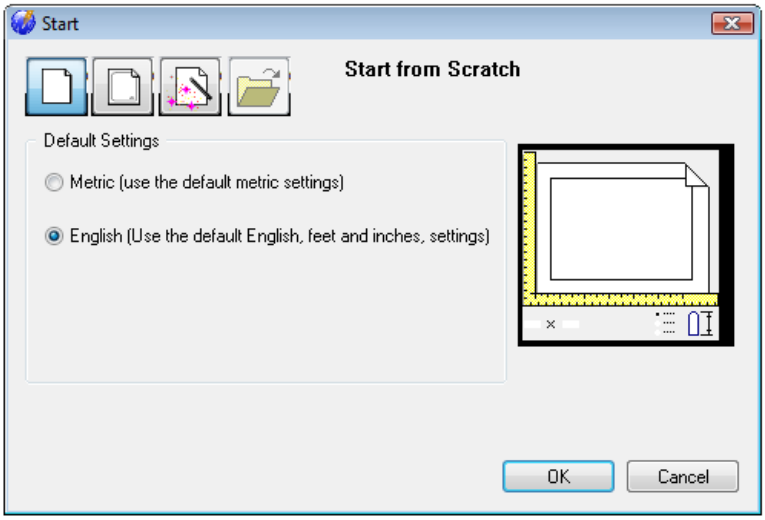
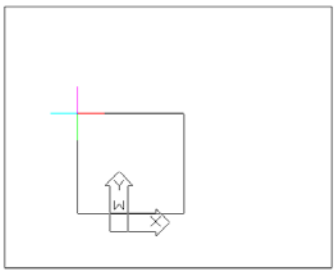


Figure 11.2 – Starting the Drawing

Modifying How the UCS Icon is Displayed



When working in progeCAD, you may notice that the UCS icon has left its position in the corner of the display. This is because the default setting places the UCS at the origin of model space if the origin is in view. Many computer aided design operators do not appreciate having addition UCS lines in with their drawing or solid, so you can change the system to show the UCS in the lower left hand corner of the display at all times.

Figure 11.4 - The UCS in the way

Select Tools from the Menu Bar and then select Drawing Settings to pull up the Drawing Settings window. Under the Display tab, choose to Change settings for Display. In the UCS Icon section select the On radial button and then hit OK. Now the UCS icon will always appear in the corner of the display.

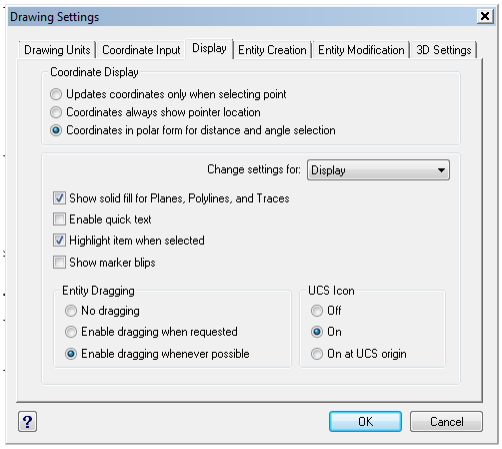
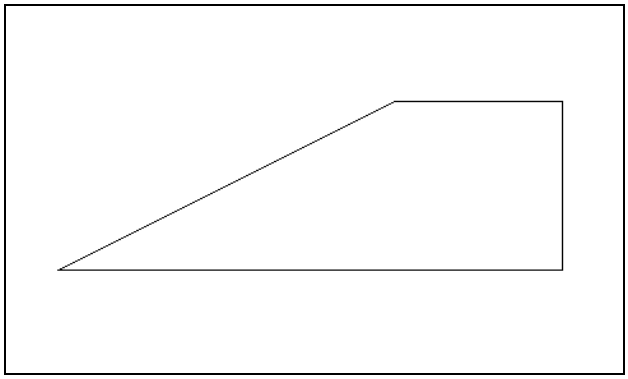


Figure 11.4 - Drawing Settings window, UCS Icon

Creating a Perimeter Using the Polyline Command

Drawing a closed Polyline is not new to you. This particular shape will end with the sharp slope contained in the design. Pick the Polyline tool on the Draw toolbar to begin the process.



With the Ortho mode "on", draw a **6.0** unit line to the right, a **2.0** line up, a **2.0** line to the left, and then type "**C**" to close the Polyline. progeCAD computes the last line length and angle when you use the close option. A closed entity will appear in the graphical display as shown in Figure 11.5. Zoom Extents to have the box fill the monitor, so you can proceed to the next step in the drawing process.

Figure 11.5 – Creating a closed polyline

Add a Bevel Edge Using the Chamfer Command

To add a beveled edge to a corner, you can use the Chamfer command to create the feature rather than drawing a line at an angle. In a previous problem, you placed the chamfer on a 3D solid edge, but on this exercise you will change the Polyline entity by placing the beveled edge in the perimeter.

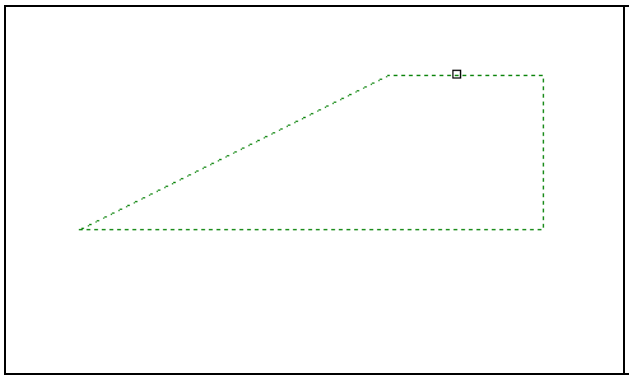


Figure 11.6 – Selecting the first Edge

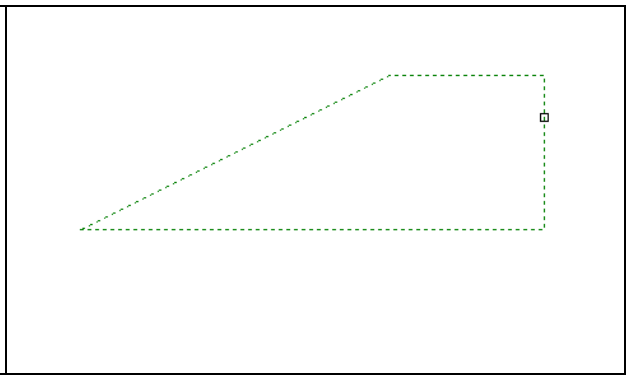


Figure 11.7 – Selecting the second Edge

Pick the Chamfer tool on the Modify toolbar and select the polyline. Type "**D**" for distance, and enter **0.5** as the chamfer distance on each entity. Then select the horizontal line as shown in Figure 11.6 and the vertical line as shown in Figure 11.7. Hit ENTER and the 0.5 by 45-degree chamfer will appear on the edge of the master solid as shown in Figure 11.8.

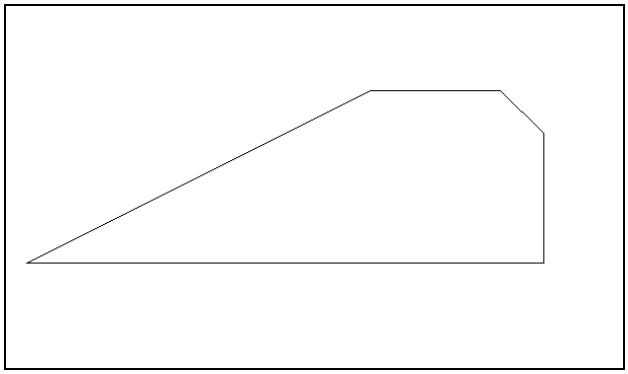


Figure 11.8 – The Chamfered Edge

Remember that you can use the Chamfer command in 2D or 3D mode. When you apply the tool in the 2D mode, you must select two lines to describe the bevel edge. When you are utilizing the tool in the 3D manner, you only select the edge, which is represented by a single tessellation line. Either method will result in the same result.

Extruding a Solid from Closed Polylines and Circles

On the View toolbar, select the Southwest Isometric View tool to allow you to see the part on the XY plane with the Z-axis rising vertically as shown in Figure 11.9. You can only extrude closed entities like Polylines and Circles using this very powerful tool on the Solids toolbar.

Select the Extrude tool on the Solids toolbar and the command line will prompt you to "**Select entities:**". Select the polyline and the command line will return with "**Entities in set: 1**". Hit **ENTER** to specify the height or path of the solid. Type "**2.50**" for the height of the extrusion, hitting ENTER again to keep the angle of taper at 0 degrees. The shapes will extend 2.5 units up into the z axis as shown in Figure 11.10.

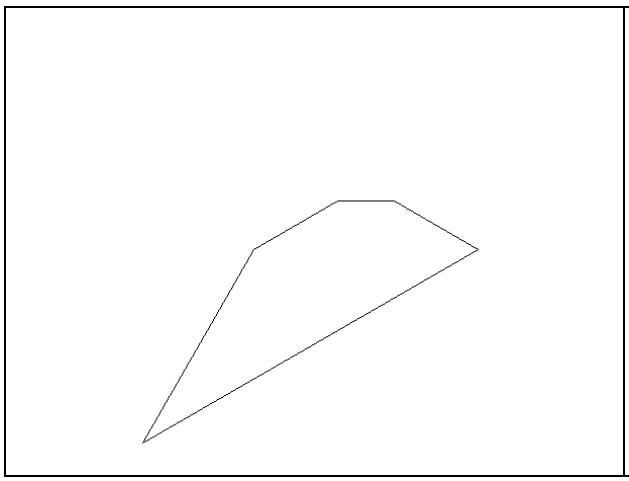


Figure 11.9 – Southwest View

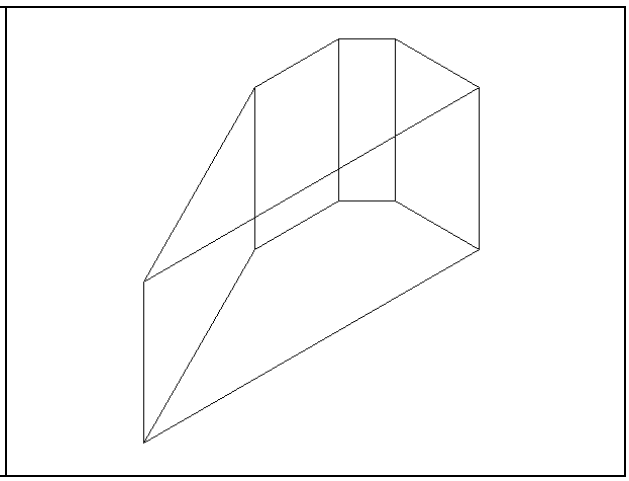


Figure 11.10 – The Extruded Polyline

Rotating a 3D Solid Using the Rotate3D and Rotate Commands

Select Modify on the Menu Bar, then 3D Operations and pick Rotate 3D from the list of commands. At the command, "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 and then with the Ortho mode still "on," pick a second point drawing a line on the x-axis as shown in Figure 11.11. Type **90** to rotate the solid as shown in Figure 11.12.

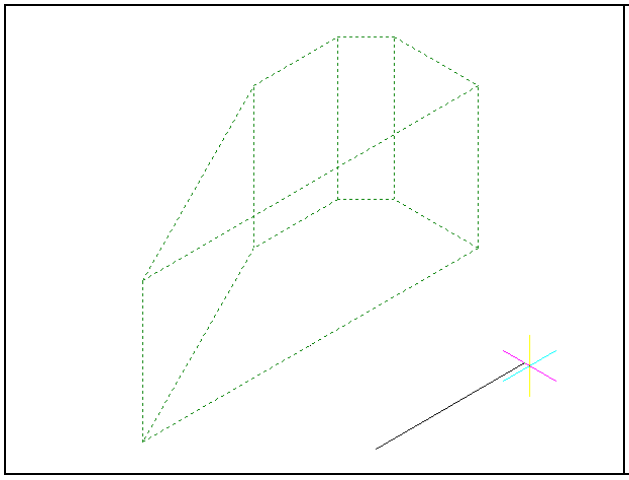


Figure 11.11 – Specifying the x axis

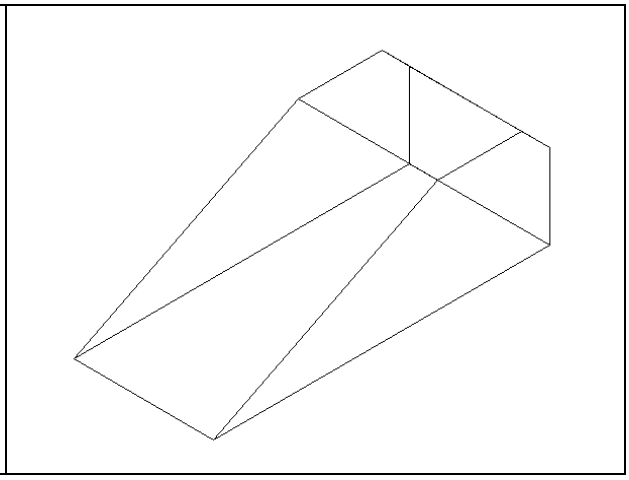


Figure 11.12 – The Rotated Solid

Drawing Two Solid Cylinders

You need to add two holes: one 0.5 diameter hole going completely through the master solid, and a second 1.0 diameter hole called a counterbore. You need to use the Cylinder command to create these features.

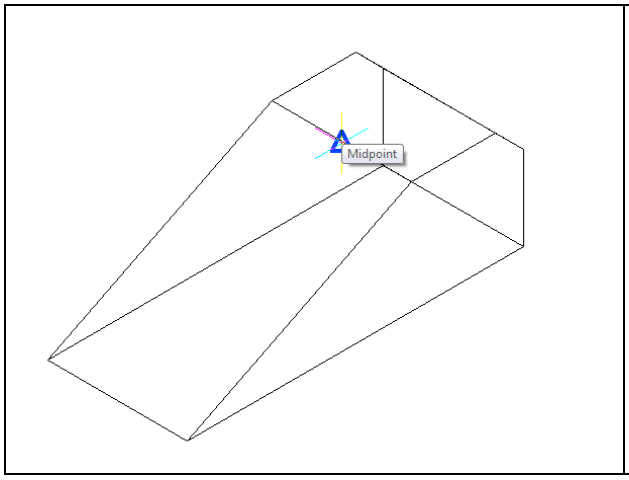


Figure 11.13 – Adding the first Cylinder

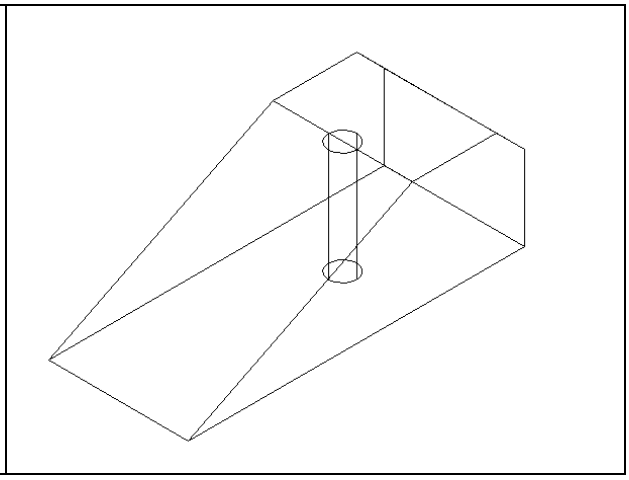


Figure 11.14 – The First Cylinder

Pick the Cylinder tool on the Solids toolbar. Specify the center point for the base of the cylinder as the midpoint of the top front edge of the master solid. Type "**D"** for diameter and hit **ENTER** to input the sketch's measurement of **0.5**. For the next prompt, "Center of second end/<Height of cylinder>:", key in **-2.0** for the elevation and hit ENTER. The cylinder will appear in the graphical display as shown in Figure 11.14.

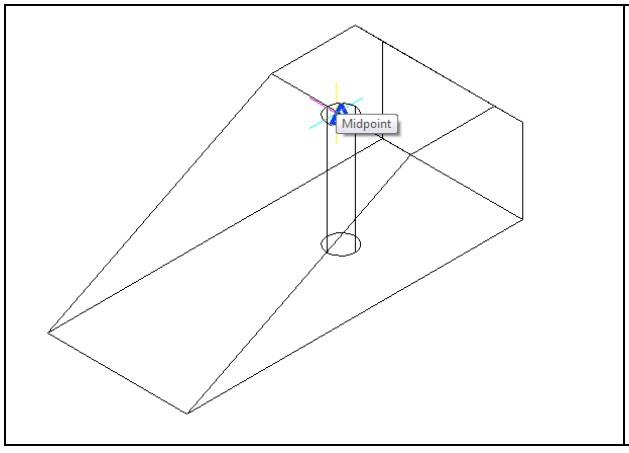


Figure 11.15– Adding a second Cylinder

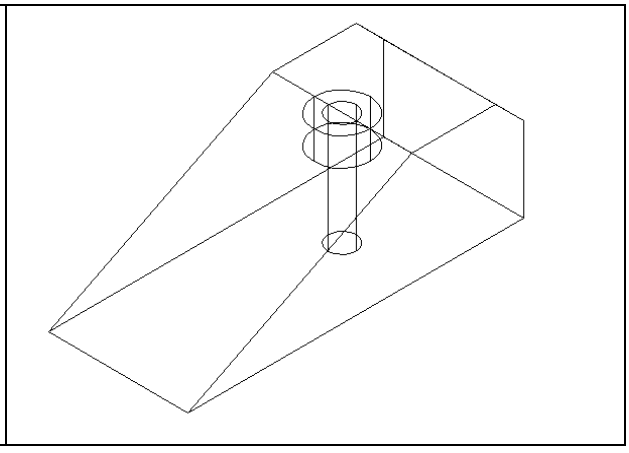
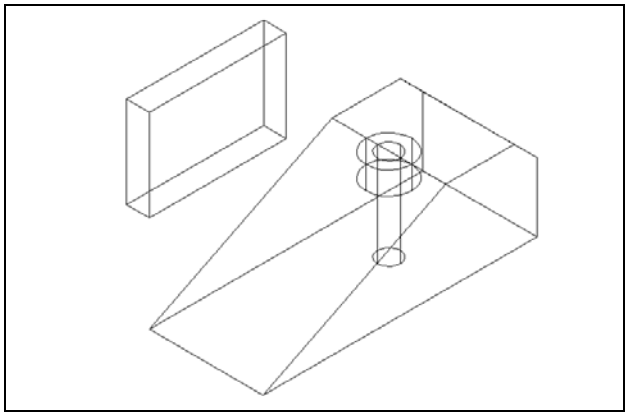


Figure 11.16 – The Second Cylinder

Hit ENTER to repeat the Cylinder command. Specify the center point for the base of the cylinder as the center of the top edge of the cylinder you just drew. Type "**D"** for diameter and hit **ENTER** to input the sketch's measurement of **1.0**. For the next prompt, "Center of second end/<Height of cylinder>:", key in **-0.5** for the elevation and hit ENTER. The cylinder will appear in the graphical display as shown in Figure 11.16.

Drawing Solid Boxes

There is a slot in the front of the wedge that you will make by subtracting a solid box. You can create the rectangular solid anywhere on the graphical display without interfering with the master wedge.



Select the Box tool on the Solids toolbar. At the command line, a prompt asks you to specify the corner of the solid. Specify and point on the display, and then type **L** after specifying the starting point and the system will ask you three questions. Specify the "Length of side of box:" as **3.0** for the X coordinate. Enter **0.5** as the value for the width and type **2.0** as the height of the box. The rectangular solid will appear as shown in Figure 11.17.

Figure 11.17 – Adding a Solid Box

Moving 3D Solids into Position on the Master Solid

The Move command is actually a three-dimensional function in progeCAD Professional. The way you can determine whether a command is three dimensional in nature is to attempt to use the function outside the Z-plane. You can use Move to relocate any entity to another point (X,Y,Z) in Model Space.

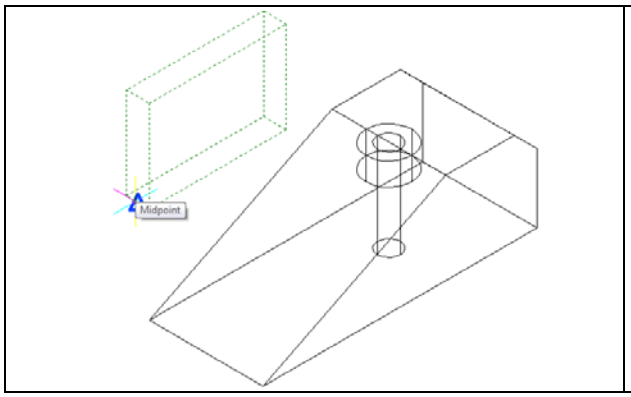


Figure 11.18 – Selecting the Basepoint

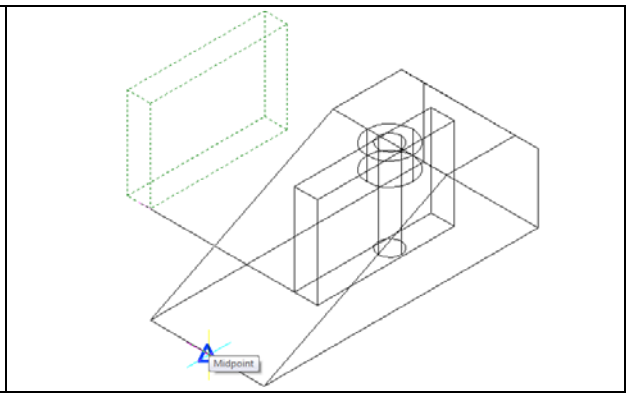


Figure 11.19 – Moving the Box

Once you choose the Move command, you will be prompted to "Select entities to move:". Pick the solid box and the command line will respond with "**Entities in set: 1**", so hit ENTER to go the second part of the command. Next, you need to choose a "Vector/<Base point>:" on the graphical display. Pick your base point as the Midpoint Esnap of the front

edge of the box as shown in Figure 11.18. Hit **ENTER** and then pick the Midpoint of the front edge of the solid wedge as shown in Figure 11.19.

Subtracting 3D Solids from the Master Solids

To remove the cylinders and the box from the master solid, select the Subtract tool on the Solids Editing toolbar.

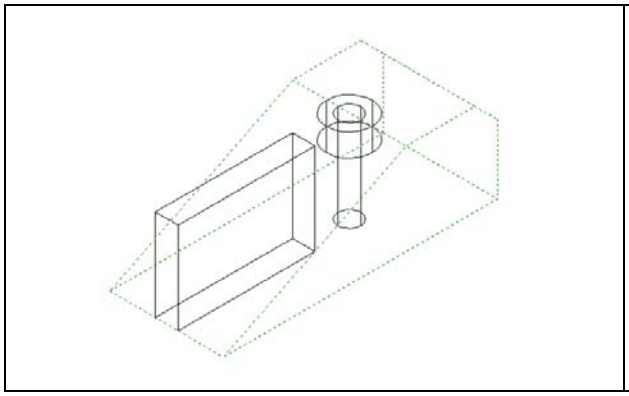


Figure 11.20 – Solid to Subtract From

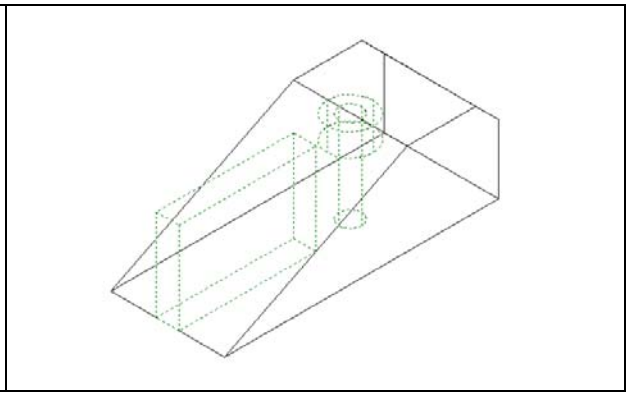


Figure 11.21 – Solids to Subtract

The command line will prompt you to "Select ACIS object to subtract from:". Select the larger master solid 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 cylinders inside of the master solid as well as the box and the command line will reply with "Entities in set: 3" (Figure 11.21). Hit **ENTER** and a region exactly the shape of the cylinders and the box will be removed from the main solid.

Hiding the 3D Solid

In Figure 11.22, 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 its material was transparent. You can use the Hide command to remove the hidden lines from your graphical display.

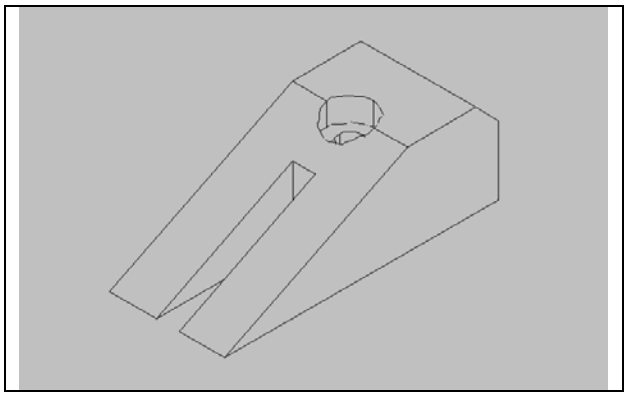


Figure 11.22 – Hiding the Solid

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 as shown in Figure 11.22. If you wish to return the graphical display to the previous setup in which all the tessellation lines can be 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.

Moving the Solid to the Origin Point

To move Problem Ten 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. When prompted to select a "**Vector/<Base point>:**," select the endpoint shown in Figure 11.23 at the lower right-hand side of the Problem Seven, which will become its insertion point if it were inserted into an assembly drawing. For the prompt to select a "Displacement point:," type **0,0,0** and hit ENTER. Problem Ten will move to the new origin point. Try using Zoom Extents if the part completely escapes your viewing area.

*** World Class CAD Challenge 101-10 * - Close this drawing file. Create a New file and draw the solid using the techniques in this chapter. 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.**

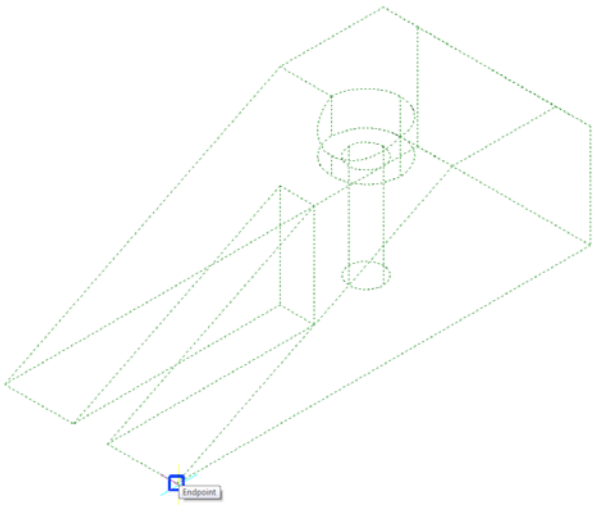


Figure 11.23 – Moving the Finished Solid to the Origin

Saving the Solid Problem

To save Problem Ten, 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 10" and press the Save button to save the drawing (Figure 11.24).

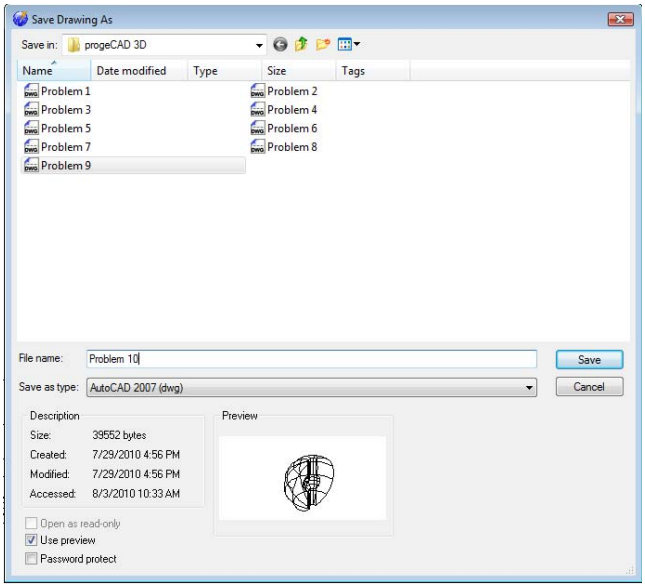


Figure 11.24 – Saving Problem Ten

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