Chapter 2

Solid Part One

This chapter will cover the following to World Class standards:

- **Sketch of Solid Problem One**
- **Drawing a Rectangle**
- **Drawing a Line**
- **Copying the Sketch**
- **Dimension and Move a Circle**
- **Finish 2D Sketch of Solid Part One**
- **Extrude the Sketch**

Sketch of Solid Part One

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 is a rectangular box that measures 7 by 2.957. There are two tabs off the bottom of the rectangle with holes in the center. There are four holes in a rectangular pattern on the rectangle. Whatever procedure you use to complete the exercise, most beginners find that the Rectangle, Line and Circle tool on the Sketch menu provides one of the easiest methods to draw the component.

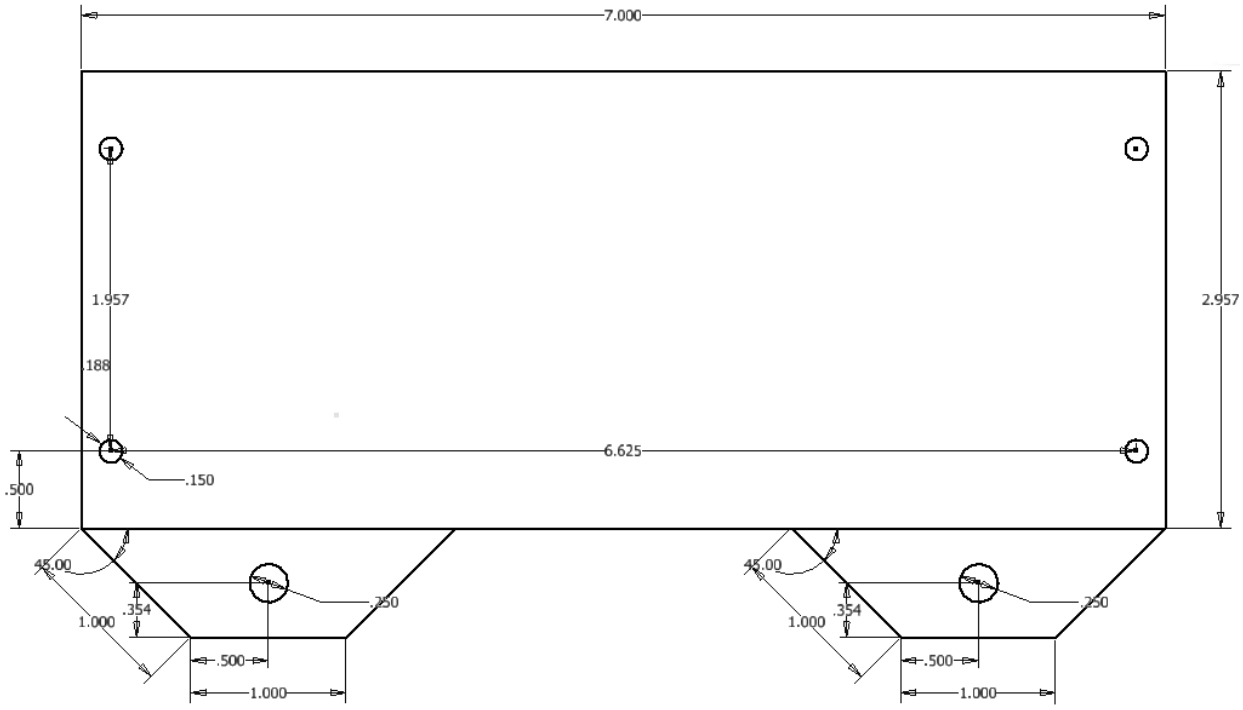


Figure 2.1 – Problem One sketch

As you have already seen from your other 2d drafting training, Inventor 2012 has a different interface than your previous experience in AutoCAD. So, these lessons will expose us techniques that will improve our sketching skills and therefore take us to 3D modeling even faster.

Some features we will enjoy using in Inventor 2012 are changing a dimension on an entity and then the line or circle will move along with the altered measurement. We will enjoy drawing components of the part such as this front view shown in figure 2.1 and when we start to model it in the next chapter, we will quickly have a 3D part. So let us start to use the Inventor 2012 program and create 3D parts and assemblies.

Starting a 3D Part Drawing Sketch

When we open the AutoCAD Inventor application, we will select New from the menu. 

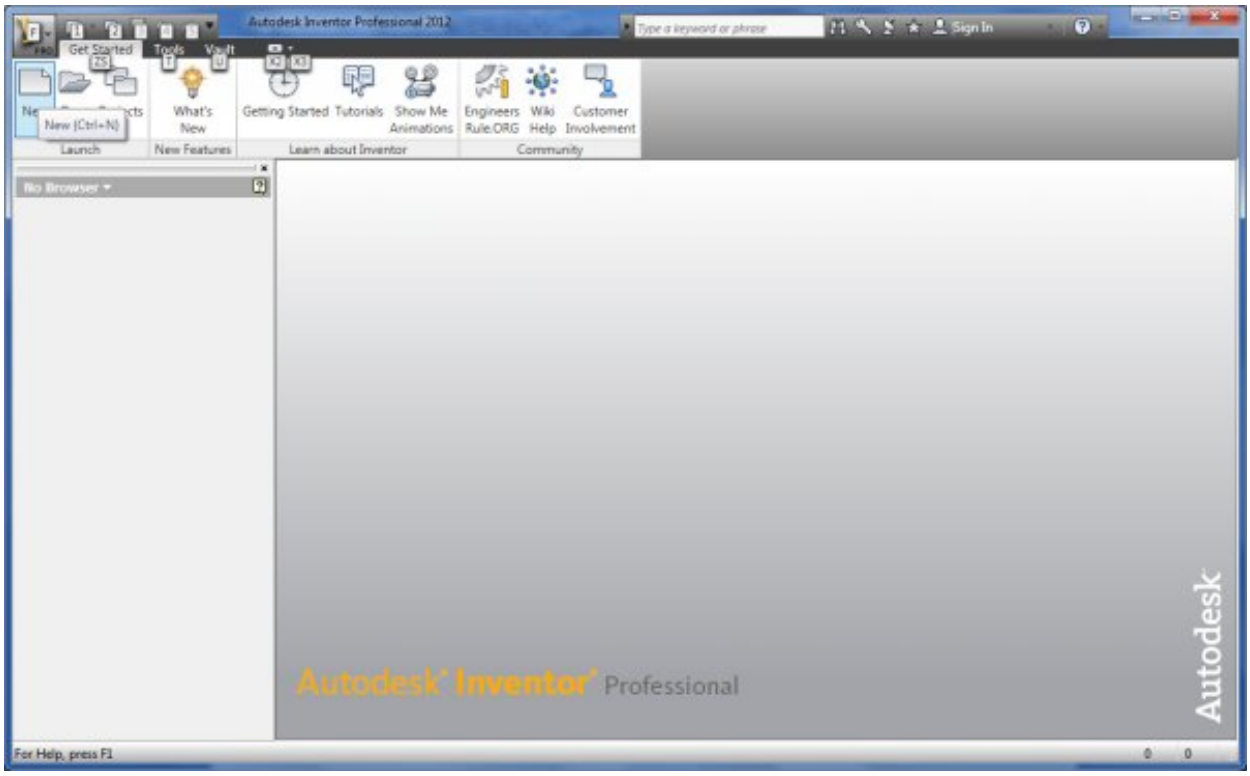


Figure 2.2 – AutoCAD Inventor Professional 2012

A New File window will appear and there are four tabs on this dialogue box. They are Default, English, Metric and Mold design. For this drawing, we will select the English tab and the Standard (in) ipt template. We will press the OK button to continue.

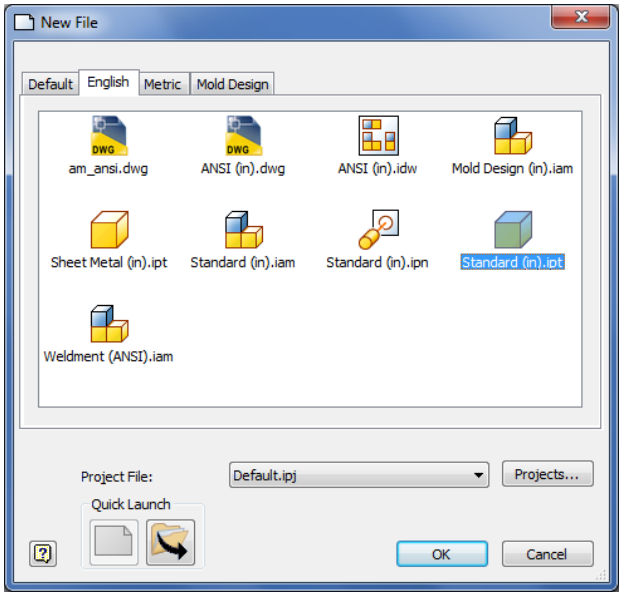


Figure 2.3 – Starting the drawing using the Standard IPT template

To turn off the grid on the new drawing, we will go to the Tools tab on the Ribbon and choose Applications Options.

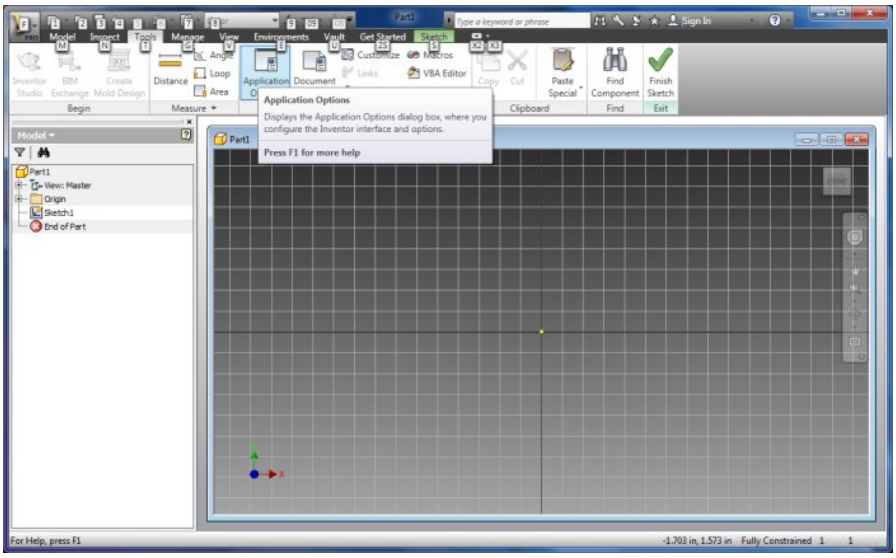


Figure 2.4 – Starting the drawing using the Standard IPT template

In the Applications Options dialogue box, we will turn off the Grid Lines.

For this chapter, we picked the Colors tab on the Applications Options and we select 1 background color and Presentation for the Color Scheme list. Having the grid and color on the drawing sketch background has no effect on the drawing, but is the designer's personal preference.

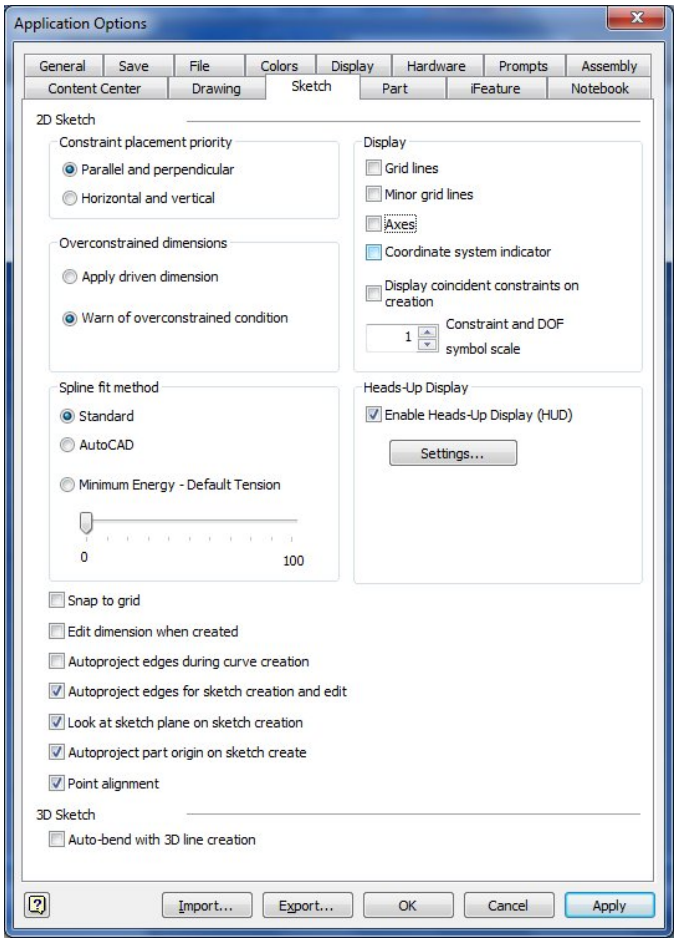
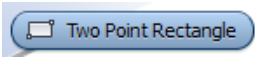


Figure 2.5 – Application Options Window

Drawing a Rectangle

We can sketch our design in Inventor many different ways and achieve the same technique. The first method we will utilize is the Two Point Rectangle.



To draw a rectangle, we right click on the drawing and we can see Create Line, Center Point Circle, Two Point Rectangle and many more choices. We pick Two Point Rectangle where we will single click on the lower left portion of the graphical display and then pull the rectangle to the upper right section of the display.

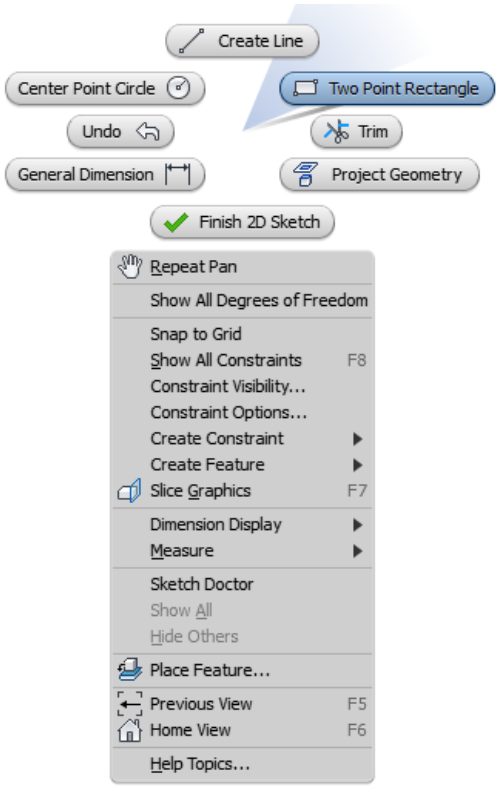


Figure 2.6 – Graphical Display Menu

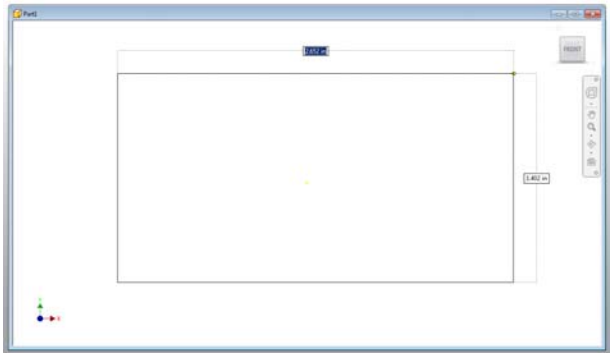


Figure 2.7 – Horizontal Dimension

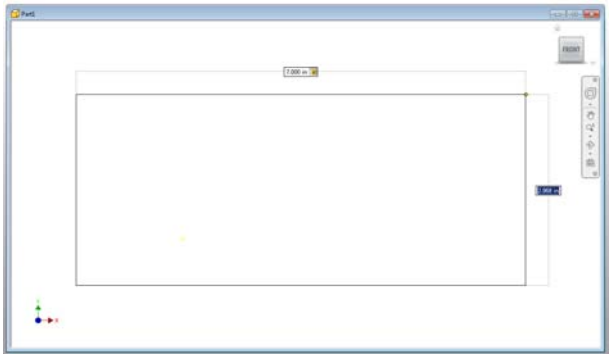


Figure 2.8 – Vertical Dimension

This any sized rectangle on the graphical display has two dimensions. The horizontal measurement is highlighted and we can type 7. We press the tab on the keyboard to switch to the vertical dimension and we input 2.957.

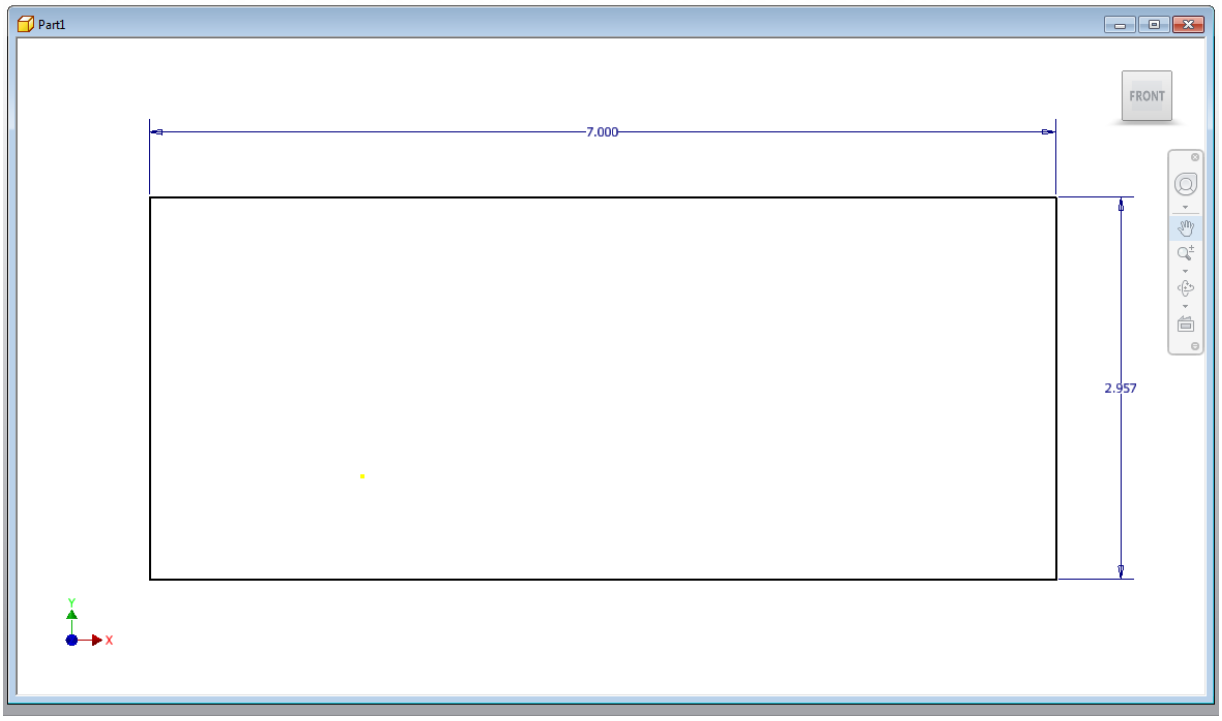


Figure 2.9 – Rectangle with Two Dimensions Showing

Drawing your First Line

The next entity we will learn to draw in Inventor is a Line. We right click on the drawing and we can see Create Line above and to the left of Two Point Rectangle.

We pick Create Line and we select the first point on the lower left corner of the 7 by 2.957 rectangle. We can observe two dimensions on the line as we pull the mouse from the corner of the rectangle. This is a polar dimension which has a length and an angle.

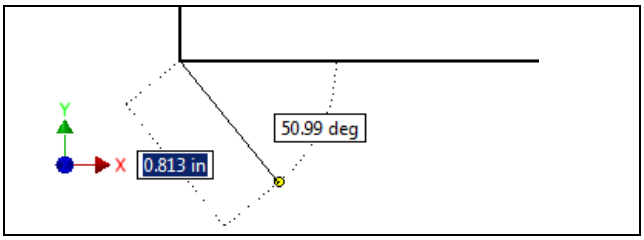


Figure 2.10 – Drawing a Line

We type 1 for the length measurement.

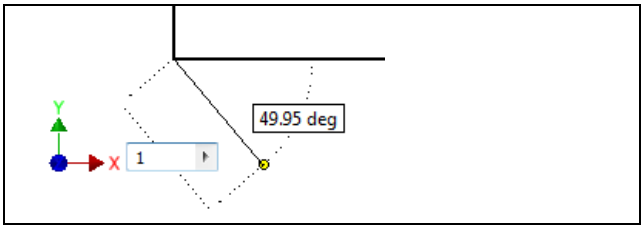


Figure 2.11 – Length Dimension

We tab on the keyboard and we type 45° for the angle measurement.

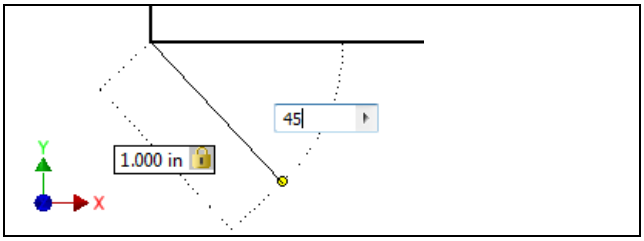


Figure 2.12 – Angle Dimension

We can press the Esc key to terminate the line or we can continue with another segment. We will continue with another segment and drag the mouse horizontally to the right. Notice the new segment will snap to the horizontal if the cursor is close to that axis. We would see a similar result if we were close to the vertical axis.

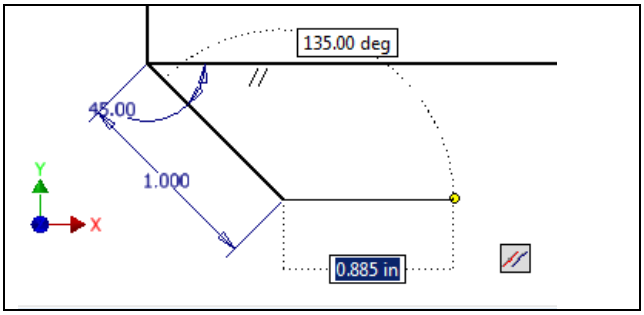


Figure 2.13 – New Line Segment

For the second line segment, we will type 1. For the third and last line fragment, we will type 1 again and point the cursor on the bottom of the rectangle. The angle dimension will indicate 135°. We then draw a line to the point where we started and we can Enter on the keyboard or choose the point with the mouse.

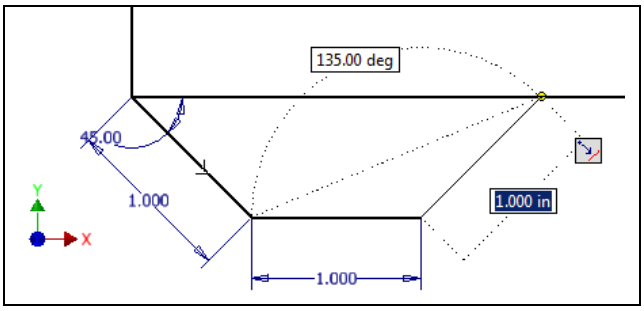
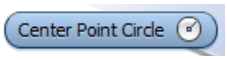


Figure 2.14 – Third Line Segment

Drawing a Circle

Next we will learn to place a circle on the drawing. We right click on the graphical display and we choose Center Point Circle



When we pick the Center Point Circle, we then point and not click on the midpoint of the 1 inch line.

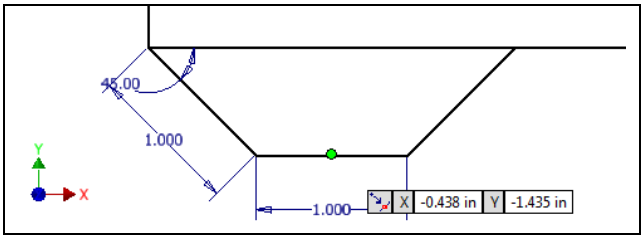


Figure 2.15 – Point to the Midpoint

We then move the mouse up and we will see two dotted lines, one horizontal and one vertical. We then can pick the yellow dot at the intersection and the circle will be in the middle of the tab.

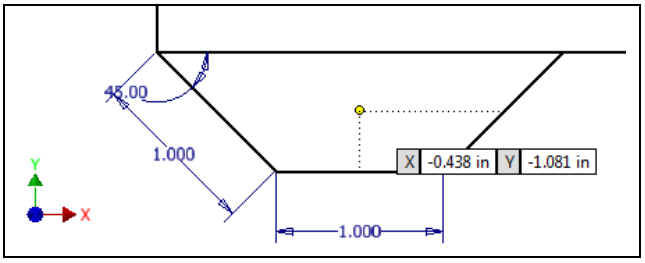


Figure 2.16 – Point to the Vertical Midpoint

Then we can input the diameter of the circle as 0.25.

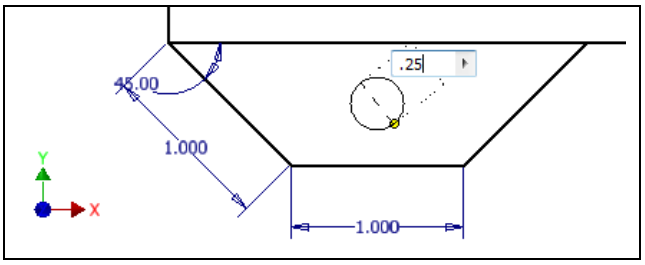


Figure 2.17 – Input the Diameter

The finished the hole will show its dimension

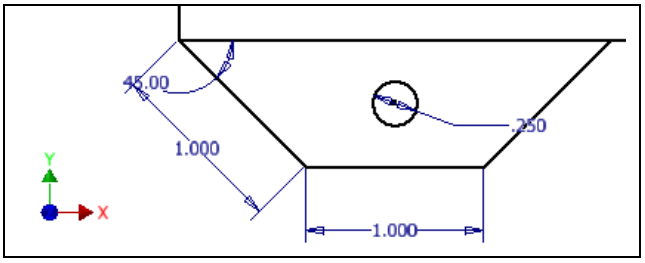


Figure 2.18 – Finished Hole

Adding Dimensions

Sometimes when we add an entity, we will not see all the dimensions, so we will load them.

We will pick the circle and the left corner of the tab and the 0.50 dimension will appear. Click to make it permanent.

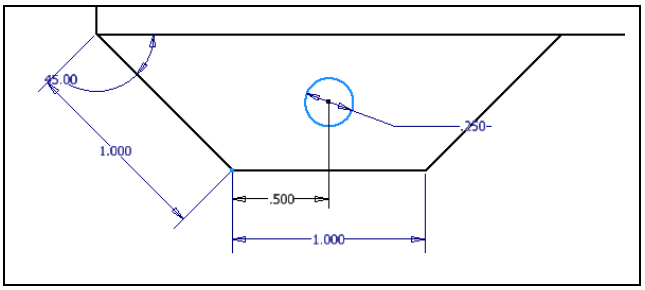


Figure 2.19 – Horizontal Dimension

We will pick the circle and the left corner of the tab and this time we will pull the dimension to the left so the 0.354 dimension will appear. Click to make it permanent.

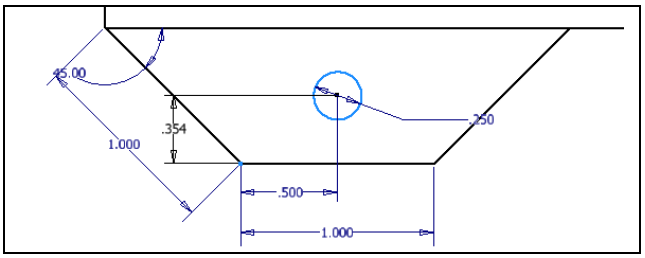


Figure 2.20 – Vertical Dimension

Copying in an Inventor Sketch

After we have drawn an single or series of entities, we can improve our efficiency by copying them to repeat the shape.

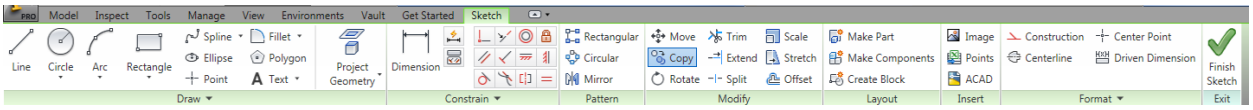


Figure 2.14 – Copy on the Sketch Ribbon

We select a point on the graphical display to the left of the rectangle being sure not to clicking on a line or dimension. We hold down the left mouse button a red rectangle will cover the entities we want to choose.

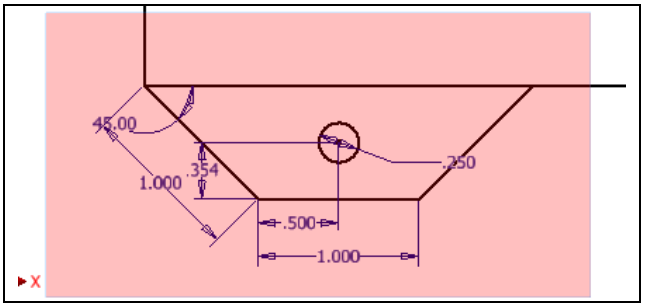


Figure 2.21 – Highlight the Entities

We then pick the Sketch tab on the Ribbon menu and we select Copy. We pick the arrow on the right for the base point.

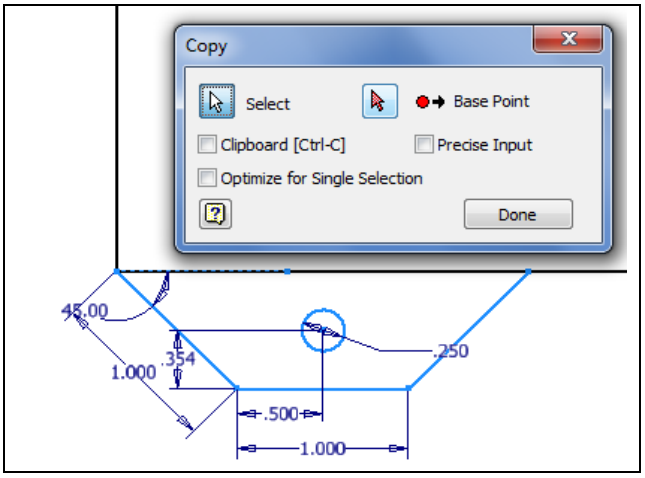


Figure 2.22 – Highlight the Entities

We will pick the junction of the 1 inch line and the bottom of the rectangle on the right side of the tab as shown in figure 2.23.

We then will pick the second point of displacement on the lower right corner of the rectangle as shown in figure 2.24.

All five entities are now copied and shown in figure 2.25.

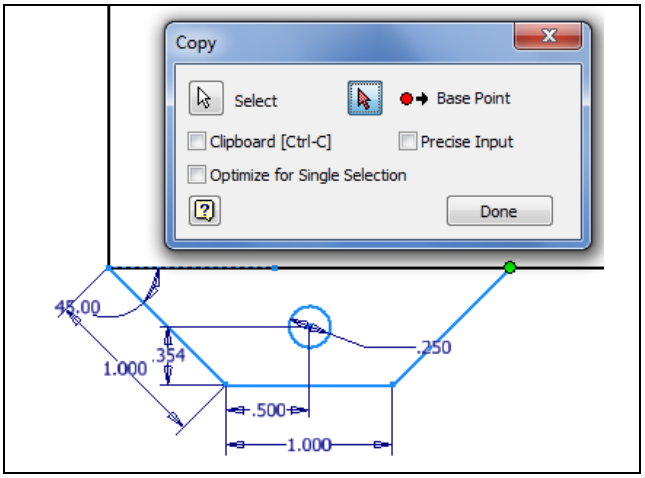


Figure 2.23 – Base Point of Displacement

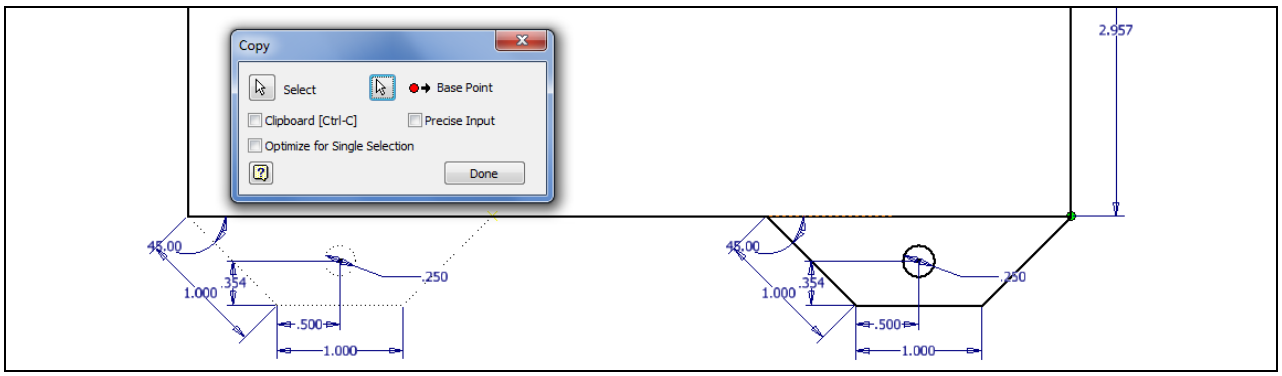


Figure 2.24 – Second Point of Displacement

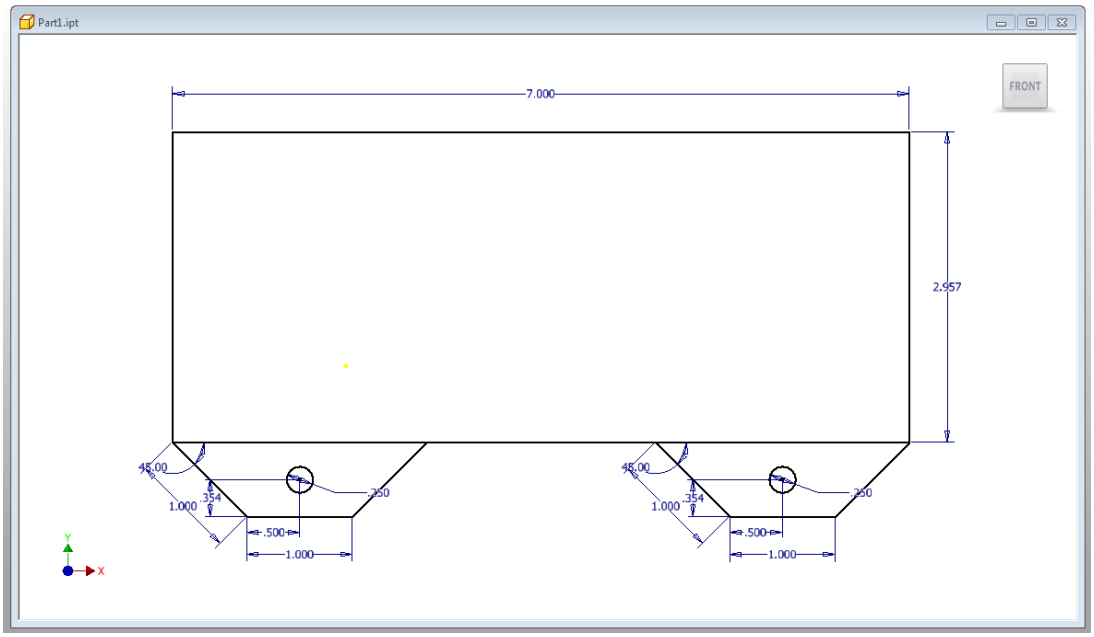
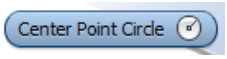


Figure 2.25 – Finished Copying

Drawing another Circle

Next we will repeat drawing a circle on the drawing. We right click on the graphical display and we choose Center Point Circle



When we pick the Center Point Circle, we then point and click anywhere inside the lower left hand corner of the rectangle. We draw a 0.15 diameter circle.

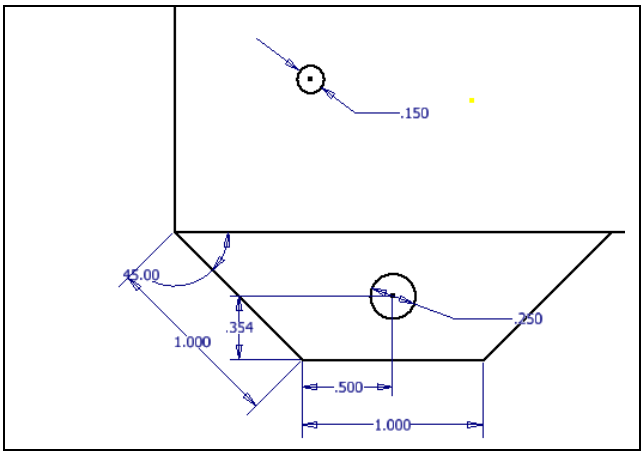


Figure 2.26 – Draw a 0.15 Circle

Dimension and Move the Circle

After we have drawn an single or series of entities, we want to move them to a precise position on the drawing . We can do that by applying the dimensions and changing the measurements and the entity will move its position.

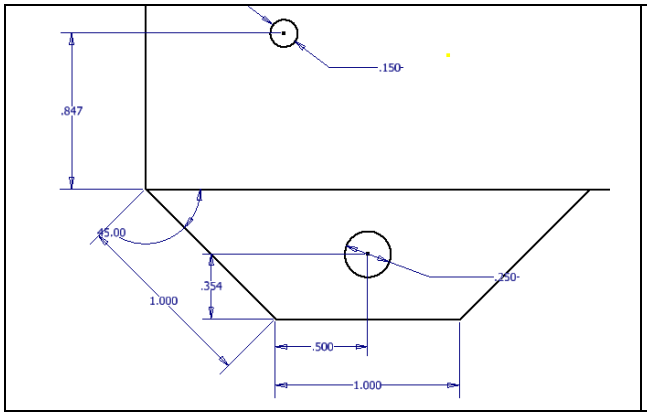


Figure 2.27 – Add the Vertical Dimension

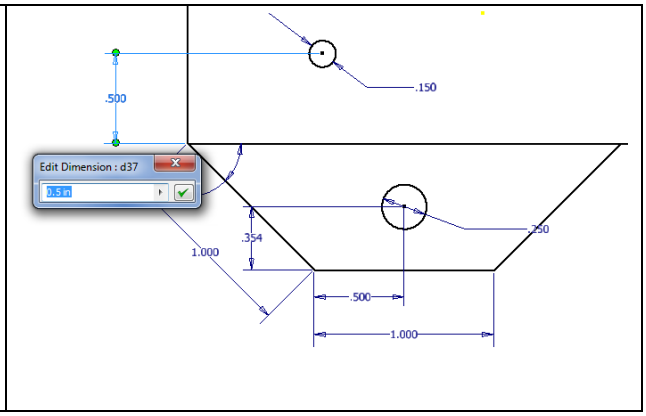
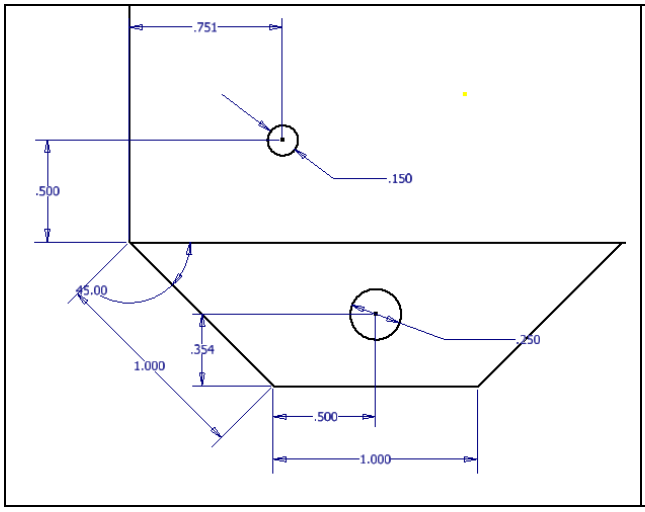


Figure 2.28 – Change the Dimension

We place the vertical dimension from the lower left hand corner of the rectangle and to the center of the 0.15 diameter circle. Our dimension shows 0.847 but your drawing can be entirely different. We double click on the vertical dimension and the Edit Dimension window will appear on the graphical display. We type 0.5 and both the dimension and the circle's position changes.



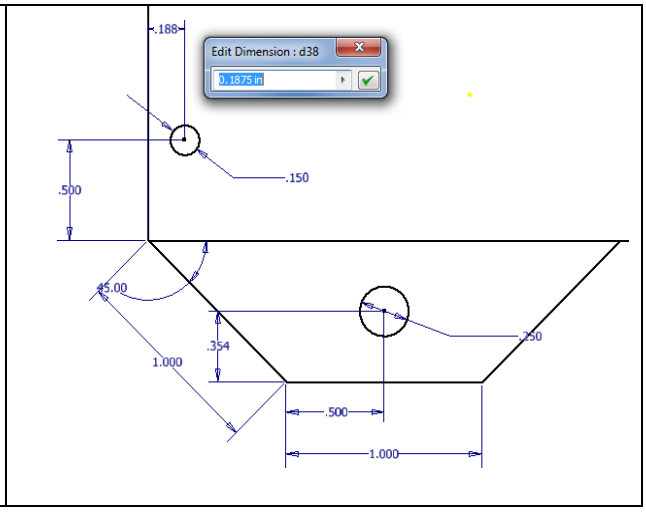
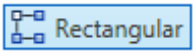


Figure 2.29 – Add the Horizontal Dimension **Figure 2.30 – Change the Dimension**

We place the horizontal dimension from the lower left hand corner of the rectangle and to the center of the 0.15 diameter circle. Our dimension shows 0.751 and again your drawing can be dissimilar. We double click on the horizontal dimension and the Edit Dimension window will appear on the graphical display. We type 0.1875 and both the dimension and the circle's position changes.

Array the Circle

We can array an entity using a rectangular or circular pattern. The first array we will do is the rectangular configuration.



When we select Rectangular on the Sketch Ribbon, we will see the Rectangular Pattern Window appear on the graphical display. We first press the Geometry arrow and select the circle to array. Then we pick the Direction 1 arrow and we choose the vertical line on the right side of the rectangle. A green arrow will appear. Then we pick the Direction 2 arrow and we choose the horizontal line on the bottom side of the rectangle. Another green arrow will appear. When the arrows appear, we will see the column and row numbers go from greyed out to active.

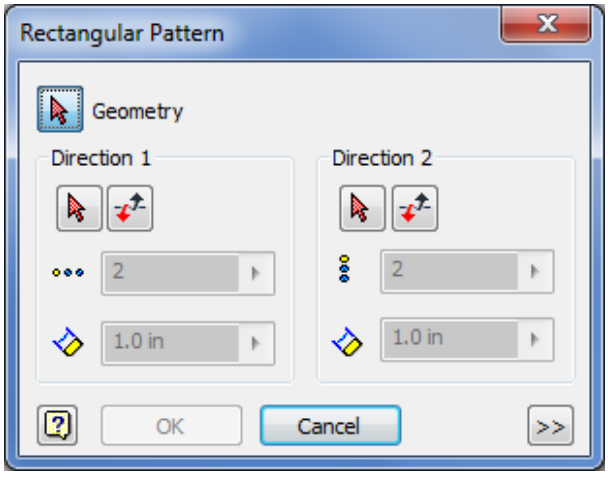


Figure 2.31 – Rectangular Pattern Window

Also, the associative dimension textbox will become active. We will leave the columns and rows textbox with 2 and 2, because we only want 4 circles total. We should know that the first hole

will also count. We will change the distance between rows as 1.957 and the distance between columns as 6.625. We press the OK command button on the Rectangular Pattern window and the finished part will be seen as shown in figure 2.32.

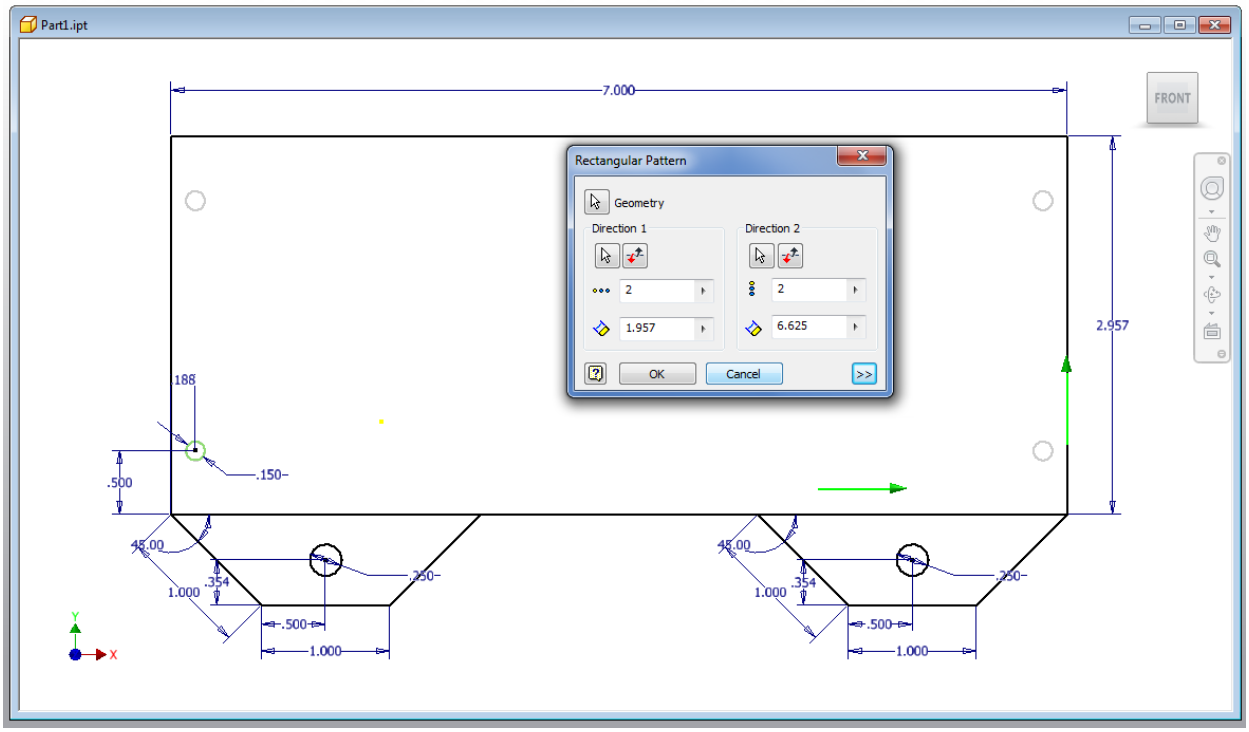


Figure 2.32 – Array the Circle

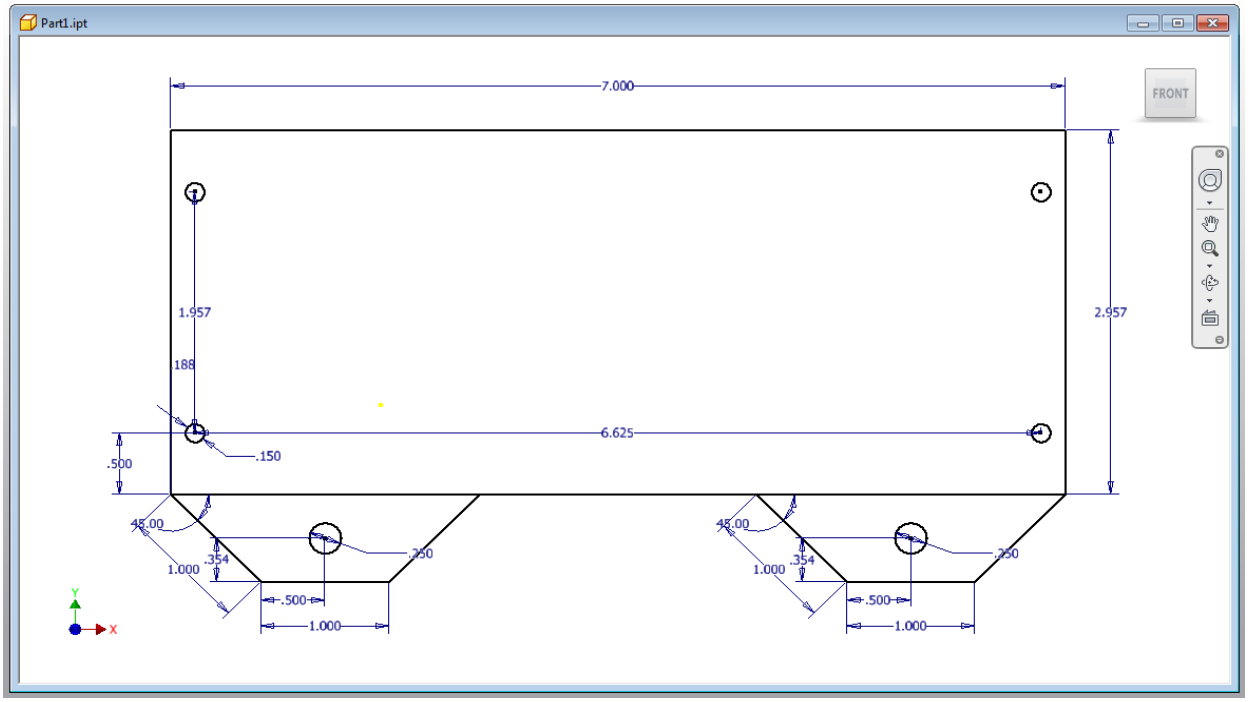


Figure 2.33 – The Finished Part

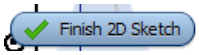
Save the drawing and we will now make a model off the sketch. To improve our sketching abilities, we should practice these drills until we know each function thoroughly.

*** World Class CAD Challenge 61-02 * - Close this drawing file. Create a New file and draw the rectangle, the two tabs and insert the circles. 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.**

Finish 2D Sketch of Solid Part One

Before we extrude the sketch, we need to right click on the graphical display and on the menu, we choose the Finish 2D Sketch button.



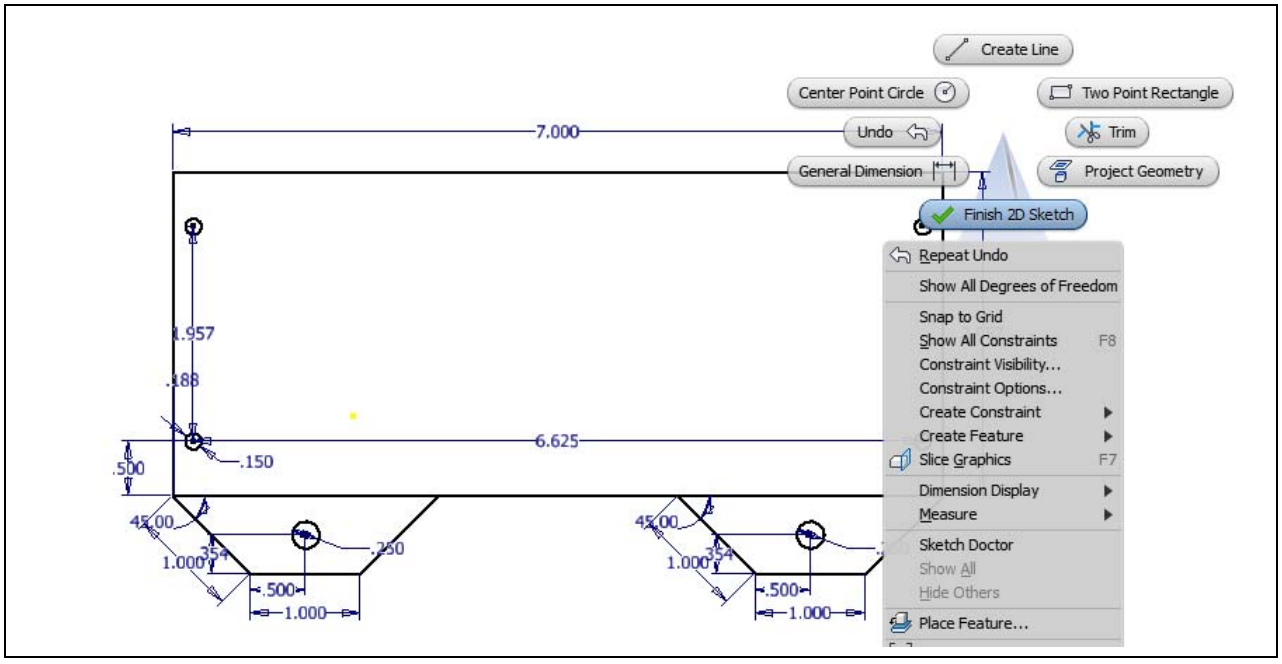
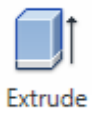


Figure 2.34 – Finished Part One Sketch

Extruding a 3D Sketch

Now that we have a finished sketch, we need to extrude the part. We can go ahead and pick the Extrude button on the Model tab of the Invertor ribbon. The Extrude window will appear on the display.



On the Extrude window, we can output either a solid or surface. The difference between the two

are that the first is like a hard piece of wood and the second choice is similar to a box. We will pick the Solid output on the left. Next, our part will be made from 0.5 finished wood, so we will change the Extents distance from 1.0 to 0.5.

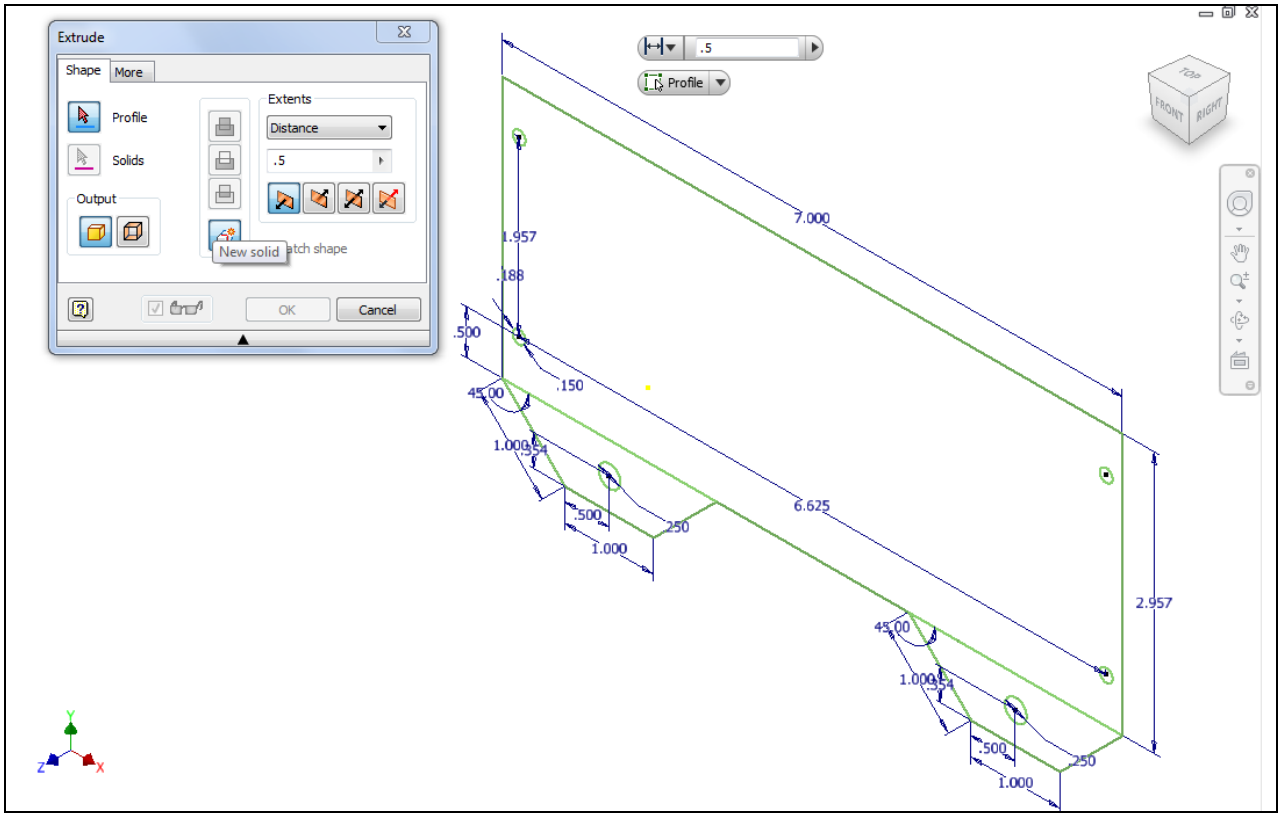


Figure 2.35 – The Extrude Window

Next, we will click on the right tab and it will become highlighted in red. Click on the red area again and the 2D polygon will become a solid with a hole where the circle was located.

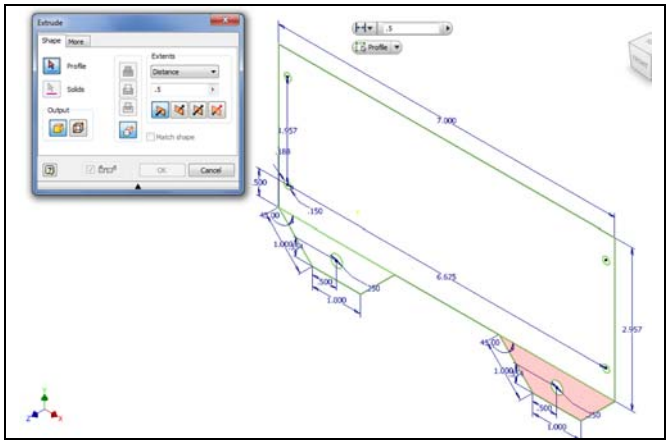


Figure 2.36 – Highlight Right Tab

Then, we will click on the left tab and it will become highlighted in red. Click on the red area again and the 2D polygon will become a solid with a hole where the circle was located.

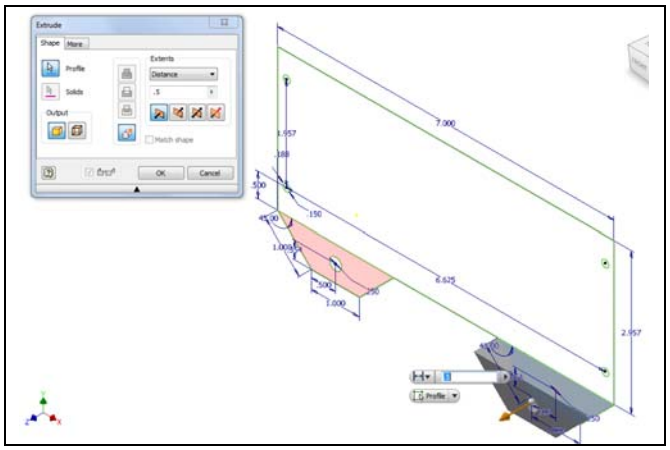


Figure 2.37 – Highlight Left Tab

Now, we will click on the rectangular section and it will become highlighted in red. Click on the red area again and the 2D polygon will become a solid with a holes where the four circles were located.

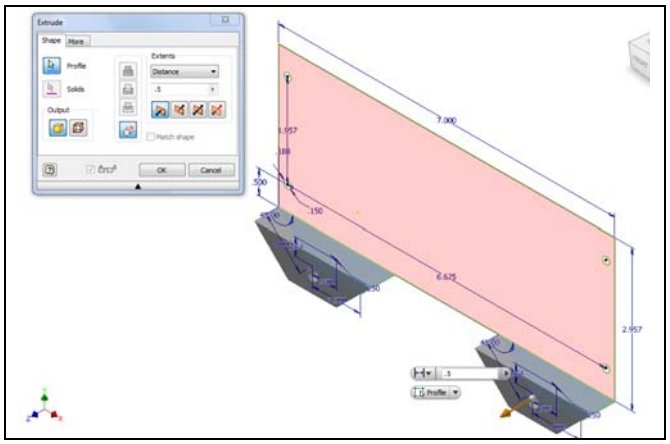


Figure 2.38 – Highlight Plate

The last step is to press the OK button on the Extrude window.

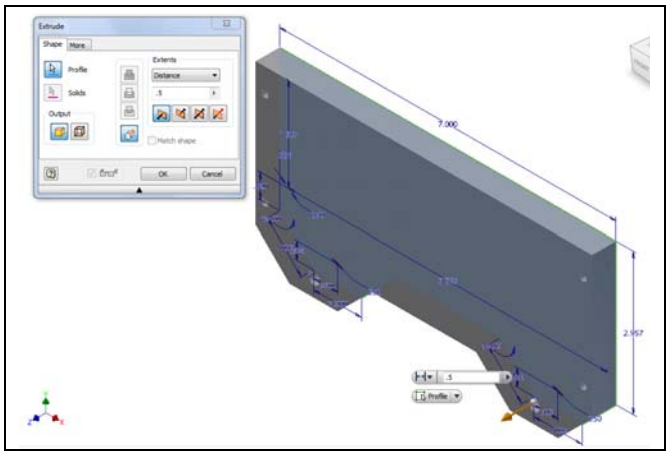


Figure 2.39 – The Solid Part

Save the drawing and we will now have a solid part to be used on a project.

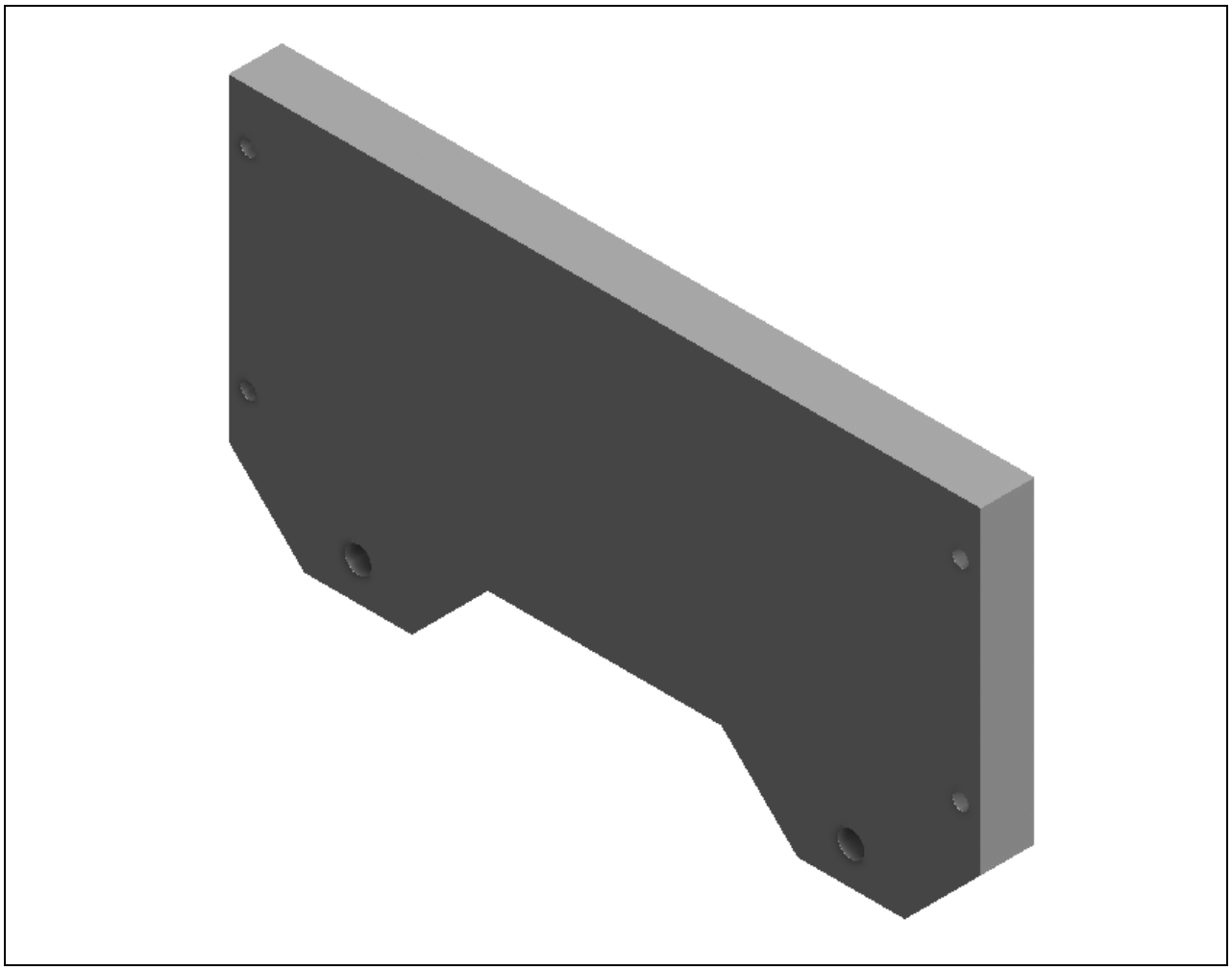


Figure 2.40 – Finished Solid

*** World Class CAD Challenge 61-03 * - Close this drawing file. Create a New file and draw the rectangle, the two tabs and insert the circles. Extrude the part. Complete the task in less than 7 minutes. Continue this drill four times, each time completing the drawing under 7 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.**