Introduction to Solidworks
Solidworks works the way engineers design and think and that is why it has become successful so quickly. Engineers and drafters say that it is easy to learn and gives them a model that they have complete confidence in manufacturing and know that it will work, just by using the tools provided with this one piece of software. Solidworks is powerful.

This is the Solidworks interface. Notice that there are links on the right that bring up Tutorials and websites. These are extremely helpful and when you are finished with our Quickstarts we suggest you try out the Tutorials in particular to learn this software.

To start our project select the sheet of paper shown. Notice there is another sheet of paper on the right side panel and you can use that one as well.
Select PART from the dialog box shown and SELECT OK. We are going to make our FIRST part or drawing in Solidworks.

Solidworks works like engineers think and they start with a sketch of an idea.

We will begin by selecting the SKETCH tab.

We have some sketching options to choose from. We are going to make a plate so choose the RECTANGLE.
Next, we are asked for a PLANE to begin our sketch. Move your mouse to the FRONT PLANE – it should HIGHLIGHT and use your left mouse button to SELECT it.

Let’s rotate our VIEW into a PLAN or STRAIGHT ON view by Selecting the ICON shown and selecting the FRONT view orientation.

We are now ready to draw our rectangle. It doesn’t matter what size we make it as the program is PARAMETRIC and we can change sizes at any time.

Make two picks with the LEFT mouse button as shown.
After we have sketched our idea we can refine it by DEFINING the size of our plate.

We just have ONE dimension tool and it can dimension ANYTHING.

Select SMART dimension and then SELECT the LINE shown and it will HIGHLIGHT.

Next select the DIMENSION LINE LOCATION.

Next you should see the following dialog box. Change the value to 4.00 and press the GREEN CHECK which is the same as OK.
Parametric is DIMENSION driven and we can change the size based on the dimension.

Next, using what we have learned select the line shown and dimension that line to 4.00.

Results should be as shown.

To further refine our sketch we will select the FEATURES tab.

There are things grayed out that we cannot do, but one thing we can is EXTRUDE.

Select Extrude boss/base.
Change only the value shown as right in the EXTRUDE dialogue box to .30 and then select the GREEN CHECK.

We definitely want to add more to this design. Notice the FEATURE TREE we are creating as we go. You can go back to anything you have done in the drawing by using the FEATURE TREE.

SELECT SKETCH
And we will add more interest in this part.

ROTATE the object to the FRONT plane by using this VIEW tool.
Select CIRCLE

MOVE to approximately the CENTER of the PLATE to create the CIRCLE. The first PICK turns the top plane blue to SELECT it.

Now pick a center point and then a second point to create the CIRCLE. It does not matter what size it is because we can change it.

Change the size of the CIRCLE to 1.00 in the CIRCLE dialogue box and then select the GREEN CHECK OK.
We want to make sure the CIRCLE is in the center and a great tool to use is SMART DIMENSION.

Next select SMART DIMENSION.

Select the TOP LINE and then PICK the CENTER

Next select the DIMENSION line LOCATION and change the VALUE to 2.00.

Continue with SMART DIMENSION. Select CENTER and then select the LINE shown.

Select the DIMENSION LINE LOCATION and change the VALUE to 2.00 and our circle will be perfectly in the CENTER using SMART DIMENSION.

Use the GREEN check to close the DIMENSION DIALOGUE Box.
Go to the FEATURES tab and select EXTRUDE.

Also, select the view icon and change to ISOMETRIC view so we can see our extrusion.

Change the VALUE of the extrusion to .30 and use the green check to close the dialog box.

Let’s continue to refine our part by adding FILLETS to our plate.
Fillet rounds the corners by a specified radius. The radius will be .10in. Select the 4 corner as shown.

Using CHAMFER we will simulate a weld between the cylinder and the plate. The CHAMFER can be selected by using the small black down arrow and then selecting CHAMFER.

Make sure the value of the radius and the distance is as shown and SELECTS the BOTTOM of the CYLINDAR. Select the green check to complete the chamfer.

Check out the different DISPLAY views by trying out each of the options, but end with SHADED with EDGES.
Another interesting way to view our object is to RIGHT mouse click in empty space and select ZOOM and then ROTATE VIEW.

This allows you to ORBIT your part by picking a POINT and holding down the pick as you move your mouse.

Suppose it was our job to save money on the manufacturing of this part. We could do that by removing material from the part.

Start by going to VIEW and select BACK.

Then select SHELL from the top menu.
Select the BACK of the part and it will turn BLUE then make sure the value is .10in.

Then check SHOW PREVIEW and it should look like this.

Select the green check to complete the SHELL.

Check it out using the ORBIT tool.

Right mouse click in empty space to get the ORBIT tool.

If this is your first part, then CONGRATULATION on completing it!

Solidworks makes you save your drawings which is a great habit to get into. Go to the SOLIDWORKS icon box and select SAVEAS and save this part where your instructor allows you to save it.

Solidworks will name your file or you can name it something else. Notice that it is saved with a file format of .prt or .sldprt
Now we are going to do a DRAWING of the part we created. Go to the piece of paper and select the 3rd option DRAWING.

Solidworks allows you to have different title blocks, but we will choose a LANDSCAPE A from the standard title blocks that come with the package. Select LANDSCAPE and then OK.

Double click on YOUR part name. If you DON’T SEE it use the BROWSE button.

Make sure the FRONT view is selected in the ORIENTATION.
Move to the right and place the front or plan view approximately here.

Move the mouse to the RIGHT and you should see a SIDE view. Place approximately here.

Place the top view as well as the isometric view on the sheet.

Check the green OK.

Select the isometric view with your mouse. When you see the MOVE tool DRAG it into place.
By pressing the ESC key twice you will unselect anything that may selected. Selected the isometric view. Select the VIEW tool and use SHADED with EDGES for this view. We could also use HIDDEN line.

There are many items we could add to this drawing, but let’s bring in items from our MODEL. Press ESC twice to make sure nothing is selected and then Go to ANNOTATION and then MODEL ITEMS.

Select the GREEN check OK and then select YES from the DIALOGUE BOX.

We just brought in all the dimensions from our model. You can pick and delete redundant ones and you can select a dimension by its TEXT and MOVE into a new position.

You could also SMART dimension anything that is missing.
You can also HATCH by selecting this tool and selecting one of the views.

If you have ever done 2D drawings you can see that using 3D is much easier to use. Solidworks makes it easy to analyze this part for manufacturing called finite element analysis and to make prototypes and actually manufacture and produce the part.

Congratulations on your first PART and DRAWING. Lesson is complete.