Good Morning, good Afternoon, Good evening...whatever time of the day it is Welcome to this tutorial. This is a beginner level tutorial and does not assume any prior knowledge of solidworks or 3d modeling. Feel free to rewind and replay any section that you don’t understand at first. It is important that you understand the steps and tools involved in creating this part before moving on to the other examples.

We’re going to try and create the object that you see on screen. They’re actually a set of wings for a model airplane...Once you’re done with the tutorials...maybe you’ll try creating the whole plane!

The first step in any solidworks project is to open a new file. Click file and then new.. or press Control + N. In the menu choose Part. And click ok.

What appears on screen is a blank document with three intersecting planes. You can press the arrow keys on your keyboard to spin the whole thing around..and roll the mouse wheel forward and back to zoom in or out.

Go ahead and click on the right plane and click the “Normal To” icon on the view menu. What you’re seeing on screen right now is the right plane alone. The two red lines u see on screen are the places where the top and front plane intersect the right plane. Please take some time to familiarize yourself with this view and try switching between the different views on the top menu to see where they go.

Almost all 3d modeling begins with a simple 2 dimensional sketch... and that’s exactly what we’re going to start with. To start drawing of a 2 d figure, return to the normal view of the right plane and Click ‘sketch’ and then choose ‘line’.

You can use the top plane’s edge as a reference. Click on it and move your mouse to make your line as long or short as you want. Click again to end the line and press escape to stop drawing.

We’re going to add a spline curve to this line. Click spline in the sketch tools section and click the starting point of the line we originally drew. When you move your mouse away, you’ll see that a line is created. Click on an arbitrary point to start the curve and then move your mouse to the other end of our old lie. Notice that when you move over the end point a small yellow circle appears near your cursor. That’s solidworks’ way of telling you that you’re on a previously created point. Click on the end point and press escape to stop drawing.

Notice that the arbitrary point we clicked on while drawing the spline is denoted by a small star. We’re going to shape our spline curve using this point. As you move your mouse over this point, it turns red. Click this point and pull it away as far as you want to create a nice little wavy shape like the one on screen.

We’re done with this figure for now. Click exit sketch on the sketch tools menu.
When you spin the screen around by pressing your arrow keys you’ll see what we’ve been doing. During our next step, we’re going to create a reference plane parallel to the plane that contains our sketch. Click the right plane and then click insert – reference geometry – plane.

When you do this, you’ll see a plane appear on screen. Let’s put this plane at 40 mm away from our right plane by entering this value as shown.

Click the nice green tick mark to complete this operation.

We’re going to try and make a copy of our old sketch on this new plane. It’s pretty simple. Choose the sketch from the menu as shown and press ctrl+c..the normal shortcut for copy on windows. Click our new plane and press control v.

There we are!

Now we’re going to the normal view of our new plane. Once you do this, click the copy of our old sketch that we created on this plane. Right click and choose edit sketch. When you’re editing, it’s currently xxxx mm long. Make it around 2/3ths of what it is. Let’s say x_x. Now click the green tick mark.

We’re now going to move this reduced sketch a little to the right. And then exit the sketch like we did before.

There we are!

Now we move to the features menu. Choose lofted boss-base. This tool adds solid material between 2 cross sections. Click the two sketches one after another and you shall see a preview of what this tool is going to do. And click the green tick mark to complete this operation.

We’re almost there. We have one wing...but need the other one. We can do this in a flash by mirroring the part we just created.

First click the feature we just created. Choose the mirror tool in the features menu and click the right plane. This will serve as our mirror.

Click the green tick mark and tada! We have our plane’s wings.

Solidworks has a lot of fun options that you can play around with once you’re done constructing your parts. Things like materials and lighting. Go ahead and explore those options before our intermediate tutorial. Where we’re going to try and make a small plastic bottle.