Good Morning, good Afternoon, Good evening...whatever time of the day it is, Welcome to this tutorial. This is an intermediate level tutorial assumes that you have completed and fully understood the basics of solidworks. I strongly urge you to check out the beginner tutorials before starting with this one. Feel free to rewind and replay any section that you don’t understand at first.

To give you an idea of what we’re doing. Consider a cylinder. A cylinder is just a rectangle that’s been spun around. Much like that, we’re going to create a cross section and spin it around to make this bottle.

As usual let’s get started by making a new ‘part document’. We’re going to draw our sketch on the right plane and in the normal view. So click on the right plane and then click normal view from the view menu on the top right.

Click sketch and choose line to get started. I want you to arbitrarily try drawing the shape that you see on screen now. It’s good to use the edge of the top plane as a reference line.

Now that we’re done with this shape. I want you to use the line tool to draw three more lines to create a little trapezoid as shown. Click on the right edge, pull back click, pull down, click and pull towards the right again. Make sure it has its end points on the sketch that we just created. Press escape to top drawing.

Now we’re going to create three more small wedges like this one. But we don’t want to go through the whole drawing process again...and even if we did..there’s no guarantee that we’re going to be accurate. So go ahead and select the three lines that created out little wedge by holding down control and clicking them like you would normally select multiple files in windows.

Now that we’ve selected our little wedge, choose the tools menu from the menu bar across the top. When you do this, you should be able to choose the sketch tools menu and the move-or-copy tool from it.

Doing this will bring up a little menu on our left like any tool normally does. We need three more instances of this wedge for our little bottle. So type in three in the space provided for it.

When you move into the modeling area again..you shall see a small circular sphere on your cursor. Click on the right edge of our sketch just below our old wedge and you’ll see three more trapezoids appear around your cursor. Move the cursor down and up to see how you can position them. When you’re done choosing a position, click again.

There we are.
As you may have noticed, we have our wedges but we need to remove these little lines on our original sketch. The trim tool allows us to remove parts of a 2d sketch that are bounded between two points.

So we go back to our tools menu and choose Trim from our sketch tools section. And using the trim tool we’re just going to snip away at the unwanted parts of our sketch.

There we are. We have a nice little cross section that needs to be spun around. To tell solidworks that we need to spin this section about its center, we need to draw a centerline.

Use the centerline tool in the sketch entities menu on the left. This draws just like a line. Once that’s done...we’re ready to get 3d!

Click features and choose revolved Bose base and click the green tick mark.

We have the bare shape of a bottle. To make it look a little better we need to create a curve between the neck and the first wedge.

To do that, click the edge of the first wedge. Once the edge is selected...choose the filet feature and give it an appropriate radius. This will help you smoothen the surface and make it look more like a bottle.

Sure...this looks like a bottle but its rock solid! To be able to pour anything inside the bottle we need to scoop out its insides. To do this, choose the circular plane on the top and choose the ‘shell’ feature. When you do this, you can specify the bottle’s thickness. Solidworks just scoops out the insides leaving the specified thickness.

I urge you to go ahead and try adding a material to this object and making it look more like a bottle! 😊