

# Sheetmetal parts for plasma cutting

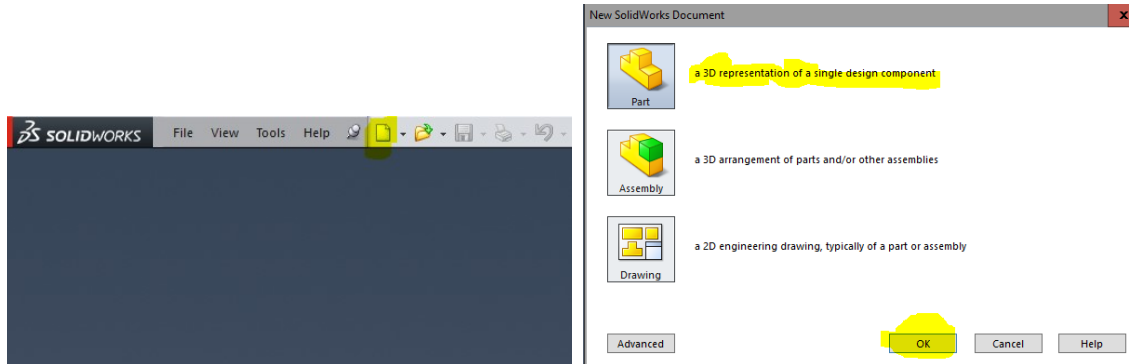
How-To: Using Solidworks CAD Program

C. Grossklaus 5/26/16

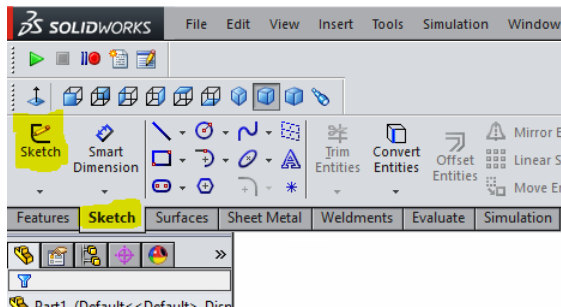


Using Solid Works sketch the basic outline of the part.

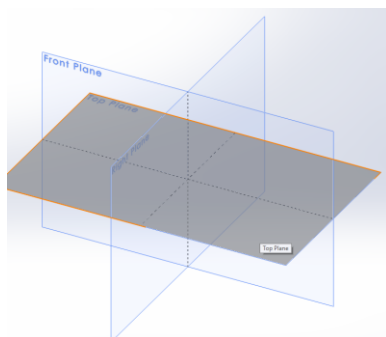
First open Solid Works and create a new part.



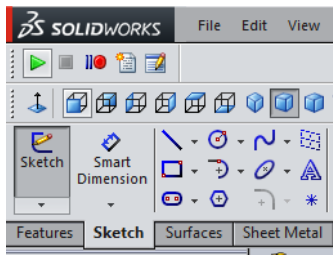
Go into the Sketch tab and start a new sketch.



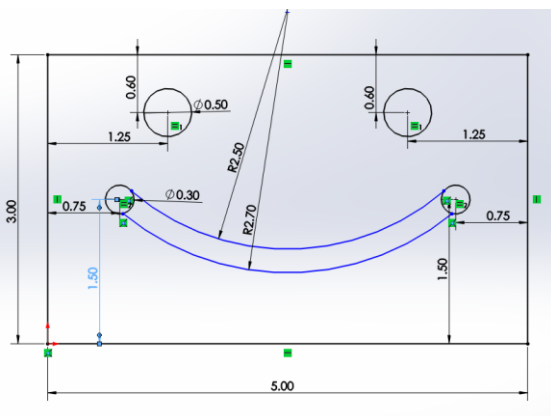
Select the plane you would like to start your sketch on. For flat parts I tend to start on the "Top Plane" although any will work.



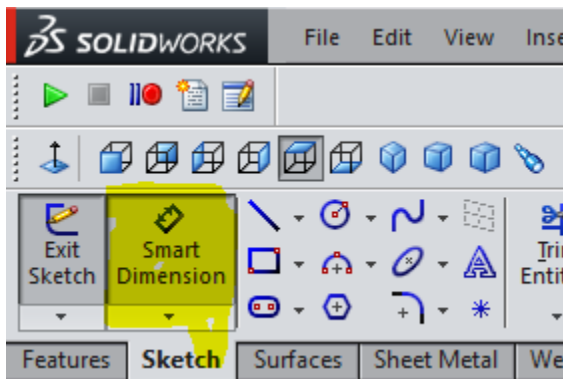
Begin drawing your part using the available tools.



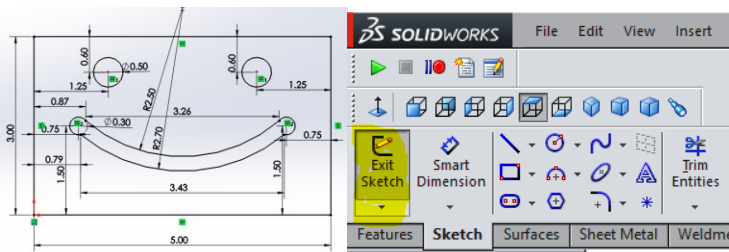
While drawing make sure to include dimensions so that your part will be fully constrained. Drawing elements that are fully constrained will show up as black and drawing elements that are not will be blue.



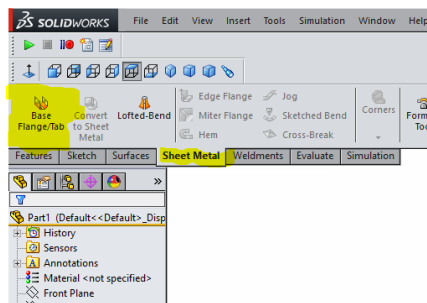
Dimensions can be added by using the smart dimension tool.



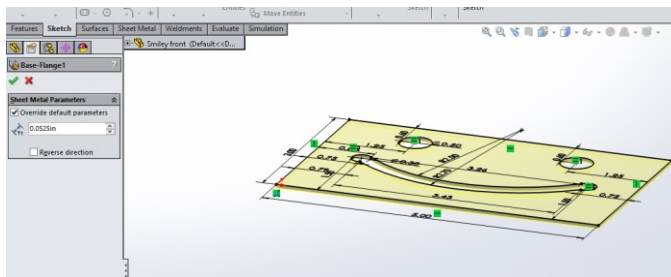
Once the sketch is fully constrained you can exit the sketch environment.



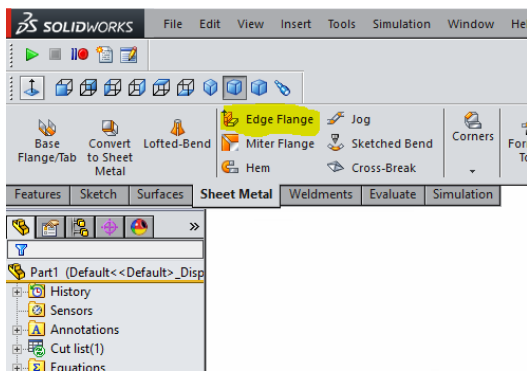
To make your part into a sheet metal part go into the Sheet Metal tab and select Base Flange/Tab.



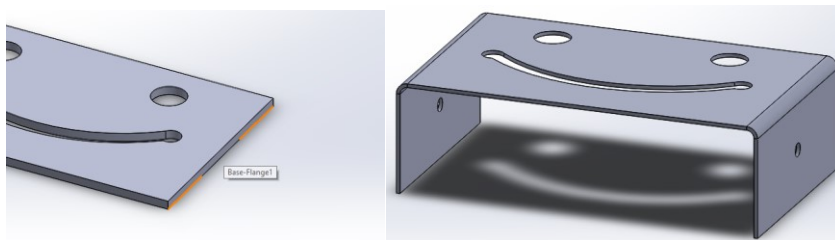
Select your part as the Base Flange and select the thickness of the sheet metal that you're going to use. In this case we're using 0.125". Click the green check when you're done.



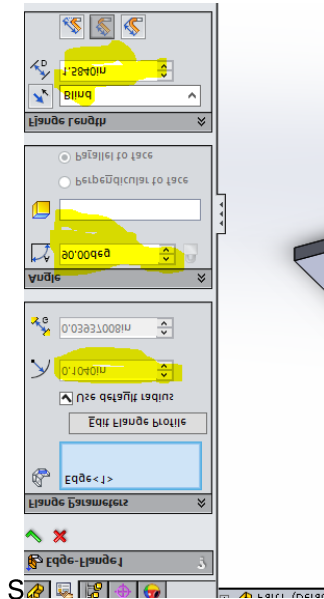
To add flanges to your part select the Edge Flange tool.



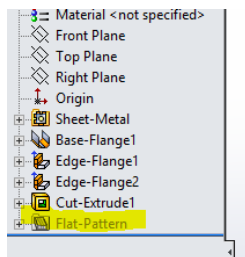
Select the edge that you want to add a flange to.



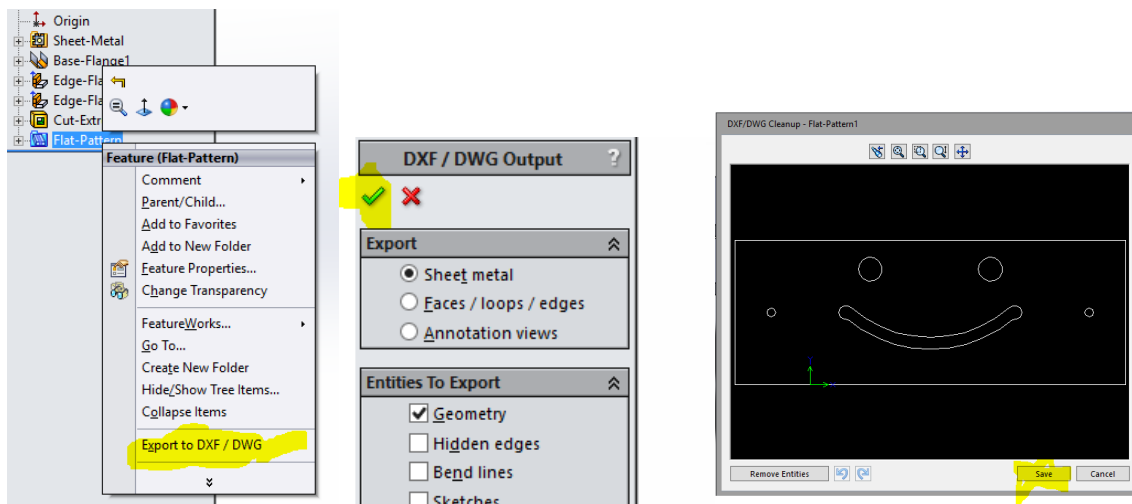
Select your bend radius (at Greenspeed our break that we bend 0.125" steel in has a radius of 0.104" so we have it set as a default), the angle of the bend and the length of the flange.



Once you are done with the design and ready to have it made right click on the flat pattern icon.



Select Export to DXF / DWG and save the part.



The DXF will be the outline the CNC plasma cutter will follow to cut your part out of metal.