

Lab #2: Design and Create 3D Model for the 3D Printer

Objective:

learn how to design and create a 3d model for the 3D printer in SOLIDWORKS.

Equipment:

- Computer: a generic computer
- Software: SOLIDWORKS
- 3D printer: Ultimaker 3 Extended

Safety Rules:

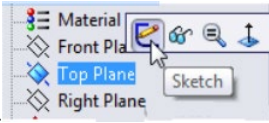
1. Turn off power supply BEFORE disconnecting the power line.
2. Don't touch any power source when your hands are wet.

Task 1: Create a simple box

1. Open the SOLIDWORKS and click New option in the software window

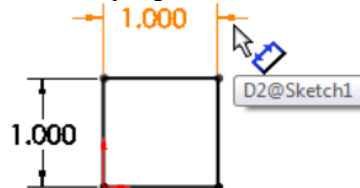
- 1) Click New  Option.

- 2) Click Part  and OK.

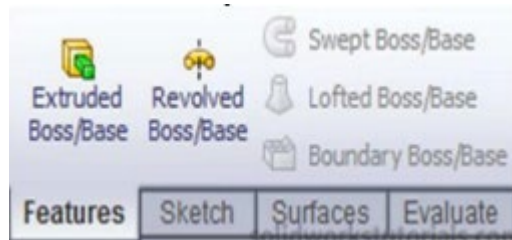
- 3) Click on Top Plane and click Sketch. 

- 4) Click Rectangle, sketch a rectangle starting from the origin.

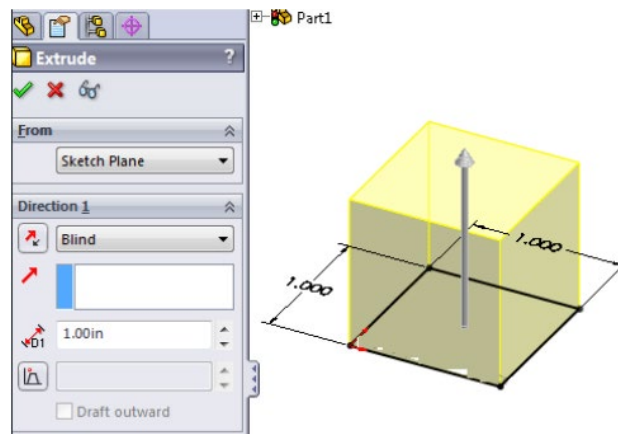
- 5) Click Smart Dimension, click side edge and click top edge to dimension it as 1.0in x 1.0in.



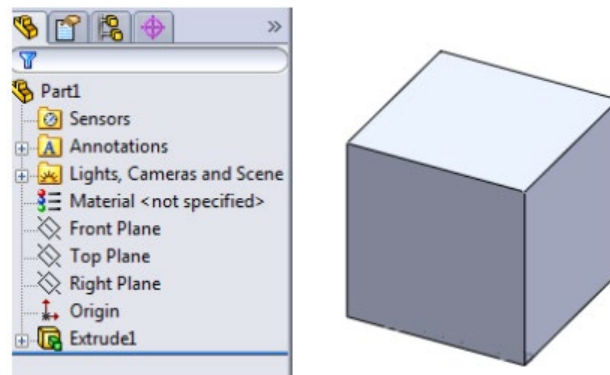
- 6) Click Features>Extruded Boss/Base.



set D1 as 1.0in and click OK.

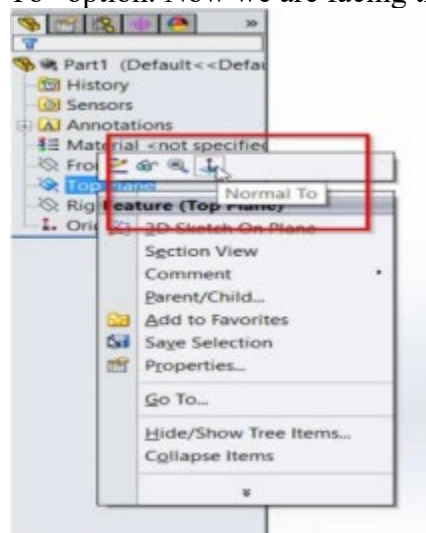


7) The final result.



Task 2: Create a basic helix shape

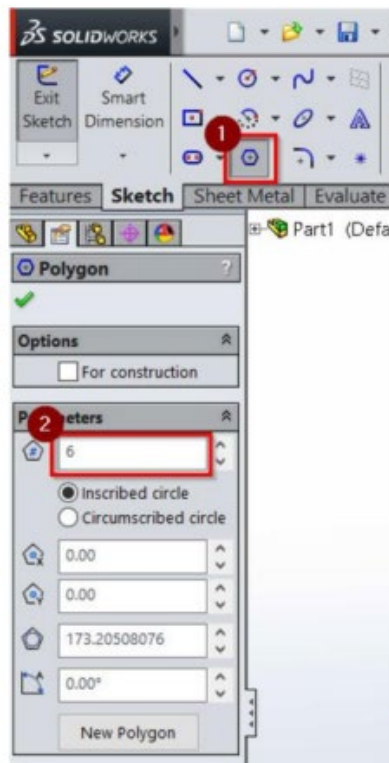
1. start a new part by going to New >Part > Ok. Since we have already set up SOLIDWORKS to use MMGS as default units, all our units should be in millimeters, grams and seconds. Because the Ultimaker 3 Extended uses MMGS.
2. On the left-hand side, you will see the Feature Manager Design Tree. Right-click on “Top Plane” and click the “Normal To” option. Now we are facing the Top Plane head-on.



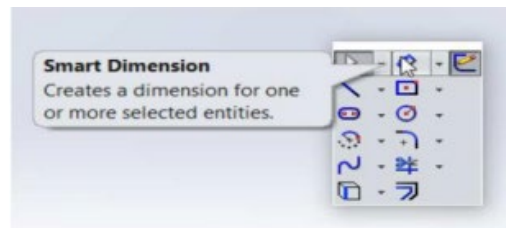
3. While the Top Plane is selected, switch to the Sketch tab at the top of the interface and select the “Sketch” tool in the top left corner. We are now creating a sketch on the Top Plane.

Anything we sketch will appear on the Top Plane. First, we want to sketch the basic hexagonal outline of the nut.

- 1) Click the “Polygon” tool in the Sketch tab. A list of parameters should appear in the left-hand panel.
- 2) Set the number of sides to 6 (hex = 6). Other parameters such as center position, angle, etc. are also modifiable, but we will not adjust these.
- 3) Next, over the origin (the base of the two red arrows) until the cursor snaps onto the dot in the middle. Left-click once to start the sketch.
- 4) Left-click again at some point to the right of the first point. Press enter to finalize the sketch.
- 5) Now, select the “Circle” tool in the Sketch tab and sketch a circle starting at the origin, similar to how we sketched the hexagon. Press enter to confirm the circle sketch.

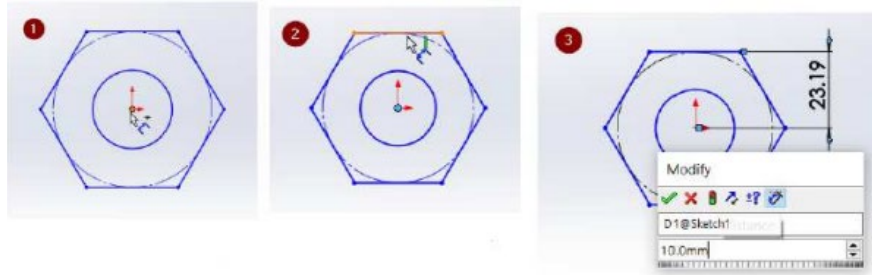


4. Now we will assign dimensions to our outline. Make sure we are still in sketch mode by looking at the top left corner and seeing that the Sketch button is depressed and now reads “Exit Sketch”. Press the “S” key on the keyboard. In the small window, click the “Smart Dimension” tool.

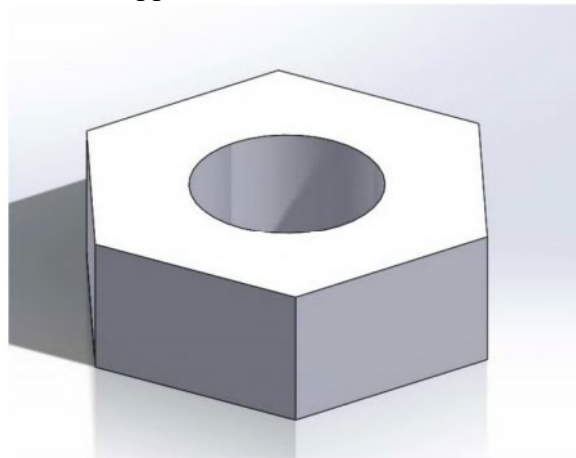


- 1) Left-click the center
- 2) Then left-click the top side (be sure to select a side edge and not one of the points between two edges).

- 3) A dimension measurement should pop up. Left-click again anywhere on the screen and a small window will appear. Enter 10mm into this field and hit enter. The hexagon is now assigned a dimension of 10mm (center to edge). Double click the dimension label to change the assigned dimension at any time. Follow the same procedure to set the diameter of the circle to 10mm. Select Exit Sketch when finished. The sketch should now appear as follows:





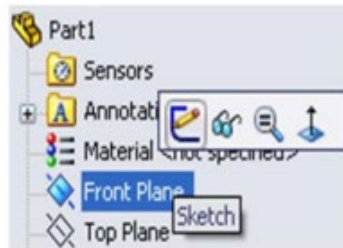
5. Now that our sketch is assigned dimensions and finalized, we are ready to bring this shape into the 3rd dimension. Select our sketch by clicking “Sketch1” in the left panel. The sketch should turn light blue as it is selected. Next, switch to the Features tab at the top and click the “Extruded Boss/Base” tool. Immediately you should see the shape gain depth. Using the parameters on the left panel, give the nut a depth of 7.5mm. Hit enter to confirm the extrusion. The shape should now appear as follows:



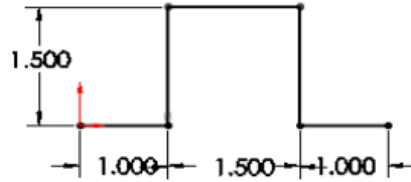
Task 3: Create a U bracket sheet metal


1. Open the SOLIDWORKS and click New option in the software window

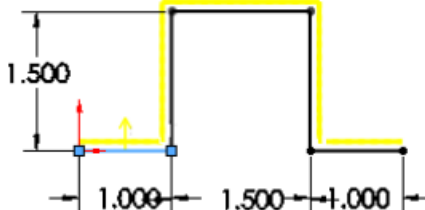
- 1) Click New  Option. Click Part  and OK.
- 2) Click Front Plane and click on Sketch.



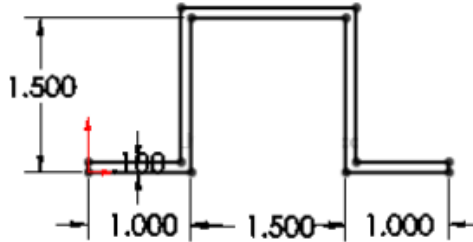
Use Line, sketch U shape. Dimension sketch with Smart Dimension as 1in x 1.5in x 1in and 1.5in height.



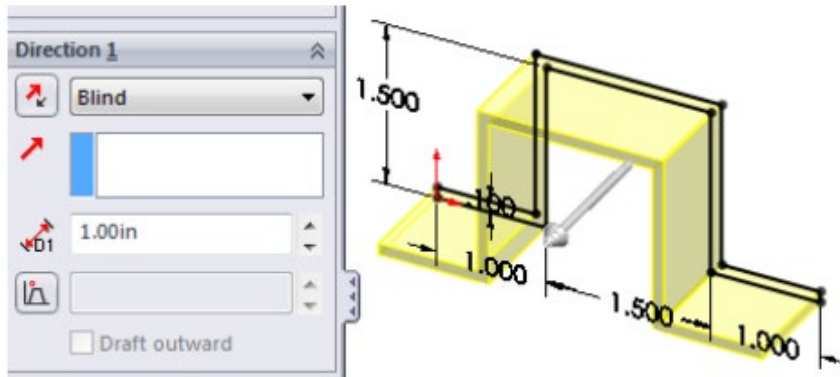
- 3) Click Offset Entities  and click U sketch. Set offset distance as 0.1in, check Reverse box and OK.



- 4) Use Line, sketch and connected open end of this sketch and make it close both ends.



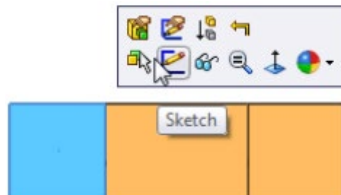
- 5) Click Features>Extruded Boss/Base set D1 to 1in and OK.



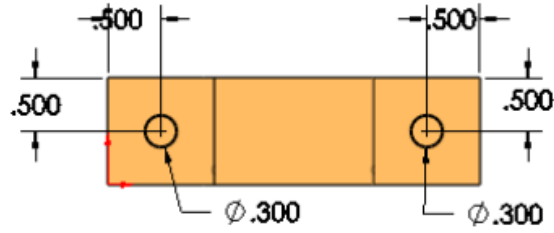
- 6) Click View>Bottom



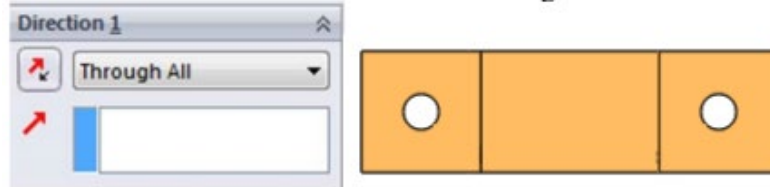
Click on the bottom face and click Sketch.



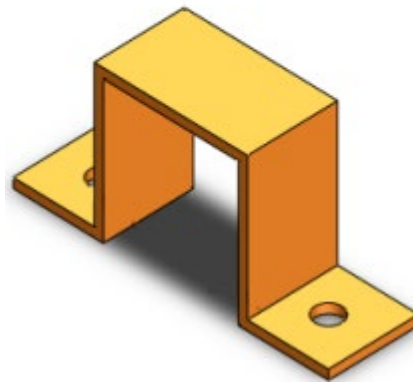
- 7) Click Circle and sketch 2 circles on the bottom face each side. Use Smart Dimension to dimension this sketch as sketched below.



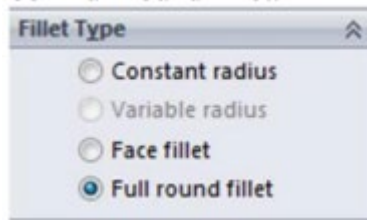
- 8) Click Features>Extruded Cut and cut Through All this circle.



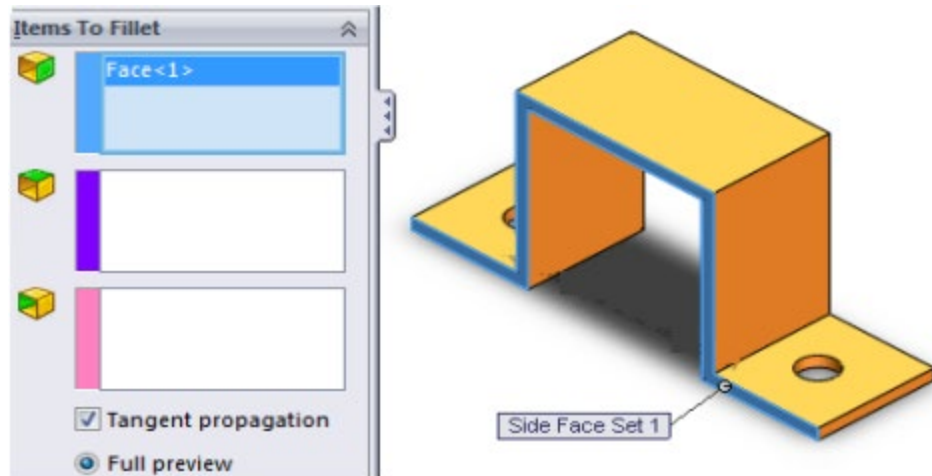
- 9) Click View>Isometric.



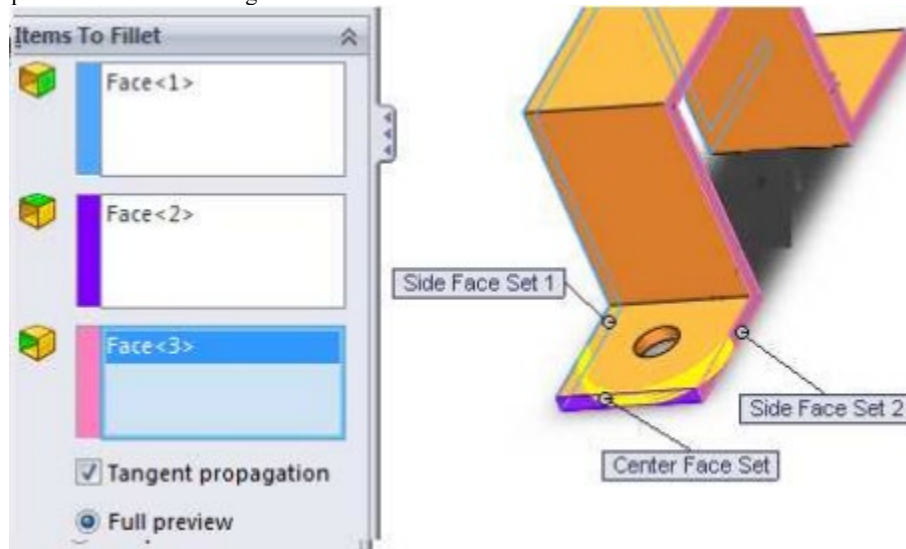
- 10) Click Fillet, checkbox Full round fillet.



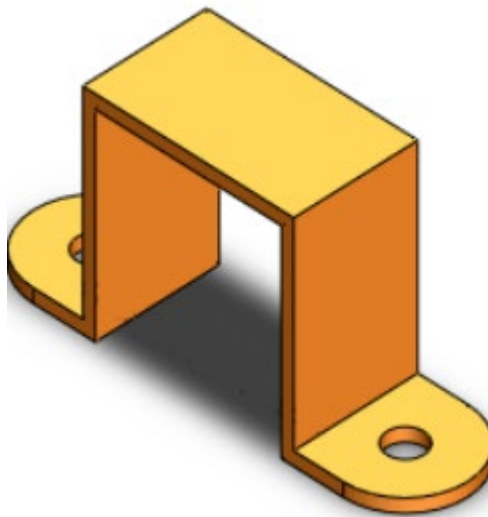
- 11) Click side left side face as Side Face 1.



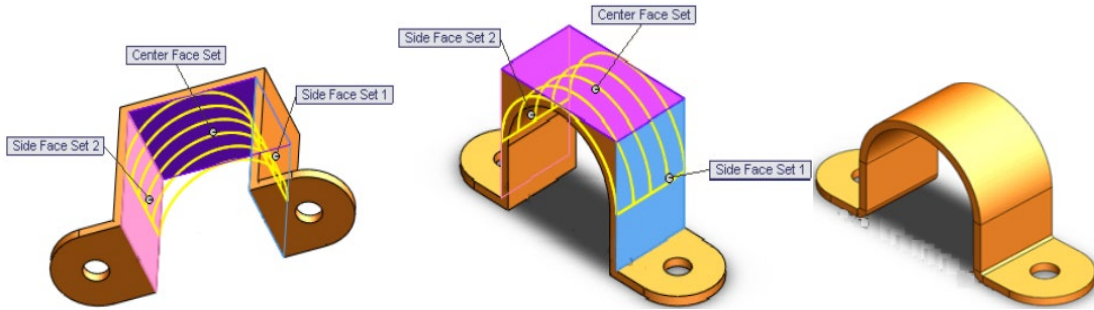
12) Click on the pink box and click the right side face as Side Face Set2 and OK.



13) Repeat on the other side.



14) Repeat step 11 – 13 for inner face and outer face of U bracket.



- 15) Click Sheetmetal>Insert Bends, click flat face as a reference when it flattens. Set bend radius to 0.03in and K factor 0.5 and OK. Then the final result is as the right figure.

