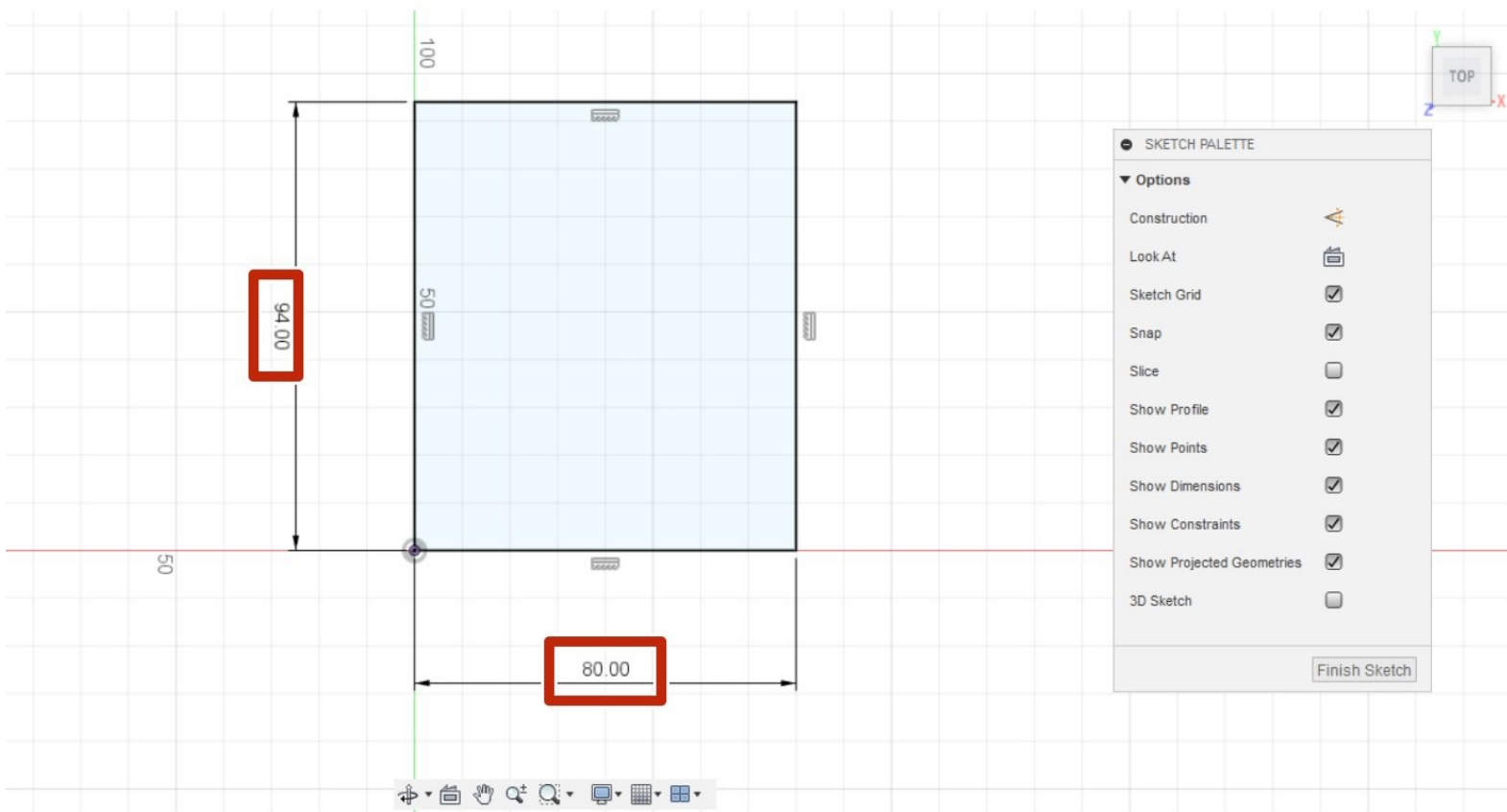




Color changing lamp cover and shade

This guide concludes the instructions for the CAD portion of the RGB color changing lamp workshop.

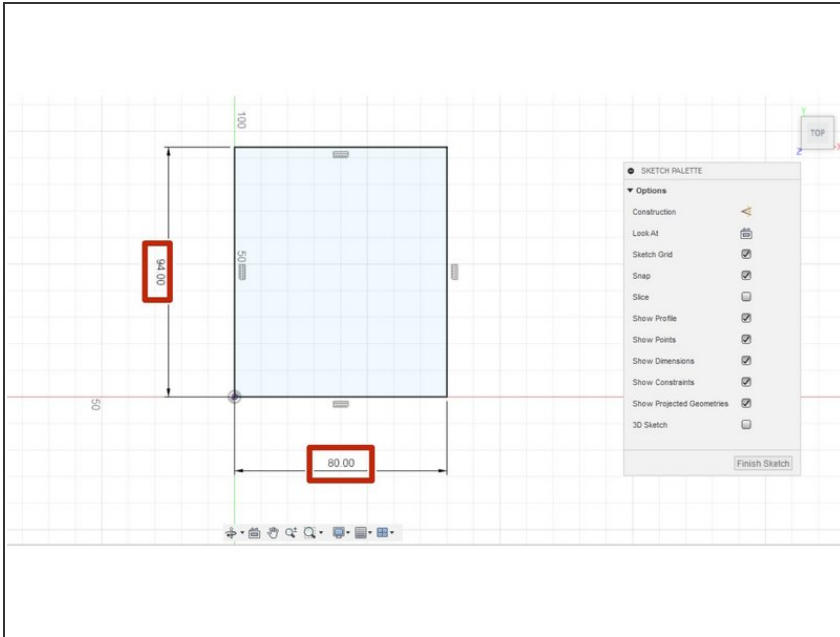
Written By: Eli Jared Fastow



Introduction

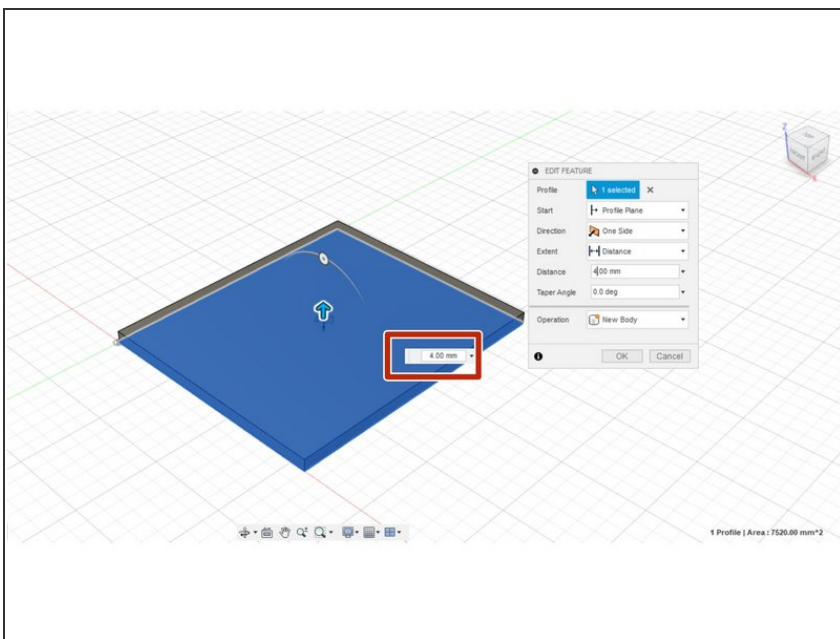
In this guide, you will finish the CAD portion of the RGB color changing lamp workshop in Autodesk Fusion 360.

Step 1 — Cover profile sketch



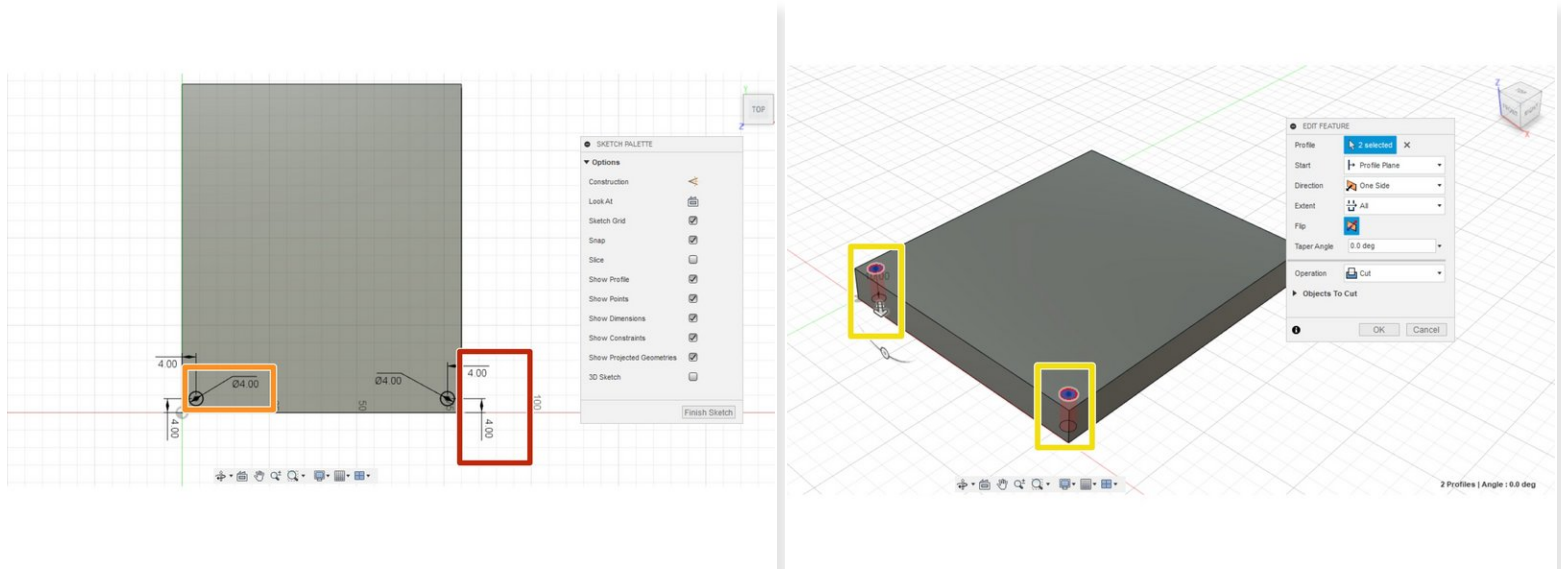
- We will create a cover for an LED and a bulb for a color changing lamp
- Open a new Fusion part and create a new sketch
- Create a rectangle 94 mm by 80 mm in size
- ❗ Insuring the 3D printed components fit together and accommodate the electronics requires dimensional accuracy. Please do not skip on properly dimensioning you sketches or features.

Step 2 — Cover profile extrude



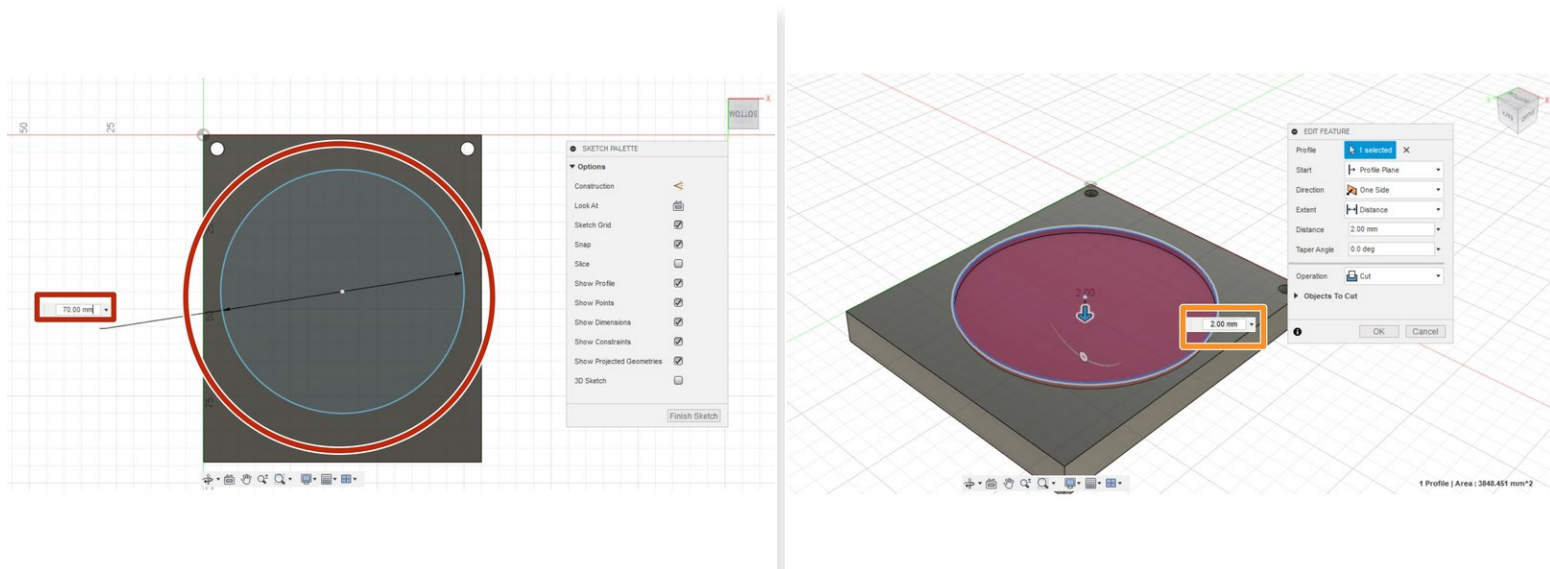
- Exit the sketch
- Extrude the rectangle by 4 mm
- This forms the cover for the LED

Step 3 — Screw holes



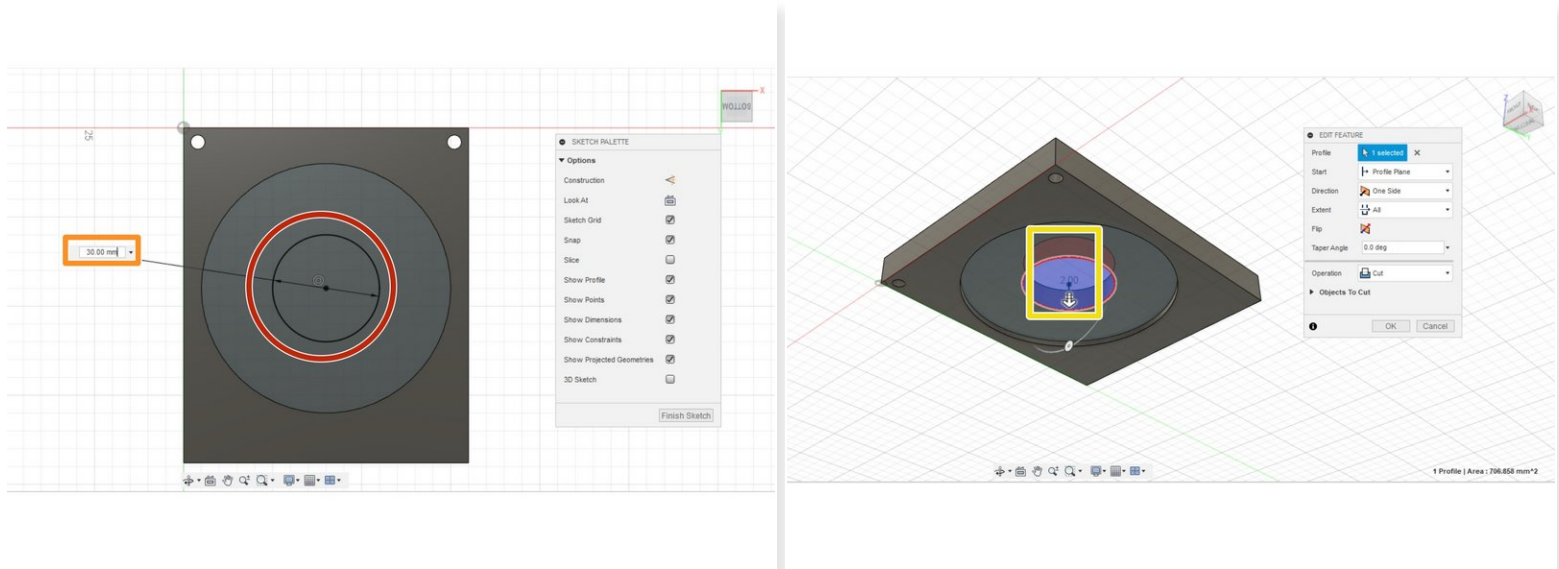
- Open a new sketch on the large face of the arduino cover
- Create a point 4 mm in both the X and Y axis from two corners of the face.
 - ❗ Tip: drawing two 4 mm lines - one from the corner, the other orthogonal to the first - identifies this point
- Create two circles 4 mm in diameter at these points
- Exit the sketch and open the extrude tool
- Select the two circles you just created and extrude them through the lid to a distance of -4.00 mm. This creates two screw holes.

Step 4 — Outer circle



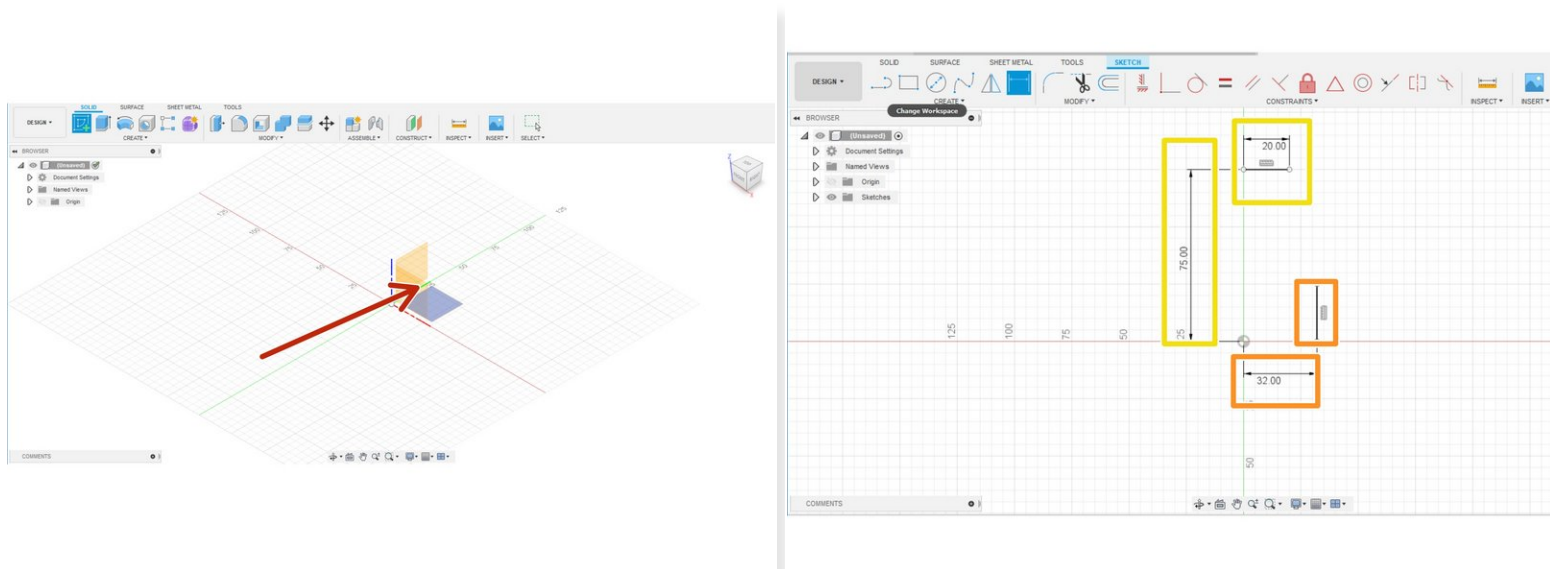
- Open a new sketch on the opposite face of the cover.
- Create a circle 70 mm in diameter centered in the middle of the cover face.
- Extrude the circle 2 mm into the cover, creating a circular relief

Step 5 — Inner circle cut



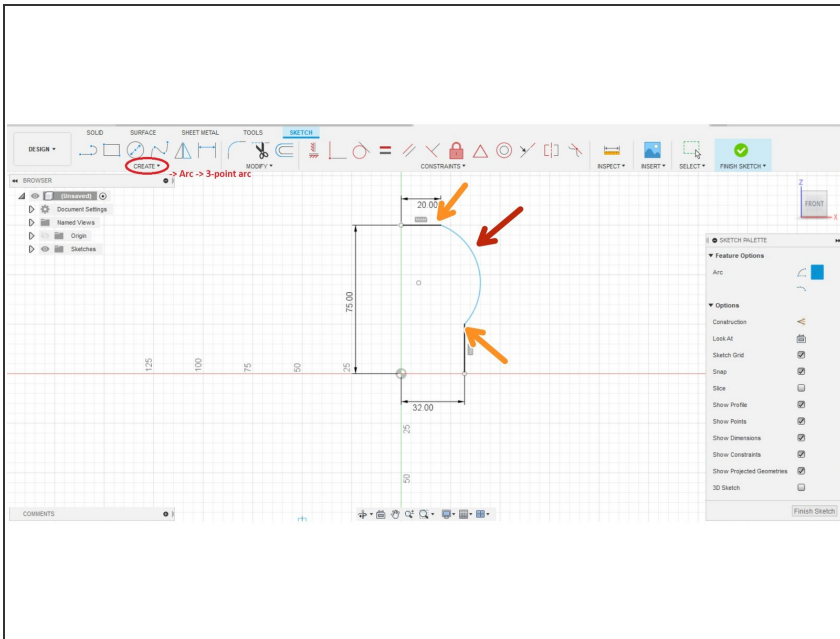
- Open a sketch on the face created by indenting the circle in Step 4.
- Create a circle concentric with the one extruded in Step 4
- This new circle should have a diameter of 30 mm
- Exit the sketch then extrude the new circle at least 2 mm into the cover, creating a hole for the LED

Step 6 — Starting to make the lampshade



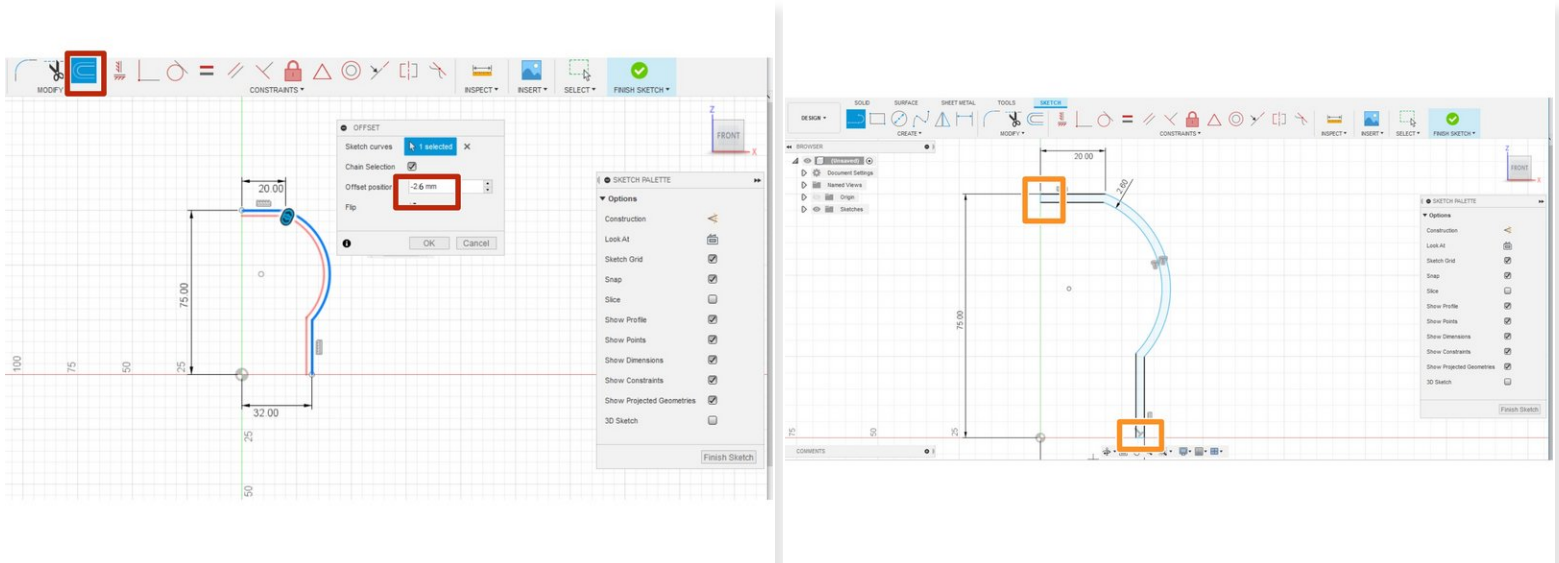
- Make a new project and start a sketch on the XY plane (front view)
- Create a line 25 mm in the y direction starting 32 mm in the x direction from the origin
- Create a line 20 mm in the x direction starting 75 mm in the y direction from the origin

Step 7 — Scoring a three-pointer



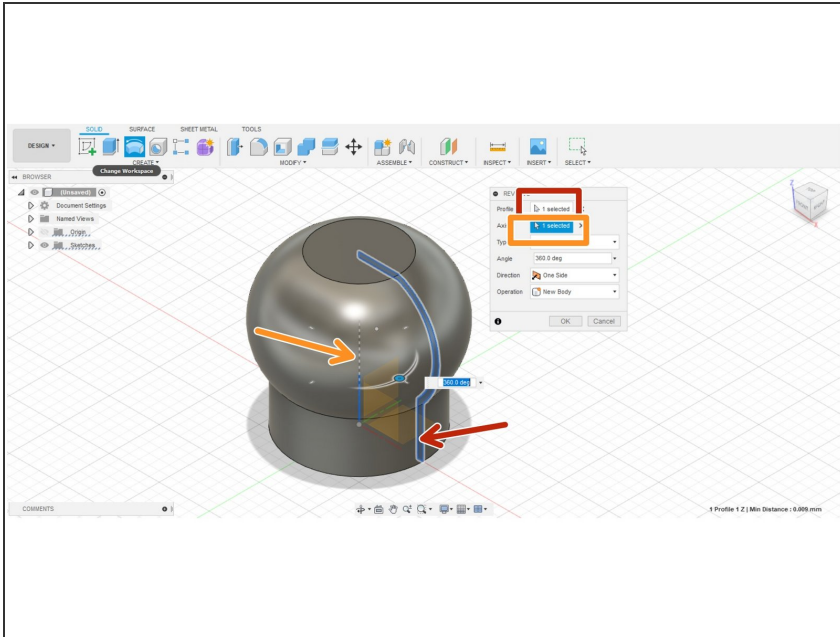
- Create a 3 point arc
- Use the endpoints of the lines as the first and second points of the arc, then place the third point somewhere off to the right.

Step 8 — Making an offset



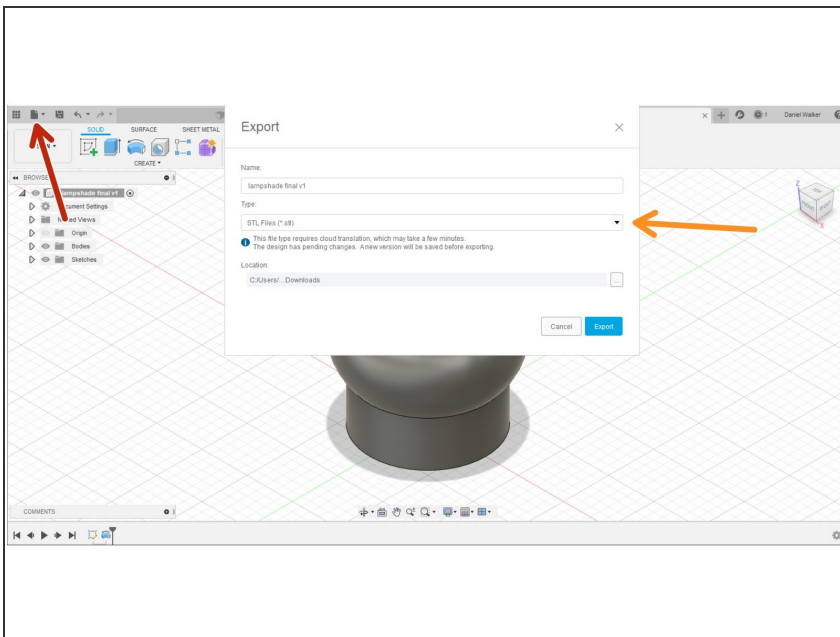
- Offset the path you just created by 2.6 mm inward
- Use the line tool to connect the endpoints at both ends to create a closed profile

Step 9 — The world revolves around puns



- Exit the sketch
- Open the revolve tool and select the profile you just created
- Use the z axis as the axis of revolution
- Revolve the profile around the full 360 degrees to create a bulb

Step 10 — Exporting your files



- To prepare the two components we just made for 3D printing, we need to export them as .stl files
- Click File -> Export
- Select .stl file as the format
- Repeat this process for the other components of this project.

This concludes the CAD portion of the workshop. Please make sure that you follow up on your print request. Next, you will be assembling your Arduino circuit and beginning to code.