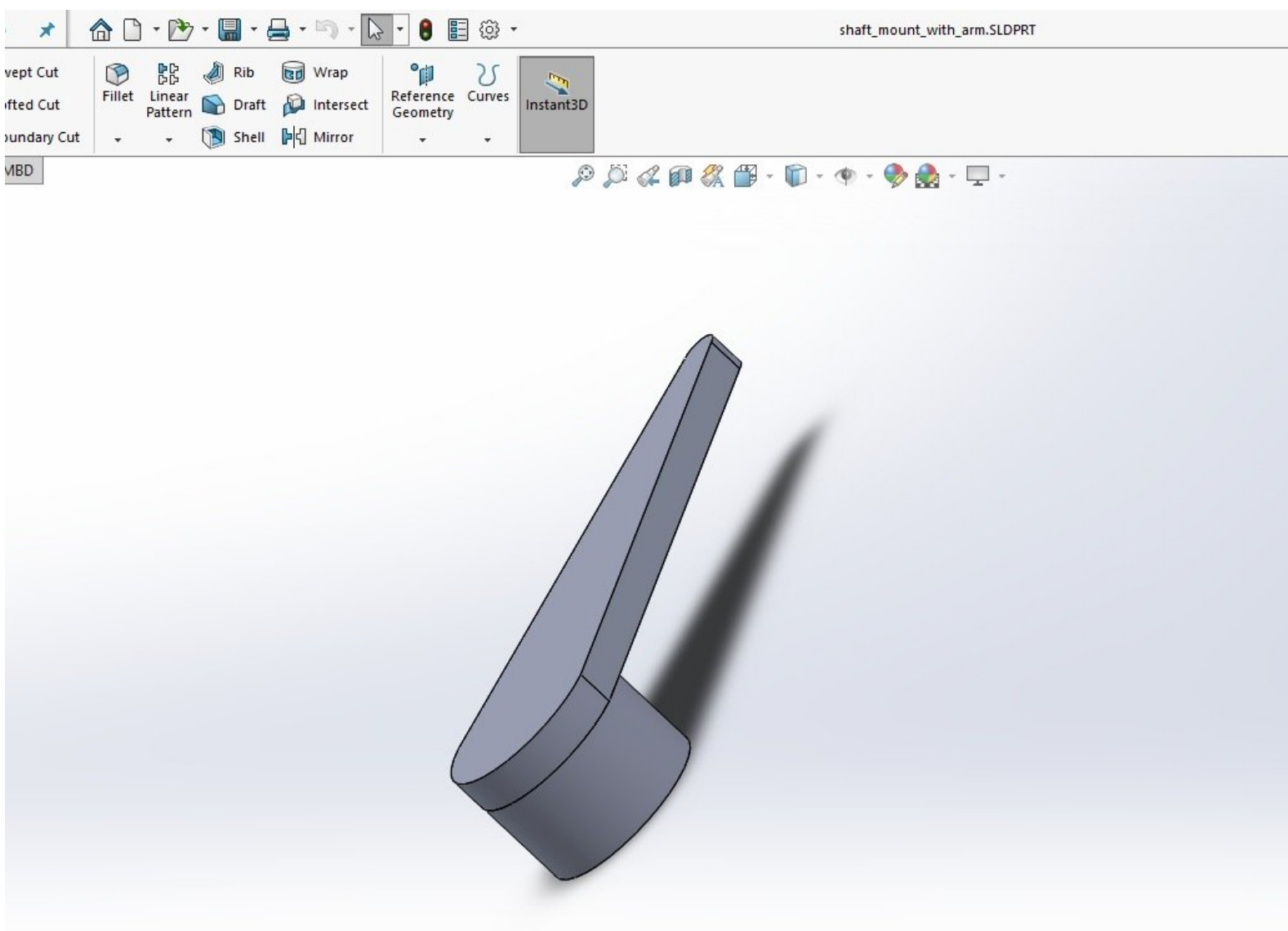




SolidWorks Basics

This guide will just barely scratch the surface on creating shapes and extruding them in SolidWorks.

Written By: Matt P. Lamparter



INTRODUCTION

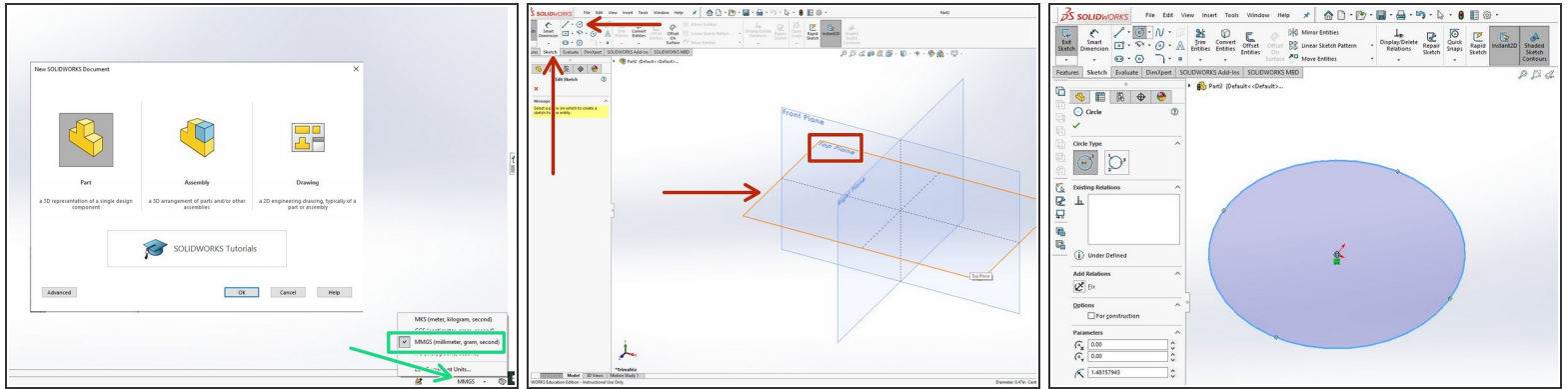
This guide will provide you with the bare minimum needed to create shapes and extrude them in SolidWorks. Concepts/commands covered will include creating 2D shapes, dimensioning them, extruding them, and making cuts. The end goal is to have an .STL file for 3D printing. The tutorial is based on a component used in the DAMNED project.

Step 1 — The basics



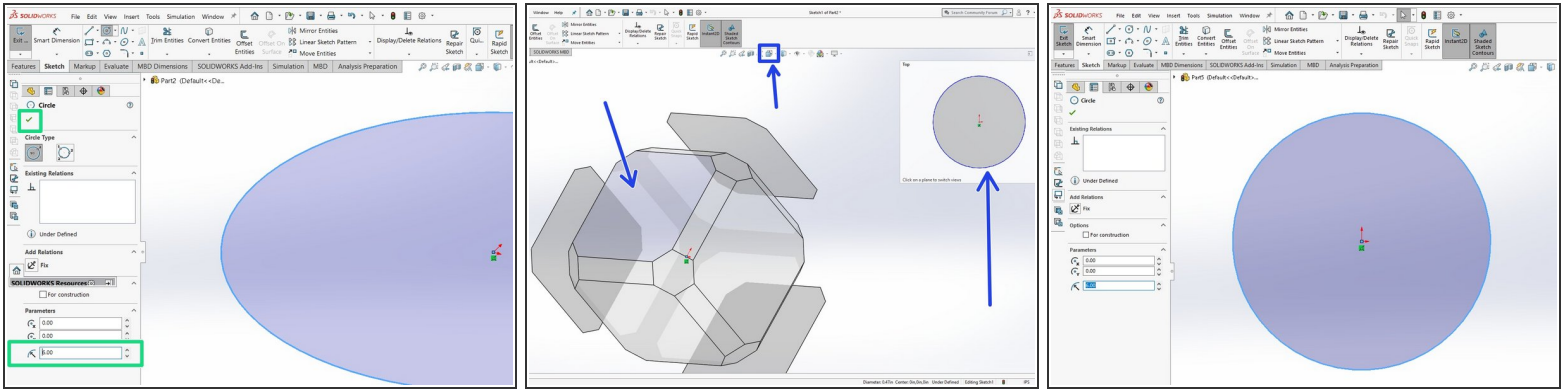
- Launch SolidWorks from any computer in the engineering buildings. You may also install it on your personal Windows machine by following the instructions [here](#).
- The basic idea in SolidWorks is to create a 2D shape, add dimensions to it, and then extrude it. In other words you "pull" the 2D shape into a third dimension and give it depth.
- Once you have a basic 3D shape then you can add additional 2D shapes on to it and further extrude these.
- Alternatively, you can add a 2D shape onto a 3D part and use the 2D shape to create a cut.
- There are **many** more things you can do in SolidWorks, but for the purposes of this tutorial that summarizes everything.

Step 2 — Create a circle



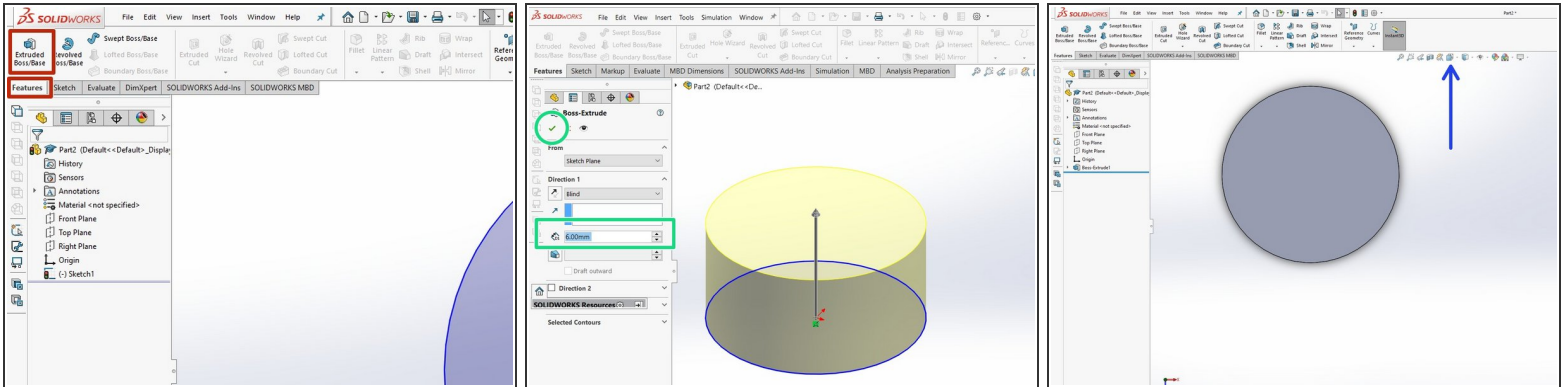
- After launching SolidWorks, begin by creating a new part.
- Once you have clicked OK to create a new part, be sure to choose metric units of measurement (MMGS) in the bottom right corner of the screen as shown here in green.
- Create a center circle by clicking the **Circle button** within the **Sketch tab**.
- Choose to create your sketch in the **Top Plane** by hovering your mouse over the correct plane until it highlights orange, then left clicking.
- Click the origin of the sketch (small red axes near the center of the screen) to anchor the center of your circle there.
- Move your pointer out away from the center of the circle. Create a circle of arbitrary radius by clicking again.
- **i** Rotating the scroll wheel of your mouse will zoom in or out on the drawing.

Step 3 — Dimension the circle



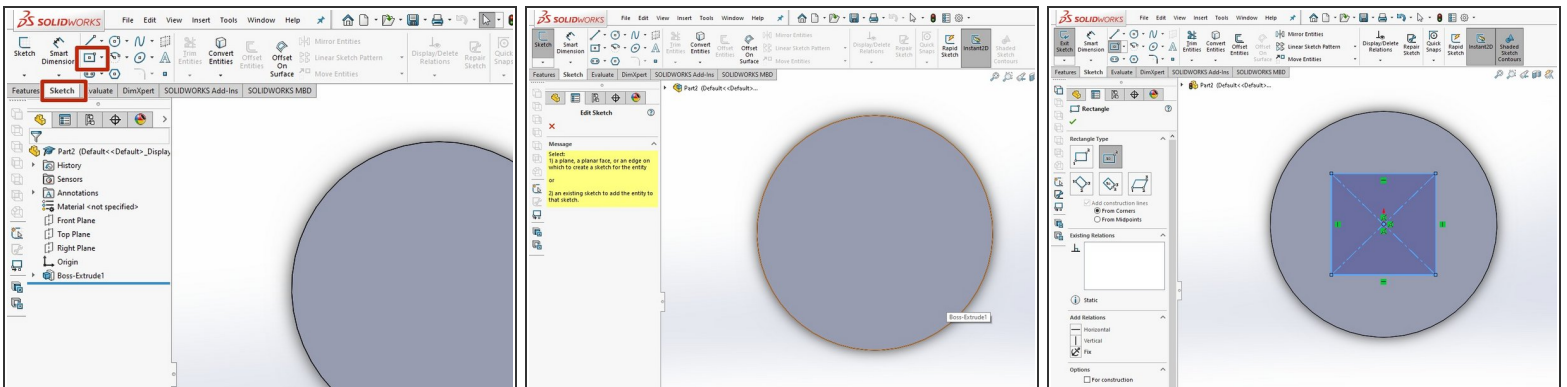
- Enter a radius value of **6.0** in the appropriate field in the left window column. Click the green check mark to confirm and exit the circle tool.
- This circle will be the outside of the motor shaft extension. Let's view it from the top plane to make it easier to see.
- Click the **View Orientation** button near the top of the screen. A cube will appear allowing you to choose a plane from which to view the sketch.
- Hover over the plane you wish to choose. Look at the preview shown at the top right of the screen. Confirm it looks like the example shown here and then click on that plane to orient your sketch properly.

Step 4 — Extrude the circle



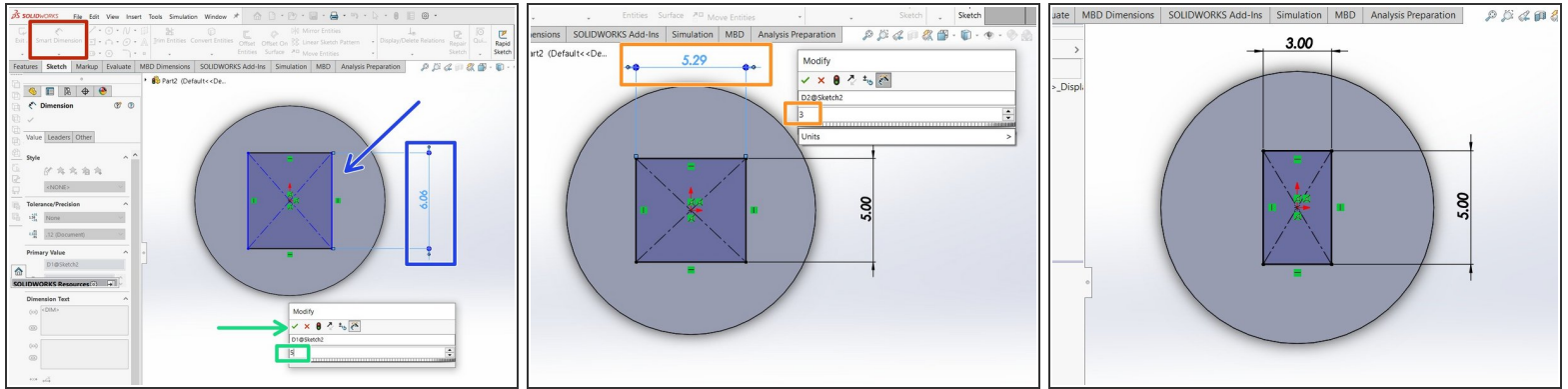
- Switch the to **Features** tab. Click **Extruded Boss/Base**.
- Enter a value of 6.0 for the extrusion height. Click the green check mark in the upper left to accept the value.
- Use the **View Orientation** button again to rotate the newly extruded cylinder so that you are viewing it from the top plane.

Step 5 — Add a sketch to make a cut



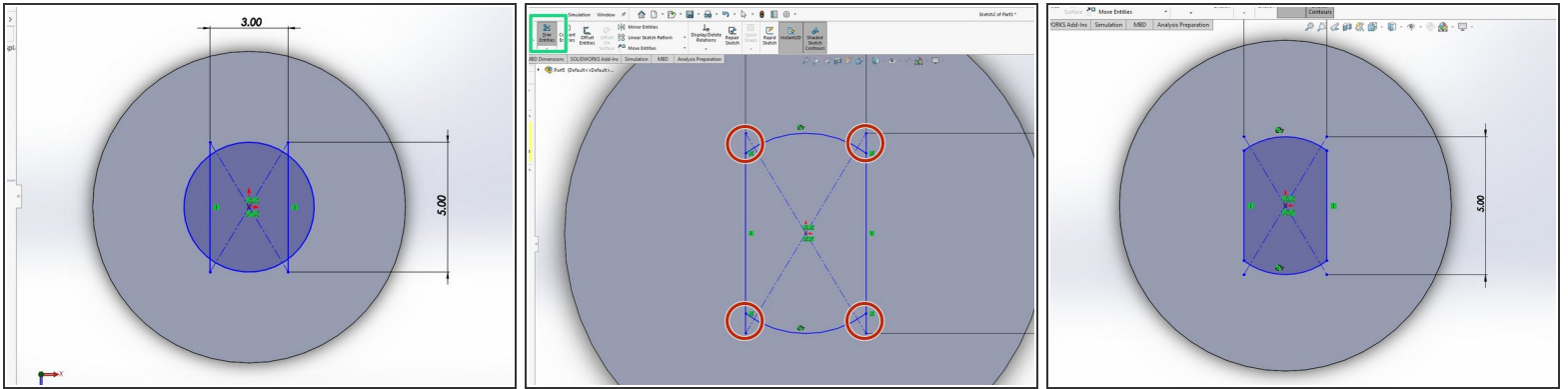
- Return to the **Sketch** tab and click the **Center Rectangle** button.
- SolidWorks will require you to choose a plane on which to create your sktech. Click the top surface of your cylinder.
- Click the center of the circle (where the red axes are shown) to anchor the center of the rectangle and create a rectangle of arbitrary dimensions inside of the circle.

Step 6 — Dimension the rectangle



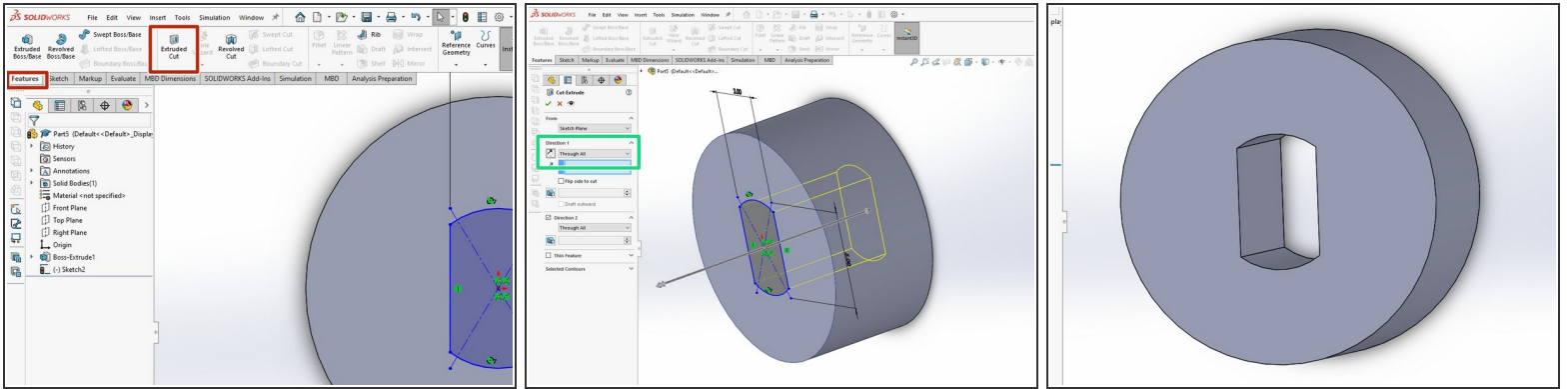
- Click the **Smart Dimensions** button.
- Click the right side of the rectangle and drag the dimension out to the right of the circle. Click again to bring up a Modify box.
- Enter a value of 5 and click the green check to accept the value.
- While still in Smart Dimension mode click the top edge of the rectangle and drag the dimension above the circle. Click to display the Modify box.
- In the dimension field enter a value of 3.
- Click the green check box when you are done. Your rectangle should now be dimensioned properly.

Step 7 — Add a circle, make some trims



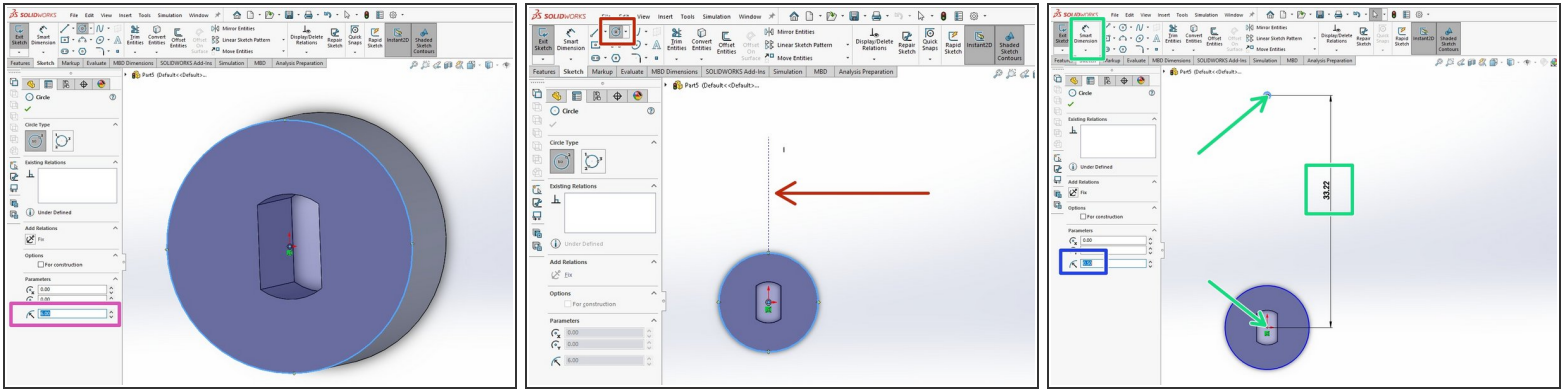
- Add a second, concentric circle inside of the first with a radius of 2.62 mm. Be sure to click the green check box when you are done to exit the circle tool.
- Now click on the top edge of your rectangle to select it. Press the delete key on your keyboard. Do the same thing to remove the bottom edge of the rectangle. The result should look like the first image here.
- Click the **Trim Entities** tool. With this tool activated, left click just outside of the left edge of the smaller circle. While holding down the left mouse button drag the cursor across the left edge of the smaller circle. This edge should be trimmed flush to the 5 mm edge of your rectangle. Repeat this process on the right side.
- The result of these two trims should look like the second image shown here.
- Repeat this trimming procedure to eliminate the four small segments of the 5 mm edges extending beyond the remaining arcs of the smaller circle. These segments are highlighted in red in the second image.
- Your final result should look like the third image shown here.

Step 8 — Use the new sketch to make a cut



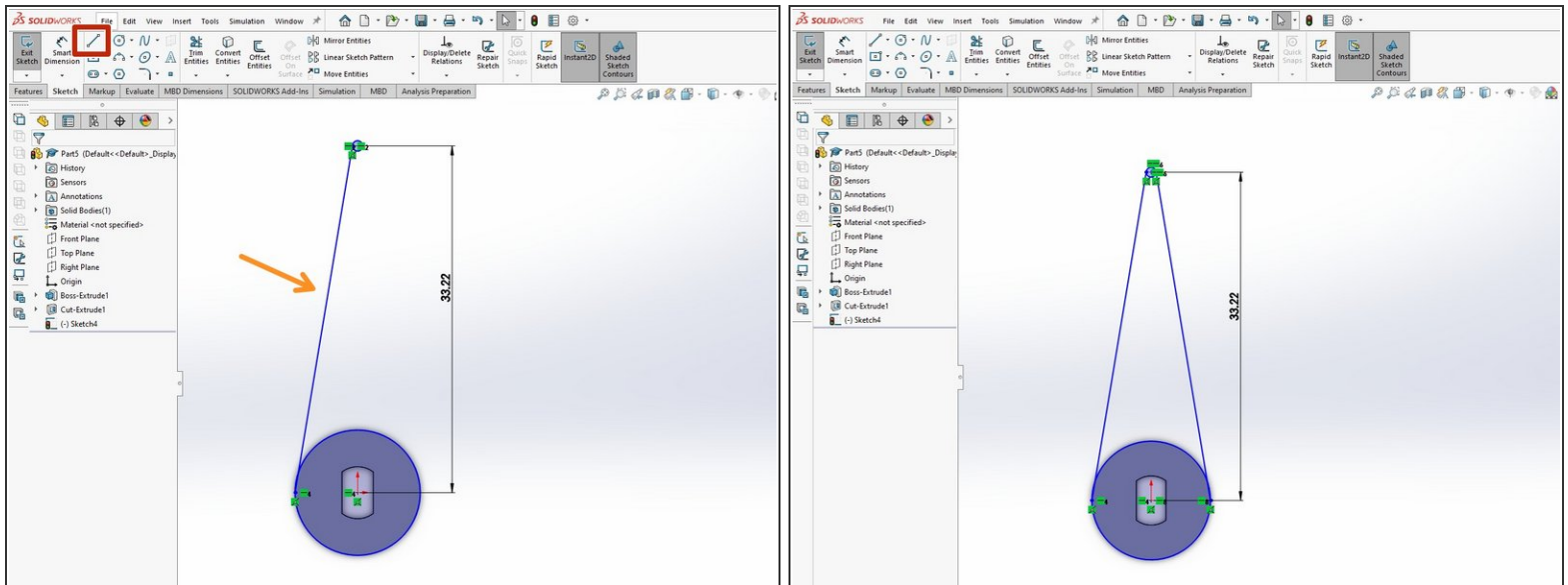
- Switch to the **Features** tab and click the **Extruded Cut** button.
- Under the **Direction 1** heading in the left column pull down the drop down menu and choose **Through All**. This will tell the cut operation to cut through all objects in its path.
- If your mouse has a scroll wheel you can click and drag with this scroll wheel (don't rotate the wheel) to rotate your 3D part and view the proposed path which will be cut. The path is shown in this example by the yellow lines.
- Click the green check mark in the left column to confirm the cut operation. You can rotate your part to see that the cut has been made through the cylinder.
- **i** This cut will allow the shaft of the stepper motor to mount on the arm you are creating.

Step 9 — Start the arm sketch



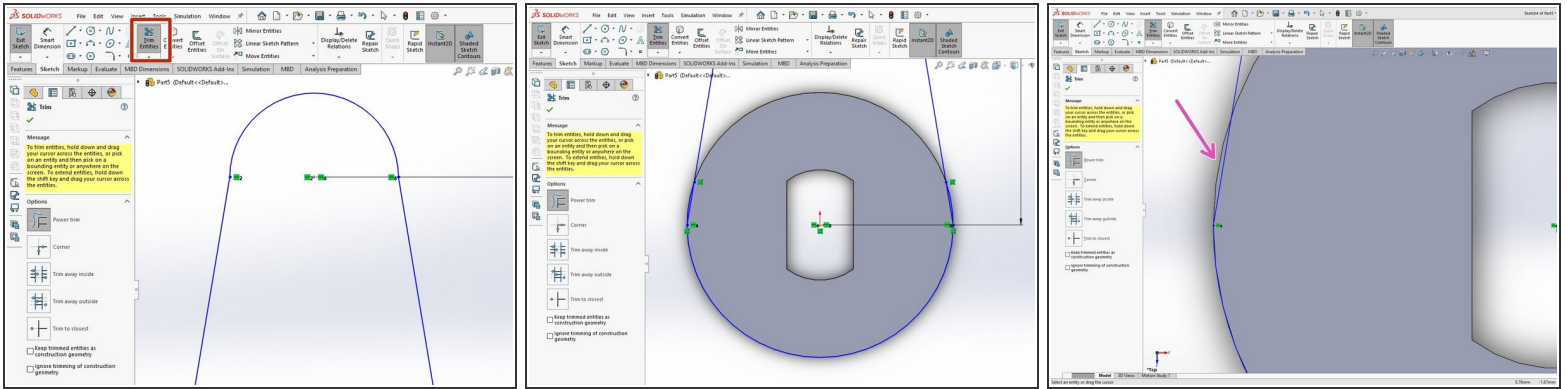
- Return to the **Sketch** tab and create a **Center Circle** which is concentric with your cylinder. Click on one of the faces of the cylinder when prompted for a sketch plane.
- Anchor the circle in the center of the cylinder then click the outer edge of the cylinder to make the radius of the new circle identical to that of the cylinder. Confirm that the radius of your new circle is 6 mm.
- Click **View Orientation** and choose to view the cylinder from the top down so that you are viewing the face of the cylinder where you just added a new circle sketch.
- Create another **center circle** and this time, after clicking the **Center Circle** button, hover above the existing circle you just drew and move the mouse near the (invisible) vertical axis. A dotted line should appear emerging from the center of the existing circle. This will lock the new circle in line with the existing circle.
- Click to anchor the center of the new circle on this line. Create a small circle of arbitrary diameter. In the radius **Parameters** field in the left column enter a value of 0.5.
- Click the green check mark to complete the circle.
- Click **Smart Dimension**. Then click the center of the large circle followed by the center of the small circle. Define this distance to be 33.22 mm.

Step 10 — Add sides to the arm



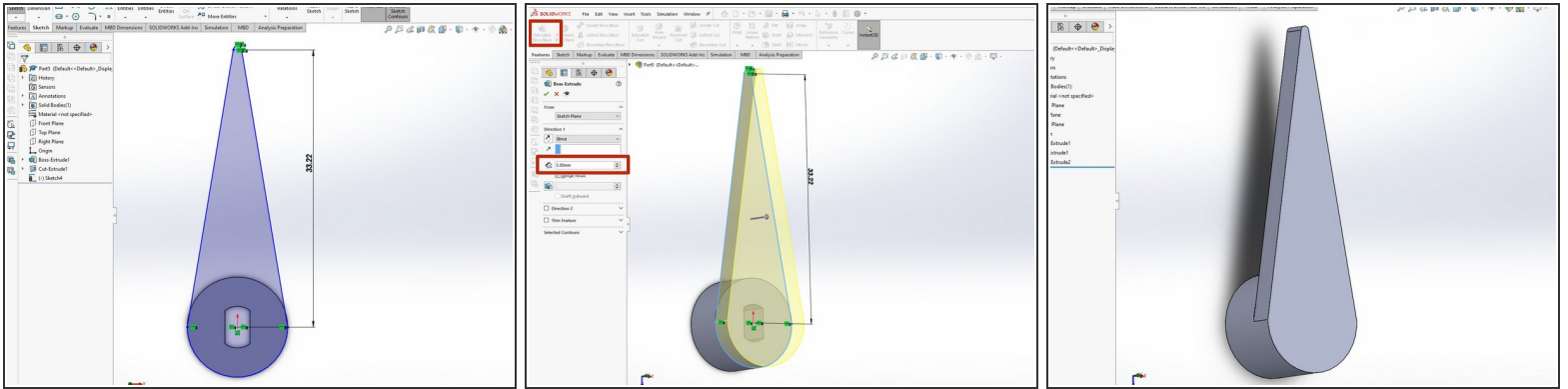
- Within the **Sketch** tab choose to create a **Line**. Use your mouse scroll wheel to zoom in on the small circle. Hover over the left edge of the small circle. An orange anchor point should appear. Click this point to anchor the line to be tangent to the left side of the circle.
- Now zoom in on the larger circle and hover near the left edge of this circle. Click the orange anchor point on this circle to terminate the line at the tangent point on the circle's edge. Hit the escape key to stop drawing this line.
- Repeat the same process to create an identical line on the right side.

Step 11 — Complete the arm sketch



- Click the **Trim Entities** button within the **Sketch** tab. Use the mouse scroll wheel to zoom in on the small circle.
- Click below the small circle and drag the mouse across the bottom of the small circle. The bottom arc of the circle will be trimmed up to the point where it intersects on either side with the tangent lines drawn previously. Release the mouse button. Your sketch should look like the first image here.
- Now zoom out and then zoom in again on the bottom circle. This time click above the top edge of the bottom circle and drag the mouse across the circle top edge. The top arc should be deleted up to the point where the tangent lines intersect with it. Release the mouse button.
- Finally, zoom in on one side of the large circle. The tangent line has created a cord as it intersects with the circle at two points. Click outside of the large circle and drag the pointer across this outer arc to remove it. **Be careful to stop after crossing the arc and do not continue to trim the cord itself.**
- Repeat this trim on the right side of the large circle as well.

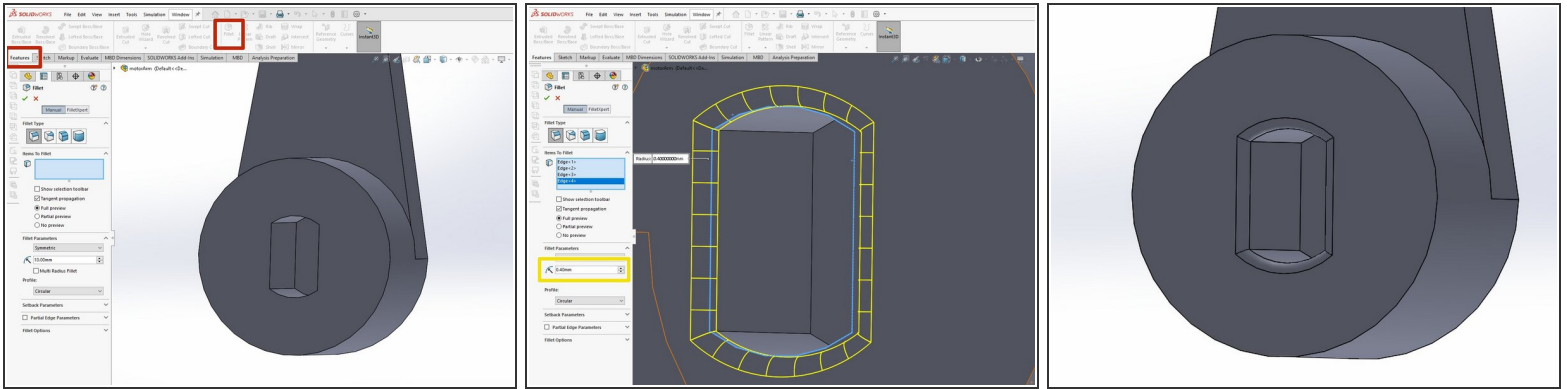
Step 12 — Extrude the arm



- i** If you do not have a closed path for the pointer arm at this point you should stop and return to the previous step. The extrude will only work properly if you have a closed path. A complete path should appear as a blue outline with a grey shaded interior as shown in the first image here.
- i** If your extrude in this step does not seem to be working check that you definitely removed the two small arcs from the previous step and left the two cords intact.

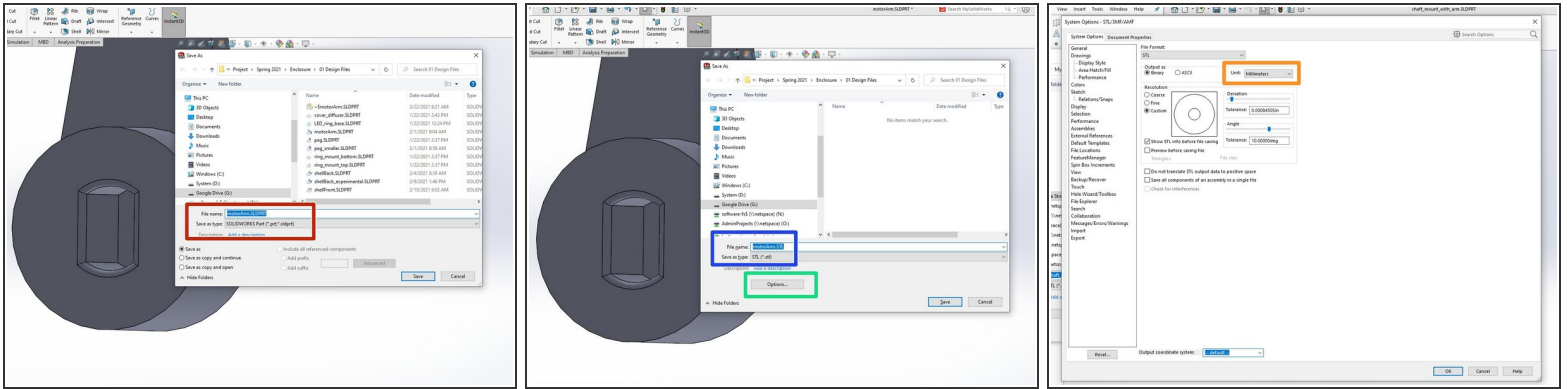
 - Within the **Features** tab choose the **Extruded Boss/Base**.
 - Enter a value of 3 mm for the height of the extrude.

Step 13 — Add a fillet



- Rotate the part so that you can see the shaft slot in the bottom face. On the **Features** tab click the **Fillet** button.
- Enter a radius value of 0.4 mm. Then click the four edges of the motor shaft slot. You should see a ribbed preview appear in yellow. Click the green check mark to complete the fillet.
- ⓘ This fillet will gently round the edges of the motor shaft slot to make it easier to attach the arm.

Step 14 — Save your part as .sldprt and .STL



- Save your part as a .sldprt file so that you can modify it later if necessary.
- Also save your part as a .STL file. You will need the .STL file to 3D print. Note that you cannot modify an .STL file.
- Be sure to click the **Options** button before saving the .STL file.
- Because many 3D printer slicing applications operate in metric units, you should confirm that the **Unit** parameter of your STL file is set to **Millimeters**.
- Click **OK** and then save your .STL file.
- SolidWorks will pop up a window telling you how many triangles your part contains. Click **Yes** to continue saving.

Congratulations! You now know enough SolidWorks to be dangerous. YouTube and the builtin SolidWorks tutorials are your friends. SolidWorks is a powerful, full featured tool. Anything you want to accomplish has already been done. Use an internet search to find a video detailing the steps necessary to accomplish anything more complex than what is provided in this tutorial.