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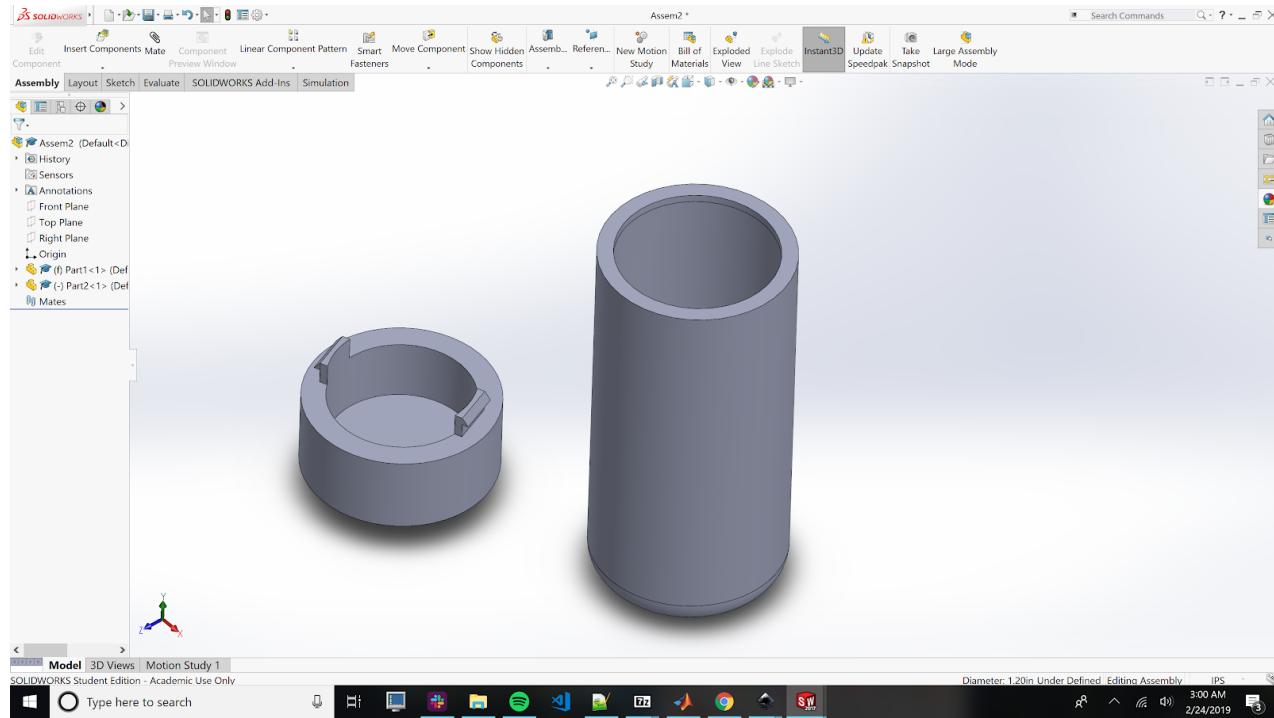
# SolidWorks

## Overview

In this workshop you will learn the basics of Solidworks, including Extrudes, Cuts, Revolves, Fillets, Shells, and Assemblies. You can put these basics to test to make a canister that is a perfect storage container capable of fitting an Arduino nano, and accelerometer, a battery, and a few other sensors that can close with a snap-fit cap!

## Prerequisites

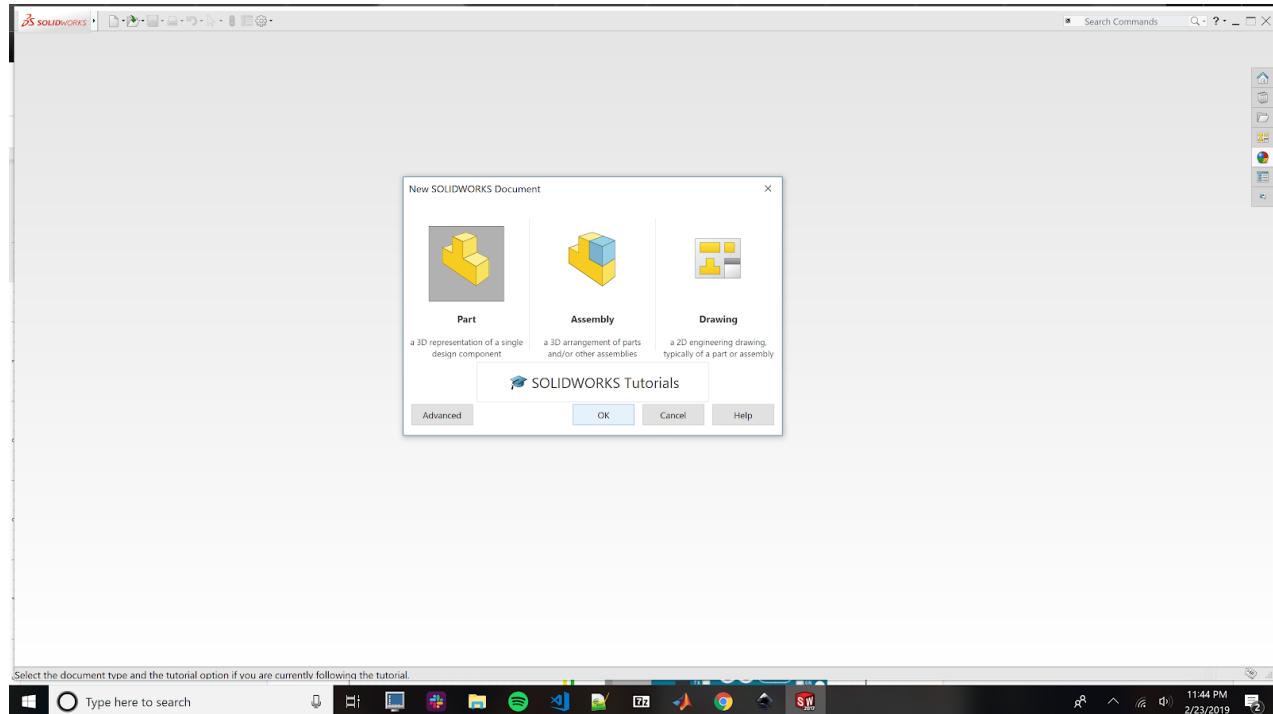
- Windows Computer with SolidWorks installed
- Computer with the UCSD Virtual Lab setup



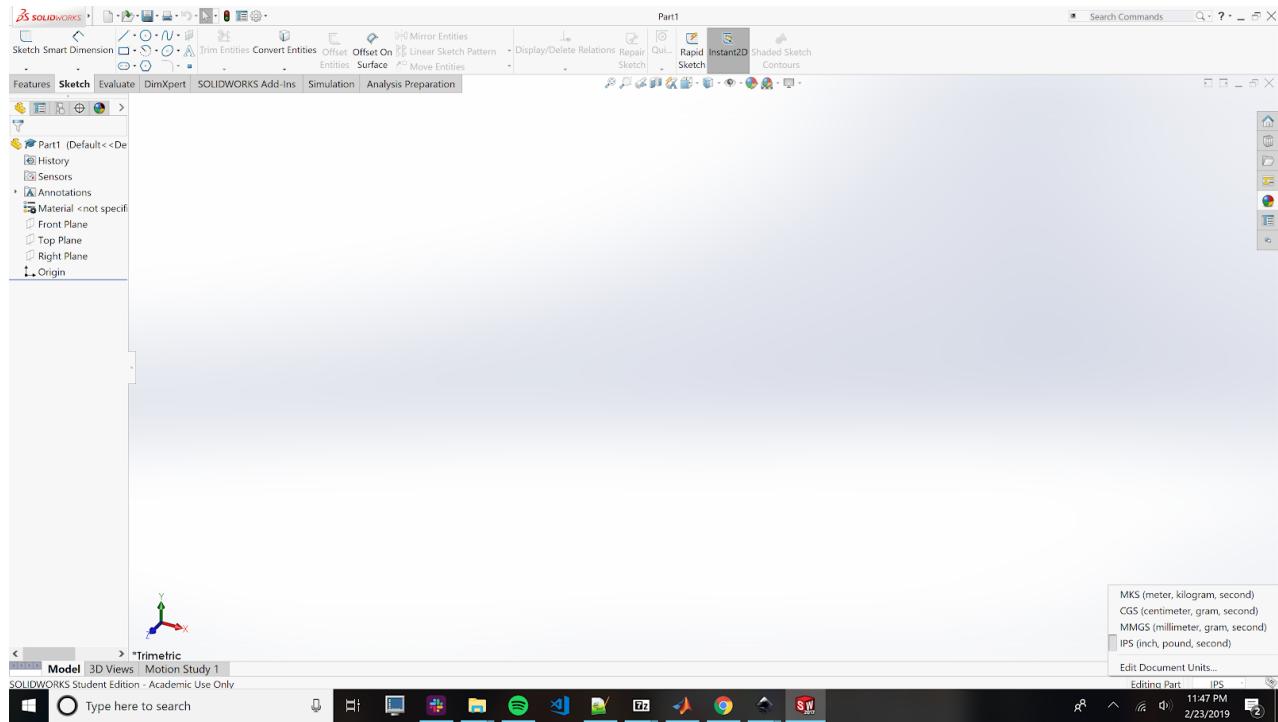
The Final Product

# Starting your first 3D Model – Sketches and Dimensions

When you open up Solidworks this screen should pop up. Click that you are creating a part, not an assembly (at least not just yet!).



Before you even change anything in your design, make sure to change the units to whatever units your design is dependent on. Make sure to change this now, as to not cause any confusion (see Mars Climate Orbiter).

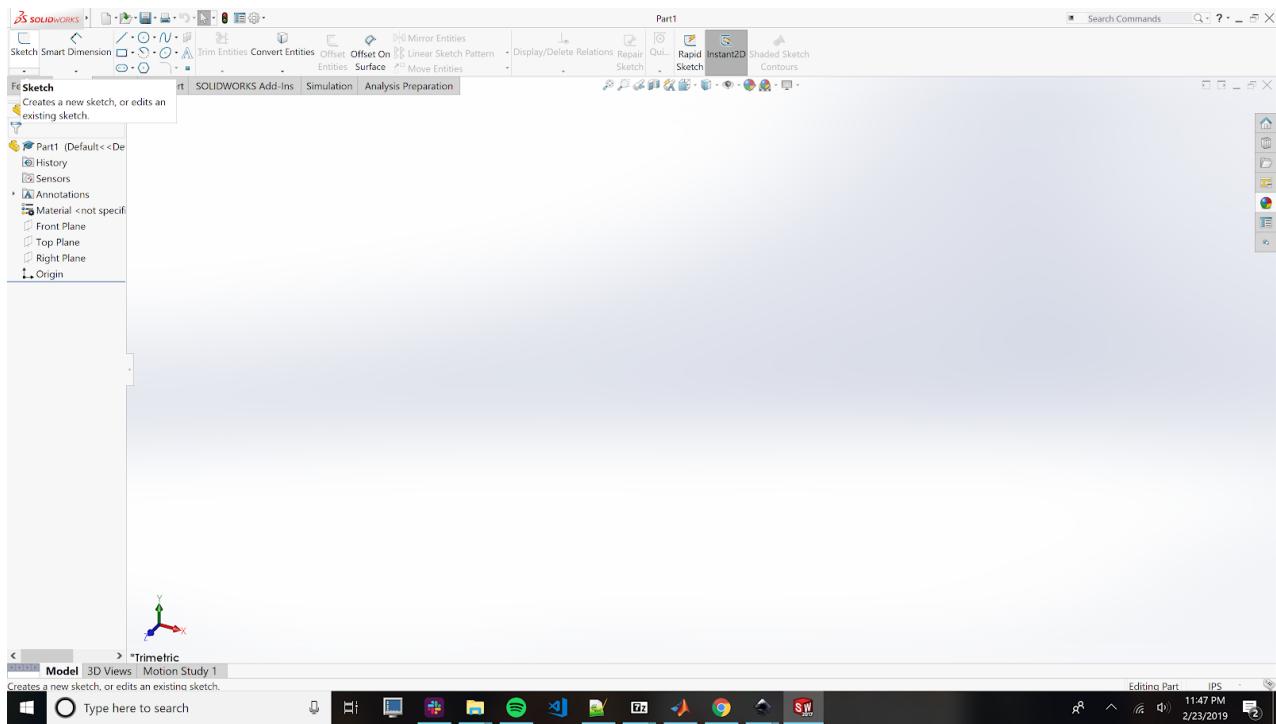


Just some quick tips to help you navigate around if you have a mouse:

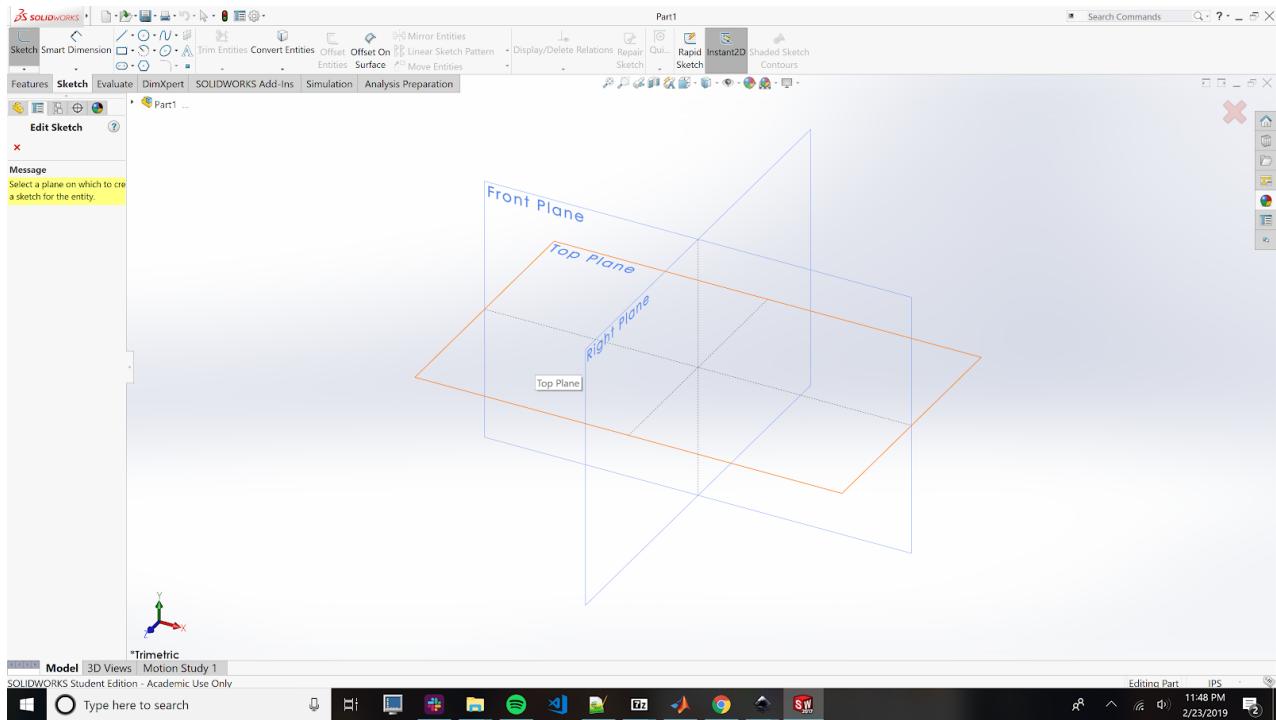
You can change the orientation of the camera by holding down middle click on your mouse and dragging around.

You can also use the scroll wheel to zoom in and out on your mouse to get a better view of what you are trying to design

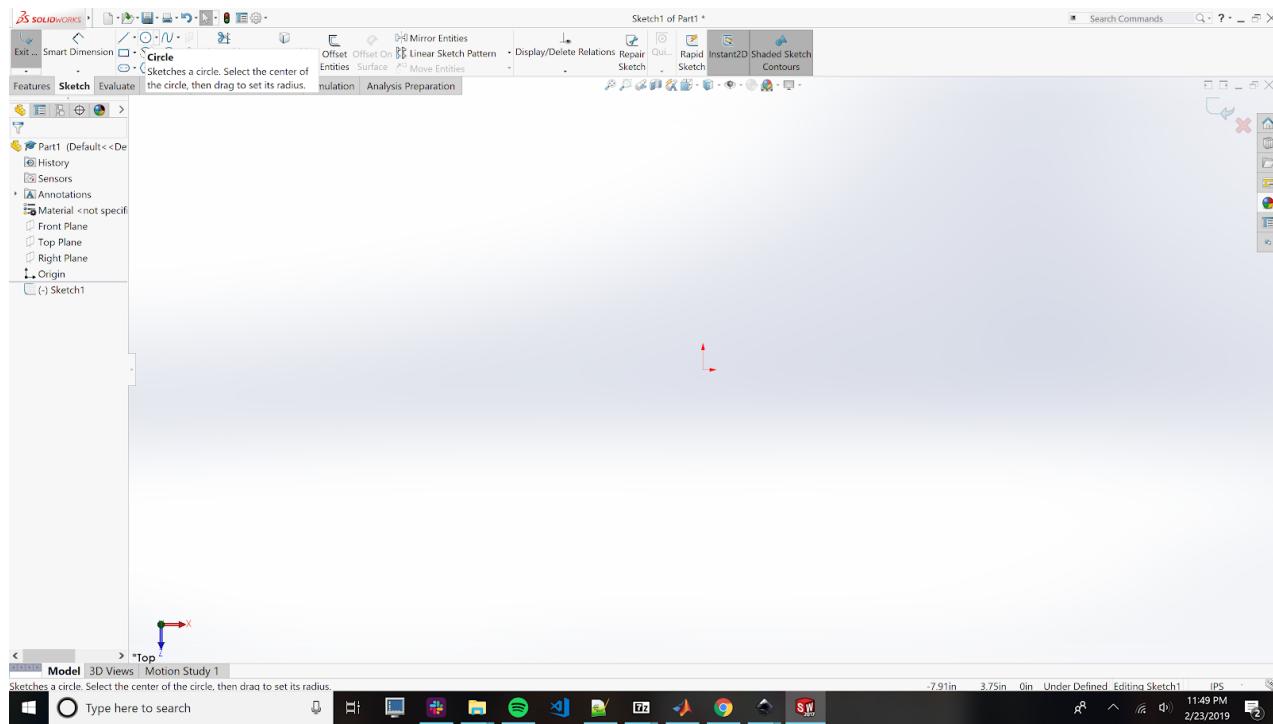
The sketch will be the first part of your CAD design. A sketch is a 2D drawing which you can extend out to create a 3D object, or even make a sketch to cut out of an already existing object! Click this in the upper left hand corner to create a sketch



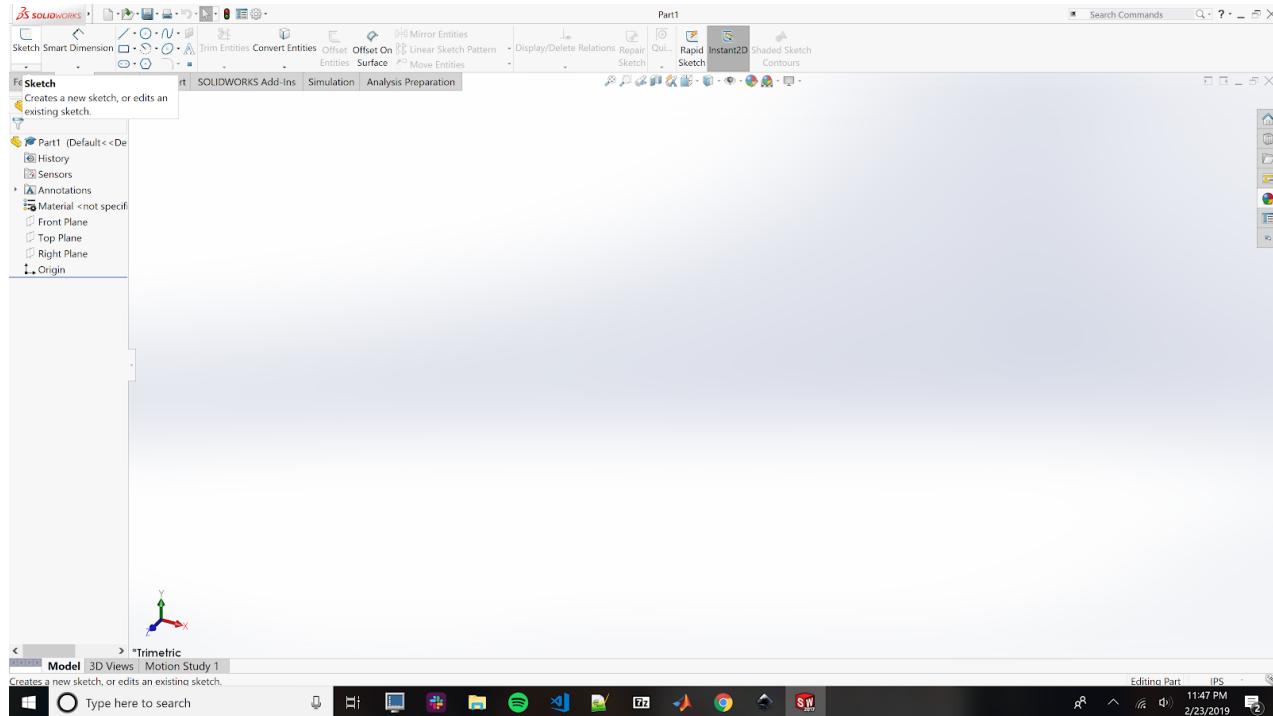
Select which plane of orientation you want to make your sketch on:



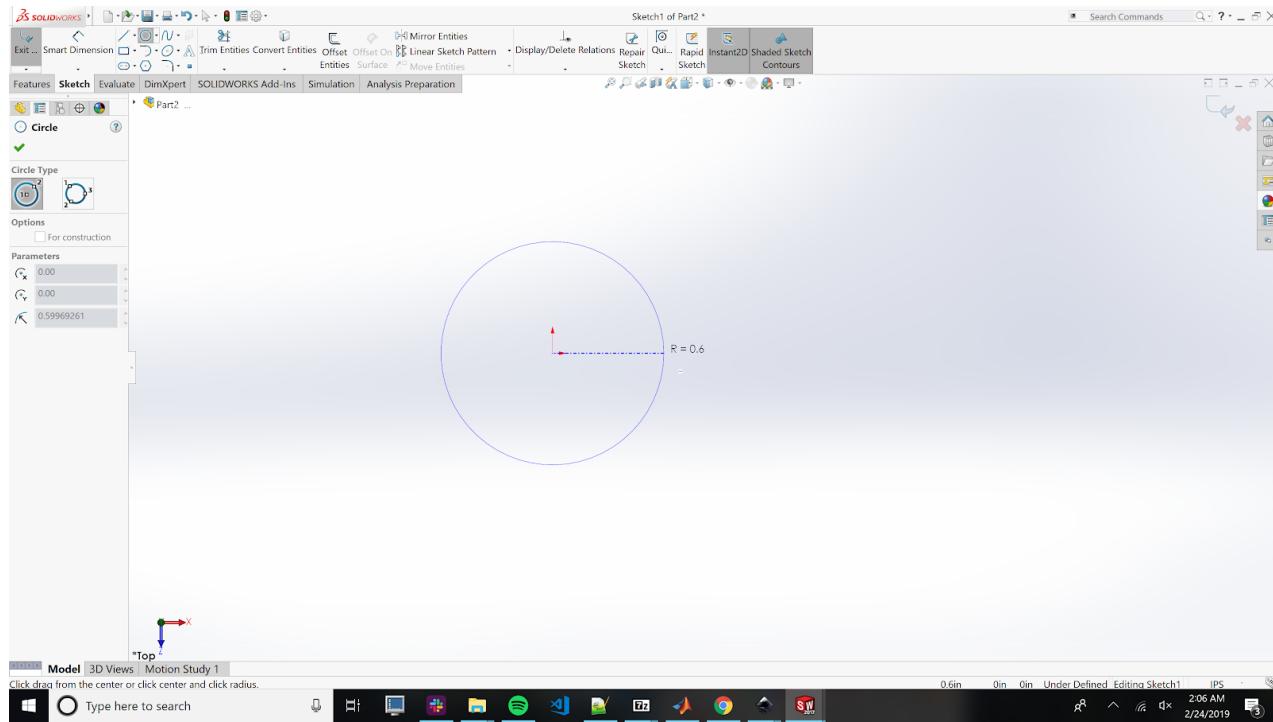
You should be able to start making a sketch from the origin, which will be your first point of reference in your design.



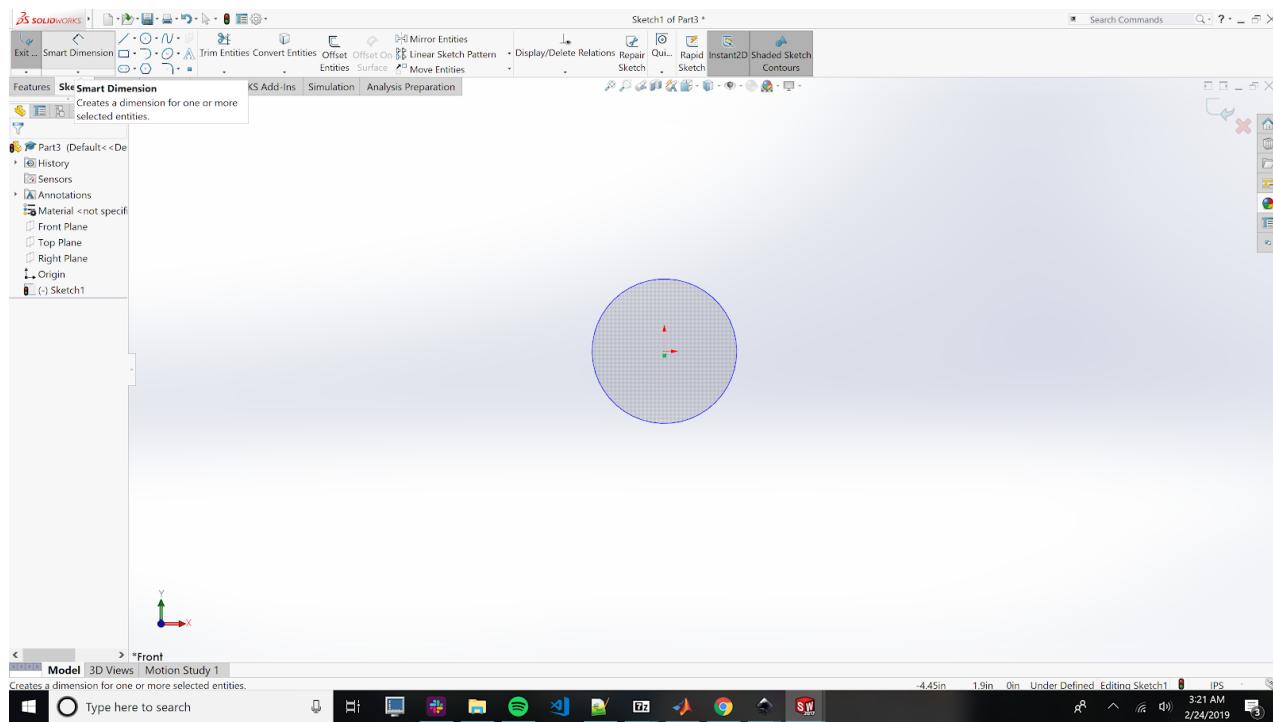
Here, you can start drawing, which you have many options for. As you can see below, you can draw lines, rectangles, circles, or any other shapes that you want with the splines or the other tools given.



Here, I drew a circle by clicking at the center of the origin and dragging outwards towards the approximate measurement I want.

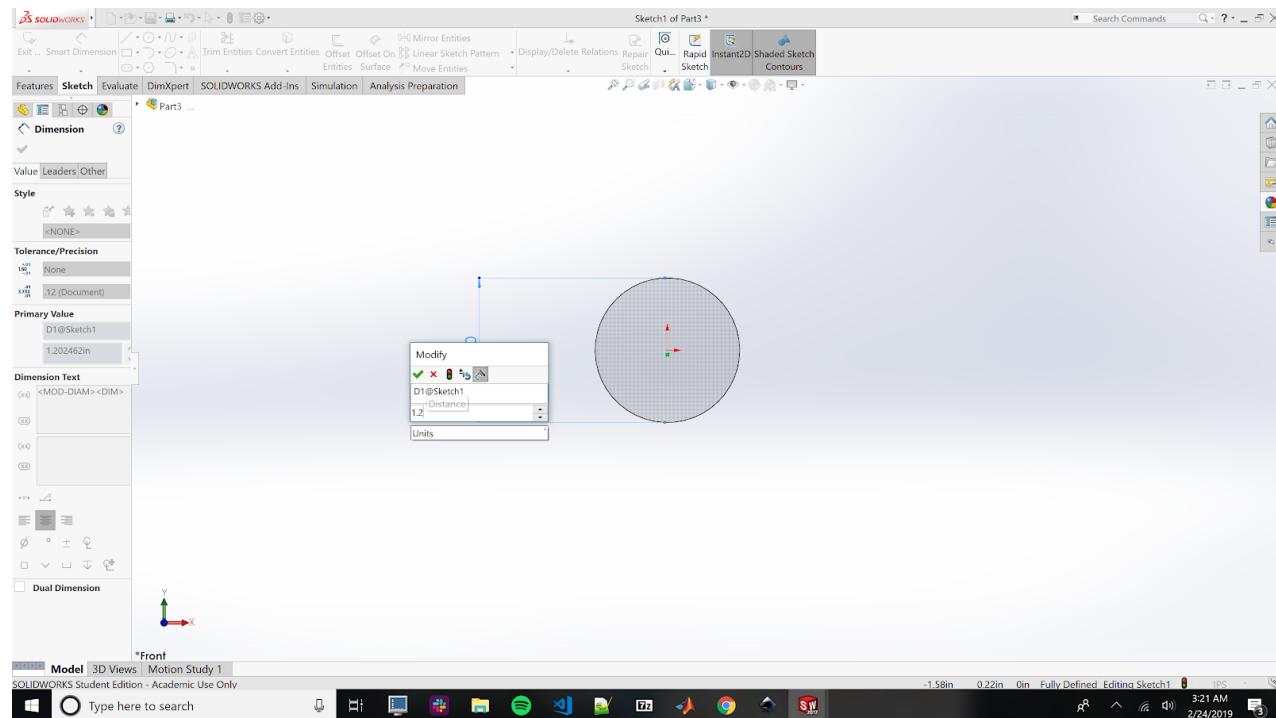


One important part of CAD design is creating dimensions. Creating dimensions allow you to define the size of a shape or line, and this size will not change when other aspects of your design change. You will see the use of this in the future, but be sure to give a dimension for every shape and line you create, this will help reduce errors in the future when you make changes to your design. To add a dimension click the smart dimension tool when you are in a sketch:



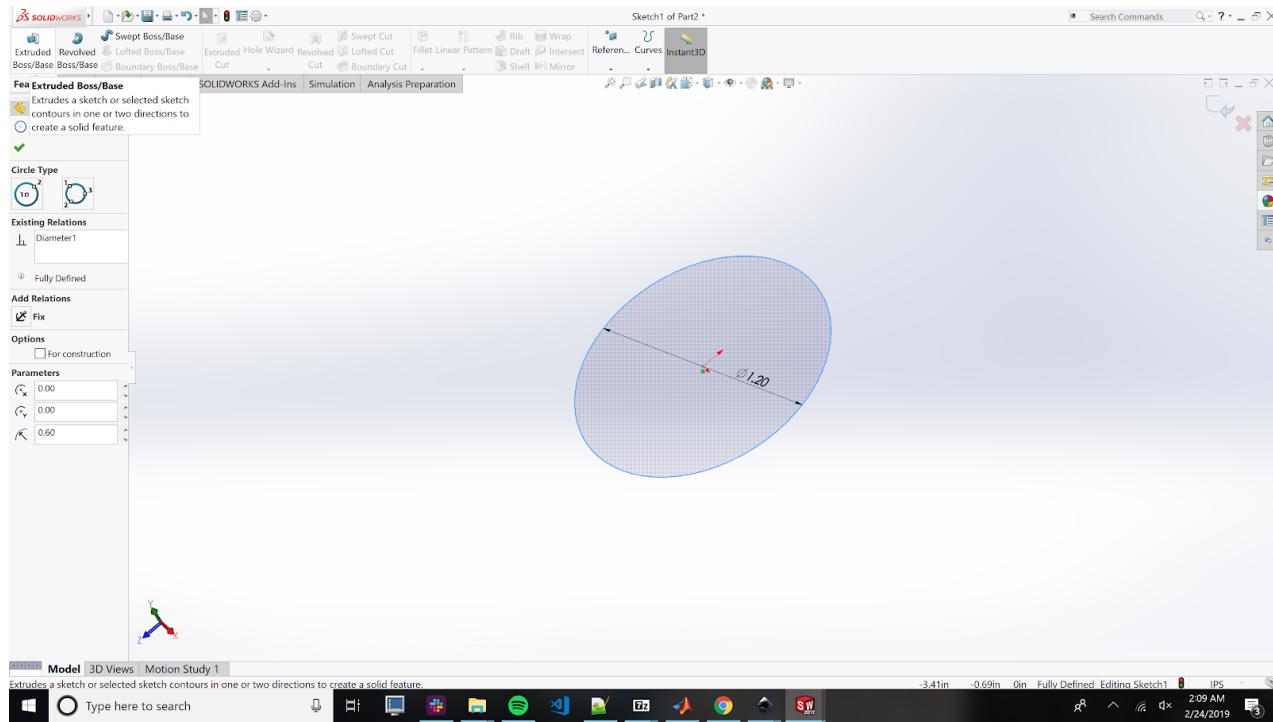
You can tell if a shape or line is defined if it is colored black. Sometimes when you create a shape or a line, it may even already be defined because you created the new shape in reference to another defined shape or line.

Here, we defined the diameter of the circle we created in the past step to be 1.2 inches by selecting both ends of the circle in the smart dimension tool.

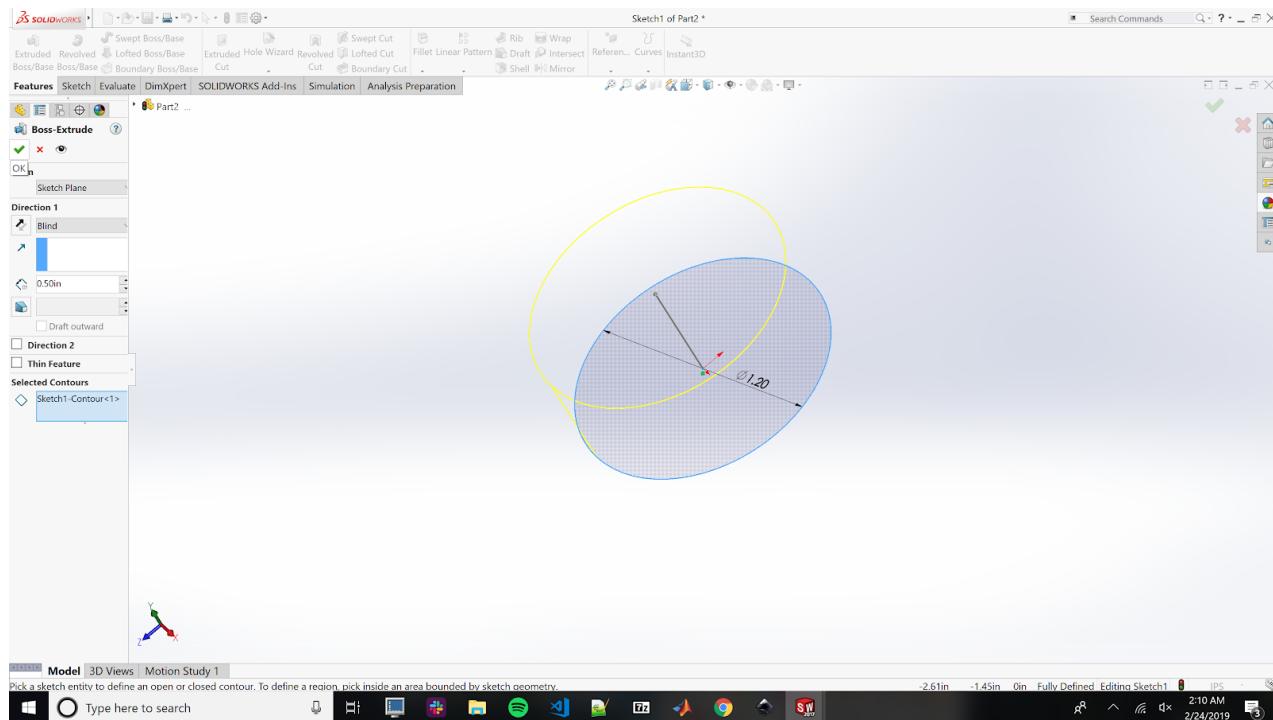


## Extruding and Cutting – The Cap

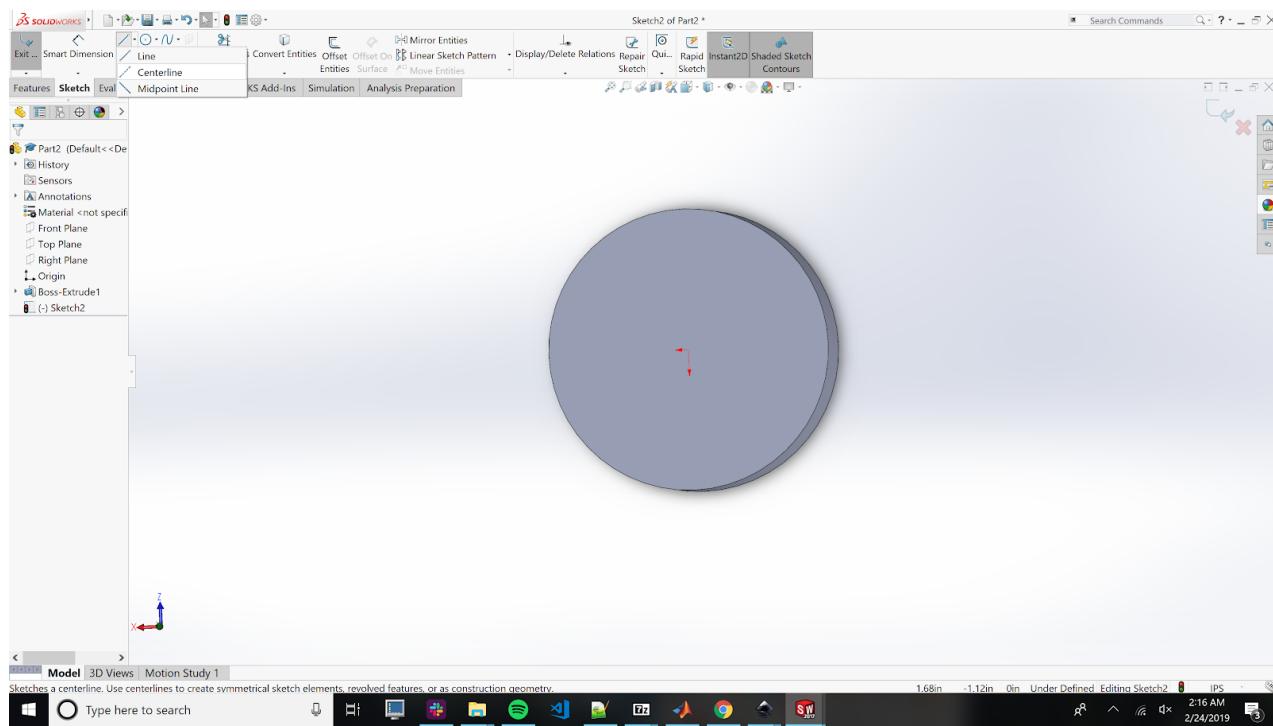
Now that you have made this 2D sketch, you can turn this 2D sketch into a 3D object by extruding your 2D sketch!



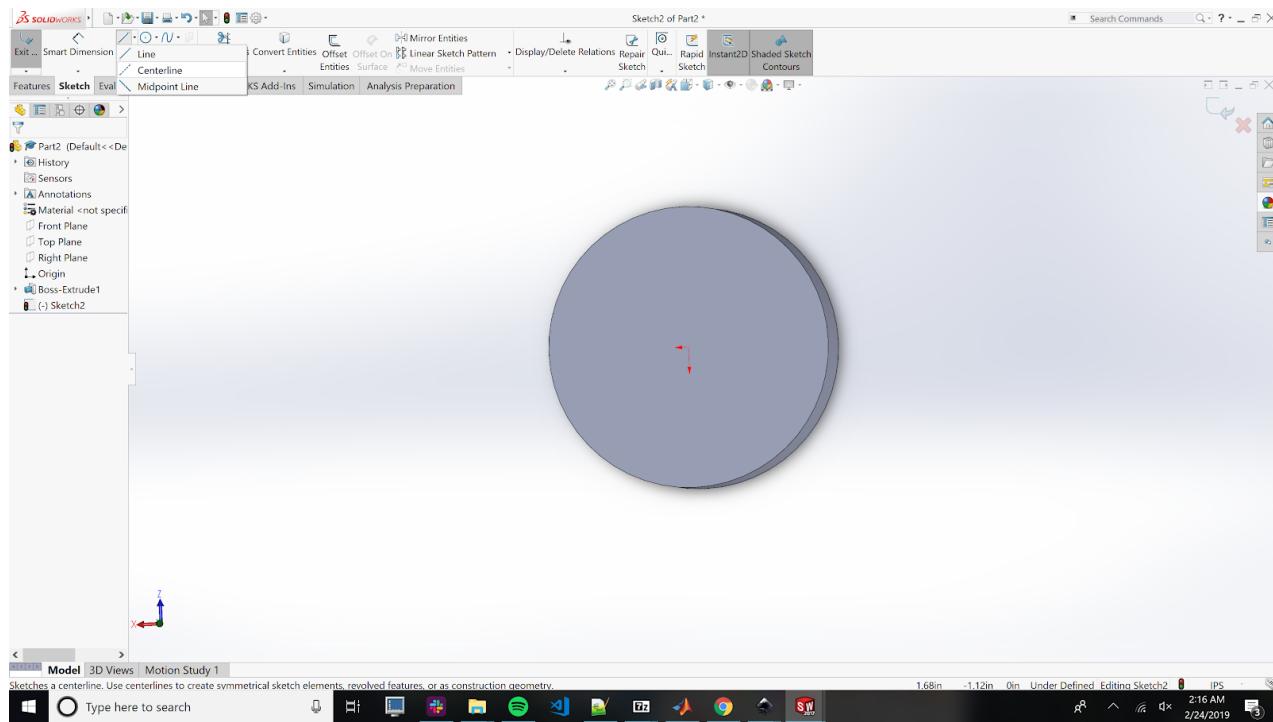
You can use the extrude tool to select a face, and extend it outwards to create a solid object. Here, I selected the sketch that we just made, and extended it by .5 inches. You can select the direction by either using the Direction option or adjusting the arrow showing the direction of the extrusion.



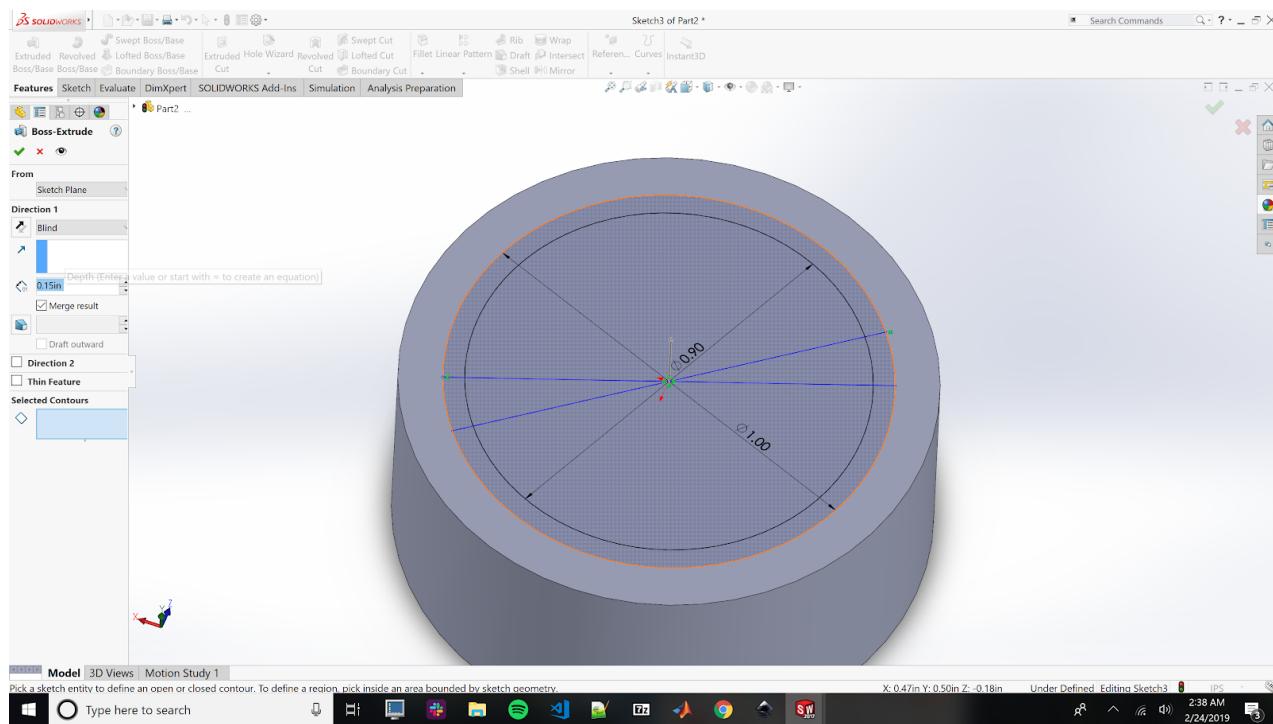
Here is the extruded object that we just created, a cylinder



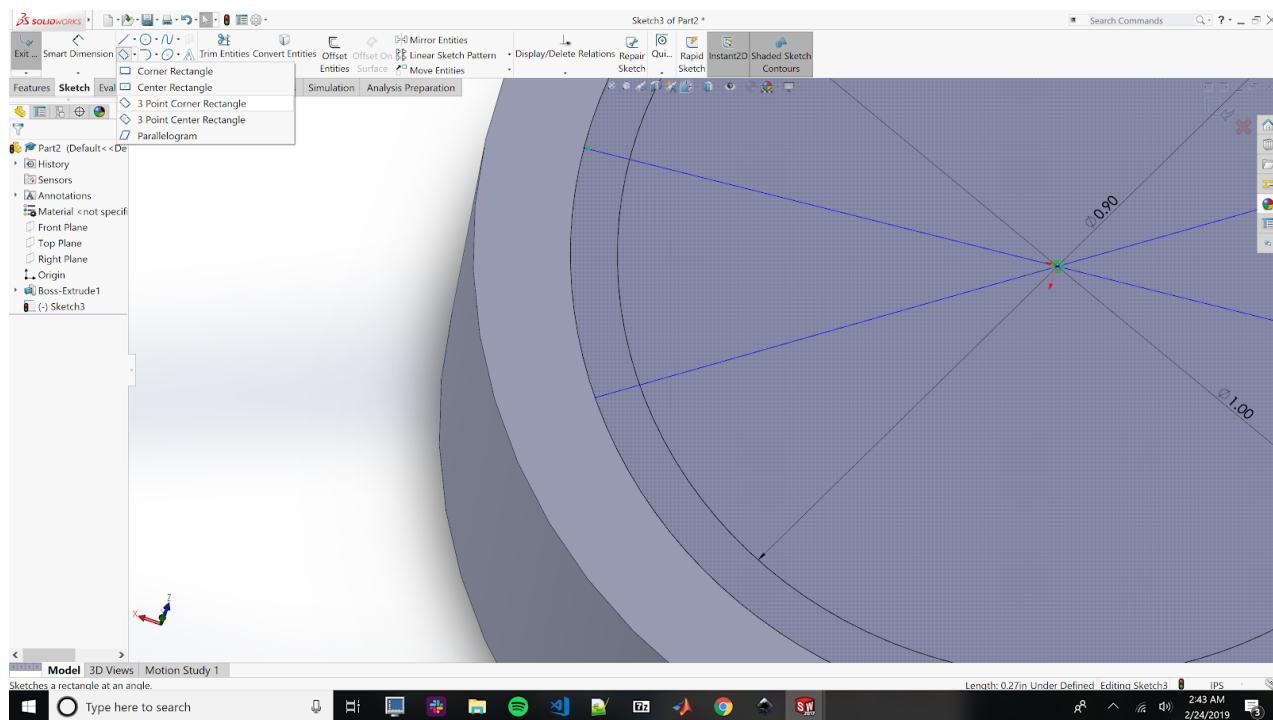
Here's another useful tip: Use centerlines to create reference lines that won't make shapes in your sketches!

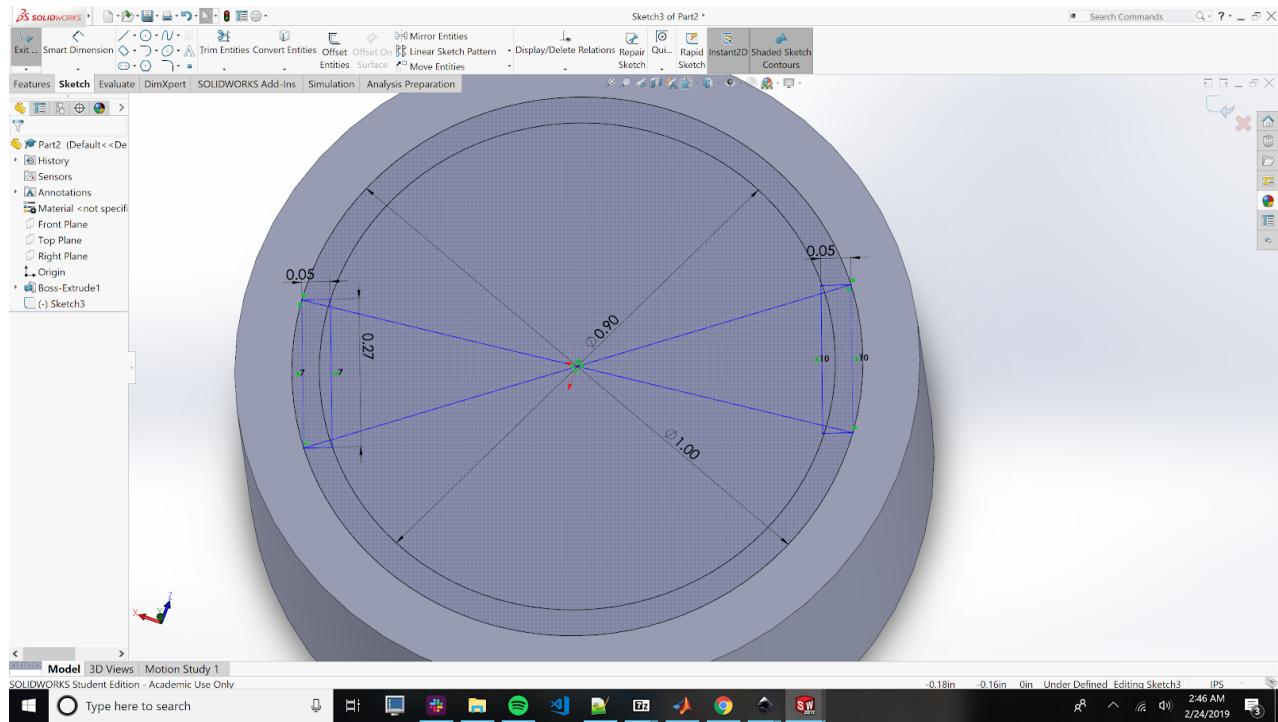


Next, we are going to make the snap-fit portion of our cap. Here I used 2 concentric circles from the center of the original object. I then took these concentric circles to create the clips that would clip onto the body. The ends of these circles will act as the extents for the clips!



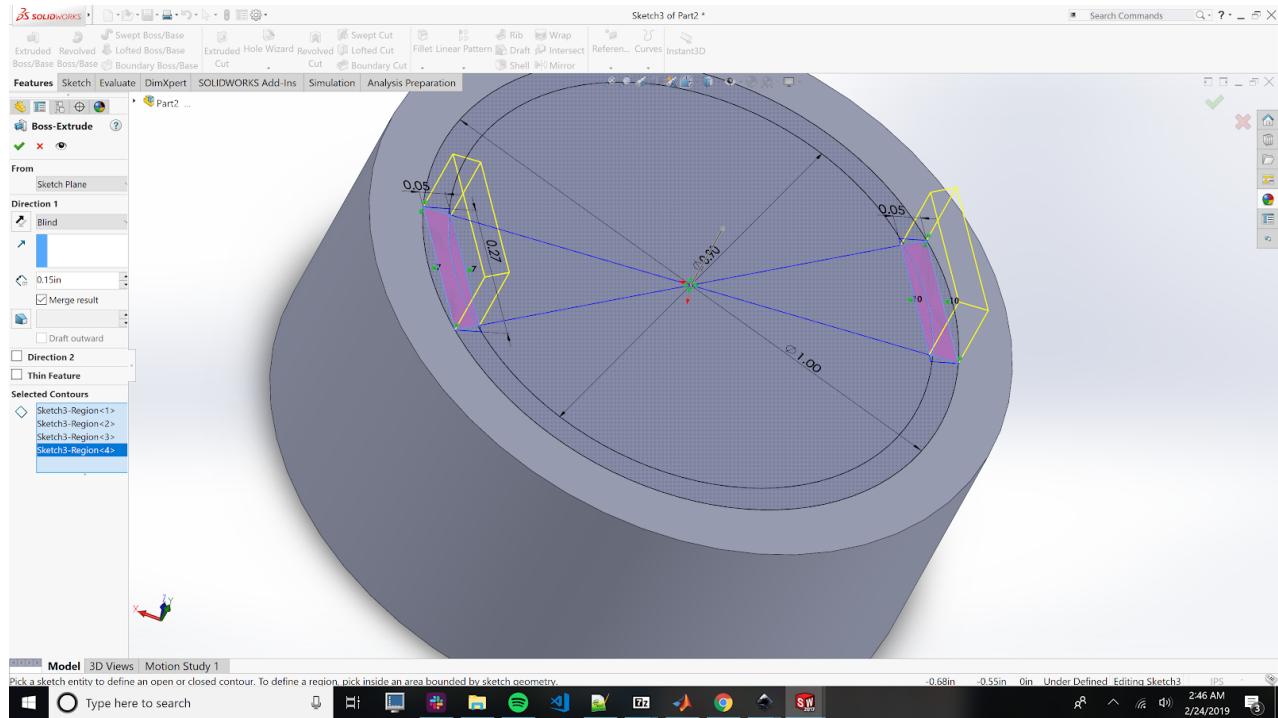
I can then use the 3 Point Corner Rectangle to define where I will extrude the clip, which are the ends of the circles.



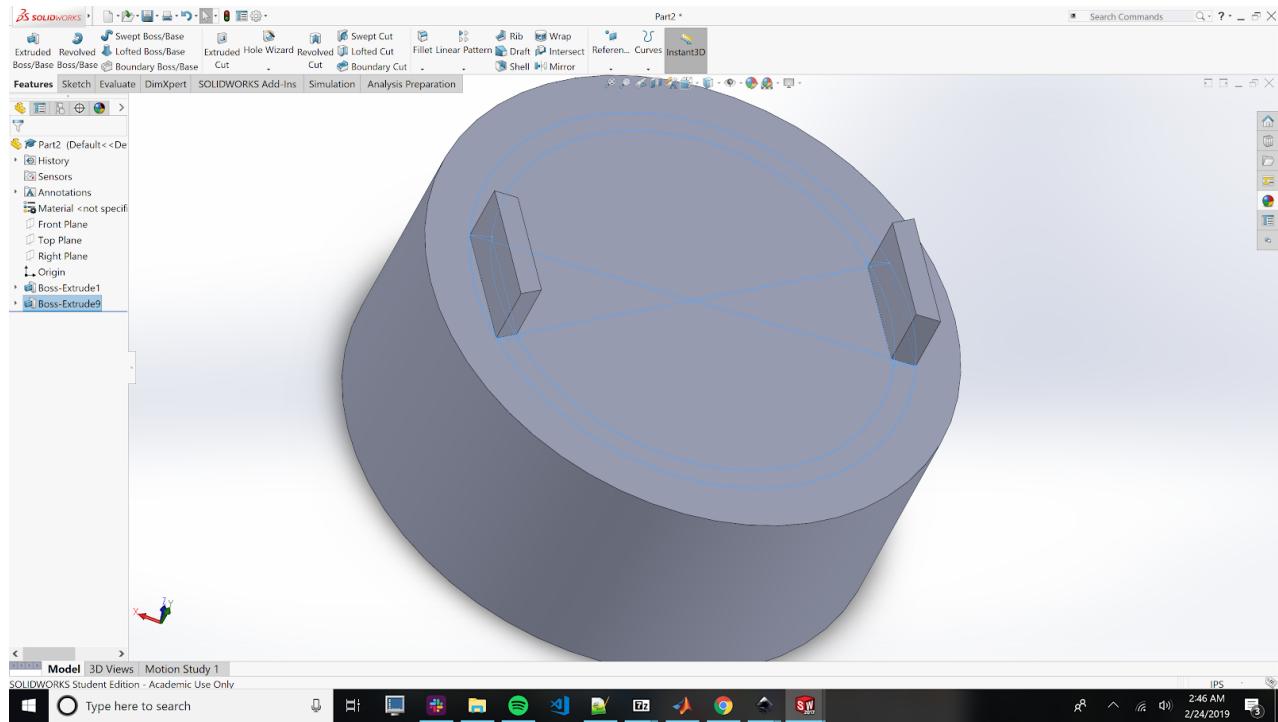


Here we can make the rectangles for which our base of our clips will be extruded from

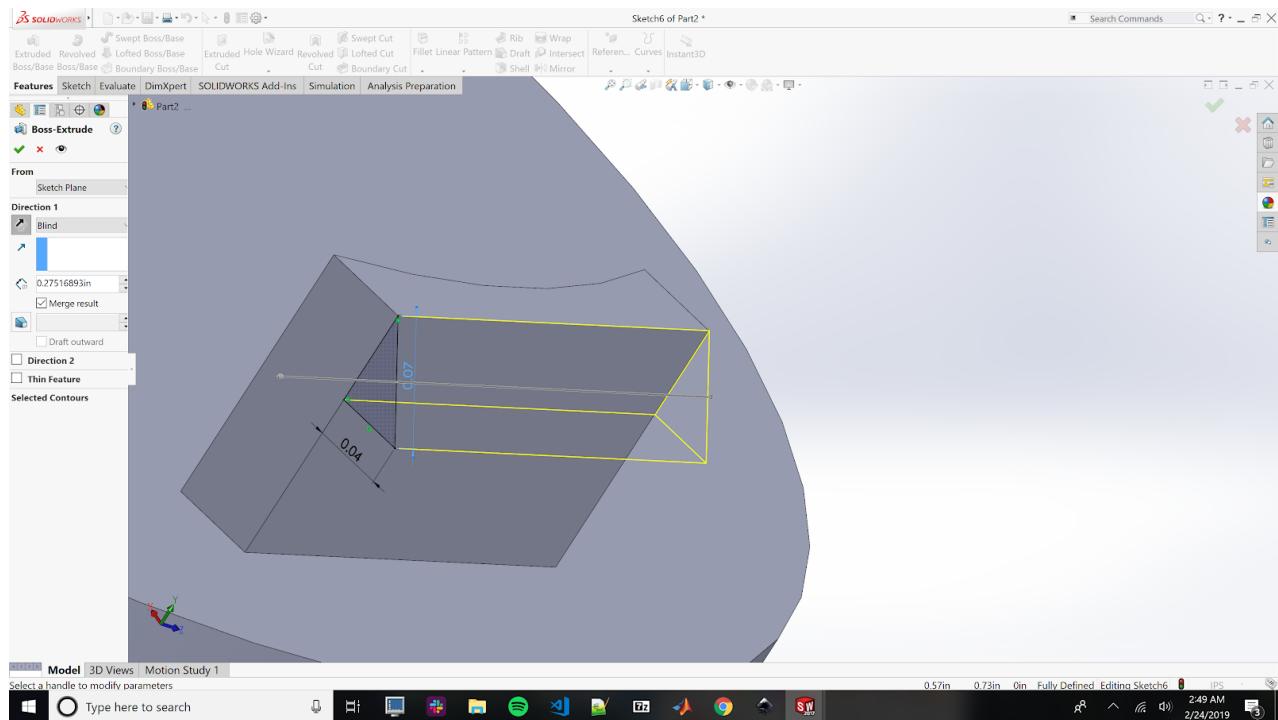
We can then extrude the base of these clips to attach the snap portion of the clips by again clicking on the polygons of our sketch and extruding them.



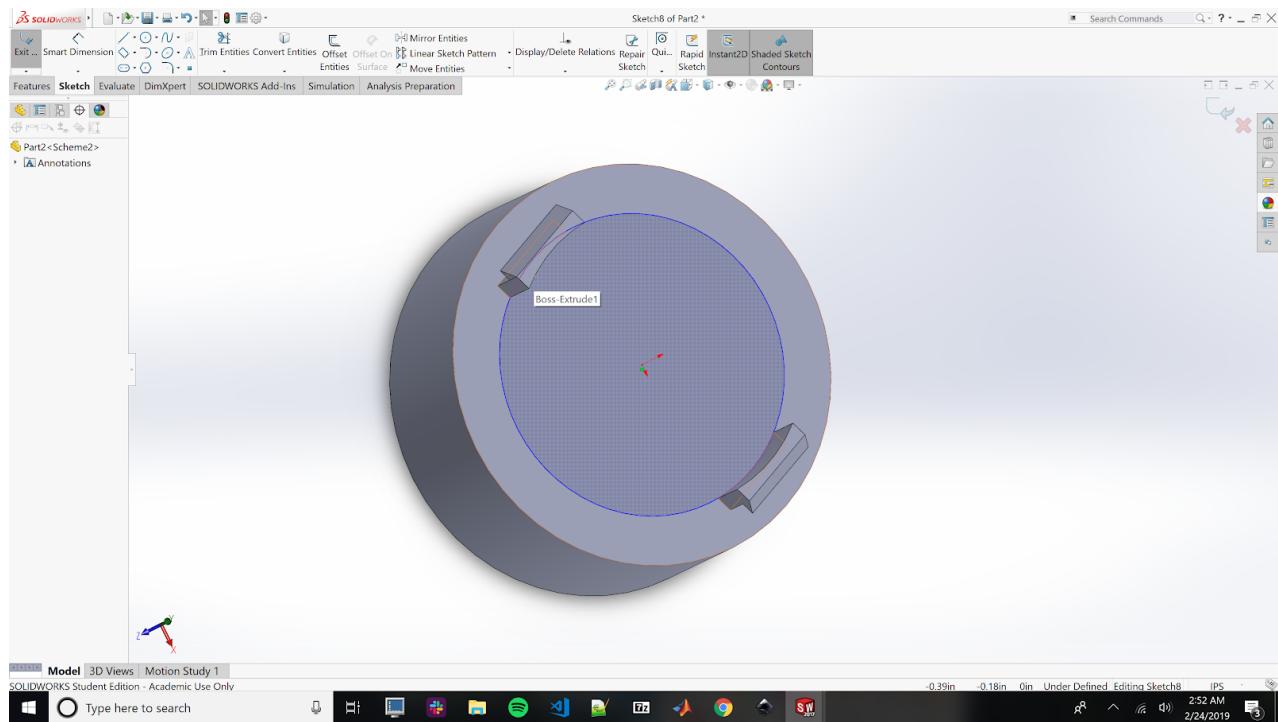
Here is the result of our extruded clip bases:



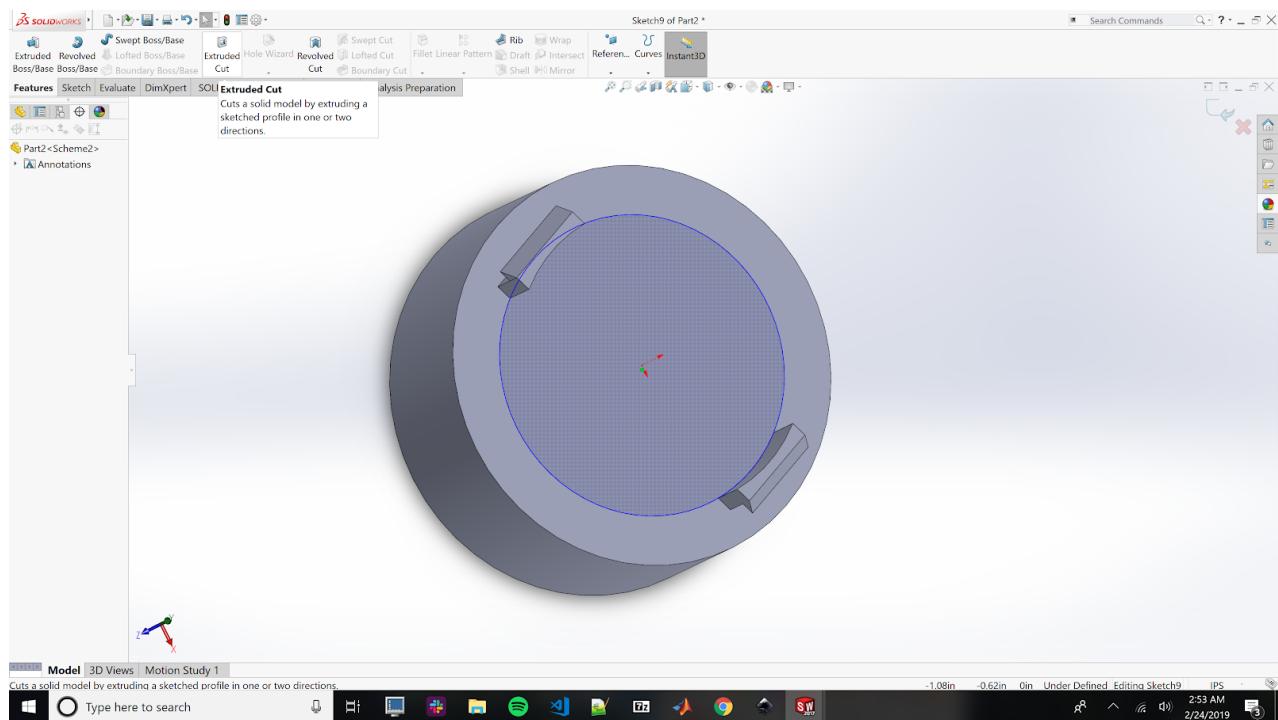
We can now make the clip portions of the cap, by extruding a triangle out from the sides of the clip base



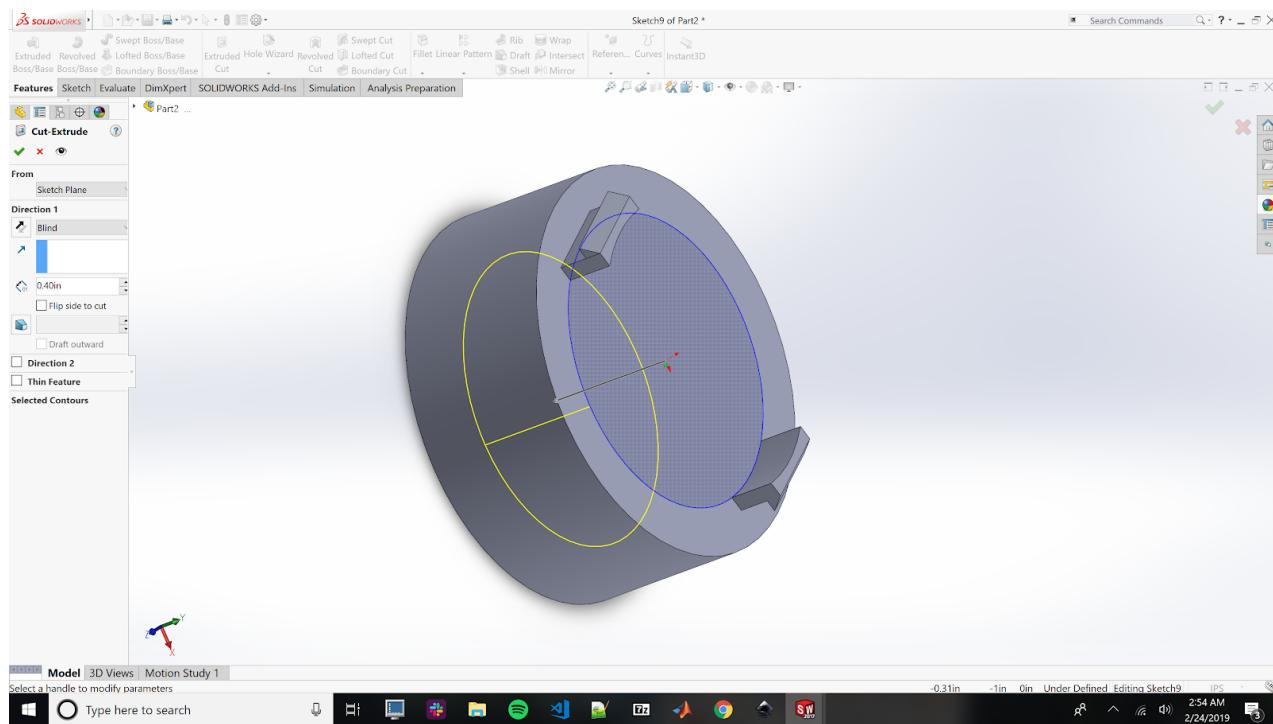
Now you may be wondering, gee that is a lot of wasted material. Are we just going to print that whole cap? No, that would be dumb. Let's get rid of all of this additional material by using the extruded cut tool. Here I made a circle outline the material that I will cut out of the cap.



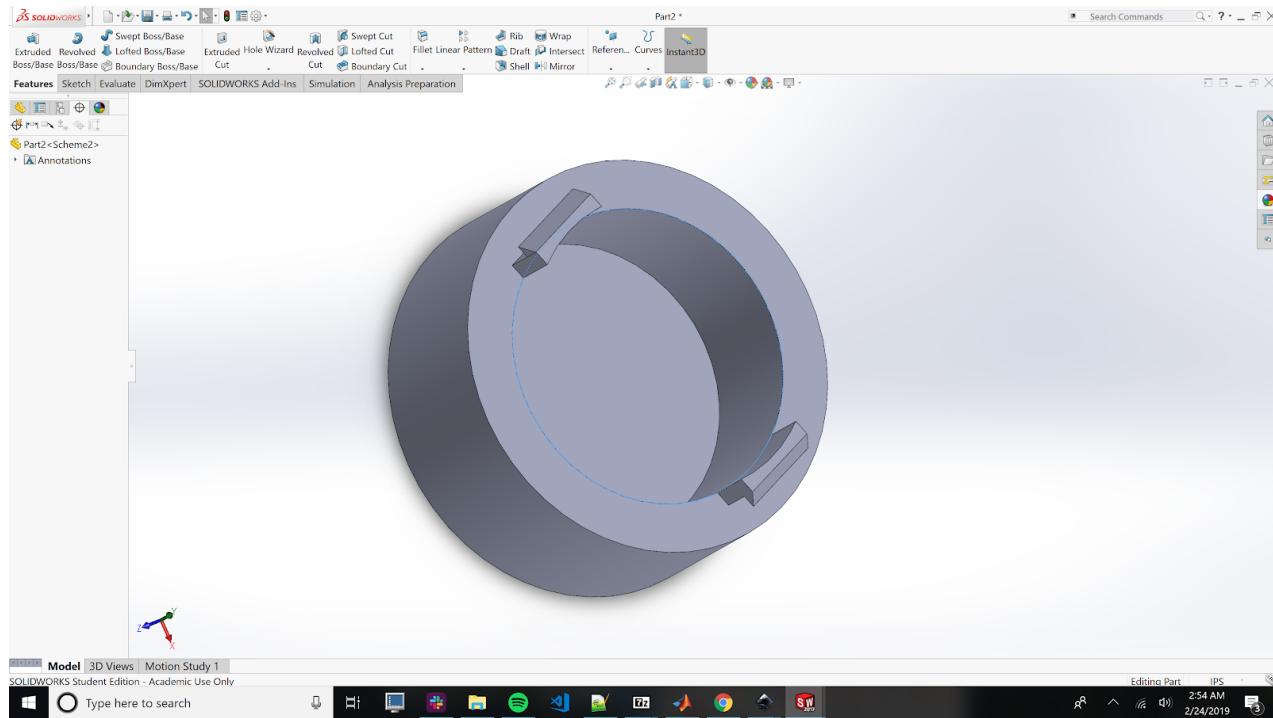
We can then use the Extrude Cut tool to clear this all out



With basically the same steps as the extrude, we can finish our cut, just in the opposite direction of our original extrusion.



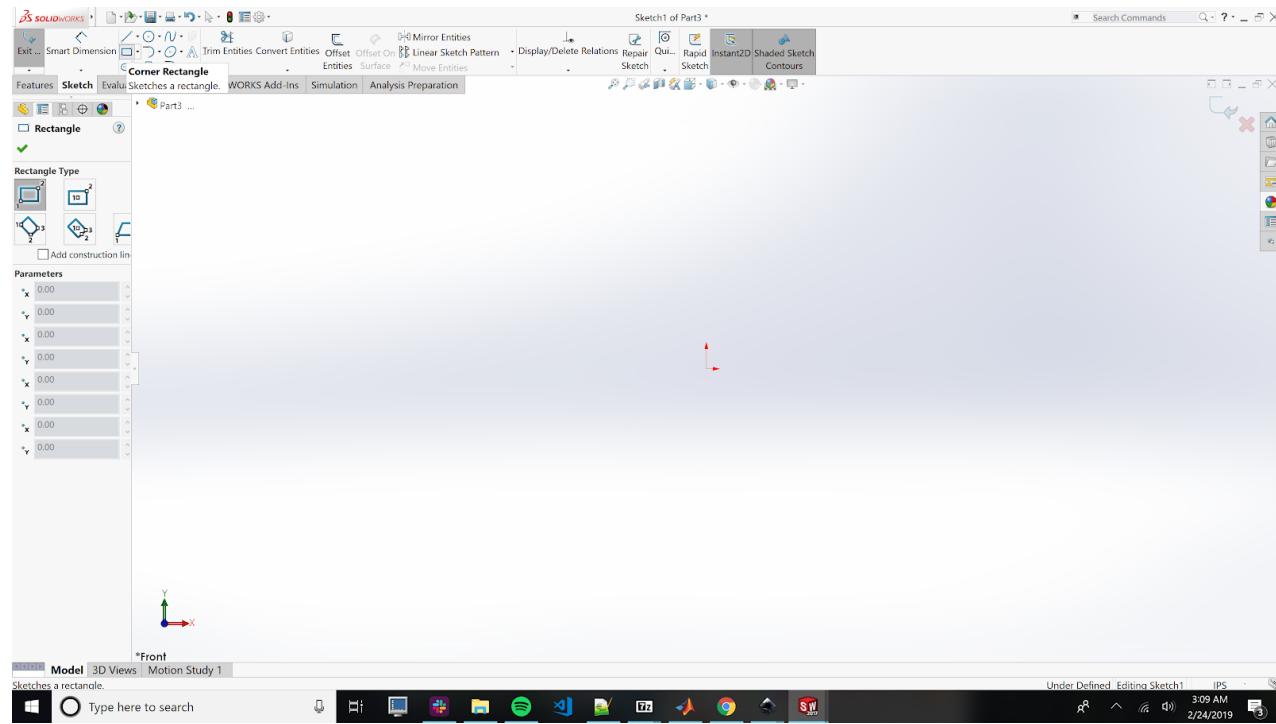
Here is the final product of our cap!



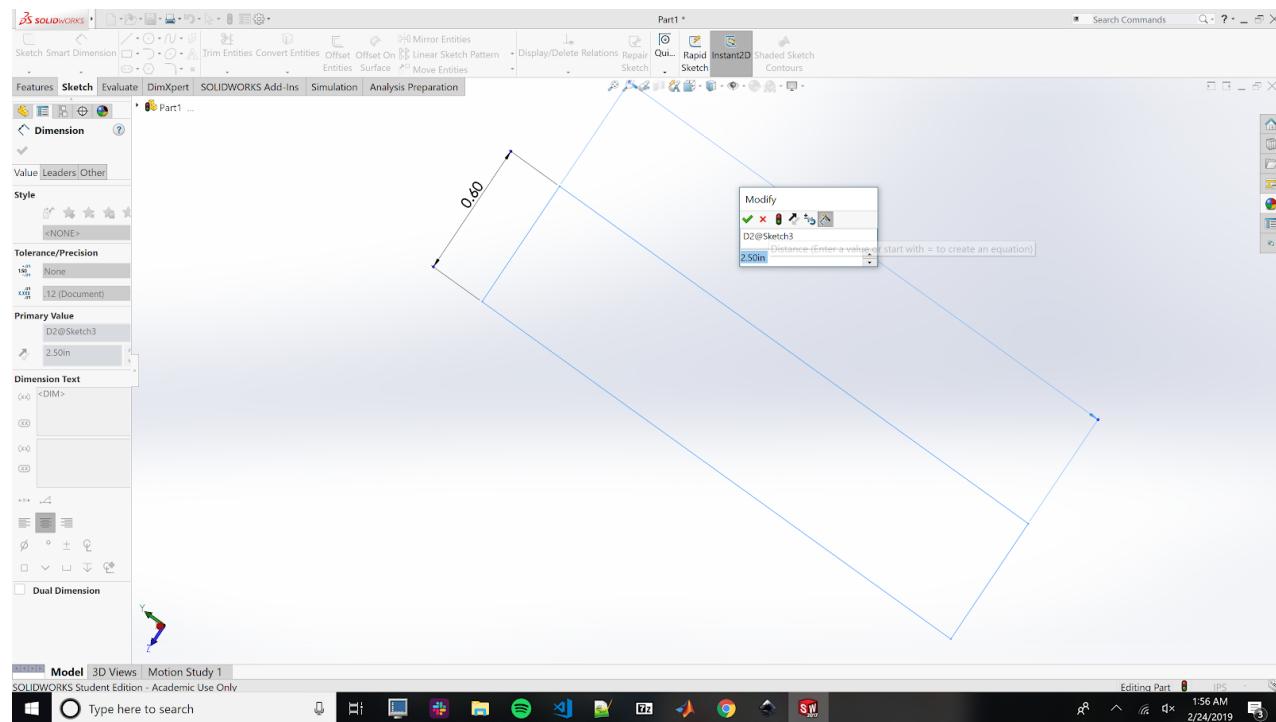
## Revolving, Fillets, and Shelling

The only way to create a 3D object isn't just extruding. We can also use the revolve tool to rotate a plane around a reference to create another 3D object. Let's start this by

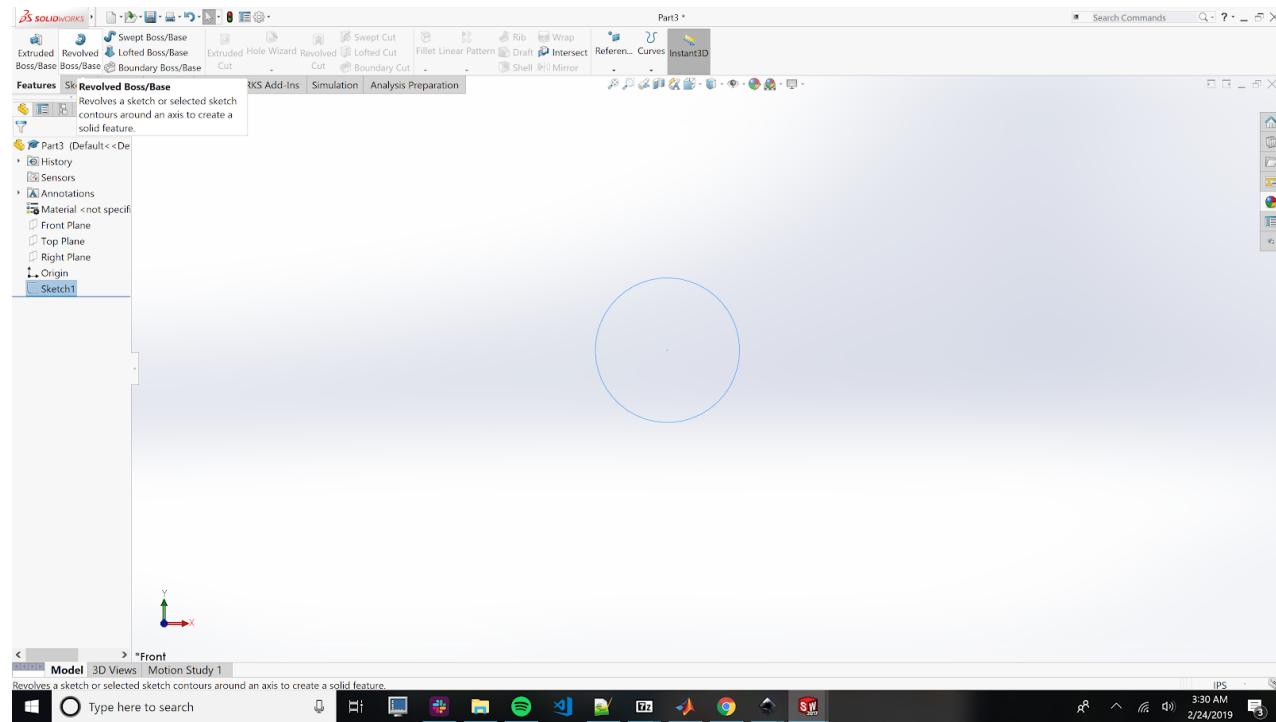
making a rectangle in a completely new part which we can revolve around one of its sides.



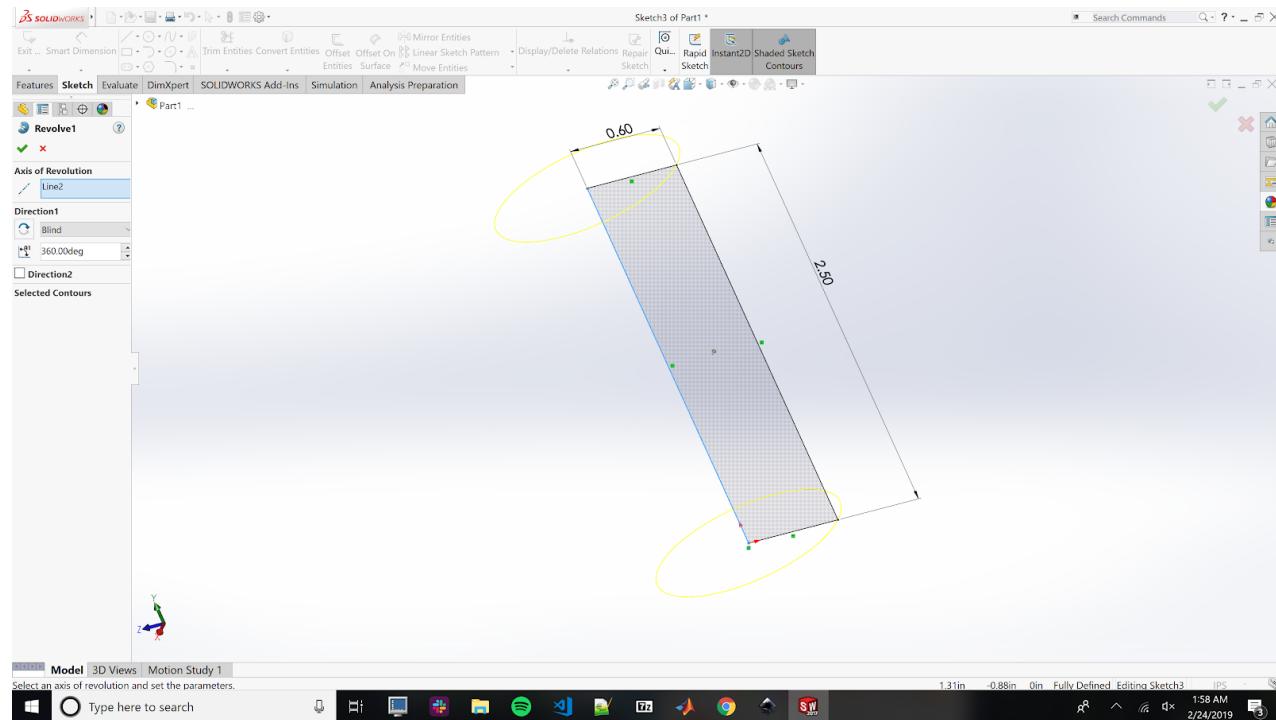
We have defined the rectangle as such in order to match the size of our cap when we revolve it. You will see why I chose these measurements next.



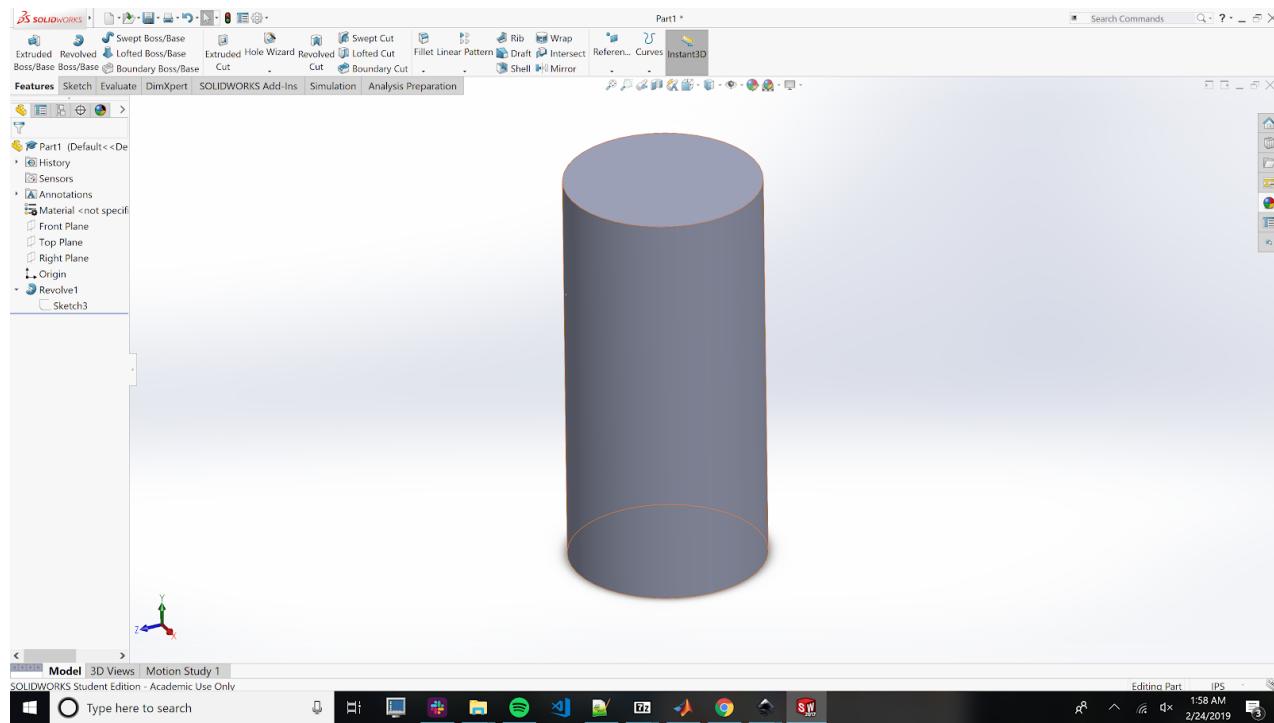
You can select the revolved tool right next to where the extruded tool is in order to revolve it.



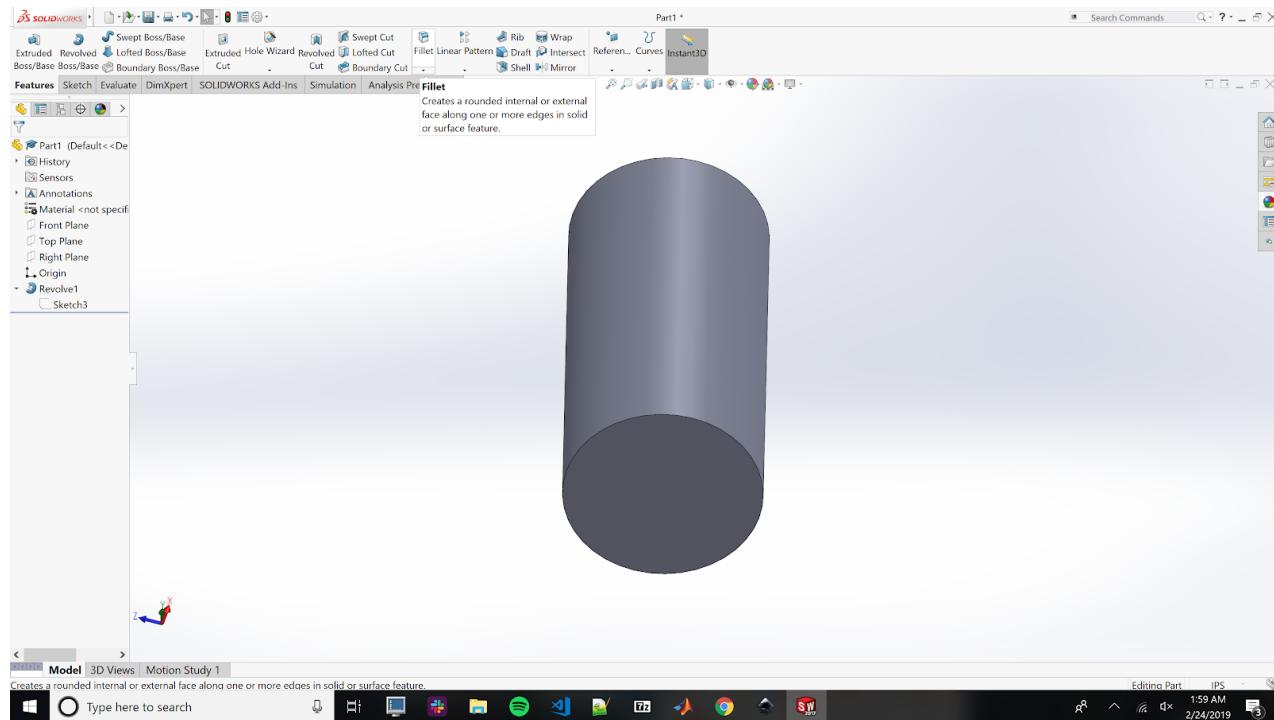
Here I clicked the polygon that we wanted to revolve, the rectangle that we just made, and then selected the axis of revolution as its long axis. This is another way to create a cylinder other than extruding a circle, instead you can just rotate a rectangle. I chose 0.60 inches, because this would be doubled after the revolve.



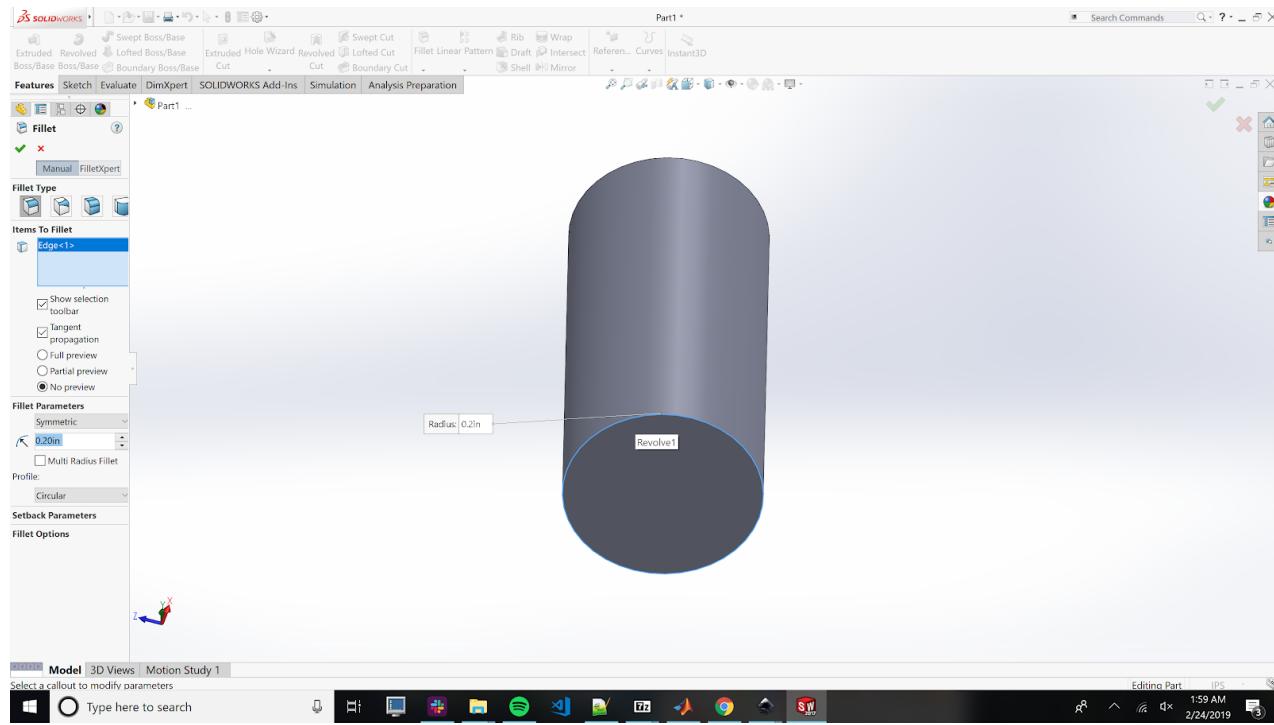
Here is the end result of our rotate, a cylinder



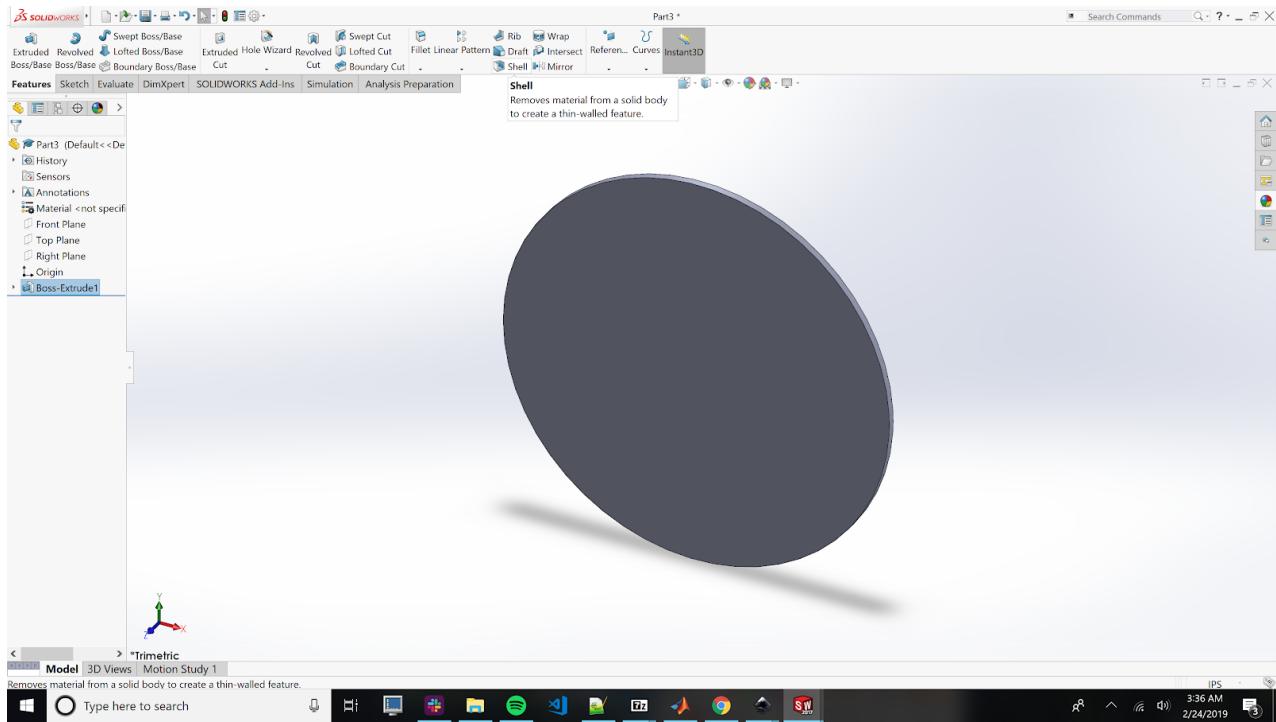
Now we can do some other transformations to our cylinder to make it more useful. Let's smooth the edges of our cylinder to make it easier and more comfortable to hold in a pocket with the fillet tool.



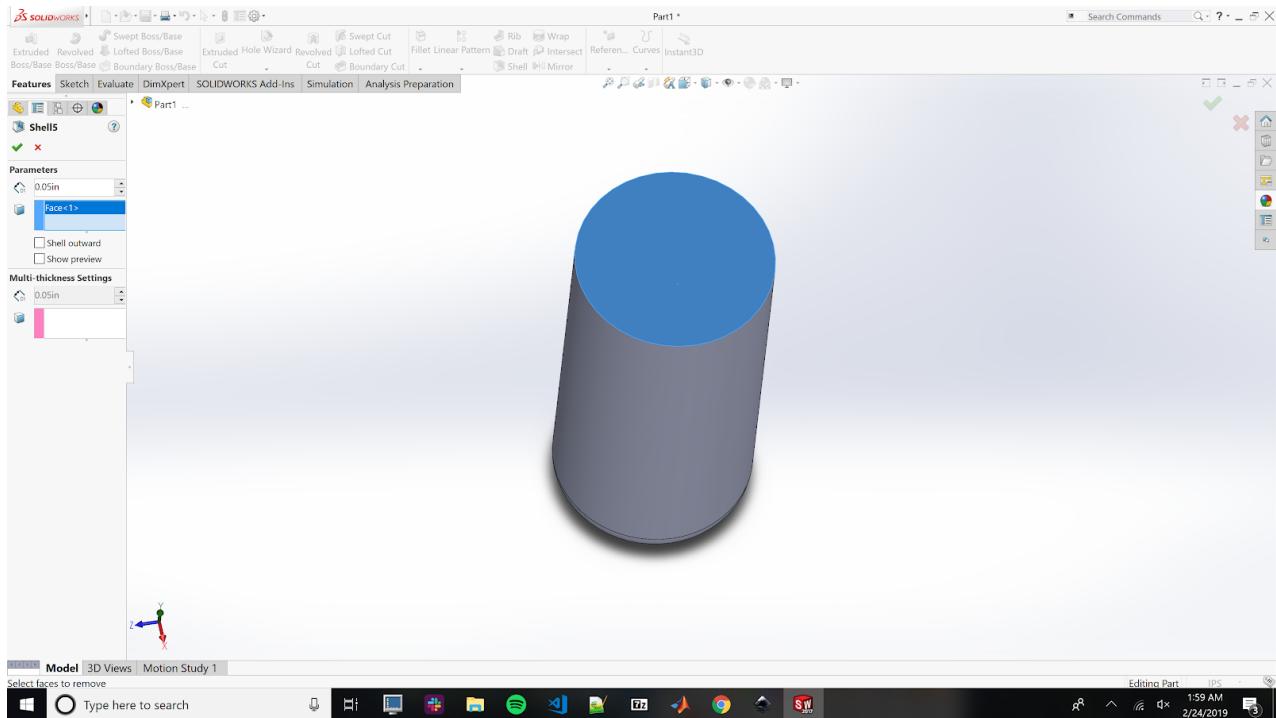
Open the fillet tool and then select the face or edge that you want to fillet. Let's fillet the very bottom of the cylinder to make it easier to put in pockets. I selected a 0.20 radius for the fillet, this should make a curve with a radius of 0.20 on the bottom edge of the cylinder.



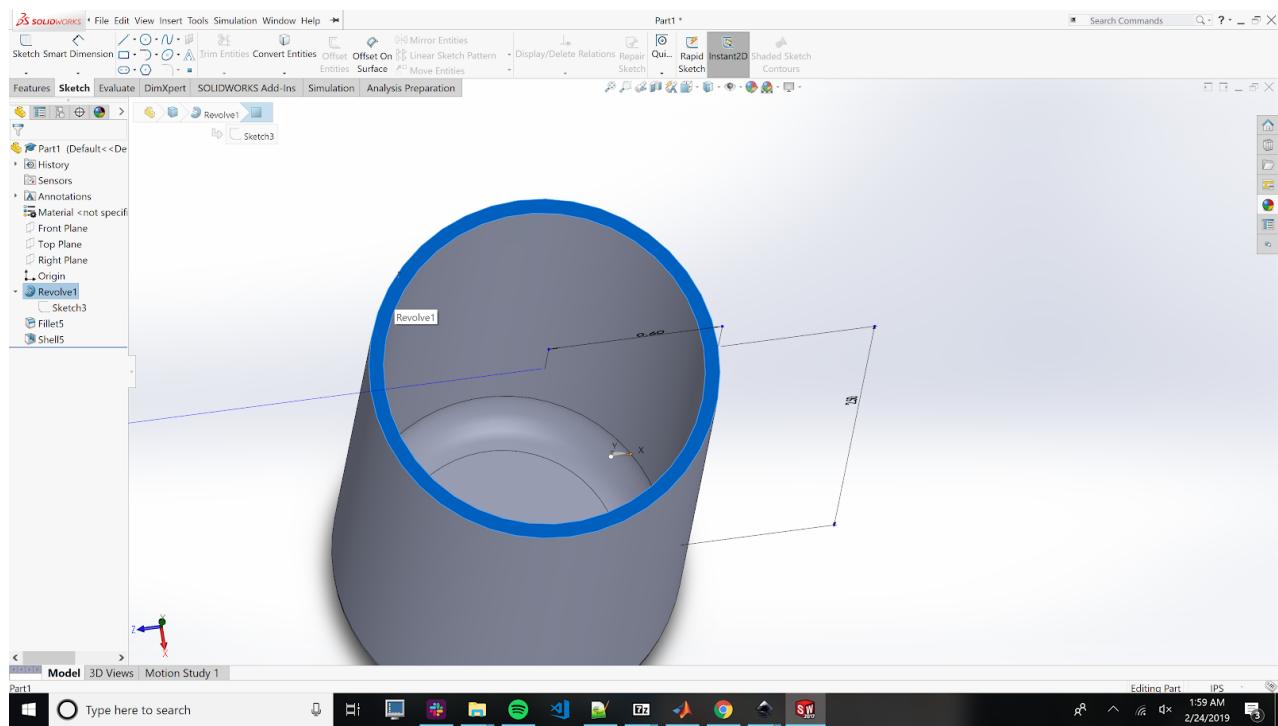
Also much like how we cleared material on the inside of the cap in the previous step, we can also clear out material out of the inside and make the thickness of the material consistent across the entire part. We can do this using the shell tool.



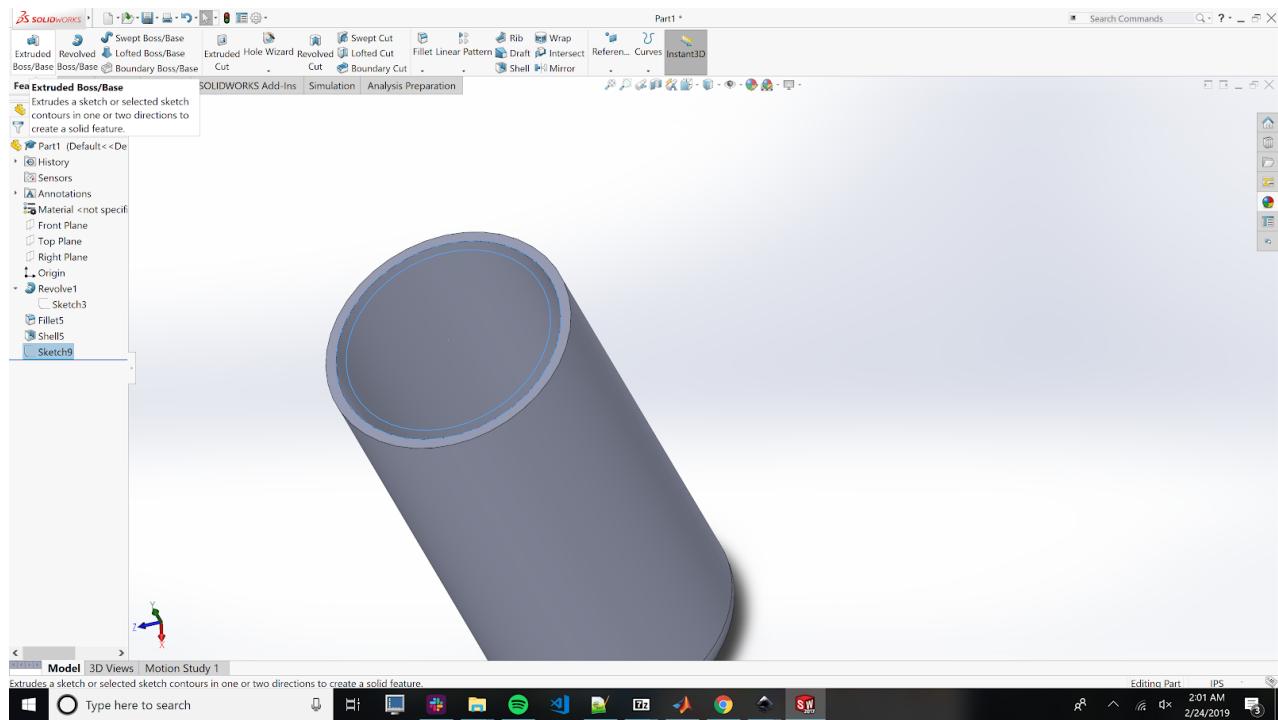
We can click on the face that we want to shell from. Let's click on the face that we didn't fillet. This should clear out all of the material on the inside of the cylinder up until our fillet on the bottom of the cylinder.

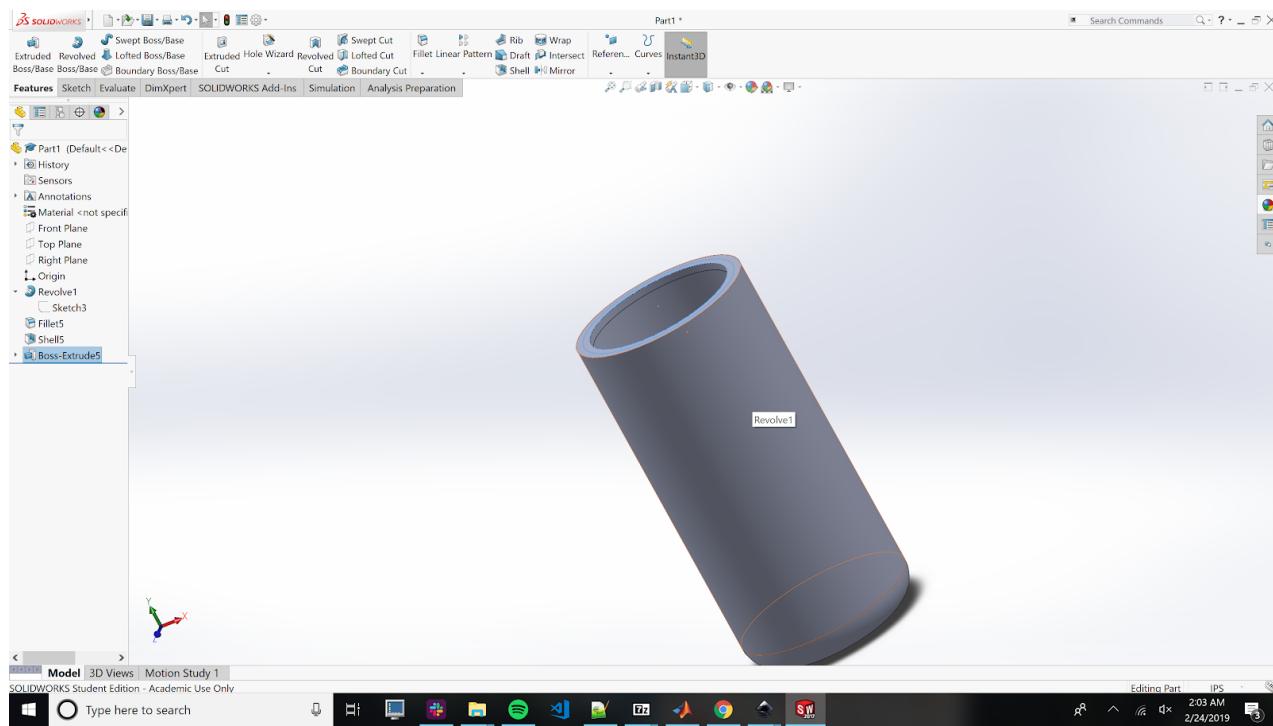


This should now turn your cylinder into a shell, with the inside free to put whatever you want inside.

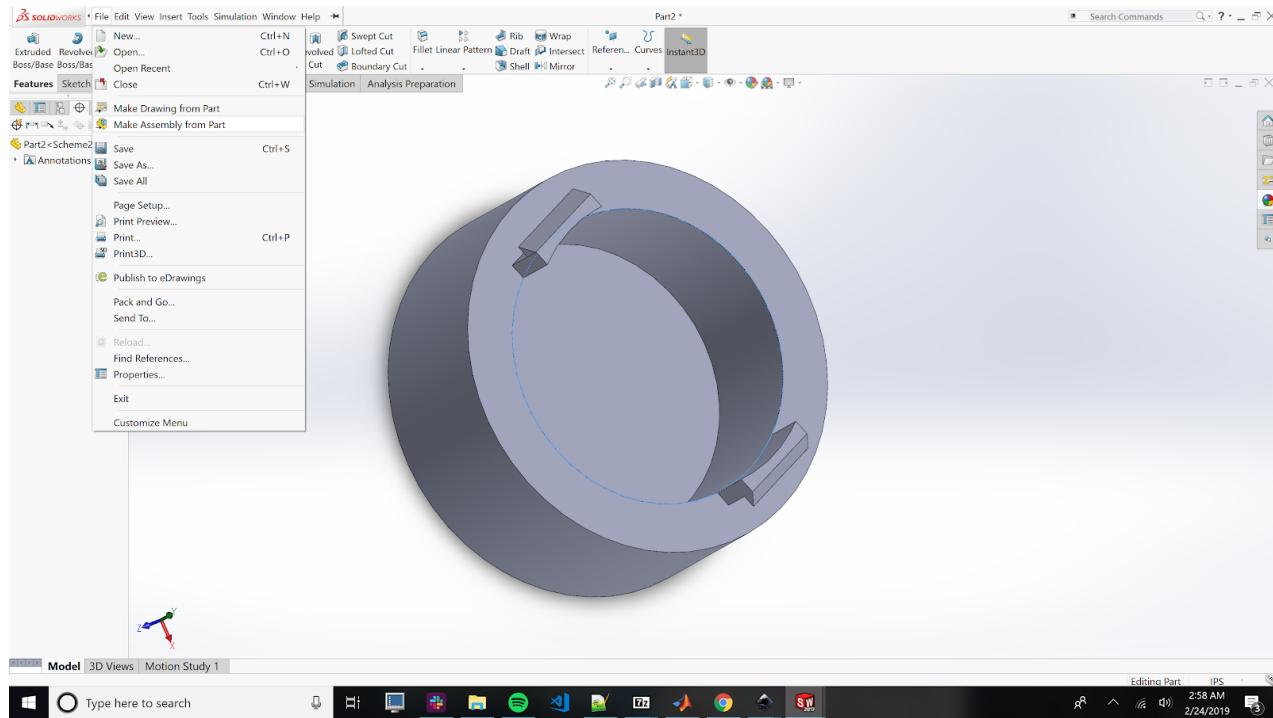


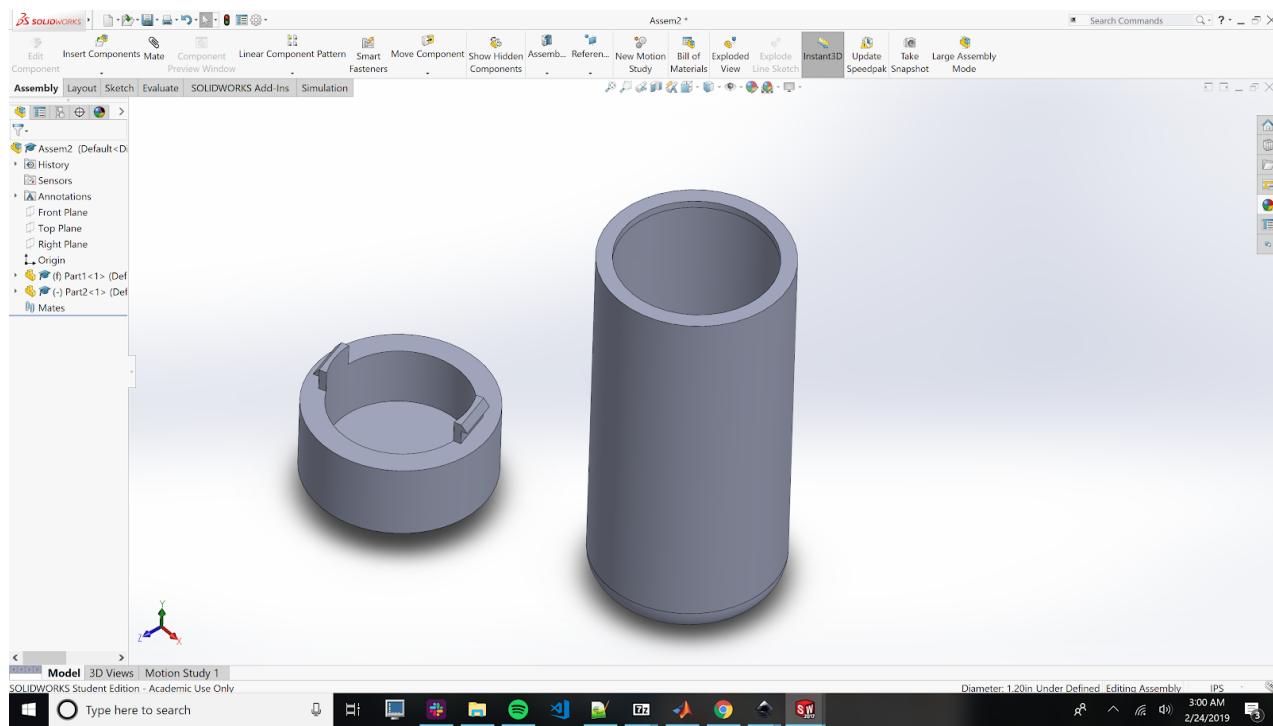
Now let's finish our design by finishing the edge which the snap fit portion will fit on. By making a circle at the top of the cylinder and extruding it.





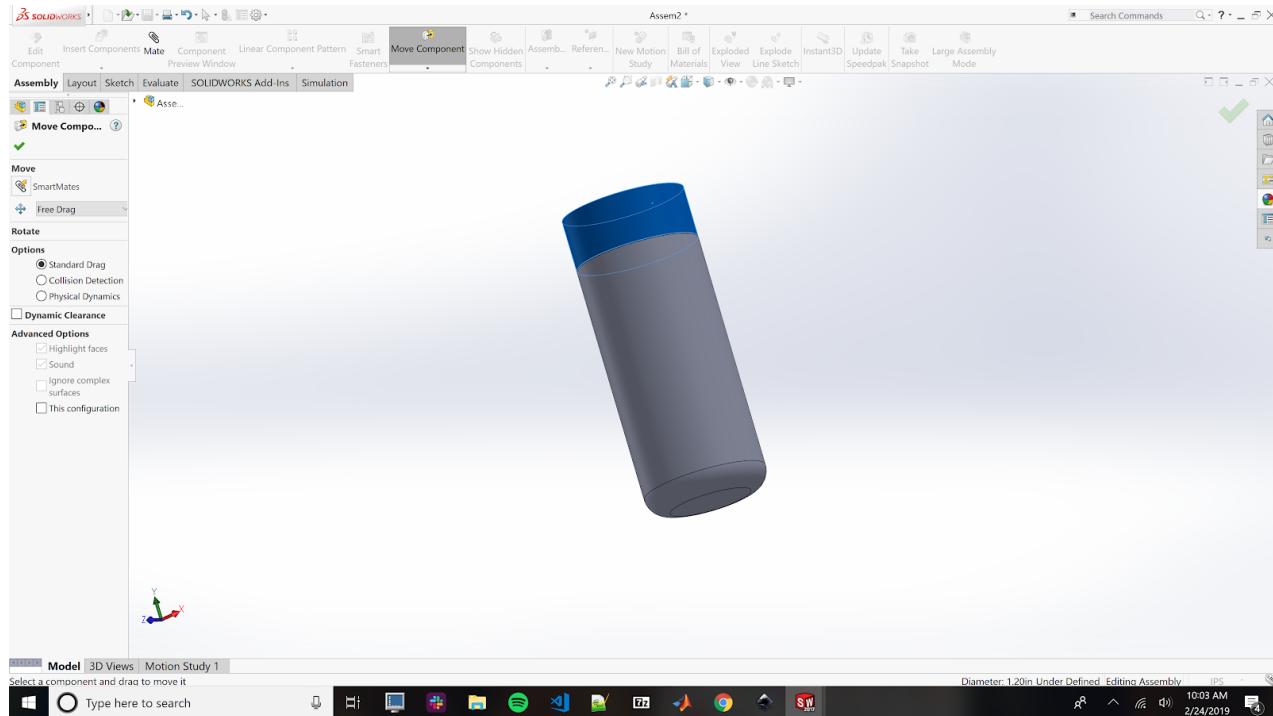
I won't go much into detail for assemblies for the interest of time (mine and yours), but you can then add your parts into an assembly to see how they look together! Move them around and turn off collisions to see how they fit together.





Now that you have learned the basics of solidworks, you can now use this to create your own 3D CAD model, or edit the model that we made right here to improve functionality. Some notable increases could be a clip that could attach to the top of the cap, improvements to the cap to make it easier to open the container, and adjusting the inside to better hold your components.

### *Finished Product*



# References

<https://www.solidworks.com/sw/resources/solidworks-tutorials.htm>

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