Cad Lab

With Junfeng Ma

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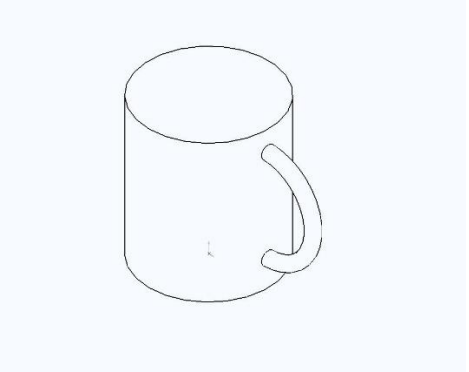
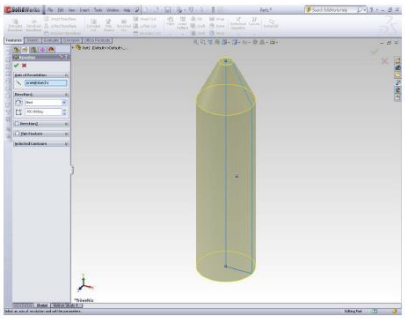
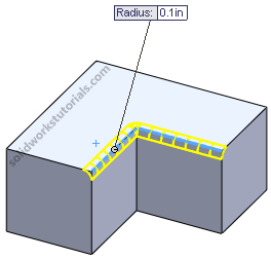
Creating Parts:

To kick off the CAD lab, it was necessary to learn the skills necessary to create basic parts. The first thing that was taught was how to create a sketch. The most important thing is to make sure you are in the correct units before you begin to sketch. For us, these units were IPS. Then after creating a sketch you can add many features. The first taught and most used, is the extrude and extrude cut. These just simply make your sketch into three dimensions or cut out a section of the part. We learned many other features such as sweep, revolve, shell, fillet, chamfer, and loft. All of these are necessary to create different parts. For example, to create this handle it was necessary to use a sweep. A sweep starts with just a sketch of a circle, followed by a sketch of a line with arcs. You do not even have to extrude these individual parts. You simply click the feature to sweep and choose your circle as the path and the line/arcs as the path to follow and it gives you a part similar to this one. Also, to create this candle stick it was most efficient to use the revolve feature. You simply create half of the candle, finish the sketch, click on the revolve feature and it will revolve it a full 360 degrees or less if you choose. The last of these features that I would like to talk about is the fillet feature. This is a very simple feature that simply creates a curved look on the outside of your part to make it look smoother and less like a box. It is used to make the edges of almost any product you can think of more aesthetically pleasing. It may seem like a feature that no one cares about, but it is so simple to use that every part should entail at least one fillet. To the left, is one simple example of a fillet that was used on a basic part. All that you have to do is click the feature and tell solid works what you want the radius of the fillet to be. These three features were used time and time again in order to make what looks like a difficult part, very easy to make and to make basic parts look more attractive and feasible to use.







Creating Assemblies:

After creating a multitude of parts, it was time to learn how to put them all together. Instead of creating a part file, we learned to create an assembly file. It is necessary to place all components that you want to put together in this file. Just as the definition says, we are going to assemble each individual component into one working part. We do this by mating each component, one by one, to the component that it needs to attach to. There are a few different types of mating, but we use coincident, concentric, and tangent the most. For example, if we want a screw to go on a lamp, we will choose the proper mate and put them together. This ensures that they will be stuck together and not able to move. This is a very simple thing to do and looks like a piece of cake after designing all of those difficult parts. For our CAD final project, I decided to create different parts of a grill and mate them all together to one assembly. I have attached a few pictures of this project. You cannot directly tell, but all of these parts are stuck together and only a few of the parts rotate, like the knobs and the top of the grill. This is the most practical way that a grill would get used. We also learned how to make drawing of the parts and assemblies directly on the computer instead of just draw them by hand. This saves time and is a lot easier to do for difficult parts. You can edit the dimensions and have them put right on the page as well.