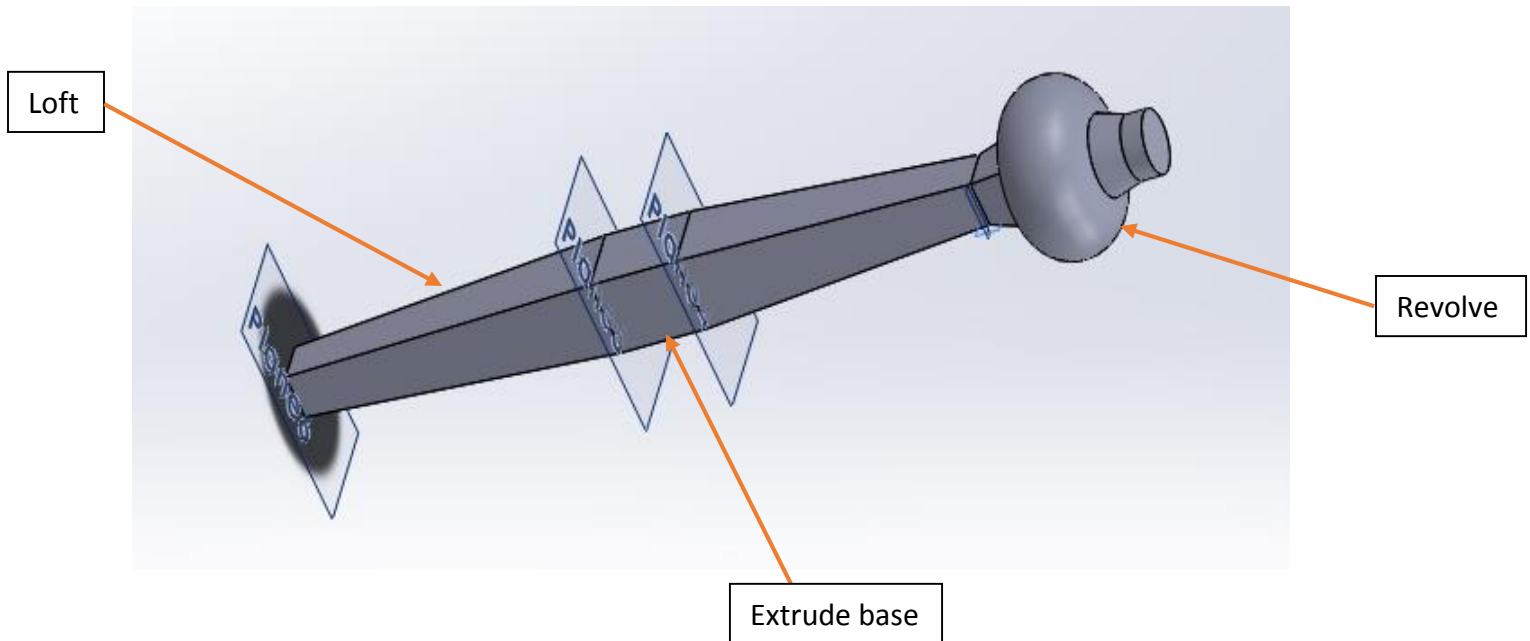


In the CAD Lab section of class, I learned how to create 3D models using SolidWorks. There are 3 main areas of SolidWorks that I learned how to use. The 3 areas are parts, assembly, and drawing.

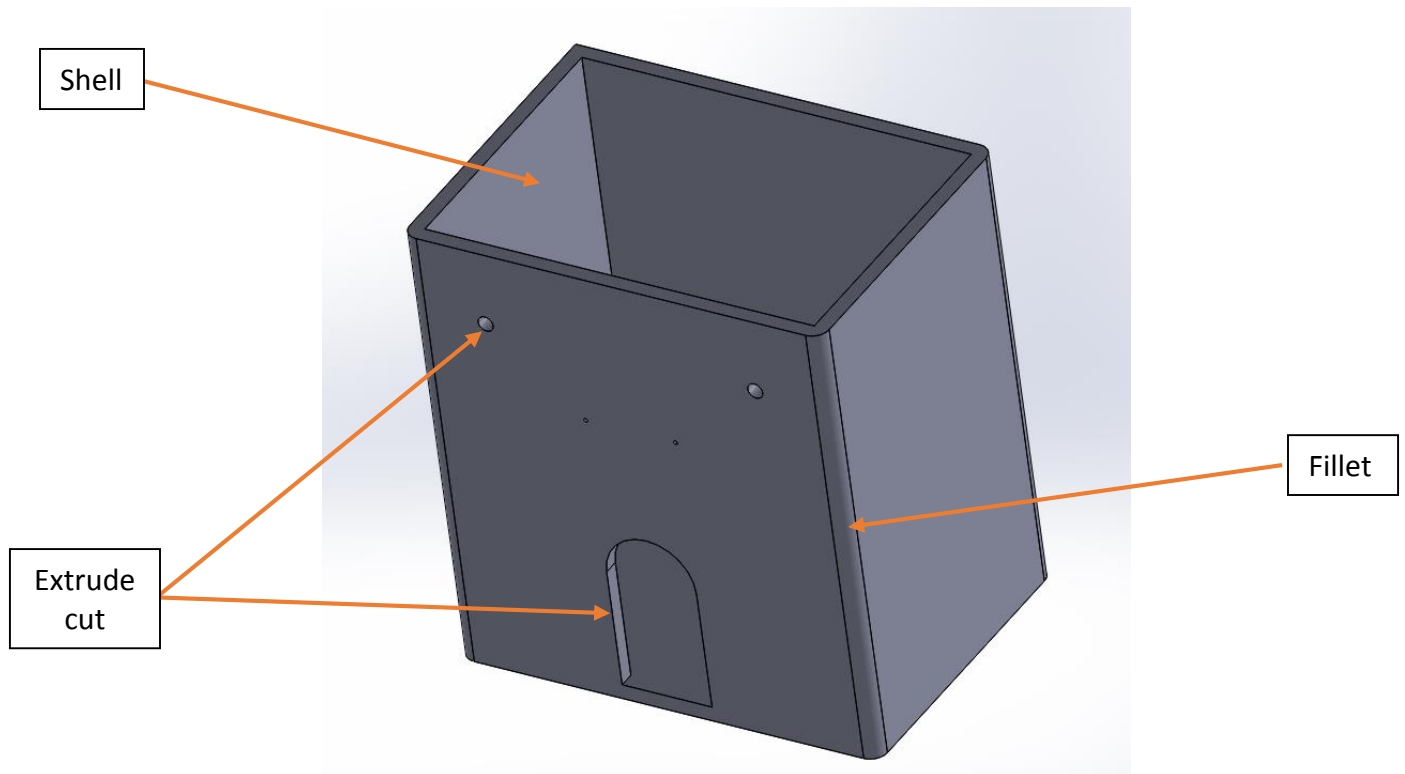
Parts

The parts section of SolidWorks is where I created the individual parts to a model. There are many different features in SolidWorks I learned how to use in order to create a part exactly how I want it. The main features I used are extrude base, extrude cut, revolve, sweep, fillet, loft, and shell. Here is an example of the leg of a table I designed that uses extrude base, revolve, and loft.

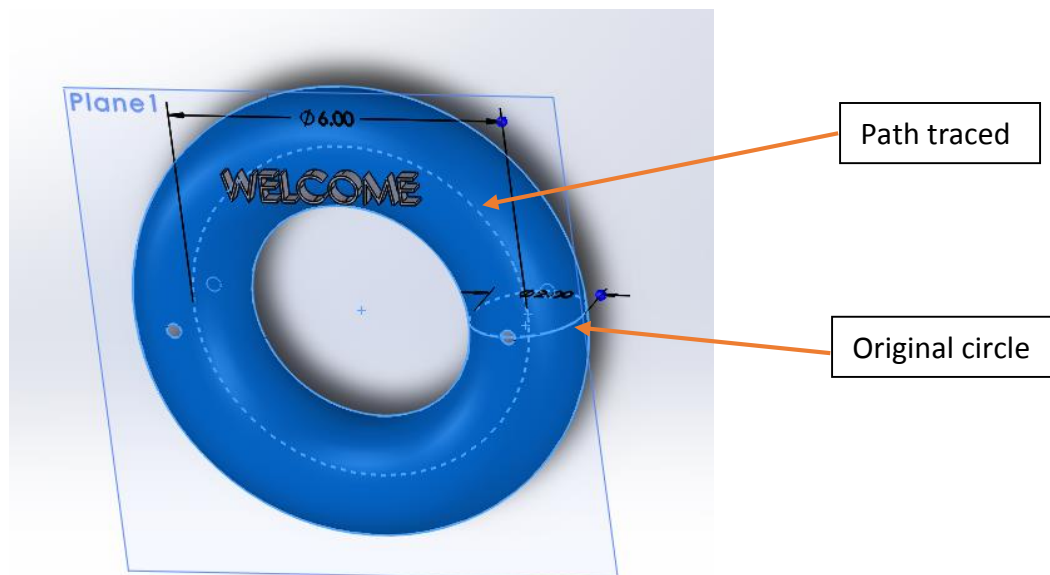


The extrude base feature was used to raise the outline of a shape to make it three-dimensional at a set height. The revolve feature was used to revolve the circular shape of the leg top around a given drawn side. The loft feature was used to do the same thing as extrude base, but to make the dimensions of the shape on the connecting plane different larger or smaller in order to create the look it has.

Here is an example of the body of a birdhouse I designed. I used the features of extrude cut, fillet, and shell. I used extrude cut to cut out the door and holes for the bird sticks. I used fillet to round the edges of the body of the house to a set length. Finally, I used shell to cut out the inside of the house while leaving a set width for the walls of the house.

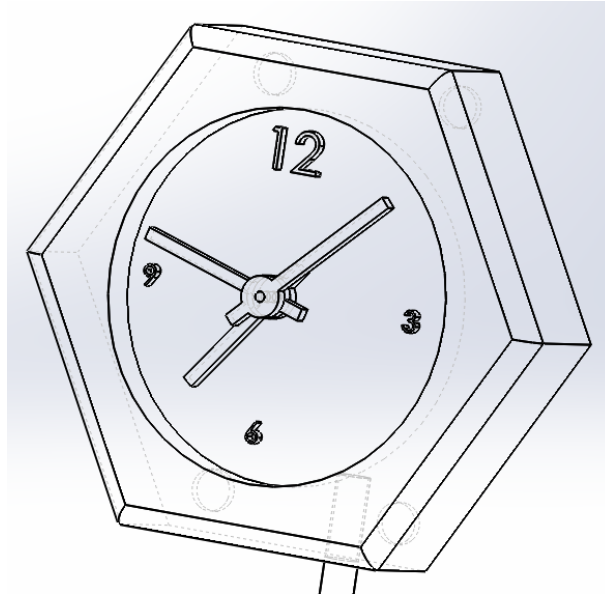


Lastly, I used the sweep feature to make this wreath. The sweep feature allows you to trace a shape around a separate plane. Here's you can see how I drew a circle and traced it around a circular path in order to make the ring shape of this wreath.

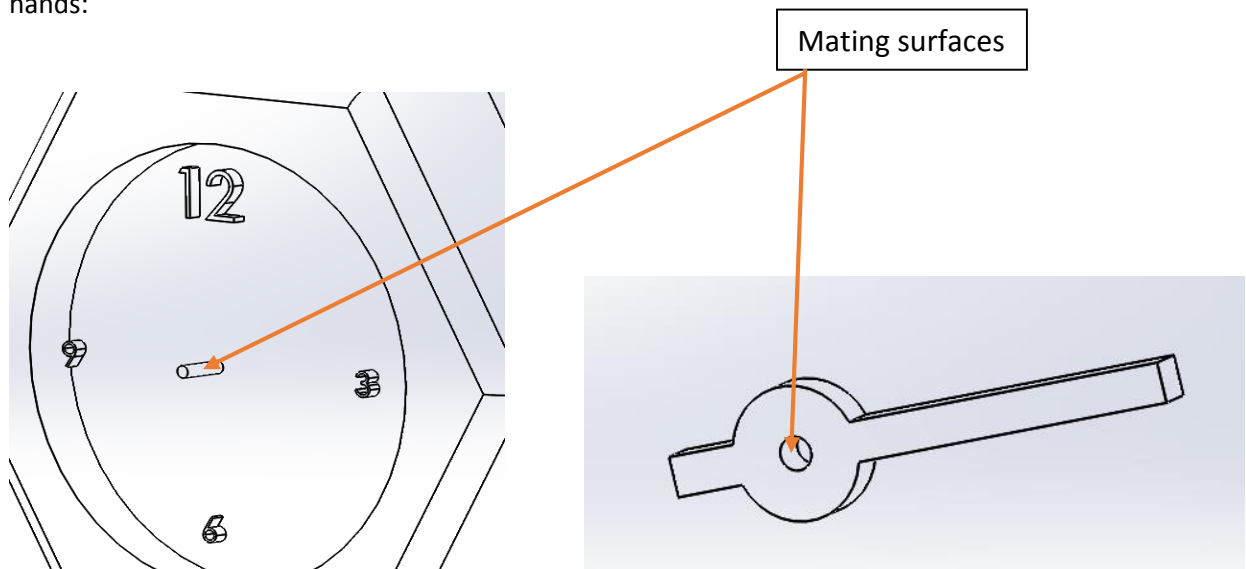


Assembly

The assembly section of SolidWorks is where all of the parts created are put together, or assembled. To do this, the first thing you do is add all of the components that will be used in the assembly. The components are all of the parts. Then to connect parts, you must mate a side or edge of one part with the side or edge of another part. Here is an example of the assembly of a clock I designed.

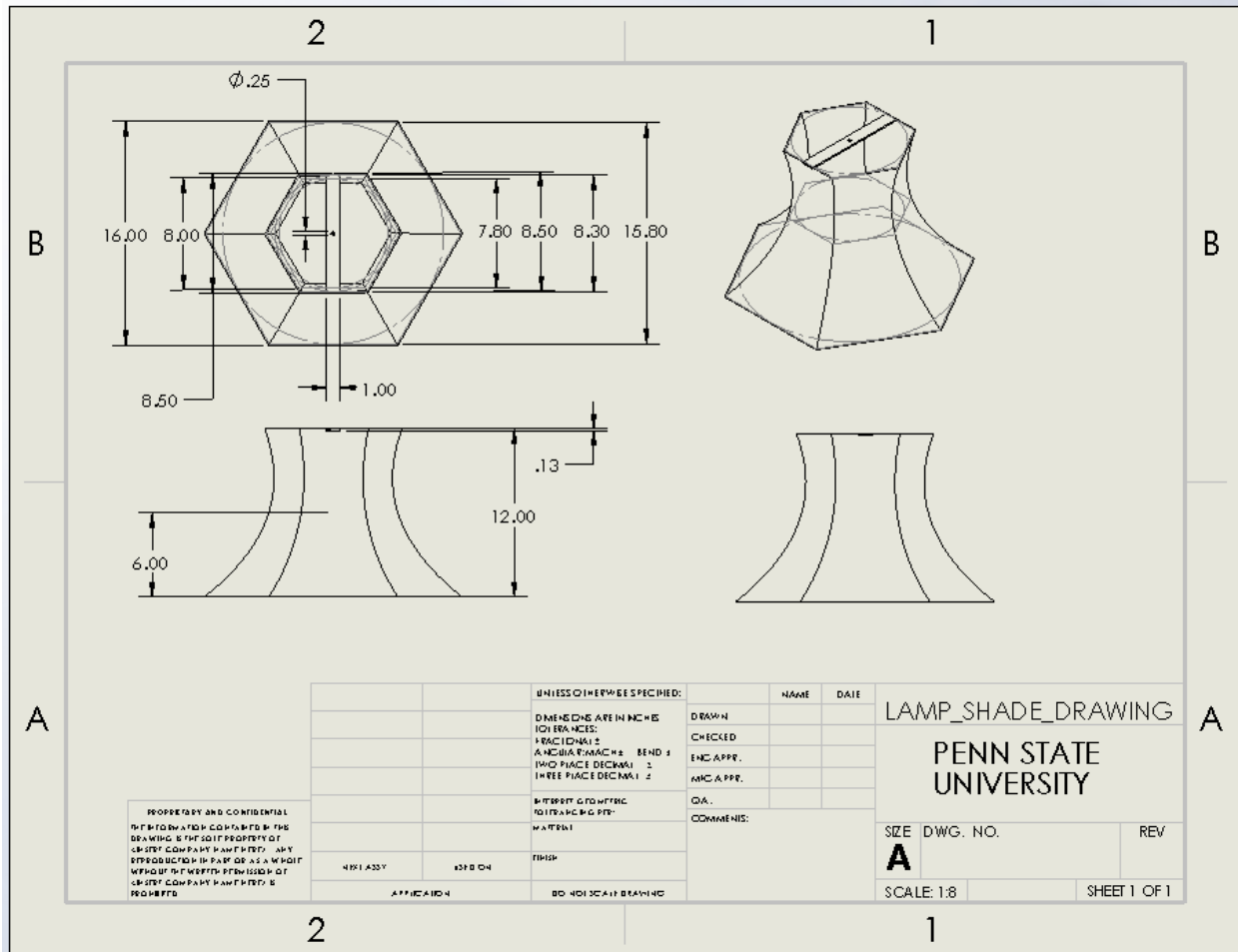


As you can see, to attach the hands to the clock, they need to be mated with something on the body of the clock. To do so, I extruded a small, cylindrical pole that can be mated with the inner circle of the hands:



Drawing

Finally, the drawing section of SolidWorks is where the multi-view drawings of any of the parts or assembly can be created. The drawing mode is extremely helpful because it can automatically add any dimensions needed and will create any multi-view drawing or isometric drawing wanted. Here is an example of the drawing for a lamp shade I designed.



As you can see, all dimensions are only given once, so every view of the model isn't cluttered with repeated dimensions. The top, front, and right views are given along with the isometric view. At the bottom right, there is room to give the scale used in the drawing. There's also a title table where you can give your name, title of the project, or any other information needed to be shown.