Personal SolidWorks Project: Padlock

The object I decided to model was a padlock that I had in my room. This padlock opens with a combination. I chose this object because I thought it would be interesting to assemble it. I wanted to be able to spin the top part which would realistically model how someone would open the padlock. I also was interested in figuring out how to assemble the hook part so that when it opens only one side could move out. I made three different parts before I assembled it. I first made the base which was built by base extrudes and then I used fillets and base cuts for the minor details. I had to then extrude holes for where the hook would be inserted. Then from those extruded holes I had to cut out for where the hook would be inserted. I then put an ellipse shape in the middle so that the padlock hook could “hook” into the hole. The next part I made was the top piece. I built this by revolving around a sketch that I made. Then I finished it off with some fillets. The last part I made was the hook. I made this through a sweep base. I then made a different plane and cut out where the hook would connect to the base part. The hardest part about this was trying to make the hook fit inside the base. Also the assembly was more challenging than I expected.

Figure 1: Base Part of Padlock  
Figure 2: Inside of Base  
Figure 3: Hook Part  
Figure 4: Top part of padlock  
Figure 5: Finished Padlock (opened and closed)