Figure 1 – Helmet Fixture isometric view
Figure 2 – Helmet Fixture front view
Figure 3 – Helmet Fixture top view
For my final SolidWorks project, I wanted to do something that was both important to me and something that would help me learn the software even better. I decided on doing a part from a helicopter pilot’s helmet. At my dad’s work, they design and manufacture both airplane and helicopter pilot helmets that our Special Forces use. I have had the lucky opportunity to work with my dad and help build numerous parts that go into the helmets manufacturing. The part I chose was one that I have been assembling the longest, thus I was most familiar with it. I thought this would be a great opportunity to design something on SolidWorks that an actual engineering firm designs and uses.

When designing this part on SolidWorks I used many of the same types of features that we learned in class, but I also learned about other new features, such as mirror for when I built the spring. Probably the most common features that I used were extrude base and extrude cut. I used extrude base for all of my parts to get them to be the thickness needed. Most parts had many cuts that had all shapes and sizes, so I often ended up extrude cutting multiple times on just a single part. I even used an extrude cut that I set an angle to.

The part that I had the most difficulty with was the spring. It seemed like a simple design and that it wouldn’t take me long; however, I was wrong. It had many curves and angles to it that made it more than challenging. I remember when I finished that part, I felt a great deal of relief. That part definitely made me better and more knowledgeable at sketching.