The Ice Skate

For my personal solid-works project, I decided to create an ice skate. The reason I chose this is because I have been playing ice hockey for most of my life, as well as following the sport on a professional level very closely for many years. Because of that, I felt that a skate would be an appropriate representation of my personality.

In designing this project, I came across numerous difficulties. One of the main difficulties I found myself stumbling upon was the inability to create the front of the skate, specifically the toe area as well as the tongue of the skate. This process proved to be quite difficult because of non-linear loft required as well as the requirement to match the shape of the sides of the skate. To solve this problem, I used a loft tool in a sketch within my actual assembly. To elaborate, within my assembly, I was able to create a new sketch on top of my assembly. In this sketch, I carried out the basic loft procedure of creating multiple planes and connecting sketches on each of the planes. This new procedure allowed me to shape this part of the skate to fit almost exactly with the sides of the skate, because within the assembly I could use these sides as a reference point in my sketching and dimensioning.

Along with this new ‘sketching within an assembly’ technique, another tool that I used that was new to me was the mirror function. This tool allows you to duplicate a sketch and extrusion directly parallel to the original, making one symmetrical piece. This tool was helpful to me in creating the base of the skate, where I was able to create half of the base using a basic sketch and extrude process, but instead of repeating the whole process to create the other half, I was able to mirror the first sketch, simplifying my work and maximizing my time.

This solid works project really tested my abilities with this software and helped me gain a new understanding of some great tools within the software.