Solidworks Wristwatch

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Figure 1: The watch is shown from an isometric viewpoint in this depiction.

Figure 2: The watch is shown from the top, slightly shifted to show the entire wristband in this image.
**Figure 3:** Here, the watch is shown from a side view.

**Figure 4:** The watch is shown from another isometric angle in this image.
Figure 5: The measurements of the nine different chainlinks are shown here.

Figure 6: Here, a multiview of the sides of the chainlink is shown.

(All dimensions in mm)
Figure 7: This drawing shows the measurements of the watch in multiview.

(All dimensions in mm)
I decided to create a watch in Solidworks on a whim. I had been browsing Amazon for cheap but expensive-looking watches (I need to look professional, of course) the night before the Solidworks idea was due. As I scrolled down the page, my alarm went off reminding me to search for a Solidworks idea. Immediately, I decided that I would make a watch. I googled “Solidworks watch” and came across the following image:

Figure 8: This image is the inspiration of the creation of a watch in SolidWorks.

I decided I could design something similar but add my own twist to the original design. I started by designing the chainlinks for the wrist band. First, I drew the initial shape of the chainlink and boss extruded it. Initially, I tried to copy this design and paste it into a new part, where I would be able to adjust the individual dimensions, because I wanted to make each chainlink smaller than the last. However, I could not find a simple way to do this so I ended up designing the same part nine different times with different dimensions for each chainlink. In each link, I included a boss extrude and an extruded cut to provide a means to mate the links between each other. I also filleted the edges of each link for aesthetic.

Next I designed the watch face. I initially boss extruded a circle and basically chopped up the resulting cylinder into a watch shape. I added two extruded cuts, one on the outer rim, for aesthetic, and another, slightly smaller one that would make up the actual watch face. In the center of the cut, I boss extruded a small nub that the watch hands would attach to. I then used
the text tool, a new tool, to add in the “3”, “6”, “9”, and “12”. Next I used the circular pattern tool, another new tool, to add in the markings for the minutes around the rim of the extrude, above the numbers. To tidy up the watch face, I filleted the bottom of the face. I also used the chamfer tool, a new tool similar to fillet, to make a straight cut at the top edge of the watch.

I then moved on to attaching the watch face to the chainlink. I designed a special piece to fit to the edge of the face, which would attach to the first chainlink. I designed the piece with the same radius as the watch face and mated the two. I filleted the edges of the attaching piece as well.

Finally, I moved on to the piece used to clip the watch onto one’s wrist. The first half of the piece was simply two filleted rectangular boxes connected by a boss extruded bar. The other half of the piece was a filleted rectangular box with an extruded cut that matched the size of the bar. I did not extrude the cut straight through the piece. Instead, the cut is a half-circle because I intended for the piece to be snapped in and out of place when the watch is taken off.

The most challenging part of the entire project was the initial part of designing the chainlinks. I spent several hours researching the size of wristwatches and their band length. Copious amounts of math and trial and error were required to make sure all of the sizing was correct.

One new feature I learned was the circular pattern tool. Using the Solidworks tutorial, I designed the minute tick marks around the edge of the watch face. I also used the text tool to add in the numbers on the watch face. This was very straightforward and did not require any outside help. The last new feature I used was the chamfer tool. This tool is almost identical to the fillet tool except that it creates a straight cut rather than a rounded one. Again, this tool was very straightforward.

Solidworks is a very useful CAD program. It allows for the user to create complex designs from scratch. Even given all the headaches it causes, the learning curve is immense and Solidworks is definitely a tool that is and/or will be useful to many engineering firms. It also allows for users to disassemble real-life objects to understand how the different components work. Solidworks helps the user create mock designs to fix mistakes without wasting any time or material on legitimate prototypes until the design is satisfactory.