iPhone 5s with Powerbeats 2
SOLIDWORKS PERSONAL PROJECT
Figure 1: Completed iPhone 5s part.

Figure 2: Completed left earphone part.
Figure 3: Both earphones connected with headphone wire.

Figure 4: Overall drawing of assembly.
Background:

The inspiration for this project came from the everyday activity of walking to and from classes on campus. After using a GPS tracking application on my phone, I discovered that I walk around five miles on any given day. During most, if not all of that time, I am listening to music through my smartphone and headphones. Consequently, I spend a lot of time doing so, which inspired me to reproduce my iPhone 5s, as well as a pair of Powerbeats 2 headphones. I was interested to see these items replicated using a modeling software which is ultimately why I chose them for this project.

To assist with the modeling of the smartphone, I used both my actual phone as well as a dimensioned picture, Figure 5, from the internet. With the headphones, I did not have a physical pair available so I used a couple regular pictures, Figure 6 and Figure 7, found using Google to help shape and dimension them.

Figure 5: iPhone 5s Dimensions
http://www.instructables.com/file/F6CPDPVHAWSH212/
A number of different features were utilized in order to complete this model. Boss extrude was the most basic, and used for a variety of different portions of the model. This feature is how the iPhone body, any buttons, and the base of the headphone were raised from the initial sketches, giving them depth. Cut extrude was another feature often used, primarily in the creation of the iPhone. Any holes that are in the made were made through this feature. Most importantly, the holes used for connecting the iPhone to the wire and the headphones to the wire were created using cut extrude. Also, this feature allowed for the variation in colors and the use of symbols as well. Extremely small cuts were made in order to shade parts of the model more accurately.

Two other features utilized were fillets and chamfers. They allowed for the chipping and rounding of edges, making the model more realistic as well as visually appealing. A lot of the edges on the iPhone were chamfered and filleted to make it more accurate, as well as the ear bud on the headphones. Another feature used was the sweep, which, more specifically was used in the production of the headphone wire.
combined with the spline editing tool allowed for an efficient way to make the wire.
Lofts were used frequently in this model, mainly in the headphones portion. The ear
holder was entirely produced using the lofting feature, as well as the ear bud section. The
ear bud section also required the shelling feature in order to represent it precisely.

However, the most difficult part of this model came in the form of the base of
each of the headphones. The design of it has many curves in it as opposed to straight
lines. Also, the end of base closest to the logo has a rounded shape. In order to
accommodate these figures, a new feature was applied. After the sketch was completed
using lines and tangent arcs, the sketch was extruded to the appropriate height. From
there, a split line was created and applied to the faces of the figure. I learned about using
the split line feature from the video “SolidWorks – Organic Shapes with Reference
Surfaces” by Gosimulation on YouTube. The split line resembled the actual shape of the
headphones, as opposed to the previous blocky structure. After the split lines were
applied, the separated faces were selected, deleted, and patched. As a result, the new
figure resembled the actual object. Aside from learning about using the split line, I also
learned how to make a new, angled plane. This was required to complete in order to
properly produce the earbud itself, relative to the rest of the headphone.

By using Solidworks for this semester, I have exponentially increased my
knowledge pertaining to modeling software. This was my first experience using anything
like it, so it was all new to me. I’ve learned much about the terms used, relationships
between items in sketches, and how to use Solidworks in general.