Manufacturing the Tik-Tok/LegZ Foot and Shoe

Brian Lui
MAE 4900 Final Report (3 credits)
Feb-May 2018
Contact: Bl569@cornell.edu
Cornell Mechanical Engineering Undergraduate, Class of 2019
For all files and models, please contact the Author

Other group member: Kate Na

ABSTRACT

This report reviews the prototyping and manufacturing process of a carbon fiber foot and foam shoe for the Tik-Tok and LegZ robot. It builds upon the work done by Diana Chu, Ethan Kramer, and David Shi in the Spring 2017 semester, as well as the work done by the Author and Ethan Kramer in the Fall 2017 semester. Specifically, the design of a flexible PCB board that will go inside the foot and successes and challenges that arose in manufacturing the foot from start to finish is described.

INTRODUCTION

The Tik-Tok robot is an open-source biped robot that aims to be safe, robust and cheap. It is to have a Cost of Transport (the energy used per unit weight and distance) of 0.25. This is comparable to humans and is done by using high-efficiency chain drives and powerful brushless motors.[1] Tik-Tok is 1.5 m tall, weighs 30 kg, has 12 actuated joints, has a peak joint power of 2 kW, and is expected to be able to walk about 15 km on a single charge of its 300 W-hr 2 kg battery. The LegZ is a robot that has similar robustness as the Tik-Tok robot but with a target manufacturing cost fifty times cheaper, extrapolated to around $2,000 in mass production quantities compared to $12,400 for the Tik-Tok.[1]

Work on the robot foot was started by Ethan Kramer, David Shi, and Diana Chu in Spring 2017. The foot was designed using a Function, Constraint, Objective (FCO) model.[2]

- Function: To allow the robot to walk or run over various terrains while maintaining functionality of the robot.
- Constraint: Attach to given ankle geometry and withstand loads of a robot stepping on a rock. Provide a large, flat surface area for a 60 by 40 mm PCB. Provide a curved bottom surface of the foot to allow for smooth gait. Additionally, allow room for sensors along the bottom, back and front of the foot to detect foot placement. The foot must be able to be manufactured using available methods at Cornell University at this time.
- Objective: Minimize mass, provide a factor of safety in yield of at least 3.

Figure 1. Version 4.2 of the foot.

Under these constraints, the current version of the foot is seen in Figure 1. The difference between version 3.11, which is the final version of the foot from the Fall 2017 semester, and version 4.2, the current version of the foot at the end of the Spring 2018 semester, is that the bottom curve of the foot has been modified, per Ryan Elandt's request.

Other than modifying the foot, the Author's work focused on creating a flexible PCB that will go inside the foot, as well as a first attempt at a complete manufacture of the entire foot and shoe. The PCB is required to be able to put pressure sensors in the foot, which will give feedback on what the robot is stepping on. A flexible PCB was decided to be the best option for the PCB because of the curved bottom surface of the foot, necessitating a board that can curve with it. The board was designed to go inside the foot because it would be more protected in there and potentially cheaper than using FlexiForce sensors on the outside of the foot.

Design of the board was done using EAGLE, a software new to the Author. Although the lab's computer contained Eagle version 7.9, the Author primarily worked using version 9.0 because of its free access to student. The later version of Eagle contained several helpful features that helped with the layout of the board, the most significant being prevention of laying down traces too close to other traces and to the edge of the board. Manufacturing the entire foot also was attempted, and issues arising from the process are described.

To avoid reiterating information and to keep earlier report from becoming redundant, the Author has attempted to keep this paper focused only on new material that has been worked on. Please consult the Author's Fall 2017 report to see how each half of the carbon fiber foot is created.

Brief Introduction into PCBs

A printed circuit board electrically connects electrical components using conductive tracks, pads and other features etched from one or more sheet layers of copper laminated onto and/or between sheet layers of a non-conductive substrate. The layers of a basic 2 layer PCB can be seen in Figure 2. Below are some terminology that are important to know for someone who may not have had any experience with PCBs before:

Pad - a portion of exposed metal on the surface of a board to which a component is soldered.
Silkscreen - the letters, number, symbols, and imagery on a circuit board. Usually only one color is available, and resolution is usually fairly low.
Soldermask- a layer of protective material laid over the metal to prevent short circuits, corrosion, and other problems.
Trace- continuous path of copper on a circuit board. They connect components to each other.
Via - a hole in a board used to pass a signal from one layer to another[2]

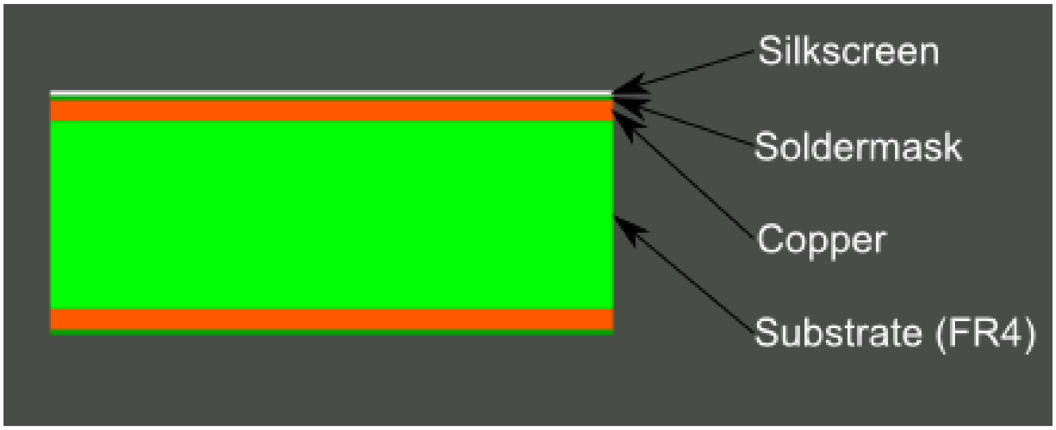

Figure 2. Basic layers within a PCB. The substrate is commonly made of fiberglast or plastic.

METHODS

Design

Foot Design

The design of the foot was modified per Ryan Elandt's request. It was modified because the bottom curve of the previous foot model was harder to model and didn't fit his desired curve of a perfect ellipsoid. There were some issues that arose from filleting the curve that made it hard to model. The difference between the two versions can be seen in Figure 3. This work was started by Ethan Kramer in the previous semester and completed by the Author. Because the foot design was changed, all the molds used in the previous semester cannot be used; everything had to be updated to reflect the change in the foot.

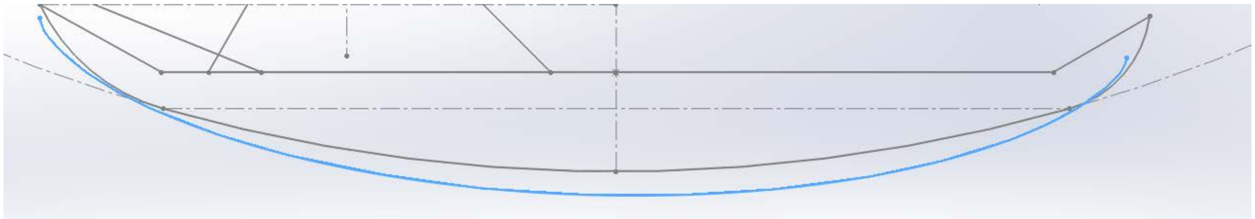

Figure 3. The curve on the bottom of the foot of version 3.11 (grey) and version 4.2 (blue).

Flexible Circuit Board Design

The circuit board that was designed to go inside the foot is flexible instead of the more commonly seen rigid ones. Stiffeners are used so that components can be soldered to the board. The schematic was designed by Jason Cortell. It was designed using the EAGLE software, and can be found under TikTok/Electronics/Boards/B_27_FootForce/B27A_Foot_Force. A quarter of the schematic can be seen in Figure 4. The layout of the flexible PCB was primarily done by the Author, with Jason refining the board and adding finishing touches to it before it was fabricated by PBCWay.

The electronic components included within the board are pressure sensors, switches, resistors, capacitors, microchip, op-amps, and a 10-pin connector that will connect to a board located at the top of the foot. The microchip communicates with the switches, receives data from each sensor, and relays the information to the printed circuit board located on top of the foot. The op-amps amplify the signal if it is too weak.

The first challenge was to determine the dimensions of the board as well as the location of the electronic components. The bottom surface of the foot was flattened using the Flatten feature in Solidworks and manipulated into a friendlier orientation. This outline of the foot was the bounds of the flexible PCB board. From this and the dimensions of the foot, 15 sensors were evenly spaced out in an alternating pattern to ensure adequate coverage over the entire foot. An additional sensor served as a toe stub sensor. A rough layout of the location of each electronic component was done in Solidworks because it was easier to make exact changes in the board within Solidworks than it was within Eagle. Making the board outline within Solidworks also made the process of marking the locations of the stiffeners easier. The board outline can then be imported into Eagle as a dxf file. Such a procedure was best because the dimensions of the board was continuously modified as the traces were laid out; some regions required more space and others less than initially planned. The outline of the foot, the outline of the actual PCB board, and location of the sensors can be seen in Figure 5.

The Author decided that it was best to lay the microchip very close to the front of the foot because the board on the top of the foot was likely to be located in the front of the foot where the foot had a flat surface. The op-amps were also located very close to the top of the board so as to be connected close to the microchip; traces should not be made unnecessarily long.

The second challenge was to design the board in such a way that there was some tolerance in the sensor location. Manufacturing the foot would involve hand drilling holes into the foot, introducing a degree of inaccuracy in the actual position of the sensors. As suggested by Jason, a back and forth snaking pattern was used to connect each sensor to the rest of the board instead of a straight line. This would help deal with the inaccuracy by minimizing any bulging in the board if the holes were misaligned. This specific feature can be seen in Figure 6. All but two of the sensors were connected in such a fashion; the remaining two were located very close to the microchip and therefore did not have enough space within the foot to be connected in such a manner.

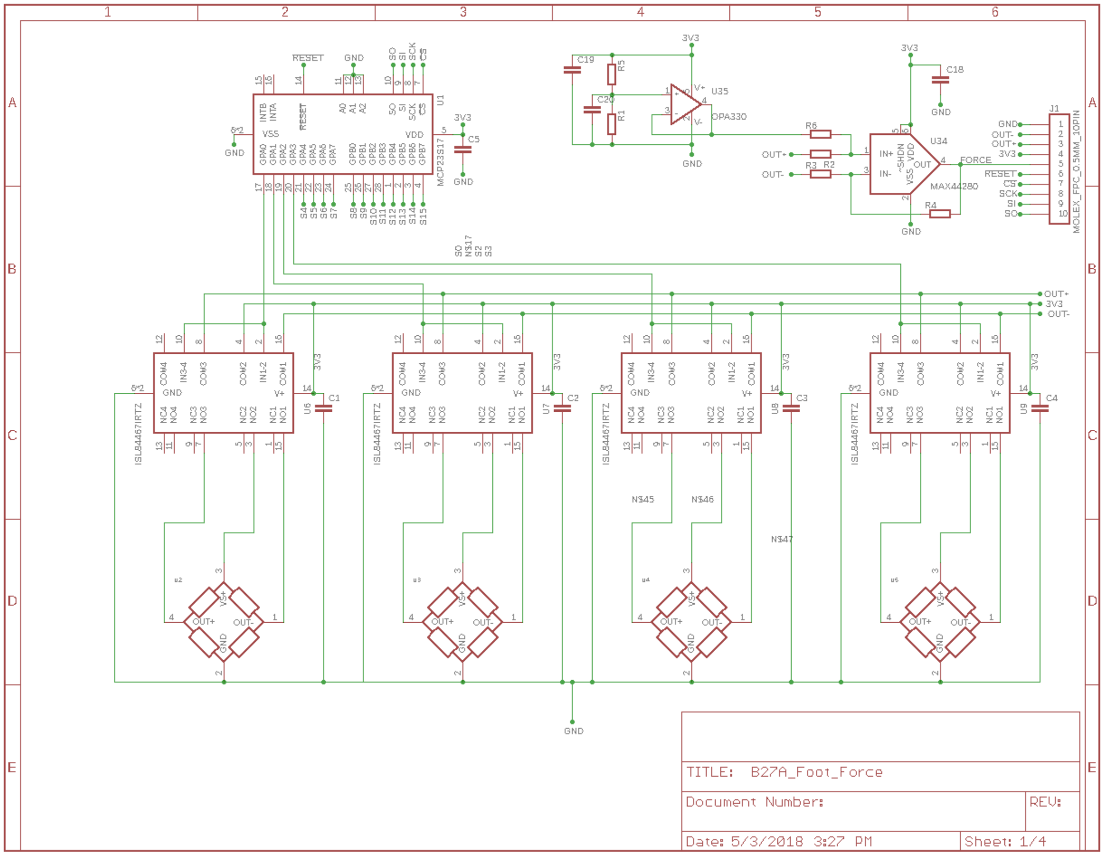

Figure 4. Schematic of the flexible circuit board showing the major components used

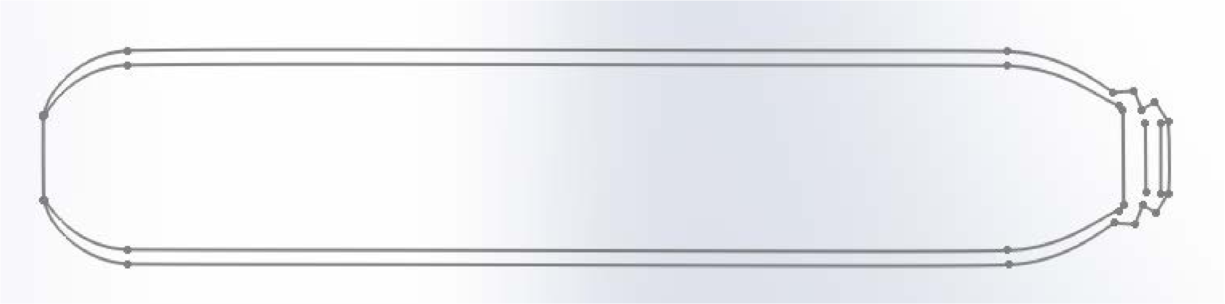

Figure 5a. Outline of the foot. The top of the foot would be on the right of this outline.

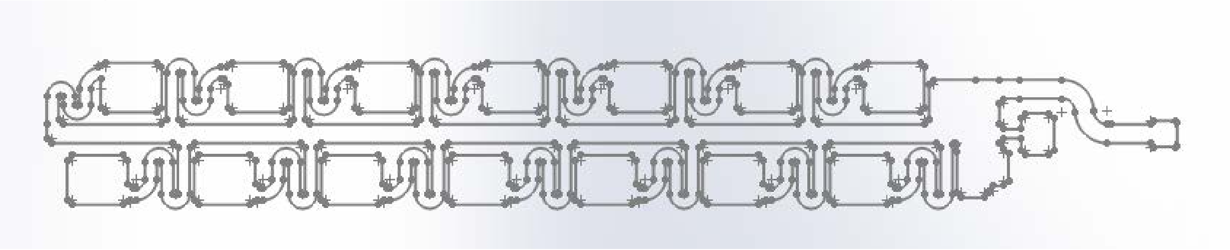

Figure 5b. Outline of the flexible PCB board. The top of the foot would be on the right of this outline.

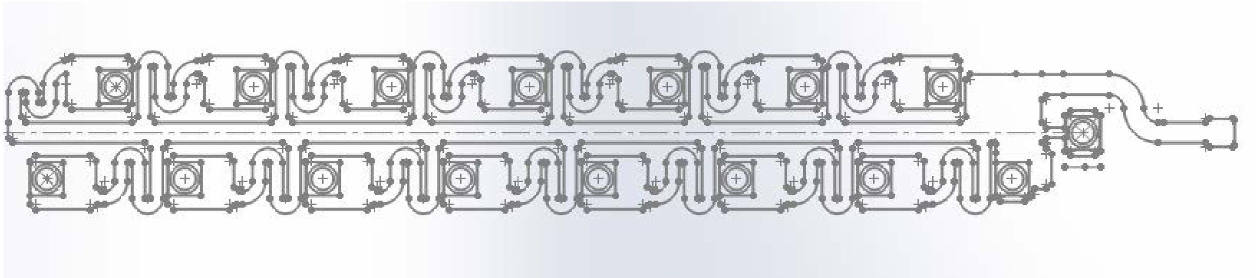

Figure 5c. Location of the sensors on the flexible PCB board. The top of the foot would be on the right of this outline.

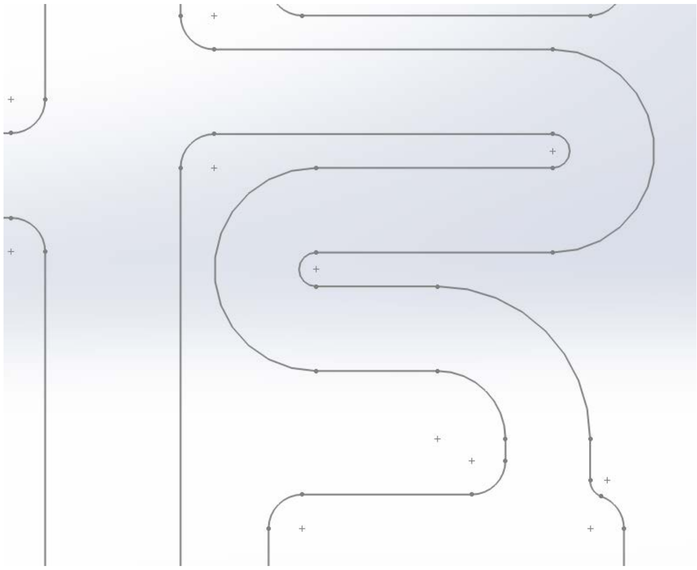

Figure 6. Close up of the snaking region of the board connecting each sensor region to the rest of the board.

The third challenge was to design the board in two layers. This was done to reduce the cost of the board. This was especially challenging given the limited space within the foot and the number of traces that need to be laid out on the board to connect all the components. One area of concern was in the snaking pattern of the board. The GND, OUT+, OUT- and 3V3 traces needed to connect to every sensor, as well as a signal trace unique to each sensor. Because of the limited space in the board, there was not enough space to have GND, OUT+, OUT-, and 3V3 traces run straight down the board and branch into each sensor; there was not enough space to add the number of vias required to do so, especially in the region of the board closest to the microchip. The Author’s solution to this was to have the traces enter the sensor area and come back out to connect to the rest of the sensors. This can be seen in Figure 7.

Furthermore, to make the many signal traces fit within the PCB as it makes its way to its respective sensor, half the signal traces were placed on either side of the board, leaving just enough room for the GND, OUT+, OUT-, and 3V3 traces be on either side of the board. This, as well as the lack of space within the board, can be seen in Figure 9.

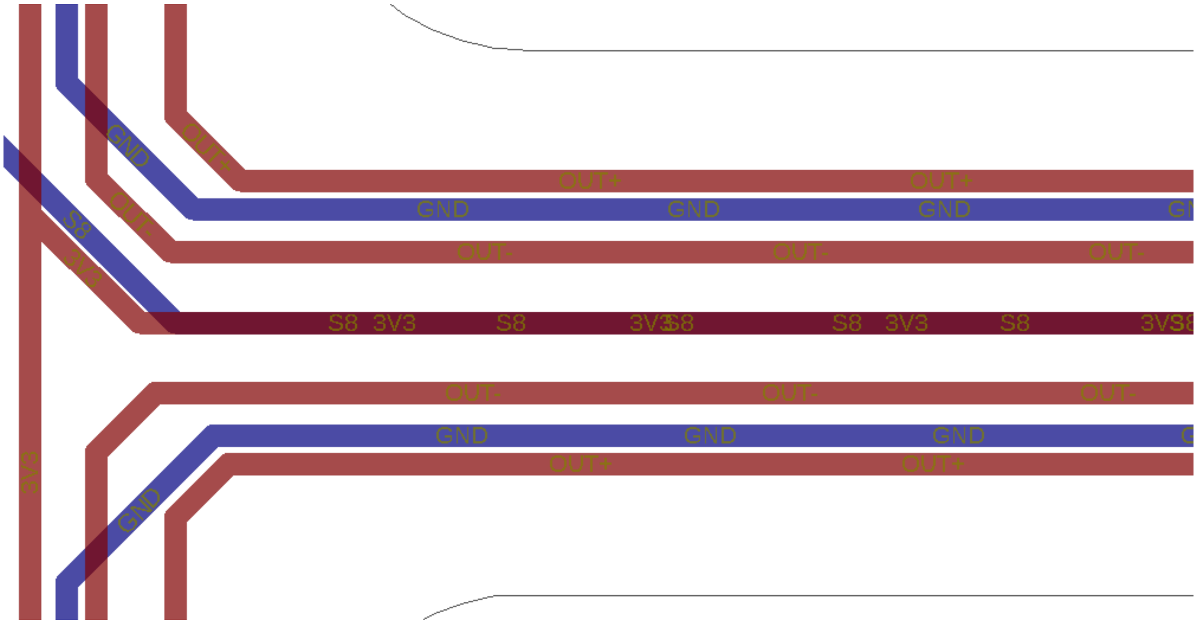

Figure 8. Traces entering each sensor area. The red wires from top to bottom: OUT+, OUT-, 3V3, OUT-, OUT+. The blue wires from top to bottom: GND, S8, GND.

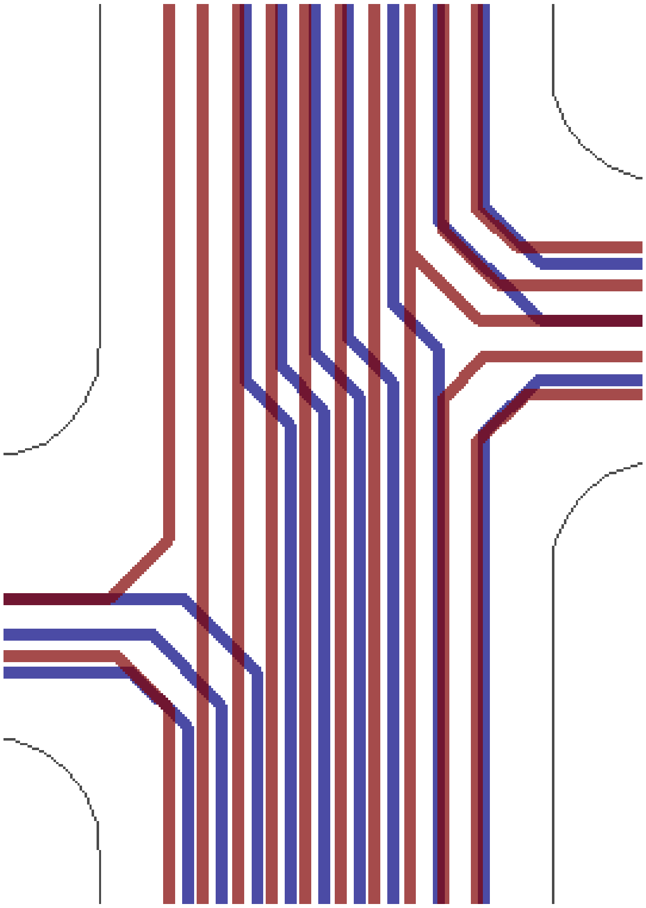

Figure 9. The traces running down the center of the board. The red wires from left to right: S9, S10, S11, S12, S13, S14, S15, 3V3, OUT-, OUT+. The red wire coming up from the bottom that goes to the left is GND. The blue wires from left to right: OUT+, OUT-, 3V3, S8, S7, S6, S5, S4, S3, GND.

For cleanliness and efficiency, an Eagle script was used to create the traces for every snaking region. The fourteen snaking regions of the board, connecting the microchip to each sensor, were designed to be identical. Eagle allows the execution of a script to run specific commands in order, among them commands LINE and ARC for creating traces in a line (by specifying two points) or partial circle (by specifying a diameter, starting point respectively. Andrew Sikowitz helped the Author design a Python 3 script to auto-generate these commands, based on an input .csv file specifying the locations of certain critical points in the snaking region of the board. These commands ensure that the arcs are created symmetrically and traces are connected to each other. The Python program was designed to be flexible to changes to the board or input, which proved useful as the dimensions of the board were changed several times, requiring small adjustments to the program. The Author also wrote a script to place the components in an exact position to match the Solidworks CAD.

After all the traces have been connected, the Ratsnest button should be used to double check that there are no airwires left unconnected. This may occur when components are moved after they are placed, resulting in many microscopic airwires that may appear connected but are not according to Eagle. The Design Rule Check (DRC) should also be used to check that things are wired properly and that traces are not too close to each other or to the edge of the board. The dimensions used for the DRC are those specified by the fabrication requirement of PCBWay and can be checked on their website. The traces in this board have 0.1016 mm diameters, vias 0.2 mm diameters, and the traces are at least 0.2mm away from the edge of the board. The only component that should violate the DRC is the 10-pin connector: it needs to just fit on the board so that it can connect onto the board on top of the foot.

Some things that Jason revised include the spacing of the components and adding a ground plane. The Author laid out some components too close to each other to be reasonably soldered without interference. Adding a ground plane helps reduce the distance that some wires have to travel, helping to reduce some of the noise. The final board that was sent for fabrication can be seen in Figure 9.

If the foot sensor board is to be refined, based on issues arising during manufacturing, the Author suggests that the sensors be placed closer to the centerline in future iterations of the board. The microchip and the sensors towards the front of the board should be brought lower to give more space for the toe stub sensor. Alternatively, the height of the foot can be increased to give more space for the toe sensor. Because soldering on the components onto the flexible PCB and assembly of the board within the foot was challenging, another option might be to use the FlexiForce sensors on the outside of the foot.

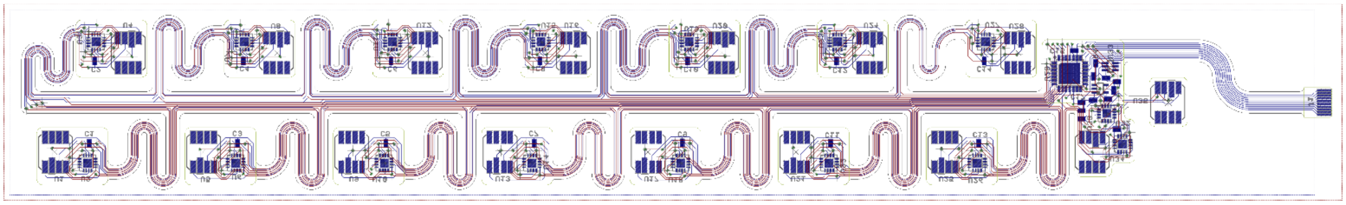

Figure 9a. Completed board layout, overview. The top of the board is located on the right of the figure.

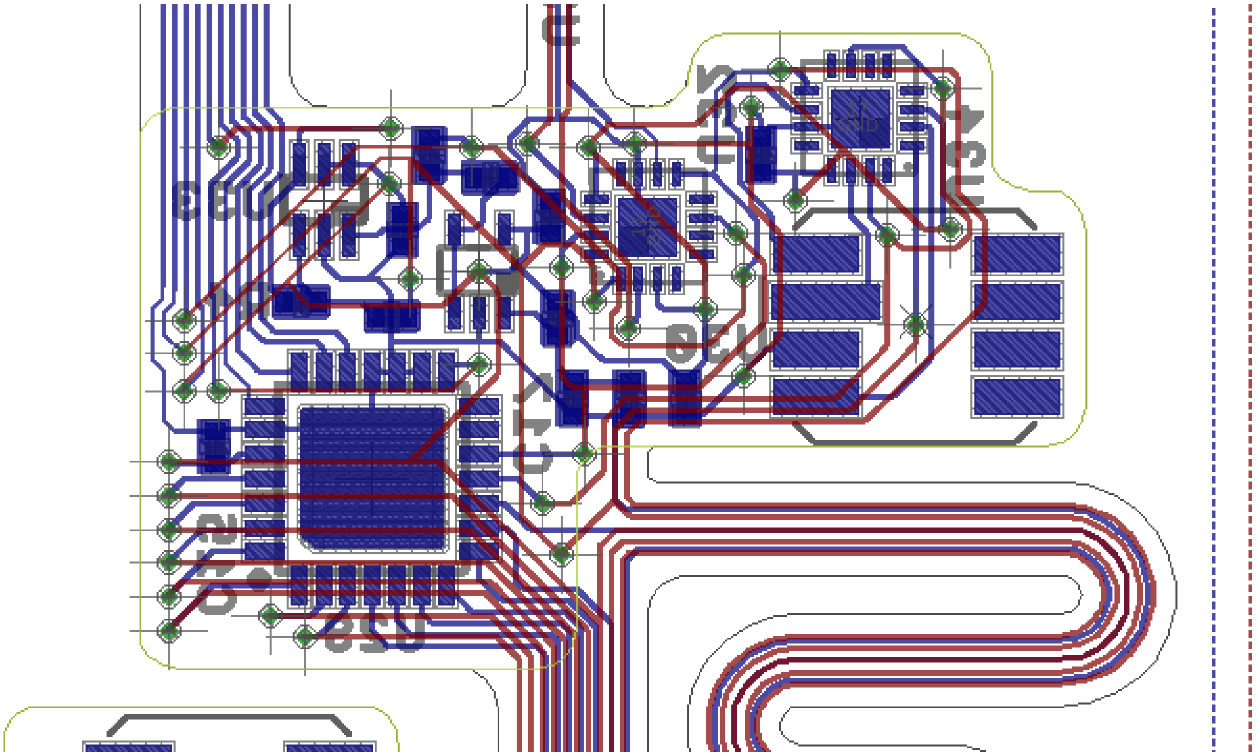

Figure 9b. Completed board layout, zoomed in at the microchip region

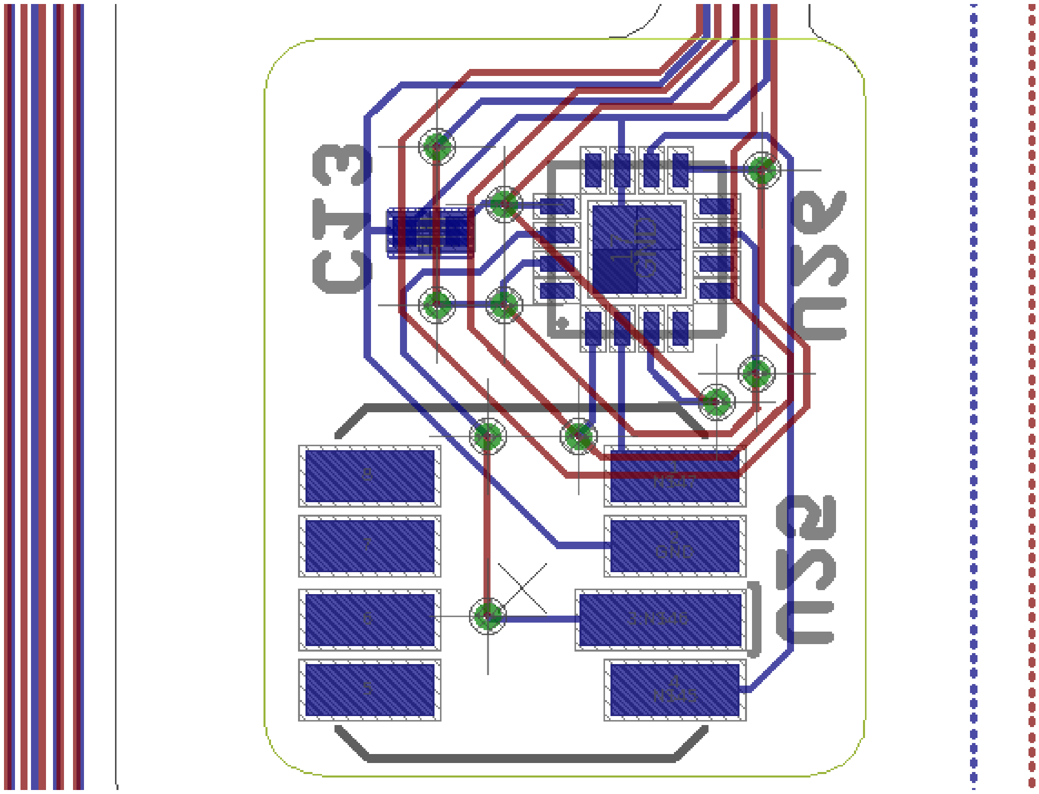

Figure 9c: Completed board layout, zoomed in at a sensor.

Hole Making Jig

This jig was used to cut the larger half of the foot so that the sensors can stick out of the foot. Because all of the molds had to be redesigned, the Author attempted to improve upon the design of the hold cutting jig, hoping to find a better way of securing the foot for drilling. The design of this jig used in the Fall 2017 semester can be seen in Figure 10. The Author attempted to use a design similar to the foot mold: the idea was that the larger foot half would fit in the jig and be secured in place from all four dimensions. The Author also found a way to directly introduce holes into the mold during 3D printing instead of manually having to drill the holes out. This involved using the wrap feature on Solidworks. The jig does not have holes for the toe stub sensor and the hole that needs to go on the top of the foot, so those would need to be drill out visually. In the future, it might be worthwhile to make a separate jig for making those two holes more accurately.

However, the foot half did not fit as intended; the fit was too tight. Although the foot half would have been able to fit in jig, the removal of the foot would be very difficult. This tight fit may have been because the foot was under vacuum pressure stresses when it was originally in the foot mold and expanded slightly after it was removed from the mold, or because the foot deformed slightly as it was removed from the mold. It may have also been because of inadequate printing resolution of the jig such that it was a little too small for the foot. As a result, the printed jig was cut in half to be usable, as can be seen in Figure 12.

Figure 10. Jig used in Fall 2017 to create the holes for the pressure sensors in the outer mold.

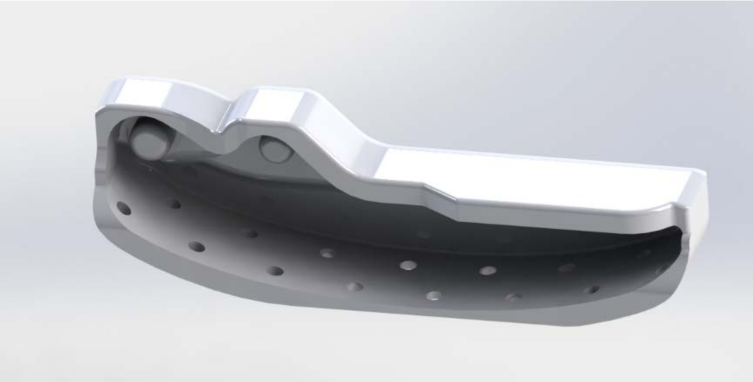

Figure 11. Intended jig to create the holes for the pressure sensors in the outer mold.

Figure 12a. Actual jig to create the holes for the pressure sensors in the outer mold.

Figure 12b. Actual jig to create the holes for the pressure sensors in the outer mold, bottom view.

Shoe Making Mold

Work on this mold was primarily done by Kate Na. Using the previous shoe making mold as a reference, she redesigned the mold so that there will be a little space in the front and back of the foot. This was done so that the foam shoe would cover both the toe stub as well as the foot heel completely, something that was not possible in the previous version of the shoe making mold. The rest of the mold's design remained the same: it would be made in two pieces and can be pulled apart easily when the shoe reaches its final size.

Using the new mold, the foot would be held in place at the ankle and Achilles holes. The softer foam layer would be poured into the mold, and then the foot would be placed into the mold and aligned in place through Delrin rods at the ankle and Achilles hole. Excess foam would exit through the space in the front and back of the foot; this foam can be removed after the foam has expanded to its final size. This mold can be seen in Figure 13.

Figure 13a. Half of the shoe making mold.

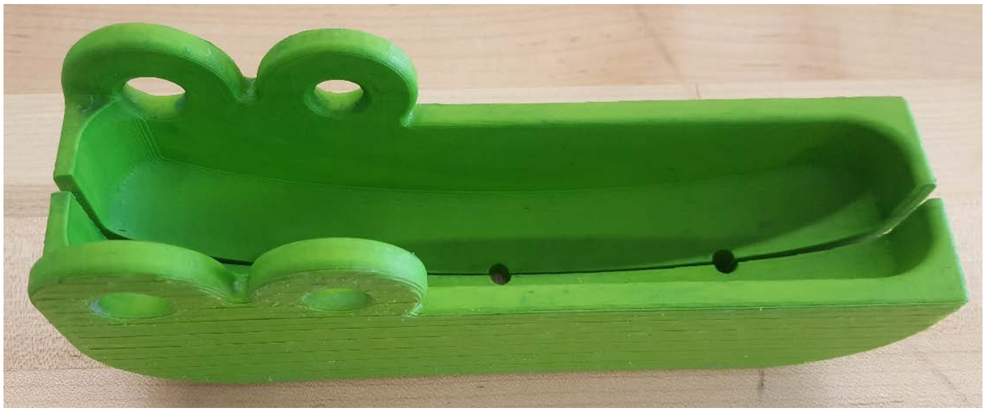

Figure 13b. Both halves of the shoe making mold.

Manufacturing the Foot/Shoe

For the steps required to make each foot half, please look at the Author's Fall 2017 report. This section focuses on the manufacturing process after obtaining each foot half.

1. Drilling holes into the feet

Holes are required in the bottom of the foot for the sensors, as well as on the top of the foot to connect the flexible PCB to the PCB on top of the foot. The hole making jig was used to make holes on the bottom surface in the larger half of the foot. The foot and jig should be clamped down on a table and multiple clamps should be used to hold the foot and jig together; the foot often would deflect instead of allowing a hole to be cut through unless clamped very close to the region in which a hole is to be drilled. The hole on the top of the foot is 15/64 inches (0.234 mm). In the smaller mold, a rectangular portion in the front of the foot was removed using the Dremel, as can be seen in Figure 14. This is to allow for the assembly of the foot, accounting for the connector to the board on the top of the foot. A future iteration of the smaller foot mold would have this region already removed so there would be no carbon fiber in that region.

The Author discovered that the holes in the hole making jig were not large enough. Although designed with the holes having the same diameter as the pressure sensor's circular extrusions, the sensors could not fit through the holes drilled using the original mold. This was likely because drilling holes into the carbon fiber doesn't always result in a smooth and symmetrical hole in the foot. There also wasn't a drill bit with exactly the right diameter as the sensor holes. The required drill bit for the sensors were 17/64 inches (0.2656 mm), and some further fine enlarging of the hole using an X-Acto knife may be required. The Author suggests completely drilling holes in the mold first before drilling holes in the foot; plastic is hard enough to cut through; trying to cut through carbon fiber concurrently makes it extremely difficult. The Author would also suggest using something to shield the inside of the carbon fiber. Sometimes there was little space between the bottom curve of the foot and the other regions of the foot and it may be easy to inadvertently drill into a region that should not be.

The Author also discovered that some of the holes were located too close to the edge of the foot. For the topmost and bottommost hole on the bottom of the foot, drilling the hole resulted in a crack that connected to the edge of the half of the foot. This led to issues in gluing the flexible PCB down. To deal with this, the Author would suggest moving the edge sensors closer to the centerline of the foot, or drill out the two troublesome holes using a smaller drillbit and using the X-Acto knife to increase the hole to its desired size.

Figure 14. The smaller half of the foot after it has been cut.

2. Soldering the components onto the board

The electronic components need to be soldered onto the flexible board before it can go inside the foot. The electronic board as received from PCBWay can be seen in Figure 15. PCBWay provides a stencil to assist in soldering the components on. Solder paste should be used to help position the components on the board accurately. Heat resistant electric tape should be used so that the board can be heated in an oven to melt the solder paste. Afterward, the board should be soaked in acetone to remove excess solder paste.

Figure 15. Flexible PCB board, no components attached to it.

3. Assembling the foot

Once all the components are soldered onto the flexible PCB and both halves of the foot has been laid up and cleaned up, the flexible PCB can be epoxied onto the larger foot half. This was done using the same epoxy as was used to in a carbon fiber layup: Fibre Glast 2000 epoxy resin and the Fibre Glast 2060 60 minute epoxy cure. Epoxy was applied first to the sensors on the outside of the foot by the holes, then to the rest of the PCB on the interior of the foot. The epoxy was cured in an oven to speed up the process. It was discovered during this step that there was not enough space for the toe stub sensor, so it was carefully removed. This resulted in the foot seen in Figure 16.

The two halves of the foot was then combined using the same epoxy. The Author used electric tape to hold the foot together as the epoxy cured but found that the seal between the two halves were not as tight as it needed to be. This can be seen in the front and back of the foot in Figure 17. For future attempts, the Author would recommend using a vacuum bag and epoxy being applied only to the interior of the foot. Caution should be used to protect the sensors if a vacuum bag is used. However, using such a method would not allowed for the epoxy to cure faster in the oven.

Figure 16a. The flexible PCB epoxied onto the foot.

Figure 16b. The flexible PCB epoxied onto the foot, side view.

Figure 17a. The foot after the two halves have been epoxied together.

Figure 17b. The foot after the two halves have been epoxied together, profile view.

4. Pouring the Foam into the Foot

The foam on the inside of the foot is Fibreglast 6lb mix and pour foam. It was poured in through the ankle and Achilles holes before it started expanding. The mold should be held upright, and clamps should be used to hold the foot in place. A tube attached to a funnel was used to help assist this process of pouring the foam in. This process is described in greater detail in Ethan Kramer's Fall 2017 report.

During assembly of the foot, there were several issues that arose during this step. If there were any holes in the foot, the foam would leak out of those holes, as can be seen in Figure 18. Even if the gaps are seemingly taped shut with electric tape, the foam can still expand past the tape. This foam can expand to cover the pressure sensors, so each pressure sensor should be taped over to protect them from being potentially damaged. Specifically, this was an issue in the front and back of the foot. If excessive leaking occurs, the holes should be plugged as soon as possible to prevent the foam from expanded out through those holes and instead rise upwards throughout the foot. The excess foam can be removed using a file or dremel after it has cured enough, as can be seen in Figure 19. Caution should be used to not damage the pressure sensors in the process. If the ankle and achilles holes are not the exact size, then the foam may also leak out of the holes instead of filling the Achilles and ankle hole completely. This can be seen in Figure 20. Future work should be done to try to improve this step and prevent the foam from leaking out of the foot, most likely through ensuring that the two halves of the foot are epoxied more tightly.

Figure 18. The foot after foam has been poured in. The foam can be seen on the exterior of the foot because the two pieces of the foot were not vacuum sealed properly before the pouring of the foam.

Figure 19. The foot after the excess foam has been removed.

Figure 20. The ankle and achilles hole region after the 6lb foam has been poured.

5. Creating the Foam Shoe

The purpose of the foam shoe is to protect the pressure sensors on the foot as well as to distribute the pressure applied on the foot such that a sensor is able to pick up a reading when any point on the shoe is touched. The shoe is designed to be made in two layers. The first layer is made of Smooth-on Flex Foam-it! 17 and is at most 15 mm thick. The second layer is a denser polyurethane that covers the other layer and is designed to prevent the inner layer from being punctured. This denser layer should be made of smooth-on PMC-780 Dry because of its strong adhesion to the Flex Foam-it! 17 and favorable properties.

To create the foam shoe, the shoe making jig is used. It is intended for the Flex Foam-it! 17 to be poured into the foot and cured before another mold is used to get the PMC-780 Dry polyurethane onto the shoe. This second jig has not been designed yet, but should attempt to avoid air bubbles from forming. Jason suggested attempting to manufacture the harder polyurethane layer within a vacuum tank to prevent air bubbles from forming. The tank should be pressurized and a syringe used to inject the polyurethane into the foot. If it is injected from the bottom up, excess polyurethane will exit through the top of the mold and there won't be air bubbles that form.

The first attempt at manufacturing the complete foot and shoe used Smooth-on Brush-on 60 as the harder polyurethane because of time constraints. Smooth-on SO Strong Liquid Urethane Colorant was used to turn the Brush-on layer black; although mainly used for improving the aesthetics of the foot, it does demonstrate that additives can be added to the polyurethane. The brush on polyurethane did not adhere as well as the PMC-780 Dry. The thickness of the brush on layer was also harder to control for because of its method of application and the material itself not being very viscous. The Author found that the brush on polyurethane and the Flex Foam-it! 17 foam adhered better when the two were poured out before either of them set. The results of this test can be seen in Figure 21. The time intervals 5 minutes and 15 minutes were chosen because the brush on polyurethane has a pot life of 20 minutes.

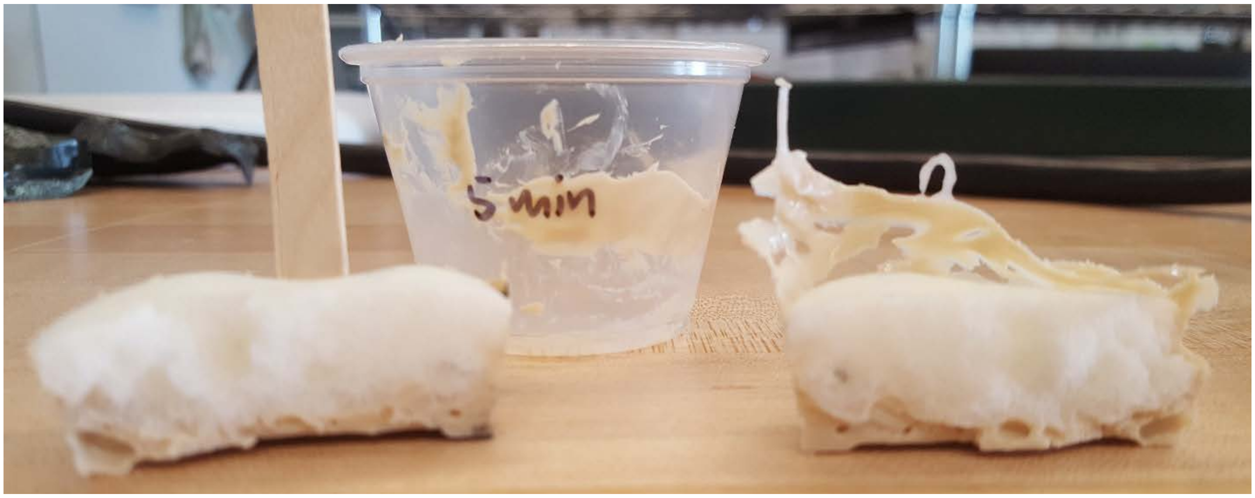

Figure 21a. The brush on polyurethane and the Flex Foam-it! 17 foam combined 5 minutes after the brush on polyurethane had sat for 5 minutes.

Figure 21b. The brush on polyurethane and the Flex Foam-it! 17 foam combined 5 minutes after the brush on polyurethane had sat for 15 minutes.

Because of the two shoe layers adhered better when added before the two set, the shoe was manufactured in the following fashion. First, a thin layer of the brush on polyurethane was applied to the bottom of the shoe making mold a day in advance. This was done to ensure that there would be enough of the denser polyurethane layer in the shoe. Then, after the rest of the foot has been prepared, another layer of the brush on polyurethane was applied to the mold. 5 minutes after the brush on polyurethane components have been mixed, the Flex Foam-it! 17 layer was poured into the foot. As quickly as possible after that, the foot is placed into the mold so that the foam can form as a shoe.

When the foam was poured, clamps were used to hold the mold together. This setup can be seen in Figure 22. The foam expand very quickly after the two parts are mixed, and unless clamped together, the foam would separate the two parts of the shoe making mold instead of rising as desired. The speed in which the foam expanded may be an issue; the foot was not aligned fast enough before the foam began pushing the mold outward.

Figure 22. The setup to create the foam shoe. Clamps are used to hold the two pieces of the mold in place.

CONCLUSION

The electronic circuit board was designed and manufactured, allowing for a complete assembly of the foot and shoe. Assembly of the foot was promising, but there is work to be done before the foot is a finished product. Although each individual step of the process was attempted prior, attempting to manufacture a complete foot showed many of the ways in which the foot assembly can go wrong. The things that went wrong are solvable issues, and future work should focus on streamlining and improving some of the manufacturing processes.

FUTURE WORK

- Refining the manufacturing process so that the foam pouring steps can happen much more smoothly.
- Improving some of the jigs and molds used so that cutting and pouring foam can be more precise.
- Modifying the foot design to have a little more space in the foot
- Use PMC-780 Dry with additives to turn it into a brush on material.
- Look into whether using FlexiForce sensors on the outside of the foot would be a better option

ACKNOWLEDGEMENT

The Author would like to thank Jason Cortell and Kate Na for their support during this project. Jason offered valuable feedback, guidance in the project, and supplies to making the project happen, and Kate was a willing learner and helped the Author do carbon fiber layups of the foot. The Author would also like to thank David Shi for continued use of his lab space and many resources required to do layups. The Author would also like to thank Andrew Sikowitz who wrote the Eagle script to streamline the circuit board layout process. Finally, the Author would like to thank Andy Ruina just for being an awesome person.

CITATIONS

1. Ruina, Andy. Safe inexpensive and robust legs for ubiquitous co robots. 2017.
2. Sfuptownmaker. “PCB Basics”. Sparkfun. <https://learn.sparkfun.com/tutorials/pcb-basics>