Carleton UNIVERSITY
Canada's Capital University

4th Year Capstone Project

iTAD Project Report: Fall 2018

MAAE 4907J

April 7, 2019

Revision 01

Supervisors:
Dr. Ala Qadi
Dr. Rong Liu
Dr. Peter Liu
Dr. Adrian Chan

Contents

List of Figures

Chapter 1

Mobility Team

The mobility team's goals and accomplishments.

Wire Management Through a PCB

Student: Arsalan Barolia
Group: Systems

1.1 Purpose

This report will explore the implementation and advantages of a PCB, Printed Circuit Board, on the robot. The iTAD robot's main goal is to be placed in a hospital that can walk freely and give patients their medication correctly via face detection.

Initially, it was proposed to implement all the components from the previous years, along with the current year, onto a big PCB however, that has been altered after consulting with Professor Qadi. After updating the communication schematic with Marcello, it was observed that the current communication wiring state of the robot is disorganized and hard to follow and needs to be improved.

For a better insight, majority of the communication wires are tangled with one another and is hard to see where certain wires are connected as some do change colour mid-way. Some wires can also be easily disconnected when a small external force is applied to the wire. If an unwanted connection occurs, the robot may malfunction and put everyone around it in danger. This will be further explored in later sections.

Thus, the new scope for this year is to implement a smaller PCB that would mainly focus on organizing the communication wires between the logic shifter and the PWM. This will aid current year students as well as the students in the coming years to have an easier time modifying/making connections while ensuring that no unwanted disconnections occur.

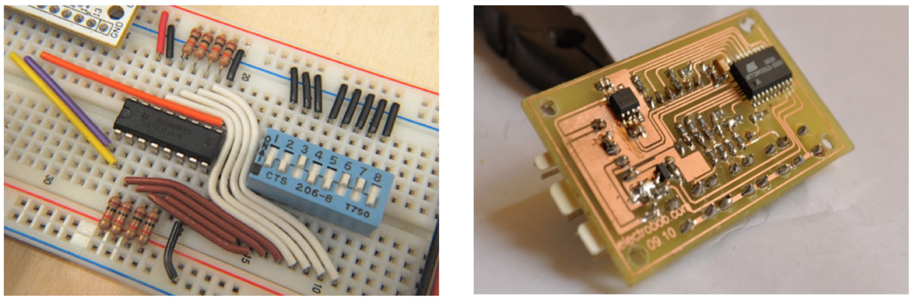
1.2 Discussion

In this section of the report, different aspects of the usage and functionality of a PCB will be discussed as well as the advantages it brings. In addition, this section will discuss where and how the PCB will be implemented onto the robot and the crucial steps required to accomplish this.

1.2.1 What is a PCB?

When talking about electronics, PCBs are referred to as **P**rinted **C**ircuit **B**oards. These types of boards are internal wires that connect and supply power to the electrical components mounted on it.

Before the invention of PCBs, industries and electricians relied on point-to-point wiring systems which often resulted in failures at the wire junctions causing the circuit to short when the insulation of the wire began to age and crack [1]. An easy way of picturing a PCB is by thinking of it as a condensed breadboard – instead of having the wires on the top of the board, they are integrated within the board. Figure 1.1 below helps visualize the differences between the breadboard and a PCB.



(a) (b)

Figure 1.1: Wire location of a Breadboard (a) and a PCB (b)

Various applications of where PCBs are used include smart phones, computers, speakers, clocks, etc. PCBs are mainly used anywhere where an electrical circuit is present. Breadboards are more used now for experimental purposes which are then turned/manufactured into a PCB once the design and schematic of the circuit is tested and finalized.

1.2.2 Layers of a PCB

A PCB is composed of 4 main layers of different materials – substrate, copper, solder mask, and silkscreen – that are laminated together using heat and adhesive. In Figure 1.2, the location of the 4 layers are indicated and then later discussed.

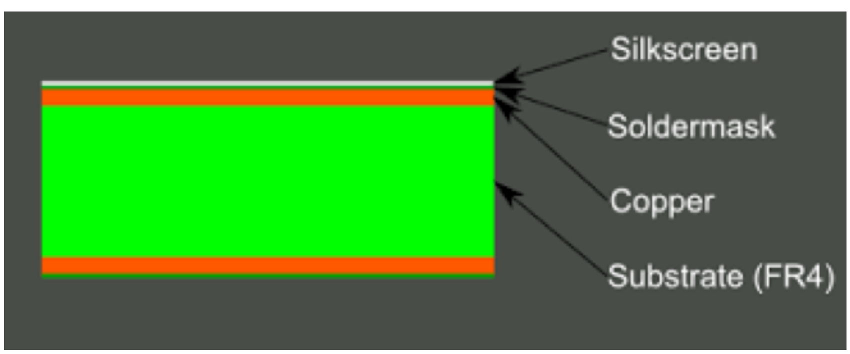


Figure 1.2: The 4 main layers used in a PCB

Substrate

The substrate is the base of the PCB and fiberglass is mainly used. In Figure 1.2, FR4 refers to a glass-reinforced epoxy laminate material that is often used in PCBs. FR4 boards are known for their durability and thermal conductivity however, these boards are not known for PCBs that produce a lot of heat due to its lack of dissipating the heat away and can potentially damage components on the board [2].

In a PCB, one of the main reasons it fails is heat – besides dust and moisture. Heat can be caused through environmental factors or by inputting any amount of current through a PCB. This is why layers in a PCB are introduced. High voltages demand more layers in a PCB to help keep the board composed and not burn up. A general rule of thumb that industries use is that if a PCB is provided with high amounts of power, instead of the FR4 substrate, materials like ceramic and other metal core PCBs should be used even if additional layers are added [3].

Furthermore, the durability of a PCB is determined by the substrate used. For instance, a low-cost PCBs will use materials like epoxies and phenolics which are not known for their durability but have a lower cost when compared with the FR4.

Copper

The next layer in a PCB is a thin sheet of copper. Most PCBs are double sided, thus copper is laminated, using heat and adhesive, to both sides of the substrate. Essentially, the number of sides a PCB is correlated with the amount of copper layers it has. A typical PCB ranges from 1 to 16 layers. Copper thickness is important. Most PCBs have 1 ounce of copper per square foot but PCBs that are designed to handle high power usually use 2 to 3 ounce of copper [4].

Solder Mask

The solder mask is the layer that is placed on top of the copper foil. This gives the PCB their green look and helps PCB manufactures to solder correct locations of the PCB without producing solder jumpers which are the main cause for shorting a circuit.

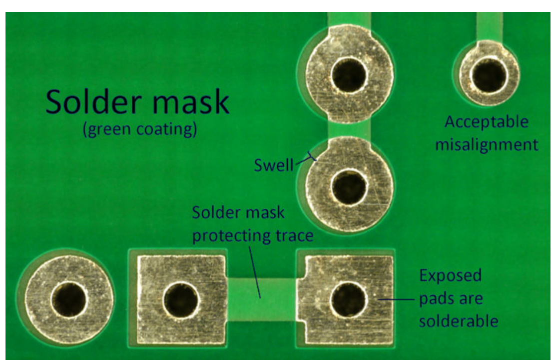


Figure 1.3: Solder mask prevents solder jumpers and has gives it a green look

Silkscreen

The silkscreen is the white writing that can be found on PCBs to help the manufacturer locate where the components are supposed to be placed instead of always referring to the schematic. This also helps the customers identify what each component is on the board once completed. Overall, it helps the user gain a better understanding of the board's components.

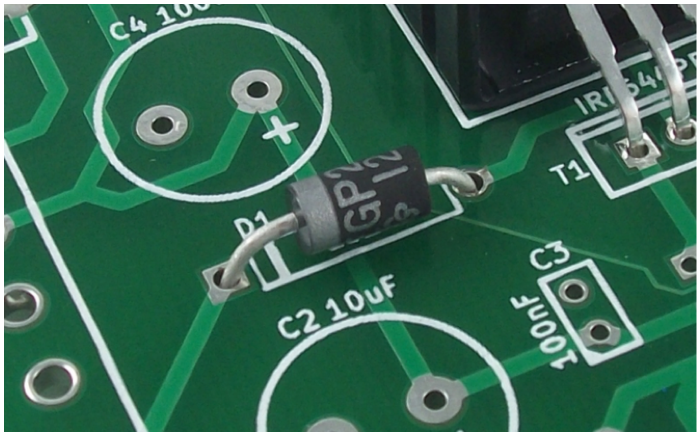


Figure 1.4: Writing in white to help locate different components on the PCB

1.2.3 PCB Manufacturing (DIY vs. Company Manufactured)

Manufacturing a PCB at home as oppose to a company manufacturing it for you can be a huge factor and depends on what the final goal entails. For example, an individual that is making a PCB for fun and learning purposes are better off manufacturing the PCB at home as it will have a lower cost avoiding shipping and handling costs. However, an individual that is making a PCB for an important project, the iTAD robot in this case, would be beneficial to get the PCB manufactured by a company as the connections are more reliable, durable, and professional looking. These are important features to keep in mind knowing that the iTAD robot will be used and modified in future years.

There are certain steps that are taken when making PCBs at home. The steps are as follows [5]:

- Step 1: Develop a schematic of your project using a software (E.g. EAGLE or Circuit Maker)

- Step 2: Convert the schematic to a board layout allowing you to choose where the components will be located and the wiring for them.

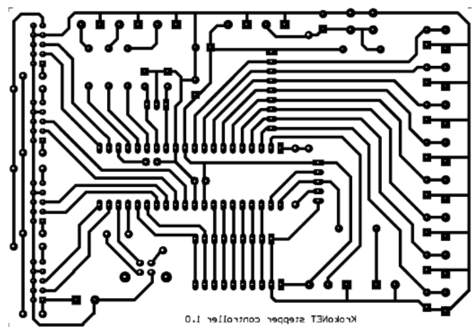
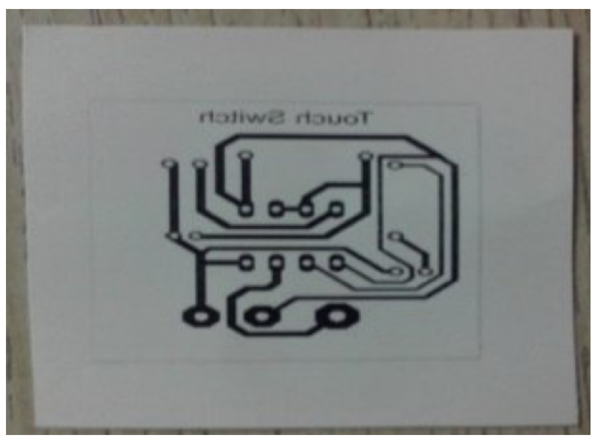


Figure 1.5: Example of a conversion from schematic to board layout

- Step 3: Using a laser printer, print it on a Peel-n-Press paper. The schematic should be on the glossy side of the paper.



- Step 4: Cut the copper clad – copper covered steel – for the circuit board.

- **Step 5:** Clean copper clad using steel wool.

- **Step 6:** Iron the circuit from the Peel-n-Press paper onto the copper clad.

- **Step 7:** Etch the plate using ferric chloride.

- **Step 8:** Drill holes and solder electrical components.

To reiterate, making a PCB at home could result in the PCB being less reliable, durable, and messy in comparison to PCB manufactured by a company.

1.2.4 Benefits and Advantages of a PCB on iTAD

There are numerous advantages of having a PCB on the iTAD robot that assists with wiring organization. For instance, misplacement of wires will be avoided as many of the wires will be integrated into the board itself. The PCB can also be manufactured in a way where unwanted disconnection of components is prevented, with clips or clamps, to ensure that the robot is always functioning properly with all its components attached. This feature would have also prevented the problem that occurred over the summer where the robot's microprocessors and other components were fried due to incorrect connections of wires.

In addition, by removing the loose/free wires hanging from iTAD, specifically between the logic shifter and PWM, to make the robot look more professional, clean, and organized. The logic shifter and PWM concept will be revisited in the Current State to Vision State section.

This will be very important for a robot, especially in an environment like a hospital, as it implies that work, time, and thought was put into this robot and not just different electronics that were put together randomly. This can also help with patients, visitors, and doctor's confidence in the robot to do what it is intended to do and not malfunction.

1.2.5 How These Goals will be Accomplished

EAGLE vs. Circuit Maker

In order to make the schematic on the computer, EAGLE and Circuit Maker are 2 programs that have been highly used/recommended. Each of these programs have different features and advantages/disadvantages that make them unique from one another.

To begin with, both programs are free and can produce detailed block designs from the libraries provided by the company. Both also include footprints of different schematics used by other people in the community. However, users have complained that Circuit Maker is filled with unusable footprints that are either corrupted or not useful. Another issue many individuals have mentioned, for Circuit Maker, is that the program is unstable and unpredictably crashes.

With the assistance of Marcello, the updated communication schematic below, Figure 1.6, was produced using Circuit Maker as the students from the previous year used it to make the initial schematic. It should be noted the arm and base connections are not included in this schematic as the group responsible for the arm and motor are working on it. However, the model for the motor controllers for the arm and base are 2x ESCON 70/10 and 2x ESCON 50/5, respectively.

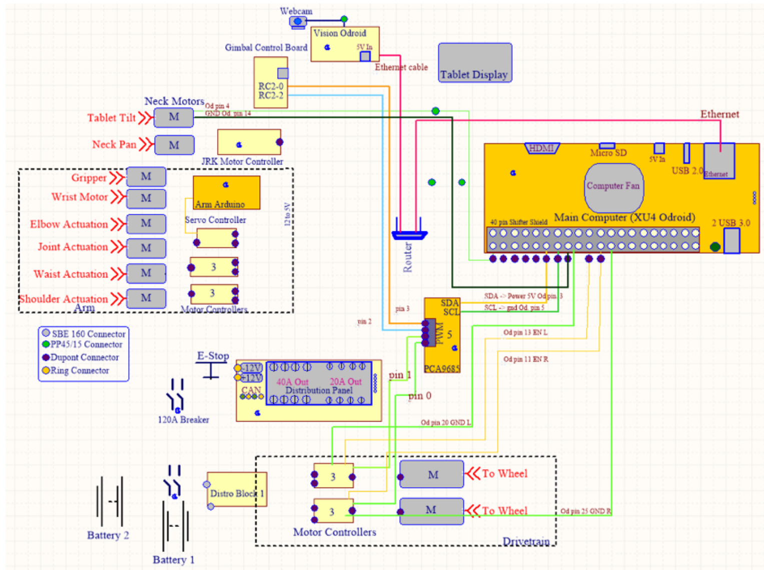


Figure 1.6: Updated comm. schematic without arm and motor connections

The colours of the wires are closely resembled to the colours of the current wires used on the robot however, not all the colours were found on Circuit Maker. In other words, it is not an accurate representation of the wire's colours on the robot to the schematic. While updating the schematic, it was noted that although the program was easy to learn and use, it started to lag frequently once more components were added to the screen.

Furthermore, EAGLE is known for having a learning curve for using the program but has many features that can be used once learned. For instance, a SPICE Simulator and Rule Checking feature is implemented into the program to ensure the validation of the connections in the circuit and to check that all the wires are connected to a component and no open wires are present [6]. One problem with EAGLE is that although there is a huge selection of libraries to choose from, not all the components are added to the software. This will result in finding specific components in different online libraries and import it into EAGLE for use.

Since the PCB will be manufactured through a company, PCBway was a company that was recommended, by Professor Qadi, for their fast response to orders and shipping time. After talking to an associate at PCBway, it was pointed out that that PCBway recognizes EAGLE's Gerber files whereas there are some compiling issues when it comes to Circuit Maker's Gerber Files. Gerber files are essentially 2D binary images that provide the PCB manufacturing company with your board layout that identifies the location of your components and wire traces [7].

After considering the advantages and disadvantages of each program, it was decided that the PCB will be done using EAGLE despite its learning curve. Overcoming the learning challenges and utilizing the features provided will be beneficial for creating a successful PCB that works properly without destroying any of the components on the robot. In Figure 1.7, a summary of the key points discussed in this section is put into a Venn diagram.

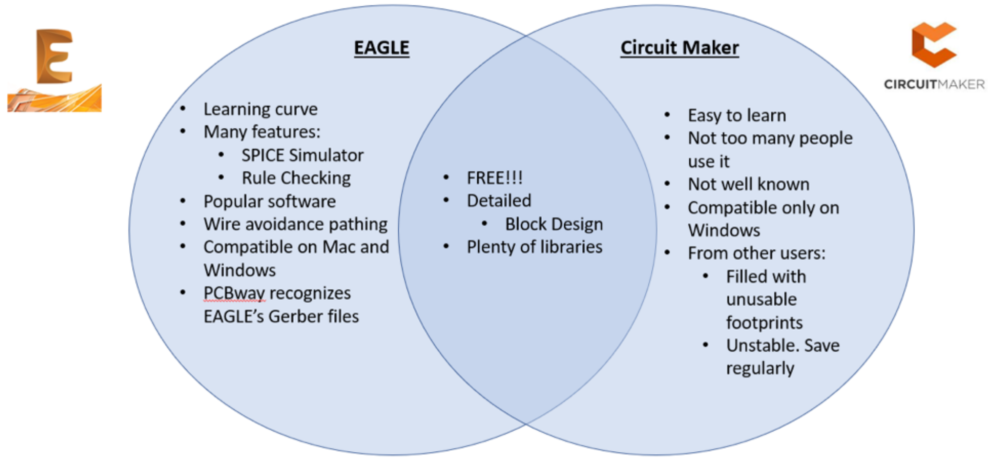


Figure 1.7: Comparison between EAGLE and Circuit Maker for PCB wiring

Current State to Vision State

From Figure 1.6, the wiring of components can be easily distinguished and seen. However, even with the assistance of Marcello, this was not an easy task to accomplish. The reason being is because the wires are tangled with one another and some wires also did get lost into a wire compression tube that changed colour mid-way through. Figure 1.8 illustrates the current wiring of the iTAD robot.

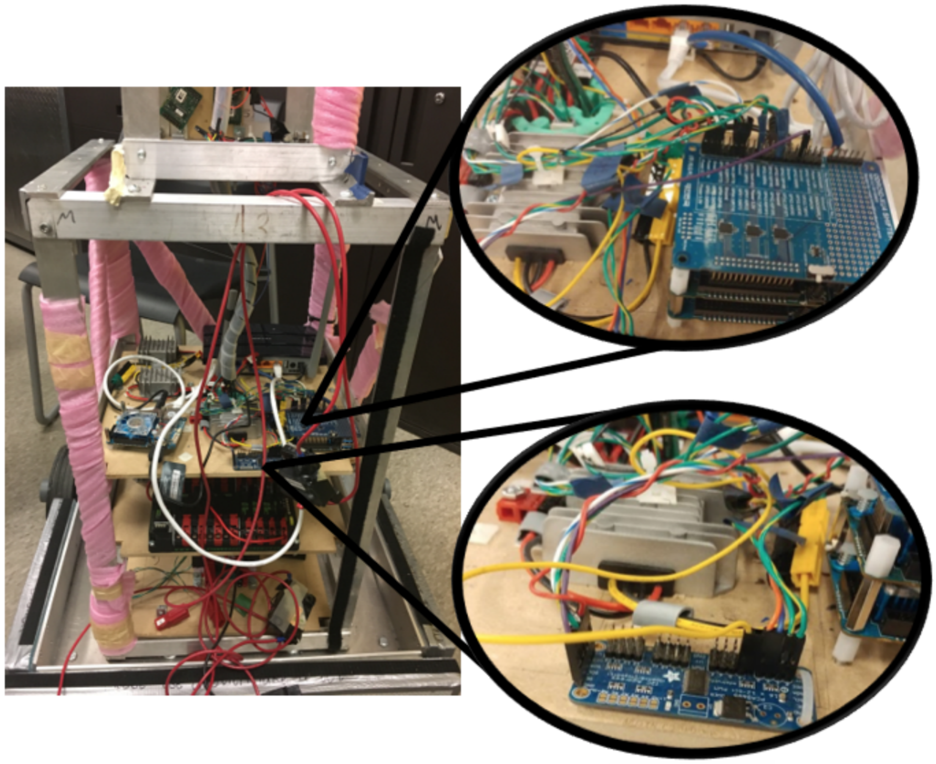


Figure 1.8: Current wiring state of the iTAD robot

Due to the recent change in scope, a proper schematic has not yet been produced but a visual, in Figure 1.9, will help capture key aspects and ideas of what the PCB will contain.

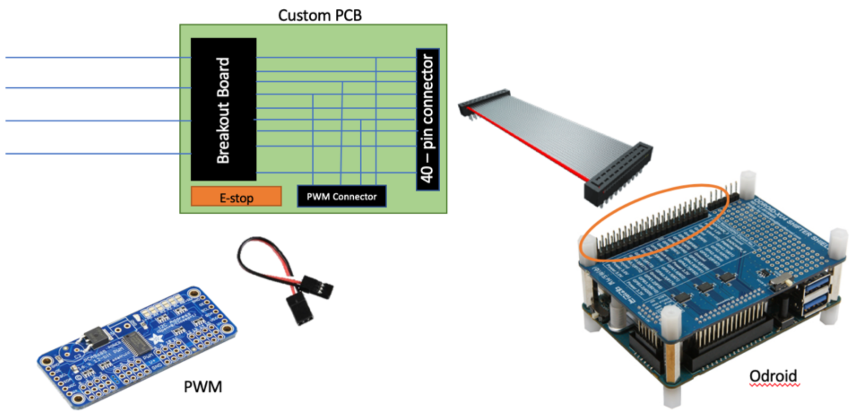


Figure 1.9: Vision of organizing connections between logic shifter and the PWM

In Figure 1.9, a 40-pin ribbon cable from the Odroid logic shifter will be connected to the 40-pin connector on my PCB. From here, the specific wires that are currently connected from the Odroid logic shifter and PWM will be established. A breakout board will also be added on the PCB for current and future students. A breakout board is important because the pins that are not in use, as seen in Figure 1.8, will be gone to waste if there is no way of connecting to those pins. Furthermore, an E-stop port is still currently under review to see if Marie, who is working on emergency stoppage of the robot, can directly talk to the PWM or if a designated port will be needed for that communication. There are multiple ways of implementing this idea, like mounting the PWM onto the custom PCB however, it is not yet confirmed what advantages and disadvantages arise from this method and will have to be further explored.

PCBway vs. Carleton Electrical Department

As mentioned above, PCBway is a company that Professor Qadi had recommended for their fast response and order delivery. Currently, an email was sent to Nagui Mikhail, Computer Systems and Physical Facilities Manager, to see if the Department of Electronics, here at Carleton, make PCBs. This will be beneficial as if an issue or problem does arise in the PCB, shipping and communication time will be saved as the Electrical Department can be reached in person rather than a company whose main source of communication is through email. It was concluded that the Electrical Departement does not make PCBs and the PCB needs to be manufactured by PCBway.

1.3 Implications and Changes

In this section, the changes that were made to the PCB will be mentioned along with the board layout that was created after the schematic was produced.

The initial idea was to have a 40-pin terminal header that will perfectly fit on the board. However, after researching for the component, the lead time for shipping was 16 weeks and, due to time constraint, this was not an option. Therefore, a new component needed to be researched. A 40-pin female header could have simply substituted this problem however, the wires could be easily removed if pulled/tugged on.

After thorough research, another screw terminal part was found. However, this screw terminal has a pitch of 2.54mm, which is in-sync with the schematic, but instead, the component found only has 10 positions instead of 40 that is required. It was concluded that 4 individual parts would need to be purchased to accomplish the goal for the PCB. This idea was passed and accepted by Professor Qadi. As seen in Figure 1.10, this change was reflected on the schematic and Table 2 is the update list of components.

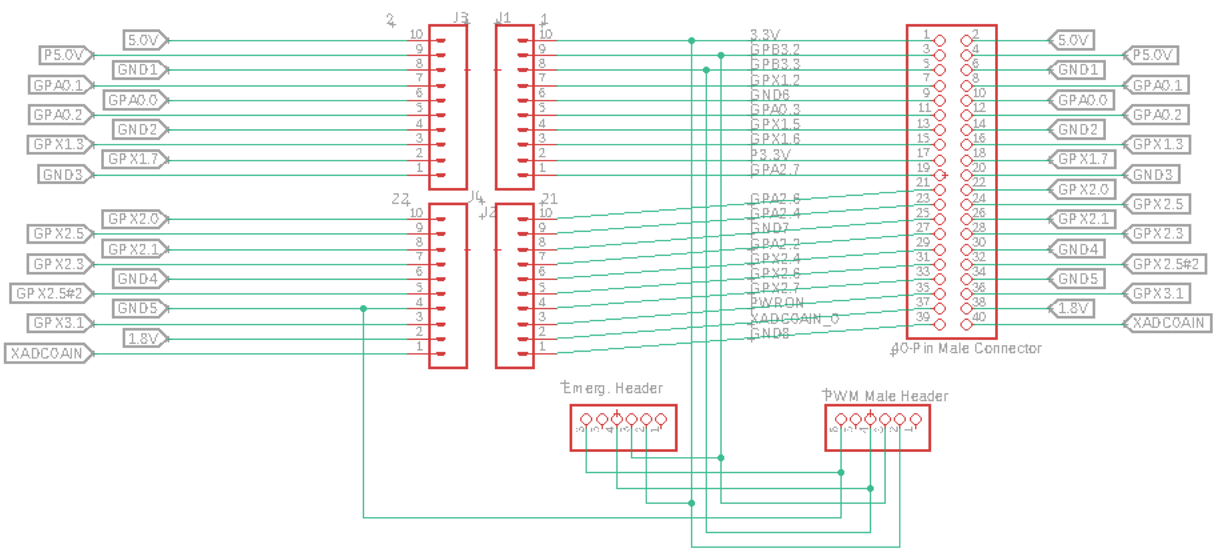


Figure 1.10: Updated Schematic with 4 separate 10-pin screw terminal

Table 2: Updated list of components that will be mounted on the PCB

Component	**Part No.**	**Picture**
40-Pin Male Header	S1012EC-40-ND	
Breakout Board x4	ED10567-ND	
6-Pin PWM Header	609-3263-ND	
6-Pin Emerg. Header	609-3263-ND	
40-Pin Ribbon Cable	571-1658623-9	

In the most left and right of the schematic, there are no wires that display a physical path to where it is connected. The reason for this is because they are currently virtually connected with the help of Flags. Virtually connected is defined as a connection that is not physical pathed to a specific location but rather a logical path that is connected to a specific location [8]. This is can only be done when making a schematic and is beneficial as it makes the schematic look neat, organized, and easier to follow by other individuals.

The labels of the flags, in Figure 1.10, are correspondent to the labels that are found on the Odroid logic shifter as illustrated in Figure 1.11.

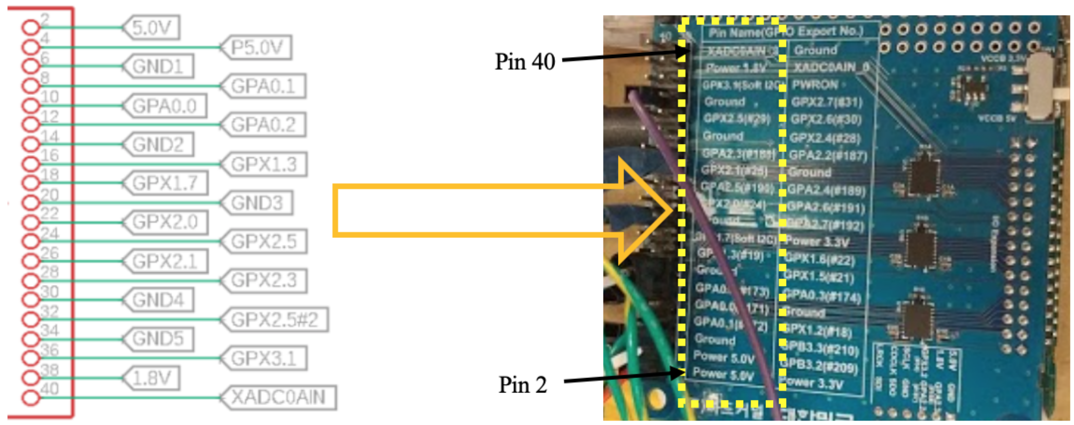


Figure 1.11: Flag labels correspond to labelling on Odroid based on pin number

In addition, the pin numbers are also matched to ensure consistency. In other words, Pin 1 connects to pin 1, pin 2 connects to pin 2, and so on. This helps future students to easily comprehend how all the components are connected and where they are situated on the schematic. After these changes were shown on the schematic, the board layout was produced. It was important to ensure that no connections were left unconnected and that no wires are touching each other. If connections were to touch each other, the circuit could be shorted resulting in the PCB not working.

In Figure 1.12, a schematic with the overall board layout but without the physical connections can be seen.

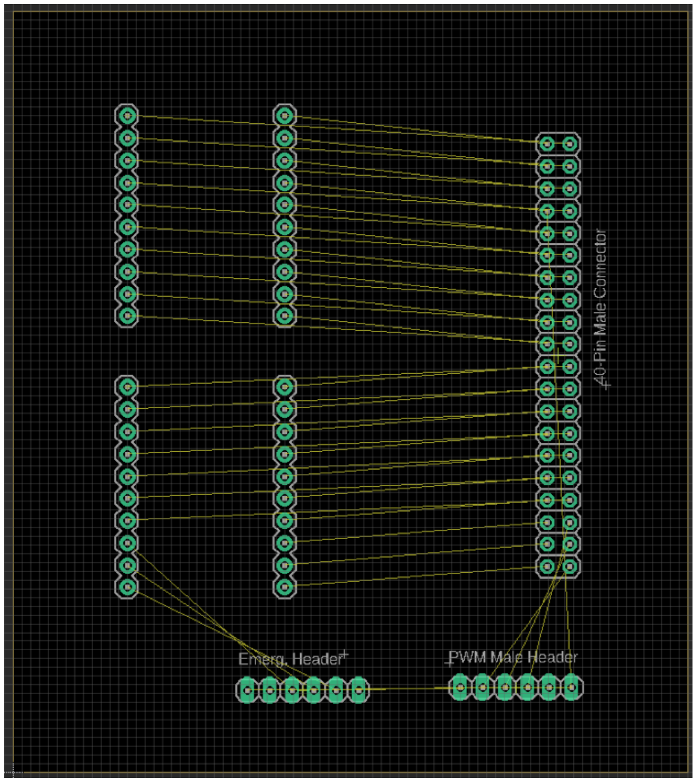


Figure 1.12: The overall board layout with no physical connection

The grid size was put into units of millimetre (mm) to help have a better visualization of the spacing between components and the board size. Once the components were put in place, the basic net classes were entered to standardize all wires and connections to have the same attributes, lengths and properties. An image illustrating the changes and values are in Figure 1.13.

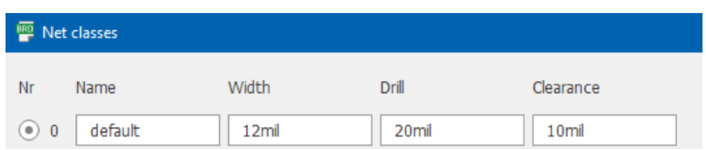


Figure 1.13: Standardized settings for the board

Once these settings were entered, the wiring was conducted manually to ensure all connections were connected correctly and that no wires were touching each other. When researching, it was said that a faster signal is sent when the wire is put at a 45-degree angle when changing directions of the wire as opposed to a 90-degree angle [8]. In Figure 1.14(a), a 90-degree angle is depicted and in Figure 6(b), a 45-degree angle connection.

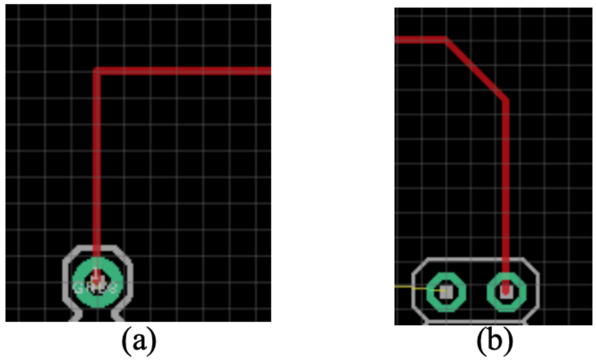


Figure 1.14: (a) A 90-degree and (b) a 45-degree connection of wires

A 45-degree angle is better than the 90-degree connection because the signals that travel along the lines gradually change directions, on the 45-degree angle, instead of an abrupt change in direction. Knowing this, the connections were made on the PCB and a finalized board layout was produced, Figure 1.16.

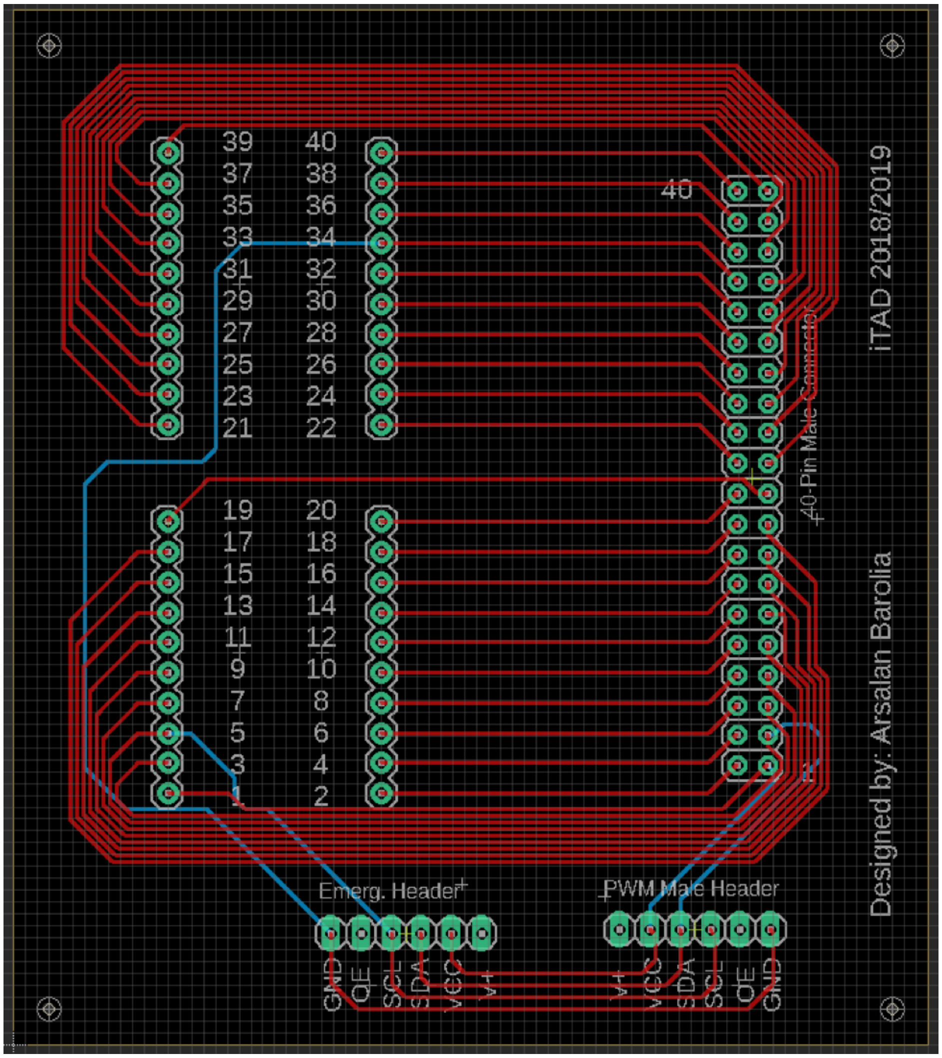


Figure 1.15: Finalized board layout along with its physical connections

Labels and text were also added using the text function on EAGLE that could be seen on the silkscreen layer. This writing helps indicate the person using the PCB where all the pin numbers are to avoid any errors in the wiring. Please note that the red wire in Figure 1.15 was routed on the top layer of the copper whereas the blue indicates the connections at the bottom layer. This helps organize the different wires without having too many wires on one layer. Once the board was produced, the ERC (Electrical Rule Check) and DRC (Design Rule Check) was used to check that no additional or unnecessary wires were connected on the schematic and the board, respectively. No errors were found. The schematic and board were reviewed and checked by Professor Qadi and sent to PCBway for manufacturing. Figure 1.16 displays the specifications that were advised to PCBway to help manufacture the PCB based on the Gerber files sent.

Product Detail

Product Detail

Product No. :	W232382ASH1	**Gerber File :**	W232382ASH1_Schematic Final Gerber Files.zip		
Board type :	Single PCB	**Panel Way :**			
Size :	77 x 87 mm	**Quantity :**	10	**Layers :**	2 Layers
Material :	FR-4 TG130	**Thickness :**	1.6 mm	**Min Track/Spacing :**	6/6mil
Min Hole Size :	0.3	**Solder Mask :**	Blue	**Silkscreen :**	White
Gold fingers :	No	**Surface Finish :**	HASL with lead	**Via Process :**	Tenting vias
Finished Copper :	1 oz Cu	**Additional Options :**			
Create Time :	2019/1/22 02:58:17	**Build Time :**	2-3 days	**Estimated Finish Time :**	2019-01-24 China Time Zone(GMT+8)
Manufacturing :					
					Total: **US $5**

Figure 1.16: Product detail of the PCB sent to PCBway based on the Gerber files

These specifications were chosen and selected based on the Gerber files sent. Gerber files help manufactures know exactly what is on every layer of your PCB and the specifications you set during the design process. Figure 1.17, below, depicts an example of the Gerber file for my board layout.

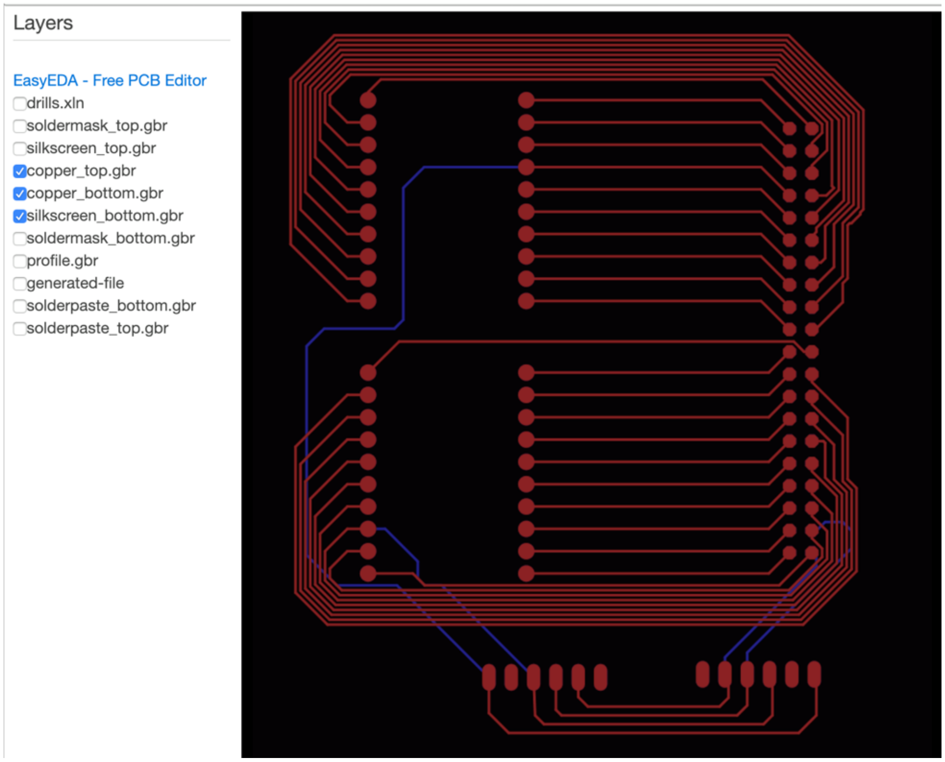


Figure 1.17: My PCB Gerber file using an online Gerber viewer

This Gerber file was viewed using an online Gerber viewer. It is very important to view your Gerber file using an online viewer or another software as some compiling issues do occur. This allows the designer of the PCB to verify that the connections on the PCB are correctly routed and that the manufacturer knows how the PCB needs to be produced.

Once this order was processed, the PCB was shipped and delivered along with its components, in 1 week and 2 days, respectively. In the figure below, Figure 1.18, an image of the PCB manufactured is shown.

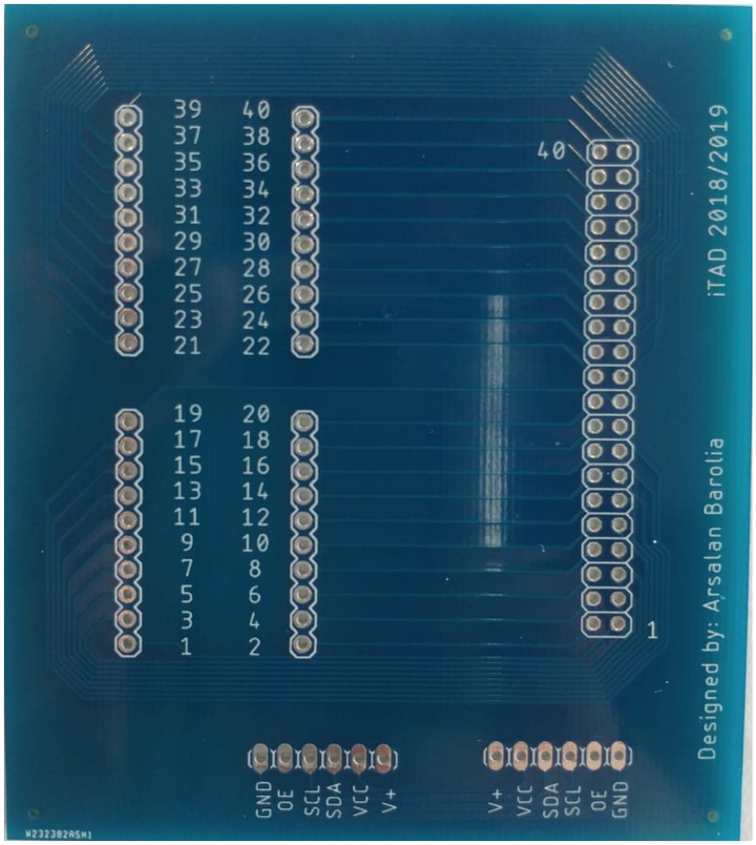


Figure 1.18: Manufactured PCB from PCBway

Professor Qadi mentioned that the PCB may experience a presence of 3.5V and I need to ensure that my PCB can handle that. To do this, I will need to verify if 3.5V will be running though the PCB. It was found that the 3.5V was not passing through the PCB, thus no changes were made and the PCB was ordered.

1.4 The Board

In this section, the testing of the connectivity of the board and the process as to how the components were mounted will be discussed.

Before mounting all the components received from Digi-key, all 10 boards were tested for their connectivity to ensure that no wires are shorted and that no components are destroyed once the PCB is implemented on iTAD 3.0. The way the PCBs were tested was using a voltmeter from one pin port to another port that it is routed to and check to see if the voltmeter gives a reading of near 0. This did become a tedious method as 10 boards needed to be tested the same way, however, this testing is crucial as one connectivity issue can ruin other components on the robot. All boards were tested to for the sole purpose of ensuring that if the current board being used needed to be replaced, it would be switched with another functioning board. Also, it was discovered that 3.3V was being introduced, from the head motor, to the PCB which was not considered for. To address this issue, Marcello and I were able to re-route the wire so that the manufactured PCB is solely responsible for communication signals between the PWM, Odroid, and motors. After all the boards were tested, the components received from Digi-key were mounted. This was done using a portable vent, board solder holder, lead free solder, tip tinner, solder sponge, and a solder vacuum pump. In Figure 1.19, the components can be visually seen.

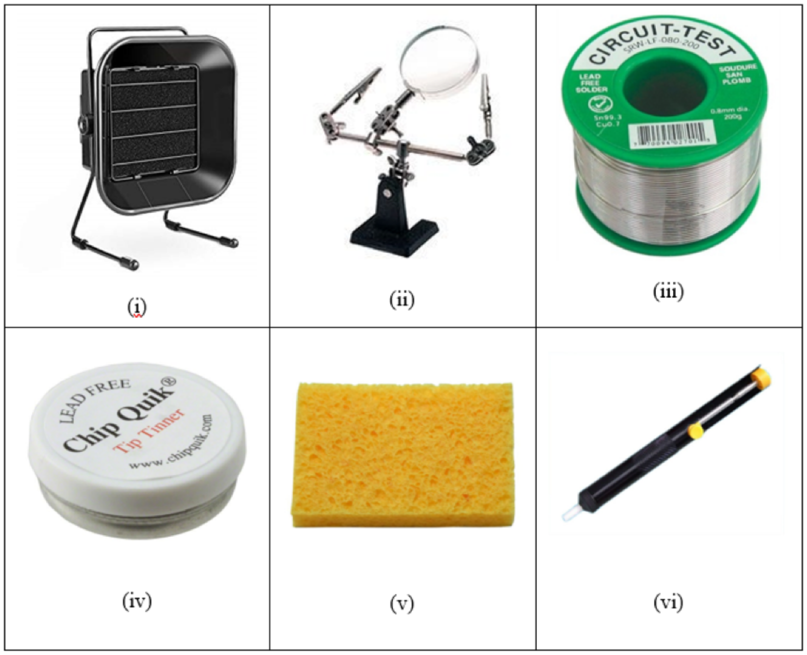


Figure 1.19: (i) portable vent, (ii) board solder holder, (iii) lead free solder, (iv) tip tinner, (v) solder sponge, (vi) solder vacuum pump

The process for this was to turn on the portable vent to allow the scent and fumes from the solder to be ventilated so the user does not inhale it. Next, the solder iron was plugged in ensuring that the iron was in its respective holder and not lying around on the table. While the solder was heating up, water was poured onto the sponge. Note, the soldering iron was not left unintended. Always ensure that you near the soldering iron when turned on/heated. In addition, the board PCB, in Figure 1.19, was clamped onto the solder holder to prevent the board from moving when soldering. Moreover, the tip tinner was used to remove all the baked-on residue as it also helps with the prevention of oxidation, which is the black burnt solder that can often be seen on the soldering tip when not in use. Once all these steps were completed, the components, as seen in Figure 1.20, was mounted on the PCB which can be seen in Figure 1.21.

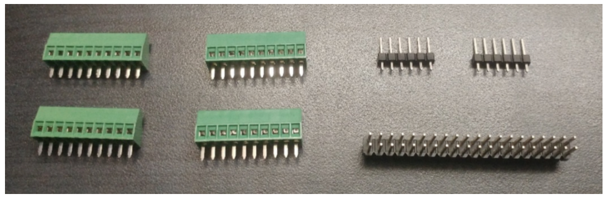


Figure 1.20: Components received from Digi-key

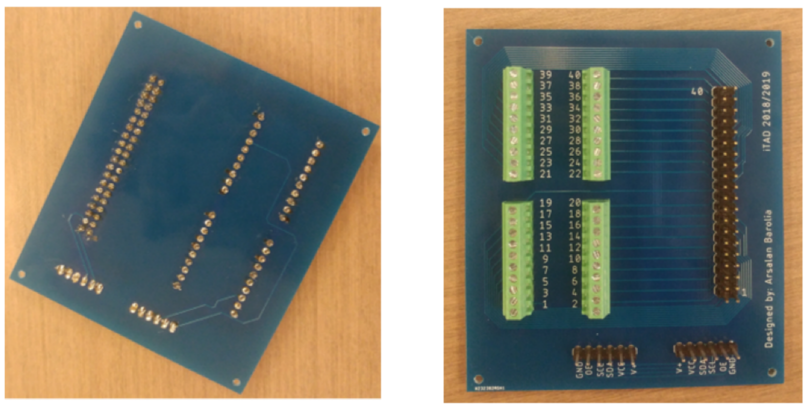


Figure 1.21: Mounted components on the PCB

1.5 The Implementation of the Board

The last step for implementing the board on the robot is to ensure the external wires are accounted for. A 40-pin ribbon cable, for the Odroid to the PCB, was bought from Mouser Electronics however, the 2 6-pin ribbon cables needed to be made, by hand, as it was difficult finding electronic stores that sold them separately. The ribbon cables were accompanied by other electronic components which caused the price to increase. With the addition of the shipping and handling cost, the price for the ribbon cables was not ideal. In order to make the 6-pin ribbon cables, the role of 1.27mm rainbow color flat cable was ordered. The figure below, Figure 1.22, depicts ribbon cable used along with its identifying name on Amazon to allow repurchasing.

Figure 1.22: The 10-wire flat ribbon cable used

A wire stripper was then used to cut to the desired length so that no excess wiring was present when the final connection was made on the robot. Next, an X-Acto knife was carefully used to remove the additional 4 wires that came with the role, because we only need 6 wires. Furthermore, since the gauge of the wire purchased was large, the diameter of the wire was small which required a more specific tool for stripping the wire which was not present in the toolbox of the lab. To resolve this issue, and X-Acto knife was still used to split the wires and carefully and precisely strip a portion of the insulation. Figure 1.23 provides a view of the how the wire looked at this point of the process.

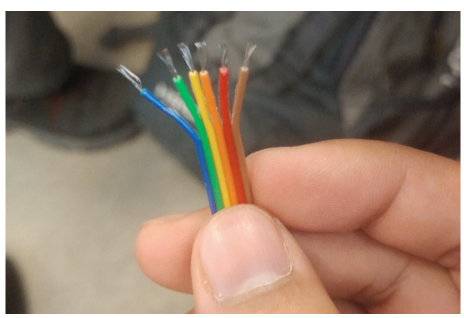
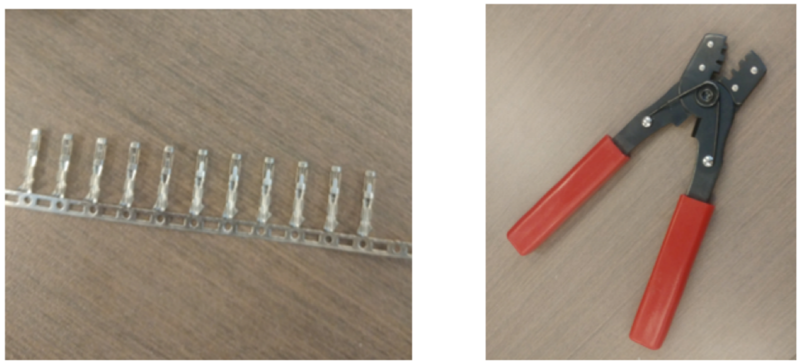


Figure 1.23: Visual of split wires and precise stripping with an X-Acto knife

Moreover, DuPont connectors were used to wrap around the wire. This allows the wire to be thick enough to place into the black blocks while still maintaining its connectivity. In Figure 1.24, a strip of the DuPont connectors along with the clamper tool used is shown.

Once the component and clamper tool are located, the DuPont connector is wrapped around the wire such that the first flap wraps around the insulation of the wire and the second flap wraps around the wire itself. The clamper tool is then used to squeeze the flaps around the wire. Ensure that the wire and its respective DuPont connector are tightly connected so that it does not easily slip off. Figure 1.25 depicts this process for the first 2 wires and Figure 1.26 shows the final product once the DuPont connector block is equipped.



(i) (ii)

Figure 1.24: (i) DuPont connectors, (ii) Connector wire clamper

The block is simple applied by feeding the DuPont connectors in the holes of the block.

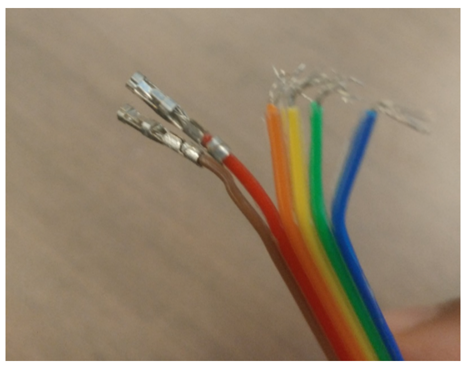


Figure 1.25: First 2 wires with the clamped DuPont connectors

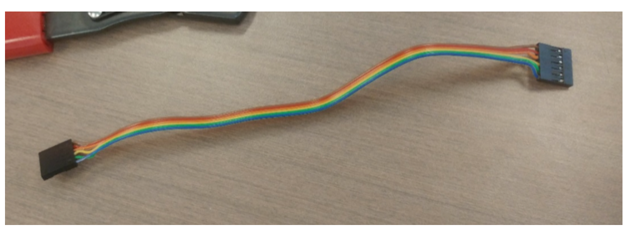


Figure 1.26: Completed 6-pin ribbon DuPont connector

Lastly, the voltmeter was used again to ensure that the connection from one end to the other end was close to 0. This process was then repeated as 2 6-pin ribbon DuPont connectors were required. The second one was to consider Marie's project, related to the emergency stop button, however, she no longer needs it and the extra wire will now be used if the current ribbon cable stops working or needs to be repaired. Figure 1.27 shows the PCB on the robot after all the previous connections were made.

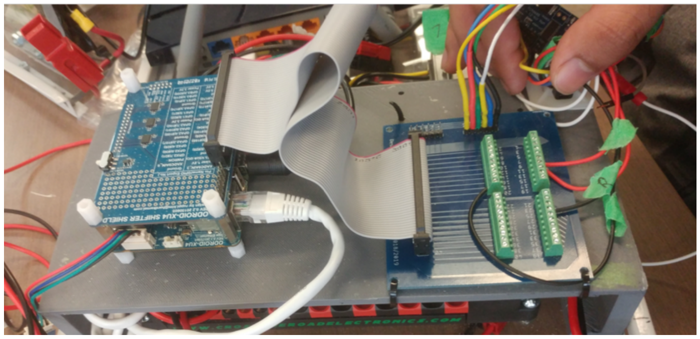
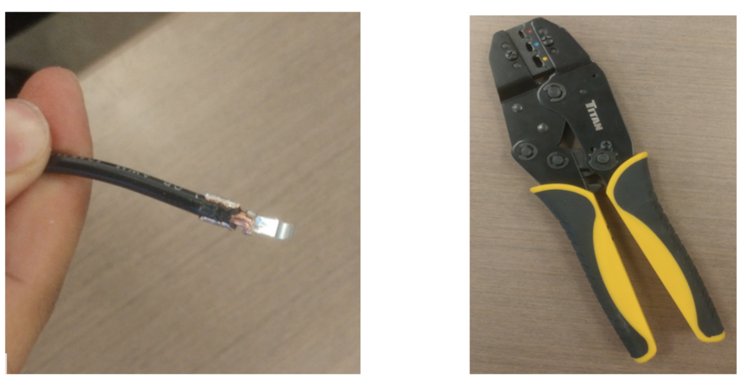


Figure 1.27: Final implementation of the board on the robot

1.6 Extra Work Regarding Crimping

After the implementation of PCB, I was seeking in helping others fulfill their project to ensure we had a functioning and upgraded/modified robot. I found myself helping Marcello in creating extension cables from the battery panel/shelf to account for all the extra sensors being added by other students. To do this, the same process was done as described in the "The Implementation of the Board" section which referred to clamping. The only difference is that instead of making DuPont connectors, the PP15 reel connectors were used and clamped so that a Molex header can be attached to it. In addition, a 12-gauge wire was used and clamped which can be seen in Figure 1.28. This figure also displays the different clamping tool used as the connector is too big for the clamping tool shown in Figure 1.24 (ii).



(i) (ii)

Figure 1.28: (i) PP15 reels on a 12-gauge wire, (ii) Molex clamping tool

1.7 Problem and Implications

Even though all the PCBs were tested before and after plugging the components and existing connections from iTAD 2.0, the software was having troubles in communicating with the

PWM. Only one of the wheels on iTAD 3.0 would move while the other remained static. Marcello and David tried many various tests and adjustments, but are not able to distinguish why this problem is occurring. All the connections are providing an appropriate readings but when the software starts to run, an error occurs. Since the Odroid is properly functioning and the PWM turns on as well, David believes that there is a lot of signal interference on the I2C bus of the PWM as the length of the ribbon cable is long. Due to time constraints, we have replaced the PCB with Carmen's obsticle avoidance Arduino, as it has a built in PWM, allowing us to remove both the PCB and PWM. This was not done in the beginning of the year as we were trying to build upon what was done the year before us. However, for future work, I do recommend that the length of the ribbon cable is reduced to limit the interference in the signal so that the PCB can be used as a backup if something was to go wrong with the arduino.