Student Circuit Board Design and Manufacturing
Two years of independent projects and the lessons learned

Gabriel Russ
Department of Electrical Engineering and Computer Science
Ohio Northern University
Ada, OH 45810
g-russ@onu.edu

Proceedings of the 2018 ASEE North Central Section Conference

***Abstract*—While all electrical engineering students are taught extensively in how various electrical components operate, it is difficult for interested students to attempt to manufacture their own projects. In order to gain a greater understanding of the process taken to design and manufacture professional circuit boards, I've spent the last two years building them into my own personal projects. This paper documents multiple projects over two years, detailing steps taken in the design and manufacturing process. It also outlines lessons that can be taken from difficulties in the design process, allowing for future students to avoid making similar mistakes.**

I. Introduction

All of the electrical engineering classes students take involve lengthy study of theoretical elements of electronics. Learning how MOSFETs and communication systems theoretically is of course important, but motivated students can run into issues with bringing their projects into the real world. While a multitude of advice exists on the internet in how to design electronics projects, the vast majority of this knowledge is presented through the lens of a specific project, without any overarching lessons. This is especially true of the design of Printed Circuit Boards (PCBs). Most projects come with a PCB already designed [13], removing any design from the process.

During the years of 2015 and 2016, five PCB design projects were completed. Three of these projects involved a Linear Synchronous Motor (LSM) project – the Supercell LSM Drive, 4H Coil Drive, and Broken Beam Position Sensor. An LED badge and two revisions of a PCB business card were also created. These projects involved greatly different processes, and featured a wide range of design challenges. All boards were designed using using open-source PCB editor KiCAD [1].

II. Supercell LSM Drive

A. Project Description

The Supercell LSM drive was the first experiment with professional PCB design. The project was needed to drive a LSM, which is electrically and physically a three phase brushless motor that has been unrolled. While an Arduino was possible to use for generating the control signal, a three phase inverter would be needed to provide enough power to the motor. The Texas Instruments DRV8303 [11] was chosen for the control of the inverter, due to its ability to drive the MOSFETs required to create a three phase inverter, and as it included application guides [10] that could be adapted to work with the project.

The motor driven was intended to be able to be operated at up to 50VDC, so some components had to be carefully selected to avoid the risk of overloading the parts. The board required the use of a professionally manufactured PCB due to the small pitch of the surface mount DRV8303.

B. Design & Manufacturing Process

As the board was required to contain a surface mount component, surface mount LEDs, resistors, and MOSFETs were chosen to make the assembly and design process simpler. Custom components were created within KiCAD for components such as the MOSFETs (CSD18540Q5B) and DRV8303, and assembled into a schematic as shown in Fig.1. The schematic for the three phase inverter was based almost entirely on a Texas Instruments application guide [10], which allowed for the board to be built without requiring manual prototyping.

For ease of viewing, the schematic was broken up into multiple pages and sections. Figure 1 contains the three phase inverter and the DRV8303 section, but the board also contained connections for an Arduino Mega. This part was required as the DRV8303 requires a three phase input signal to operate, as well as Serial Peripheral Interface Bus (SPI) communications.

An Arduino Mega was selected as the main controller for the board, primarily due to its versatility as a controller. Its many input and output pins allowed me to send the proper signals to the DRV8303, while the simplicity of the programming language allowed for quick tweaks to the code.

The completed schematic was then laid out onto a 8”x10” circuit board and sent to Advanced Circuits for fabrication. A double-layered circuit board was selected to allow for simpler layout,

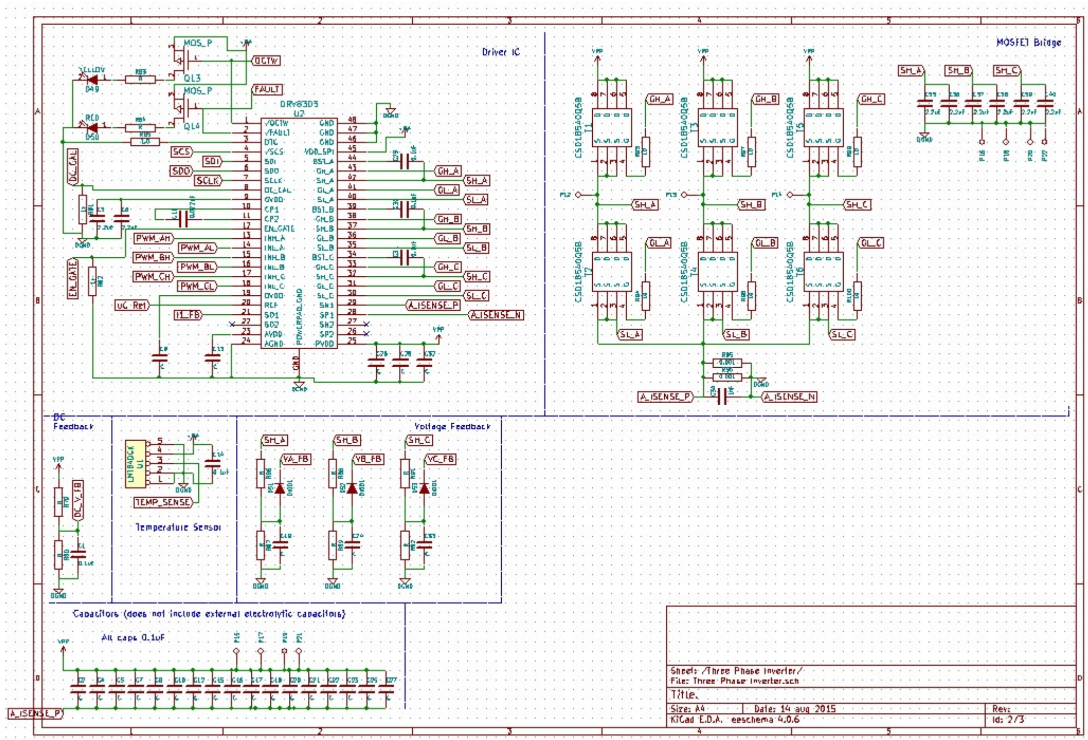

Fig.1 KiCAD Schematic

but multiple revisions were required and improve the design.

Design of the circuit board took some time, but each revision was able to be completed more rapidly by taking the methods learned from previous iterations and applying them. KiCAD’s PCB layout editor was used to first place large and important components in their desired places. The three phase inverter was oriented to allow for large buses to pass by, and the DRV8303 was placed as close as possible to allow the high frequency PWM signals to avoid attenuation. To try and improve the current capability of the board, the solder mask was removed, and on manufacture was coated with solder to increase the area of the “wire”. Additionally the back of the board was covered with a duplicate bus with many vias to connect them.

Once the positions of the important components had been solidified, the smaller components such as LEDs, resistors and capacitors were placed. Care was taken to position these components to minimize the trace distance. Capacitors especially were placed as close as possible to their components, usually right next to them.

While initial versions of the circuit board had traces drawn to minimize distance, utilizing vias to switch board sides if necessary, this would quickly create a maze of signals. To combat this, the positions of nearly all the parts was reset, and the board was relaid out so that nearly 100% of the traces could be completed on a single side. The back was then filled with a solid copper ground plane, to minimize noise that could potentially affect the electronics. The resulting circuit board is shown below in Fig.2.

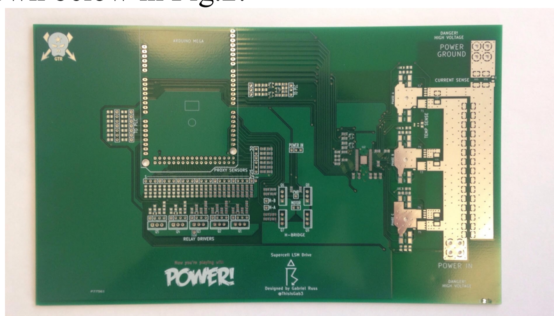

Fig.2 Unpopulated Supercell LSM Board

Upon receipt of the PCB, the board was assembled using components from Digi-Key [8]. Due to the small size of the surface mount parts, they could not be soldered by hand. To make the solder connections, ChipQuik solder paste was manually spread across all pads, and each component was placed on the paste by hand. The board was then placed on a kitchen griddle, which was set to 450°F. The board was then closely monitored as it heated, and when the paste reached its reflow point (the point at which the paste melts), all of the components were drawn to their correct positions through the surface tension of the paste. The griddle was then switched off and the board was allowed to cool naturally through the air in the room. The griddle soldering method worked extremely well, as all of the parts were properly located, with only the DRV8303 chip requiring some manual touchup with a microscope and a hand iron due to excess paste connecting pins improperly.

C. Lessons Learned

While the design of the driver worked fairly well, the largest challenge was in the assembly of the board, which required two hours of manual paste and part application of extremely small (0805) components with tweezers. Using a circuit board meant that connections were much more reliable between parts. Using an Arduino as the main controller for the board was extremely useful, as the microcontrollers have a wide range of input and output capabilities and can be quickly modified.

III. 4H Coil Drive

A. Project Description

While the Supercell LSM Drive performed acceptably, it was calculated that direct control over individual stator coils would allow for a more accurate launch system. To solve this, a board was created that would allow for bidirectional currents to be sent through the electromagnets, with the additional feature of being capable of driving motors in both directions to at least 24V.

B. Design & Manufacturing Process

The design of this board was based off an existing design shown in Fig. 3. The design was simulated using an online simulator [7] and built using KiCAD.

A new PCB manufacturer was chosen to supply this project, DirtyPCBs [9]. This company is a project from Dangerous Prototypes, and provides ultra-low cost PCBs from China, with fairly high quality 10cm x 10cm PCBs costing $17 for approximately 10 in a multitude of colors. Their high quantity was useful for this project as only four H-bridges were able to fit onto a single circuit board. The boards (twelve were sent by the manufacturer) arrived quickly and were made properly. The unpopulated board can be seen in Fig. 3.

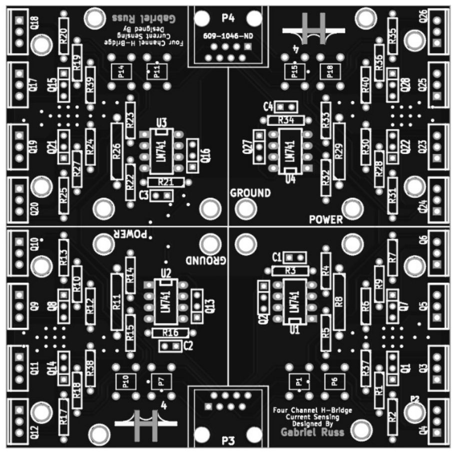

Fig.3 Unpopulated 4H PCB

Assembly of the board was where a major issue was encountered, as the schematic (shown in Fig. 4) used unusual MOSFET symbols. While the board was designed with N-channel MOSFETs on the high side and P-channel MOSFETs on the low side, they should have been reversed. Upon initial testing, the test power supply would instantaneously lock itself out, and it took several hours to discover the cause of the problem

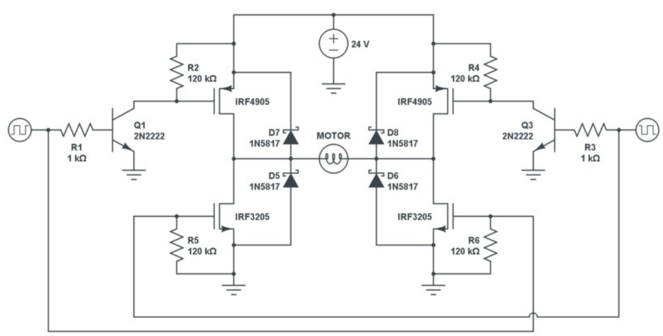

Fig.4 MOSFET H-Bridge Schematic [2]

The cause of the short circuit was due to the fact that no-bulk MOSFETs (the type used in the sample schematic) have the arrow on the source pin reversed vs the more commonly used MOSFET enh as shown in Fig. 5. Fortunately, the MOSFETs could be switched without requiring modification to the boards, and the board ultimately worked properly.

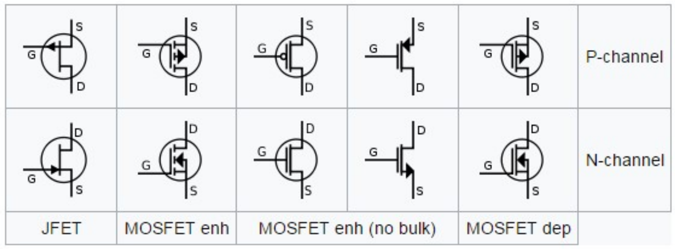

Fig.5 MOSFET schematic sysmbols [3]

C. Lessons Learned

Basing the design of a project on a preexisting design can have risks if the design isn't physically prototyped first.

IV. Broken Beam Position Sensor

A. Project Description

For the LSM project, a way to accurately determine the position of the projectile was quite difficult. Attempting to manually place sensors meant that accuracy of the spacing between the detection points was inconsistent. To improve this, a simple board was designed to implement a broken beam sensor. This allowed for contactless sensing of the projectile, which could have caused it to slow down.

B. Design & Manufacturing Process

To decrease the cost and complexity of the ordering, the board was designed so that the same board could be used both as a transmit and as a receive unit, just by changing a few components and soldering or desoldering a few small jumpers. As a transmit board, the unit has a MOSFET and its associated resistor to switch on and off the IR LEDs. As a receive board, an Ethernet connector is soldered onto the back, similar to the 4H Coil Drive, allowing for connection to a microcontroller. The LED locations are filled with phototransistors selected to match with the IR LEDs of the transmit side.

To allow for easy positioning and connecting, the board has several plated holes drilled through it at regular intervals, and each side has a power, ground and trigger (only used in transmit mode) connector. To make a chain of the boards is as simple as connecting the pads between boards.

Fig.6 Unpopulated Broken Beam Sensor PCB

The boards were ordered from DirtyPCBs, and due to their simplicity were able to be easily soldered by hand. To test the transmit boards, the trigger pin was set to 5V, and an iPhone 4 camera was pointed at the IR LEDs. This brand of phone camera is able to see into the frequency of the LEDs used, and displays them as a light pink, allowing for easy testing.

To test the receive boards, an Arduino Uno was connected to the Ethernet cable, and a transmit board was positioned approximately 5 inches away. An input to serial program was then used to make sure that each phototransistor reacted to the covering of the IR beam.

C. Lessons Learned

The main lesson learned from this design was in the usefulness of planning how the board would ultimately be used. Had separate transmit and receive boards been used, the end cost would have been higher. Because adding unfilled component space did not increase the overall size of the board it was easy to save the extra cost of more boards.

V. LED BADGE

A. Project Description

This project was done as a present for a friend, who requested a unique way to display text in a way that could be worn as a badge for conventions & other events. The design that was created was a basic 8x16 LED matrix, controlled by an ATmega328P. This microcontroller drives two shift registers and a transistor array to scan text.

The secondary goal of this project was to see how rapidly an idea could be brought into the real world, to try and improve personal speed at designing schematics, programming, and laying out printed circuit boards.

B. Design & Manufacturing Process

The design of the badge was fairly simple electrically, as all of the signals were intended to remain at 5 volts. A phone battery pack was selected as the power source, due to their cheapness, and due to how easily they can be recharged.

Two 74HC595 shift registers are used in the project to drive the individual columns of LEDs, and a ULN2803 was selected to switch between the 8 rows. While the software doesn't have a specific frequency at which it updates the display, it was able to maintain a high enough frequency that display updates were not noticeable.

To try and give the badge a more aesthetically pleasing look, a unique curved board edge was drawn manually across the top. While the DirtyPCBs manufacturer did not specifically allow unique PCB outlines, it was created without issue. The unpopulated board can be seen in Fig. 7.

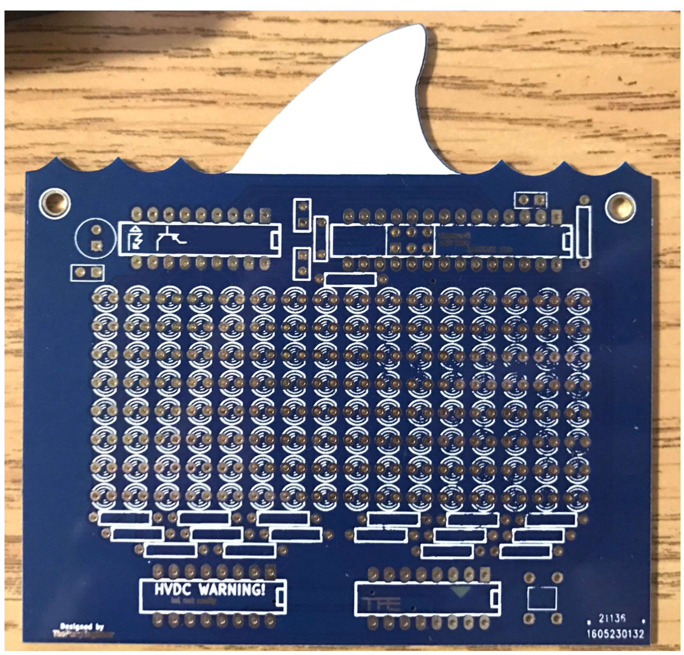

Fig.7 LED Badge PCB

Upon the arrival of the board, the assembly was completed by hand. This took quite some time due to the 128 through hole LEDs, and could have been completed more rapidly had surface mount LEDs been used instead of through hole parts. Programming and working out the bugs in the software was the most time consuming part of the whole process, with errors in the code interacting with the shift registers causing garbled data to appear on the screen. To debug the unit, it became necessary to solder jumper wires to many components, requiring extensive touchup afterwards to ensure that the finished project was presentable.

The manufacturing efforts paid off however, as the boards were able to survive many days of being walked around without any LEDs failing. One issue that popped up was that occasionally the board would randomly turn off. This was tracked down to the necklace that the user had installed was shorting the top copper pour (5V) with the bottom copper pour (GND). This was activating the protection circuitry in the battery pack which would turn off the unit.

C. Lessons Learned

The biggest lesson learned through this project was that it is important to include test points on projects. Having an easy way to connect to the tested device would have allowed for much faster debugging of the software. The project also showed that PCB projects are not limited to a simple square, and using the shape of the board can be very effective.

VI. PCB BUSINESS CARD

A. Project Description

Business cards are one of the most common networking tools utilized by students and professionals. The problem with business cards is their commonness. To set a unique impression, some engineers have begun creating their own business cards out of printed circuit boards [12]. So far two revisions of a personal business card have been created so far, one pocket reference card, and one USB isolator. A great deal of positive feedback has been given from people who have received the various versions. While they do cost more than traditional business cards, their uniqueness sets a very powerful first impression.

B. Design & Manufacturing Process

For the 2015 edition of the card, the electrical design was fairly simple. An ATmega328P breakout is included on the front, and the back is used as a space for LED, MOSFET, surface mount and transistor diagrams, as well as the Ohms Law wheel [5]. For further usefulness, an imperial and metric ruler is across the top and bottom edge. The card is shaped slightly shorter and smaller than a standard business card (3”x2.5”) [6] to allow it to fit with regular business cards in a pocket or folder.

The fiberglass of the board also has the benefit of being far more durable than a paper card. In order to make the cards more efficiently, two cards were connected together using drilled holes. This required manual filing of the fiberglass to achieve a clean edge. The completed card can be seen in Fig. 8.

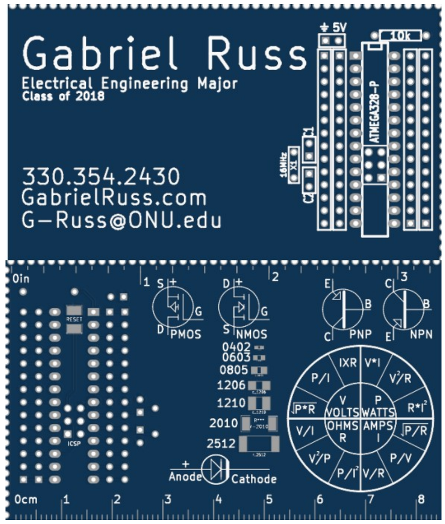

Fig.8 PCB Business Card (2015)

For the 2016 edition of the card, a USB isolator was designed. This was chosen to allow for a working and object to be given out, without requiring any modification to be made by the receiver. Due to the prevalence of USB charge devices, and the risk that plugging a cell phone into an unknown device presents, a device that can isolate the user's phone is extremely useful. Using two resistor dividers, a known voltage is generated on the USB D+ and D- pins, which tells phones plugged into it to charge at 500 mA [4]. This is the lowest value possible for charging, but ensures that the phone doesn't overload the charging device.

The card was designed to fit easily in line with the user's phone cable, with solder on the back of the card to ensure a snug fit. The through hole resistors are situated within slots cut into the card, allowing them to sit almost completely flush with the surface of the card. This was chosen to try and protect the resistors and to make it easier for the card to be stored. The resistors were tested by cutting each lead one at a time on sample cards, to ensure that a phone plugged in would not enter a dangerous state.

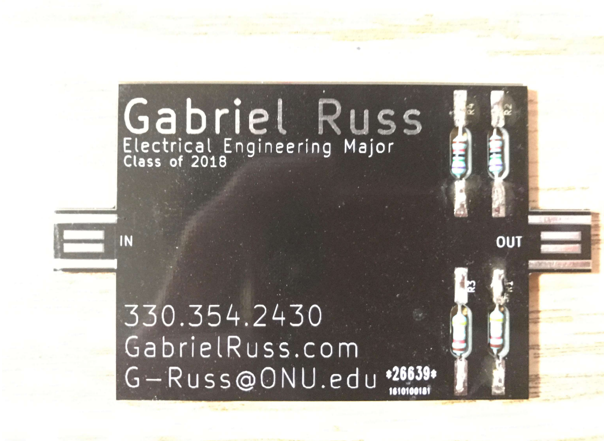

Fig.9 PCB Business Card (2016)

C. Lessons Learned

(**2015**) Having a single card per board would have allowed for the cards to be ready to hand out upon arrival, without requiring manual processing. Additionally, when including data on a business card, having someone else double check the data is useful, as the card features a NPN with a backwards arrow

(**2016**) Building a mockup of the card before placing the order would have allowed the card to have a slightly tighter fit on the USB ports.

VII. Conclusion

Through these design projects, many useful lessons were able to be learned in PCB design. While some of the designs required some modification, they were all able to perform well. These techniques and lessons should be useful for any student who is interested in layout, or improving their layout skills.

References

[1] Charras, "KiCad EDA", *Kicad-pcb.org*, 1992. [Online]. Available: http://kicad-pcb.org/. [Accessed: 13- Mar- 2017].

[2] E. Kringen, "Mosfet H-Bridge", *CircuitLab*, 2014. [Online]. Available: https://www.circuitlab.com/circuit/m4c244/mosfet-h-bridge. [Accessed: 13- Mar- 2017].

[3] "MOSFET", *En.wikipedia.org*, 2017. [Online]. Available: https://en.wikipedia.org/wiki/MOSFET. [Accessed: 13- Mar- 2017].

[4] L. Fried, "iCharging | Minty Boost | Adafruit Learning System", *Learn.adafruit.com*, 2013. [Online]. Available: https://learn.adafruit.com/minty-boost/icharging. [Accessed: 13- Mar- 2017].

[5] "Ohms Law Wheel", *Ohmslawcalculator.com*, 2017. [Online]. Available: http://www.ohmslawcalculator.com/ohms-law-wheel. [Accessed: 13- Mar- 2017].

[6] "Business card", *En.wikipedia.org*, 2017. [Online]. Available: https://en.wikipedia.org/wiki/Business_card. [Accessed: 13- Mar- 2017].

[7] I. Sharp and P. Falstad, "Lushprojects.com - www.lushprojects.com - Circuit Simulator", *Lushprojects.com*, 2015. [Online]. Available: http://lushprojects.com/circuitjs/. [Accessed: 13- Mar- 2017].

[8] "DigiKey Electronics - Electronic Components Distributor", *Digikey.com*, 1995. [Online]. Available: http://www.digikey.com/. [Accessed: 14- Mar- 2017].

[9] "PCBs cheap! - DirtyPCBs.com", *Dirtypcbs.com*, 2015. [Online]. Available: http://dirtypcbs.com/store/pcbs. [Accessed: 13- Mar- 2017].

[10] *36-V Power Stage for Brushless Motor*, 1st ed. Texas Instruments, 2015.

[11] "DRV8303 Three Phase Pre-Driver with Dual Current Shunt Amplifiers | TI.com", *Ti.com*, 2014. [Online]. Available: http://www.ti.com/product/DRV8303/description. [Accessed: 15- Mar- 2017].

[12] T. Ward, "Extreme Business Cards", *Instructables.com*, 2017. [Online]. Available: http://www.instructables.com/id/Extreme-Business-Cards!/. [Accessed: 15- Mar- 2017].

[13] A. Industries, "Adjustable breadboard power supply kit [v1.0] ID: 184 - $14.95 : Adafruit Industries, Unique & fun DIY electronics and kits", *Adafruit.com*, 2017. [Online]. Available: https://www.adafruit.com/products/184. [Accessed: 15- Mar- 2017].