Edgerton Assembly Manual

Documenting this project has been a hobby. There may be errors or mistakes in the documentation. If you suspect an error, or if something does not make sense, please send me an email at tyler@tdog.ca!

Table of Contents

Description

This is a complete assembly manual for Edgerton, the High-Speed LED Flash. Some soldering and 3D printing skills are required. Please note that the photographs below may show components that differ somewhat from the most recent models available. The assembly photos were taken during development of the flash and the design has been slightly changed since.

For a complete overview, please visit https://td0g.ca. The documentation and firmware repository are available at https://github.com/td0g/high_speed_flash.

Warnings – PLEASE READ

The greatest advantage of an LED flash over an air-gap flash is safety, **but there are still dangerous voltages in this LED flash**! The capacitor charger, if set correctly, will supply 120V to several capacitors. Be careful, try to keep the case closed, and do not touch any circuitry when the capacitor charger is on!

This strobe pushes the capabilities of the LED's. I have worked hard to find the safest limitations of the LED's, but **complete failure of the LED's is still possible**. That means the expensive components can be accidentally destroyed and require replacing. I've taken many precautions in the design to prevent this, but please consider building a strobe at your own financial risk.

The LED's are not as powerful as a Xenon flashtube, and they do not turn on as long as a typical camera strobe. **Expect to crank up the ISO by several (4 or more) stops** in order to capture usable images.

Tools Required

- 3D printer with 200mm x 200mm bed
- Soldering Iron & Solder
- Small Side Cutters
- Hot Glue Gun
- Allen Wrenches
- Dupont Crimper Kit (Optional but Recommended)

3D Printing the First Parts

Note: before proceeding, please choose a suitable 3D printing filament. I prefer PETG as it is tough, fairly easy to print, and is not affected by heat. ABS is also a good filament, but requires more care to print. PLA+ is a form of PLA that is more resistant to heat, but retains PLA's amazing printing properties.

1. Download the latest set of .STL files from https://github.com/td0g/high_speed_flash/tree/master/MK1_Edgerton.
2. 3D print the **Template.STL** (using a cheap filament – this part will not be used in the final assembly) and 8x copies of the **LED Clamp.STL**
3. Begin 3D printing the **Front.STL** and **Back.STL**. This will take many hours and you can proceed to the next steps while these parts are printing.

LED Bank Assembly

4. Take 3x **LED’s** and lay them out on the **Template**. Make sure the orientation is the same (anodes / + on one side, cathodes / - on the other).

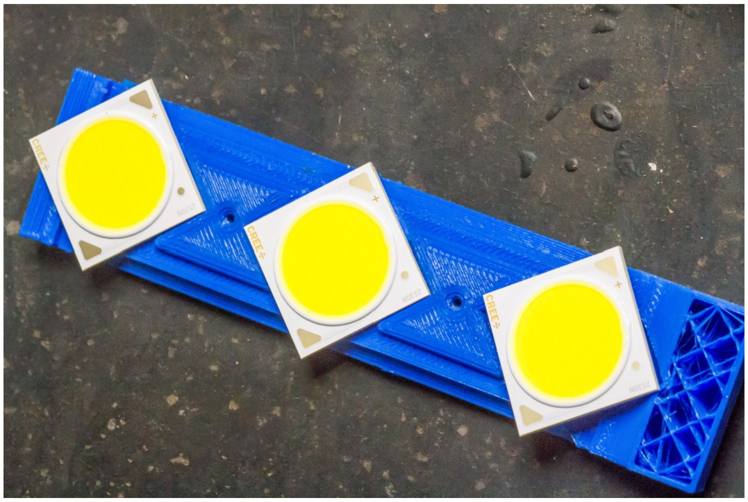


Figure 1: Anodes on top, cathodes on bottom

5. Flip the **LED’s** face-down and clamp them down using the **LED Clamps** and **M2 Screws**.

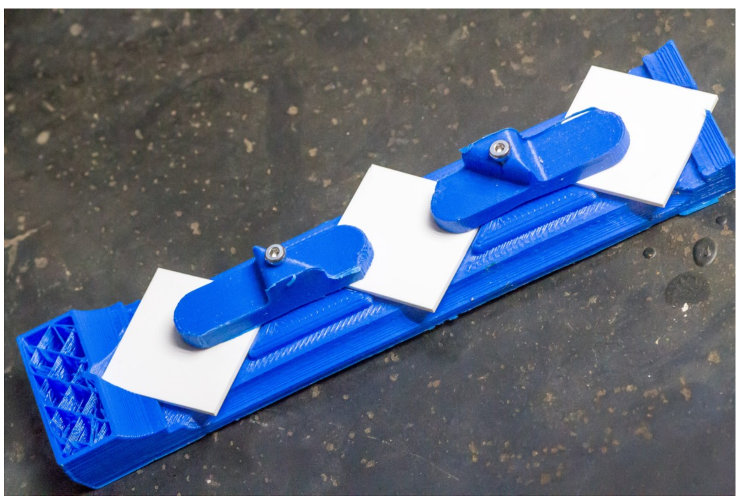


Figure 2: Do not mix up LED orientation when flipping LED's

6. Take a 9cm long section of **22AWG** wire and strip the insulation. Twist the conductors (wire strands) so that there are no loose strands.

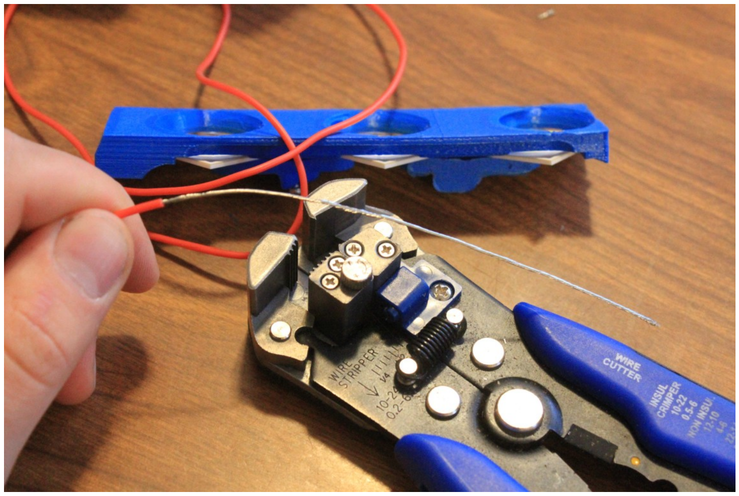


Figure 3: Removing insulation from 22AWG wire

7. Solder the bared wires onto the **LED** cathodes (negative). Don't make the wire too tight or too loose.

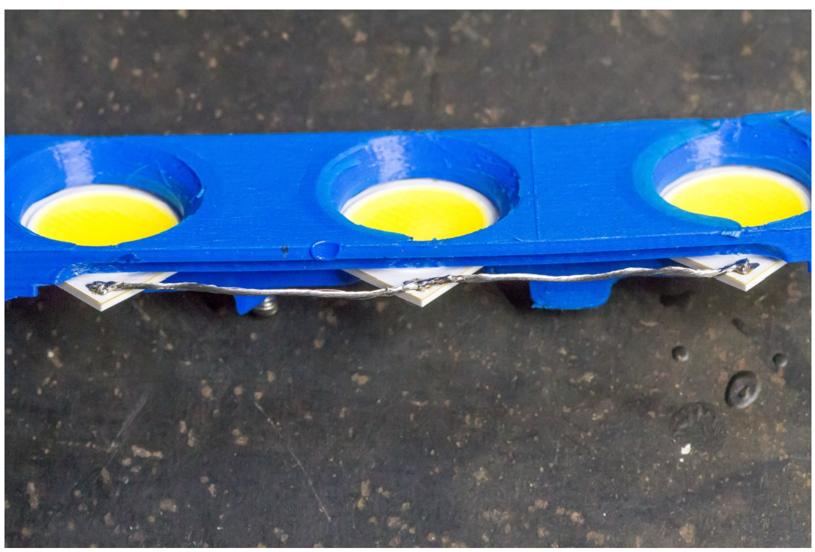


Figure 4: 22AWG wire soldered to LED cathodes (negative)

8. Solder the **2-ohm Resistors** onto the **LED** anodes (positive) as shown below.

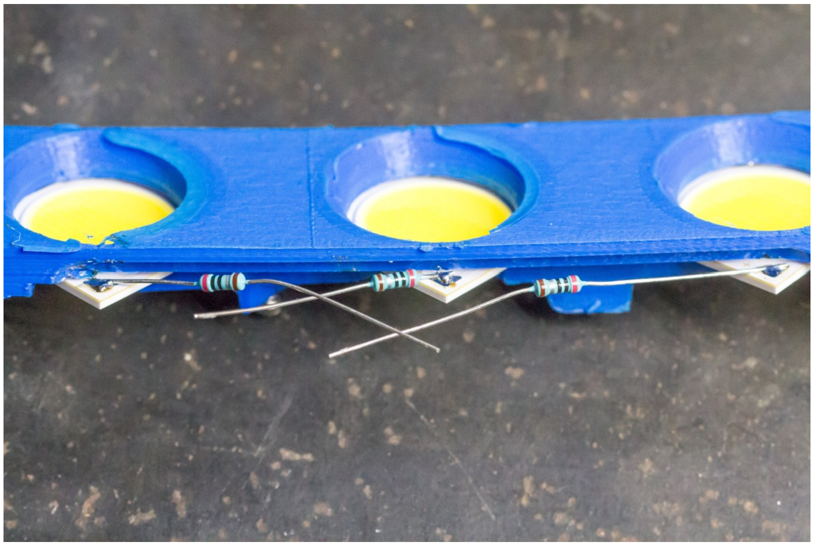


Figure 5: 2-ohm series resistors

9. Take a **Film Capacitor** and solder the **MOSFET** source pin as shown (Polarity of the film capacitor doesn't matter – you can use either side). Don't connect the **MOSFET's** middle pin yet. *NOTE – the layout of the MOSFET pins (Source, Gate, Drain) are shown in Appendix A, at the end of this manual.*

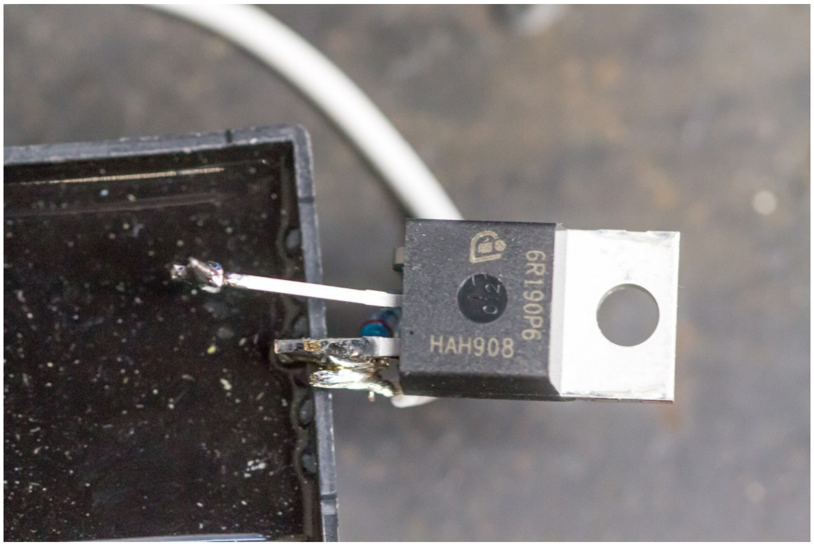


Figure 6: MOSFET Drain on bottom, Source in middle, and Gate on top

10. Solder **22AWG Wire** onto the **MOSFET** source / **Capacitor** cathode.

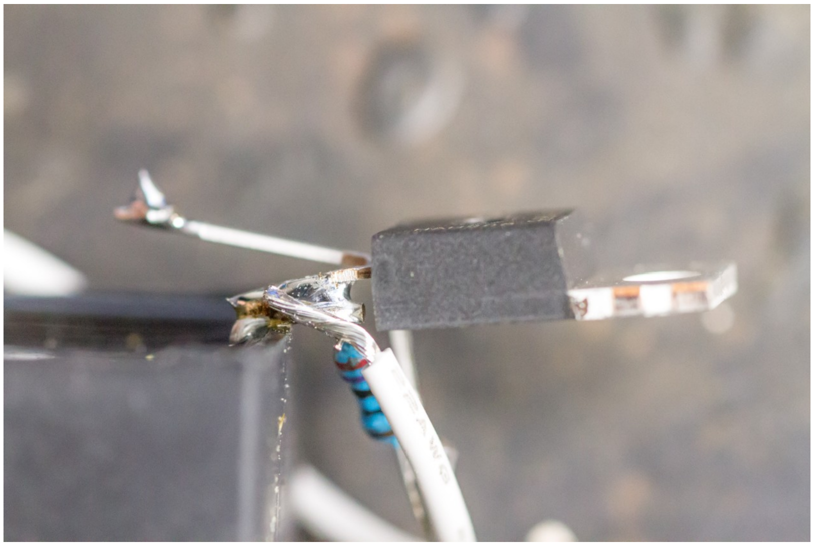


Figure 7: Ground wire soldered to MOSFET Source / Capacitor cathode

11. Solder a **200-ohm Resistor** from the MOSFET Gate to the Source.

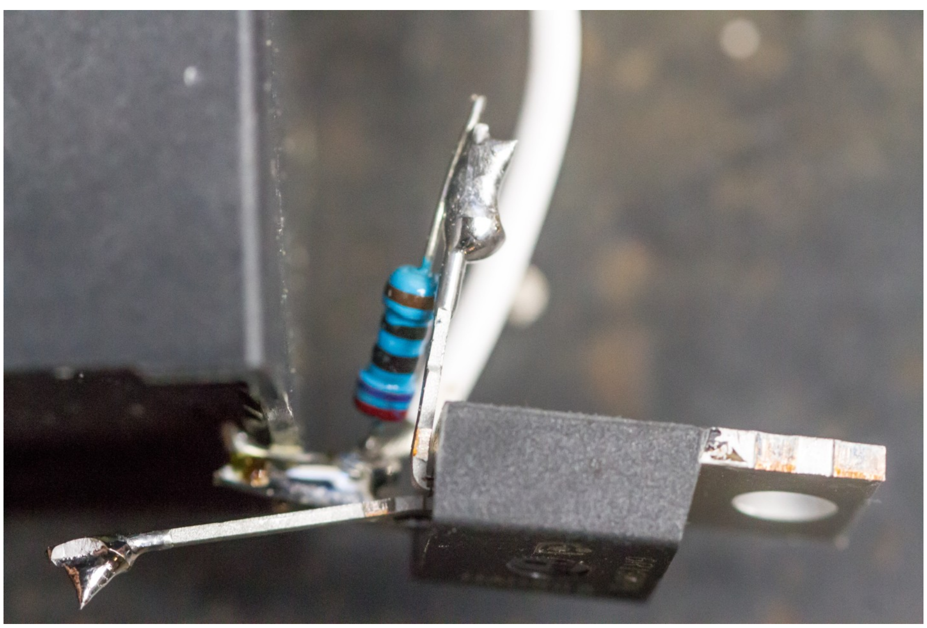


Figure 8: 1/4W resistor from MOSFET Gate to Source

12. Solder the **Ferrite Bead** onto the **MOSFET** gate

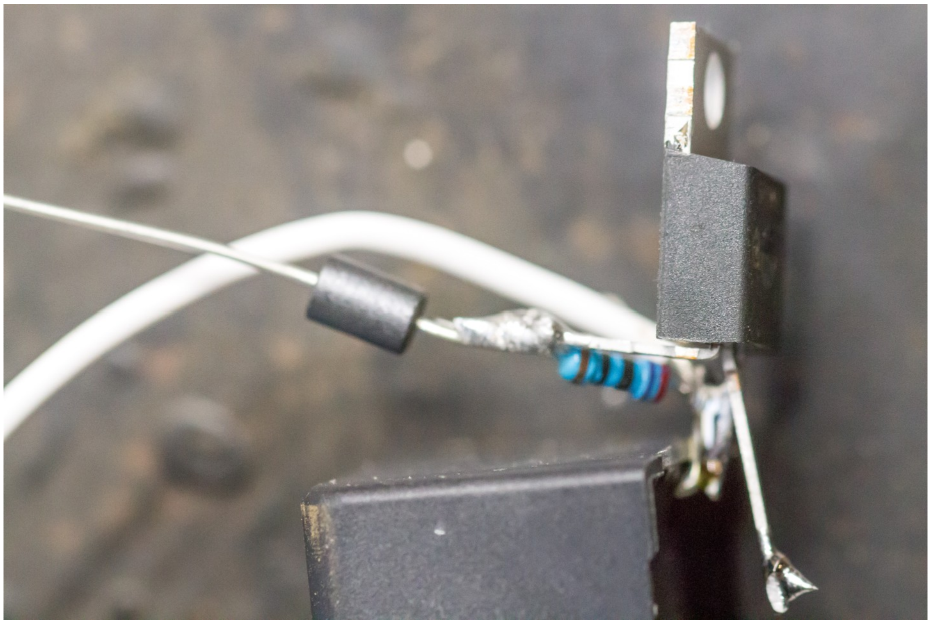


Figure 9: Short distance from ferrite bead to MOSFET

13. Lay the **Capacitor** upside down and place the **LED Template** on top. Solder the **MOSFET** drain (middle pin) to the **LED** cathode rail.

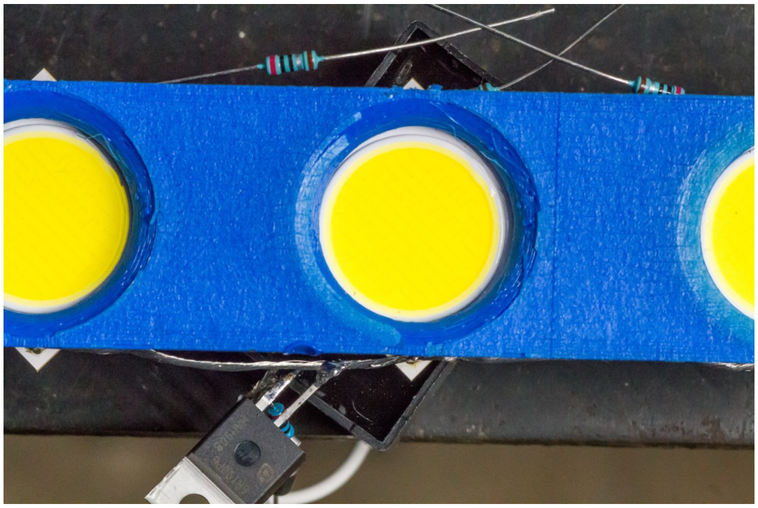


Figure 10: MOSFET Drain (middle) to LED cathode rail

14. Solder the **Capacitor's** anode to the LED series **2-ohm Resistors**. Clip the excess leads from the **2-ohm Resistors**. Note how the rightmost resistor lead is routed on the opposite side from the LED Anode pad so that they don't touch.

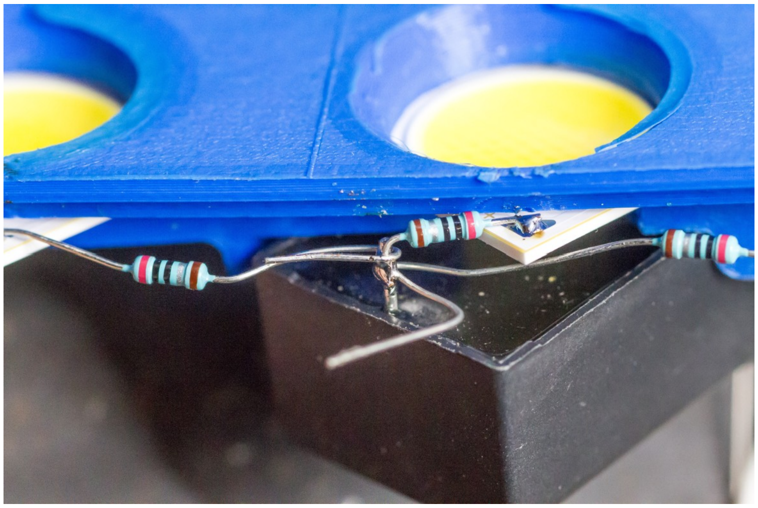


Figure 11: Careful routing of resistor leads

15. Solder a **22AWG Wire** to the **Capacitor's** anode.

Figure 12: High-Voltage Supply Line on capacitor Anode

16. Repeat steps 4 – 15 three more times to build a total of four banks.

Control Board

17. There are multiple options available:
    - Assemble your own control board on a 5 x 7 cm perfboard using the Circuit Diagram and Perfboard Layout available at https://github.com/td0g/high_speed_flash/tree/master/MK1_Edgerton.
    - Etch your own control board using the Eagle files available at https://github.com/td0g/high_speed_flash/tree/master/MK1_Edgerton.
    - Purchase a ready-to-use control board from https://www.tindie.com/products/td0g/high-speed-flash-controller/. If you chose this method, you may skip step 18.
18. If you chose to assemble or etch your own control board, flash the ATMega328P with the latest firmware available at https://github.com/td0g/high_speed_flash. There are multiple ways to do this, but I personally recommend burning a Lilypad Arduino bootloader, installing the microcontroller on the control board, then using an FTDI adapter and the Arduino IDE to upload the firmware. Nick Gammon has a good guide for this process, please visit http://www.gammon.com.au/breadboard.
19. If you build your own **Control Board**, please consider performing the Quality Control tests outlined in Appendix B. If you do not have the equipment, please double-check all of the solder joints on the board before proceeding.
20. Mount the **Control Board** onto the **Back** body of the flash using 2x **M2 x 12mm Screws**.

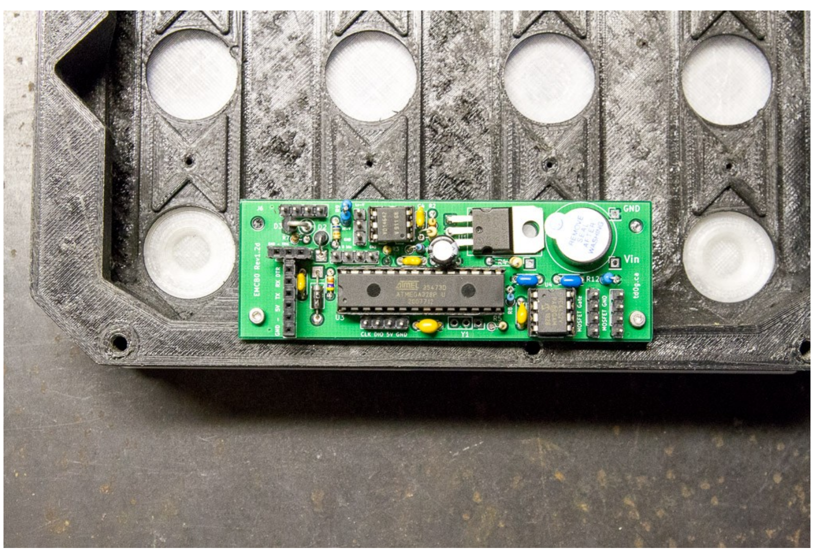


Figure 13: Control Board mounted on back

Complete Assembly

21. Begin printing the **Encoder Knob.STL**.

22. *OPTIONAL* - Slice the **LED Cover.STL**. This component has a built-in magnet. You will need to determine the layer at which the magnet hole is covered and modify the gcode to pause at that point. You might be able to use M226, but I believe most firmware doesn't support this gcode. I simply split the gcode into two files, with the first ending when it's finished the top layer of hte magnet hole (and adding G0 X0 Y190 F3000 to move the print head out of the way) and the next file continuing after that. This way, I can leave the printer running, install the magnets when the printer finishes the first file, then continue by running the second file. Once the gcode is ready, go ahead and print the LED cover.

23. Install the **LED Banks** into the front as shown. Use two **LED Clamps** per bank and one **M2 x 12mm** screw per LED Clamp. Alternate the anode/cathode direction so that there is never an anode and cathode adjacent to each other.

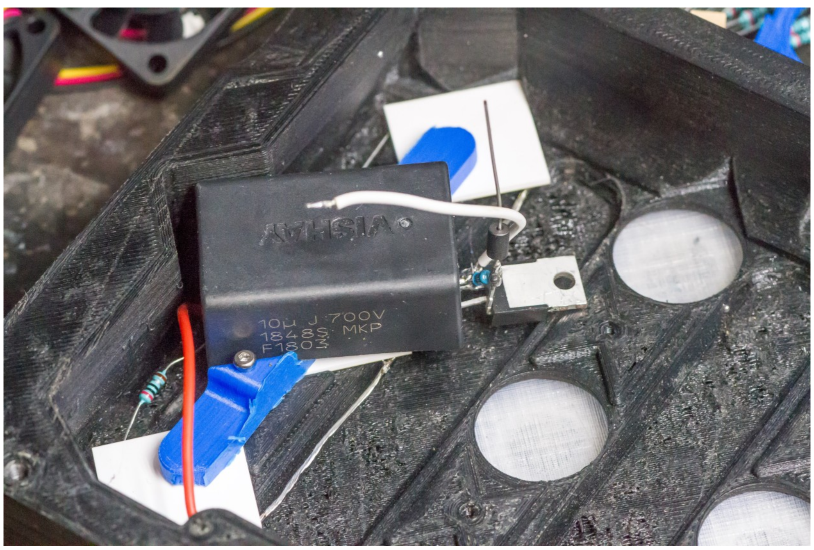


Figure 14: LED bank installed in case front

24. Install the **Arca-Swiss QR** plate onto the **Back** using **M5 Screws** and **Nuts**. You may need to drill holes into the QR plate for this purpose.

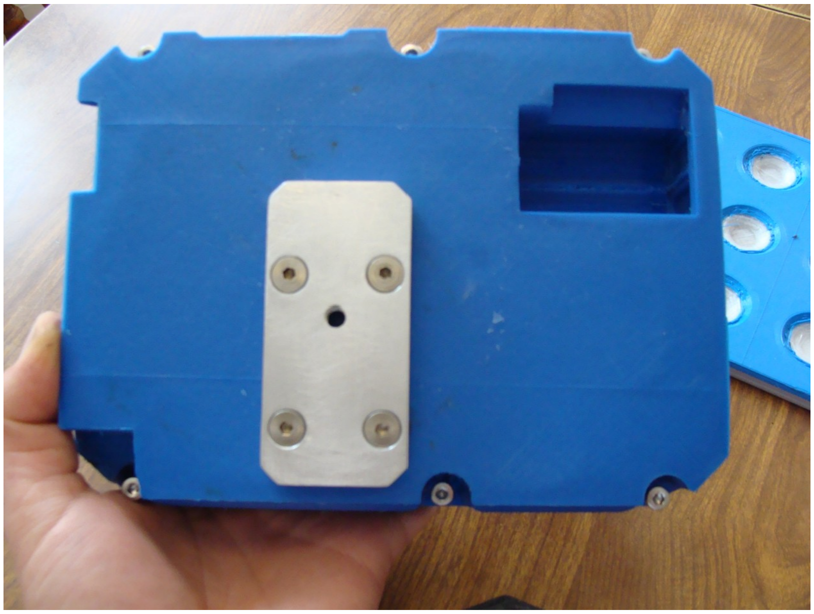


Figure 15: Arca-Swiss QR plate

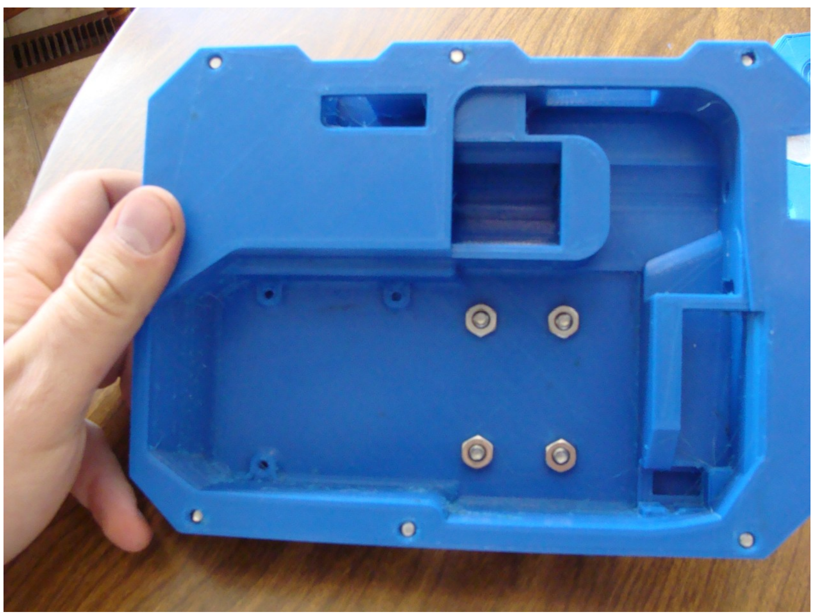


Figure 16: Case interior view, M5 nuts securing Arca-Swiss QR plate

25. Install the **High-Voltage Boost Converter (capacitor charger)** onto the back using **3x M3 x 8mm Screws**.

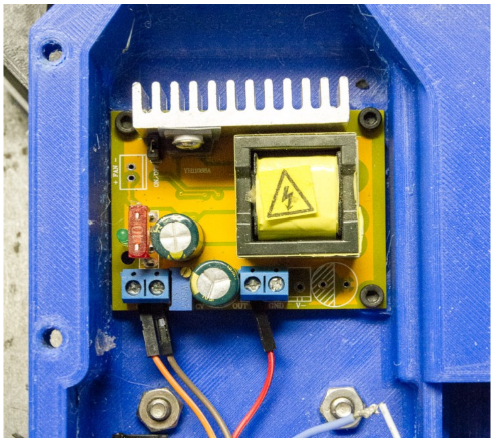


Figure 17: High-Voltage Boost Converter

26. Screw the **Encoder** into the hole in the **Back** (See figure 18). You may need to tap the hole first if it is difficult to thread in.

27. Solder four 22-gauge wires onto the encoder as shown in figure 18.

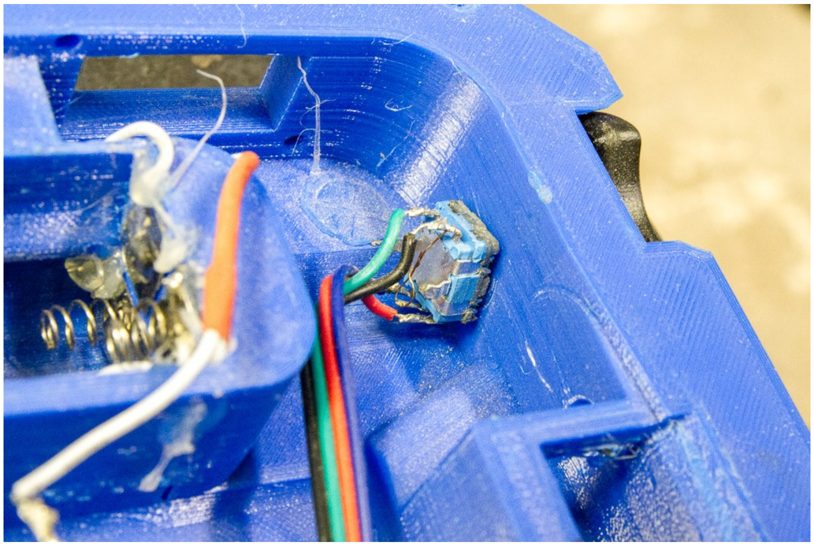


Figure 18: Encoder screwed into Back

28. Install the **Dial** onto the **Encoder**

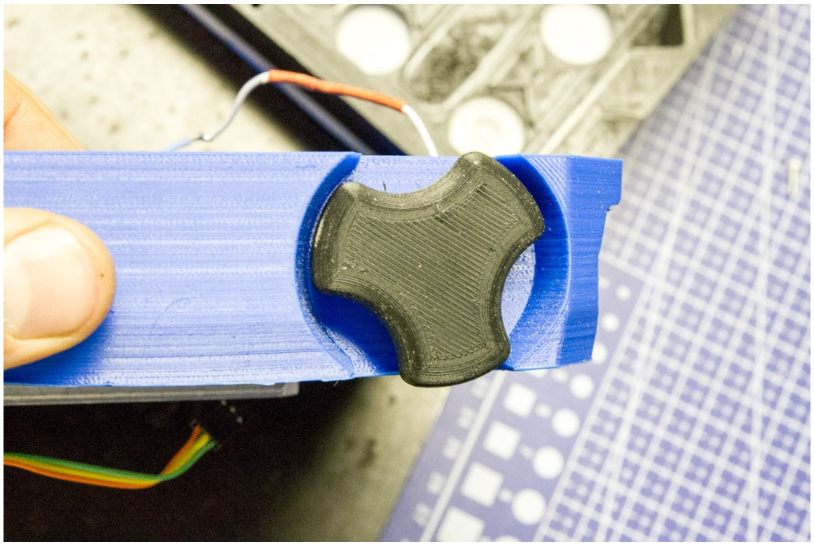


Figure 19: Dial mounted on Encoder

29. If your **TM1637 Display** has a male header facing the top of the display, remove it and install a header facing the bottom (mine is female, you may use male headers OR directly solder wires onto the display).

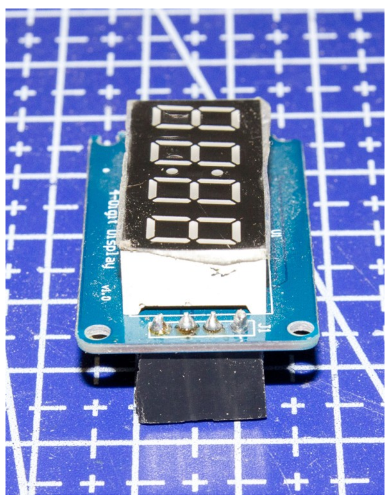


Figure 20: TM1637 Display with header facing bottom

30. Slide the **TM1637 Display** into the hole in the back and secure it with **M2 x 12mm Screws** or hot glue.

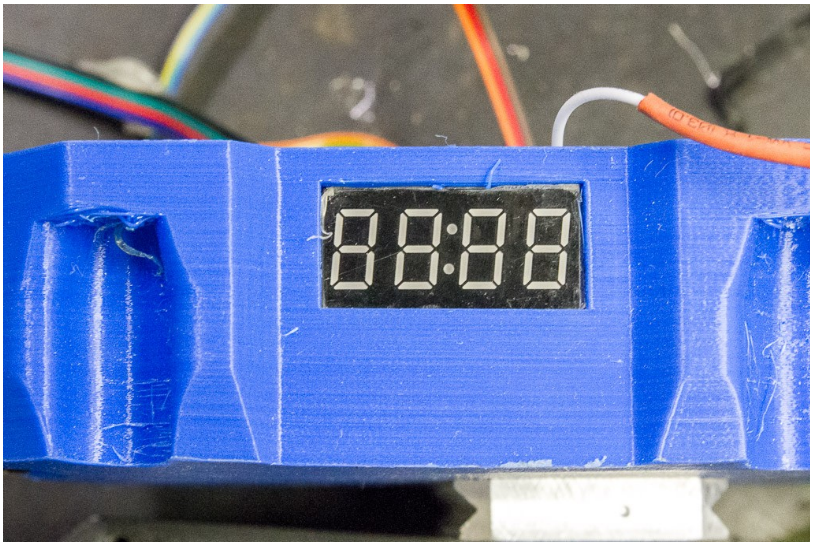


Figure 21: Control Board mounted on back

31. Slide the **3.5mm Jack** into the small hole in the back. Solder wires to all three pins then secure the jack with hot glue.

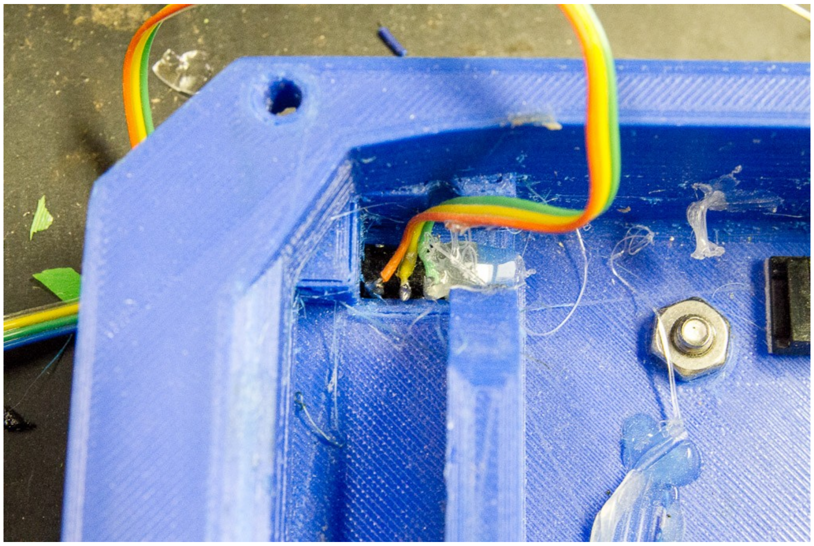


Figure 22: 3.5mm Jack secured with hot glue

32. Install the **AA Battery Contacts** onto the back using hot glue.

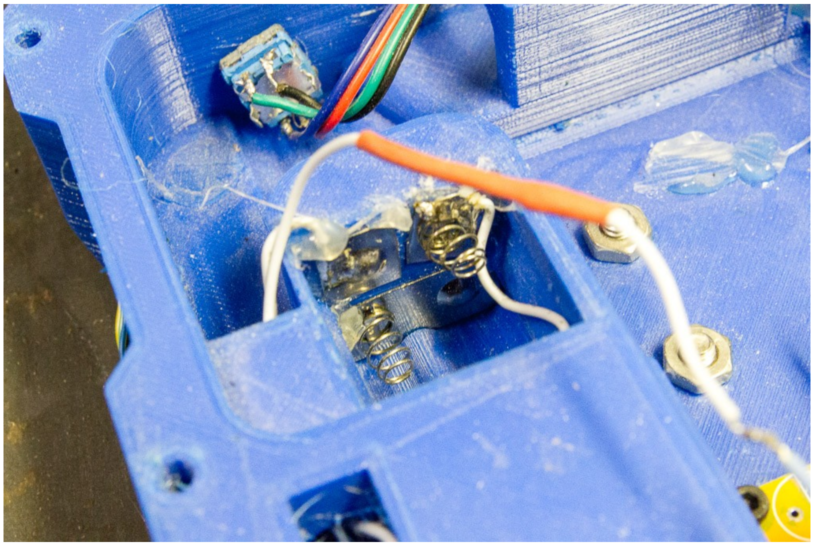


Figure 23: Interior battery contacts – Note that the top left (POSITIVE) is separated from the top right (NEGATIVE)

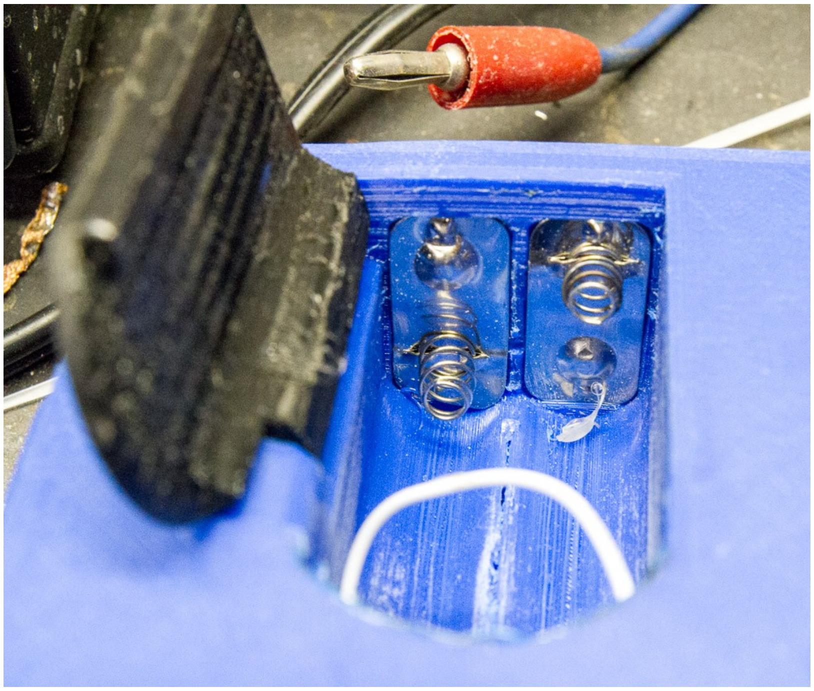


Figure 24: Exterior battery contacts

33. Install the battery door with an **M3 x 8mm Screw** and **M2 x 12mm Screw**.

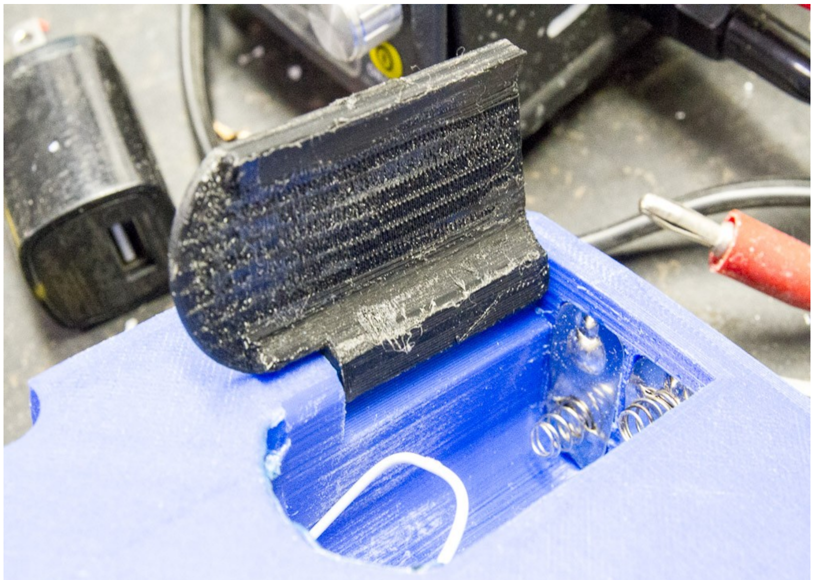


Figure 25: Exterior battery contacts

34. Solder a wire from the battery output positive and run it to the power switch hole. Solder the other end onto the **Power Switch**, and run another wire through the hole to the **Power Switch**.

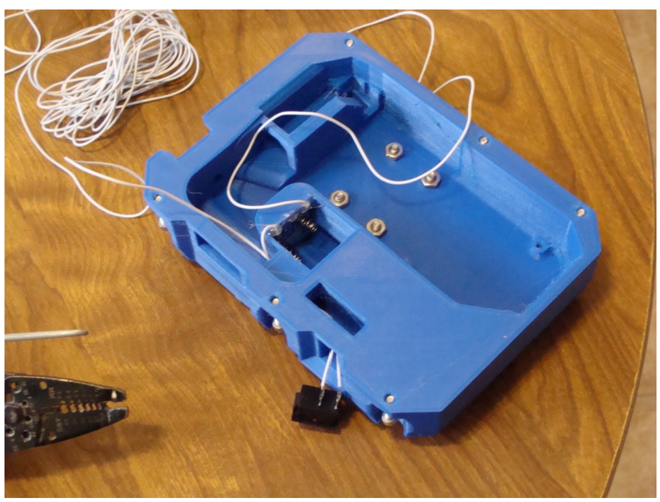


Figure 26: Wiring to power switch installed, proceed to slide power switch into case

35. Slide the **Power Switch** into the hole in the **Back**.

36. Connect the **High-Voltage Boost Converter** supply power to the **Control Board**. NOTE: Figure 27 shows the high-voltage (RED) wire connected. Leave it disconnected for now, only connect the other two wires.

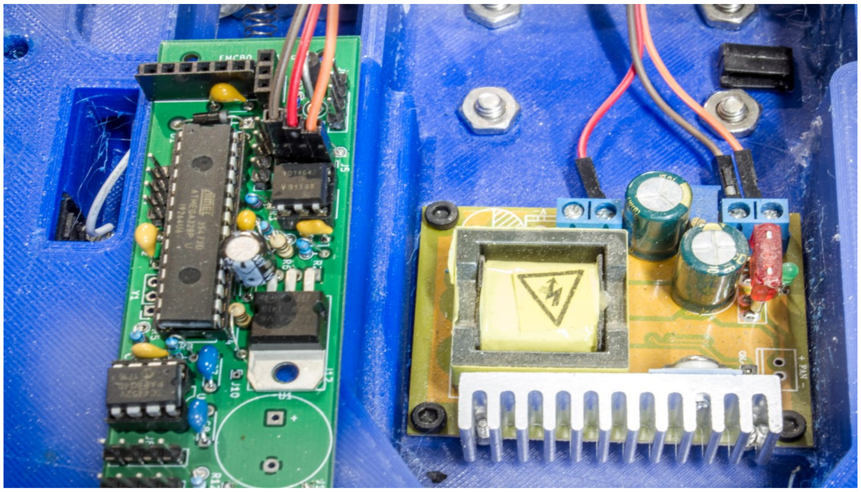


Figure 27: Connecting High-Voltage Boost Converter to Control Board

37. Connect the **3.5mm Jack**, **Encoder**, and **TM1637 Display** to the **Control Board** (see the following figures). I used Dupont connectors and headers on the **Control Board**, but if you don't have a crimper tool then you can solder the wires directly onto the **Control Board**. The Capacitor and MOSFET connections are all connected in parallel. NOTE: It is recommended to solder the Capacitor Gnd and Gate connections onto the male headers.

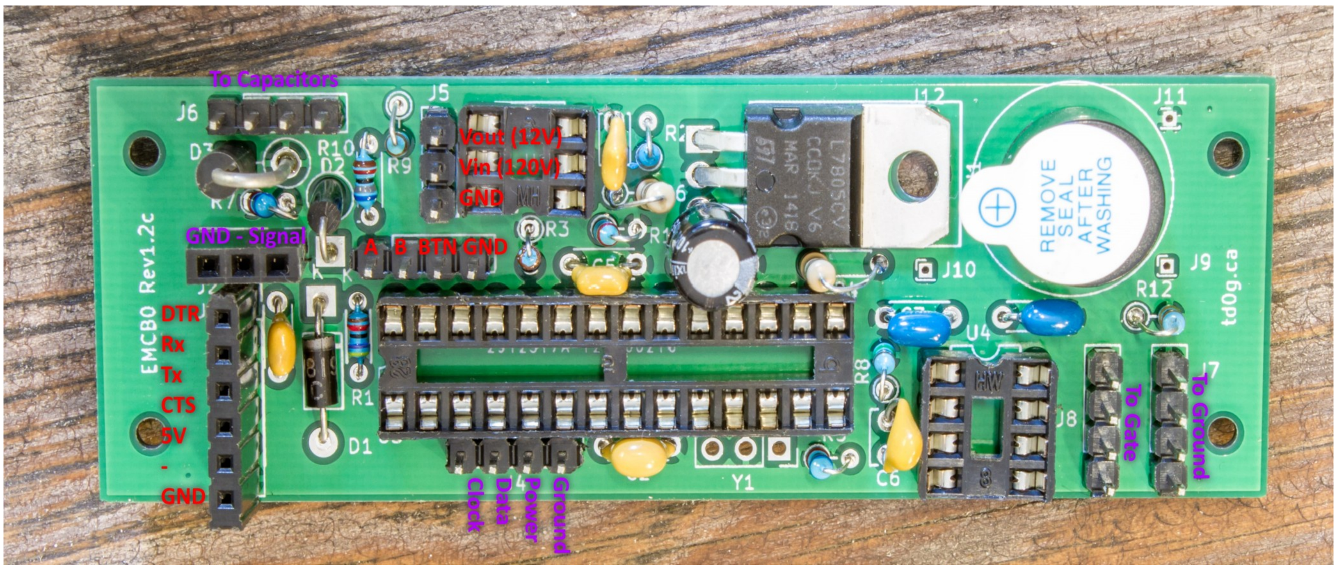


Figure 28: EMCB0 Control Board Hookup Guide

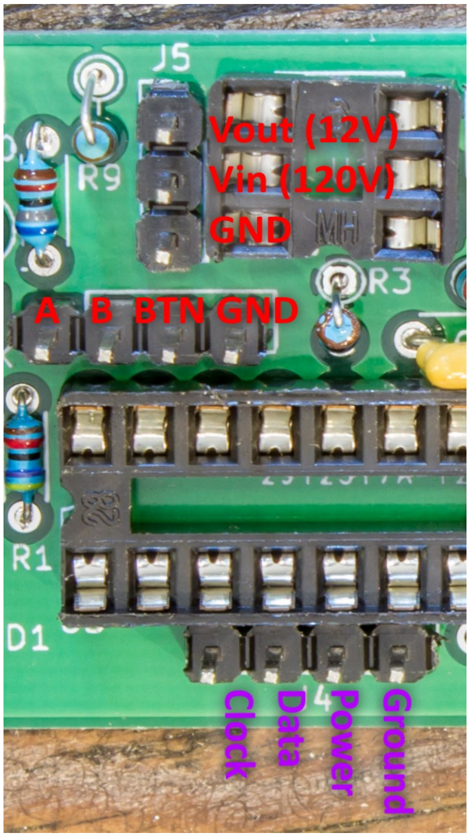


Figure 29: High-Voltage Boost Converter (TOP), Encoder (MIDDLE), and Display (BOTTOM) headers

Figure 30: Capacitor Anode header

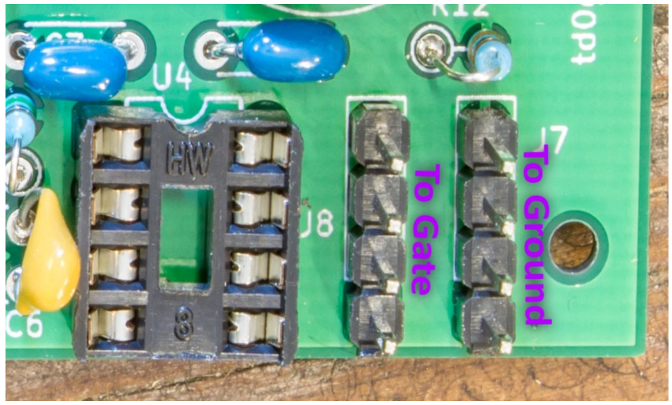


Figure 31: MOSFET Gate and Capacitor Ground headers

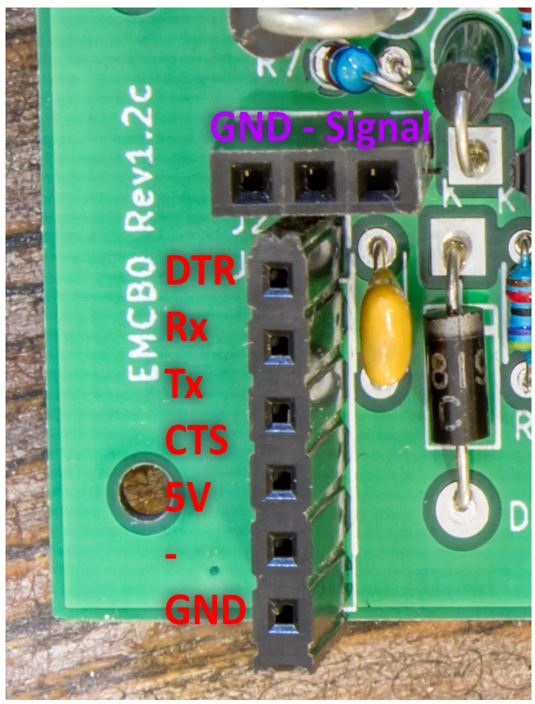


Figure 32: Trigger (TOP) and FTDI (BOTTOM) headers

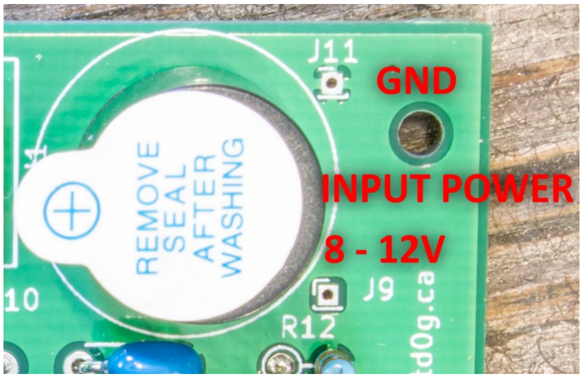


Figure 33: Power Input connections (Battery Negative to GND, Power Switch to 8 - 12V)

Firmware and Voltage Setup

38. The setup routine can be run by holding the button for several seconds while the flash is turned on. The screen will remain blank until the setup routine begins, at which point you may release the button.

39. Before calibrating the firmware, the first step is to set the capacitor voltage. Once you finished step 38, press the encoder button again. The **High-Voltage Boost Regulator** will turn on. **CAREFULLY** measure the output voltage as shown in figure 34 while adjusting the potentiometer on the converter. Adjust the voltage to the desired LED drive voltage (120 V recommended, anything higher is at your own risk).

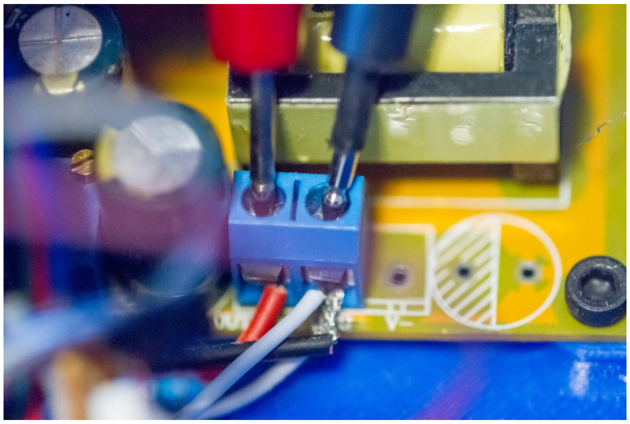


Figure 34: Output voltage from boost converter

40. Turn the **Power Switch** off, then connect the **High-Voltage Boost Converter** output to the control board (see Figure 27).
41. Hold the **Encoder Wheel** while turning the **Power Switch** on (just like step 38). Wait until the unit enters the setup routine again.
42. Measure the input voltage at the batteries as shown in figure 18. Using the **Encoder Wheel**, adjust the voltage displayed to match the input voltage. *NOTE: if you purchased a control board from me, the voltage should be correct already.*

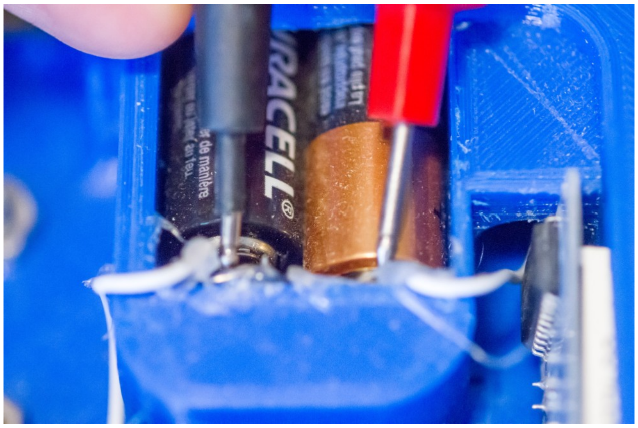
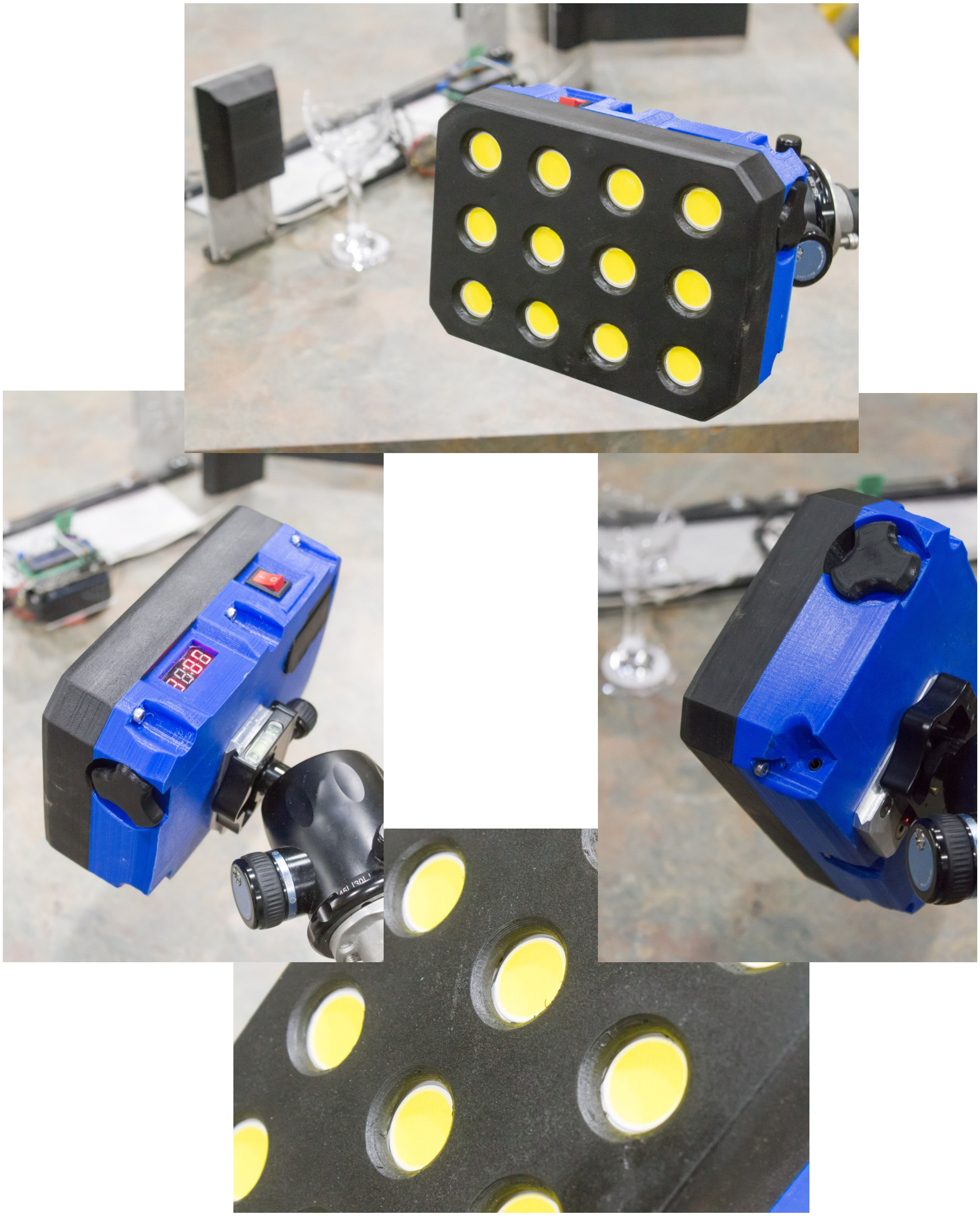


Figure 35: Input (battery) voltage measurement

43. Press the button to proceed to the next step.
44. Check that the **High-Voltage Boost Converter** is still set to the correct voltage (see Figure 34). It may have decreased slightly, go ahead and adjust to the correct voltage.
45. Using the encoder wheel, adjust the voltage display to match the **High-Voltage Boost Converter** output voltage. *NOTE: If you purchased a control board from me, the displayed voltage should already be correct.*
46. Press the button again to save the settings to memory and resume normal operation.
47. Finally, mount the **Back** to the **Front** using 6x **M4 x 20mm Screws**.

DONE!

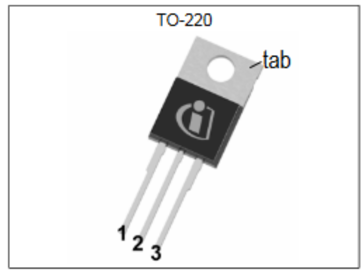


Appendix A – IPP60R190P6 Datasheet Excerpt

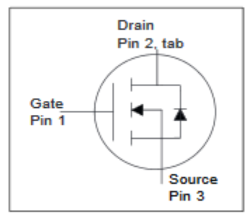
600V CoolMOS™ P6 Power Transistor

IPP60R190P6

1 Description



CoolMOS™ is a revolutionary technology for high voltage power MOSFETs, designed according to the superjunction (SJ) principle and pioneered by Infineon Technologies. CoolMOS™ P6 series combines the experience of the leading SJ MOSFET supplier with high class innovation. The offered devices provide all benefits of a fast switching SJ MOSFET while not sacrificing ease of use. Extremely low switching and conduction losses make switching applications even more efficient, more compact, lighter and cooler.



Features

- Increased MOSFET dv/dt ruggedness
- Extremely low losses due to very low FOM Rdson*Qg and Eoss
- Very high commutation ruggedness
- Easy to use/drive
- Pb-free plating, Halogen free mold compound
- Qualified for industrial grade applications according to JEDEC (J-STD20 and JESD22)

Applications

PFC stages, hard switching PWM stages and resonant switching stages for e.g. PC Silverbox, Adapter, LCD & PDP TV, Lighting, Server, Telecom and UPS.

Please note: For MOSFET paralleling the use of ferrite beads on the gate or separate totem poles is generally recommended.

Table 1 Key Performance Parameters

Parameter	Value	Unit
V_{DS} @ $T_{j,max}$	650	V
$R_{DS(on),max}$	190	mΩ
$Q_{g.typ}$	37	nC
$I_{D,pulse}$	57	A
E_{oss}@400V	4.9	µJ
Body diode di/dt	500	A/µs

Type / Ordering Code	Package	Marking	Related Links
IPP60R190P6	PG-TO 220	6R190P6	see Appendix A

Courtesy of Infineon - https://www.infineon.com/dgdl/Infineon-IPP60R190P6-DS-v02_00-en.pdf?fileId=db3a30433f2e70c5013f37a818ba23e7 – Page 2

Appendix B – Quality Control Testing Protocol

1. Board can be programmed

- *(FTDI adapter)*

2. Board can be powered by 8 - 12 v

- *(Lab PSU)*

3. Board IO functional (encoder, display, audio)

4. Connect high-voltage converter, set to 120V, calibrate firmware

5. Confirm 130V output into board, 120V @ capacitor anode header

- *(multimeter)*

6. Test high-pass filter using modified firmware to send 1,000 microsecond pulses

- *(Oscilloscope, CH1 -> Gate, CH2 -> microcontroller output, 1us/div, 0us offset)*

7. Calibrate and print pulse traces @ 0.5, 1.0, 2.0, and 4.0 microseconds

- *(Oscilloscope, CH1 -> Gate, CH2 -> microcontroller output, 500 ns / div, -2us offset)*

8. Attach pushbutton to external trigger, test NO trigger response time, print trace

- *(Oscilloscope, CH1 -> Gate, CH2 -> trigger, 500 ns / div, 0us offset)*

9. Attach pushbutton to external trigger, test NC trigger response time, print trace

- *(Oscilloscope, CH1 -> Gate, CH2 -> trigger, 500 ns / div, 0us offset)*