Cyclops is a high-power LED driver that enables precise control of light power for optogenetic stimulation. The circuit was developed by Jon Newman while in Steve Potter's lab at Georgia Tech in order to complete his thesis work, which required the delivery of ultra-precise, continuously time-varying light waveforms for optogenetic stimulation [1]. This was, and still is, not possible with commercial hardware for optogenetic stimulation. Since its first use, the circuit has been improved in terms of speed, precision, programmability, and ease of use. This document provides construction, usage, and performance documentation for the Cyclops LED driver. This document evolves with the repository. To view old revisions, checkout tags or old commits using their SHA.

Contributors

- jonnew http://www.mit.edu/~jpnewman/

Table of Contents *generated with DocToc*

Features

Circuit Features

- Ultra-precise
- High power
- Up to 1.5A per LED
- Wide bandwidth
  - ~2.5 MHz -3 dB bandwidth
  - Maximum 200 ns 1.0A rise-time
- Current and optical feedback modes
- Built-in waveform generation
- Over-current protection
- Modular
  - Arduino compatible
  - 4 synchronizable optical channels
  - Accepts external analog, gate, or trigger inputs

Stimulus Generation Options

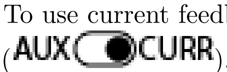
- External stimulus sequencer
- External digital trigger
  - TTL logic level
- External analog waveform generator
  - 0-5V analog signals
- Internal 12-bit DAC
  - Synchronized across up to 4 drivers
  - Powerful Arduino library
  - Programmable triggering logic
  - Respond to USB input

Performance Specifications

TODO

Usage

Current Feedback Mode

To use current feedback mode, push the rear slide switch to the CURR position (

). Using the circuit in current feedback mode ensures that the forward current across the LED is precisely regulated according the voltage at the VREF pin. This configuration is a standard method for driving LEDs because the relationship between current and LED irradiance is smooth and monotonic. This means that more current across the LED will generate more light power (while staying within the LED's maximum ratings, of course). However, the relationship between current and irradiance is not linear. For most LEDs, it looks like a logarithmic function. Additionally, the efficiency of the LED is inversely related to its temperature. So, as the LED operates and heats up, the amount of light it produces drops even when the current is held constant. The severity of an LED's temperature dependence and current/irradiance nonlinearity depend on the type of LED (roughly, the color and who made it). These properties should be clearly documented in the LED's data sheet. With a quality LED and proper thermal management, the effects of temperature and static current/irradiance nonlinearity are fairly minimal and can be ignored in most situations.

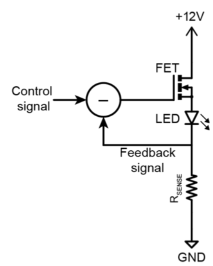
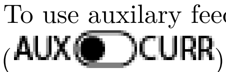


Figure 1: Current feedback configuration.

Auxiliary Feedback Mode

To use auxilary feedback mode, push the rear slide switch to the AUX position (

). When extremely stable, linear control of light power is required, the auxiliary feedback input can be used to used to compensate for the temperature dependence and static nonlinearity of the current/irradiance relationship of the LED. For example, when the auxiliary voltage is supplied by an amplified photodiode that is somewhere indecent to radiation from the LED, or is sampled from the fiber transporting LED light, the gate voltage is adjusted such that the measured light power matches a DAC-supplied reference voltage. This is the case in the circuit diagram. This configuration is referred to as optical feedback mode. The PDA36A adjustable amplified photodiode from Thorlabs is a good option for supplying optical feedback. However, you can make your own amplified photodiode for a fraction of the price, and a design is included within the cyclops repository. Optical feedback completely linearizes the relationship between a supplied reference voltage and the light power produced by the LED by compensating for the current/irradiance nonlinearities and temperature dependence.

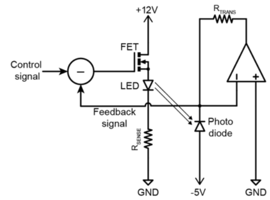


Figure 2: Optical feedback configuration.

Stimulus Generation Options

There are three ways to generate light signals using the driver. The behavior of each of these options is dependent on the feedback mode being used. The behavior of each input option is described in relation to the feedback mode of the driver.

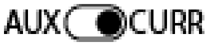
1. TEST The test button is always available and will override all other input modes. Using the `TEST` button the behavior of the circuit is:

- AUX CURR Source the current specified by the MAX CURR. dial.
- AUX CURR Generate the optical power specified by the h * mW level that is specified by the MAX POWER dial. The intensity of the LED will be dependent on the auxiliary feedback signal used which defines the ‘h’ parameter.

2. EXT DAC External input mode is engaged when the SOURCE switch is moved to the EXT position and user supplied voltage waveforms are present at the EXT BNC input. If the user attempts to supply more than 5V to the EXT input, the circuit will clamp the input signal to 5V. Using `EXT` mode, the behavior of the circuit is:

- AUX CURR Source the current specified by (EXT Voltage / 5V) * MAX CURR.
- AUX CURR Generate the optical power specified by (EXT Voltage/5V) * h * mW. The intensity of the LED will be dependent on the auxiliary feedback signal used which defines the ‘h’ parameter.

3. EXT DAC The internal DAC is engaged when the SOURCE switch is moved to the DAC position and can be used to generate pre-programmed waveforms and waveform sequences triggered by a digital pulse to the TRIG input. This feature relies on optional Arduino installation and programming the device using its API. Using the `DAC` mode, the behavior of the circuit is:

- 

Source the current specified by (DAC Voltage / 5V) * MAX CURR.
- 

Generate the optical power specified by (DAC Voltage/5V) * h * mW. The intensity of the LED will be dependent on the auxiliary feedback signal used which defines the 'h' parameter.

Construction

If you have questions during device assembly, please direct them to the open-ephys forum so that others may benefit.

Components

The bill of materials (BOM) is available on this google doc. Most of the parts can be purchased from Digikey, but there are a few components that need to be bought from other sources such as Newark, Adafruit, and Samtec. All vendor, part number, and quantity information is listed on the BOM. If you are having trouble getting a part, check the BOM since there are alternative suppliers listed for some parts. As a convenience, the spreadsheet contains links to a pre-populated Digikey cart (which are optimized for quantity discounts).

The cyclops PCB can be constructed by purchasing from one of the pre-uploaded options:

- OSH Park - made in America, excellent quality. Minimum of 3 boards per order
- Seeed Studio - made in China, very good quality. Minimum of 5 boards per order.

Alternatively, the gerber files can be uploaded to the PCB fabrication service of your choice. The layer of each gerber file is identified by its file extension:

```
*.GKO = board outline
*.GTS = top solder mask
*.GBS = bottom solder mask
*.GTO = top silk screen
*.GBO = bottom silk screen
*.GTL = top copper
*.G2L = inner layer 2 copper
*.G3L = inner layer 3 copper
*.GBL = bottom copper
*.XLN = drill hits and sizes
```

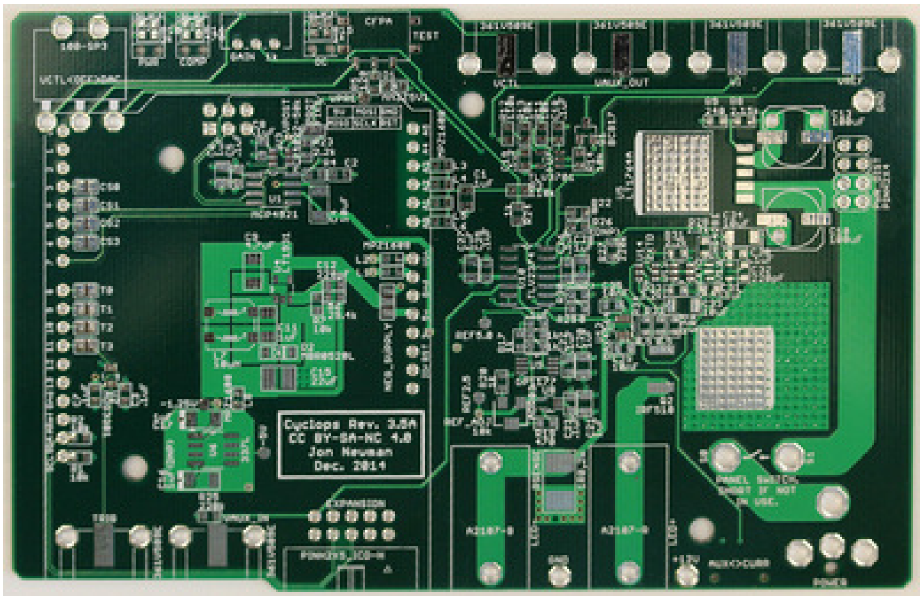


Figure 3: A bare Cyclops PCB, top side, fabricated by Seeed Studio.

PCB stencils, which are useful for applying solder paste to the boards, can be purchased from a service like OSH stencils using the gerber files located in ./cyclops/stencil/. If you plan to hand solder the board, or don't mind dispensing solder paste yourself, then you do not need to purchase these stencils.

The BOM includes several optional components, which are not in the pre-populated Digikey cart. These include:

- An **extruded aluminum enclosure**, which houses the completed board. The enclosure is recommended because the large voltages and current transients used to drive high power LEDs can cause capacitive and inductive interference with nearby recording equipment. Acrylic front and rear panels can be purchased from Ponoko using the links supplied in the BOM. The instructions below show how these plastic pieces are modified to provide proper electrical shielding.
- An **M8-4 connector**. This is a rather expensive connector that allows cyclops to drive Thorlabs LED modules or Doric LED modules.

Board Assembly

To assemble a Cyclops board, you will need the following materials

- A soldering device.

- At minimum, a soldering iron regulated to ~370 deg. c) will do the job.
- A hot-air rework tool or reflow oven are recommended and the assembly instructions below assume you are using one of these two options. A low cost, high-quality hot-air rework station can be purchased from SparkFun here.

- Copper braid ('solder wick') for solder removal (e.g this)
- Liquid flux (`no-clean` variants are easiest since they don't have to be thoroughly removed after use)
- Stereoscope or loupe (optional but nice for tracking down shorts.)
- Isopropyl alcohol for cleaning flux off the board (e.g. this; optional)
- An anti-static mat (e.g. this; optional but recommended to protect your work. . .)

PCB component population and soldering is fairly straightforward and requires standard surface mount construction techniques.

- A tutorial on hot-air soldering can be found here.
- A great tutorial filled with general tips and tricks for surface mount soldering can be found here.

The following steps provide a visual guide to construct your own board. The goal is to create a fully populated PCB like this one:

Figure 4: Finished device (revision 3.3).

Following board construction, you should run through the electrical tests outlined in the next section before applying power.

0. Place the bare PCB on a flat surface, preferably one that is static dissipative or anti-static. Alternatively, the board can be mounted in a PCB vice.

1. Below are pictured some of the materials you will need to construct a board.

2. The silkscreen layer on the PCB (white text) has almost all the information you will need to properly populated the PCB. However, its a good ideal to

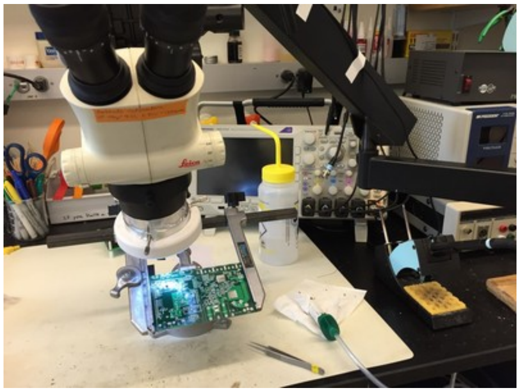


Figure 5: Instead of populating components on a table, holding the PCB using a PanaVise can be helpful.

Figure 6: Wire solder and an soldering iron can be used to construct the PCB, but solder paste combined with a hot air rework station or a reflow oven makes things much easier. We use Chipquik 291ax10.

Figure 7: Circuit components. A complete bill of materials is provided in the BOM.

Figure 8: A soldering iron can be used to assemble the PCB, but a hot air rework station makes things much easier. These can be purchased from Sparkfun.

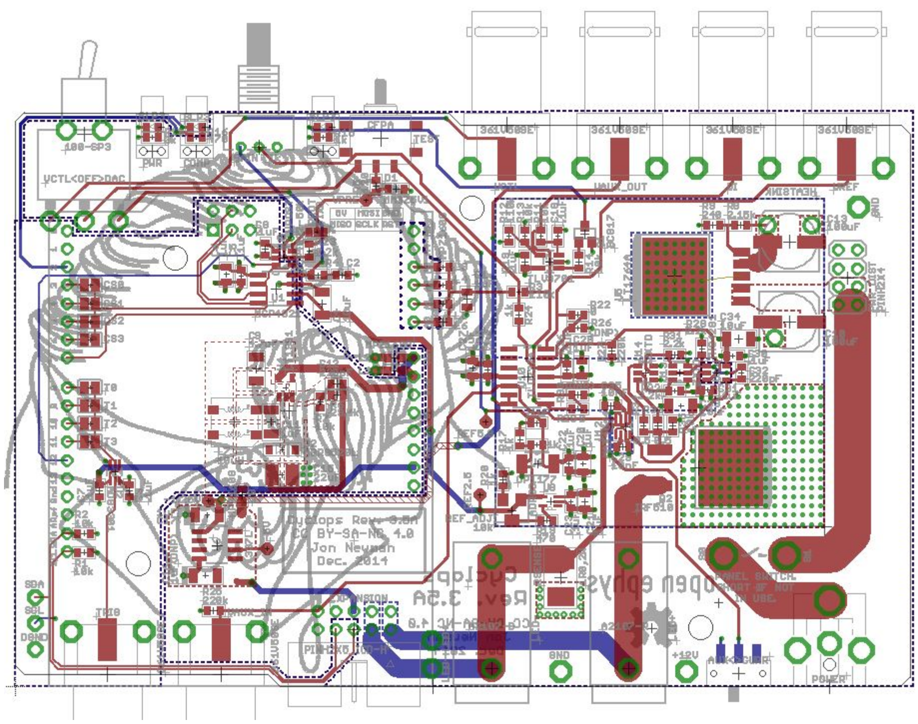


Figure 9: The cyclops PCB design in CadSoft EAGLE

to open the cyclops design in EAGLE. This will allow you to get detailed information on components before placing them on the board.

You can then the `information` tool to get detailed information on each component, e.g. to ensure you are placing the correct value resistor or capacitor.

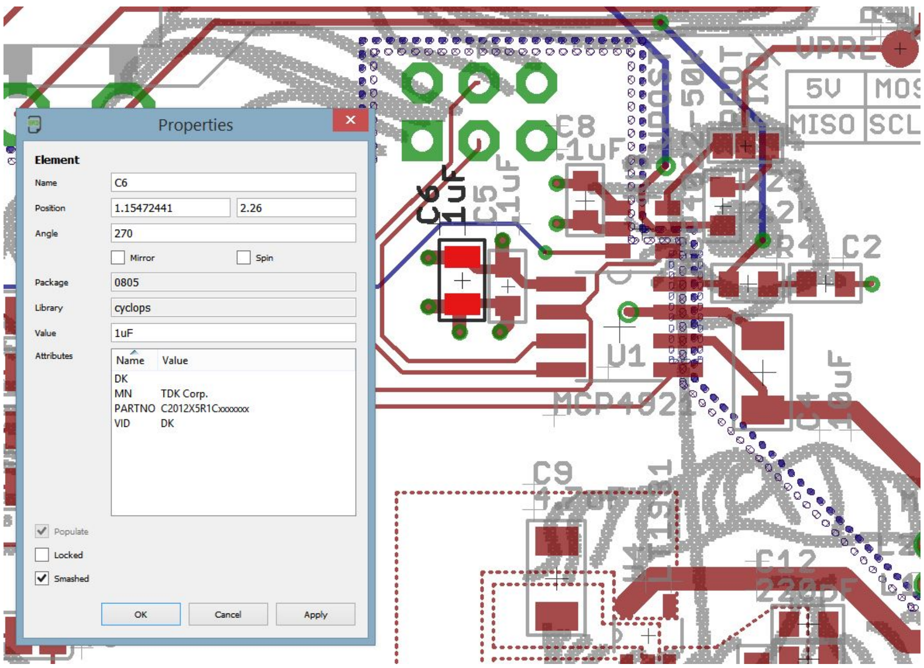


Figure 10: By selecting the information tool and clicking the cross at the center of a component, you can pull up detailed info (e.g. part number)

3. After cleaning the surface of the board with isopropyl acholhol or similar, apply solder paste to each of the pads. For an excellent series of tips on effective methods for dispensing solder paste, see Mike's video on the subject. Do not apply solder paste to through-holes or the pads shown outlined in red in the following image. These will be hand soldered later in the assembly process.

   TODO

   The correct amount of solder paste to apply is 'enough'. Each component contact should sit in a small amount of paste, but blobs of paste that envelop the component pad or pin may later result in a short. The following images show examples of good and bad solder placement.

   If you need to pause at any point, you should store place the PCB in the fridge to prevent the flux in the solder paste from breaking down.

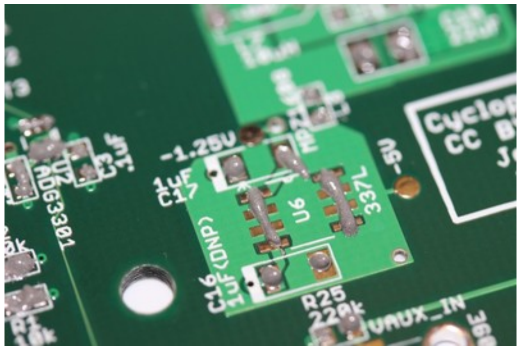


Figure 11: Good solder placement.

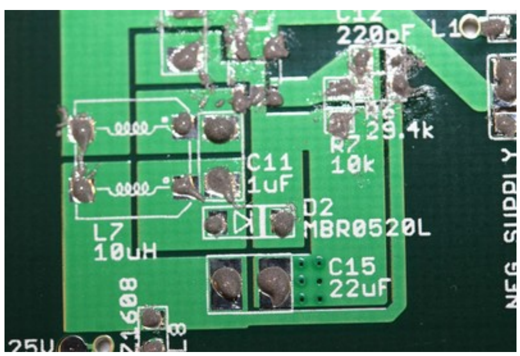


Figure 12: Bad solder placement. Too much paste!

4. You can look at the picture below and put solder on the exact same gold plates as in the picture. I actually missed some gold plates at this step, but don't worry for now. You can put solder just like this.

   Fig. 7 After putting solder paste to all necessary slots (a few are missing here)

   Fig. 8 Closer look 1. Too much solder in general, but will work

   Fig. 9 Closer look 2. Too much solder in general, but will work

5. Populate all **top-side surface mount** components on the board. There is a single surface mount switch on the back of the board that will be hand soldered later. Additionally, all through hole components (e.g. power jack, BNC connectors, etc) will be populated later. Start by placing the integrated circuits (ICs). Use the stereoscope or loupe to ensure that pads are making contact with the pins of the placed components. Precise component alignment is not necessary. Components will self-align during the reflow process.

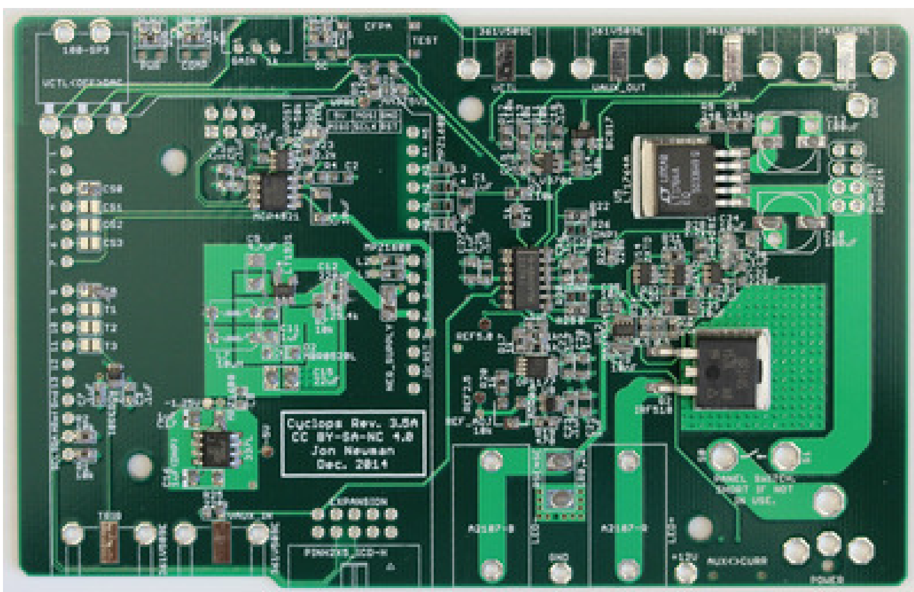


Figure 13: Ingegrated circuit population.

6. After placing the ICs, place the passive components (resistors, capacitors, inductors, and diodes).

7. Next step is to reflow solder. Let me show my homemade reflow oven. You can make a similar one or use a commercial one. Reflow the solder paste on the board using your oven or hot air gun as described in the links above. After the solder has cooled, examine solder pads using the stereoscope

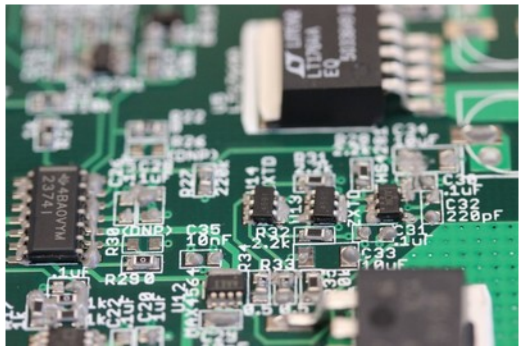


Figure 14: Zoomed view of ingegrated circuit placement.

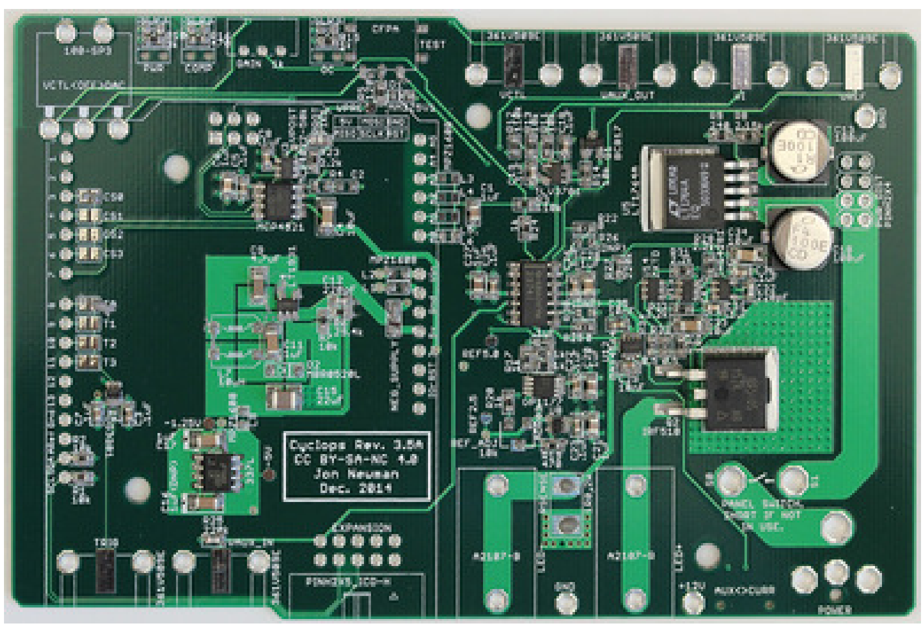


Figure 15: Ingegrated circuit population.

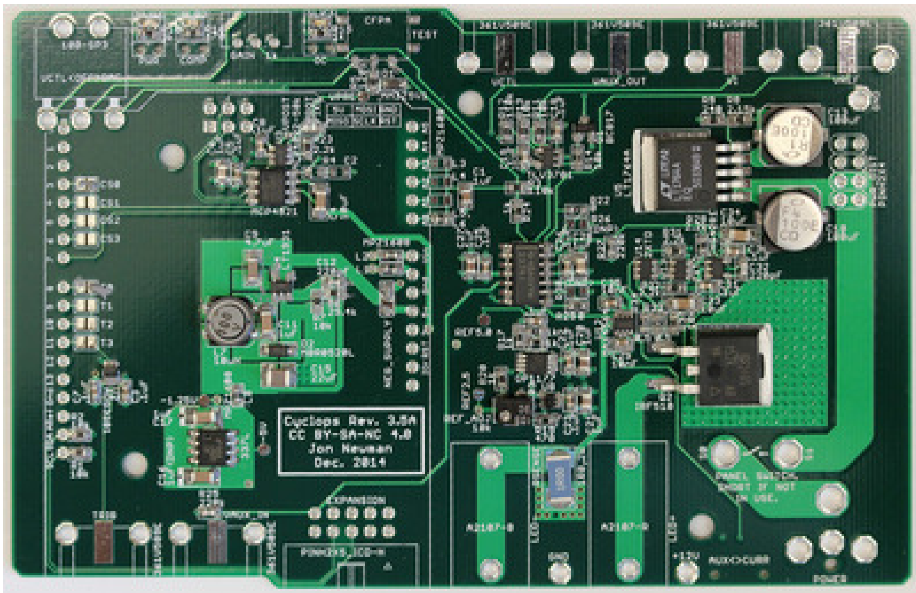


Figure 16: Ingegrated circuit population.

of loupe for solder bridges between pins, solder that has not melted, or pads lacking a decent solder joint Fix any issues using a standard soldering iron If there are solder bridges present, get rid of them using some solder wick before moving on. Solder through-hole components in place using a standard soldering iron. A low cost reflow oven can be made form a toaster oven as shown here. This link also contains useful information on the basics of the reflow soldering process,

Fig. 14 Carefully bring the PCB and place it in the oven like this.

Here is a closer look.

A board…

As it goes above 200C, the solder starts to reflow. Like a live worm

8. After reflow, take the PCB out and check for possible shorts. We need to fix it.

   Fig. 15 See the blob of solder there?

   Fig. 16 Use this solder remover and flux

   Fig. 17 Bring it to the solder blob and sort of press it down with solder. Then the solder gets absorbed to the remover.

   Fig. 18 Problem fixed! There is enough solder after removal, so mostly you don't need to worry about connection.

By the way, there are some differences between different versions of the board. For example, 1 kOhm there in the middle of the picture was replaced to a 0 ohm one in the newer version.

9. Now populate all electromechanical parts. The barrel power jack (name: "POWER", value: PJ-063BH on the schematic) should be mounted on the *bottom* of the board. It fits on both the top and the bottom, and will properly supply the board with power if mounted on the top. However, if the barrel jack is mounted on the top side of the board, it will not fit inside the enclosure.

   Fig. 19 After populating all electromechanicals. Front side.

   Fig. 20 After populating all electromechanicals. Back side.

   Front side..

   Front side, with different camera focus..

   Back side..

   Back side, with different camera focus..

10. Don't forget to install the heatsink.

11. The light pipes over the front LEDs need to be seated firmly for the board to fit inside the enclosure.

12. Each board has an address (0 through 3) that is defined by two solder jumpers and the location of a ferrite chip. This allows cyclops boards to be stacked to share a power supply while being driven by a common microcontroller. For each board that will share a microcontroller, a unique address must be specified and the solder jumpers and ferrite chip must be soldered in appropriate positions to reflect this address. See the picture below to better understand this addressing scheme.

13. Finally, let's install the power switch. You need to hookup wire capable of handling the currents that the driver requires. AWG 20 (~0.8 mm diameter) braided copper wire or thicker is recommended. Even if you don't want to use the power switch, jumper the switch solder points using AWG 20 wire or thicker.

    Fig. 21 Power switch.

    Fig. 22 Front side.

    Fig. 23 Back side.

Enclosure

To construct the enclosure, you we will use the following materials

- Phillips head screwdriver (if you are using the enclosure)

- A white paint pen (e.g. these)
- Conductive coating for EMI suppression (e.g. this).

Circuit testing

To perform basic electrical testing, you we will use the following materials

- Multimeter. A low cost mulitmeter is available from sparkfun.
- Jeweler's flat head screwdriver.
- Oscilloscope (optional, but recommended for performance verification)

0. Before powering on the device, check for shorts between power traces on the board. Put your multimeter in continuity mode Check for shorts between the Digital rail (TPXX) and ground (TPGND) Analog rail and ground If there is a short, you must track it down and get rid of it before applying power. If you find a short, test the same contact points on an unpopluated PCB to ensure that it is not due to a PCB fabrication defect. If so, contact your PCB for a return.

License

Hardware Licensing

Software Licensing

References

[1] T. Tchumatchenko*, J.P. Newman*, M.-f. Fong, S.M. Potter. Delivery of time-varying stimuli using ChR2. (* - equal contributions, co-first authors) Front. Neural Circuits (7:184) 2013. doi: 10.3389/fncir.2013.00184