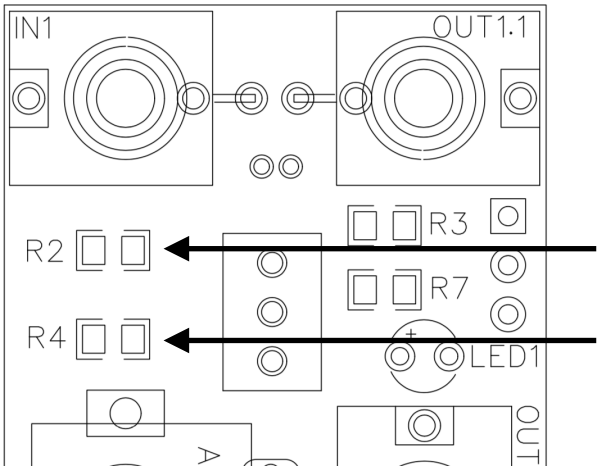
LOOM Build Guide - Rev 1.1

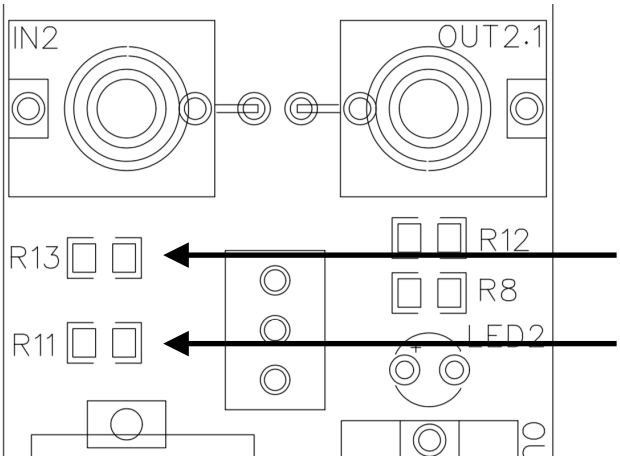
First things first: solder down the resistors.

Resistors

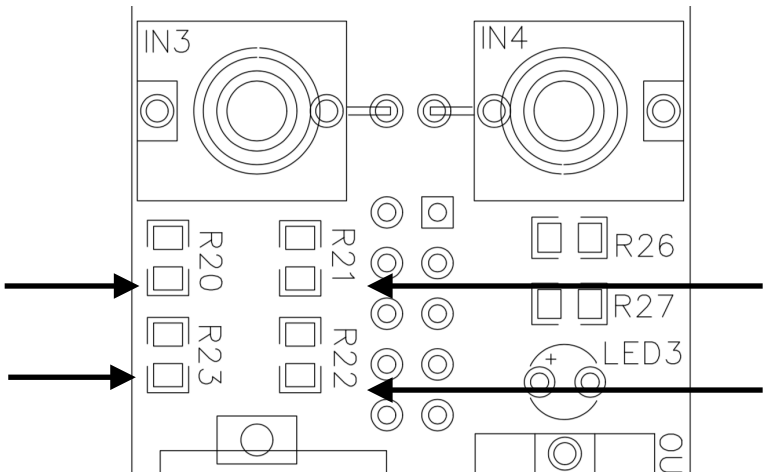
1. Start on the front of the board, soldering R2 and R4. Both are 100K



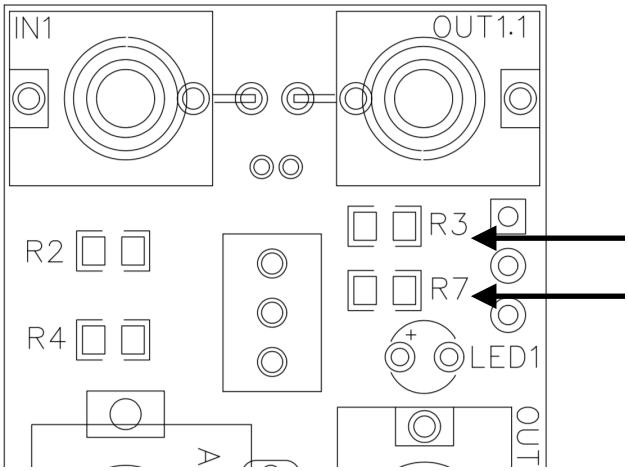
2. Next, solder R13 and R11. Again, both are 100K



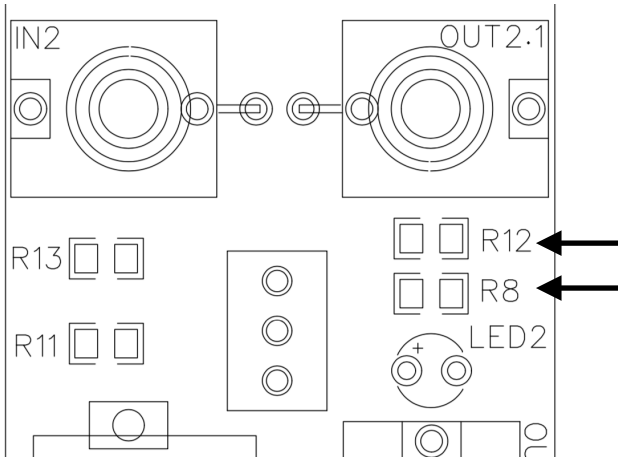
3. Solder R20, R21, R22, and R23. All are 100K



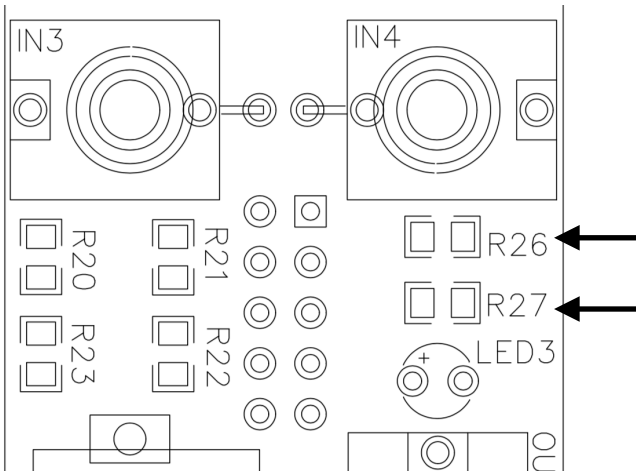
4. Moving onto the 1K resistors on the front! Solder R3 and R7



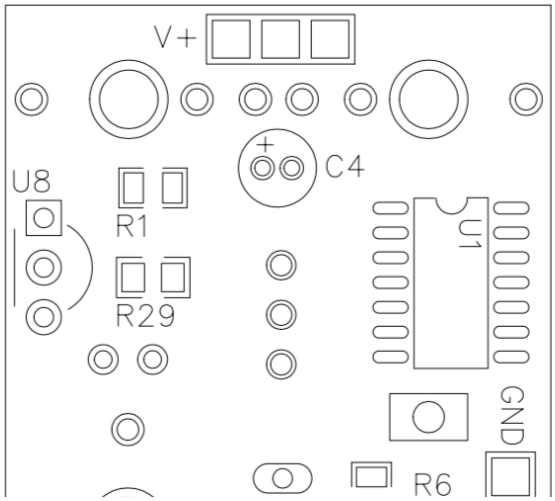
4. Solder R12 and R8. Both are 1K



5. Solder R26 and R27. Again, both are 1K

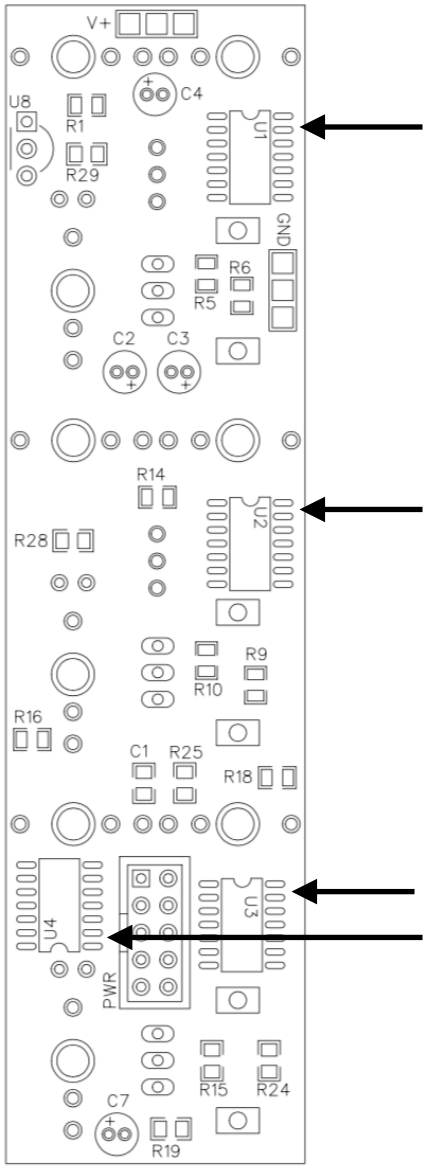


6. Flip the board over. Solder R29, a 1K resistor

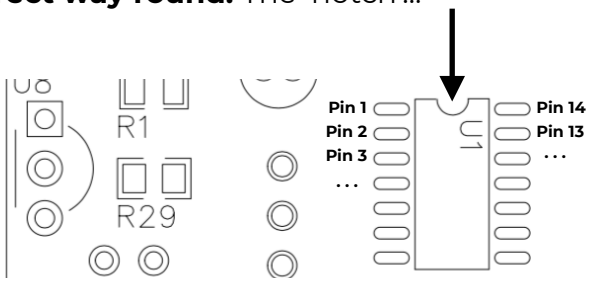


SMD Chips

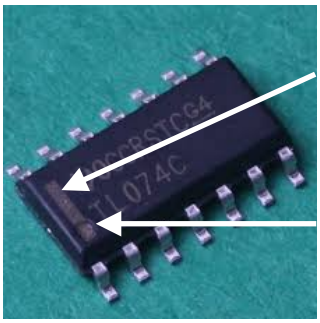
7. Next, move onto U1-U4. These are all TL074 in a SMD package (see Bill of Materials for more details!) . Soldering these chips is definitely the most delicate part of this build; have a look at resources online for soldering techniques for these.



Remember to **ensure that the chip is the correct way round.** The 'notch'...



...indicates where pin 1 is, as most chips have a hemispherical notch to the right of pin 1. On SMD chips, the indication may be different, such as a solid bar of white near the top of the chip, or a small dot next to pin 1 (or both!).



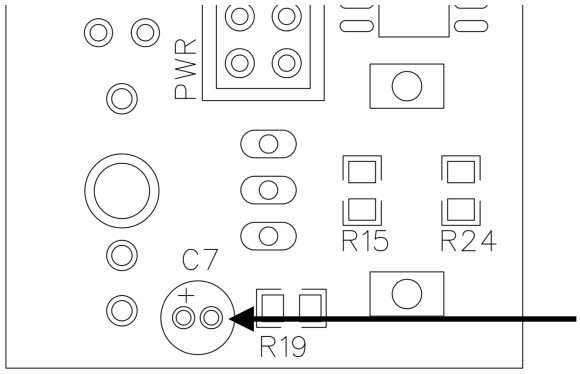
An example of a TL074 with a bar instead of a notch...

...and a dot to indicate pin 1.

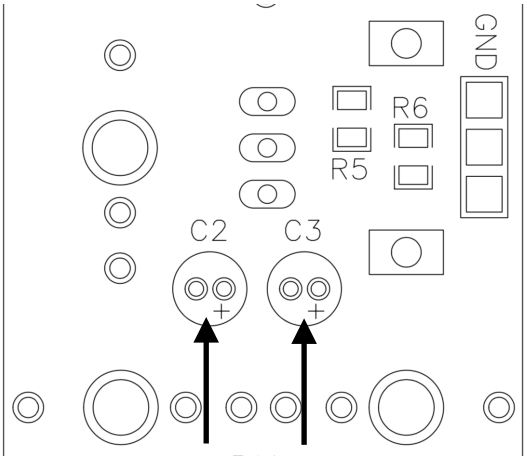
Image source: http://fec.4art-studio.com/product/TL-074-SMD/2437

Capacitors and other chips

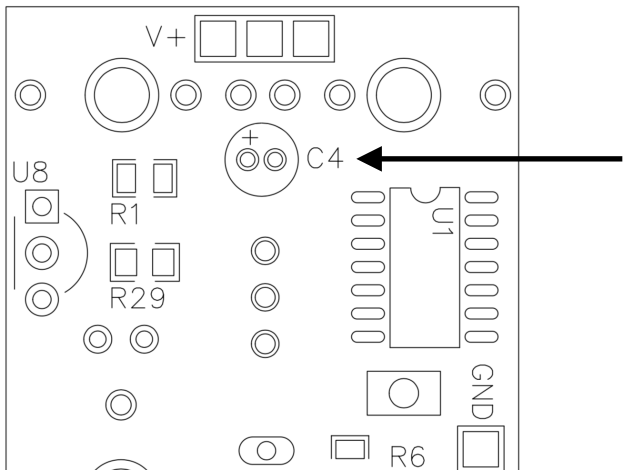
8. Solder the capacitor C7 (0.1uF, electrolytic). **Ensure that the polarity is correct** (negative lead is in the negative hole; positive lead in positive hole).



9. Solder C2 and C3 (10uF, electrolytic). **Again, ensure that the polarity is correct.**

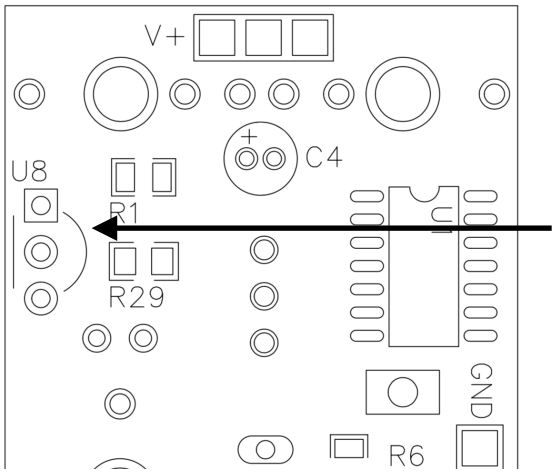


10. Solder the last electrolytic capacitor, C4 (0.1uF, electrolytic). **Ensure that the polarity is correct** (I promise I won't say it again!).



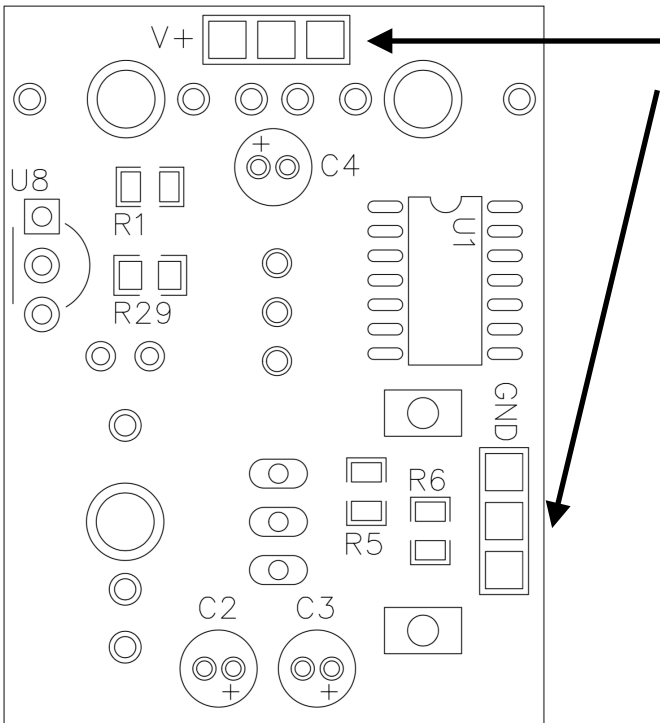
NB If you are building the module with no reference voltages, you can leave this step out.

11. Solder U8, the voltage regulator! The square pad indicates where the output voltage pin is, and the outline of the shape of the component shows the orientation. The voltage regulator can be any low power, positive regulator you like: I recommend the 78L05 (for a +5V reference voltage) or a 78L09 (for +9V reference voltage). Other regulators will work but check the data sheet first (i.e. do they work when given +12V?), and check the pins before you solder them in, i.e. make sure that the square pad is soldered to the chip's output and check that the middle pad is connected to ground.



If you would like a +12V reference, then you could place a jumper wire between the bottom and top (square) pad. (I have not tested this but there is no reason it should not work!). ***If you do this, be very careful to insulate the middle pad/jumper wire from the two pads that you are soldering together*** *, otherwise you might short your +12V power supply rail to ground...which is nightmare fuel.*

12. Now, decide on which section you want to output a reference voltage, or not! (see 'Module overview' for more information). This can be done via the jumpers on the board (see picture).

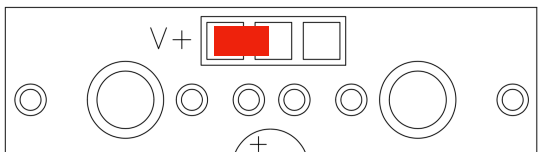


The Jumper for the top section

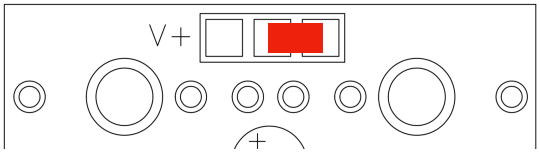
The Jumper for the middle section

To enable a voltage reference on a section, simply jumper the middle pad to the V+ pad. If you do not want a reference, be sure to jumper the middle pad to ground. This jumper can be made with solder bridging the two pads, or a small piece of wire.

Ensure that the middle pad is only bridged to one of the outer pads. If the middle pad is jumped to both V+ and GND, the voltage from the voltage regulator will be shorted to ground, damaging or destroying the regulator.



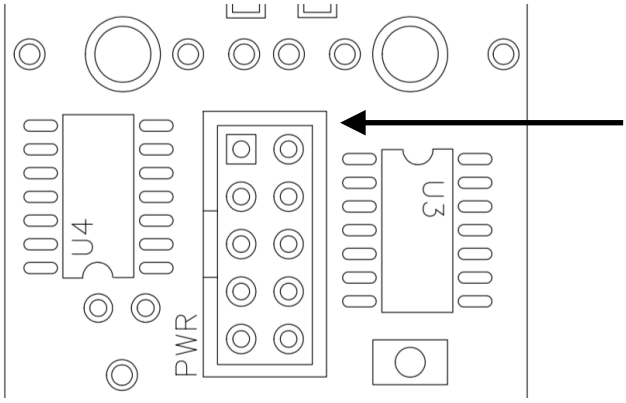
E.g. this jumper (red is solder) will provide a voltage reference for the top section.



This will provide no voltage reference, and will make sure the input is silent when nothing is patched.

Power pins

13. Solder in the pin header for the power cable. I advise using bare pin headers for this build. If you plan to use a Eurorack 'shrouded' box connector, see below:

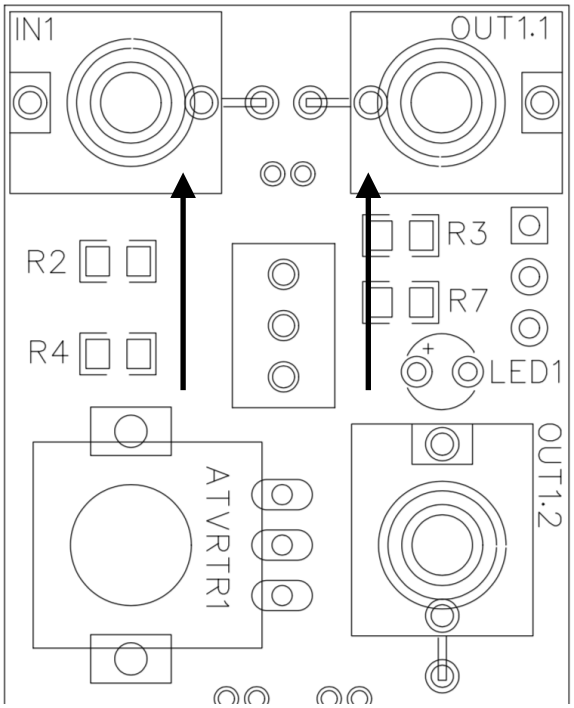


If you plan to solder in a box connector, ensure that it is orientated the correct way so that the cable will not connect backwards. The footprint indicates that the 'key' slot should go next to U4, ***which is different for different connectors****. Plug a power cable into the box and make sure that the red stripe lines up with the -12V marking on the PCB before soldering it.*

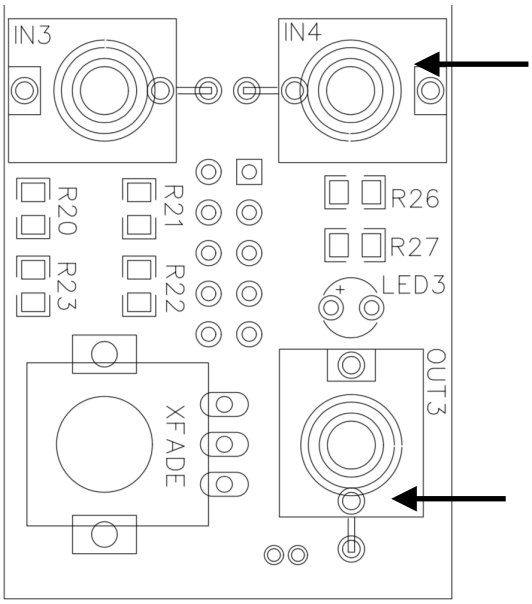
Jacks and potentiometers

There are many ways to solder these parts in; some people like to fit everything to the panel before soldering all the jacks and potentiometers down. I like to check that everything fits as I go along.

14. Turn the board over and begin by soldering the two jacks (IN1 and OUT1.1) at the top of the board.

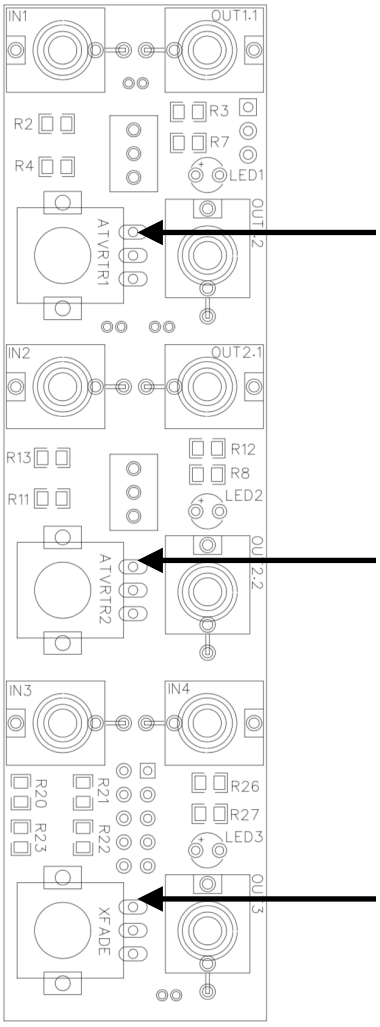


15. Next, solder in two jacks at the bottom of the board (IN4 and OUT3)



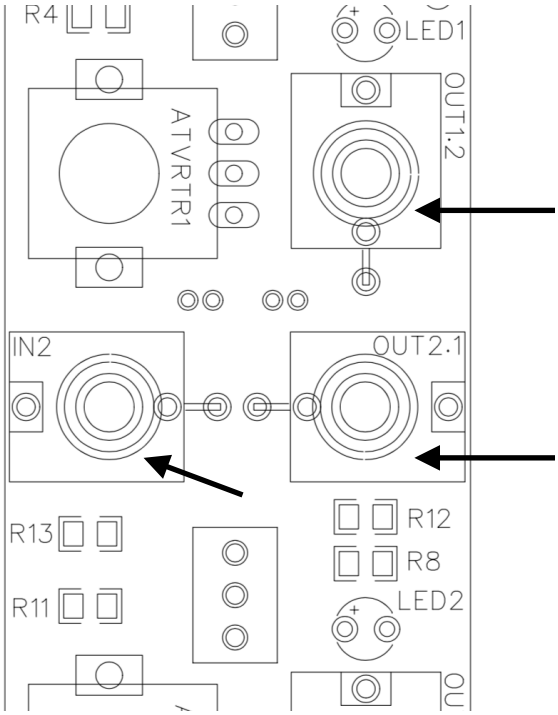
16. Next, solder down the potentiometers (3 x B100K). Solder the three front pins of the potentiometer first.

The side holes for supports do not fit the tabs for tall trimmer potentiometers from Thonk, but they may fit others. If they do not fit, clip the supports off so that the potentiometer fits flush to the board, and you can either leave the potentiometer as it is, or solder some thick gauge copper wire through the holes and to the potentiometer's casing to provide support.

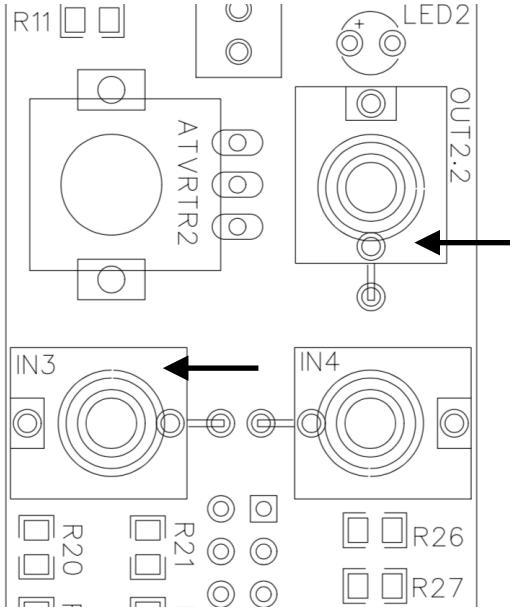


Whether the supports fit or not or whether you need to solder new ones will depend on which potentiometer you are using in the build. If you are securing the potentiometer to the panel with nuts and washers, you may not need the extra supports.

17. Next, solder in jacks IN2, OUT2.1, and OUT1.2.



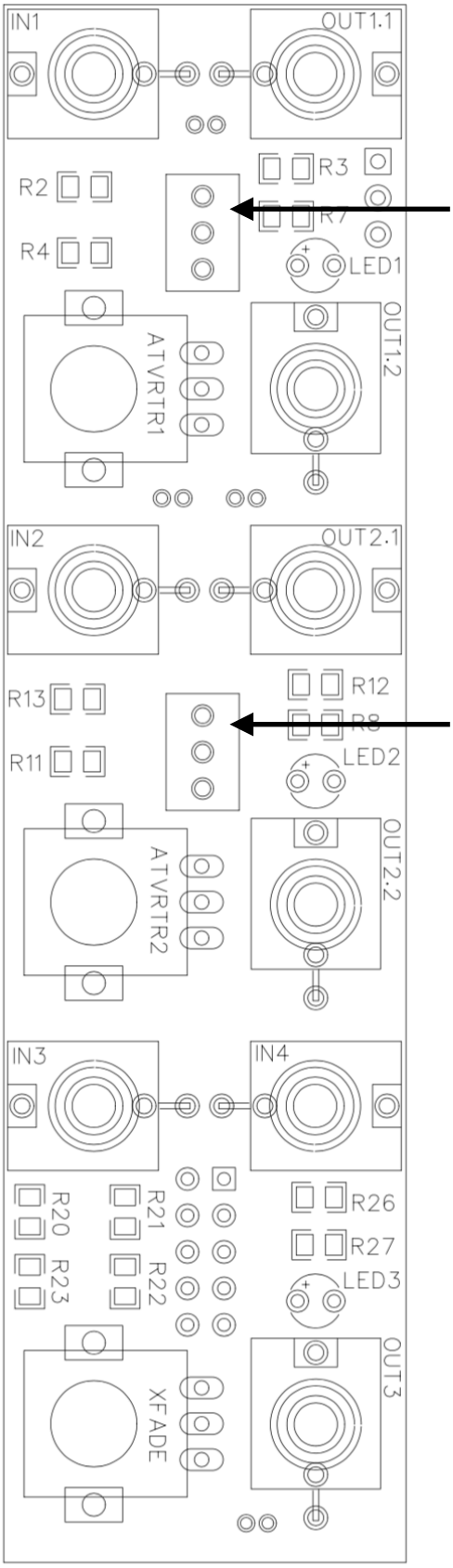
18. Finally, solder jacks OUT2.2 and IN3.



19. Finally, place the panel over the PCB to ensure that all of the jacks and potentiometers fit.

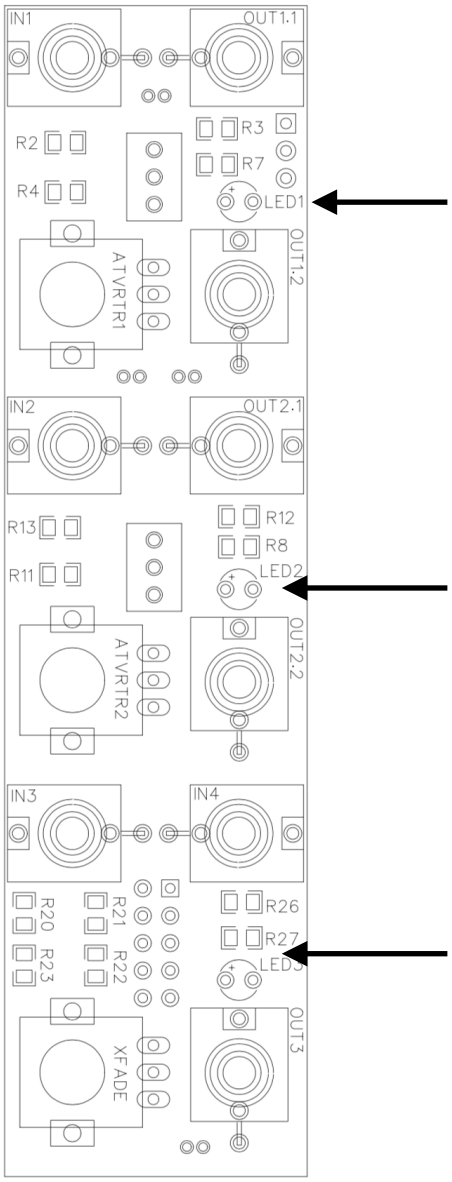
Switches

20. Finally solder in the switches. Start with the top switch, place it into the board, place the panel onto the module, and solder the pins into the board. This will ensure that the switches will fit into the board, and also allows you to check whether they are soldered in straight.



LEDs

21. The final soldering step is to solder the bicolour LEDs in. Place the LEDs into the PCB slots, with the shorter leg in the positive (+) hole (NB this is the opposite to normal LEDs). This will make the LED green if there is a positive voltage on the output jack of each section, and red if there is a negative voltage.



If you want different colours for positive/ negative responses or are using different LEDs, test the LEDs with a multimeter/look at their data sheet to see which colour is which for forward and backwards directions.

If you are using normal uni-directional LEDs, solder them in as indicated on the board i.e. longer leg into the positive hole.

Next, place the panel onto the PCB. Push the LEDs up into their holes onto the panel, so that they stick through the holes. When you are happy with their placement, solder them in on the PCB, and clip excess wire off with pliers.

Finishing Touches

22. All the soldering is finished! Screw the jacks and switches to the panel, and place the knobs onto the potentiometers.

23. See the 'Module Guide' for advice for plugging the module in.

If you have any issues with the build, feel free to get in touch! Freddie