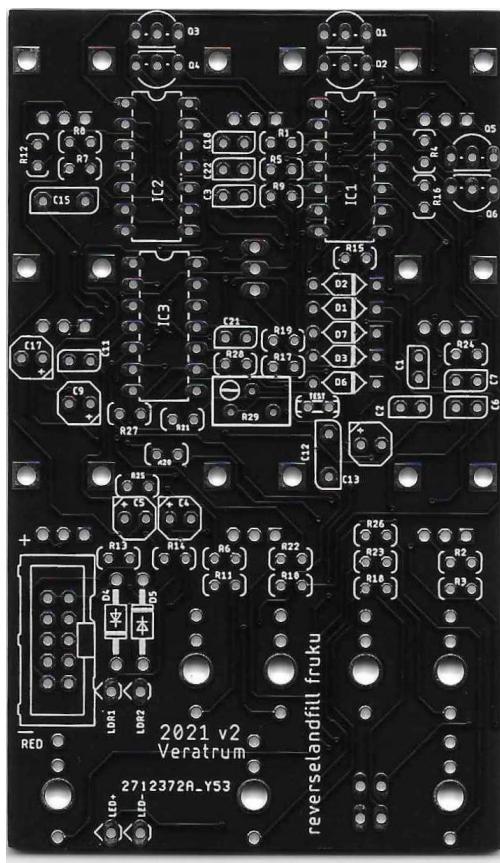
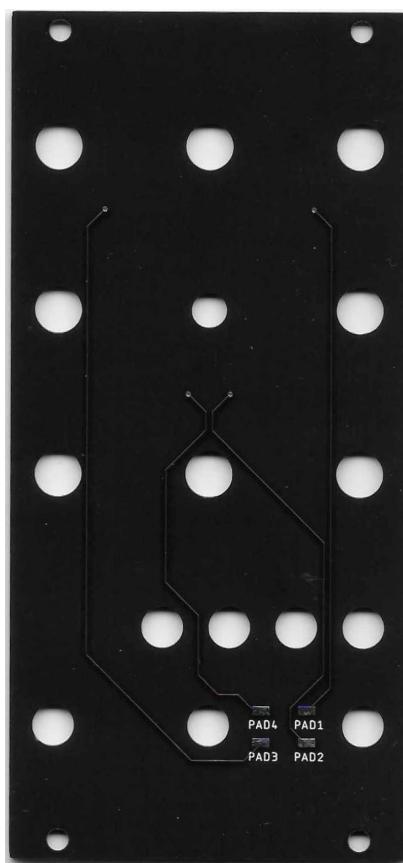
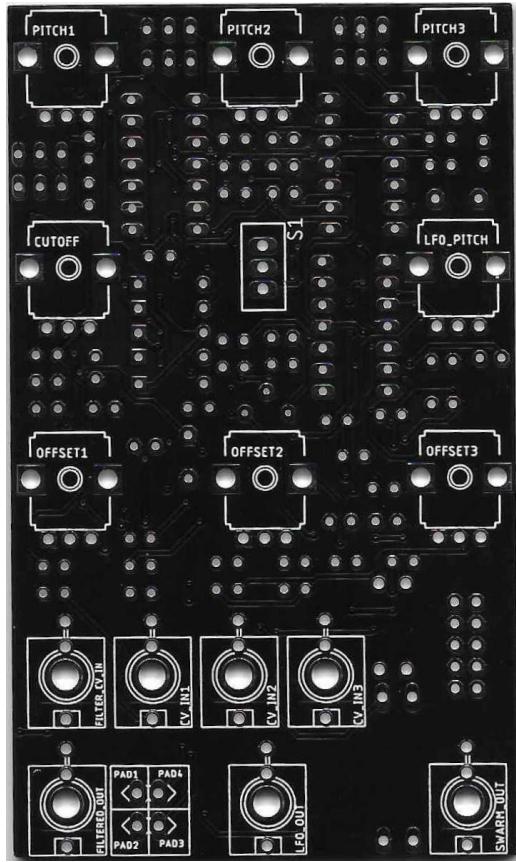
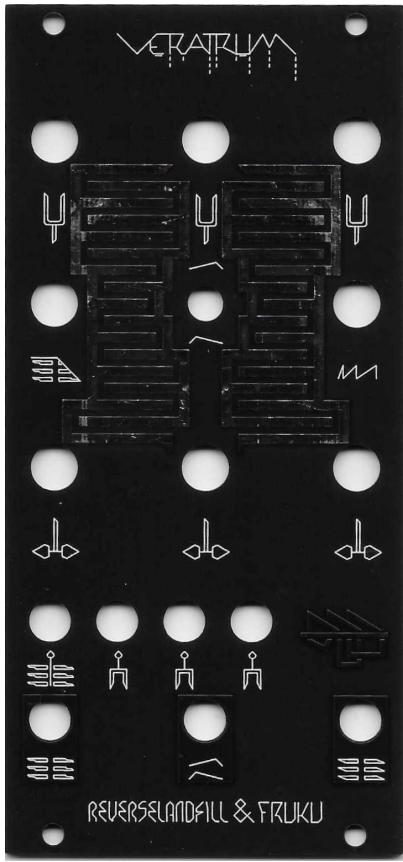
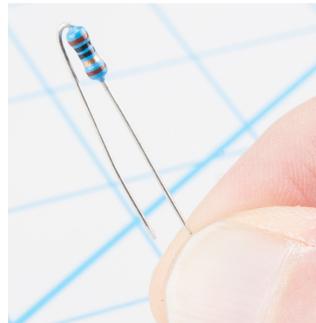


Veratrum Build Guide

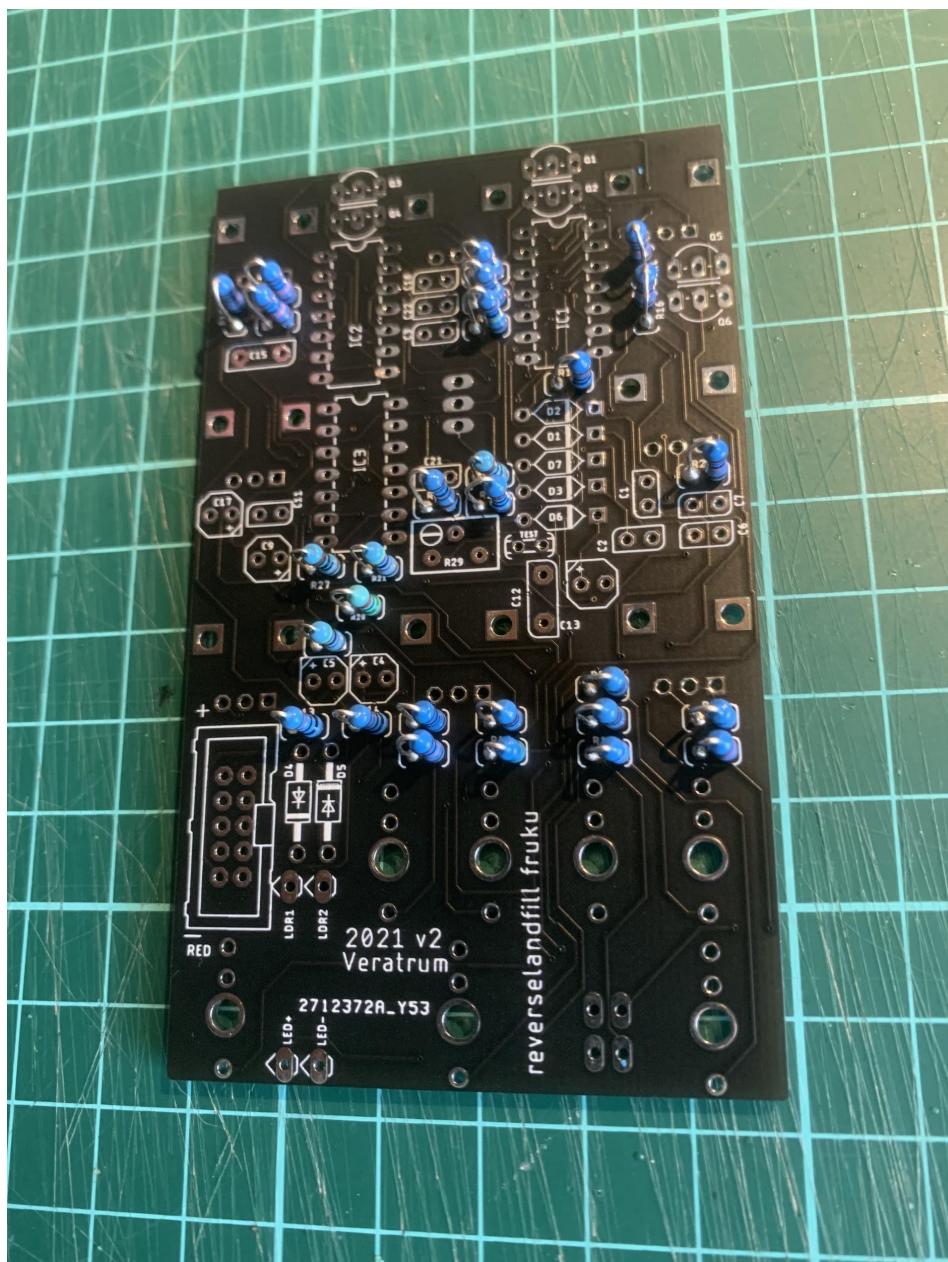


Resistors:

Start with soldering the resistors, these need to be placed "standing up". The value of the resistors are written on the tape. If you are unsure, use a Multimeter to check the value. Check the "Resistor Places" file, there you see color coded markings that correspond to the BOM.



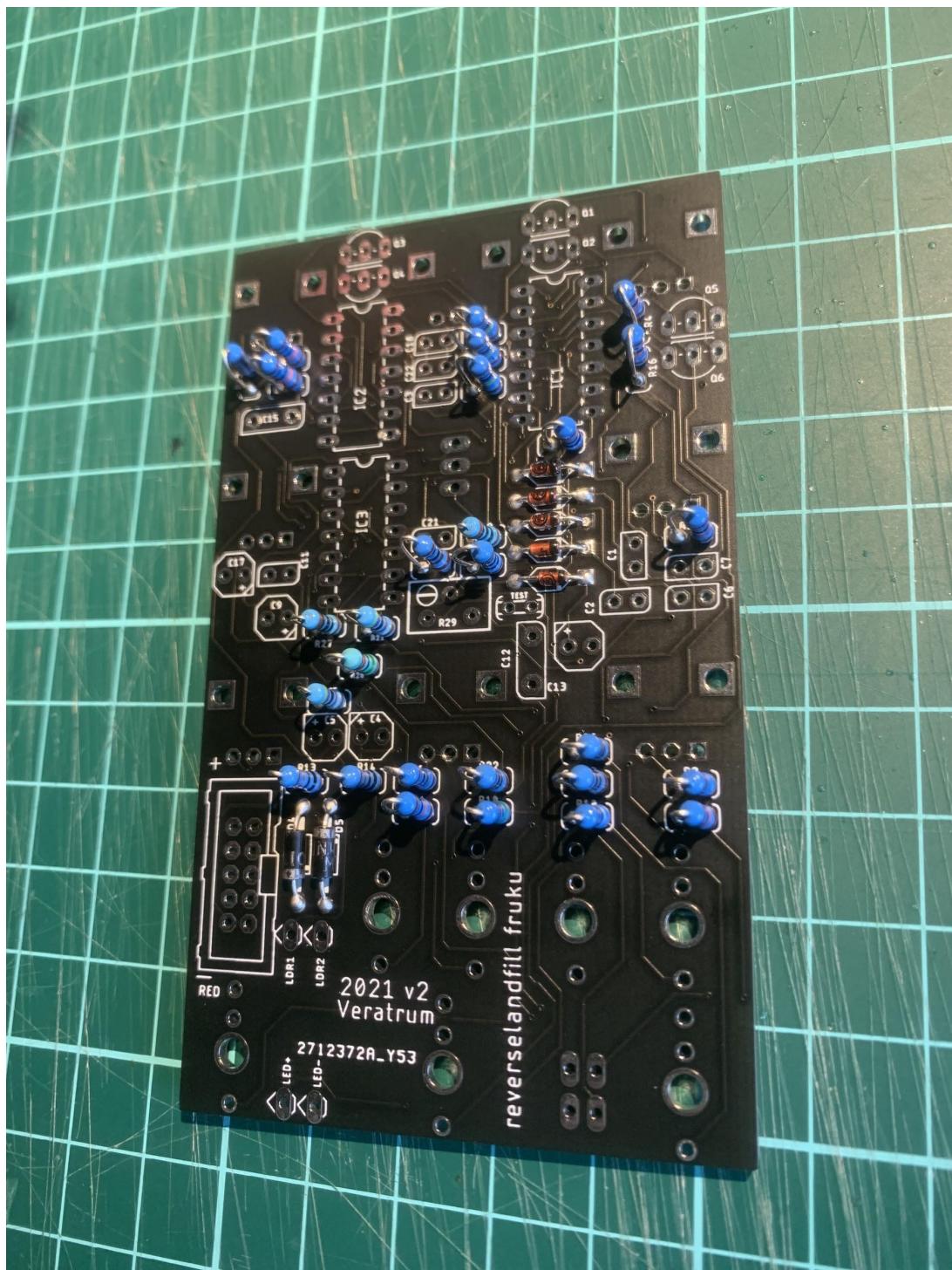
After soldering the resistors it should look like this:



Diodes:

Now solder the diodes, look at the BOM for the values and placements. Diodes are polarized so please take notice of the **positive** and **negative** side.

After soldering the diodes it should look like this:



IC Sockets:

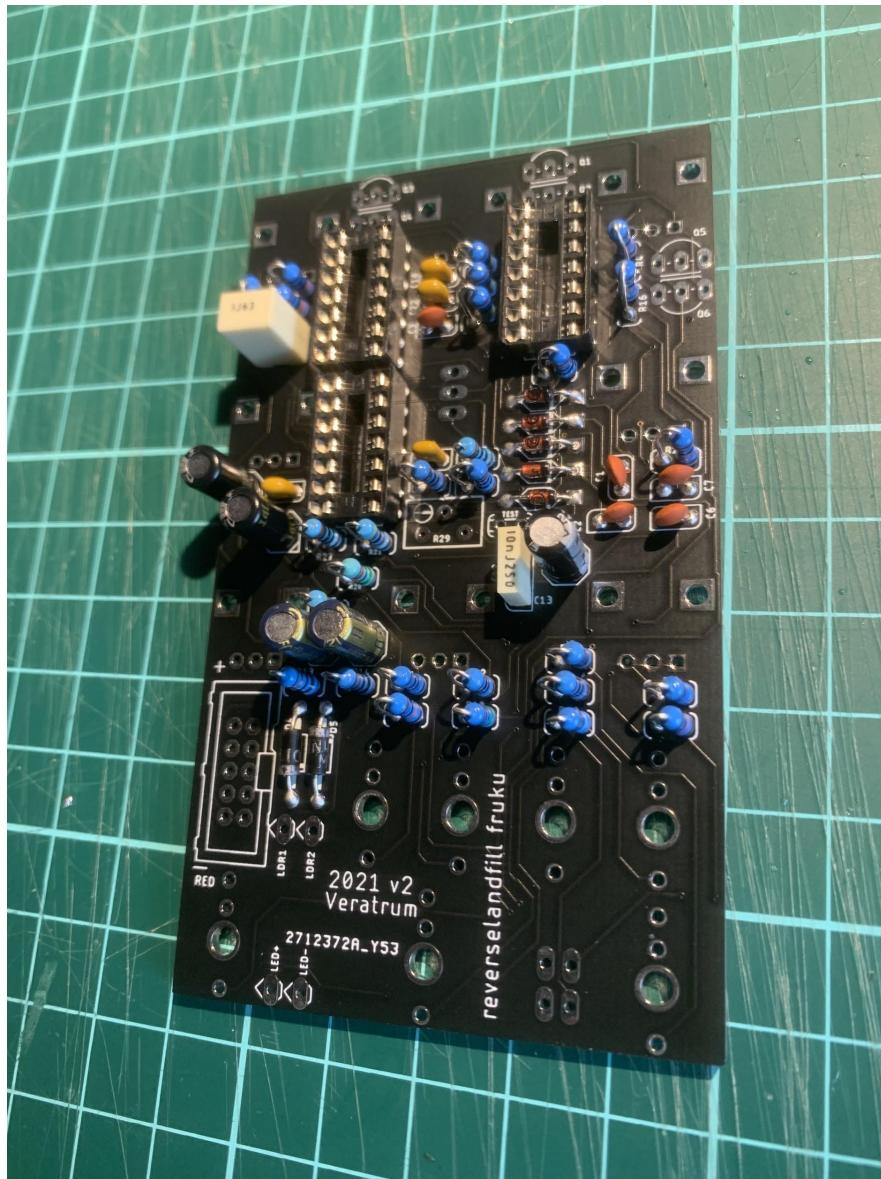
Solder the 3x 14pin IC sockets to **IC1**, **IC2** and **U1**. Take care to orientate them properly. The notch on one end should match the silkscreen. First solder just 2 opposite pins and check if the socket is aligned flat to the pcb. If not, slightly press down on the socket and reheat the pins.

Now solder all remaining pins. Leave the IC's out for now.

Capacitors:

Now solder the capacitors, look at the BOM for the values and placements. Please take notice of the positive and negative side with the electrolytic capacitors.

After soldering the capacitors and IC sockets it should look like this:

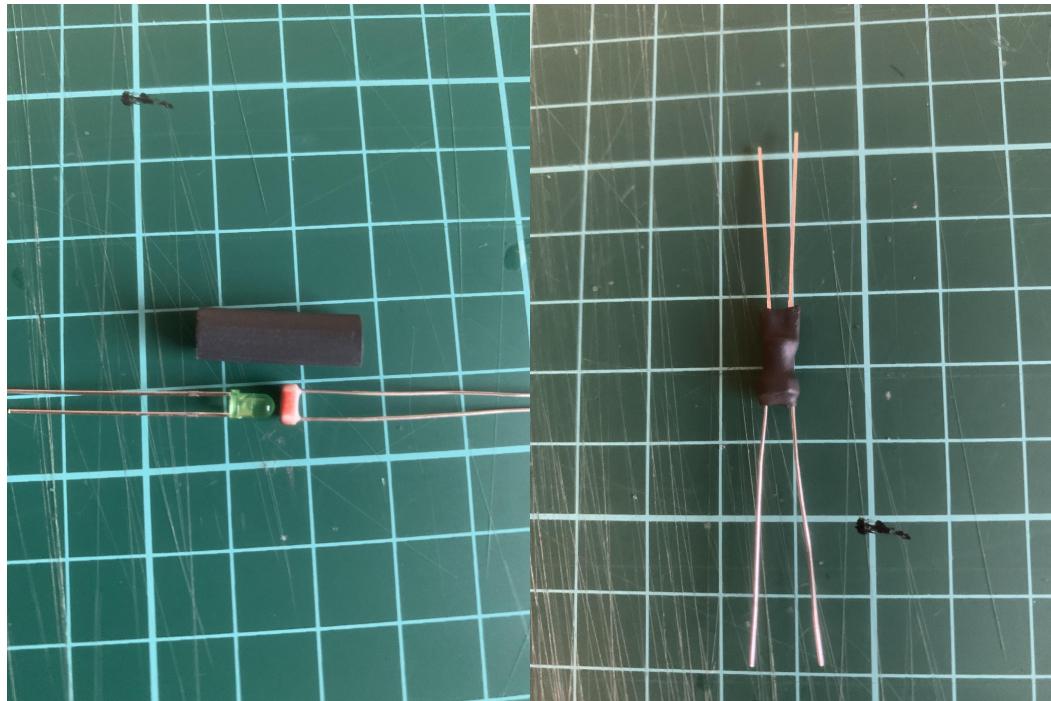


Transistors:

Now solder the transistors, look at the BOM for the values and placements. Please take notice of the orientation, the transistor part should match the silkscreen.

Vactrol:

Now it's time to make a vactrol, you can use heat shrink or electrical tape just make sure it's as dark as you can make it for the LED & LDR.



Once you have made the vactrol you can solder it in place at the bottom left side of the pcb. Please note that the long leg of the LED goes in the LED+ hole and the short leg of the LED in the LED- hole.

Power Header:

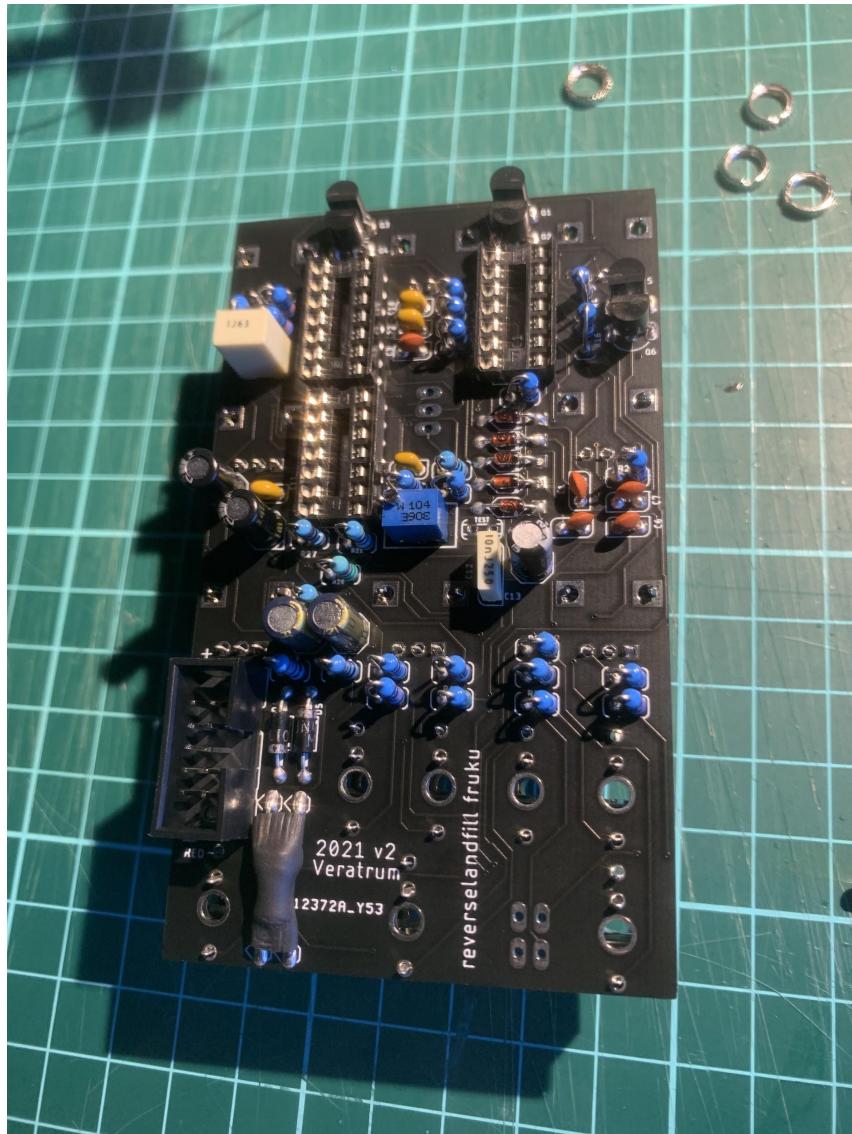
Insert the 10pin shrouded power header in place. This part also has an orientation; the open side. Make sure the part matches the silkscreen marking on the pcb.

Then solder one pin and check if the header is aligned correctly. If not, slightly press the header and reheat the pin. It should click into place. Now solder all remaining pins.

Trimpot:

Now solder the trimpot, please match the orientation as shown on the silkscreen.

After soldering the transistors, vactrol, power header and trim pot it should look like this:



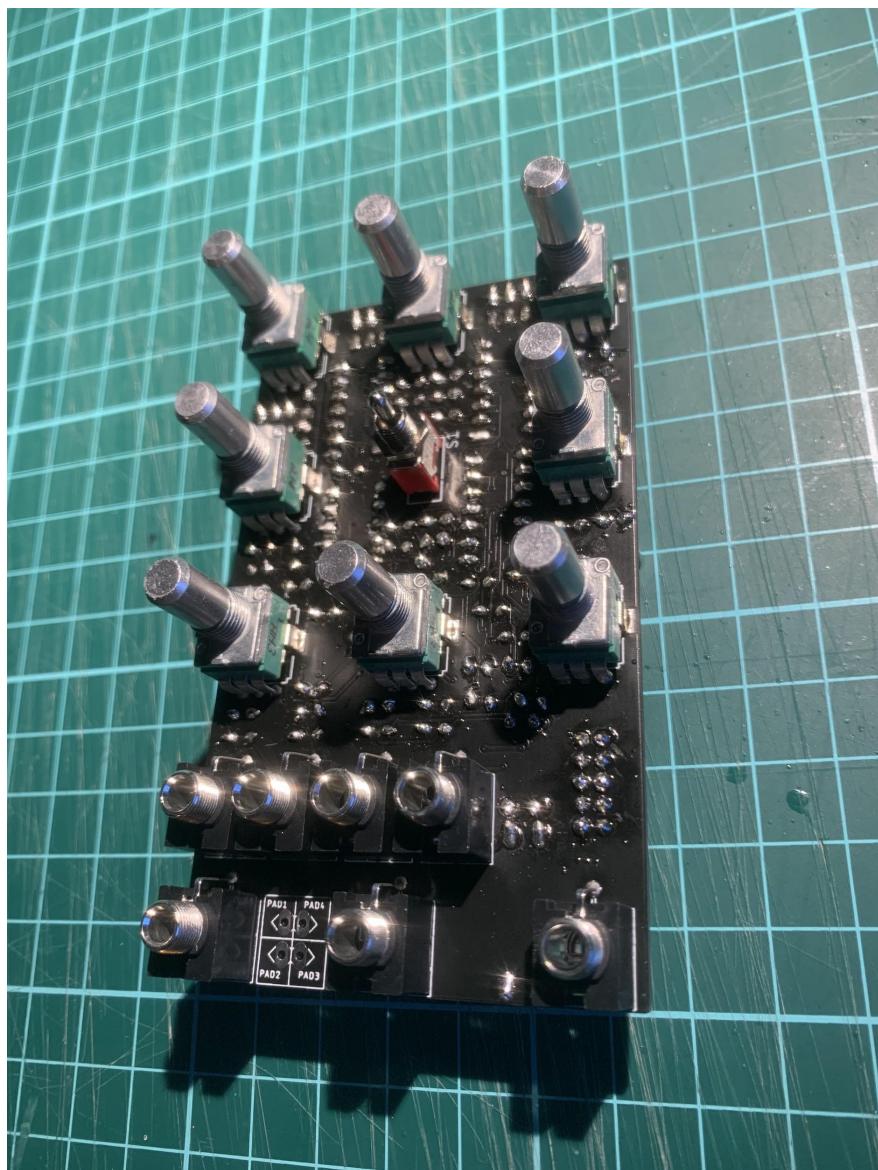
Potmeters, jacks and switch:

Flip the pcb and insert the potmeters into the pcb. Also insert the jack sockets and the switch. DON'T solder yet! Now place the panel. Use one or two nuts to hold the panel in place.

Now solder one pin of each pot and jack. Remove the panel and check if the pots are aligned. The jacks will be slightly raised from the pcb. Don't worry about them, as long as the pins are sticking through, they are fine. If you are unsure, slightly press down on the jack and reheat. It should click flat to the pcb. If all is correct, continue to solder all pins.

Also put a nut on the switch like shown below, this is because the switch is a bit lower than the potentiometers. And so the nut prevents the panel from bending.

After soldering the pots, jacks and switch it should look this: (front panel taken off)



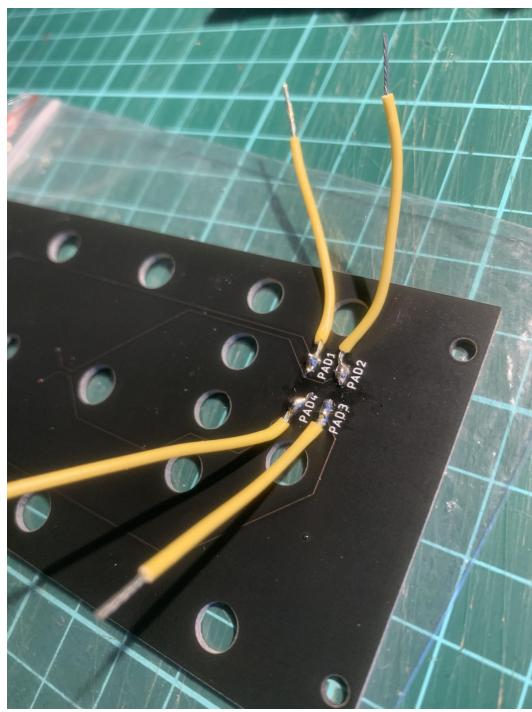
Panel & Touchpads:

Now we have to make a connection from the front panel to the pcb for the touchpads.

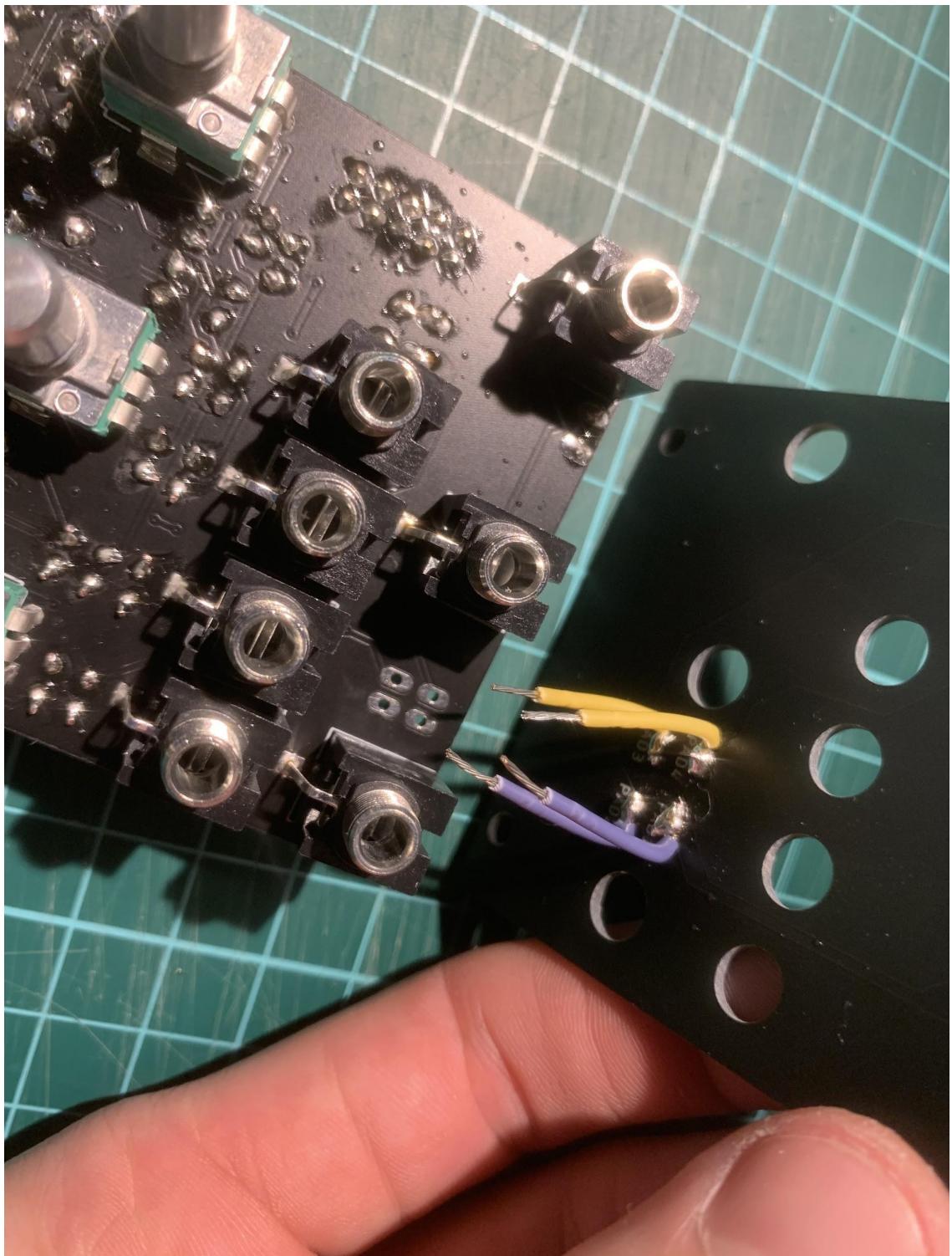
First put some solder on the pads of the back side of the front panel:



Take 4 short pieces of wire with the ends stripped and solder them on the pads like so:



Now bend the wires like shown below and push the ends through the holes and solder the wires to the pcb. Make sure you match the correct wires/pads and the correct holes. Pad 1 on the front panel connects to pad 1 on the pcb and so on.



Now you can push the panel on the pcb and tighten the potentiometer and jack nuts, be careful not to scratch the front panel as you tighten the nuts.



IC's:

Take the IC's out of the foam. Bend the legs to 90 degrees using a flat surface.

Then insert it while taking care that the notch matches the IC socket (and silkscreen on the pcb) Press both the IC's firmly into the sockets. Look at the BOM for the correct placements.

Knobs:

Turn all pots fully CCW and place the knobs. Take care that all lines point to the same angle. Then push them firmly to the pcb, while holding the back of the pcb. Depending on the type knobs you might need to tighten them with a screwdriver, the type knobs that need to be tightened have a small screw on the side.

Testing:

As a last check, look over the pcb and check the soldering, check for shorts and polarity.

Insert the power cable and connect it to your modular system.

Turn on the power. Check if nothing blows! If all is well, proceed: Patch the unfiltered output to a mixer. (the right jack on the bottom).

You should be able to hear sound now, now test the top row knobs, bottom row knobs and touchpads and listen if they affect the sound (they should).

Then patch the filtered output to a mixer (bottom left jack). also patch the LFO output into the cutoff cv in. Now test the filter knob and LFO speed knob.

You also want to check the cv input of each voice. Patch the LFO or another cv signal in one the 3 cv inputs and check if it affects the pitch.

Adjusting LFO offset with the trimpot:

With the trimpot you can adjust the LFO offset. You can use an oscilloscope for this, if you don't have an oscilloscope you can measure the resistance of the trimpot on the "test" pads on the pcb next to the trimpot.

- Around 50k for a bipolar LFO

You can also adjust the LFO by ear, when powered simply patch the output of the LFO into the cv input of the first voice. Now you can hear the LFO adjusting the pitch.

Troubleshooting:

If the module does not work, check if the orientation of the IC's, electrolytic capacitors, power header, transistors and diodes. Check your soldering. It should be perfect, like in this picture:



Design by M.Verhallen & F.Heldens 2021

www.reverselandfill.org