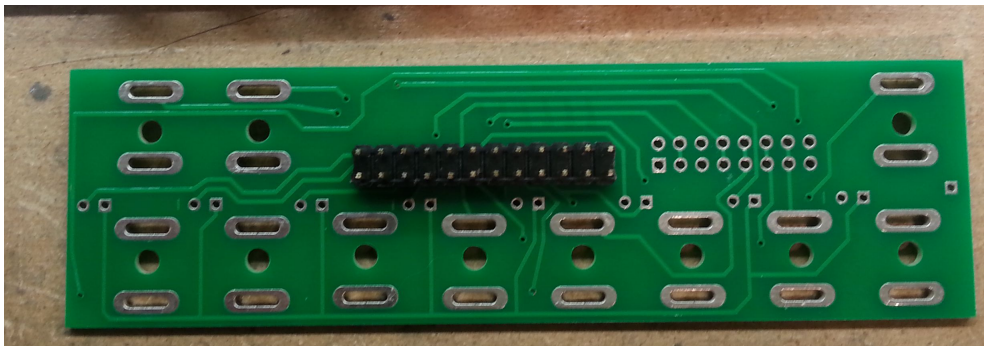
SMRL Vactrol Sequencer Assembly Guide

Jack Board and Faceplate



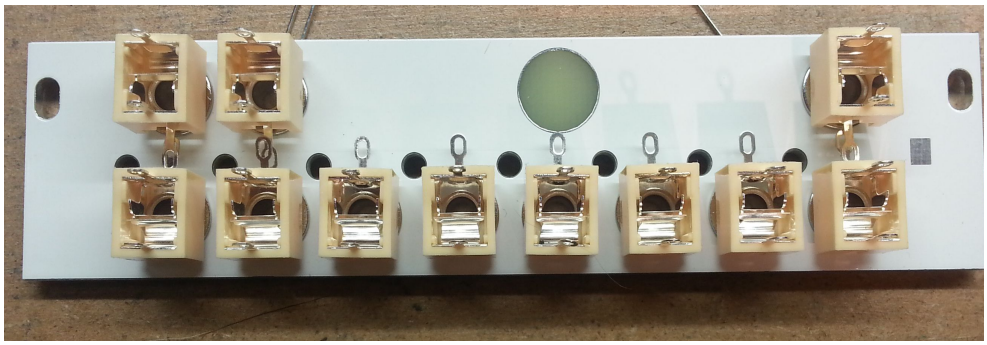
Step 1 – Install the male header on the jack PCB.

The male header should be inserted on the side of the board with no silkscreen, and soldered on the side with the silkscreen. Solder pins on either end of the header first, to ensure that it's flush with the board, before soldering the rest of the pins. Be very careful with this, and double-check the solder joints on this header. **Once this is soldered, it's going to be sandwiched between the jack PCB and faceplate.** This is your opportunity to remove the flux, particularly important if you used organic core solder.

Note the extra 8-pin header connector – that connects to the normals of the jacks. If that means anything to you and you want to use it, go ahead. I've used it to connect multiple sequencers together so the inputs of one sequencer are normalled to several sequencers. These are unbuffered, however, so you're really just connecting the same signal directly to multiple vactrols. The input impedance will vary with the state of the vactrols it's connected to, and when a vactrol is 'on' it's about 5k. If you wanted to use these inputs and avoid interaction between sequencers sharing the same signals, it would be best to buffer the signals going into this connector (e.g. with an op-amp follower. It needs to be stiff.) Keep in mind, however, that you will briefly contact this signal with the tip of your cable before the normal is broken!

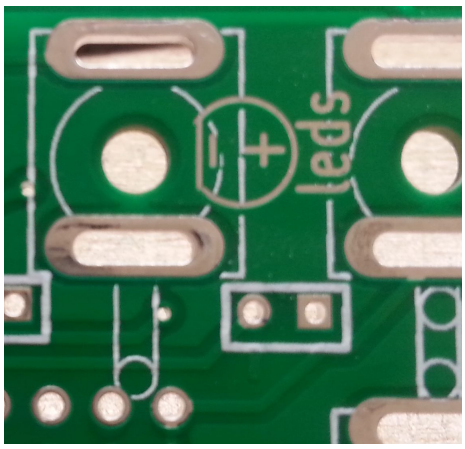
This points to a general shortcoming of this design. **If you passively mult the signals going into the sequencer, be forewarned that the sequencer can attenuate the signal when the vactrol is on.** With LZX's 500 ohm output impedance, this 'signal droop' could be as much as 10%. With other modular manufacturers, it may be more or less. While I went to the effort to match these characteristics for a 'matched set', they will also vary somewhat from unit to unit. Unfortunately there is no clear standard for output impedances in Eurorack. The 4.7k summing resistor network limits the maximum voltage drop, but also establishes our attenuation factor – it's a tradeoff. The output impedance of this module is 470 ohms, and is given by R5.

Step 2. Secure the jacks to the faceplate.



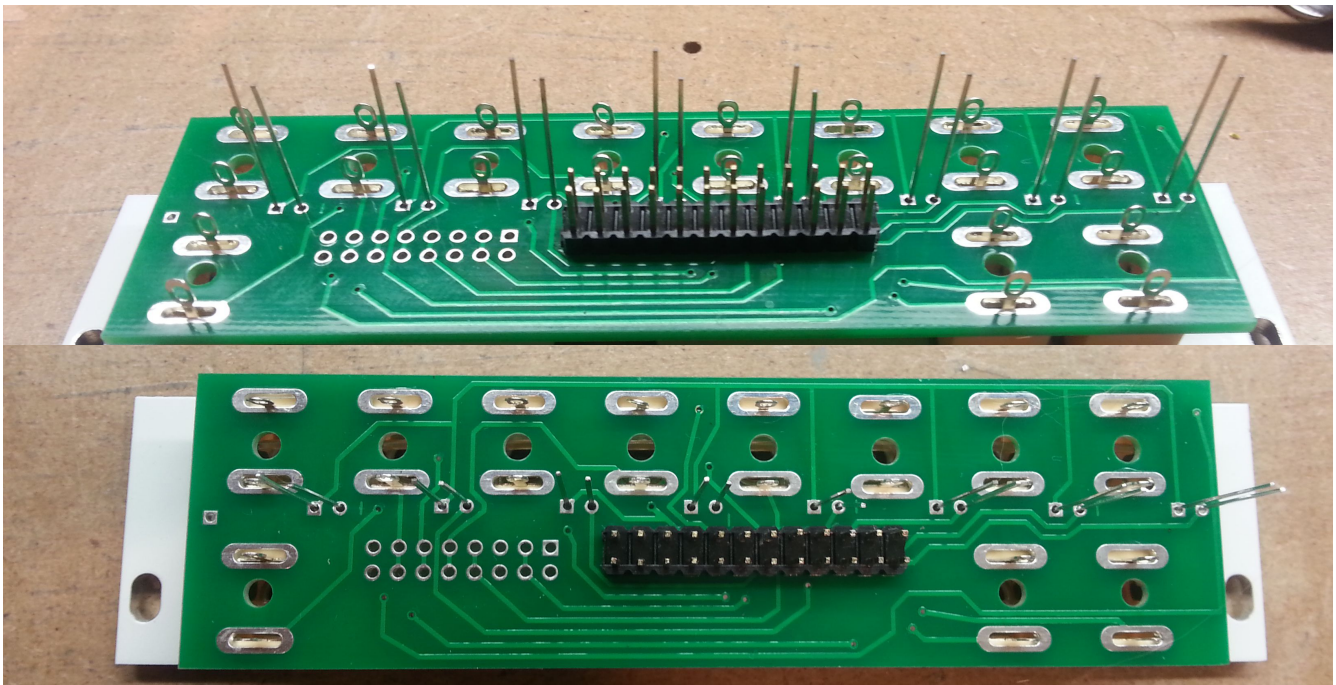
Using a washer between the jack and the faceplate, push each jack through the front faceplate and secure with a nut. Align them all so that they are straight, with the ground tabs facing the center of the faceplate. You can remove them if you desire, the jacks are grounded through the front faceplate. I didn't bother.

Step 3. Insert the LEDs into the jack PCB.



There are several indications on the PCB of proper LED orientation. There is a copper legend that shows the correct orientation. Look at the flatted side of the LED as a reference. Also, the longer lead should go through the square hole, and the shorter lead should go through the round hole. There are also 'minus' signs corresponding to the negative lead. They are all oriented the same way. I tried my best to make it obvious.

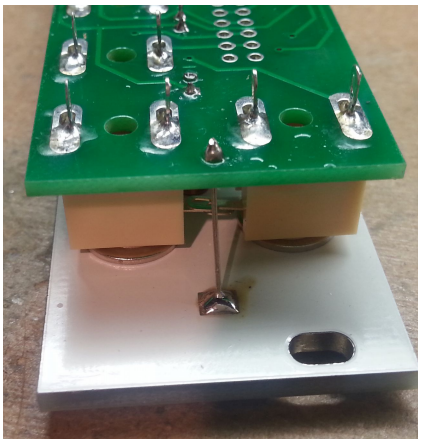
Step 4. Mate the jack PCB and faceplate.



With the jack PCB LED-side up, insert the faceplate. If the jack leads don't fit, check the alignment. Flip the two over and push the LEDs through the faceplate so that they all extend fully through.

Step 5. Solder the jack PCB connections.

Make certain that the two PCBs are as aligned as possible. I had to make the jack holes large enough to get the leads through and perhaps made them a bit too large. Solder one leg on two jacks on either end of the board. Make sure it's flush to the jacks. Now would be a good time to place it in your rack to make sure it is lined up properly. **The tolerance is fairly small on this.** I haven't had any problems, but you may want to check.

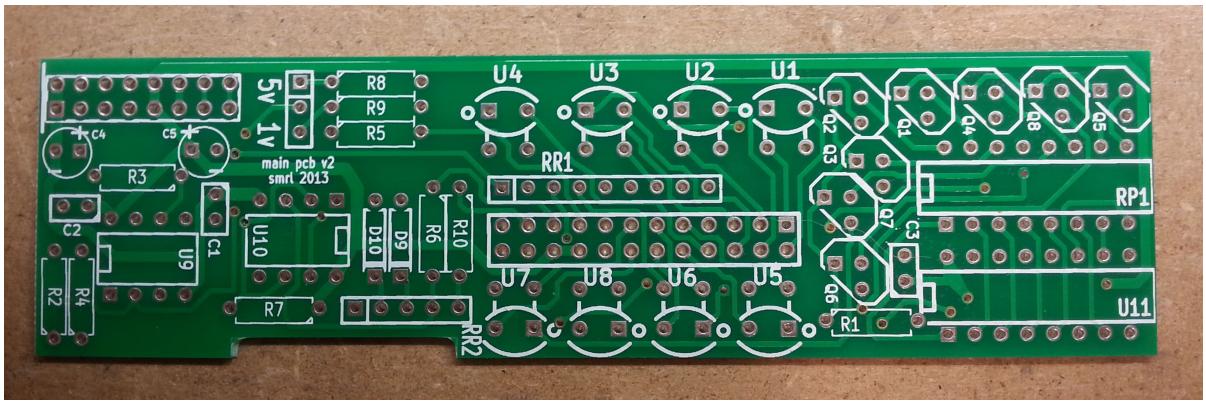


If you're happy with it, solder the rest of the jacks in place. Then you can again make sure that the LEDs are all sticking through and solder those. Cut the LED legs with a flush cutter.

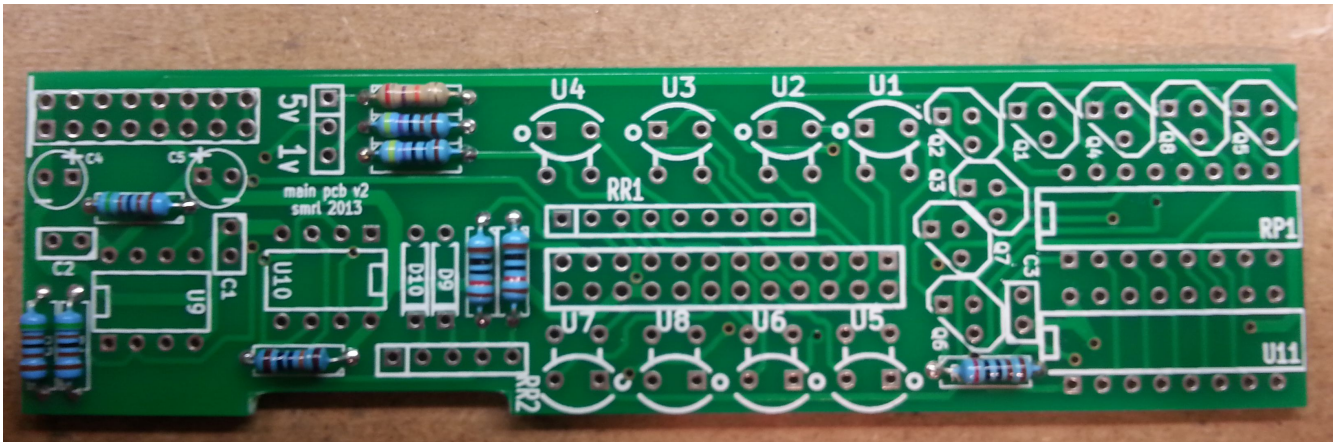
Step 6 – Solder a cut leg of one of the LEDs between the pad on the faceplate and the jack PCB. This is the ground connection for the jacks. Remove the flux from the boards if needed.

Tighten the jack nuts.

Main PCB



Step 1 - Install the resistors.



!! R4 needs special attention !! *You will need to bend one of its leads across to pin 1 of op-amp U9 or solder a short wire there. I've just been leaving this one bent over until I install the op-amp package.*

Component values and function

R1 – 2k (This resistor sets the current through the **vactrol AND front-panel** LED. 2K across 12v = ~6mA, but consider diode drops) (Red, black, black, brown, brown)

R2, R3, R4 – 5.1k (Green, brown, black, brown, brown) video summing/buffering op-amp resistors – Leave R4 lead closest to the board edge untrimmed.

R5 – 470R (Yellow, purple, black, black, brown) Don't confuse with R9! Output protection resistor.

R6, R10 – 10k (Brown, black, black, red, brown)

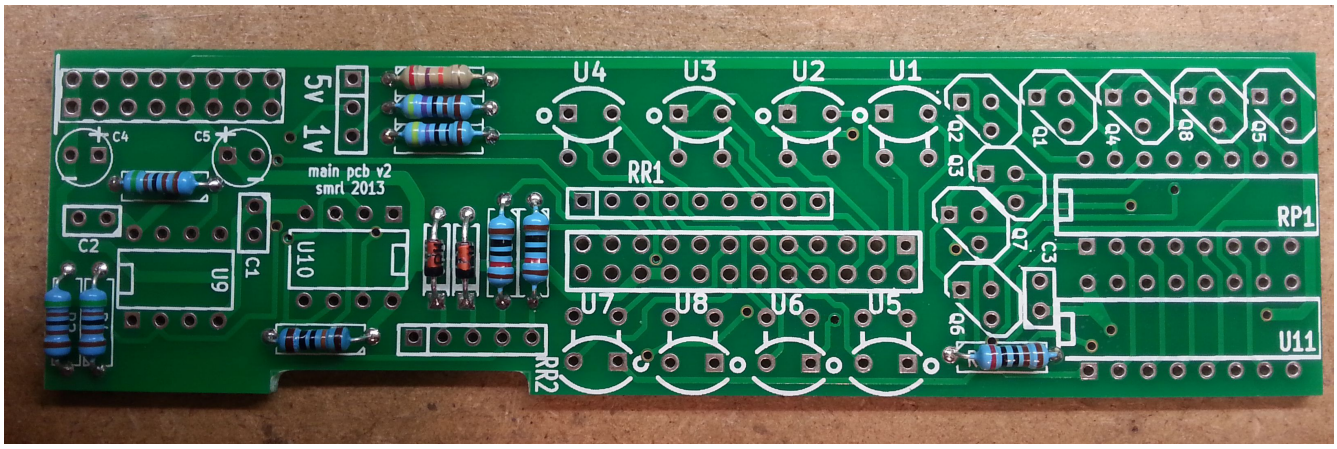
R7 – 100k (Brown, black, black, orange, brown) Comparator threshold resistor

R8 – 27k (Red, Purple, Orange OR Red, Purple, Black, Red) (5v mode threshold-setting resistor) – In conjunction with R7 sets comparator threshold at ~2.5v

R9 – 4.7K (Yellow, purple, black, brown, brown) (1v mode threshold-setting resistor) – In conjunction with R7 sets comparator threshold at ~.5v

Clip the leads with a flush cutter.

Step 2 – Install the diodes.



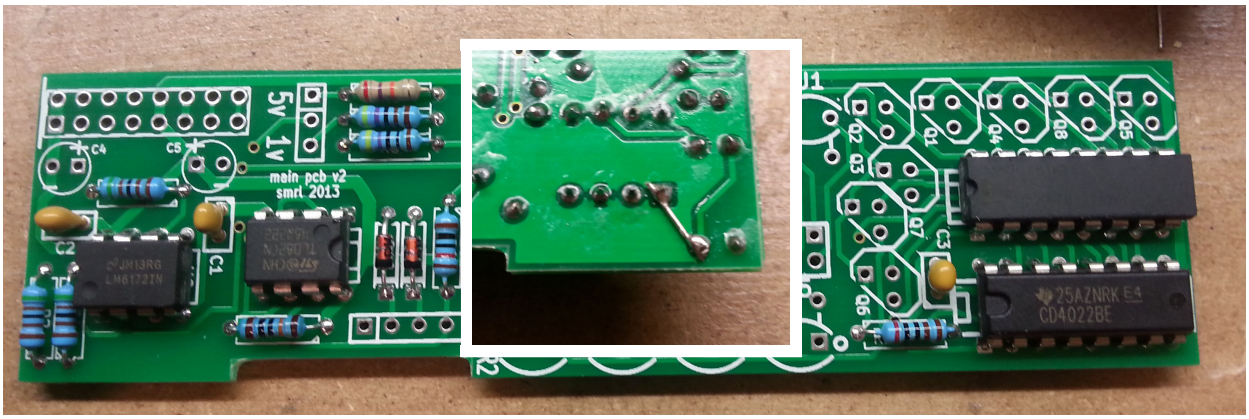
Note the orientation. Orient the black band on the side with the thick white line. The 2 diodes are marked D9 and D10. Their purpose is to keep the comparators from pulling the reset and clock pins on the 4022 below 0v. I may have included an additional diode. Disregard it, this was needed for a different counter in a previous revision.

Clip the leads with a flush cutter.

Step 3 – install the .1uF (marked 104) bypass capacitors C1, C2, C3.

Again, there are 3 caps on the board and probably 5 included in the kit. I changed the number of caps in the later revision of the board -- there simply wasn't space for the two additional caps on the supply pins of the TL082. You could kludge them in, if you really wanted to, on the underside of the board. I haven't noticed a problem with it. These just supply a little 'backup charge' as the power supply varies under load changes.

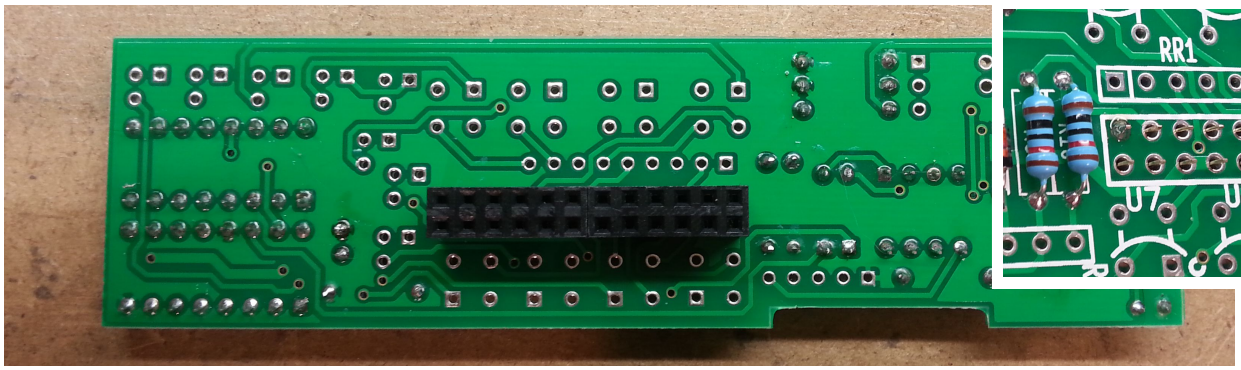
Step 4 – install the ICs and DIP resistor pack.



U9 – LM6172 (video summing/buffering op-amp)
U10 – TL082 (2 op-amps used as comparators for clock and reset)
U11 – CD4022 (CMOS Divide by 8 Counter)
RP1 – marked '102' (base resistors between CD4022 and transistor current sources)

Align the orientation of the IC's with the marking on the PCB. The notch corresponds to the silkscreen rectangle. **U9 and U10 are not pointing in the same direction**. You can use the included sockets for U11 and U10 if you'd like. I haven't been. Don't install a socket at U9. **Remember to solder the remaining resistor lead of R4 to pin 1 of U9.** If you trimmed it, just use a cut lead from a resistor or diode to make the connection.

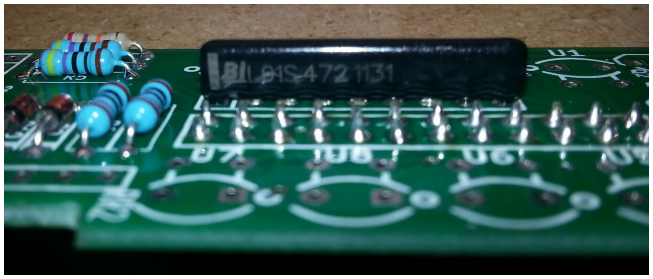
Step 5 – Install the female main PCB – jackboard header.



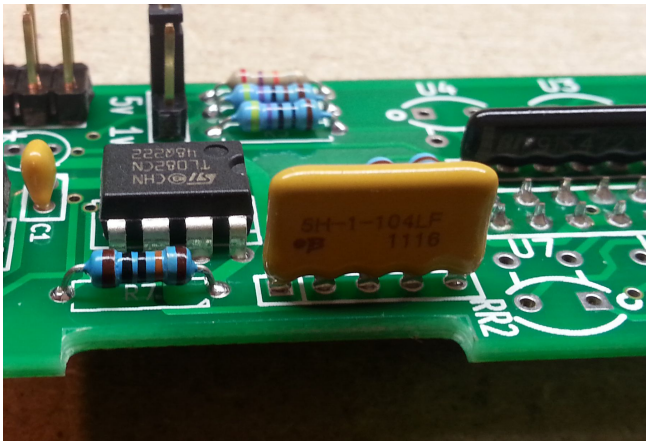
This is the only component which is mounted 'upside down' with respect to the rest of the components. Flip the board over and insert the two 12 pin female headers. Plug it into the jackboard's male header. This ensures alignment of the headers. Carefully solder pins on either end. Ensure that the main PCB is flat and flush with the headers. Continue soldering the rest of the pins. Separate the two boards.

Step 6 – Install the resistor packs.

Install the 9-pin resistor pack marked '472'. **Insert the side with the stripe into the silkscreen square.** (The stripe should be at the extreme end of the header you just soldered.) Bend the outer pins a bit so that it stays in place. Solder it.

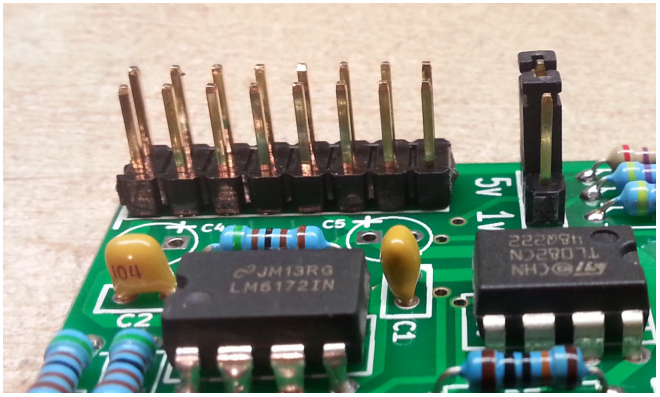


Next, install the 5-pin resistor pack marked '104'. Do the same thing, but this time, **there is a dot marking pin 1** which should be aligned with the silkscreen square. Solder it.

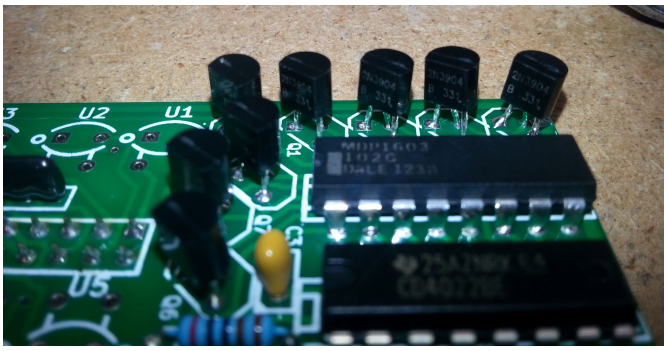


Step 7 – Install the remaining headers.

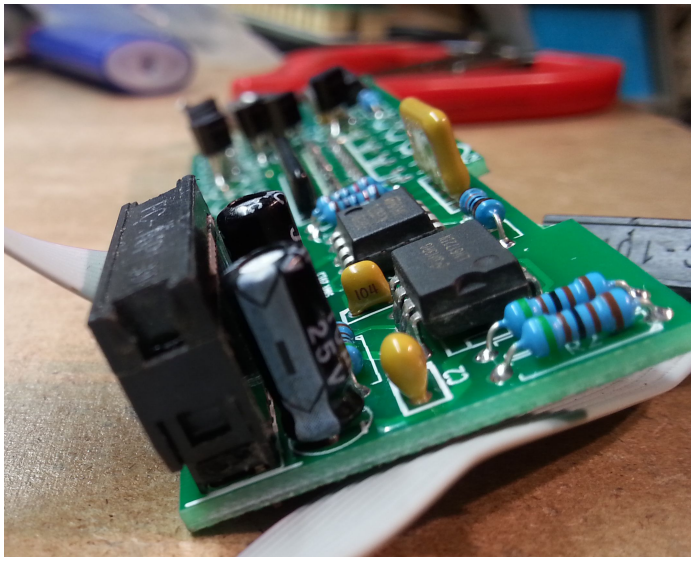
Insert the 3-pin voltage threshold header and solder it. Insert the 16-pin power header and solder it. As before, It's good practice to tack down either end first, making sure it's flush to the PCB, and then continuing to solder. Put the voltage selection jumper in place. 1v mode should work for most situations.



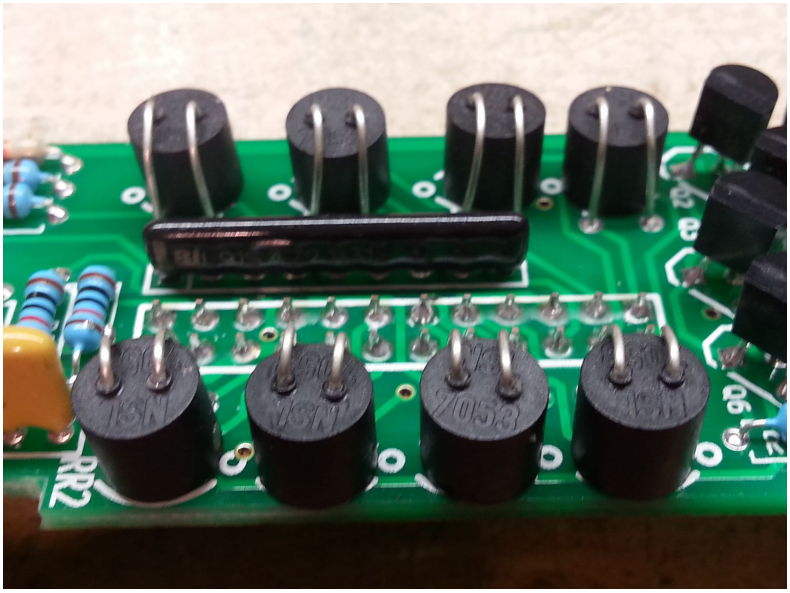
Step 8 – Insert and solder the 8 2N3904 transistors. The orientation is suggested by the PCB footprint – align the long flat side with the flat side of the transistor. Clip the leads with a flush cutter.

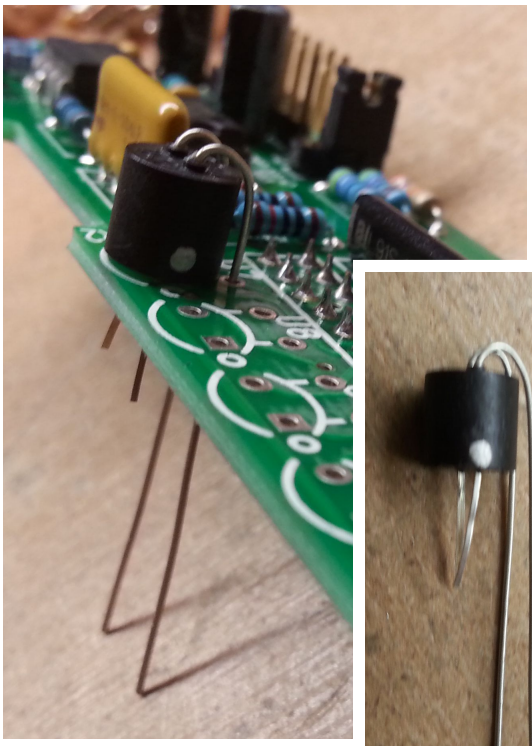


Step 9 – Install the two electrolytic capacitors. These are polarized. The negative side is marked with a stripe containing a "-". Insert the longer lead of each into the square pad with the "+", and the shorter lead (which corresponds to the striped side), into the circular pad with the "-". It's a good idea to push a power cable onto the header before you solder the capacitors down to ensure that you can get the power connector on & off.



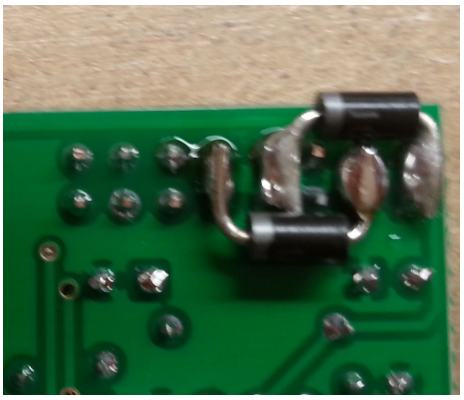
Step 10 – Install the vactrols.





These vactrols have 4 leads. The LEDs are the shorter ones, and have a gray dot on one side. **LEDs are also polarized,** so it's important that you install the vactrols the right way around. Bend the resistor leads over one side of the vactrol as shown. Note the location of the dot. There is a circular area on the PCB footprint that corresponds to the LED side, and a small circle denoting the proper LED orientation. **Make certain that the dot on the vactrol is directly adjacent to the dot on the PCB footprint.** Solder them in. Be cautious for the few close vias – you don't want any solder bridges to them. Now would be a good time to clean your PCB, especially if you've used organic core solder. That's it for the main PCB!

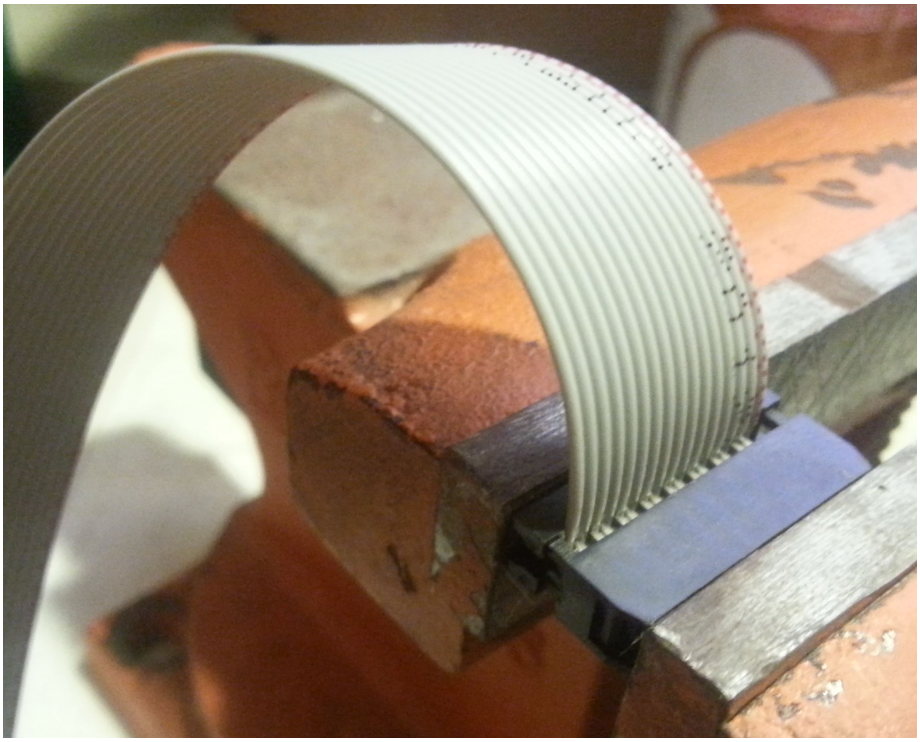
NOTE: This board has no protection on the power header. The power connector orientation is marked on the board – the white stripe at the bottom should correspond to the red stripe on your power cable. MAKE SURE that the power connection is the proper way around before powering up the PCB. If you insist on adding protection, you can do so by soldering two diodes (1N4001 or 1N4004) to the underside of the PCB like this:

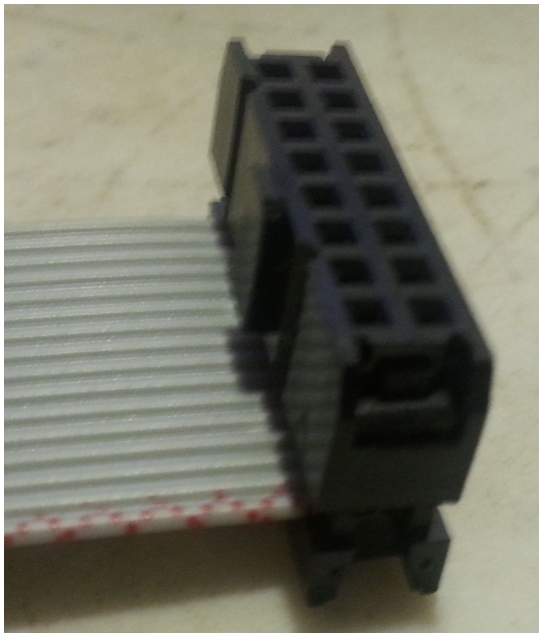


Make sure not to short any of the rows of the header together while doing this!

Power Cable

Make certain that the ends of the ribbon cable are cut straight across. Use a pair of scissors if it needs trimming. Remove the strain relief clip, insert the ribbon cable between the top and bottom half of the IDC connector. The arrow denoting pin 1 on the connector should be aligned with the red stripe on the cable. Put it in a vise if you don't have a crimp tool, and tighten it. You should hear a slight click. Bend the cable over the top of the connector and insert the strain relief clip. Repeat for the other side.





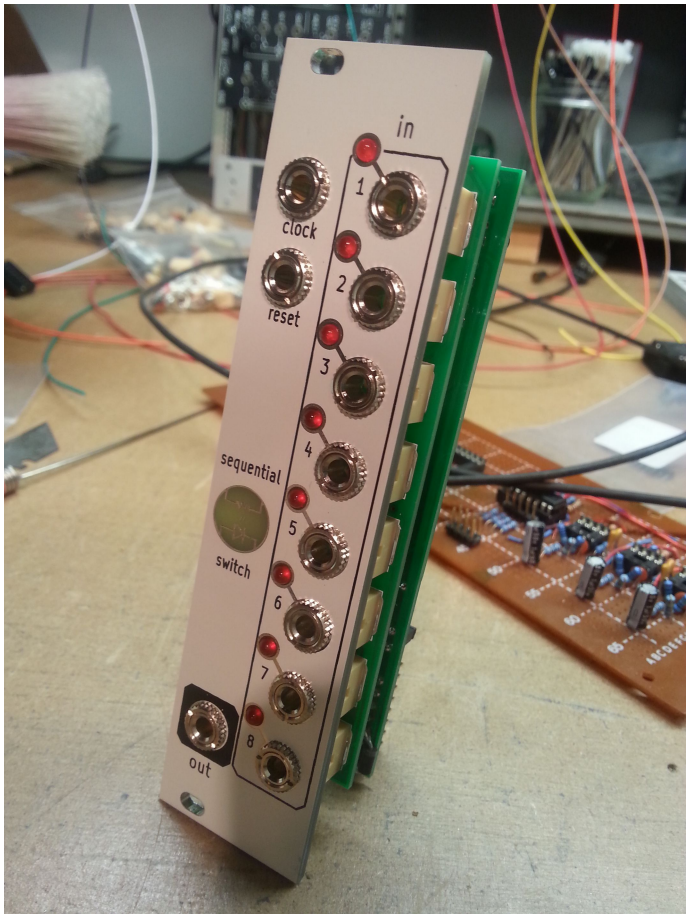
Other design notes

The transistors aren't really needed, and were left in because of the prior design where the logic was running off of 5v. If I were to redo this circuit, I would omit them and simply use 8 resistors hanging off of the 4022. The one benefit is that you can change one resistor (R1) to change the brightness of both the front panel LEDs and the vactrol LEDs.

Op-amps are used as comparators with no hysteresis. Not a great idea but it works. At ordinary speeds it works fine. When you go faster than the vactrols' response time, the comparators get erratic. Since it works through the frequencies of interest, I think it's okay.

Bill Of Materials

8 vactrols, electronic goldmine surplus



Semiconductors
1 LM6172 high-speed dual op amp
1 TL082 dual op-amp
1 CD4022 8-stage divider/counter
8 2n3904 transistors
2 1n4148 diodes
8 3mm LED

Resistors
2 - 10k
1 - 470R
1 - 4.7k
1 - 100k
3 – 5.1k

1 8-resistor parallel network, DIP, 10k
1 8-resistor ganged network, 9 pin SIP, 4.7k
1 5-resistor ganged network, 5 pin SIP, 100k

Capacitors
2 10uF electrolytic capacitors, 25v
3 .1uF (104) ceramic capacitors

11 Kobiconn 16PJ138
1 3-pin SIL header
jumper
1 16-pin DIL male header, 1 24-pin DIL male header (from larger breakaway)
2 12-pin DIL female header (mouser)