nonlinearcircuits

RESONATE build & BOM

This is based on the Korg PS3100 resonator circuit, except there are 4 BP stages instead of 3, it has a feedback path, the CV control section is totally different, the filter core components are based on mods suggested by RJB and it is all smd ☺

Note - Some builds need a fix to prevent the output drifting to a rail, see pg3 (this has been added to vers.3 TOP PCB).

Also vers.2 BTM PCB has an error (fixed on Vers.3) - see Fix #2

BOM

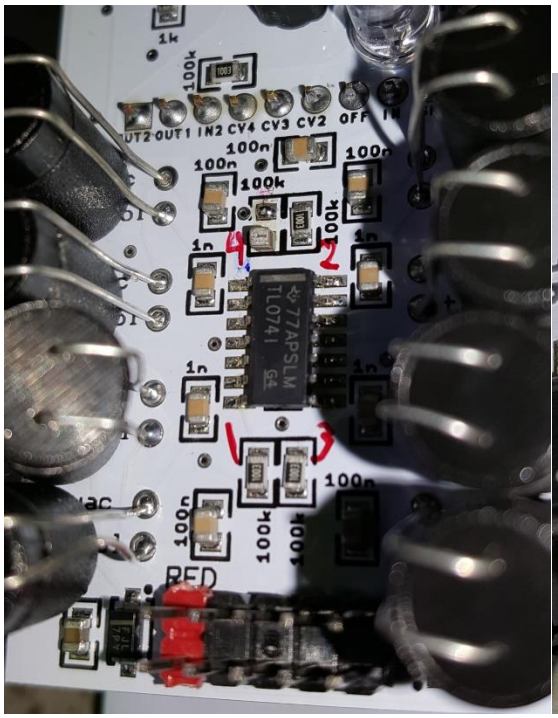
VALUE	QUANTITY	DETAILS
10pF	1	0805 25V rating or higher
1nF	4	0805 25V rating or higher
33nF	2	0805 25V rating or higher
100nF	8	0805 25V rating or higher
1uF	4	0805see notes
10µF	2	0805 25V rating or higher Mouser No: 81-GRM21BR61E106KA3L (or similar)
RL	4	see notes #6
47R	1	0805
1k	2	0805
1k5	1	0805
33k	1	0805
68k	1	0805
100k	19	0805
150k	1	0805
220k	5	0805
TL072 or TL082	4	SOIC Tayda: A-1136
TL074 or TL084	1	SOIC Tayda: A-1137
vactrols	8	**see notes**
Blue LED	1	should be blue!
10 Pin 2.54mm Single Row Pin Header Strip	2	Tayda: A-197 (cut to size)
10 Pin 2.54mm Single Row Female Pin Header	2	Tayda: A-1306
Eurorack 10 pin power connector	1	Tayda: A-198
S1JL or similar, optional - for reverse voltage protection	2	SMD, standard power diode 200-600V 1A, dot on PCB indicates CATHODE (stripe on component) SEE NOTES #2
100k POT	6	Tayda: A-1848
3.5MM SOCKET Kobiconn style	8	Tayda: A-865 or preferably get Thonkiconn Jacks (PJ301M-12) from Thonk or Modular Addict
10-11mm stand-off and M3 or M2.5 screw to suit	1	11mm is best but 10mm is fine with a washer.

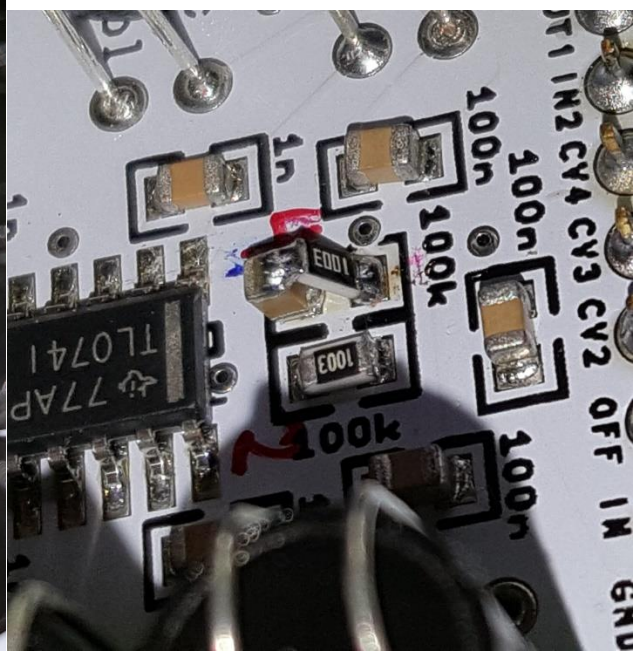
Additional notes:

1. **The prices for these 10uF 0805 capacitors drops to approx. 10c each when buying more than 10….and you should always get plenty of spares, it is easy to drop and lose smd parts.**
2. S1JL Power diode for Reverse voltage protection - Mouser Part No: 821-S1JL. Any similar rectifier with at least 1A rating should be okay.
3. **Vactrols – doesn't have to be actual Vactrols, many LDR based opto-couplers should work. Feel free to buy some GL5528 or GL5516 LDRs off ebay at 3c each. Get some red or orange or green LEDs, some heatshrink, black tape or black silastic/sealant and roll your own. If you have access to a 3D printer, there are files on muffwiggler you can download and print your own vactrol cases. Failing all that, there are plenty of Chinese made vactrol clones on Aliexpress these days for a $1 or so.**
4. Not a mod, just a tip - If you don't have any 47R but have 100R in stock, you can use 100R instead but replace the associated 1k5 resistor with 3k. Or you can solder a 2nd 100R on top of the 1st and still use 1k5. See schematic.
5. **The output can get pretty hot, especially if the feedback (Q) pot is cranked up. I like a bit of analogue clipping (digital not so much) but if you want to keep it out, reduce the gain controlling 220k resistor to something lower, say 180k or 150k. It is the one next to the 10pF cap.**
6. The resistors marked RL are for driving the LEDs in the 8 vactrols. You will need to select these to suit whatever ones you are using. Generally I make my own vactrols and use LEDs that work nicely on 4k7 resistors. Factory ones might prefer 1k……check the datasheet.
7. **The blue LED does not go to the panel and is only there to reduce the dead zone in the Freq pots. Blue LEDs have a high V-on (2.5V+) so are perfect for the job. You can use another colour but try to pick one with a high V-on.**
8. Don't forget to install the stand-off to the bottom PCB before attaching it to the panel otherwise you will have hard time trying to get the screw in place.
9. **The resistors and op amps are cheapest from Tayda. Maybe get the caps from Mouser and try to get decent ones, say 5% tolerance.**
10. Join the Nonlinearcircuits Builders Guild on FB (https://www.facebook.com/groups/174583056349286/) and ask questions there if you have any. If you prefer not to FB then email is fine.

FIX #1 – Only do this if you have Vers.1 TOP PCB. This mod has been added to Vers.3.

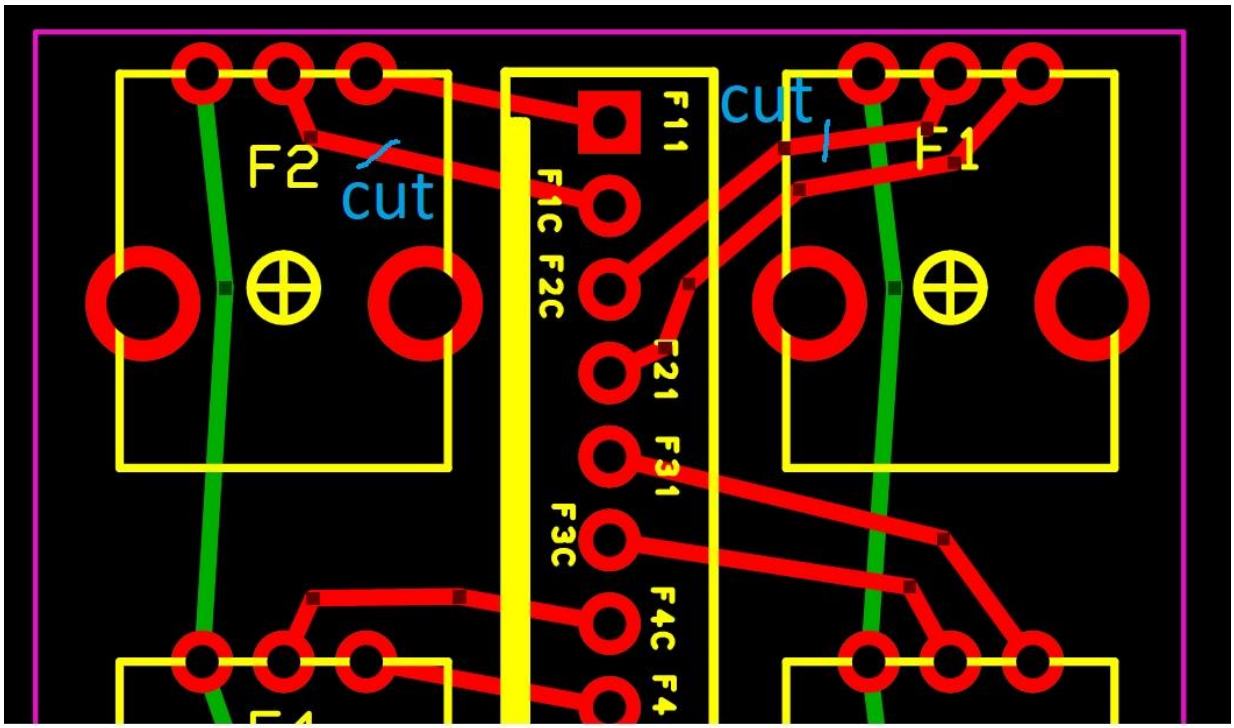
On some Resonates the output drifts to one or other of the power rails at certain settings. Usually this is caused by an imbalance in one of the Resonators, when you tweak the pot of the culprit the signal returns to normal.
If you have an oscilloscope, monitor the output, nothing on the inputs, pots turned down to 0. If the output slowly drifts to +V or –V over a few minutes, you need to do this fix.
Tweak the pots, one of them will cause the output to return to 0. If you are unlucky it may be more than one, so you will have to repeat this test.
The fix is to simply place a capacitor on the output of the offending resonator. I used 1uF on the one I tested/modded, anything from 100nF to 10uF should be fine.
See the pictures to see which resonator corresponds to the pot on the panel. The 100k resistor next to the red numbers needs to be removed and then you must build a little tent with the capacitor and the 100k resistor. **The capacitor must be closest to the op amp.**



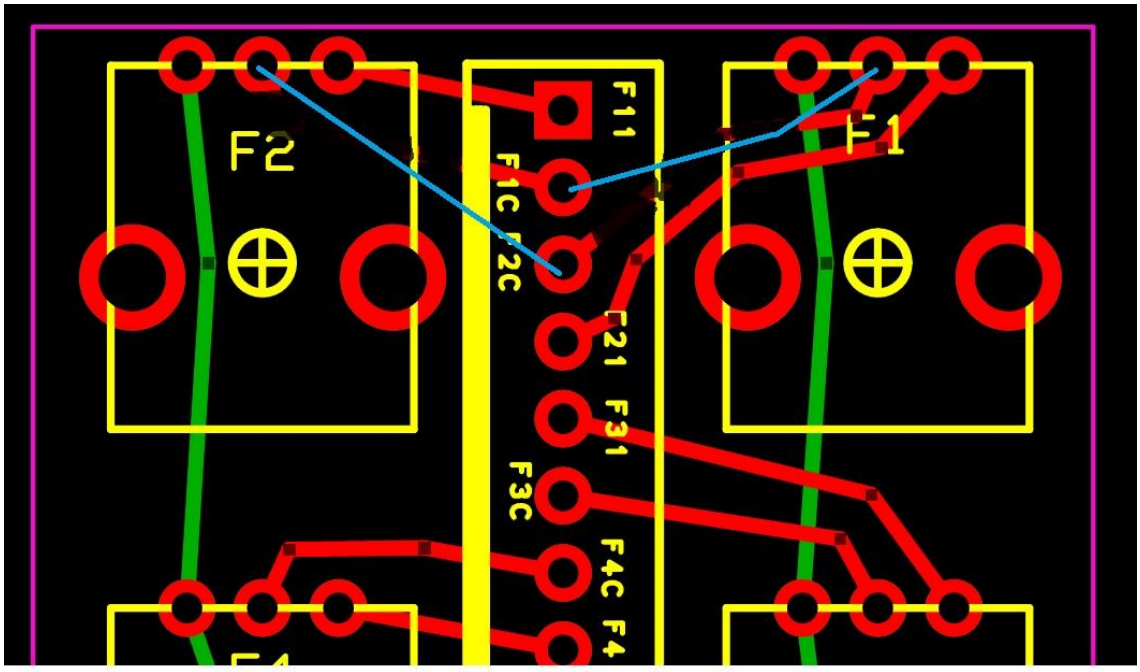


If the problem still occurs after adding 1 cap, consider yourself somewhat unlucky ….or special…….and repeat the test/fix.

FIX #2 - VERSION 2 BTM PCB FIX (THIS HAS BEEN FIXED ON VERS.3 SO IGNORE)
2 traces on the bottom PCB need to be cut and wires added to re-route.
Synth DIY Guy gets credit for spotting this; many Thanks!

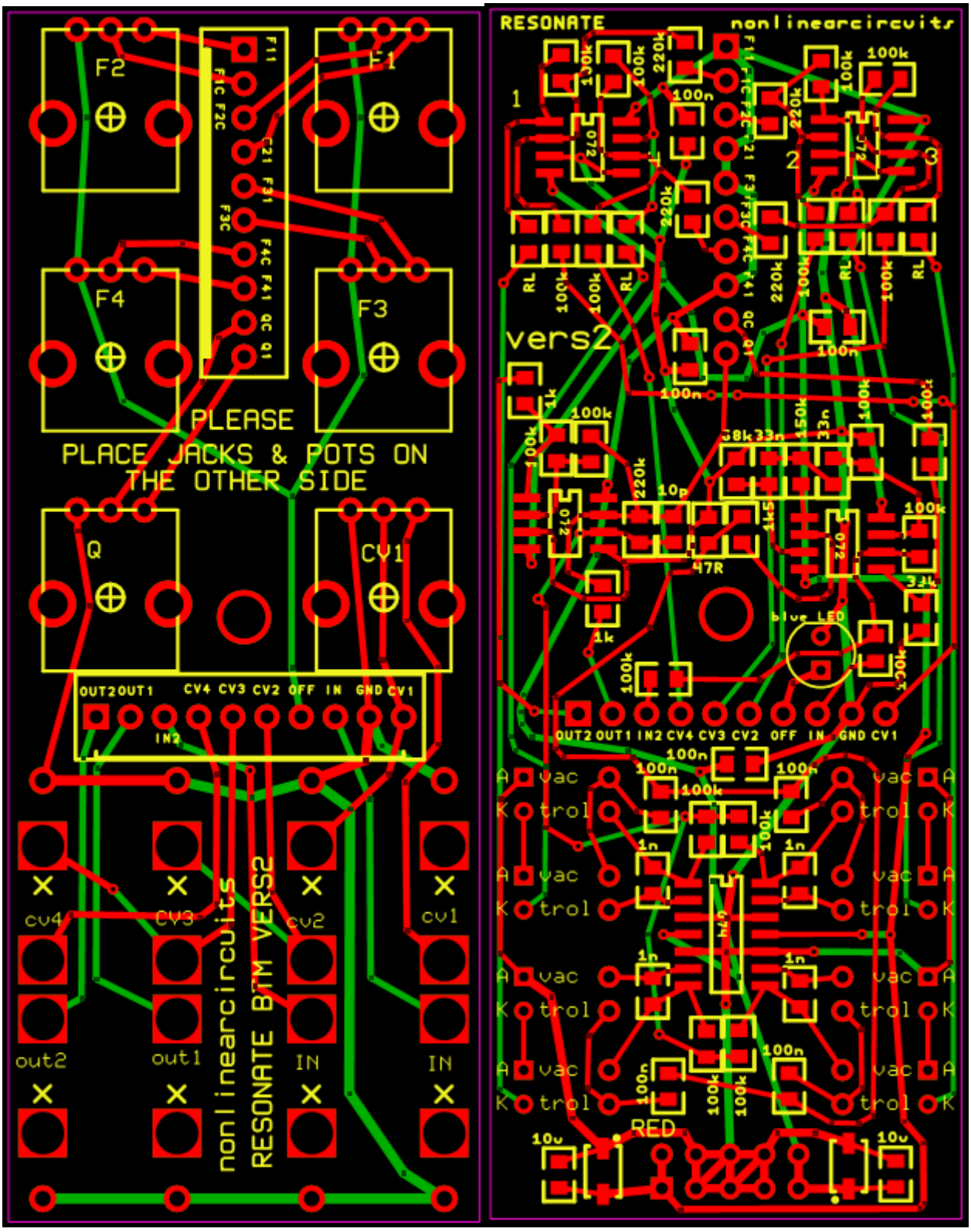


#1 - cut the 2 traces shown

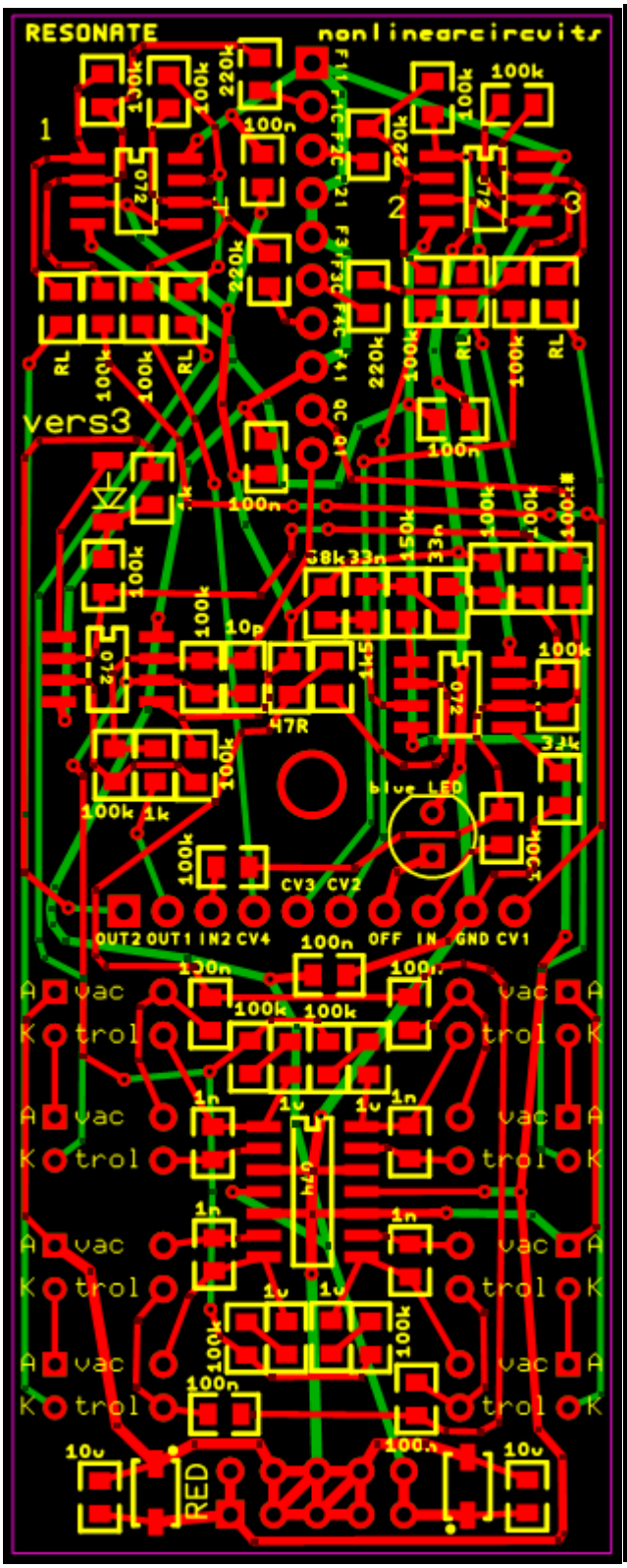


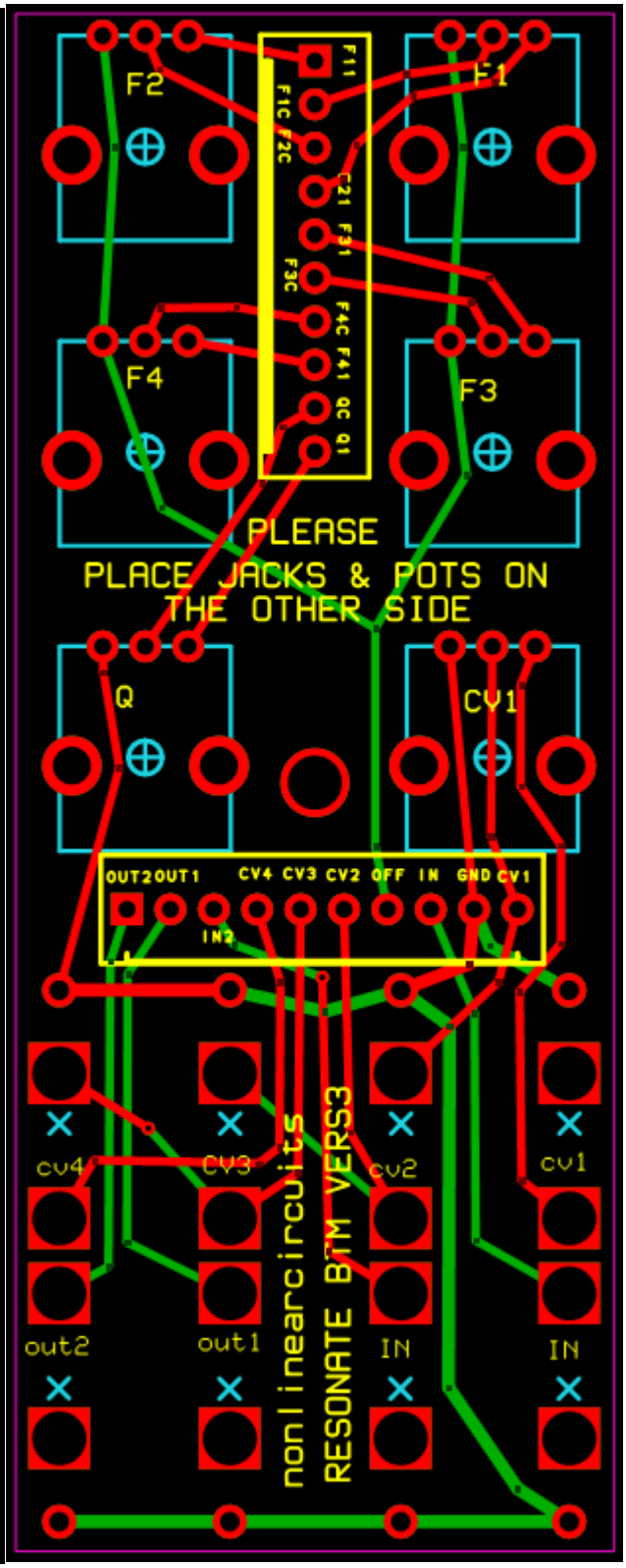
add jump wires as shown in blue

done!

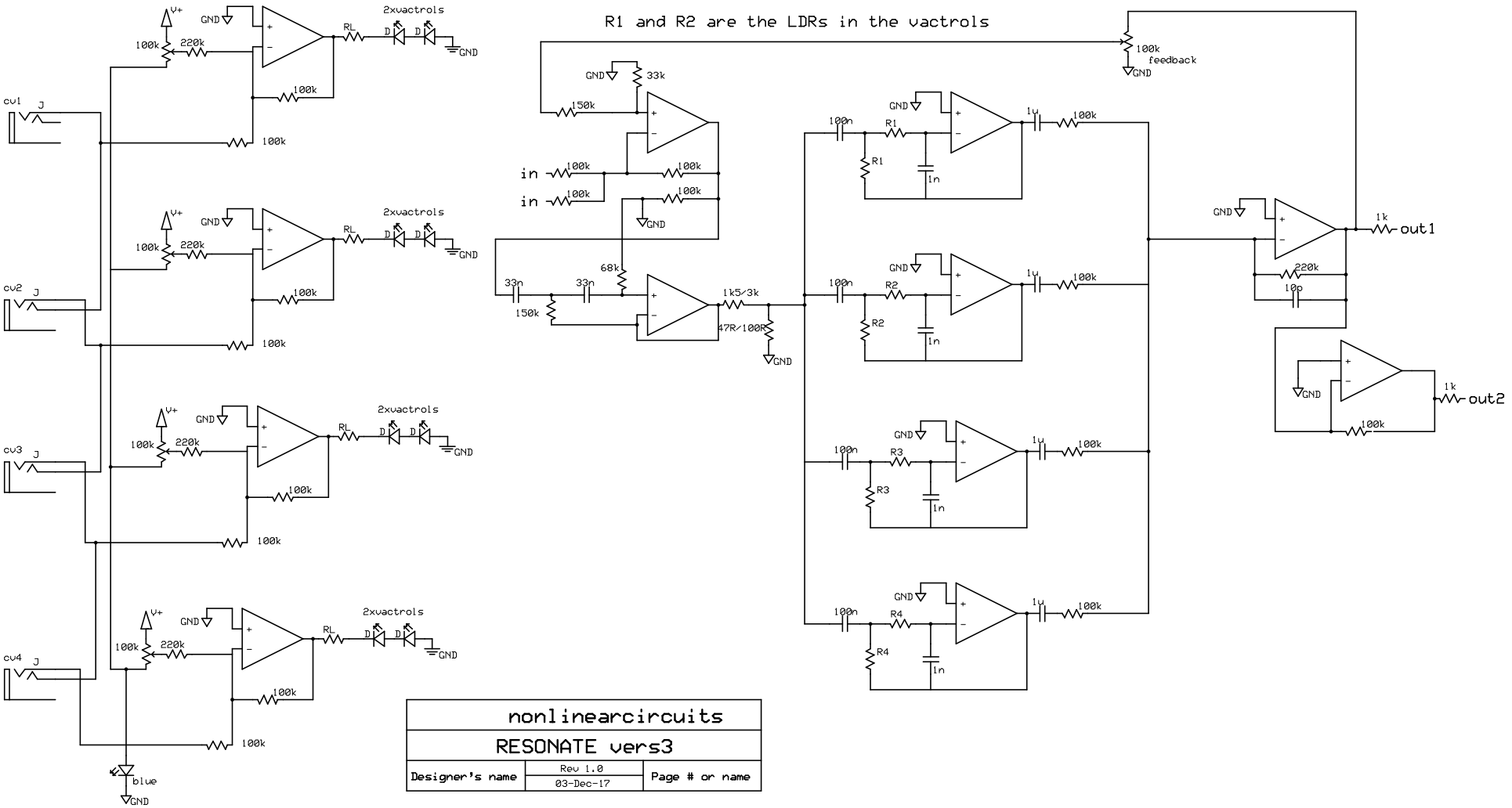


Version 2 PCBs – these need the capacitor mod and F1C/F2C traces swapped

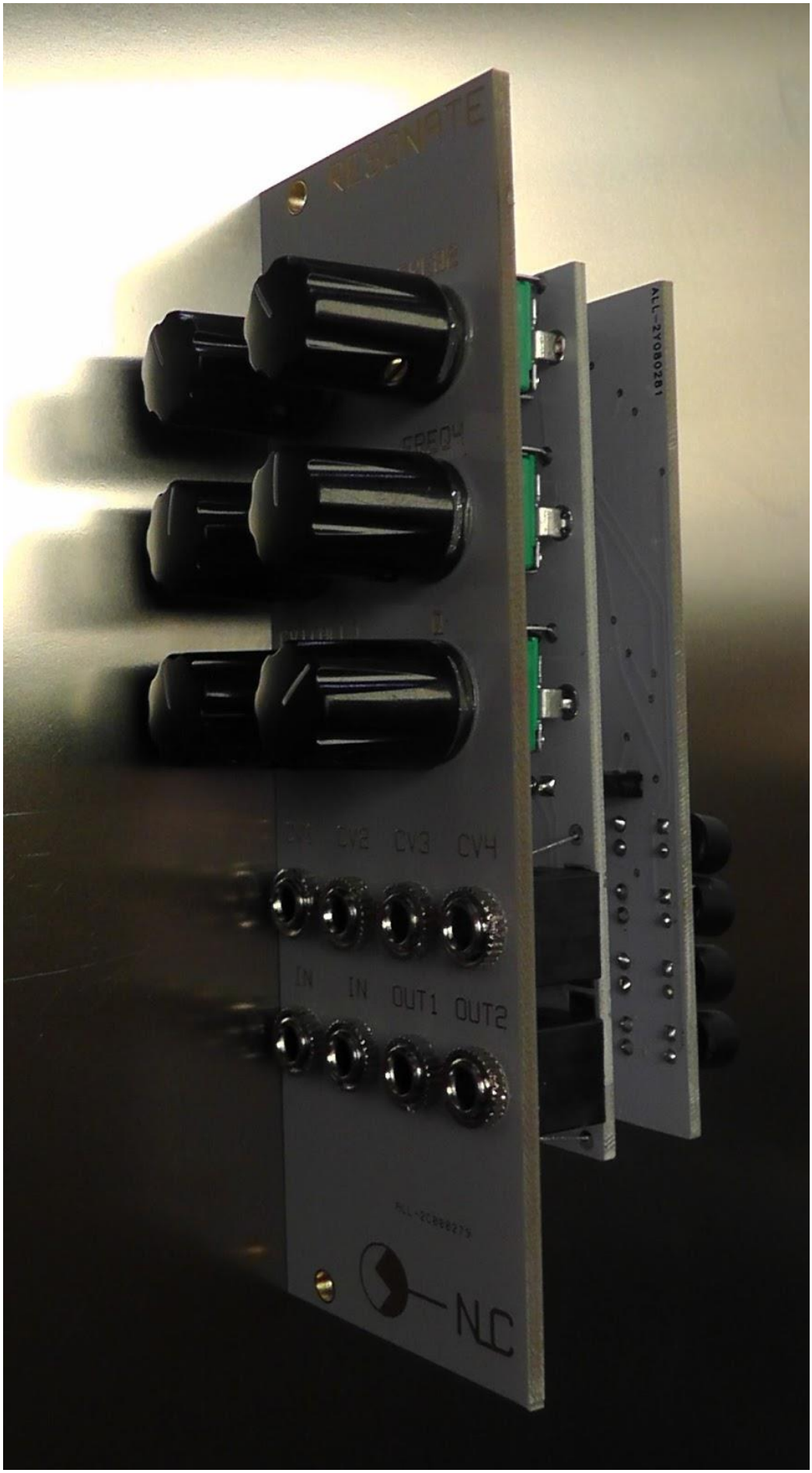


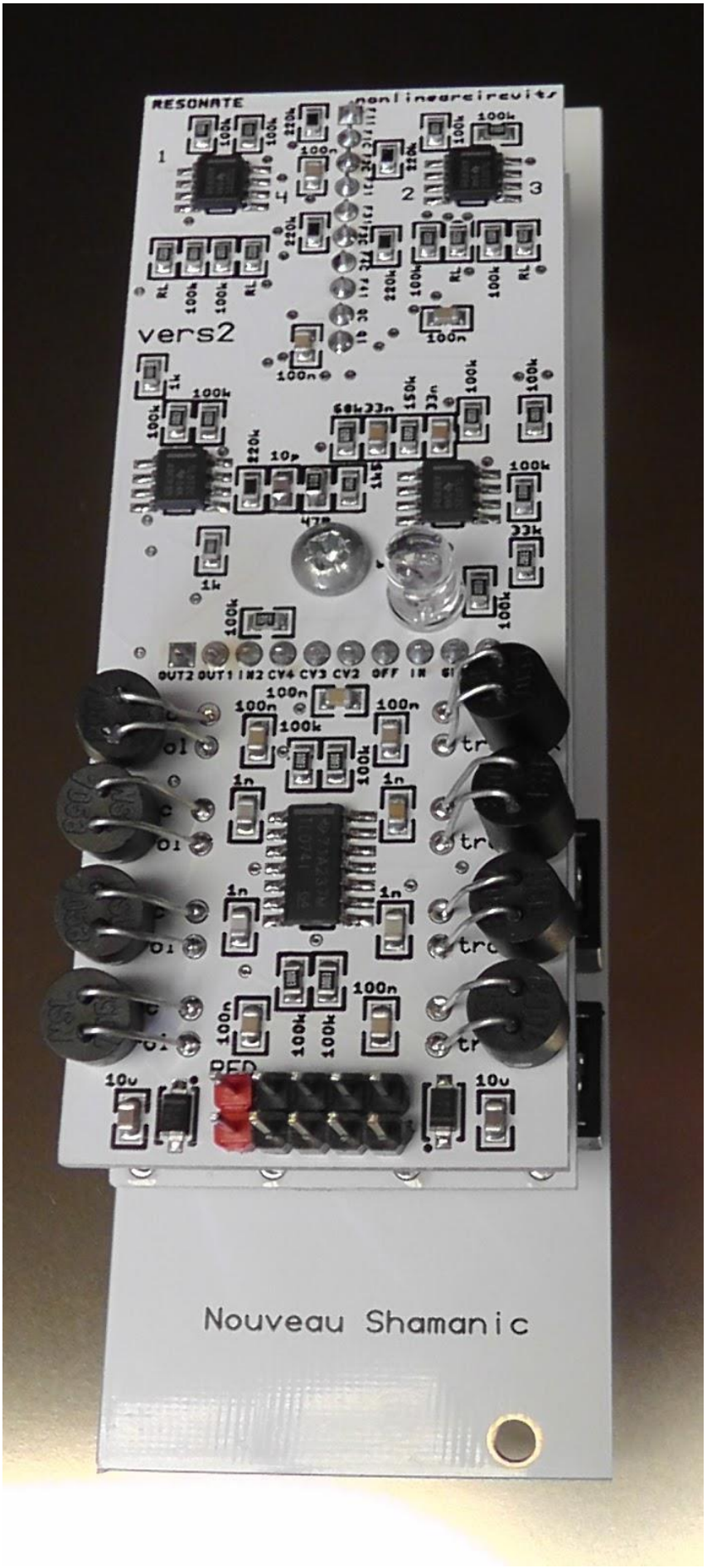


VERSION 3 PCBs



nonlinearcircuits		
RESONATE vers3		
Designer's name	Rev 1.0 03-Dec-17	Page # or name


CV2
CV3
CV4
IN
IN
OUT1
OUT2
NLC


RESONATE
vers2
Nouveau Shamanic

PLEASE
PLACE JACKS & POTS ON
THE OTHER SIDE
CV1
CV2
IN
RESONATE RIM VERSO
Nouveau Shamanic