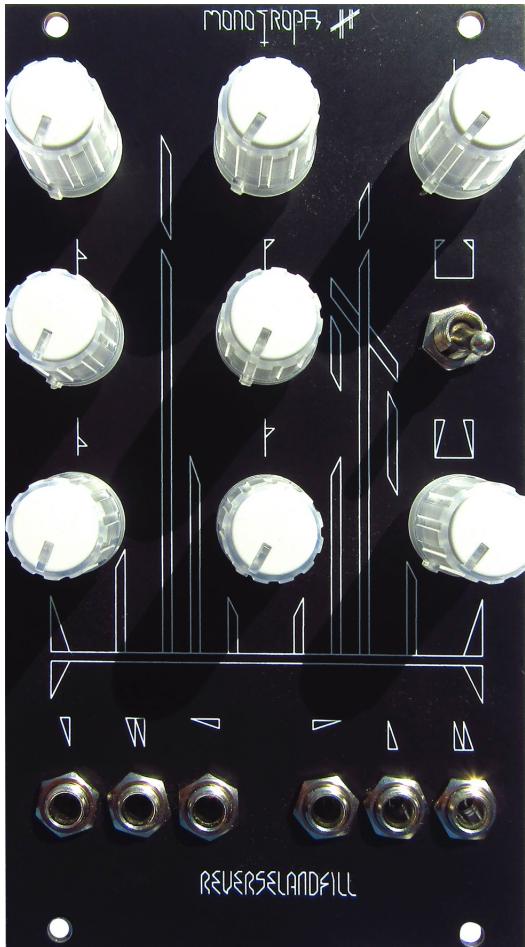


Monotropa mk2 Buildguide dec 2022



Resistors:

Start with soldering these resistors. The value of the resistors are written on the tape.

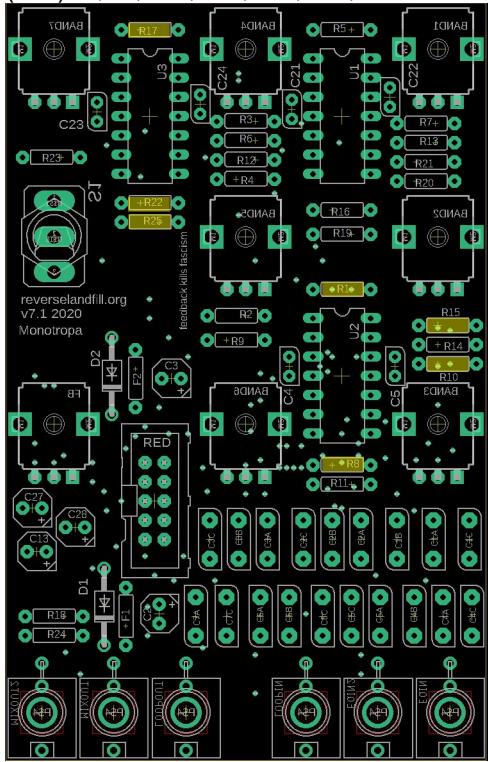
If you are unsure, use a Multimeter to check the value.

Bend the leads in an angle of 90 degrees, insert the resistor in place and solder one lead.

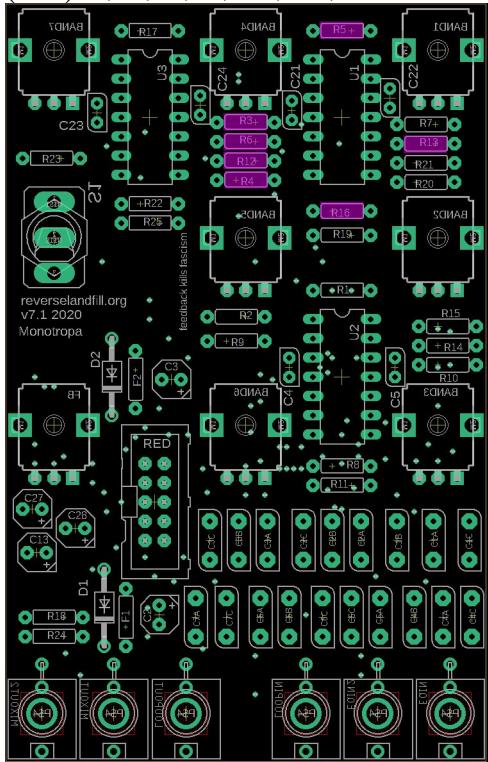
Check if the resistor is aligned correctly (flat to the pcb) and solder the other lead.

Snip off the lead just above the solder.

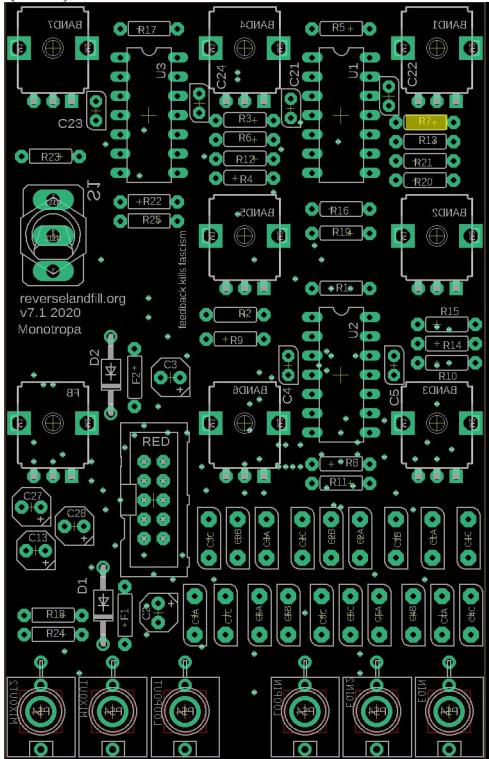
(1k8) r1, r8, r10, r15, r17, r22, r25



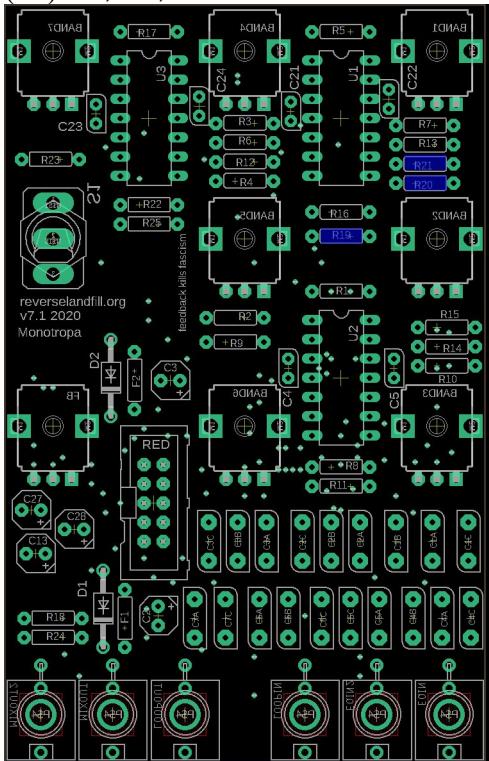
(10k) r3, r4, r5, r6, r12, r13, r16



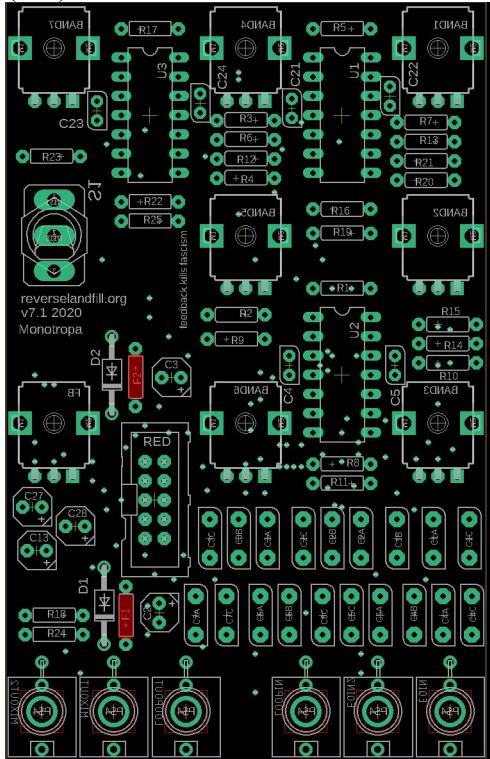
(3k3) r7



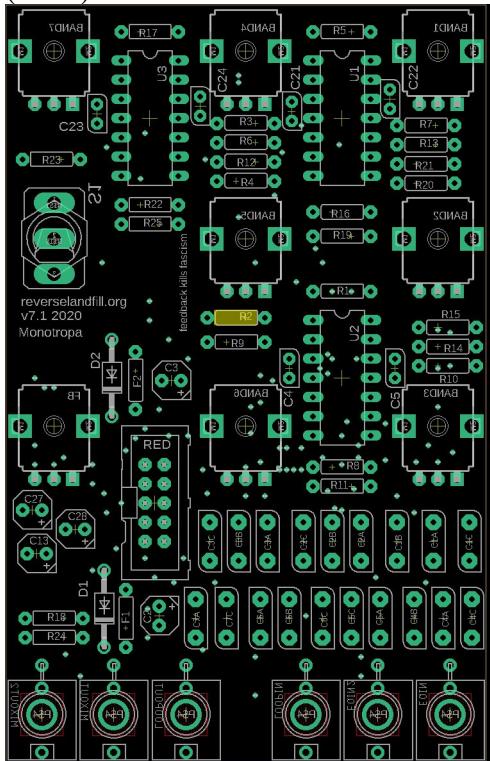
(1k) r19, r20, r21



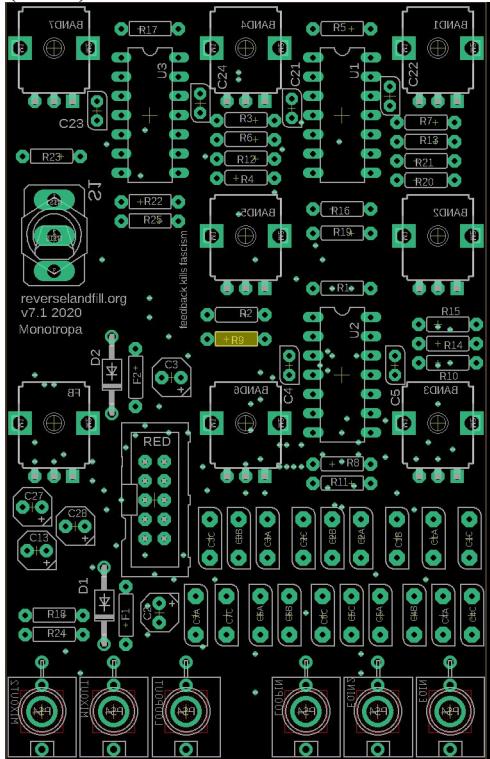
(10r) f1, f2



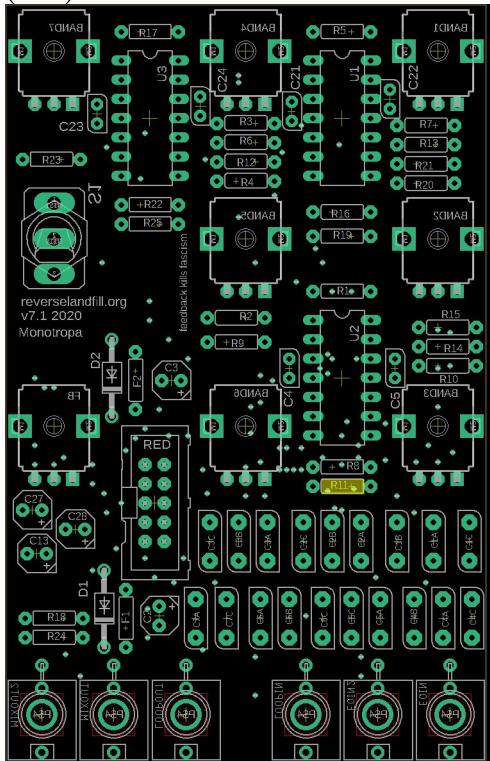
(130k) r2



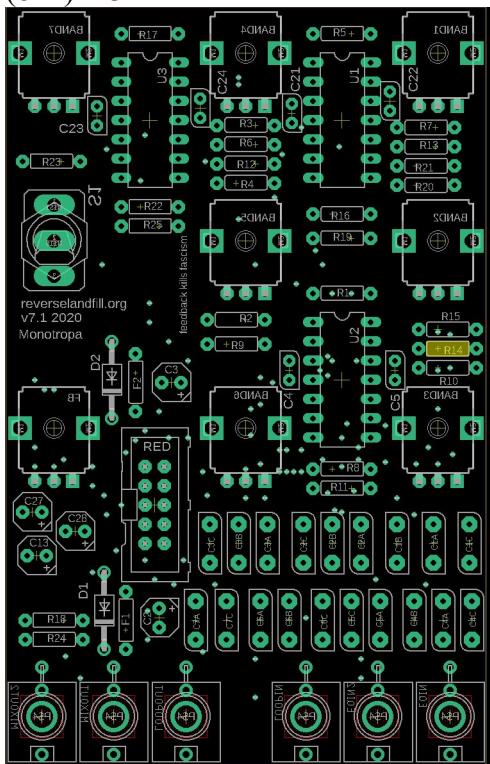
(110k) r9



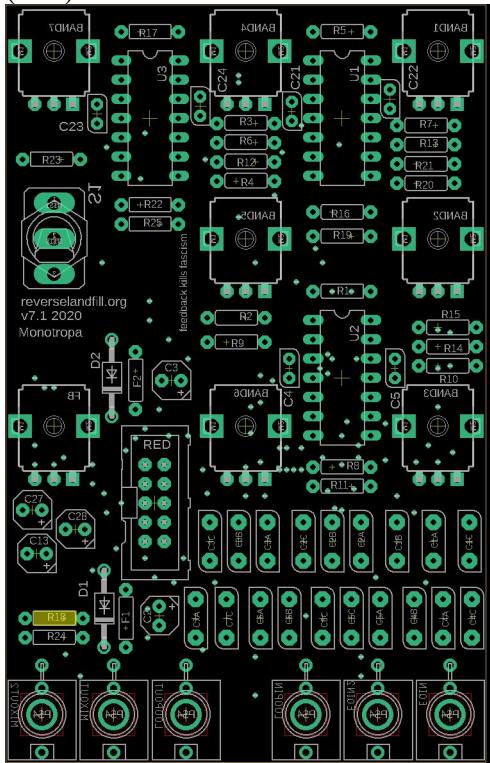
(91k) r11



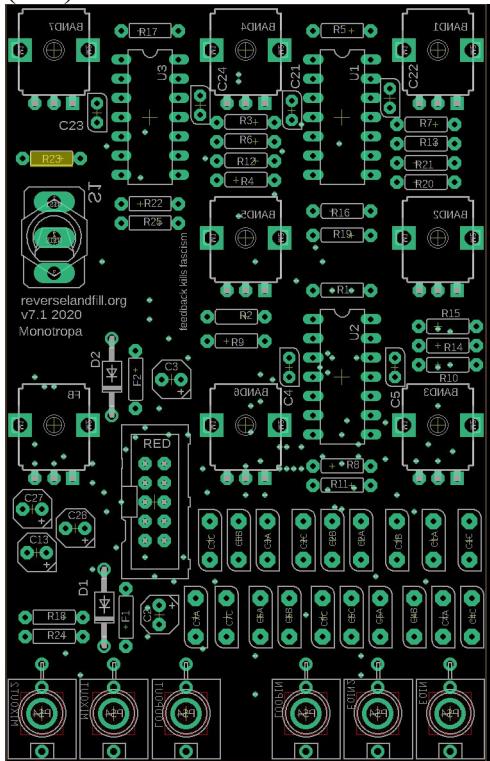
(82k) r13



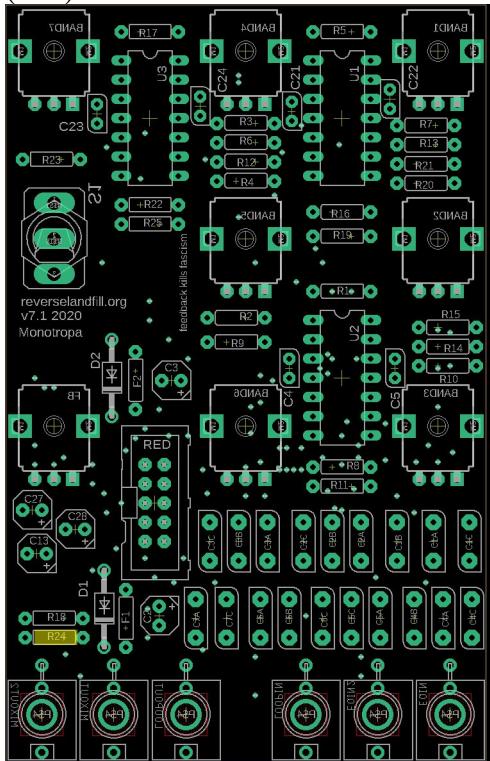
(68k) r18



(62k) r23



(51k) r24



Diodes:

2x 1n4001 at D1 and D2.

The stripe on the diode should match the white line of the silkscreen on the pcb.

IC sockets:

Solder the 3x 14pin IC sockets to u1, u2 and u2. 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:

There are a lot of different capacitors in the kit.

Make sure you read carefully and have the correct one for the position.

First start with the 6x100nF (code 104) caps.

Depending on the kit, they look like yellow blobs or orange disks.

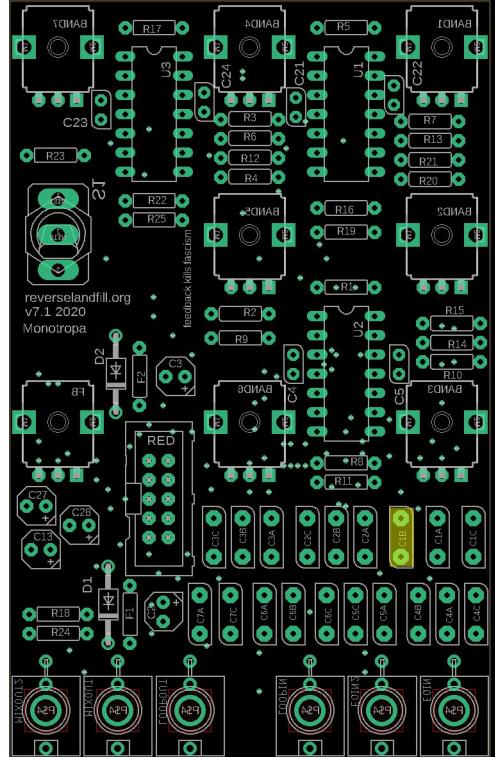
If you have the yellow blobs (with axial leads), mount them “standing up”.

These 100nF capacitors go here: c4, c5, c21, c22, c23, c24

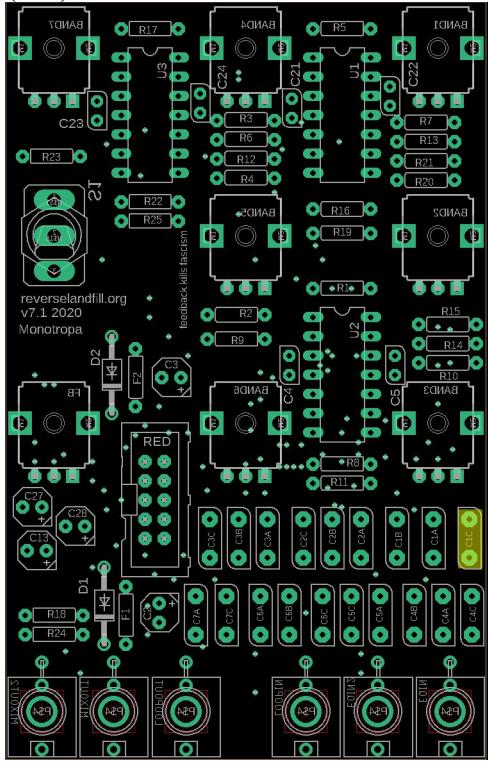
The film capacitors (the blocky ones & 2 small disk shaped ones with the codes, 221 and 471) determine the frequency bands.

The value is written on top. Start with the smaller sized caps.

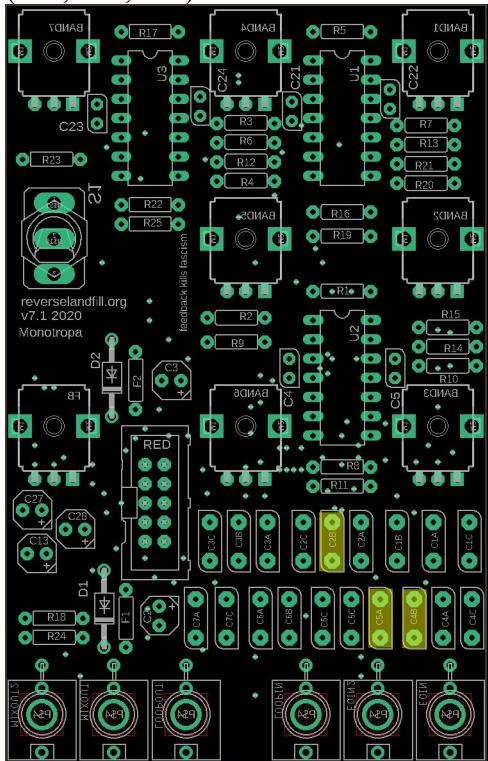
(c1b) 270nF



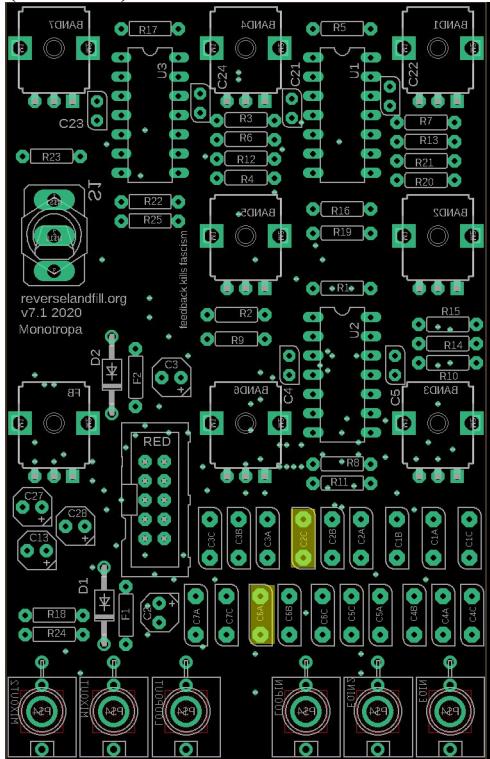
(c1c) 22nF – code 223



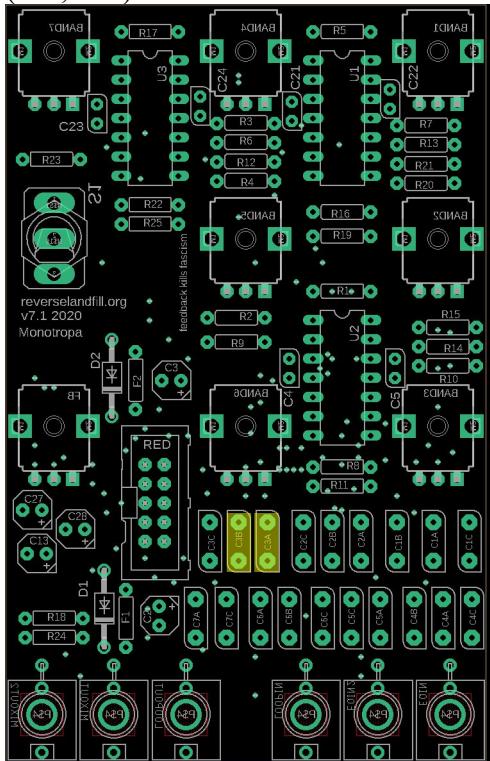
(c2b, c4b, c5a) 33nF



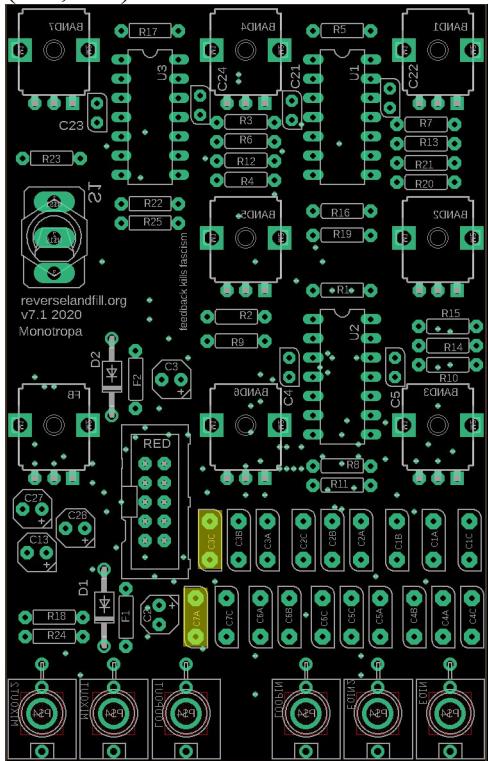
(c2c, c6a) 10nF



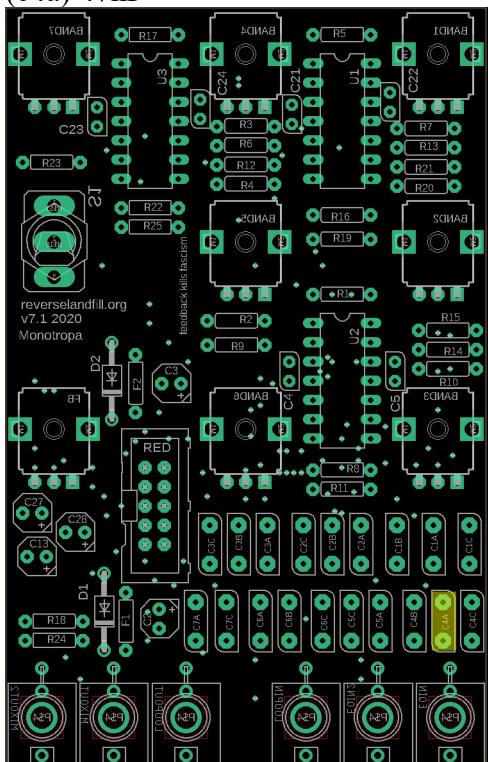
(c3a, c3b) 100nF – code .1J63



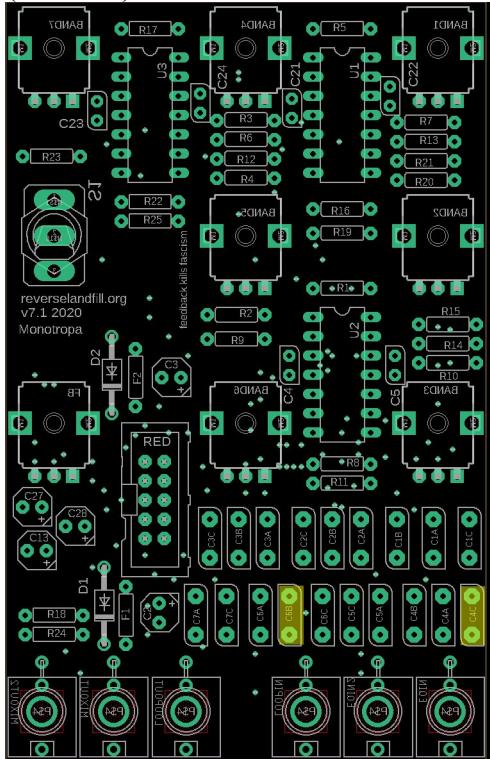
(c3c, c7a) 4.7nF



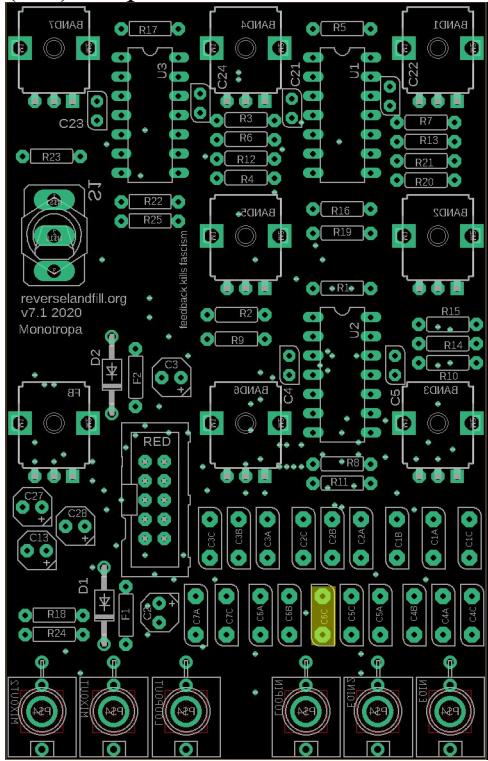
(c4a) 47nF



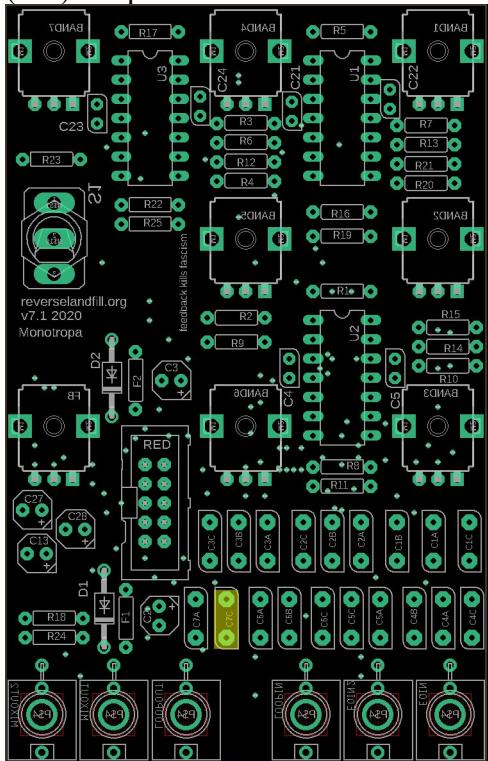
(c4c, c6b) 2.2nF



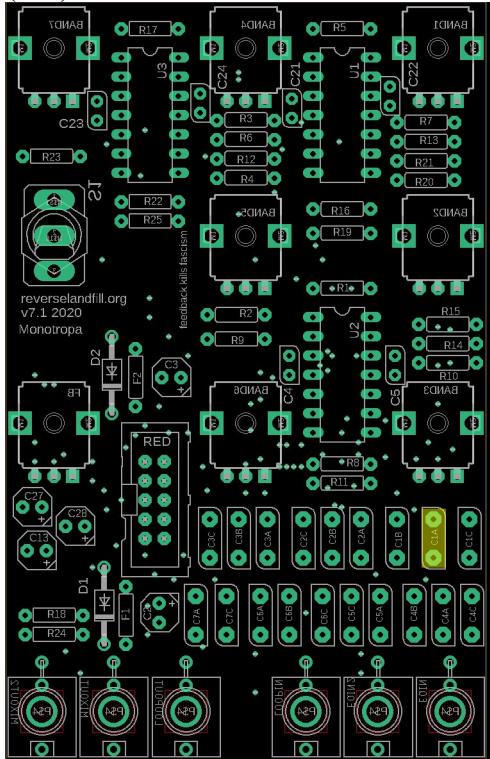
(c6c) 470pF – code 471



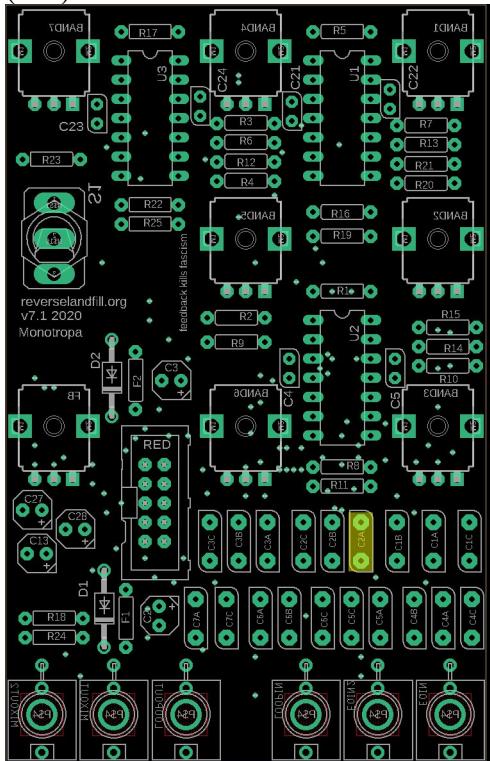
(c7c) 220pF – code 221



(c1a) 1uF – code 1K63



(c2a) 470nF



The electrolytic capacitors have an orientation. The longer leg is PLUS.
There is also a small + sign on the pcb. Make sure you place them correctly!
The value is printed on the side of the parts.

(c13, c27, c28) 1uF
(c2, c3) 10uF

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.

IC's:

Take the 3x TL084 (or TL074 / TL064) 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.

Take a break! Drink some tea or go outside :)

Potmeters and jacks:

Flip the pcb and insert the 8x b10k vertical 9mm potmeters into the pcb.
Also insert the 6x Thonkiconn jack sockets and the 1x SPDT 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, switch 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.

Panel:

Mount the panel to the pcb and secure all the nuts. Carefully tighten them using the correct tools.

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.

Time to test!

As a last check, look over the pcb and check the soldering, check for shorts and polarity of the IC's, Diodes and Electrolytic Capacitors.

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 one of the two outputs to a mixer. (the two jacks on the right)
Set both the Feedback knob (lower right knob) to fully CW.
Set the switch to the down position (feedback path engaged)
The frequency knobs will now self resonate, when turned CW. Try each band.
You should hear low, mid, mid/high, and high tones. It works!

Set the Feedback to 12 'o' clock.
Patch a drum sound to one of the two inputs. (left two jacks). Patch the output to a mixer.
Try out different settings. You should hear distortions and resonating sounds.

Refer to the sound manual for other patch ideas and a guide to all functions.

Troubleshooting:

Check the orientation of both IC's, the polarised capacitors, the power header.
Did you insert the IC's? Check the soldering. reflow pins if needed.

Credits:

CC-BY-SA

Part of the EQ was inspired by the Musicthingmodular Graphic EQ / Boss GE7 circuit.

The rest of the design is by:

www.reverselandfill.org

martijn@reverselandfill.org