# QUANTUM DEFRAKULATOR MK3
Build Document last updated june 2020
for PCB <u>version 3.0</u>

Make sure that you are looking at the correct version of the build document to match the version of your PCB.

The Quantum Defrakulator is a drone synthesizer with 3 square wave oscillators and one LFO (low frequency oscillator). Each oscillator has independent controls for frequency and volume. With the LFO engaged, oscillator 1 turns into a 3 or 4-step sequencer.

Have fun!
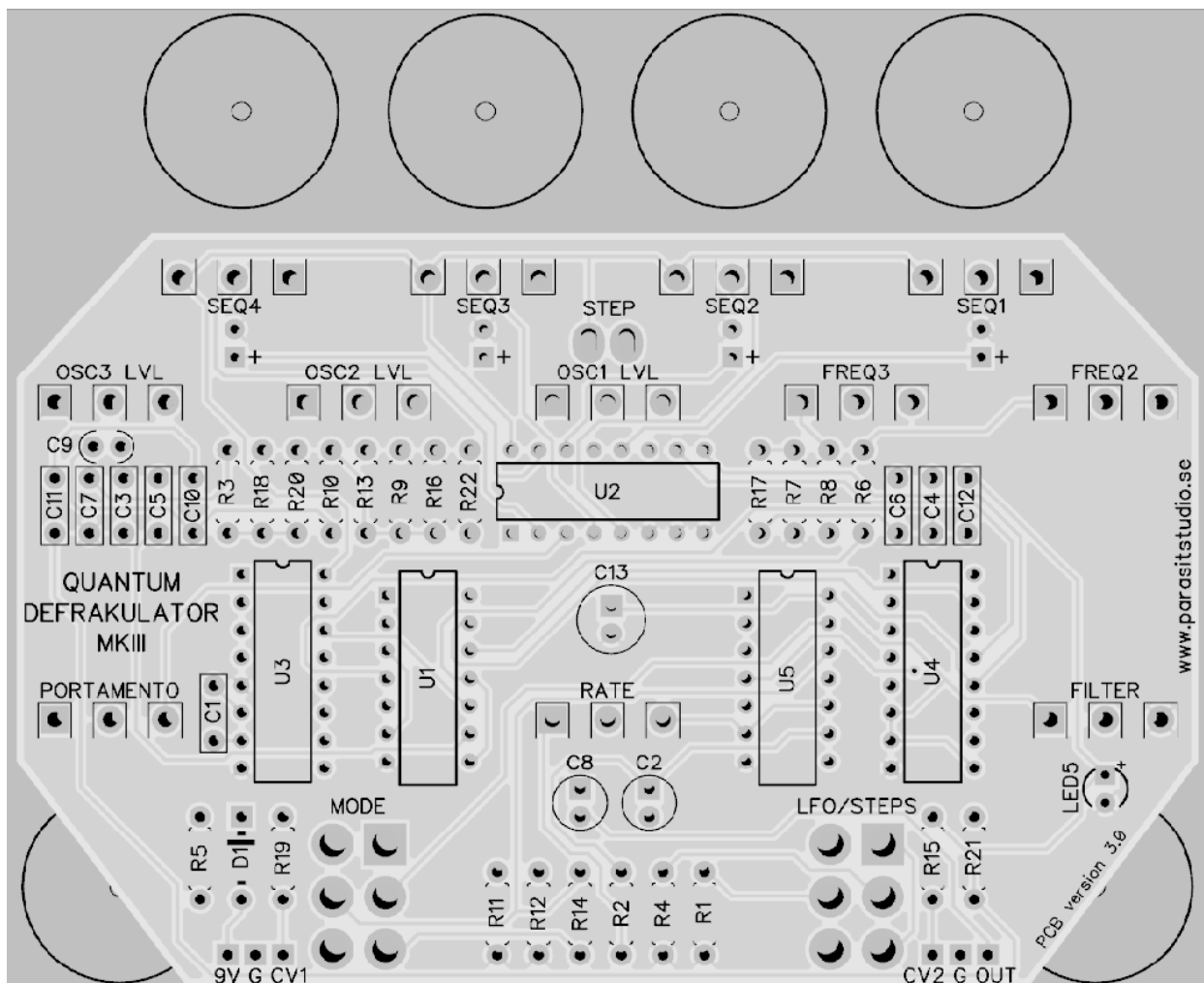
Changelog MK3 (PCB 3.0)

- The circuit has been redone almost from scratch (again) to fix an issue with the 2.0 PCB LFO, but also to add a couple of new features. It has a few more components than the 2.0 PCB, but is still a straight forward build. :)

- New feature: NAND modulation. XOR (ringmod) modulation was added in the 2.0 version of the PCB. The 3.0 version also adds NAND modulation for a different flavour. Both these settings are modulating oscillator 1 (against oscillator 2).

- New feature: Manual stepping of the sequencer. Now you can push a momentary button to manually toggle through the steps of the sequencer when the LFO is off. If the button is pressed while the LFO is on, it will temporarily stop the sequencer.

  While turning off the LFO, it will also stop on the current sequencer step (unlike the old versions which always went back to the first step) so that it's easier to tune each step of the sequencer.

Notes

- The Quantum Defrakulator output is meant to hook up to an amplifier like a guitar amp or through a mixer to a PA system.

  It can't drive a speaker directly. If you want to drive a speaker you will need to add an amplifier at the output, such as a LM386 or similar. There's plenty of simple 9v amplifier projects around if you do a search. :-)

General builds tips

- Solder the low profile components first, from short to tall. Recommended order: resistors, diodes, IC socket, film-caps, electrolytics, pots.
- CMOS chips are very sensitive to static charges and can be easily damaged. It's a good idea to wear a anti-static wristband or at least avoid wearing a wool jumper and petting your cat/dog while building...
- Always use sockets for IC chips to avoid heating them directly. It also makes it much easier to swap them out if needed.
- Pay special attention to the orientation of the diodes and electrolytics.
- This PCB is designed for board mounted angeled pots, but if you want to use regular solderlug-pots, the square holes represents pin 1 of the pot.
- Cover the backside of the pots with pot covers or tape, so they don't short out anything on the PCB.

The Quantum Defrakulator MK3 prototype

Quantum Defrakulator MK2 Bill of Materials (BOM)

Resistors		R21	47K	IC's	
R1	100K	R22	CLR*	U1	CD4069UBE
R2	2.2K	**Capacitors**		U2	CD4052
R3	22K	C1	220nF	U3	CD4046
R4	100K	C2	47uF	U4	CD4015
R5	100K	C3	220nF	U5	CD4093
R6	4.7K	C4	220nF		
R7	220K	C5	220nF	**Potentiometers**	
R8	4.7K	C6	100nF	RATE	C50K
R9	220K	C7	100nF	FREQ 2	C100K
R10	330K	C8	2.2uF	FREQ 3	C100K
R11	15K	C9	220pF	PORTAMENTO	A1M
R12	330K	C10	100nF	FILTER	B500K
R13	220K	C11	4.7nF	rest of the pots (x7)	B100K
R14	100K	C12	6.8nF	**Switches**	
R15	47K	C13	100uF	LFO/STEPS	DPDT on/off/on
R16	220K	**Diodes**		NAND/XOR	DPDT on/off/on
R17	100K	D1	1N4001	1x Momentary push button	
R18	100K				
R19	100K	LED5** 3mm normal red diffused LED			
R20	100K	4x LED's (for the sequencer)			

- * Current Limiting Resistor for the sequencer LEDs. They share a single resistor since there's always just one LED engaged at the time. Use the appropriate value for your LED type.
- I recommend using a 4.7K resistor for a diffused LED or a 15K resistor for a clear superbright LED.
- I recommend using multilayer ceramic capacitors for the 220nF caps to keep the size down. For the larger values use electrolytic capacitors as usual.
- ** LED5 is not supposed to be external. It's just part of the filter circuit, so just mount it on the PCB on the component side like the rest of the components. The LED type/size and colour is not critical.

Things not included in the BOM that you will also need:

1x Hammond BBDD/JJ enclosure, 1x DC jack, 1x 6.3mm mono jack for the output, 12x knobs, 4x LED bezels (optional), 2x 3.5mm jacks (optional, for the CV inputs)

## Fitting everything inside the enclosure

Getting everything mounted inside the enclosure can be a bit tricky because of all the pots, leds and switches. Here's what I do.

1. Solder the middle leg of the potentiometers to the PCB first (that way the pots won't fall out when you try to insert the board into the enclosure, but they will still have some movement for adjustment).
2. Place the switches inside the enclosure.
3. Put the LED's in the PCB (without soldering).
4. Carefully put everything together and make sure that everything lines up.
5. Solder everything in place.

In this picture example I put the top four pots inside the enclosure first, since the DC jack was in the way, but it's much easier to solder them to the PCB insted which I was doing on later builds. This shows the 2.0 PCB, but the same method goes for version 3.0
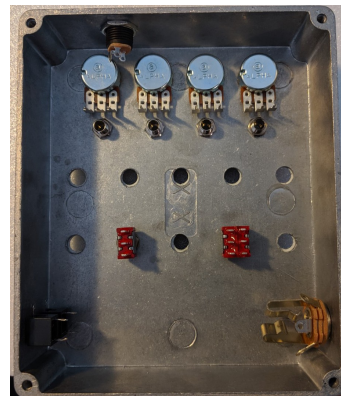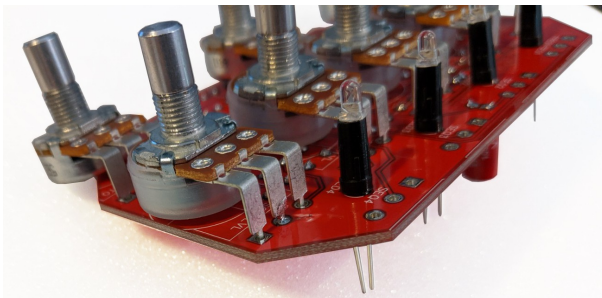
When using a pot cover I usually keep the legs of the pots a bit liften from the hole so that the pot sits flush against the PCB.
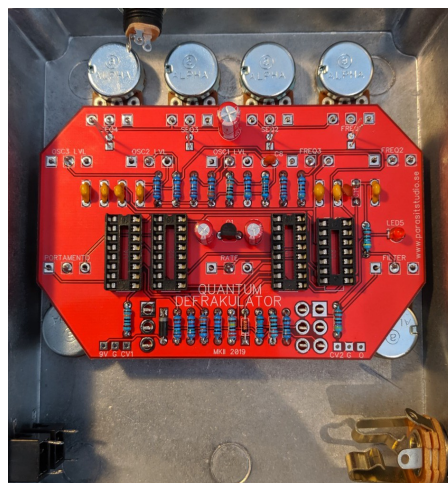
Offboard Wiring

- 9V pad to DC jack positive lug (or battery clip +)
- G pad to DC jack negative (or battery clip -)

- O pad to output jack Tip
- G pad to output jack Sleeve

If you want to drive this effect from a 9 volt battery, I recommend using an extra switch (like a SPST, or a SPDT using two of the pins) as an on/off switch, to disengage the battery while not in use.
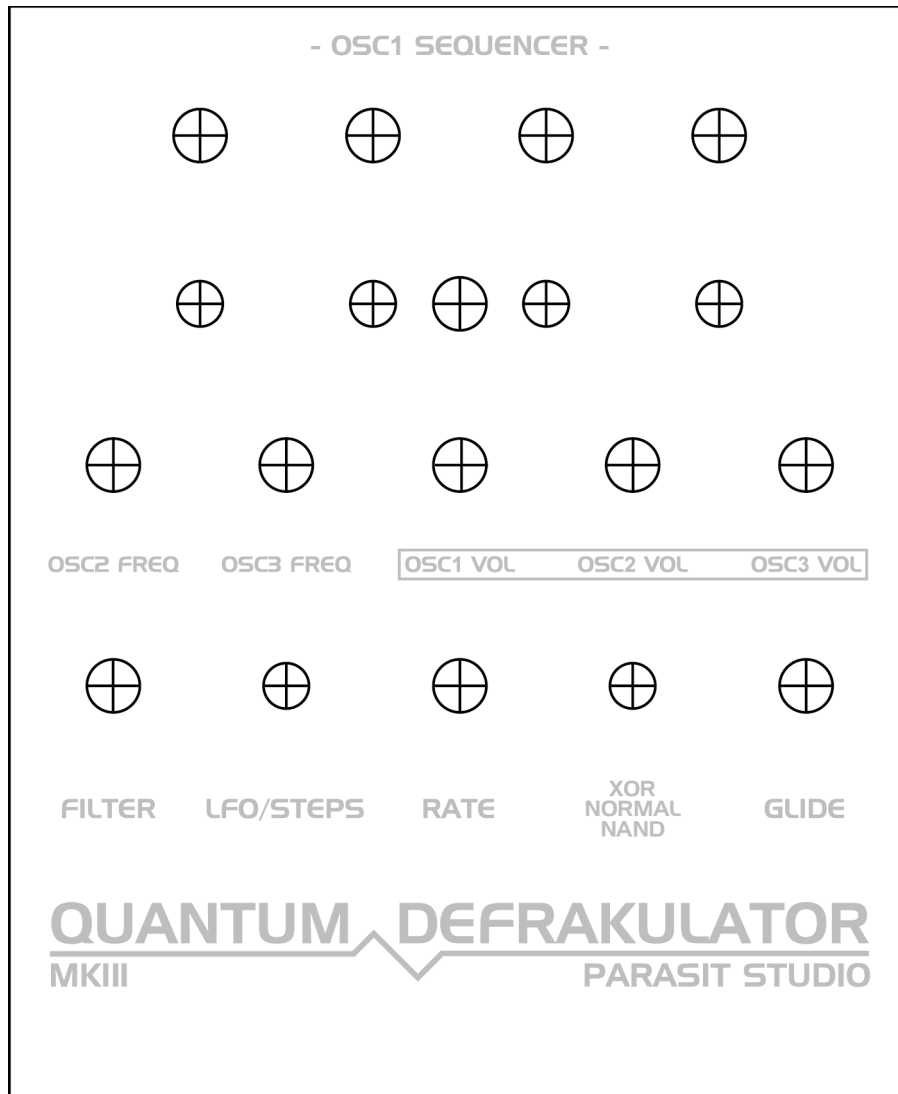
CV jack Wiring

- To hook up the CV (control voltage) input jacks, connect the CV pad on the PCB to the tip of a 3.5mm jack (or jack of your choice). Make sure that the sleeve of your jack is connected to ground (it usually is connected to ground via the enclosure unless you have an isolated jack type). If you use a stereo/balanced jack, just leave the ring lug unconnected.

CV1 = oscillator 1 frequency
CV2 = filter cutoff

The CV input range is 0-9 volts, but 0-5 volts works great aswell.

Drilling template (Hammond BBDD/JJ)

- OSC1 SEQUENCER -

OSC2 FREQ   OSC3 FREQ   OSC1 VOL   OSC2 VOL   OSC3 VOL

FILTER   LFO/STEPS   RATE   XOR NORMAL NAND   GLIDE

QUANTUM DEFRAKULATOR

MKIII   PARASIT STUDIO

- *Drill holes on the sides for the output jack and the DC jack to your own preference.*
- Make sure your printer is printing 100% print size and isn't doing any scaling.
- Use at your own risk! This template is approximate.
- The pot and switch placements are exactly the same as PCB version 2.0 (MK2), except one extra hole for the push button.

- **Measure and confirm before drilling!**

Schematic
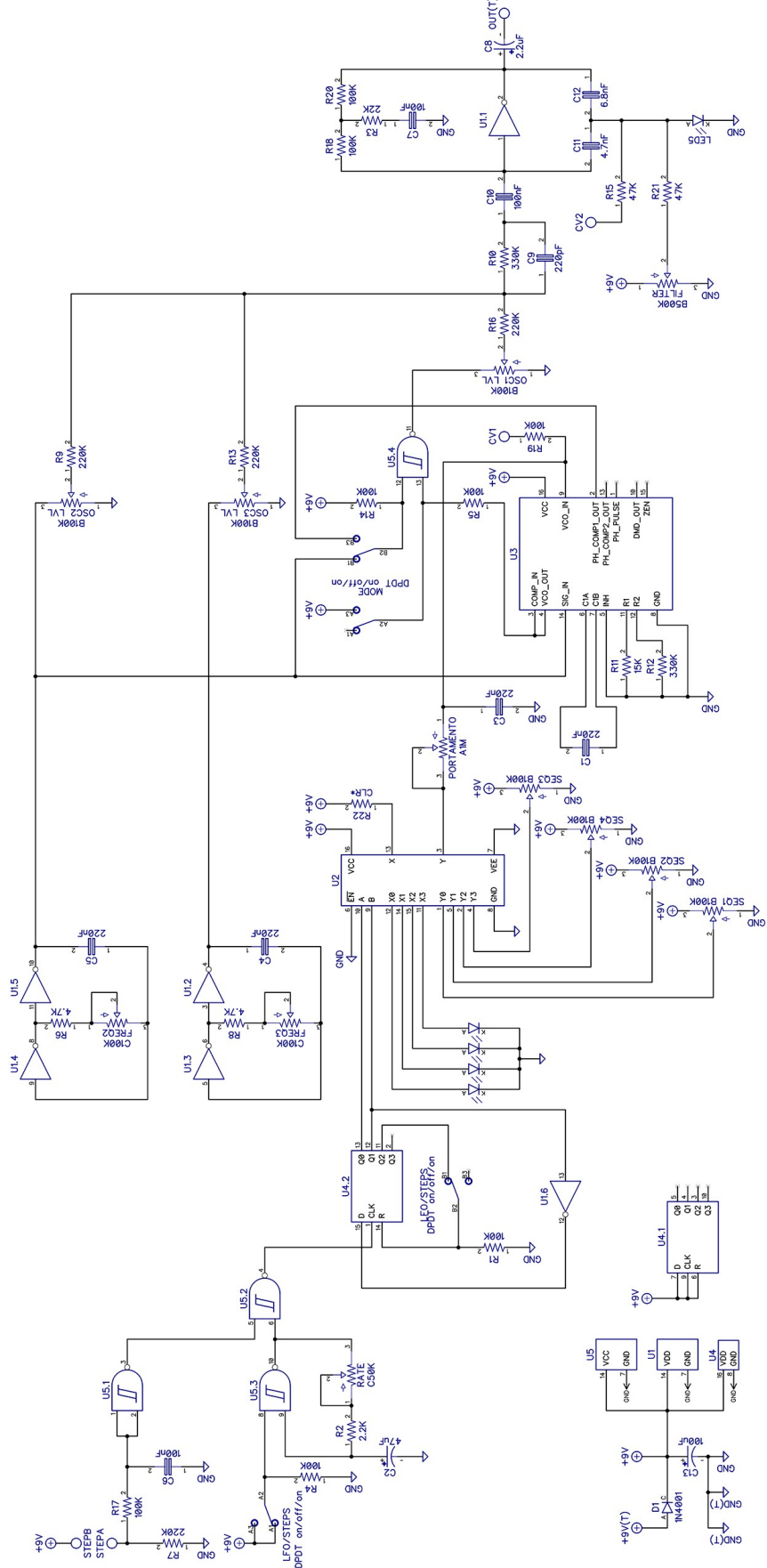
Troubleshooting

There's always a chance of running into trouble. To minimize error, follow the BOM and general building tips carefully. Take your time and don't rush. Take a break now and then. Use good solder, and it helps to have a decent soldering station insted of a cheap iron.

If you are still having trouble, please visit the madbean forum Parasit Studio subforum section and ask for help there.

http://www.madbeanpedals.com/forum/index.php?board=84.0

If you have bought the Musikding kit and have recieved a faulty or missing component, please contact musikding directly.

kontakt@musikding.de

Terms of use

PCB's from Parasit Studio are intended for DIY use only. Commercial resale is not allowed. It's meant for personal use, which means that it's not allowed to build many pedals and sell them for profit to strangers using public forums, webshops or craiglist ads ect. However, it's totally ok to build a few pedals and give to your friends and bandmates. After all, that's what this hobby is about. :)

www.parasitstudio.se
parasitstudio@gmail.com