

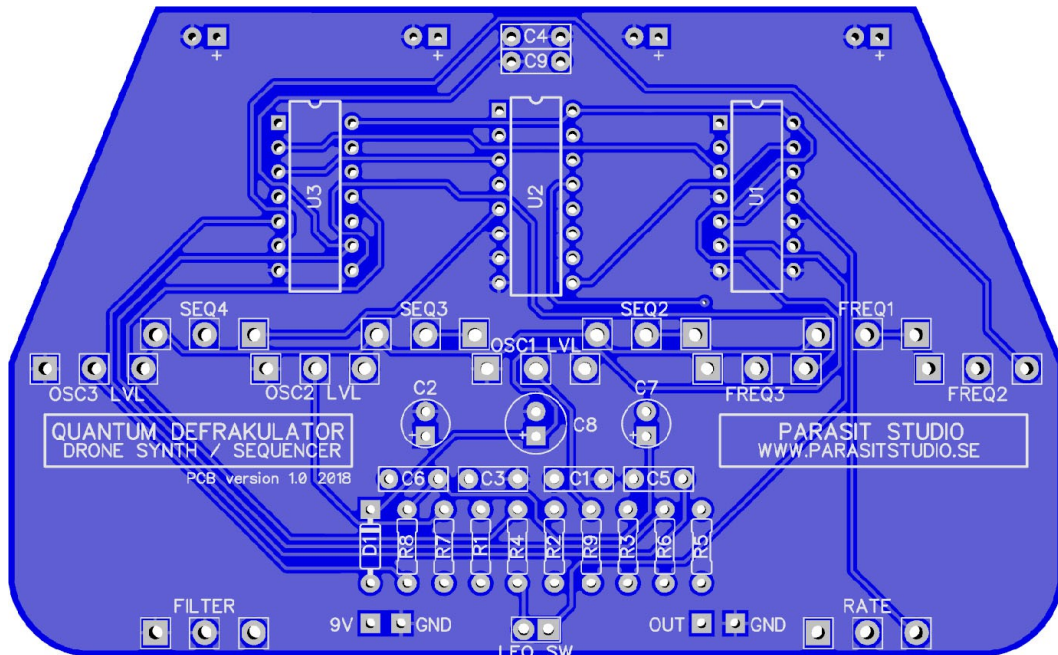
PARASIT STUDIO

QUANTOM DEFRAKULATOR

Build Document last updated june 2018
for PCB version 1.0

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

Have fun!



Note

- *The Quantum Defrakulator output is meant to hook up to an amplifier like a guitar amp.*

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. :-)

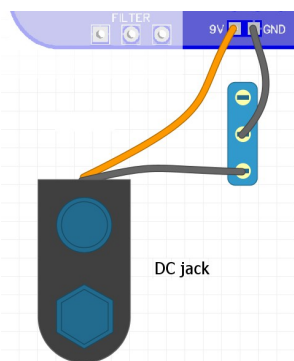
General builds tips

- 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.
- **There are a LOT of pots on this PCB. Be sure to place them in the PCB without soldering first, THEN place them in your drilled enclosure. Gently tighten the nuts to the enclosure, then solder LAST. Otherwise, it will be really hard to get this in your enclosure.**
- It can be a good idea to divide the pots and LED mounting into separate steps, to make it easier to mount everything inside the enclosure.
- For example: mount the PCB inside the enclosure without the LED's first and solder the pots, then take out the PCB and put it back inside again with the LED's, then solder the LED's.

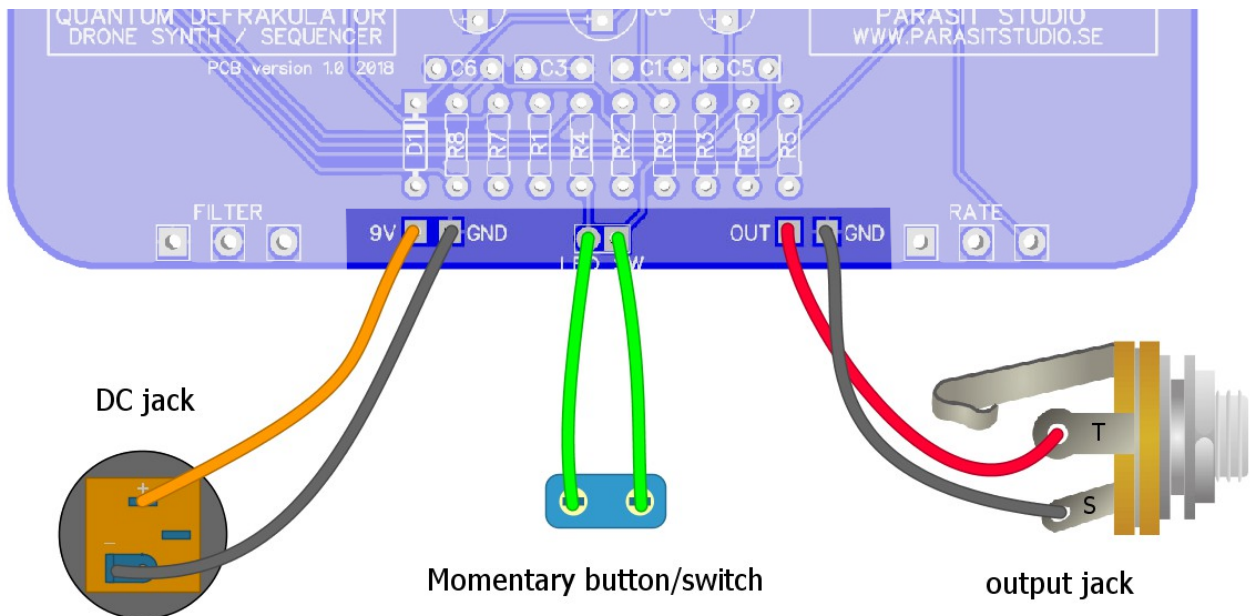
Quantum Defrakulator Bill of Materials

Resistors		Capacitors		IC's	
R1	10K	C1	220nF	U1	CD4013
R2	2.2K	C2	47uF	U2	CD4052
R3	CLR*	C3	100nF	U3	CD40106
R4	1M	C4	220nF	Potentiometers	
R5	10K	C5	220nF		
R6	10K	C6	22nF	RATE	C50K
R7	100K	C7	22uF	FILTER	C50K
R8	100K	C8	100uF	Rest of the pots (x9)	B100K
R9	100K	C9	100nF	(11 pots total)	
Diodes					
D1	1N4001				
4x LED's (for the sequencer)					
1x optional LFO on/off LED					
		Switches			
		LFO sw	Momentary SPST		

- * Current Limiting Resistor for the sequencer LED's. 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 to keep the size down of the 220nF caps. For the larger values use electrolytic capacitors as usual.
- For the momentary switch you can use any type of momentary switch or push button. It can be either normally open or normally closed, it doesn't matter.
- If you want to drive this effect from a 9 volt battery, I recommend using an extra switch (like a SPST or using two pins of a SPDT) as an on/off switch, to disengage the battery while not in use -



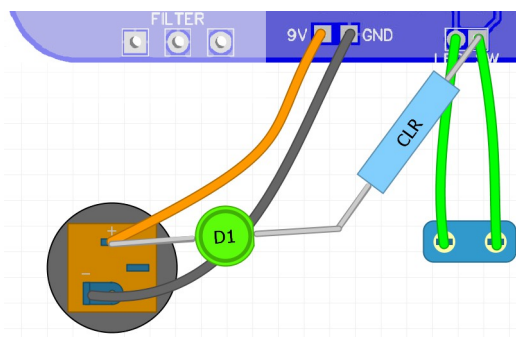
Offboard Wiring



- 9V pad to DC jack positive lug (or battery clip +)
- GND pad to DC jack negative (or battery clip -)
- OUT pad to output jack Tip
- GND pad to output jack sleeve
- LFO sw pads to momentary SPST switch or button (*which wire goes to which switch lug doesn't matter*)

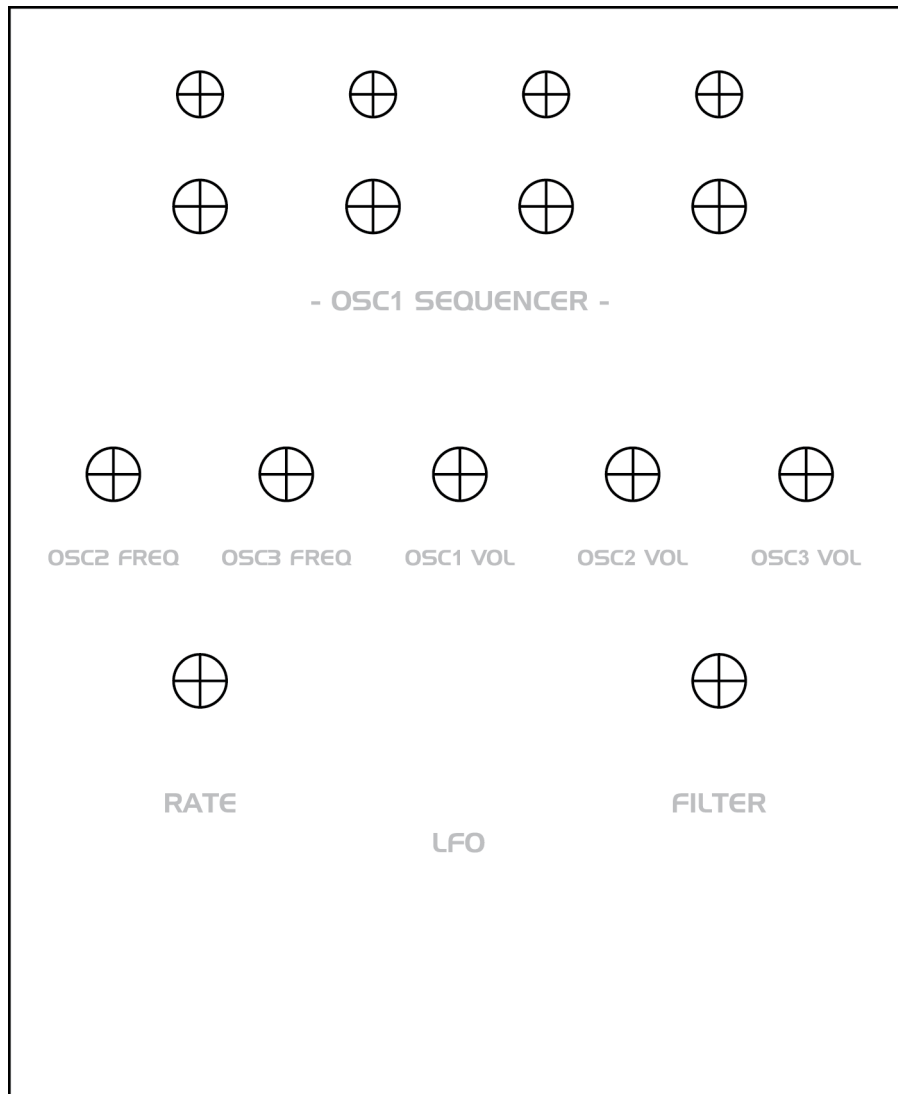
Optional sequencer on/off indicator LED

- If you want an LED to indicate when the LFO is running (like I have in my complete build, combined inside the LFO button) this is how you hook it up.



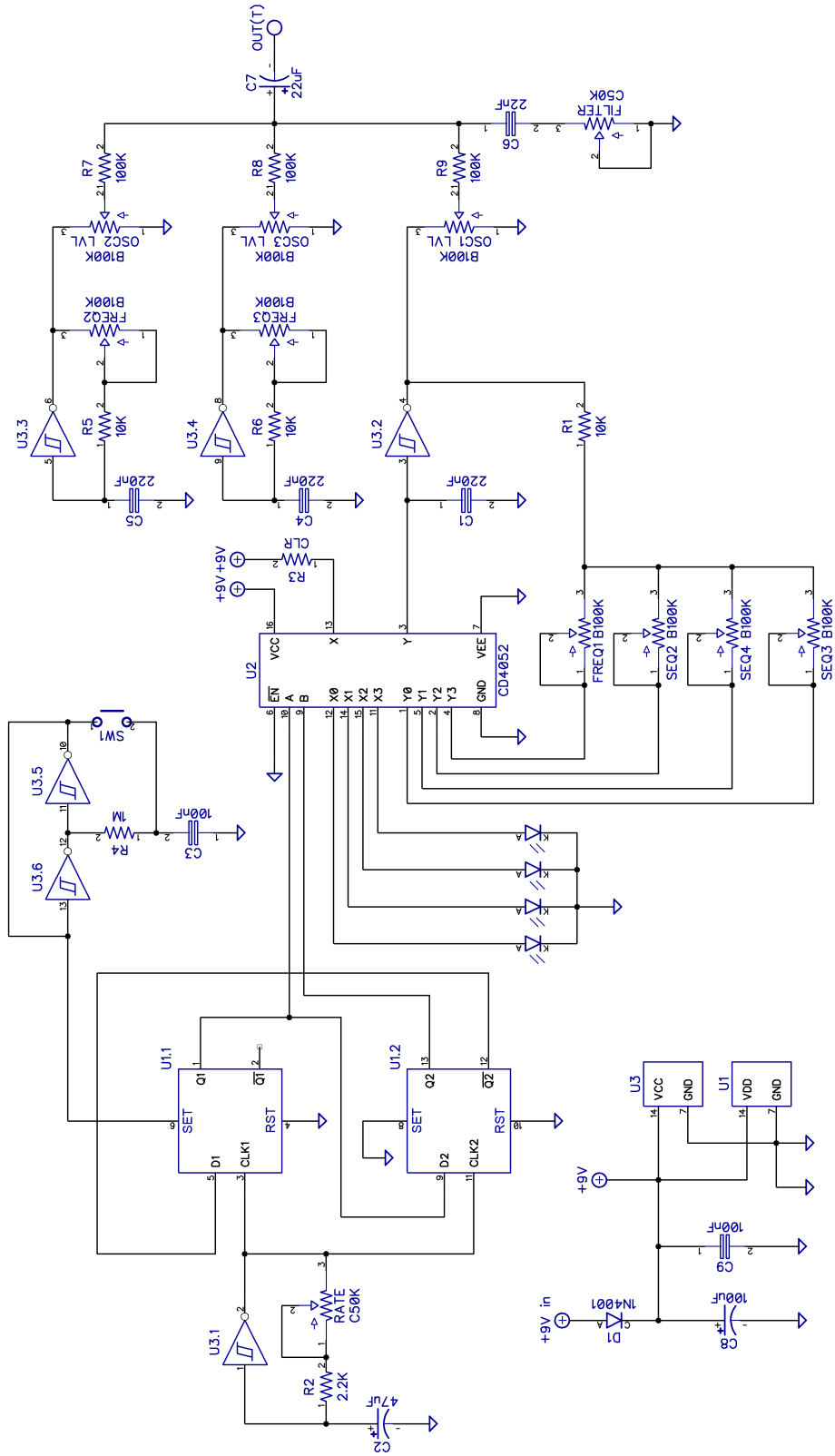
- Connect the square "LFO SW" pad to a current limiting resistor
- LED cathode/negative side to the resistor
- LED anode/positive side to DC jack (or battery clip) positive/+

Drilling template (Hammond BBDD/JJ)



- **+ one extra top hole** for the momentary button/switch!
I didn't include it on the template because people will probably use different placements and different types of switches or buttons.
- *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.
- **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 craigslist ads ect. However, it's totally ok to build a few pedals and sell to your friends and bandmates. After all, that's what this hobby is about. DIY or DIE!

www.parasitstudio.se
parasitstudio@gmail.com