

Engine

Dual Complex VCO with wave shaping and modulation features

Build Guide

PCB V1.1 May 2022

Written and Illustrated by Ruben Sponar

Introduction

This Build Guide is written for builders with experience in soldering and assembling PCBs. Due to the complexity of the build, the guide is focused on the steps that require special care or techniques. Therefore, not every component value is described and listed as a single step.

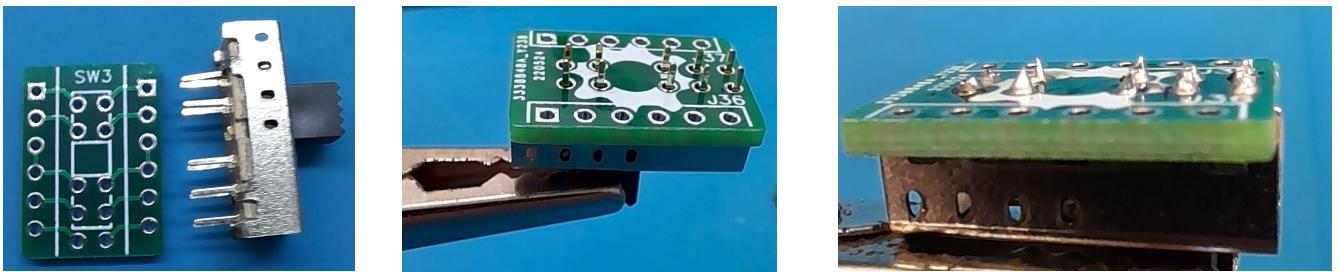
Tools Required

Soldering iron (+wire)
side-cutter
snipe nose plier
flat head screwdriver
Nut wrench / driver M6, M8, (M9/M10)
Digital Multimeter
Helping (third) hand / PCB holder

BUILD GUIDE

STEP 1 (Switch PCB)

Take the small Switch adapter PCB and the 4 position slide switch. Place the switch on the side that says SW3. It is recommended to use a helping hand to secure them together while soldering. When soldered, trim the solder legs flush with the PCB to avoid shorting them later. Put it aside for later use.



STEP 2 (Main PCB)

Resistors

There are different ways to go, you might already have developed your own method to place the components and solder.

You can always select the resistors of the same value, place them, turn the board around and solder in place. Trim the solder legs and move on to the next value.

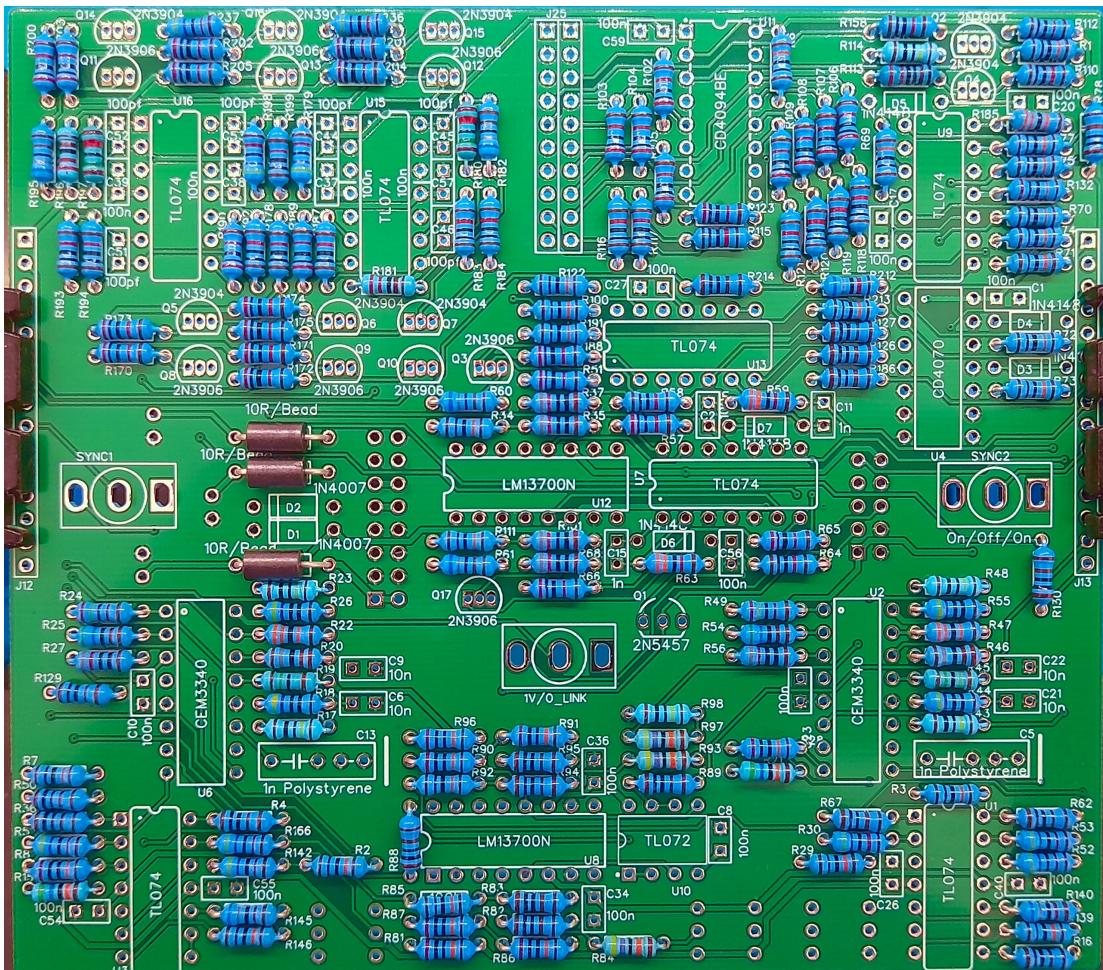
Another method is to place all resistors and solder them from the top, without turning around the board.

On the next page, you can find the list of resistors needed for the main PCB and a picture of all resistors in place.

To help keeping track of the already placed components, print the BOM and cross out the parts you already soldered.

For the "10R/Bead" you can either use Ferrite Beads or 10R resistors.

3	10R/Bead	L2,L1,L3
1	220R	R113
4	470R	R55,R44,R26,R18
6	820R	R56,R27,R101,R68,R37,R35
19	1k	R214,R127,R126,R89,R88,R78,R67,R62,R50,R36,R16,R4,R95,R94,R83,R82, R130,R129,R186
2	1k8	R46,R20
2	2k	R196,R181
2	5k6	R54,R25
4	6k8	R195,R182,R61,R60
20	10k	R199,R194,R193,R192,R187,R184,R183,R178,R158,R112,R110,R64,R58,R121, R120,R119,R118,R117,R116,R115
2	15k	R197,R180
28	20k	R205,R204,R203,R202,R201,R200,R175,R174,R173,R172,R171,R170,R66,R51, R65,R57,R123,R109,R108,R107,R106,R105,R104,R103,R102,R190,R189,R188
2	24k	R49,R24
2	33k	R63,R59
10	47k	R198,R179,R166,R53,R30,R5,R142,R52,R237,R236
3	68k	R212,R145,R140
24	100k	R213,R146,R139,R92,R85,R77,R74,R73,R72,R71,R70,R69,R6,R111,R34,R29, R7,R3,R2,R1,R122,R100,R75,R191
9	200k	R28,R8,R91,R90,R87,R86,R132,R96,R81
3	330k	R47,R22,R185
3	470k	R97,R93,R84
2	560k	R21,R15
6	1M	R114,R98,R48,R43,R23,R17
2	1.5M	R45,R19



STEP 3 (Main PCB)

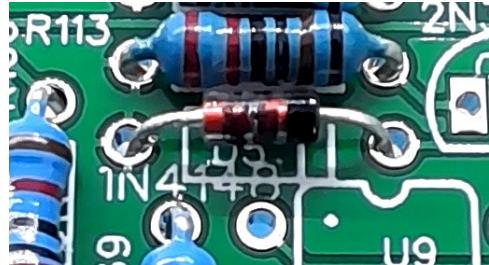
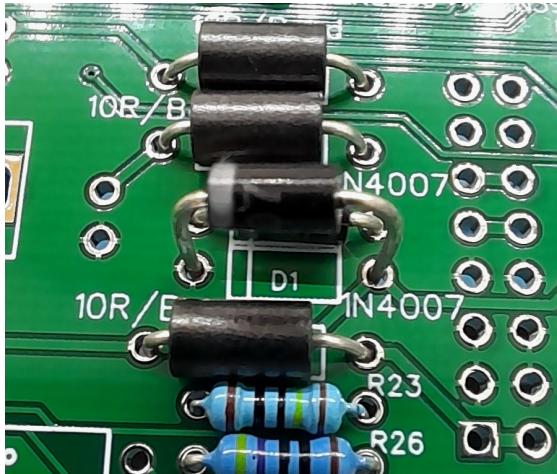
Diodes

Before continuing with the build, check the time! You won't believe how much time has passed while "only" soldering the resistors. Go to sleep if it's late or make a break. It is quite unlikely that you will build the full module in just one day.

Identify the 1N400X* and 1N4148 diodes and put them in place. Orientation is Important! Make sure the line on the diodes lines up with the stripe on the PCB silkscreen.

*can be any from 1N4001 to 4007 or 1N5819 (marked as 1N4007 on the PCB)

2	1N4001	D1,D2
5	1N4148	D3,D5,D4,D6,D7

**STEP 4 (Main PCB)**

ceramic capacitors

Find the ceramic capacitors

100pf is marked as "101"

1nf is marked as "102"

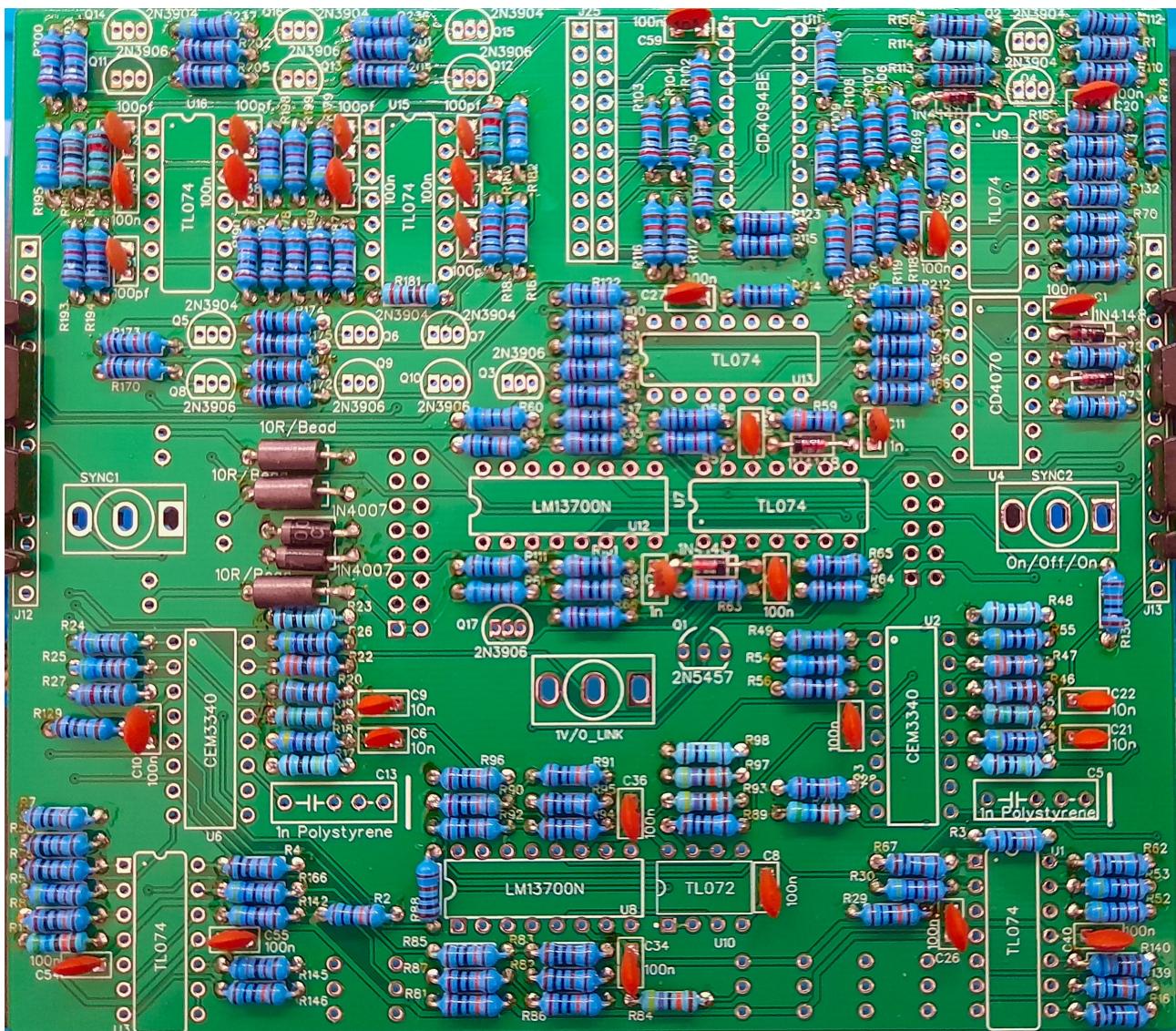
10nf is marked as "103"

100nf is marked as "104"

Orientation doesn't matter. Put them in place and bend the solder legs outwards, so they don't fall when turning around the board.

Solder all ceramic capacitors in place. On the next page you can see how your PCB should look by now.

6	100pf	C53,C52,C51,C46,C45,C44
2	1n	C15,C11
4	10n	C22,C21,C9,C6
20	100n	C59,C57,C56,C55,C54,C40,C39,C38,C37,C36,C34,C27,C26,C23,C20,C17,C10,C8,C2,C1



STEP 5 (Main PCB)

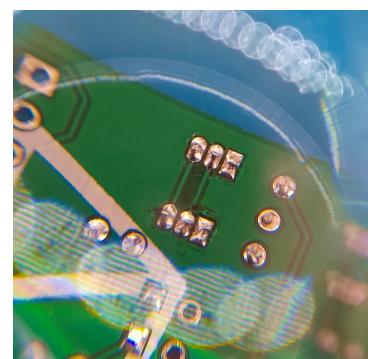
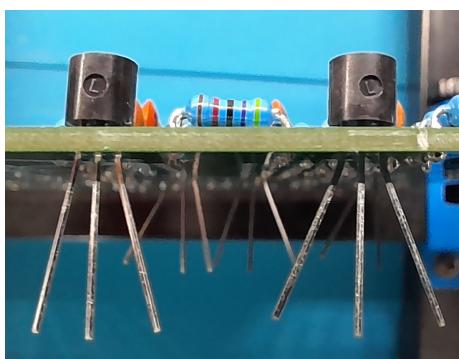
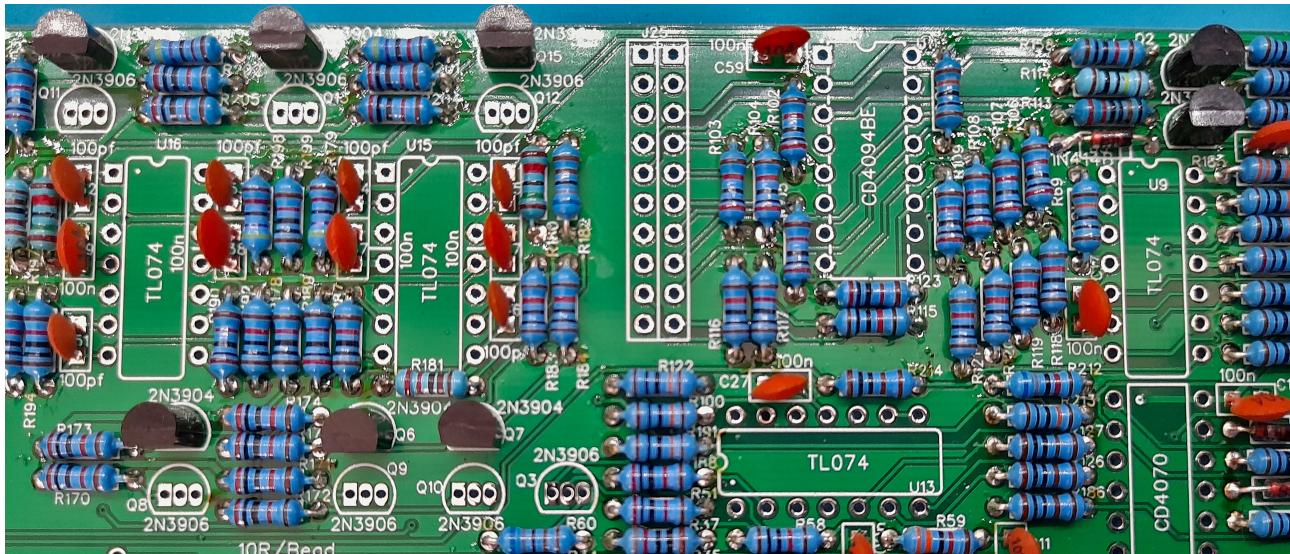
Transistors

Keep the 2N5457 for later, since it stands off further from the PCB. It is recommended to start with the 2N3904, then the 2N3906 due to the tight placement. Put them in place and make sure the orientation is correct. The outline of the transistor has to match with the silkscreen on the PCB. You can bend the 2 outer legs of each transistor, so they don't fall out when turning around the board like shown below.

Lay the PCB down on a flat surface to make sure, the transistors are flush to the PCB. The footprint for those is quite small, so stay focused and don't use too much solder to avoid shorts.

Check your solder joints with a magnifier and/or multimeter when in doubt.

8	2N3904	Q16, Q15, Q14, Q7, Q6, Q5, Q4, Q2
8	2N3906	Q3, Q8, Q9, Q10, Q11, Q12, Q13, Q17

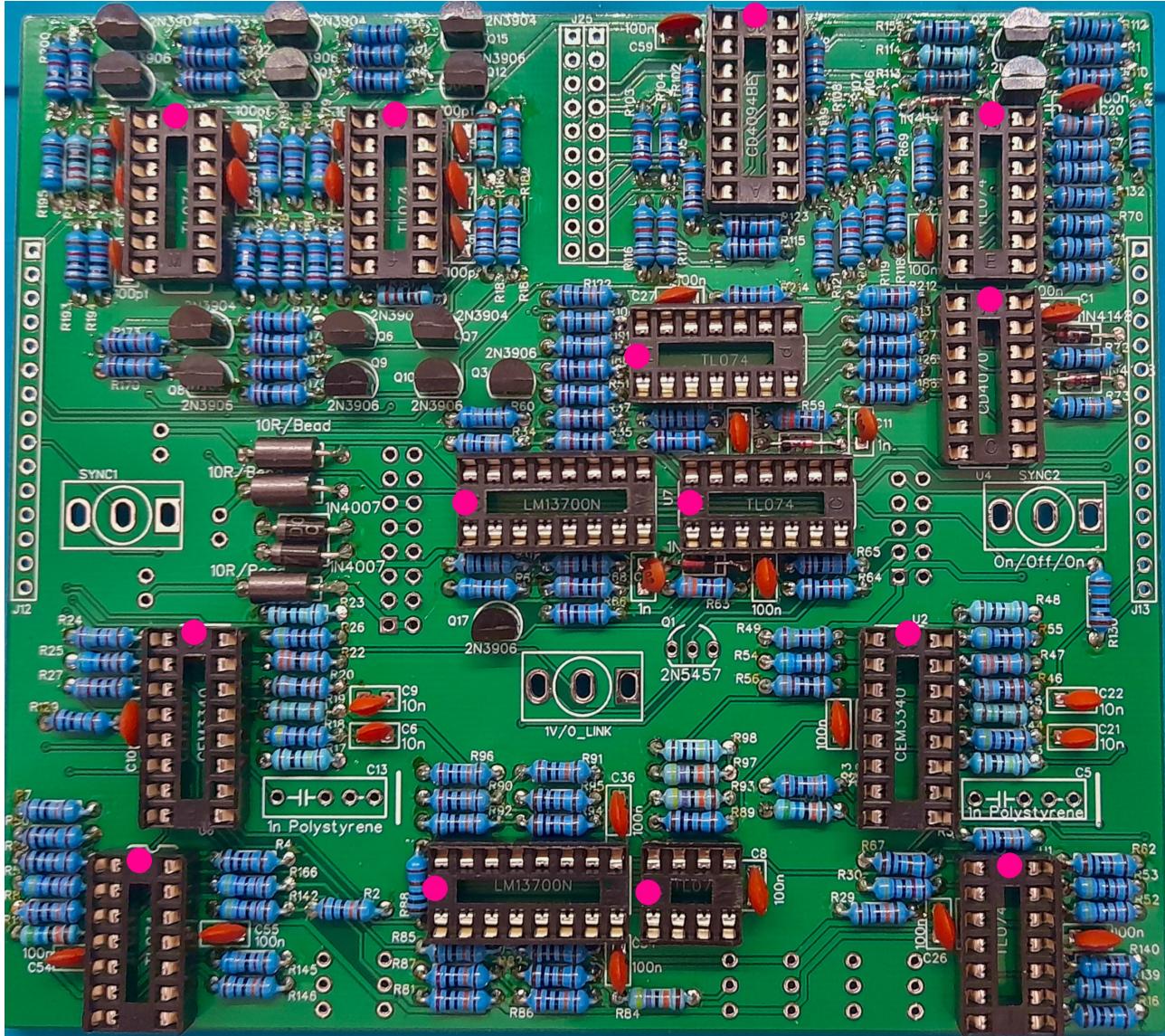


STEP 6 (Main PCB)

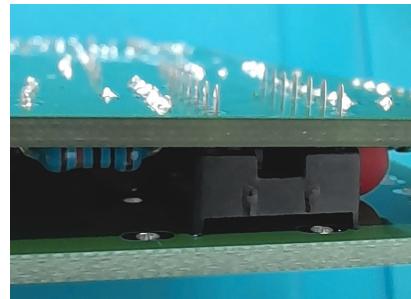
IC sockets

Look for the dip-8, dip-14 and dip-16 sockets.

Oriantation is important to know how to place the ICs later! The place for the notch is marked on the PCB as "D shape" or with a dot on pin 1. In the picture below it is marked with a pink dot. You can use tape to hold them in place, put the other PCB on top or bend 2 pins of each socket inwards, so they don't fall when turning the board around. Then solder everything.



In this image, you can see the method of using the "control PCB" to hold the IC sockets in place while turning around the board.



STEP 7 (Main PCB)

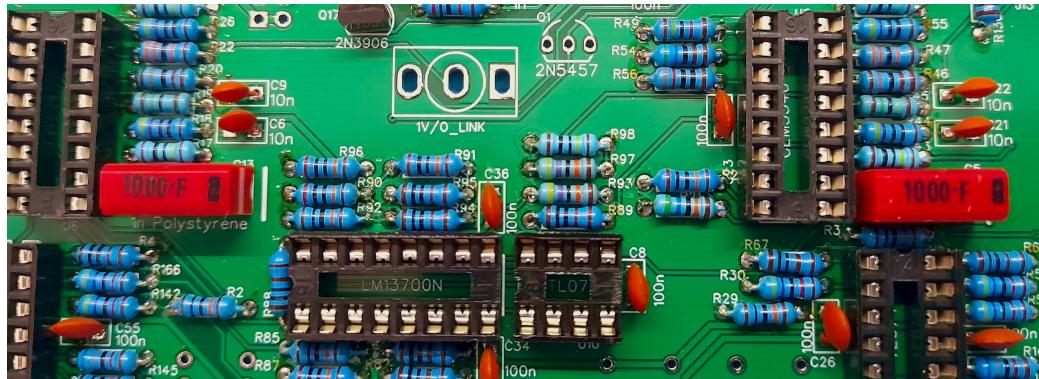
Polystyrene Capacitor

Place the Polystyrene (or other high quality) capacitors

If it has a polarisation, the marking on the capacitor should point towards the white line on the PCB.

A normal ceramic cap will work as well, but since it may affect the stability of the VCO cores, you should use high quality Plastic Film capacitors. Make sure, they are not taller than 10mm.

2	1n Polystyrene	C13,C5
---	----------------	--------



STEP 8 (Main PCB)

2N5457 Transistor

Place and solder the 2N5457 N-channel JFET.

Make sure the outline of the transistor matches with the silkscreen on the PCB.

1	2N5457	Q1
---	--------	----



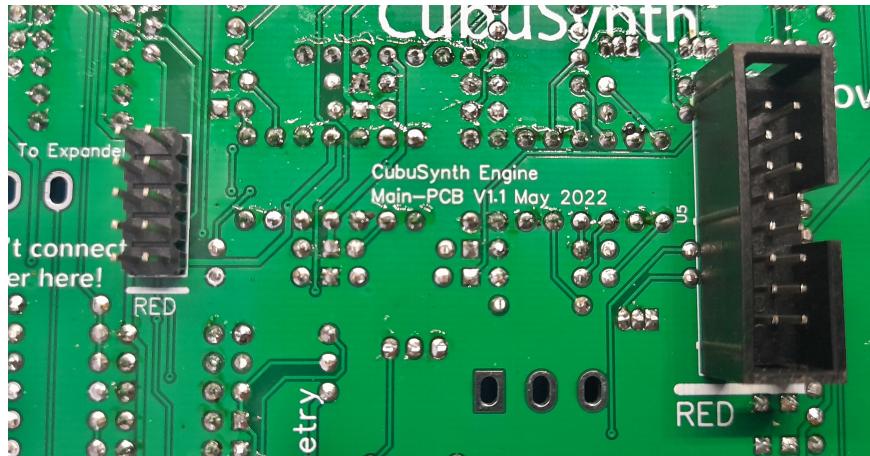
STEP 9 (Main PCB)

Pin headers

Turn around the board and place the Pin Headers for Power and the Expander like shown below. Orientation of the 16 Pin Power Header is very important to be able to insert the power cable the correct way.

Put some tape, to hold them in place while soldering from the other side.

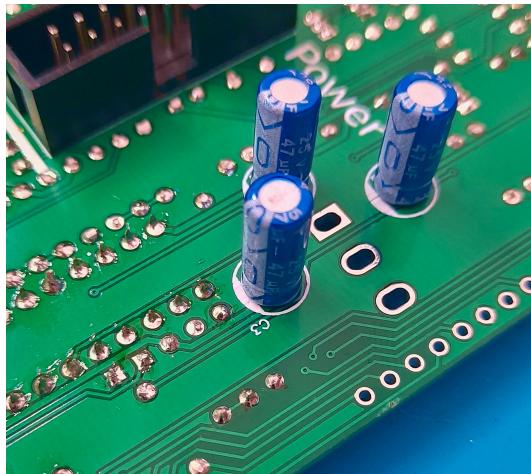
1	Eurorack Power 16 Pin IDC	U5
1	HDR-M-2.54_2x5	H3

**STEP 10 (Main PCB)**

Electrolytic capacitors

Place the 3 47uf capacitors as shown below. The white line on the capacitor has to face down towards the white mark on the PCB. The longer leg of the capacitor goes through the hole marked with "+"

3	47u	C43,C4,C3
---	-----	-----------



STEP 11 (Main PCB)

Trimmers

Place the 6 Trimmers according to the marking on the PCB, like shown in the picture below.

2	V/O trim	TRIM2_10K,TRIM4_10K
2	Hftrim	TRIM1_20K,TRIM3_20K
2	SineSymmetry	TRIM5_100K,TRIM6_100K

**STEP 12 (Main PCB)**Toggle Switches /
Pin headers

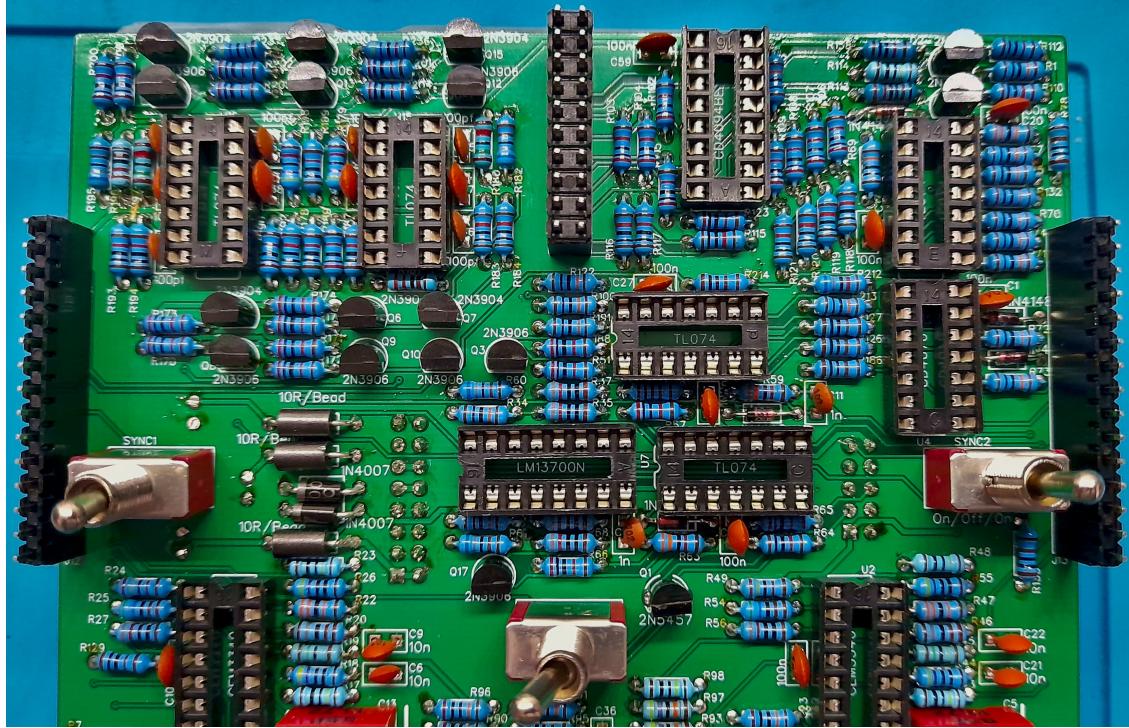
Place the toggle switches in the correct spots. The On/Off/On (3position) is the leftmost one.
Don't solder yet!

1	MINI TOGGLE On/Off/On	SYNC2
2	MINI TOGGLE On/On	SYNC1,1V/O_LINK

Take the 2x10 and 1x15 Pin Headers (Male) and Sockets (Female) and put them together. Place them on the Main PCB, with the female header down, but don't solder yet!

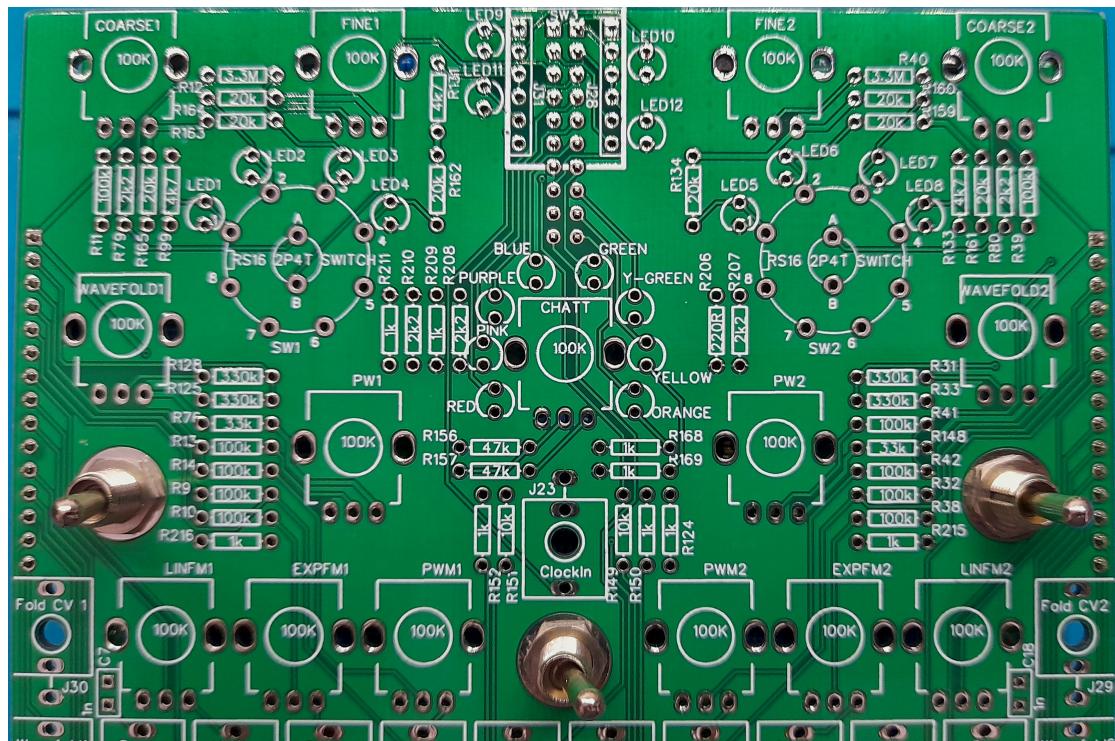
On the next page you can see the correct placement of the switches and the pin headers.

1	HDR-M-2.54_2x10	J25
2	HDR-M-2.54_1x15	J13,J12



STEP 13 Control PCB

Now take the other PCB and fit it on top, so the shafts of the switches go in their holes and all pins of the male headers are showing through. Fit the nuts to the switches and carefully turn the board around. Solder the Switches first, they will hold together the 2 PCBs while soldering the pin headers on both sides (both PCBs).



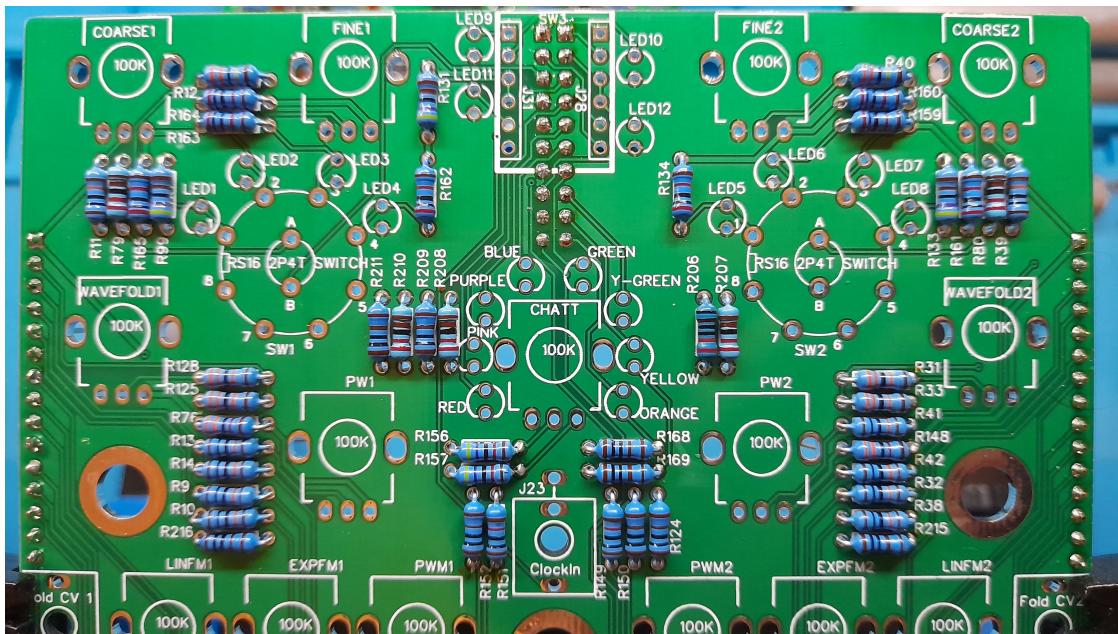
STEP 14 (control PCB)

Resistors

After removing the nuts on the switches and carefully taking apart the two PCBs, place and solder the resistors on the control PCB.

Here is the list of parts and a picture with all resistors in place.

1	220R	R206
9	1k	R215,R216,R124,R211,R209,R169,R168,R152,R150
5	2k2	R210,R208,R207
3	4k7	R131,R133,R99
2	10k	R151,R149
8	20k	R165,R164,R163,R162,R161,R160,R159,R134
2	33k	R148,R76
2	47k	R157,R156
10	100k	R42,R39,R38,R32,R14,R11,R10,R9,R41,R13
4	330k	R128,R125,R33,R31
2	3.3M	R40,R12



STEP 15 (control PCB)

1 nF capacitors

place the 1n capacitors, marked as "102" and solder in place.

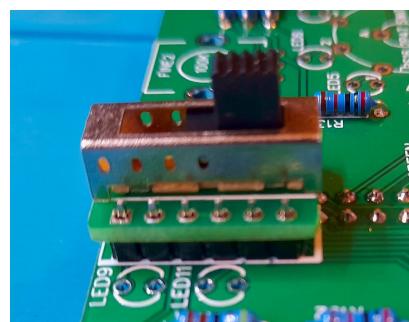
As shown below, the footprint also allows the placement of 0805 SMD capacitors. The DIY kits come with normal thru-hole components.

2	1n	C18,C7
---	----	--------

**STEP 16 (Switch PCB)**

Male pin headers

Place the 1x6 Male pin headers on the Control PCB, with the longer pins facing down. Then fit the Switch with its adapter PCB on top, like shown below. Solder from top (the side of the switch) but wait with soldering the other side.



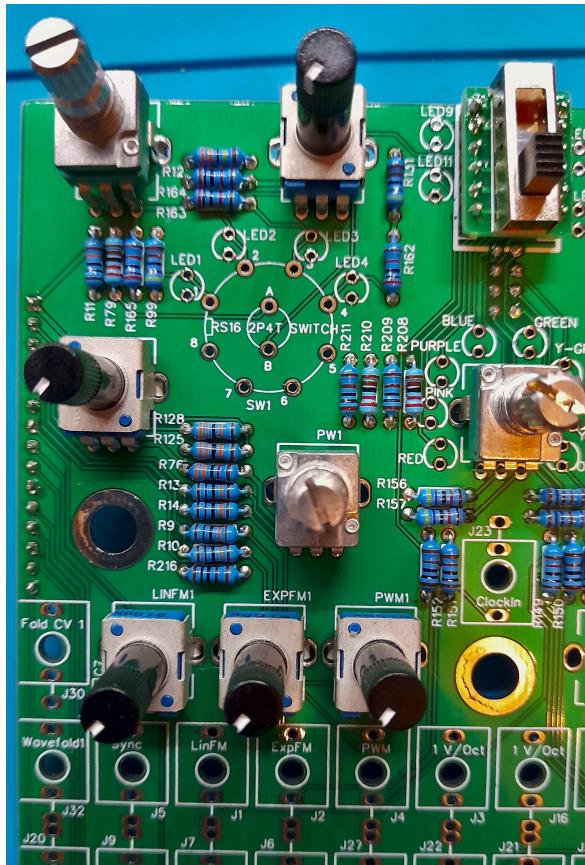
STEP 17 (control PCB)

Controls

Now take the Potentiometers, jacks, switches and the LEDs and put them in place as marked on the PCB but don't solder yet. The Potentiometers are for "Tune", "PW" and "Ch-Att" (Chaos attenuverter). The rest are Tall Trimmer Pots.

For the LEDs the orientation is important! The longer leg goes in the upper hole, while the flat side of the LED points down.

Below you can see the placement for the Potentiometers. (Mirror for the other side)



Before placing the rotary switches, cut the anti-rotation pin like shown below. The solder pins may be bent a bit due to transportation. Make sure they stand off straight when placing on the PCB.

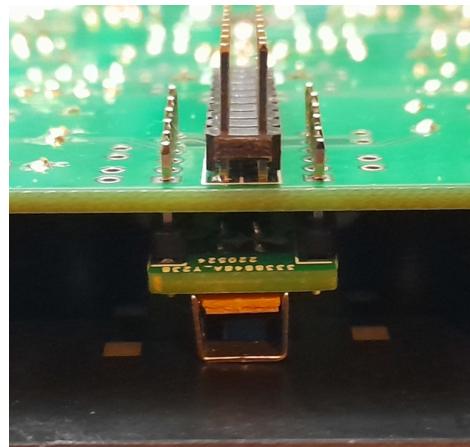


When everything is in place, take the front panel and put it on top, so all controls go into each corresponding hole of the panel. Since it is over 40 parts that need to fit, "wiggle" the front panel into place and make sure everything fits well.

Fit all nuts to the Potentiometers and Rotary switches. It is also recommended, to fit all nuts to the Jacks, since they lift up from the PCB a bit. You can also just slightly push them down, when the boards are upside down, so they lay flat on the front panel.

Be careful when turning the boards around, so that the components stay in place.

The slide switch should lay down on the front panel too, like shown in the picture below. Make sure the little handle is standing out from the front panel a bit.



Only solder 1 leg of each LED! It is quite difficult to solder them nice and straight. So after you soldered all Potentiometers, Switches, Jacks and 1 Leg of the LEDs, remove the front panel again and bend all LEDs, so they form a nice 90° degree angle to the PCB.

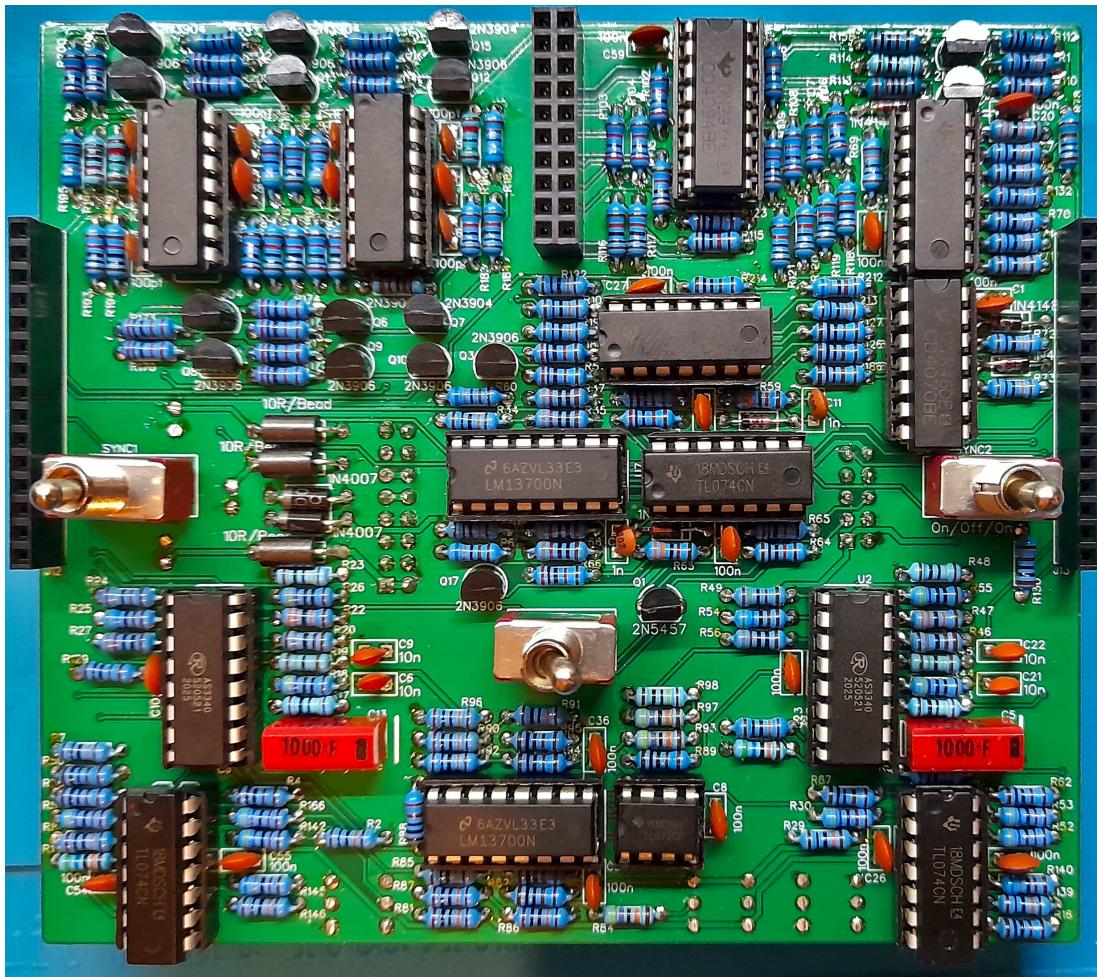
Then solder the remaining pins.



STEP 18

Installing ICs and final check

Take the Main PCB and install the ICs. Make sure to install the chips in their specific spot. Bend the Legs of the ICs inwards a bit so they form a 90° degree angle to the body of the chip. Make sure the orientation is correct, the notch has to match the silkscreen / the sockets placed before.



Check your soldering. If you see shorts or bad solder joints, fix it up. Then check the power pins for shorts with your multimeter.

When everything looks fine, you can fit the control PCB, so the switches go through their holes and all the Pin headers are connected properly. Then tighten the Nuts for the switches on top.

Then You can fit the Front panel on top again. Fit all the nuts on the Potentiometers, Rotary switches and Jacks.

Place the knobs on the Potentiometers. For best results, turn all Potentiometers counter-clockwise and push on the knobs, so the marking points towards the left end of the surrounding circle.

Step 19 - Trimming

Before your module is finished, you need to calibrate the trimmers on the back.

To do so, plug in the 16pin Eurorack Power cable correctly to your Busboard and the Module while your rack is powered OFF! Make sure orientation is correct. (Red stripe to the -12V side). Then turn on your case, and plug the VCO1 sine wave to the output of your rack. (speakers / headphones).

The Sine wave will not show an exact sine, it will be slightly triangle shaped. Adjust the "Sine Symmetry" so the wave looks the same on top and bottom ends, and swing around 0V. It may show values between -5V to +5V and -5,5V to +5,5V.

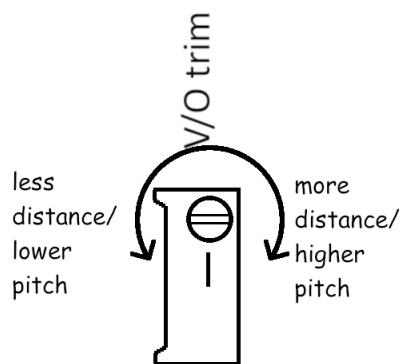
To adjust the V/oct tuning, plug a Keyboard/Sequencer or MIDI-interface, that produces 1V/oct signals to the 1V/O input of VCO1. Play the lowest C and tune VCO1 to play a C. Then press the C1 on your keyboard (one octave higher) and adjust the trimmer (without detuning on the Front) so the tuning is exactly note C, one octave higher. Repeat those steps until the VCO always plays a C when you play a C on the keyboard, and up to 10 octaves higher.

Repeat everything for VCO2.

If you prefer, you can also let a sequencer play through 4 octaves in 4 steps (C1, C2, C3, C4) and turn the trimmer, until the tuner always shows the same note in different octaves.

If you don't have a tuning device, but a frequency counter (on the oscilloscope), you can replace the notes with Hz. Tune down to exactly 100hz, the next octave should always double the frequency to 200hz, then 400hz, 800hz, 1600hz and so on.

The HF trim might need to be adjusted. If by playing in higher frequencies (higher octaves) the V/O tracking is not exact, turn up or down the trimmer, until the high octaves are also stable in pitch.



Congratulations, you can now install the module in your rack and have fun patching!

Please understand that we can't guarantee for your DIY module to work. Nevertheless we will do our best to guide you through the build or to solve problems when needed.

If you have trouble with your build, you can contact us at:

cubusynth@gmail.com

For better support, please include pictures of your soldered PCBs, front+back.

<https://cubusynth.com/>

<https://www.facebook.com/cubusynth>

https://www.instagram.com/cubusynth_modular/

<https://www.tindie.com/stores/cubusynth/>

<https://www.etsy.com/shop/CubuSynth>