

VCFA

Multimode VCF with LP,HP and BP outputs and integrated VCA

Build Guide

PCB V1.2 Sept 2022

Written and Illustrated by Ruben Sponar

Introduction

The CubuSynth VCFA is a multimode VCF with LP,HP and BP modes with integrated VCA, CV control over Frequency, Resonance and Volume. The CV input for the Frequency features 1V/Oct scaling over at least 2 octaves. So you can even create plucky melodies with the resonance in "ringing" mode.
The Gain control can reach a nice, soft overdrive.

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

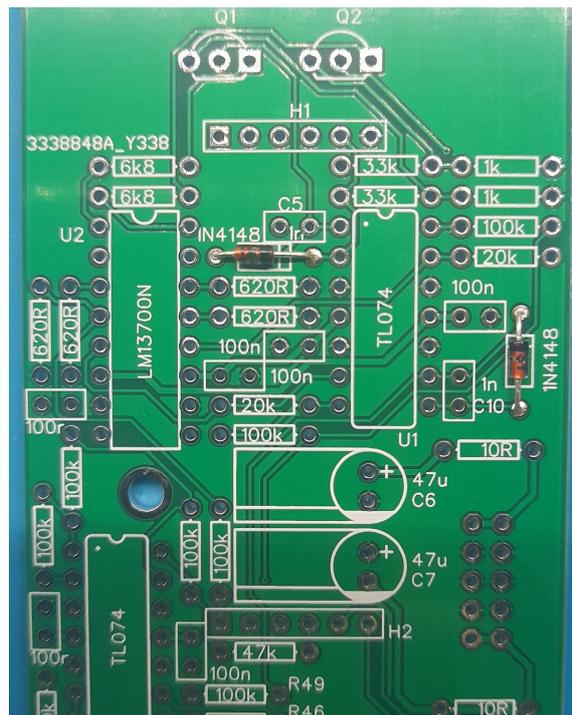
BUILD GUIDE

STEP 1 (Main PCB)

1N4148 Diodes

2	1N4148	D2,D1
---	--------	-------

Identify the 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.



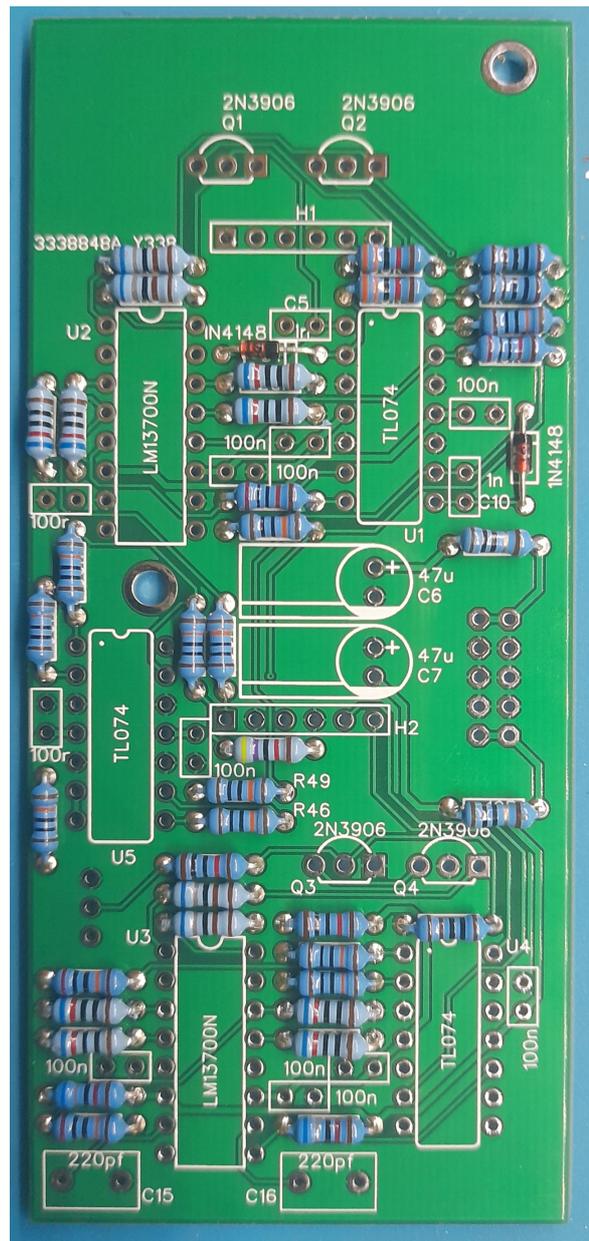
STEP 2 (Main PCB)

Resistors

2	10R	L2,L1
8	620R	R30,R29,R23,R20,R19,R11,R8,R7
3	1k	R43,R16,R1
4	6k8	R28,R27,R13,R12
2	10k	R44,R42
2	20k	R18,R9
2	33k	R15,R2
1	47k	R41
2	68k	R36,R32
11	100k	R49,R40,R39,R37,R26,R25,R21,R17,R10,R6,R46
2	200k	R31,R4

There is 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.

Here is 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.

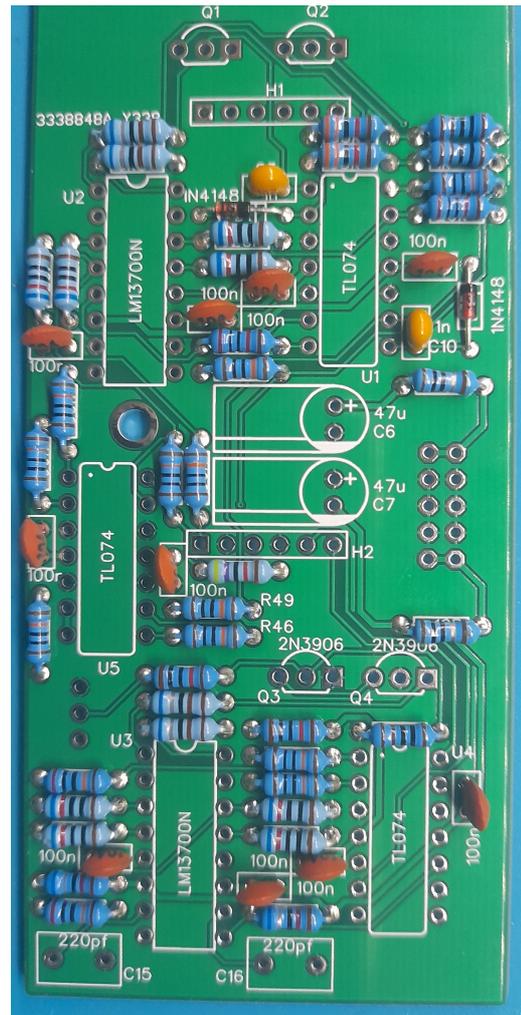
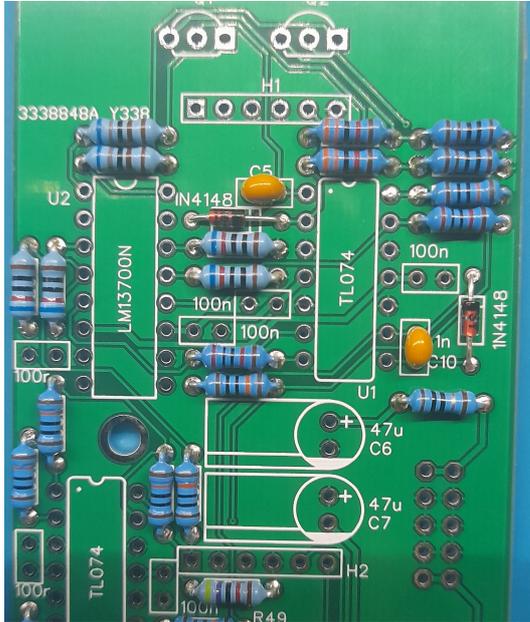


STEP 3 (Main PCB)
ceramic capacitors

2	1n	C10,C5
10	100n	C20,C19,C14,C13,C12,C11,C4,C3,C2,C1

Find the ceramic capacitors
1nf is marked as "102"
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 the placement of the 1n (here in yellow) and the 100n capacitors.



STEP 3 (Main PCB)
220pf capacitors

2	220pf	C16,C15
---	-------	---------

Place the 220pf capacitors. You can use any capacitor type with 220pf, but since those are the filtering caps, it is recommended to use high quality boxed capacitors. Orientation doesn't matter.

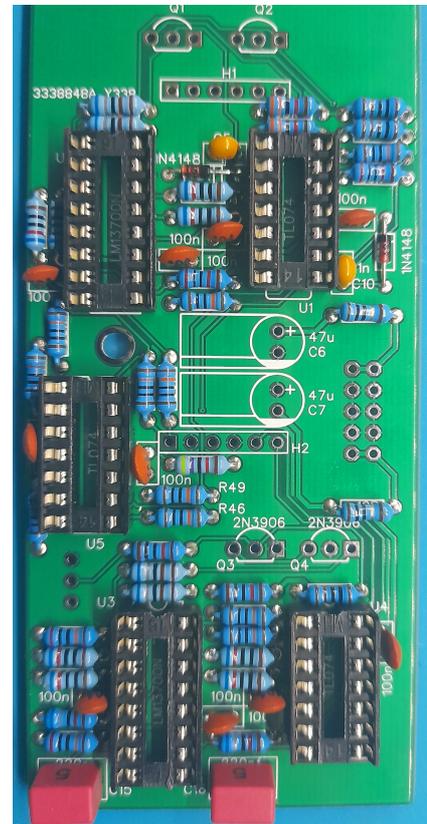


STEP 4 (Main PCB)

IC sockets

3	DIP14 socket	TL074
2	DIP16 socket	LM13700

Look for the dip-14 and dip-16 sockets.
 Orientation is important to know how to place the ICs later!
 The place for the notch is marked on the PCB with a "D shape"
 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.



STEP 5 (Main PCB)

Electrolytic capacitors

2	47u	C7,C6
---	-----	-------

Bend the legs of the two 47uf capacitors, so they form a 90° angle, as shown here.

Then place them on the PCB with the white line on the capacitor facing down towards the white mark on the PCB. The longer leg of the capacitor goes through the hole marked with "+"



STEP 6 (Main PCB)

Transistors

4	2N3906	Q4,Q3,Q2,Q1
---	--------	-------------

Take the 2N3906 transistors, 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 like shown below. Then turn around the board and solder.



STEP 7 (Main PCB)

Power header

1	EURORACK10	SV1
---	------------	-----

Turn around the board and place the Pin Header for Power like shown. Orientation of the Power Header is very important to be able to insert the power cable the correct way. Put some tape, to hold it in place while soldering from the other side.



STEP 8 (Main PCB)

Trimmer

1	CV Trim	TRIM1_100K
---	---------	------------

Place the 100k Trimmer according to the marking on the PCB, like shown in the picture.
 You can bend the legs outwards so it stays in place.

Turn around the board and solder.

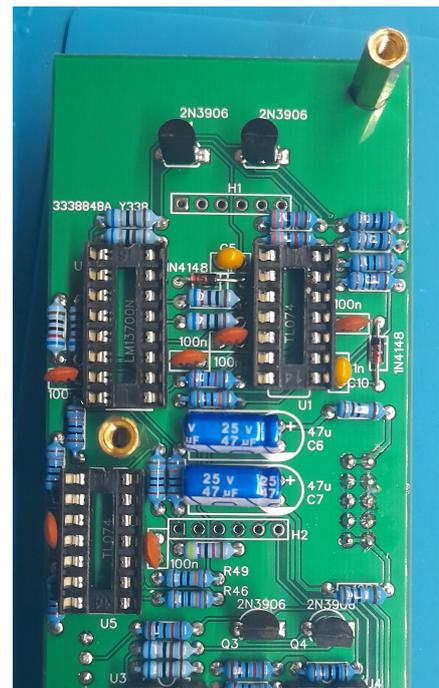
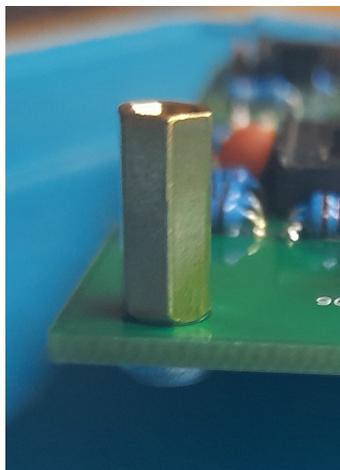


STEP 9 (Main PCB)

M3 Standoffs & Screws

2	M3 Spacer 11mm	M3 spacer 11mm height
4	M3 Screw	M3 screw, max length 6mm, Philips Head

Place the screws in the holes of the PCB so they stand out on the side with all components and screw them from the other side like in the pictures.

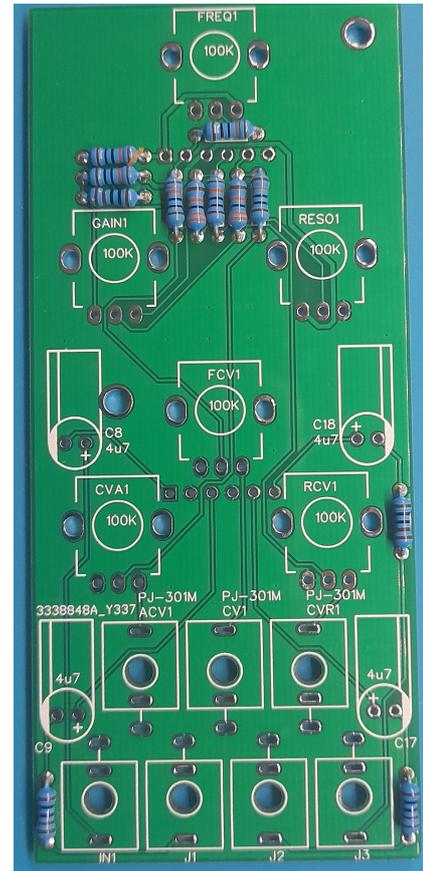


STEP 10 (control PCB)

Resistors

4	1k	R35,R34,R33,R47
6	100k	R45,R38,R22,R5,R48,R14
2	330k	R24,R3

Now take the other (control-) PCB, and place and solder the resistors.
Here is the list of parts and a picture with all resistors in place.



STEP 11 (Main & control PCB)

Pin headers

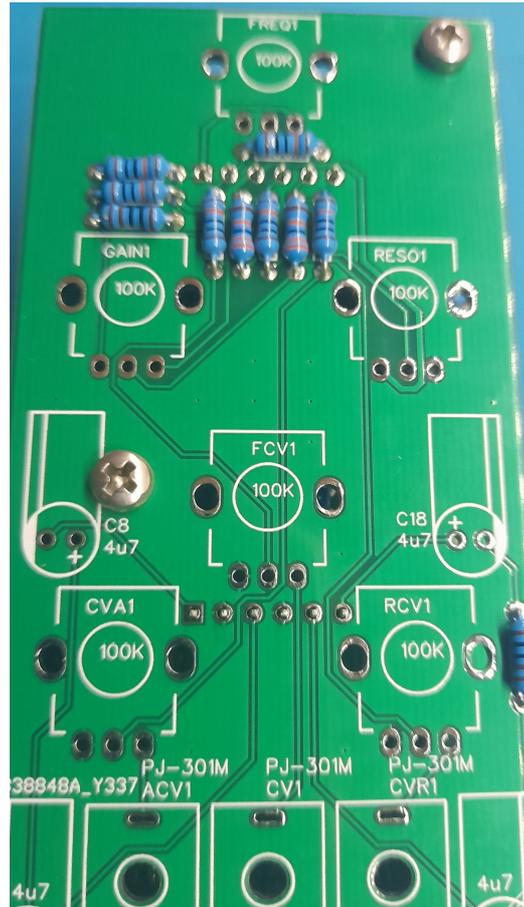
2	HDR-F-2.54_1x6	H2,H1
2	HDR-M-2.54_1x6	J9,J7

Take the 1x6 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!

Now take the other PCB and fit it on top, so the pins of the male headers are showing through. Screw the PCBs together with the M3 screws, while soldering the pin headers on both sides (both PCBs).

Then remove the screws from the Main-PCB. You can let the Standoffs fixed to the control-PCB.

The next page shows pictures of this step.

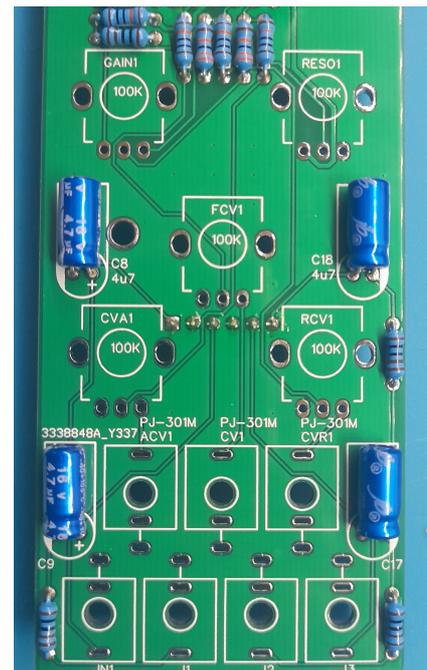


STEP 12 (control PCB)
4.7 uf capacitors

4	4u7	C18,C17,C9,C8
---	-----	---------------

bend the legs of the Capacitors so two of them have the white stripe on one side, and two on the other as shown here.

Place them on the PCB so the white stripe of each cap faces outwards from the middle. The longer leg of the capacitor goes through the hole marked with "+"



STEP 13 (control PCB)

Controls

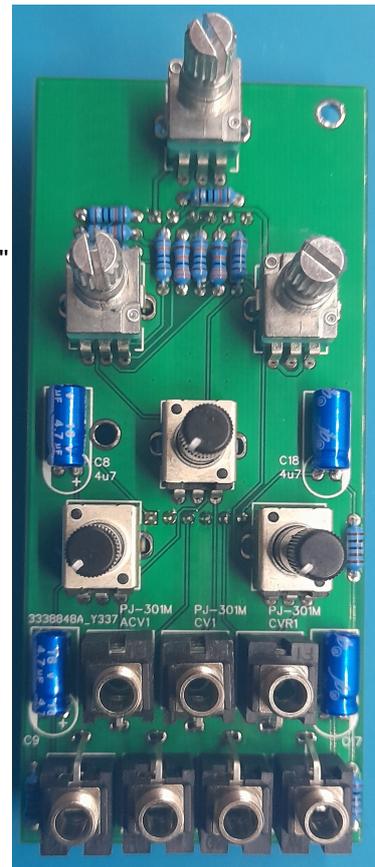
7	PJ-301M	IN1,CVR1,CV1,ACV1,J3,J2,J1
3	100K	FREQ,GAIN,RESO
3	100K	CVA,FCV,RCV,

Now take the Potentiometers and jacks and put them in place as marked on the PCB but don't solder yet.

In the picture you can see the placement.

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. "wiggle" the front panel into place and make sure everything fits well.

Fit all nuts to the Potentiometers and the Jacks.
 Be careful when turning the boards around, so that all the components stay in place.
 Then solder everything.



STEP 14

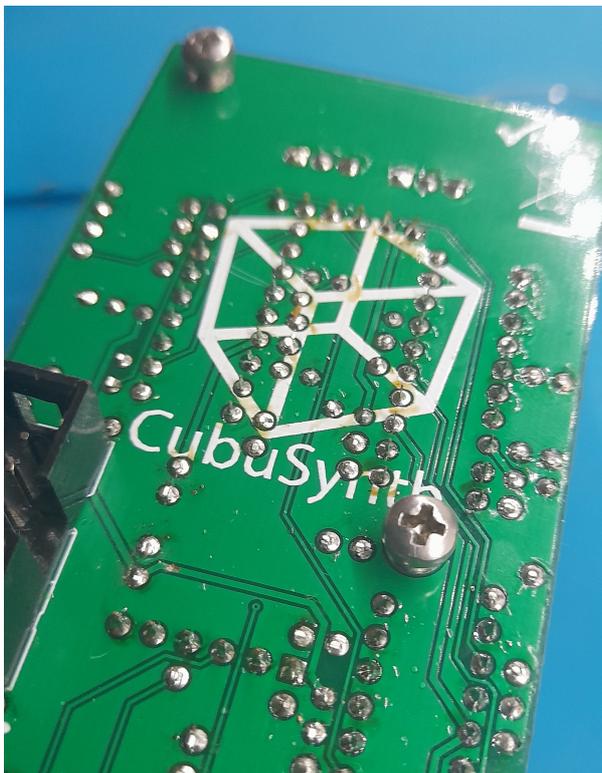
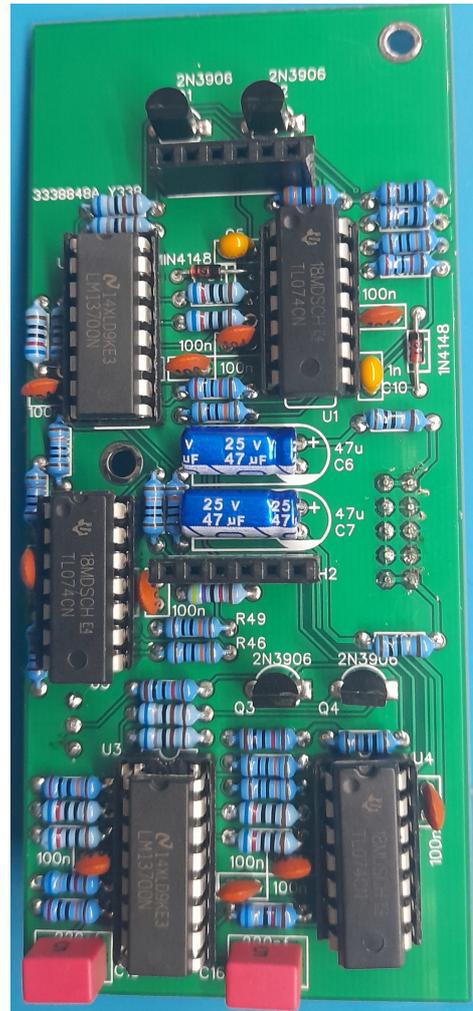
Installing ICs and final check

3	TL074	U1,U4,U5
2	LM13700N	U3,U2

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 Pin headers are connected properly. Then tighten the Screws on the Main PCB to the standoffs.



Step 15 - Knobs

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 16 - Trimming

Before your module is finished, you need to calibrate the trimmer on the back. Please note, that this is a filter and not a VCO, so 1V/oct tracking will not be the most precise. But it will be quite stable over at least 2 octaves.

First 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 LP to the output of your rack. (speakers / headphones).

To adjust the V/oct tuning, set the resonance to 100%, then plug a Keyboard/Sequencer or MIDI-interface, that produces 1V/oct signals to the Freq CV input (set the Freq CV knob to 100%) and the Gate to the Input of the VCFA. This allows the resonance to "ping" at given Frequency. Play the lowest C on the keyboard and tune the big Frequency 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 2 or 3 octaves higher.

If you prefer, you can also let a sequencer play through the octaves in 2-3 steps (C1, C2, C3...) 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 (e.g. oscilloscope), you can tune down to exactly 100hz, the next octave should always double the frequency to 200hz, then 400hz, 800hz, 1600hz and so on.

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

Help

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.

In case there should be parts missing in your Full DIY kits contact us to ask for replacement /

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

cubusynth@gmail.com

To be able to identify mistakes in your build, please include high quality 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>