Penrose Quantizer Assembly Guide

Schematic and BOM V 1.1 06.06.22

The schematic can be found here:
www.sonic-potions.com/public/PenroseQuantizerSchematic.pdf

https://www.tubeohm.com/penrose.html

The BOM is available at google docs:
Link to BOM

Prepare the PCB

First take the PCB and separate it using a pair of pliers.

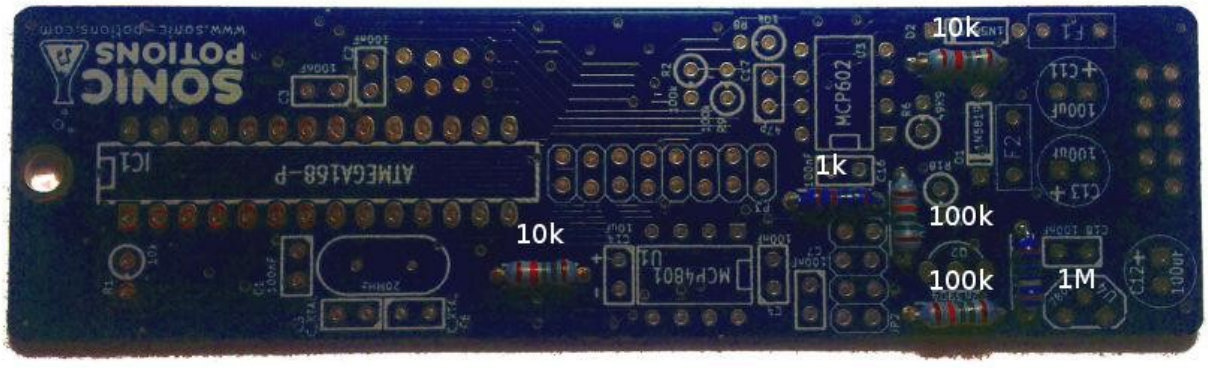
PCB 1

Now we start soldering the components to the PCB. We start with the lowest profile parts and work up to the highest.

Flat resistors

First we solder the resistors. The orientation does not matter.
The color rings can look quite similar in bad lighting. So it is advised to double check or measure them with your multimeter to be sure.

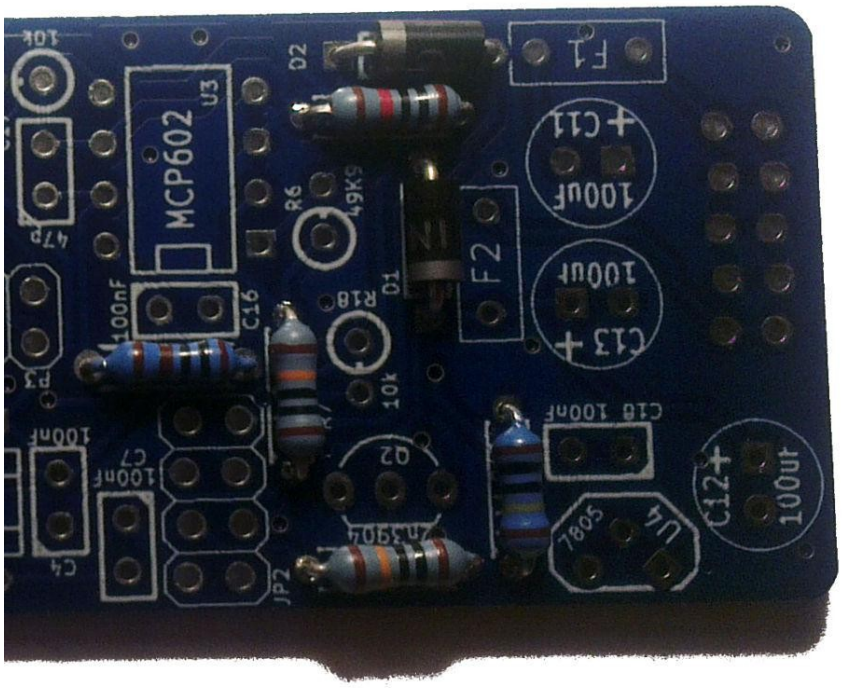
Image	Description	Quantity	Part No.	Notes
	10k	2	R10 R17	brown, black, black, red, brown
	1M	1	R19	brown, black, black, yellow, brown
	100k	2	R7 R20	brown, black, black, orange, brown
	1k	1	R21	brown, black, black, brown, brown



Diodes

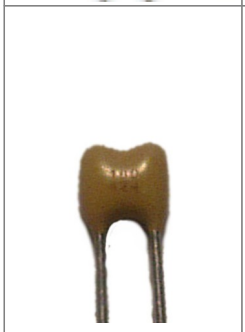
Diodes are polarized parts, so the orientation is important. You have to align the silver ring on the diode to the line on the silkscreen as seen on the picture.

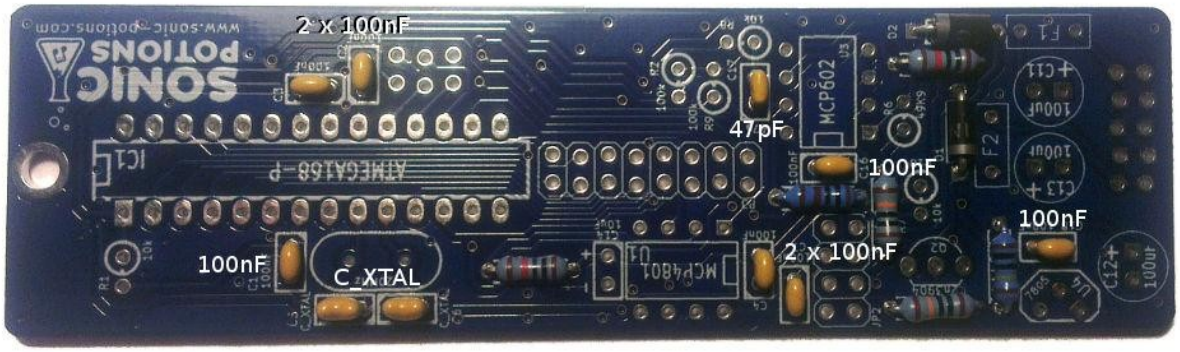
Image	Description	Quantity	Part No.	Notes
	1n5819 diode	2	D1 D2	



Ceramic capacitors

These capacitors have no polarity, so the orientation does not matter. The capacitors marked with C_XTAL on the PCB can have different values (18/22pF) depending on the used quarz .

Image	Description	Quantity	Part No.	Notes
	47pF ceramic capacitor	1	C17	has "47" written on it
	100nF ceramic capacitor	7	C1 C2 C3 C4 C7 C16 C18	has "104" written on it
	18 or 22pF ceramic capacitor	2	C5 C6	Has "180 or 220" written on it PCB is marked with C_XTAL. **Marked with a red dot using a felt tip pen**



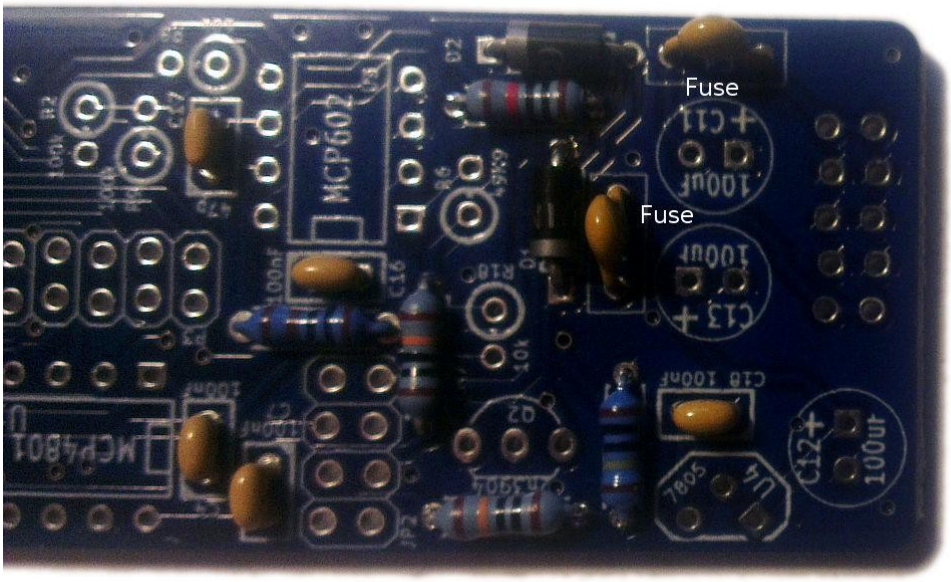
Fuses

The orientation does not matter.

Image	Description	Quantity	Part No.	Notes
	Polyfuse	2	F1 F2	

Cut of the legs of the fuses. This helps to get them into the PCB as the kink in the leg makes it hard to get them in straight.

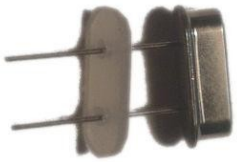
Then put them in the PCB and solder.

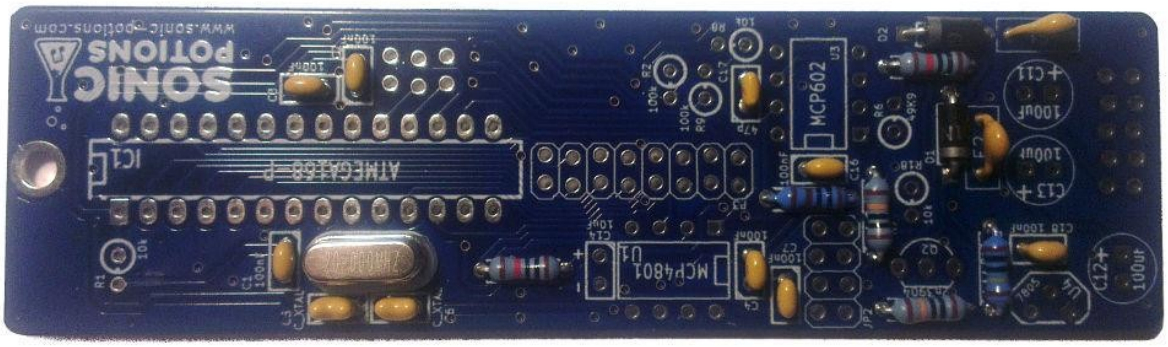


Quarz

Next is the quarz. Put the small plastic isolator disc under the quarz before soldering.

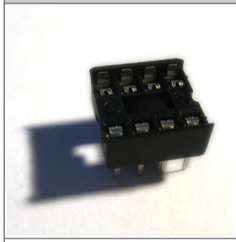
Image	Description	Quantity	Part No.	Notes
	20 MHz quarz	1	-	
	Isolator disc	1	-	Put the isolator below the quarz



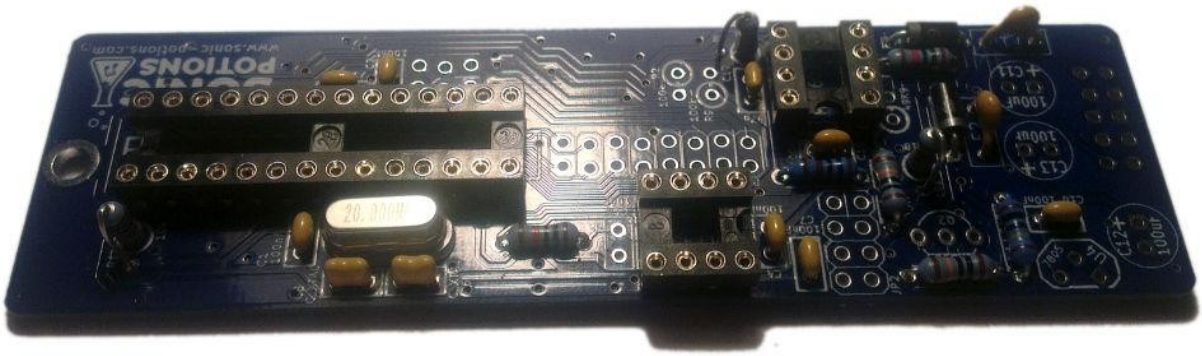


IC sockets

Align the notch of the IC sockets with the notch on the silkscreen. The notch indicates the IC orientation.

Image	Description	Quantity	Part No.	Notes
	DIP 8	2	U1 U3	
	DIP 28	1	IC1	

Please ignore the standing resistors on the following picture. They are added in the next step.

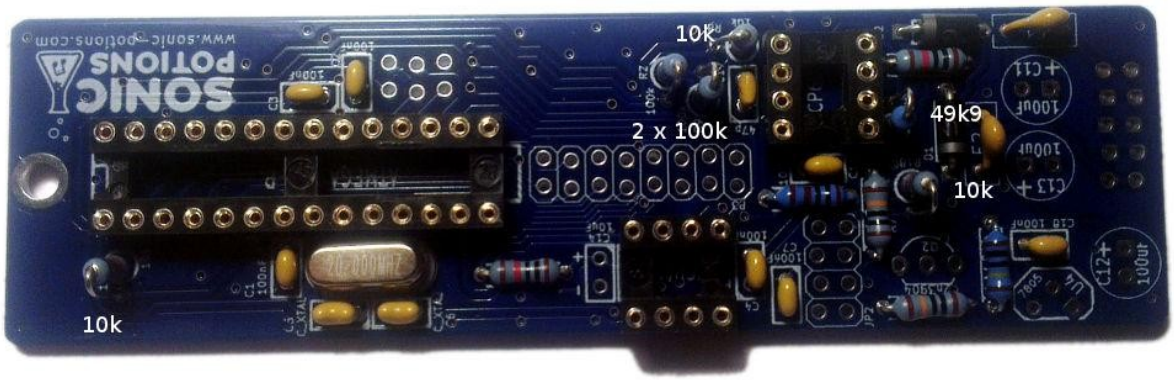


Standing resistors

Bend over one of the legs to make a U-shape. It does not matter which side since the resistor has no polarity. Leave the other leg straight. Then put the side with the resistor into the circle on the silkscreen.

Make sure the two 100k resistors have the right orientation. The two legs have to be left and right of the PCB, not up and down! otherwise your gate out won't work!

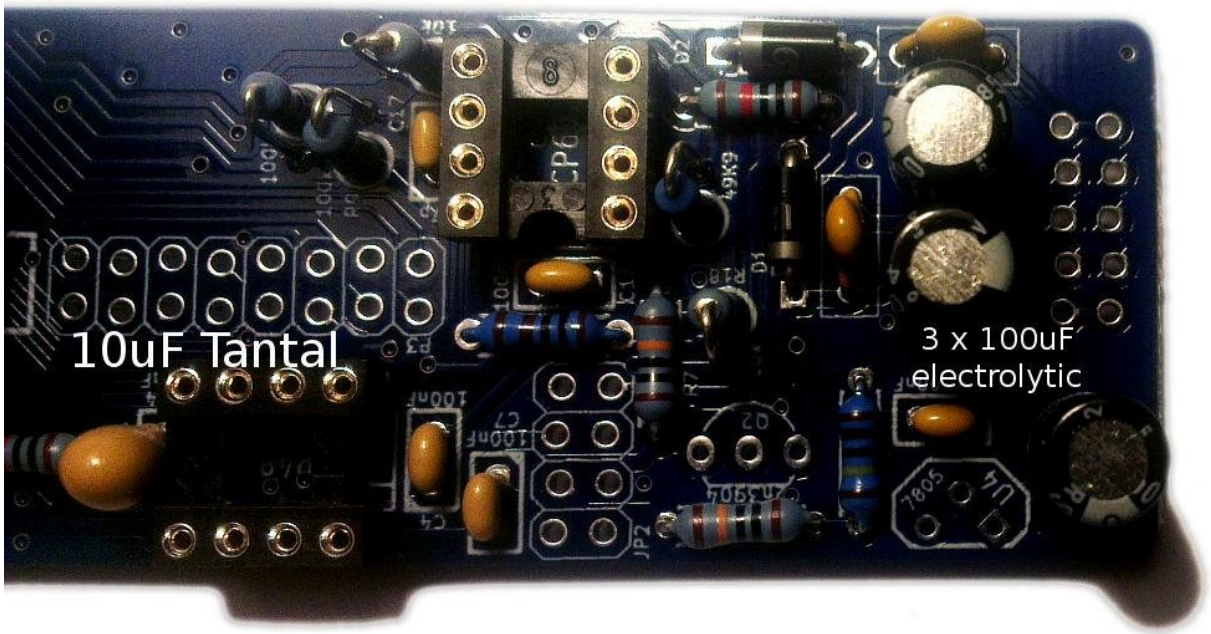
Image	Description	Quantity	Part No.	Notes
	10k	3	R1 R8 R18	brown, black, black, red, brown
	100k	2	R2 R9	brown, black, black, orange, brown
	49k9	1	R6	yellow, white, white, red, brown



Big capacitors

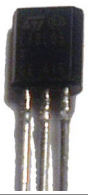
For the big capacitors the polarity is important! On both cap types the long leg is positive (+) and the short leg is negative (-). On the electrolytic capacitors the negative side is also marked with the big white stripe. The footprints on the silkscreen have a '+' sign on the side where the long leg has to be inserted.

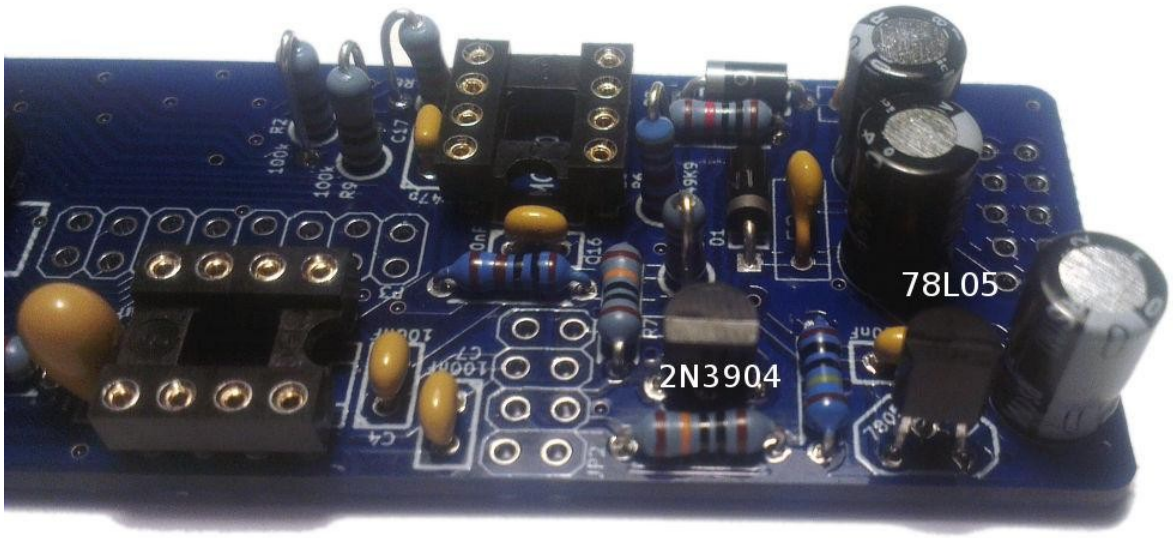
Image	Description	Quantity	Part No.	Notes
	10 uF tantal capacitor	1	C14	polarized part! Long leg is + Short leg is - has "106" written on it
	100uF electrolytic capacitor	3	C11 C12 C13	polarized part!



Transistor and Voltage Regulator

These are polarized parts. The flat side of the parts has to face the PCB edge. The flat side is also marked on the silkscreen.

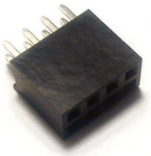
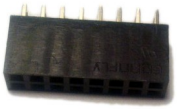
Image	Description	Quantity	Part No.	Notes
	78L05 voltage regulator	1	U4	
	2N3904 transistor	1	Q2	



Board and power connectors

Solder a single pin first. Then make sure the connector is completely flat on the PCB. If not you can simply reheat the single pin and re-adjust the component.

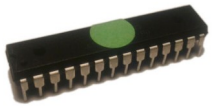
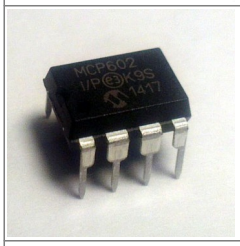
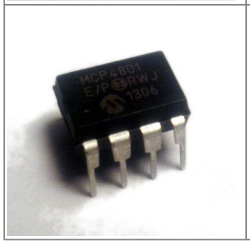
The female connectors are located on the top of the PCB, the male connectors on the bottom (according to the silkscreen)

Image	Description	Quantity	Part No.	Notes
	2x4 female	1	JP2	Board connector **top**
	2x8 female	1	P3	Board connector **top**
	2x5 male	1	P1	Power connector **bottom**
	2x3 male	1	AVR ISP	AVR ISP connector **bottom**

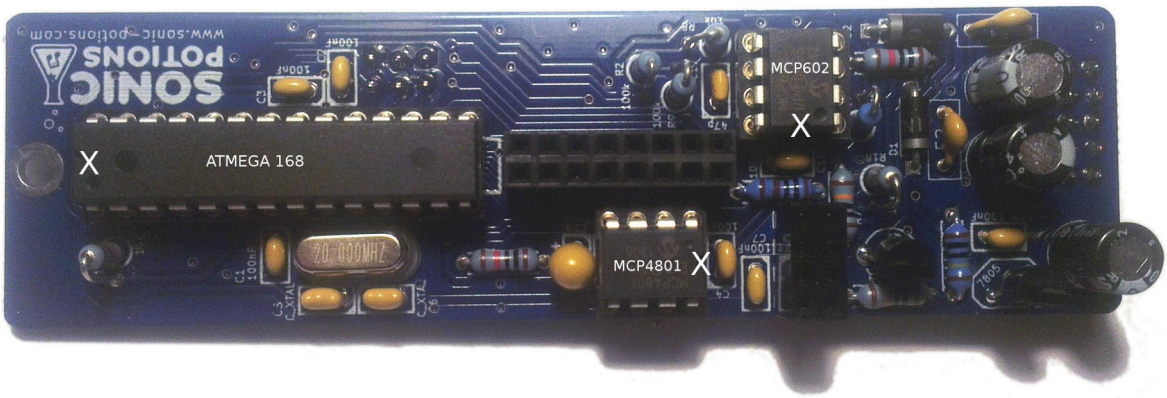
Here are the 2 female connectors on the top

And here are the 2 male connectors on the bottom:

Insert ICs

Image	Description	Quantity	Part No.	Notes
	programmed Atmega 168	1	IC1	microcontroller
	MCP602	1	U3	OpAmp
	MCP4801	1	U1	DAC

The orientation is important for the ICs. Align the notch on the ICs with the notch from the sockets/PCB. On the following picture the noth is marked with a white 'X' on the ICs. You may have to bend the legs a bit inwards to fit them into the sockets.



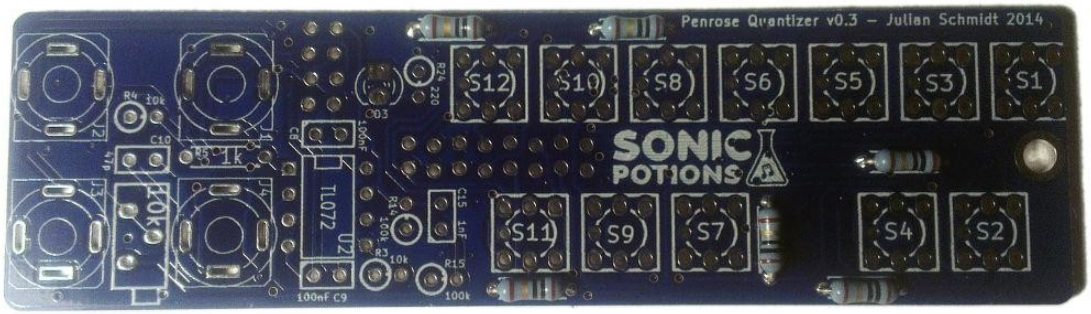
Congratulations! The first PCB of the kit is finished. Take a break, drink a coffee, then return with new energy to the other PCB.

PCB 2

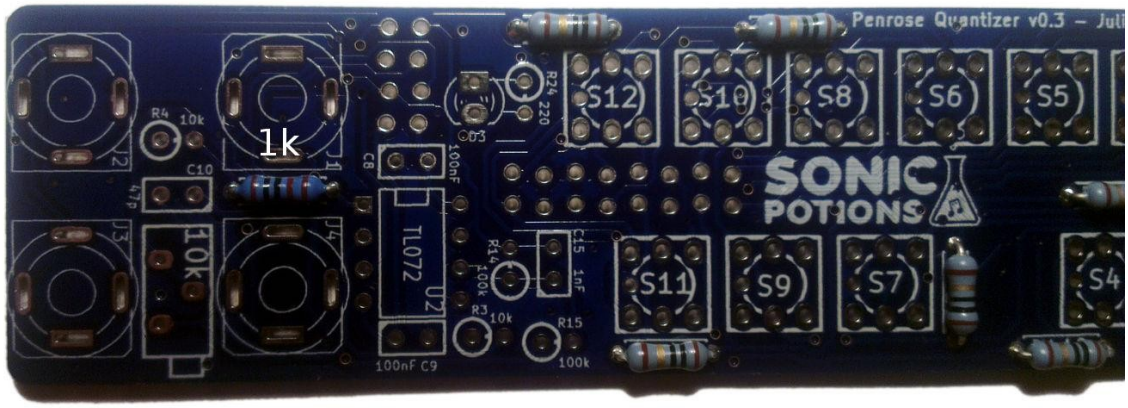
We follow the same scheme as with the first PCB and start with the low profile components.
Flat resistors
Non polarized parts. Orientation does not matter.

Image	Description	Quantity	Part No.	Notes
	10 Ohm	6	R11 R12 R13 R16 R22 R23	brown, black, black, gold, brown
	1k	1	R5	brown, black, black, brown, brown

First the six 10 ohm resistors



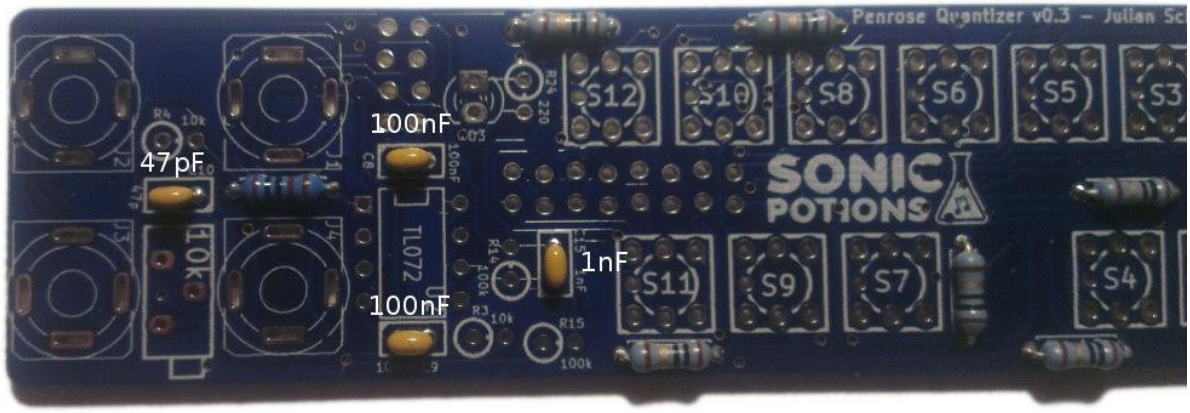
Then a single 1k resistor



Ceramic capacitors

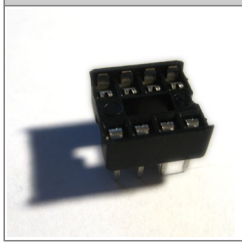
Next are the ceramic capacitors. The orientation does not matter here, too.

Image	Description	Quantity	Part No.	Notes
	47pF ceramic capacitor	1	C10	has "47" written on it
	100nF ceramic capacitor	2	C8 C9	has "104" written on it
	1nF ceramic capacitor	1	C15	has "102" written on it **Marked with a black dot using a felt tip pen**

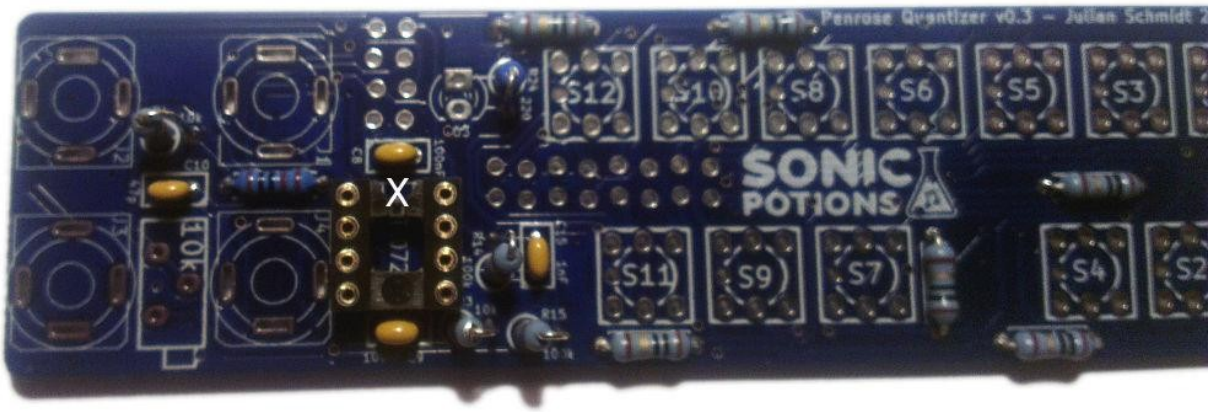


IC socket

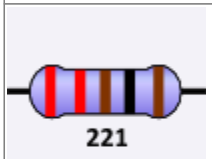
The notch in the socket is marked with a white 'X' and has to match the screenprint.

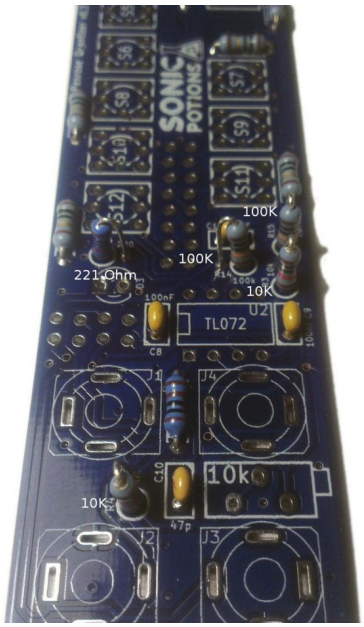
Image	Description	Quantity	Part No.	Notes
	DIP 8	1	U2	

Please ignore the standing resistors on the following picture. They are added in the next step.



Standing resistors

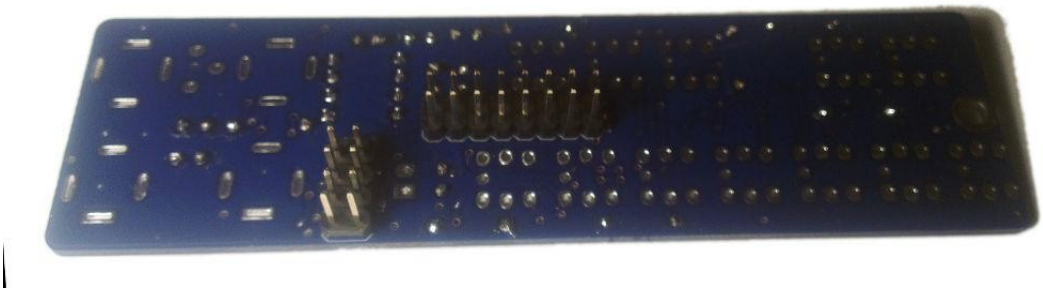
Image	Description	Quantity	Part No.	Notes
	10k	2	R3 R4	brown, black, black, red, brown
	100k	2	R14 R15	brown, black, black, orange, brown
	221 Ohm	1	R24	red, red, brown, black, brown



Pin array bottom

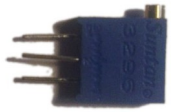
The bottom pin arrays have to be soldered now, otherwise the jacks and button are in the way and soldering becomes very tricky! Again, make sure you solder the components completely flush with the PCB. Solder one pin first, and check if the component is completely flat before soldering the rest of the pins.

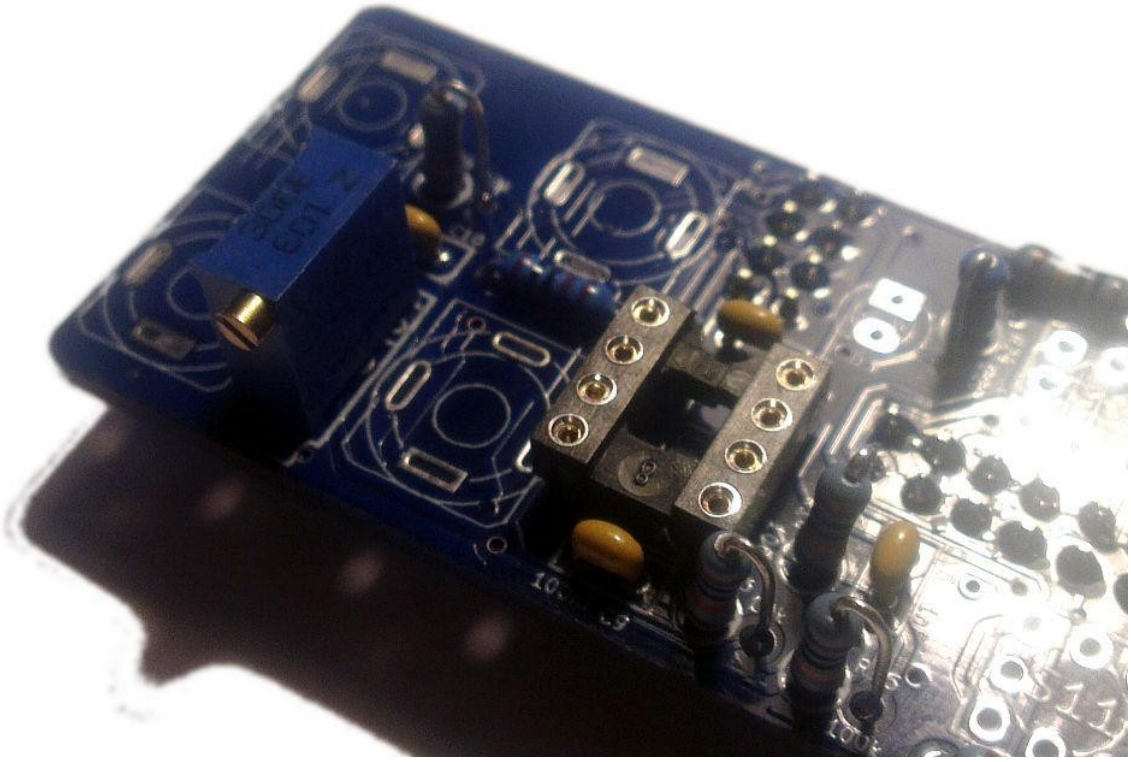
Image	Description	Quantity	Part No.	Notes
	2x4 pin array male	1	JP1	Bottom
	2x8 pin array male	1	P2	Bottom



Trimmer

The trimmer adjustment screw has to face the edge of the pcb.

Image	Description	Quantity	Part No.	Notes
	10k trimmer	1	-	used for tuning calibration

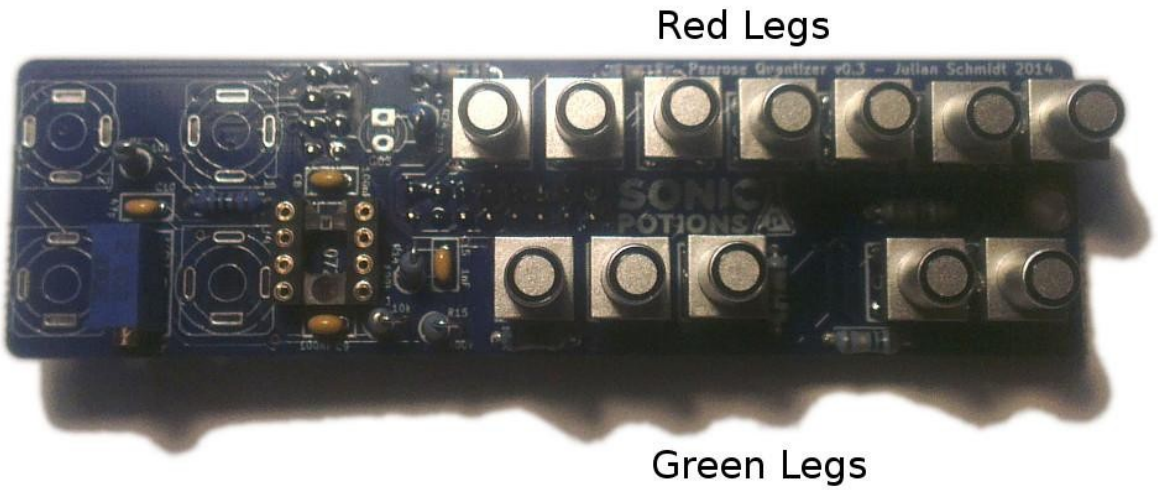


Buttons

The buttons have a red and a green marking on their sides. If you hold the module in front of you, the red leg has to be on the left side, the green one on the right side (on the picture the red leg has to be on the top and the green on the bottom). There is also a small(!) plastic pin on the bottom of the buttons that fits into the hole on the PCB for alignment.

Once again make sure you solder the buttons completely flush to the PCB. Soldering one pin first is advised.

Image	Description	Quantity	Part No.	Notes
	LED Button	12	S1 - S12	red leg left, green right



Insert IC

It's best to insert the IC now, otherwise it will be quite cramped with the jacks in place.

Orientation is important for the IC! Align the dot on the IC with the notch on the socket/PCB.

Image	Description	Quantity	Part No.	Notes
	TL072	1	U2	dual OpAmp

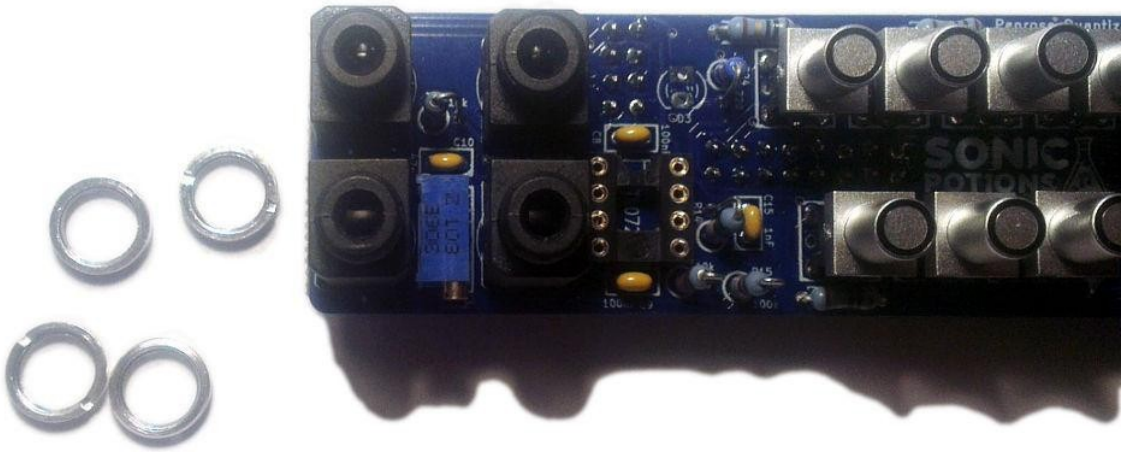
Attention!

For the following steps parts have to be placed on the PCB but **not soldered** before the frontpanel is attached. This is to make sure all components are properly aligned with the panel and fit through the holes.

Jacks

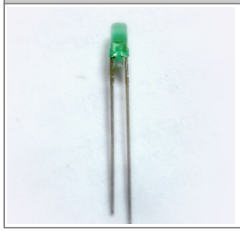
Remove the nuts and put the 4 audio jacks on the PCB. **Do not solder yet!**

Image	Description	Quantity	Part No.	Notes
	3.5mm Jacks	4	J1 - J4	



LED

The LED is a polarized part. The long leg has to be put in the square pad. **Do not solder yet!**

Image	Description	Quantity	Part No.	Notes
	Green LED	1	D3	Watch polarity! long leg in square pad.

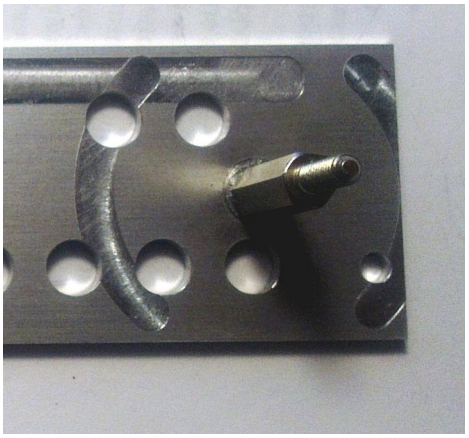
Panel

We are now going to attach the panel to the PCB. This way it is ensured that the jacks and the LED are aligned correctly with the panel before they are soldered in place.

It is advised to re-check your soldering job

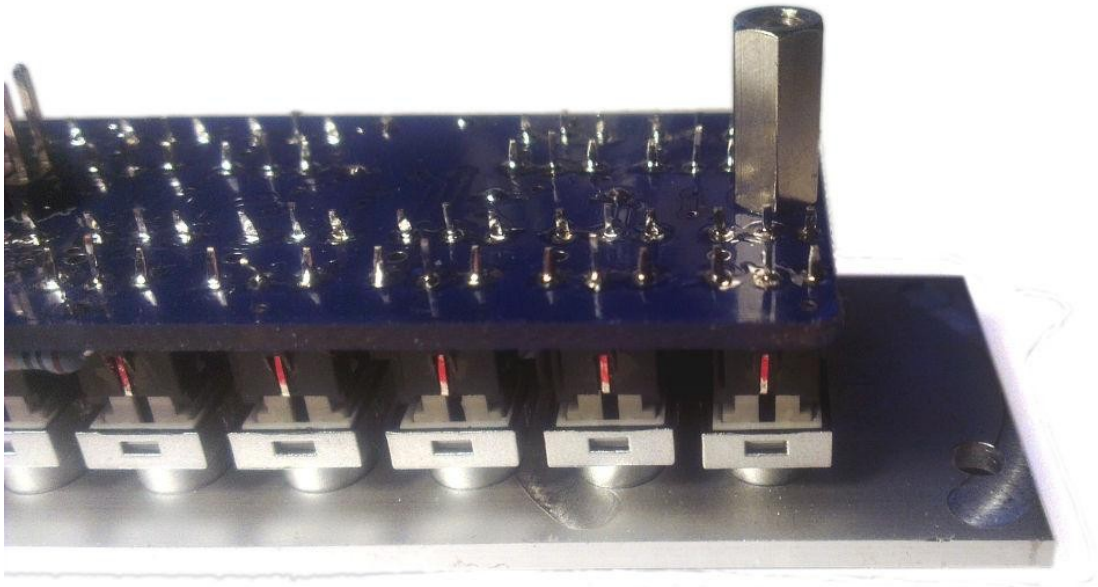
Image	Description	Quantity	Part No.	Notes
	11mm or 10 mm female/female **for 10 mm use 2 washer**	1		
	13mm or 12mm male/female **for 12 mm use one washer**	1		
	M3 screw	1		

- First screw the 12 or 13 mm male/female spacer onto the threaded pin on the back of the panel,
- If there is only a 12 mm spacer, use a washer to get the correct height.
- then attach the panel to the PCB.

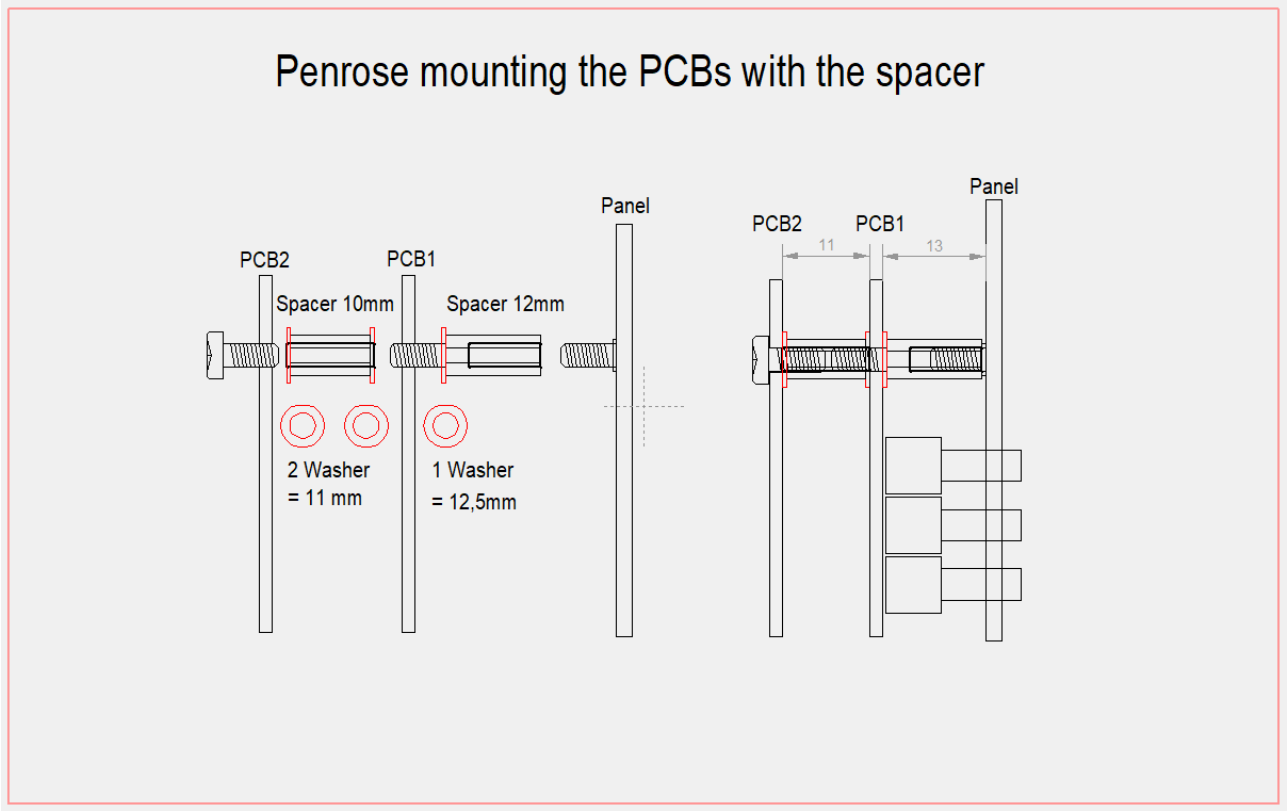


- secure the jacks on the panel using the nuts.

- Screw the 10/11mm female/female spacer below the PCB
- If you have only a 10 mm spacer , use 2 washer to get the correct height. For 11 mm spacer you don’t need the washer



If 10 mm and 12 mm spacer are in the kit, mount the pcbs like in the picture above



Solder the components

Now that the panel is attached it's time to solder the loose components.

- Move the LED into the hole of the panel, making it flush with the front, then solder it in place.
- Cut of the protruding legs of the jacks as flat as possible or they will touch the capacitors on the other PCB.

Final assembly

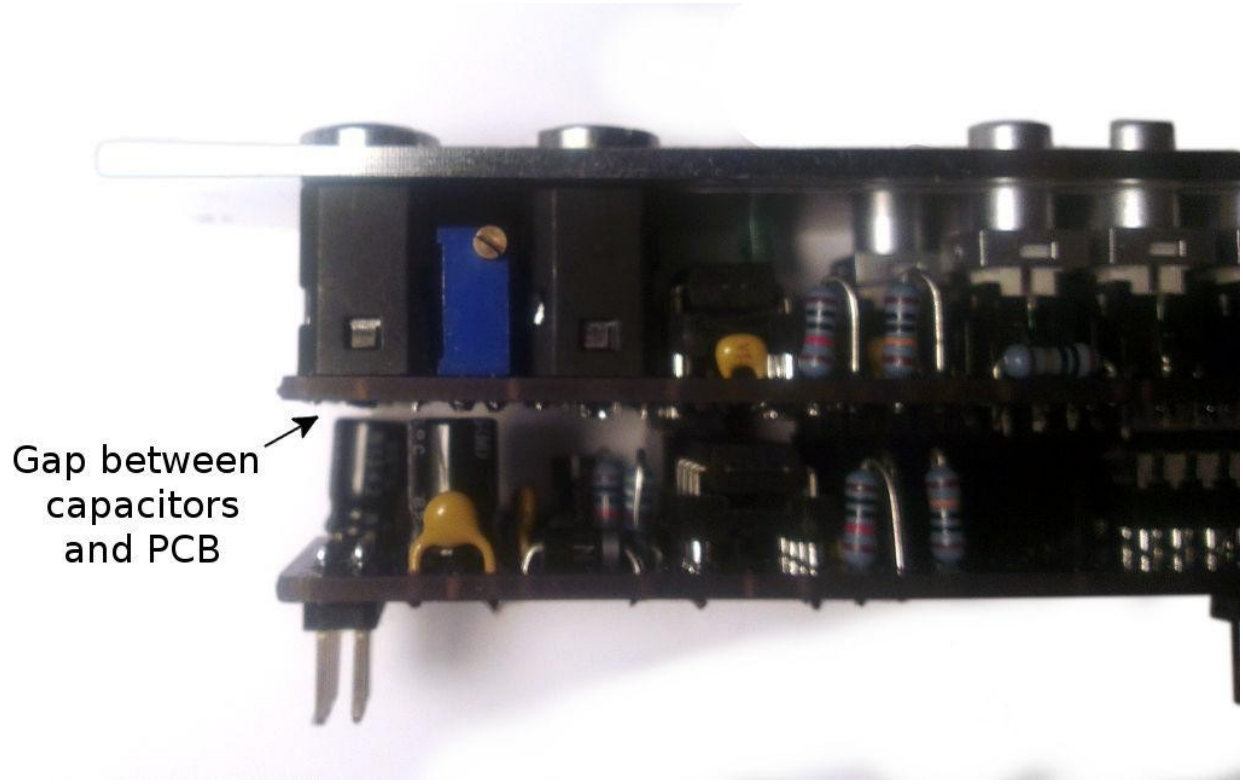
First check your work for errors again:

- Are all ICs in the proper orientation?
- Do all the solder joints look good?
- Is the diode and capacitor orientation correct?

To be sure you could also measure continuity between the gnd, +12V and -12V pins on the power connector using your multimeter. You should not measure continuity here or you have a short somewhere.

If everything looks good you can put the 2 PCBs together and secure the bottom one with the M3 screw.

Make sure the 3 electrolytic capacitors do not touch the upper PCB!!!



Test the module

Connect the module to your power supply. Make sure the polarity is correct. The red stripe is the -12V side and marked on the PCB. The button LEDs should light up.

Before you can use your module, you have to calibrate it for the 1V/Oct scaling. This is described in the next step.

Calibrating the module

The module has to be calibrated to match the 1V/Oct specs. There are several ways to calibrate it. I recommend the frequency counter method.

General instructions

The trimmer on the PCB is used to calibrate the spacing between the different notes.

- Turning it clockwise decreases, the spacing between the steps.
- Turning it counter clockwise increases the spacing between the steps.
- You have to set the trimmer so that each step corresponds to 1/12 Volt.
- This yields in a semitone per step

You can think of the tuning procedure as if the single output notes are drawn on a rubber band that is fixed on the left side (0V, lowest note). If you turn the trimmer counter clockwise you stretch the rubber band, making the gaps between the single notes bigger. If you turn it clock wise you relax the rubber band and the single notes slide closer together.

Calibration using a frequency counter

You need a VCO and a tool to measure frequency. You can either use a scope or your PC or mobile phone. Guitar tuner apps/VSTs that show the frequency are quite handy an can be found for free.

The way to achieve a proper tuning is as follows:

1. connect the CV out jack to an oscillator

2. connect a variable voltage source to the CV in. Nothing special is needed here. You can use a sequencer, midi2cv interface or just a CV source with a single knob. You just have to provide some kind of adjustable voltage.
3. deactivate all steps on the quantizer but one. This way all output CVs should be one octave (1V) apart when the trimmer is set correctly
4. Now connect the output of the oscillator to your frequency measuring device
5. tune the oscillator to 1000Hz
6. Increase your voltage to the CV input jack until the oscillator frequency makes a jump to the next octave. Now it will probably be either too high (i.e. 2100Hz) or too low (i.e. 1900Hz).
7. Adjust the trimmer until the frequency reads exactly double the frequency. It should read 2000Hz.
8. Lower your CV again. The oscillator will jump down an octave. Now it probably wont be exactly at 1000Hz again.
9. Use the oscillator fine tune knob (**NOT the trimmer on the quantizer!**) to readjust the frequency to 1000Hz
10. repeat steps 6 to 9 until the 2 octaves are exactly at 1000Hz and 2000Hz
11. To improve accuracy you can switch to higher octaves, too. For each octave the frequency should double

You sucessfully calibrated your module.

Here is a youtube video showing the calibration using a small pocket scope:

https://www.youtube.com/watch?v=qvvRYRC2Jt8

Calibration using a multimeter

You can use a multimeter to measure the voltage between the notes. This method is not as accurate as the frequency measuring method.

It is nearly the same procedure as the frequency counter method, but instead of an oscillator and a frequency counter you just connect the CV out to your multimeter and measure the output voltage. An octave should be exactly 1V apart.

1. connect the CV out jack to your multimeter measuring the voltage.
2. connect a variable voltage source to the CV in. Nothing special is needed here. You can use a sequencer, midi2cv interface or just a CV source with a single knob. You just have to provide some kind of adjustable voltage.

3. deactivate all steps on the quantizer but one. This way all output CVs should be one octave (1V) apart when the trimmer is set correctly
4. take note of the current output voltage (for example you may measure 2.13V here. Your measure may be different)
5. Increase your voltage to the CV input jack until the output voltage makes a jump to the next octave. The voltage should now have increased 1V (in our example 2.13V -> 3.13V) but it will probably be either too high (i.e. 3.51V) or too low (i.e. 3V).
6. Adjust the trimmer until the Voltage reads exactly 1V more than the initial value.
7. Lower your CV again. The voltage will jump down an octave. Now it probably wont be exactly at your initial value again.
8. take note of the current output voltage again.
9. repeat steps 5 to 8 until the 2 octaves are exactly 1V apart

You sucessfully calibrated your module.

Firmware Updates

The firmware can be updated with an audio file.

Update procedure

- Connect your audio player to the trigger input jack
- Hold the first note select button down and turn the power on
- The red LED of the button will start to flash red quite fast, indicating the module is now in firmware update mode.
- Start the audio playback of the firmware file
- The LED should switch its color to green and continue to flash
- If the LED lights up red and stops flashing an error occured -> see troubleshooting.
- Once the update is finished the quantizer will start the new firmware automatically

Troubleshooting

A constant red LED indicates an error. Most of the time this just means your audio level is too low.

Some phones and audio players have too weak outputs. In this case you may use a mixer or amplifier to boost the signal strength.

You can push the first note select button again to reset the module in firmware update mode and try again without cycling the power.