

BASTL INSTRUMENTS

NOISE² v1.3 - Assembly Guide

bastl-instruments.com



INTRODUCTION

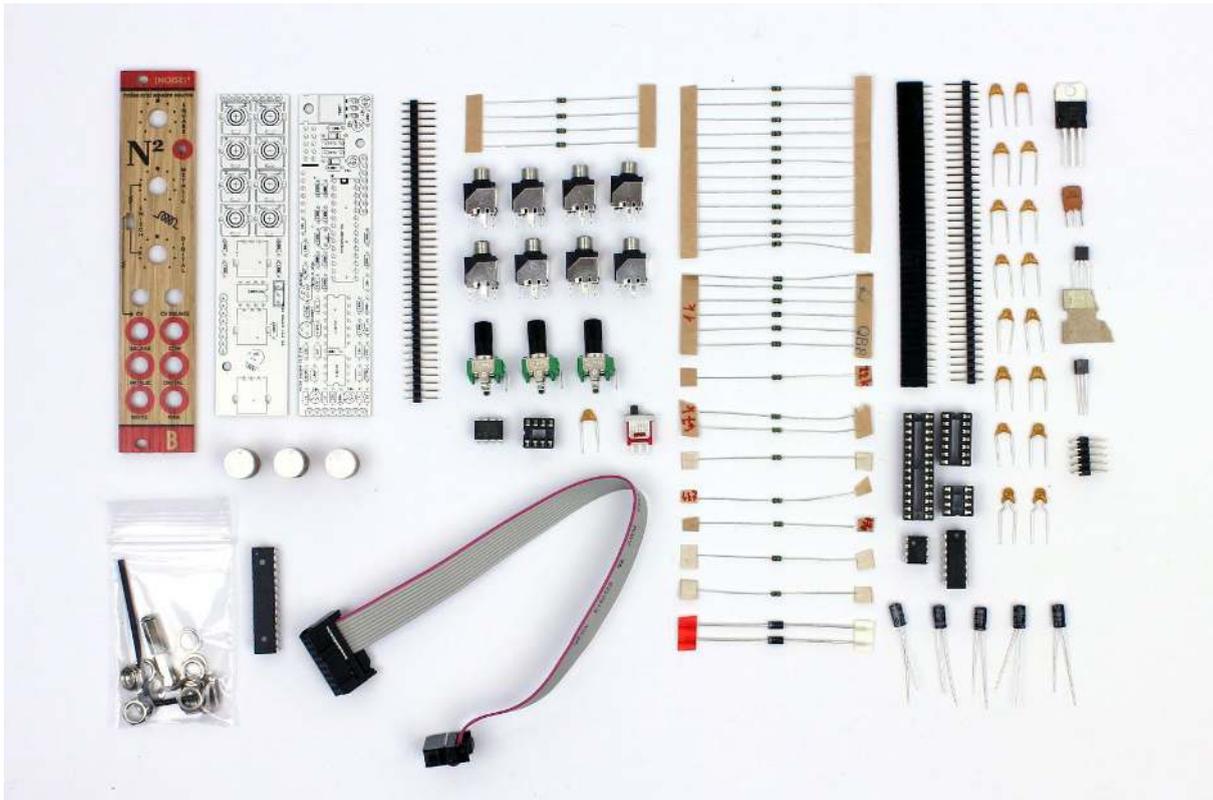
This guide is for building NOISE² module by BASTL INSTRUMENTS. It is good to have basic soldering skills and to be able to identify electronic components before starting this kit. However if you have never soldered before, check out this [tutorial first](https://cdn-learn.adafruit.com/downloads/pdf/adafruit-guide-excellent-soldering.pdf)¹. We even included some of the best quality solder to help you solder everything faster and better.

The Noise² module consists of two printed circuit boards (PCBs). All the parts comes mainly in three bags separated for Bottom board, Top board and Assembly parts. See Bill of Materials (BOM) for detailed list.

¹ <https://cdn-learn.adafruit.com/downloads/pdf/adafruit-guide-excellent-soldering.pdf>

NOISE SQUARE v1.3 BILL OF MATERIALS		
SOLDERING_TOP_NOISE 1.3		
qty	value	part
4	100k	R-EU_0204/5
1	CK 100nF	ceramic capacitor
8	jack connector	PJ-301BMB
3	POT LIN B100k	linear potentiometer
1	16pin	male pinheader
1	switch 2 P	switch
1	8 pin DIL	DIL socket - in foam
1	MCP6002	IC in foam
SOLDERING_BOTTOM_NOISE 1.3		
6	1k	R-EU_0204/5
1	4k7	R-EU_0204/5
1	22k	R-EU_0204/5
1	33k	R-EU_0204/5
2	47k	R-EU_0204/5
12	100k	R-EU_0204/5
1	270k	R-EU_0204/5
1	470k	R-EU_0204/5
1	820k	R-EU_0204/5
2	1N4007	DIODE-D-7.5
1	CK 10pF	ceramic capacitor
1	CK 1nF	ceramic capacitor
3	CK 10nF	ceramic capacitor
7	CK 100nF	ceramic capacitor
1	CK 470nF	ceramic capacitor
1	CK 1uF	ceramic capacitor
5	CE 10uF	electrolytic capacitor
1	2N3904	NPN
1	BC546A	NPN
1	7805	voltage regulator
1	20MHz	resonator
2	100mA	fuse
1		female header 8 + 8 pin
1		male header 6 pin
1		double male header 2x5 pin

1	8 pin DIL	DIL socket - in foam
1	14 pin DIL	DIL socket - in foam
1	28pin DIL	DIL socket - in foam
1	TL72	IC in foam
1	TL74	IC in foam
ASSEMBLING_NOISE 1.3		
qty	value	part
1	11 mm	spacer nut x nut
1	11,5 mm	spacer nut x screw
2	M3 x 8 mm	panel screw cross
3	M3 x 6 mm	screw
1		small nut
8		jack washers
8		jack nuts
1		allen key
1	TOP	PCB
1	BOTTOM	PCB
3		knob
1	Atmega328-PU	Atmega
1		power cable 10-16
1		front panel



Before starting this kit, prepare the following tools:

- Soldering iron
- Multi-meter
- Flush cutters
- n2. hex screwdriver or allen key (enclosed with kit)
- Phillips screwdriver (cross)
- Isopropyl alcohol + smaller and clean brush (optional)
- Wrench No. 8
- Protective eyewear

We suggest that you work in a clean and a well lit and ventilated environment to avoid accidents or losing any of the small components.

Also briefly go through this guide and make sure that you understand all the steps.

BOTTOM BOARD SOLDERING

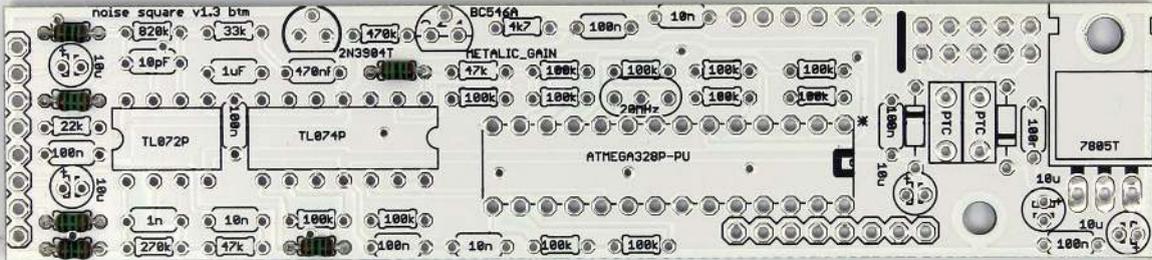
RESISTORS

Let's start with the bottom board. Before you start soldering, take your time and find all the **resistor values** [using a multimeter](https://learn.sparkfun.com/tutorials/how-to-use-a-multimeter/measuring-resistance)² (or you can check the color codes if you are seasoned enough).

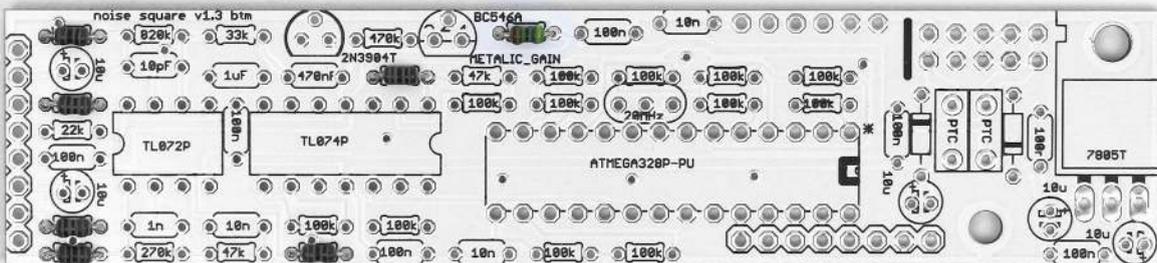
Now insert and solder 26 **resistors** (6x **1k**, 1x **4k7**, 1x **22k**, 1x **33k**, 2x **47k**, 12x **100k**, 1x **270k**, 1x **470k**, 1x **820k**). Be careful to insert these **resistors** on the right place (rectangular with an appropriate value) and solder them. Then snip the overhanging leads (be sure to make this step on all remaining leads in the course of this guide).

² <https://learn.sparkfun.com/tutorials/how-to-use-a-multimeter/measuring-resistance>

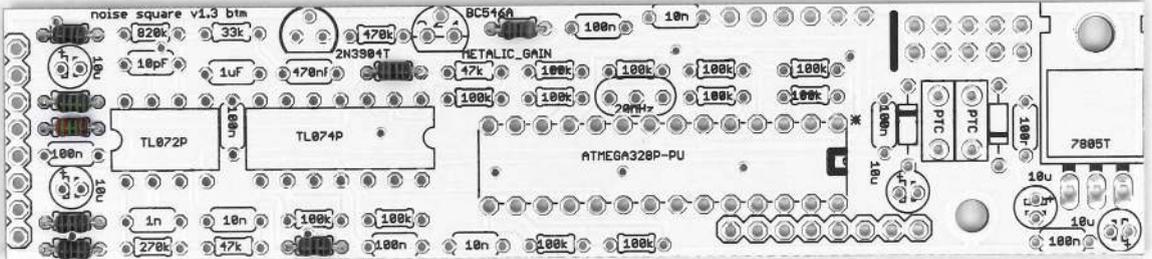
resistors 1k (6x)



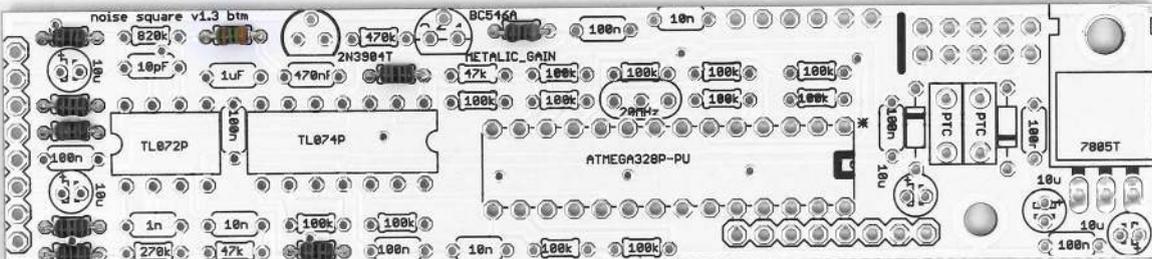
resistor 4k7 (1x)



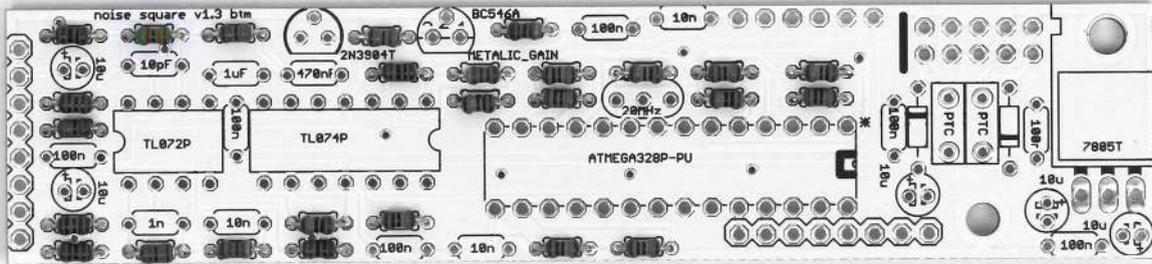
resistor 22k (1x)



resistor 33k (1x)



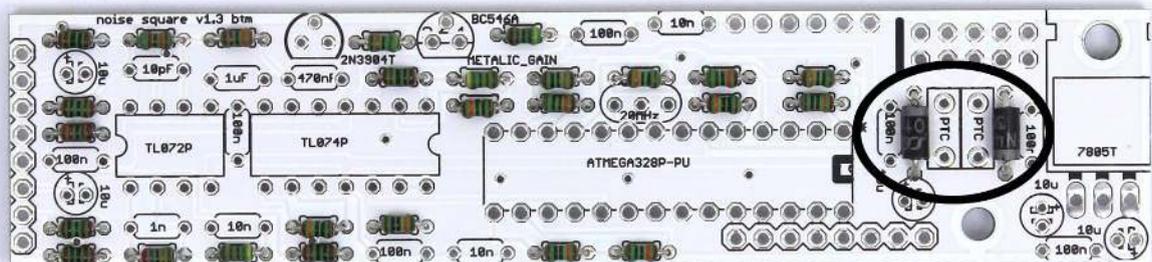
resistor 820k (1x)



DIODES

After that solder the two diodes: 2x 1N4007. Be careful, diodes are polarized! Make sure that the stripe on the diode body matches the stripe on the PCB. Check the photo below.

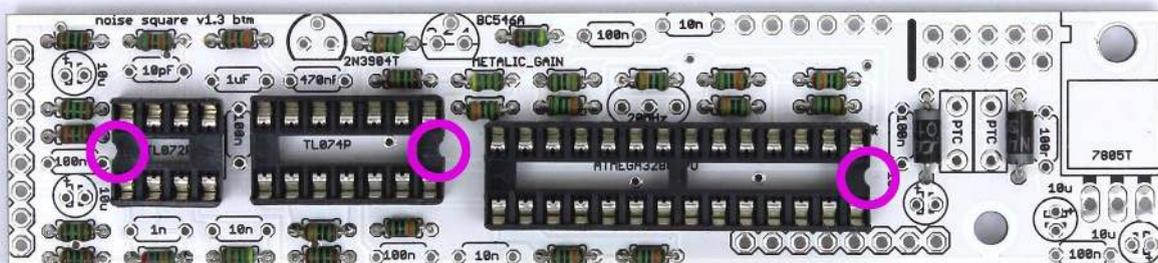
diodes 1N4007 (2x) watch out for the stripe!



IC SOCKETS

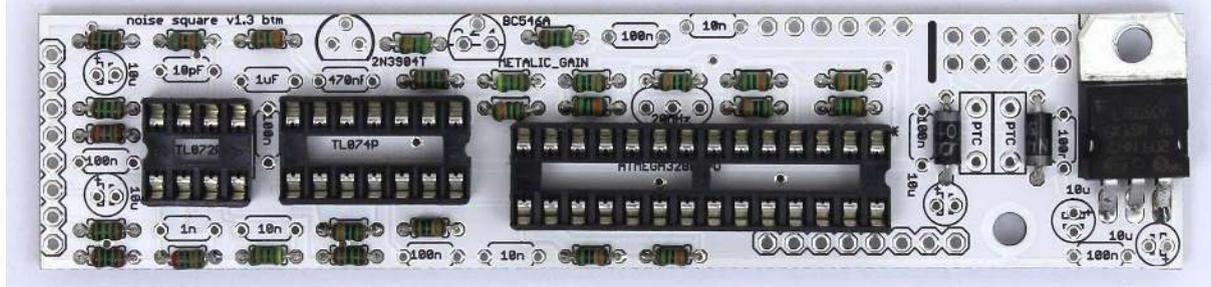
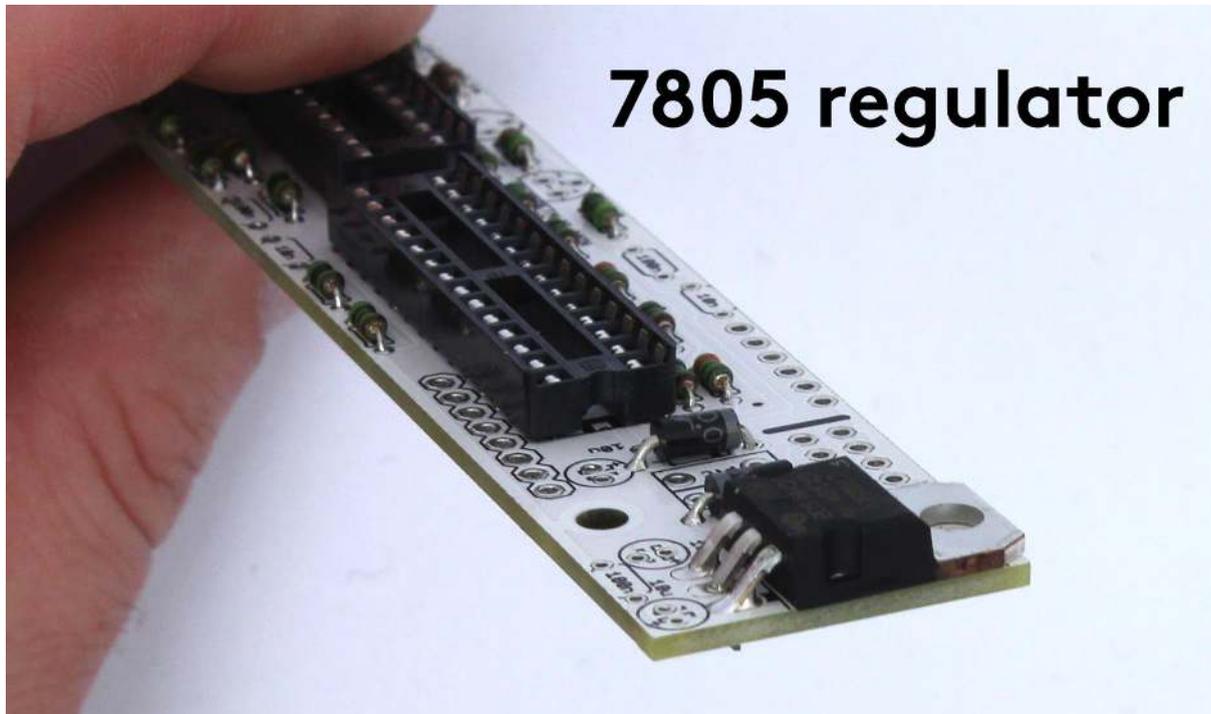
Let's populate the PCB with three IC sockets (8, 14 and 28 pin). Make sure that the notch on the socket matches the print on the board.

IC sockets watch out for the notch orientation



VOLTAGE REGULATOR

Next solder the **7805 voltage regulator**. Bend its legs as close to the body as possible, at a right angle to make sure that it lies flat on the circuit board.

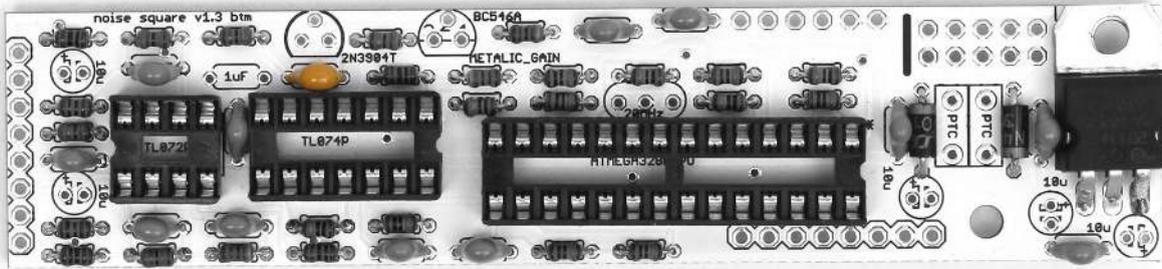


CERAMIC CAPACITORS

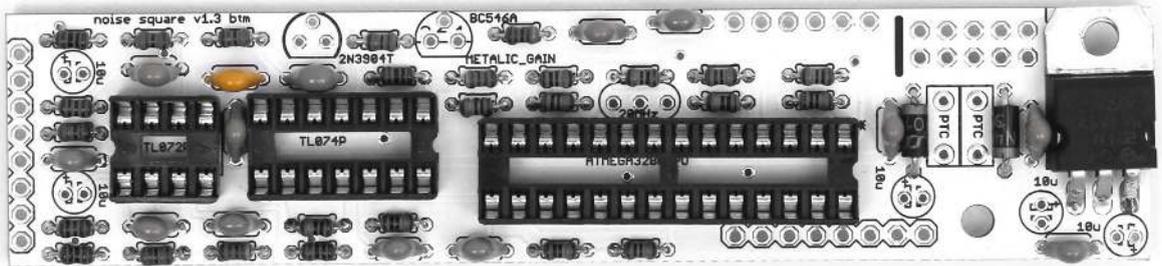
Now let's do the **ceramic capacitors**. These parts are not polarized. There are fourteen of them:

- one **10pF** (marked "100" on itself),
- one **1nF** (marked "102"),
- three **10nF** (marked "103"),
- seven **100nF** (marked "104"),
- one **470nF** (marked "474"),
- one **1μF** (marked "105").

"474" cap (1x)



"105" cap (1x)



ELECTROLYTIC CAPACITORS

There are also five **electrolytic capacitors** (10 μ F). These ones are **polarized!** There is a plus (+) sign on the PCB that should match the longer lead of the electrolytic capacitor (actually the minus (-) side is also marked on the body of the capacitor with a white strip).

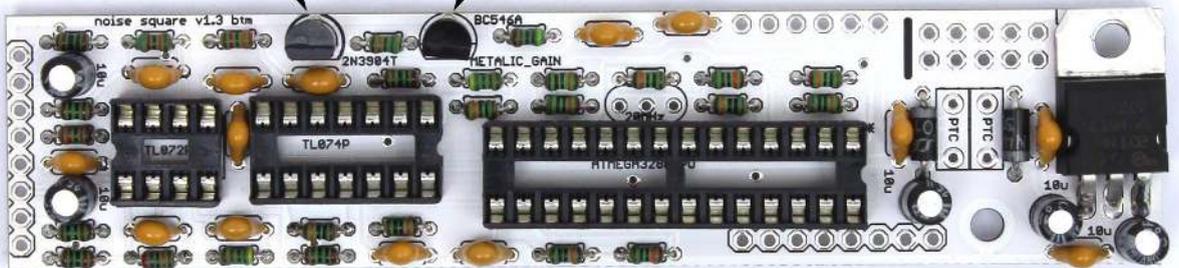


TRANSISTORS

Now you can do the **transistors** (1x 2N3904, 1x BC546A). Be aware of putting them in oriented in the right way - the **flat side of transistor must match the outline drawn on the PCB!**

transistors
(watch out for the orientation!)

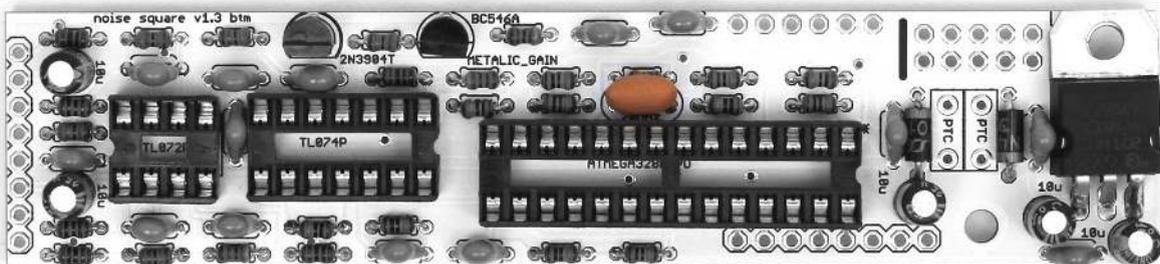
2N3904 **BC546A**



RESONATOR

Solder also the **resonator** (the orange component with 3 leads). The marking on PCB is signed "20MHz".

20MHz resonator

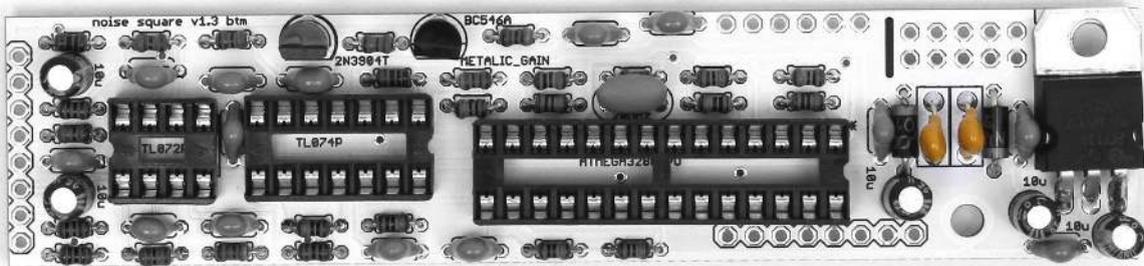


FUSES



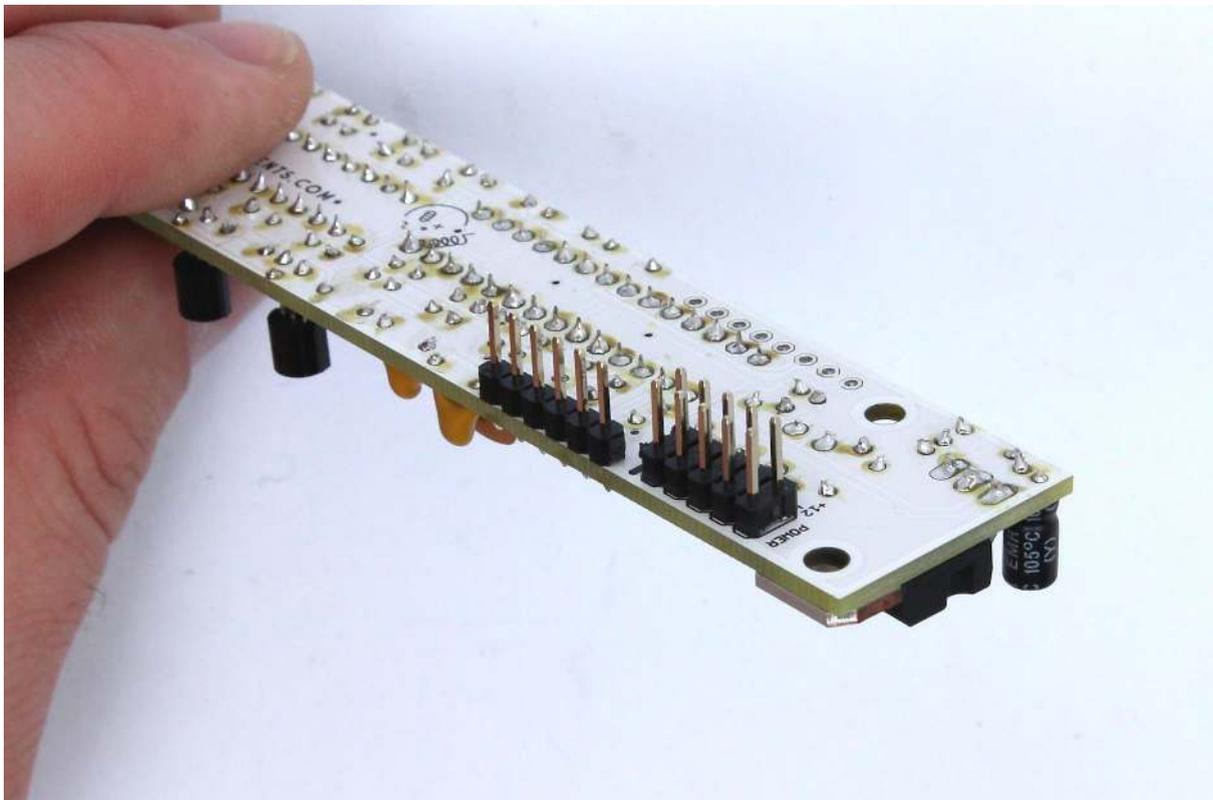
Move to soldering the **fuses**. They look **quite similar to capacitors**. Place them in the “PTC” rectangle on board.

fuses (2x)



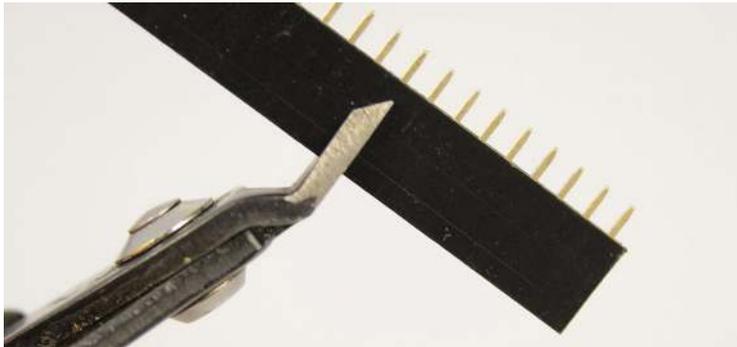
MALE HEADERS

Turn around the PCB and insert the male **pinheaders**. Be careful to solder them straight! You may first solder e.g. the middle pin, then take the board in your hand and re-heat that pin while pressing down on the header to align it. Wait for it to cool and solder the rest of the pins.



FEMALE HEADERS

As you can see one **female pinheader** left. Use your flush cutters to get **two 1x8 headers** (you will always lose one pin when cutting the female headers, so be sure to cut it always after the last required pin - check the picture to see where to cut to get 8 pin). Then keep them as you will use them in the later step.



NUT + SCREW

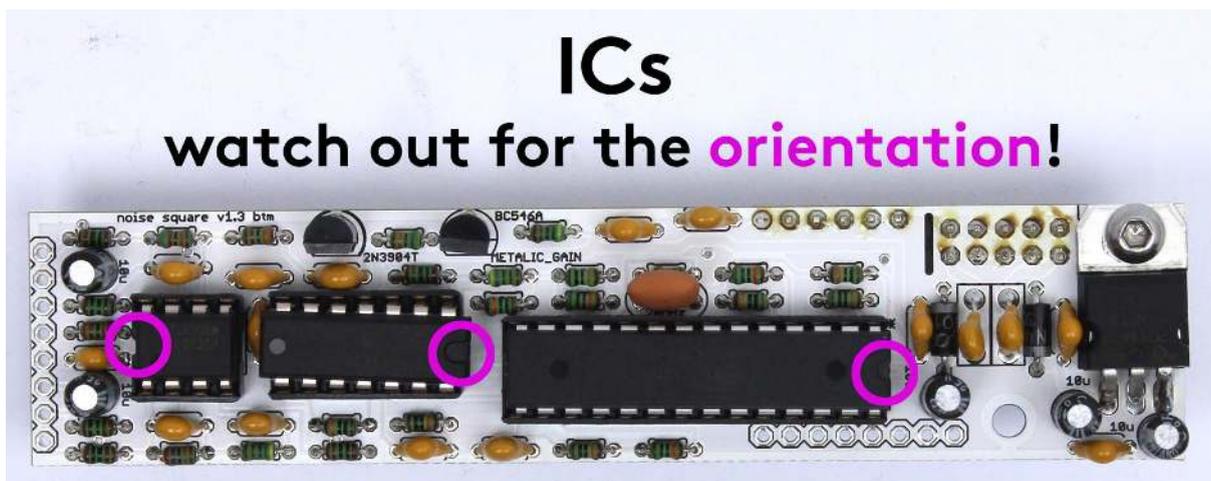
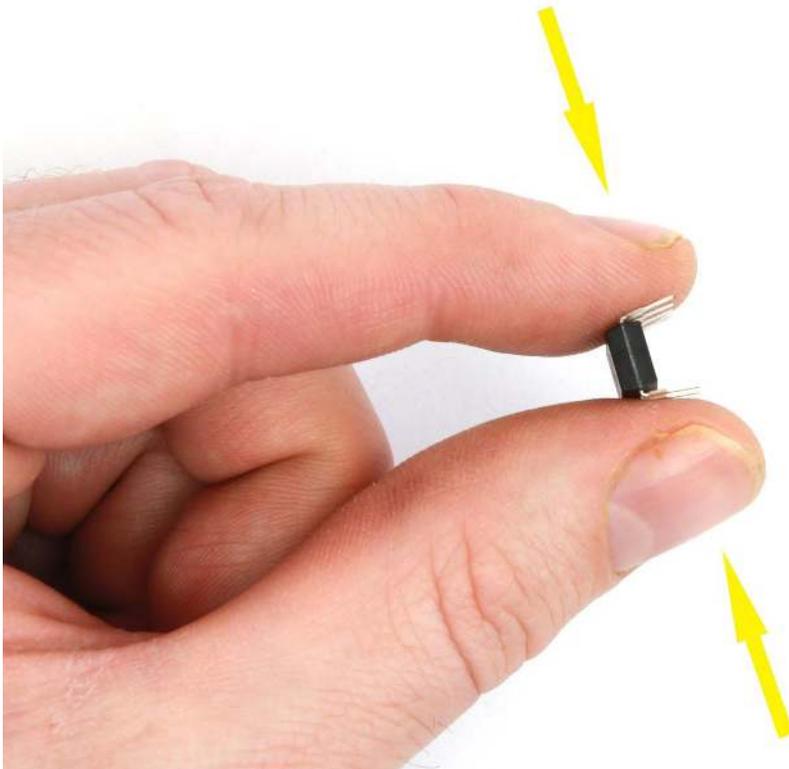
Now you can also install the small nut and screw on the voltage regulator.

nut + screw



ICs

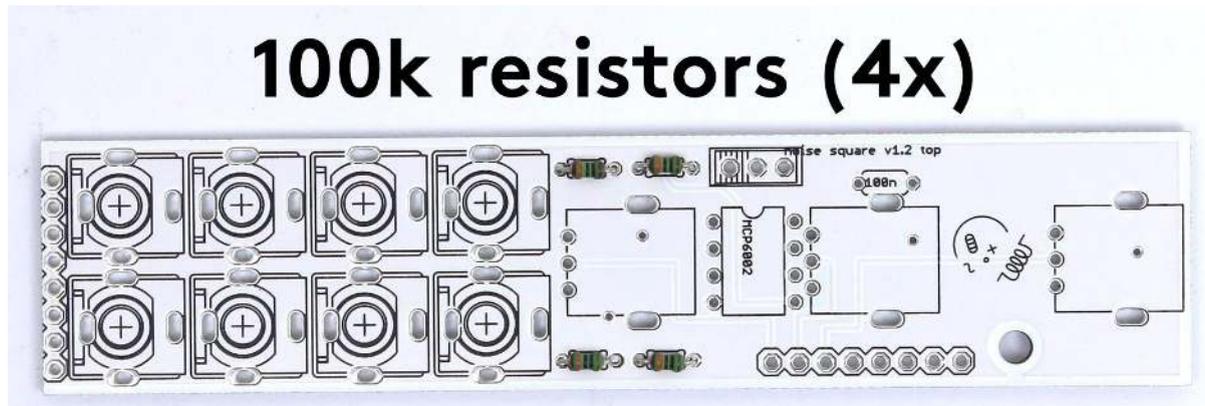
Finally, insert **ICs** into sockets (1x Atmega, 1x TL074 and 1x TL072). **Watch out for orientation!** There is a notch on Atmega and TL074 that should match with the notch on the sockets. For TL072 is relevant the dot on it. Installing the ICs can be a little tricky. The IC leads are flared out a bit wider than the socket will accept. Bend them in slightly with your fingers, and then try to press all the leads into the sockets in one shot.



TOP BOARD SOLDERING

RESISTORS

Let's populate the top board now. Again, start with the four remaining **resistors** (100k) and solder them in.



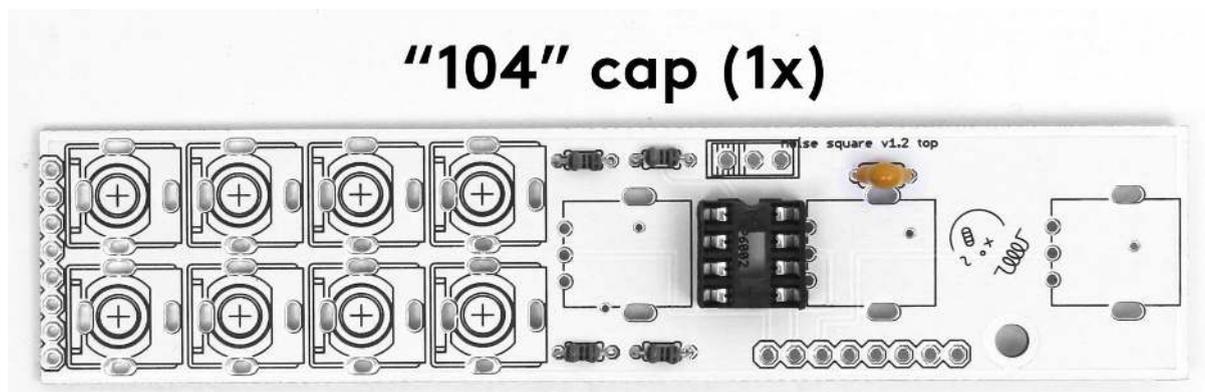
IC SOCKET

Add the **IC socket**. Watch out for the **orientation**!



CERAMIC CAPACITOR

Then you can add the **100nF ceramic capacitor**.

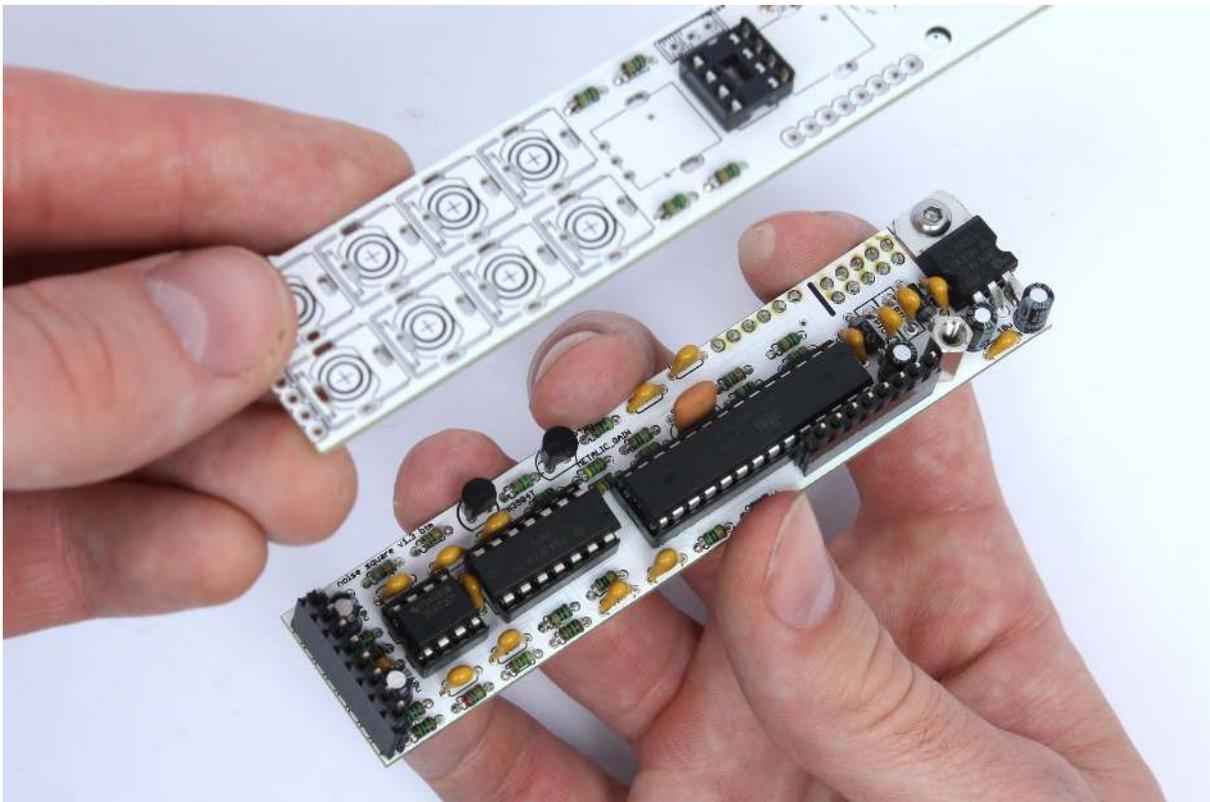
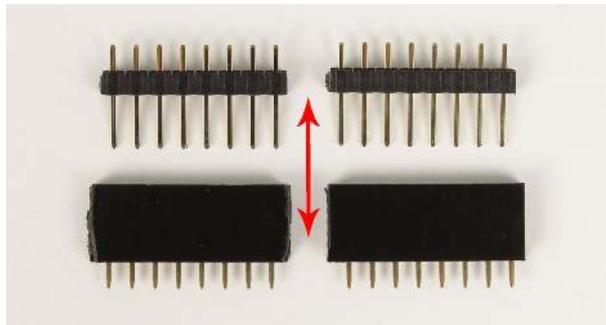


PINHEADERS

To ensure that the headers will be properly aligned, **screw the nut x nut spacer** on the bottom board first.



You need **two 1x8 male pinheaders** now. Cut them with your flush cutters. Place the **female headers** on the bottom board with the male pins inserted. Connect the boards together and mount them with the other spacer. Now you can **solder the headers** on both boards.

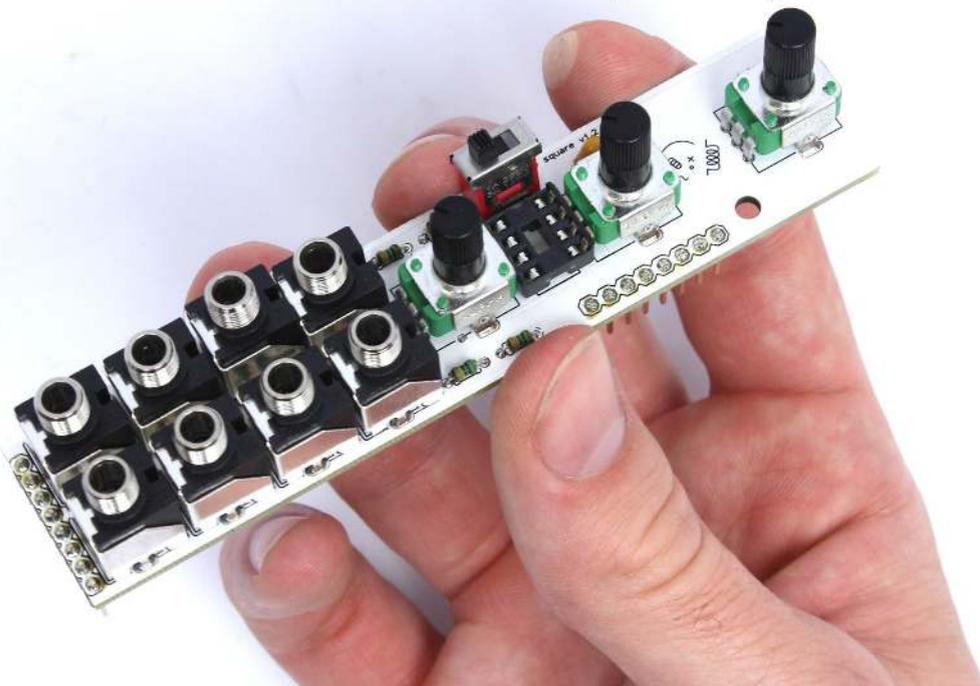




POTENTIOMETERS, JACKS AND SWITCH WITH PANEL

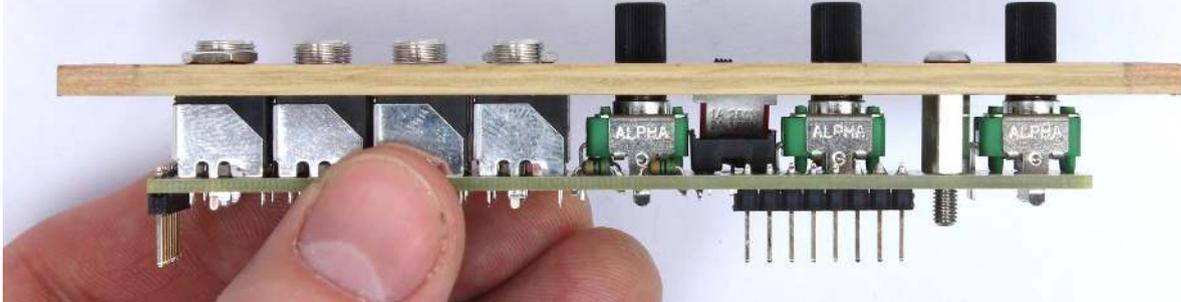
Unscrew the top spacer and disconnect the two boards. Next place the three **potentiometers** (B100k) to their respected places on the top board. Push them well until they **sit absolutely flat on the board**. Then place the mono **jacks** and the **switch** on the board. **Don't solder anything yet.**

do not solder these parts yet!

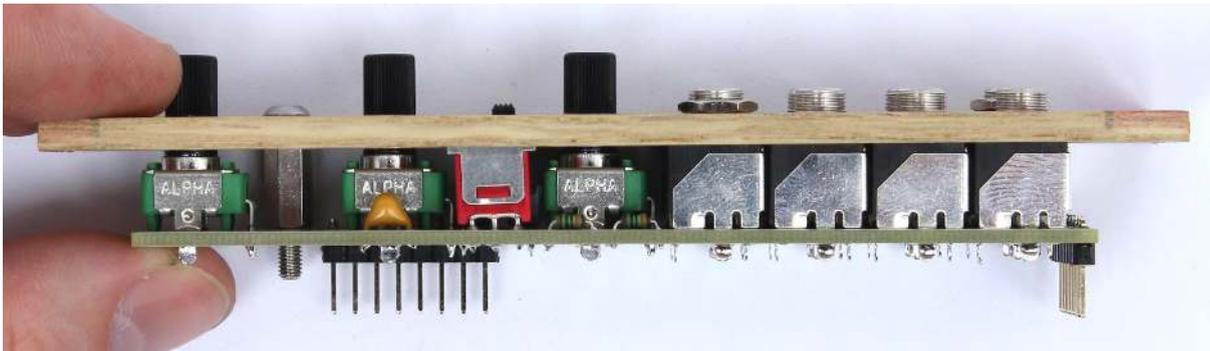


Place the **spacer** back in the opening. Take the **front panel**, **screw** and mount it with the board. Check that all the components came through. Secure some of front and back jacks to the panel with the **nuts** (keep in mind not to tighten the jack nuts too much as you may damage the panel!).

front panel + nuts + spacer + screw



Push the **switch** to be sure that they come through the panel. Make sure that everything is properly aligned. Now you are finally ready to solder all these components.



IC

After this place the IC (MCP6002) into the socket (the socket notch should match the IC notch).

MCP6002 watch out for the orientation

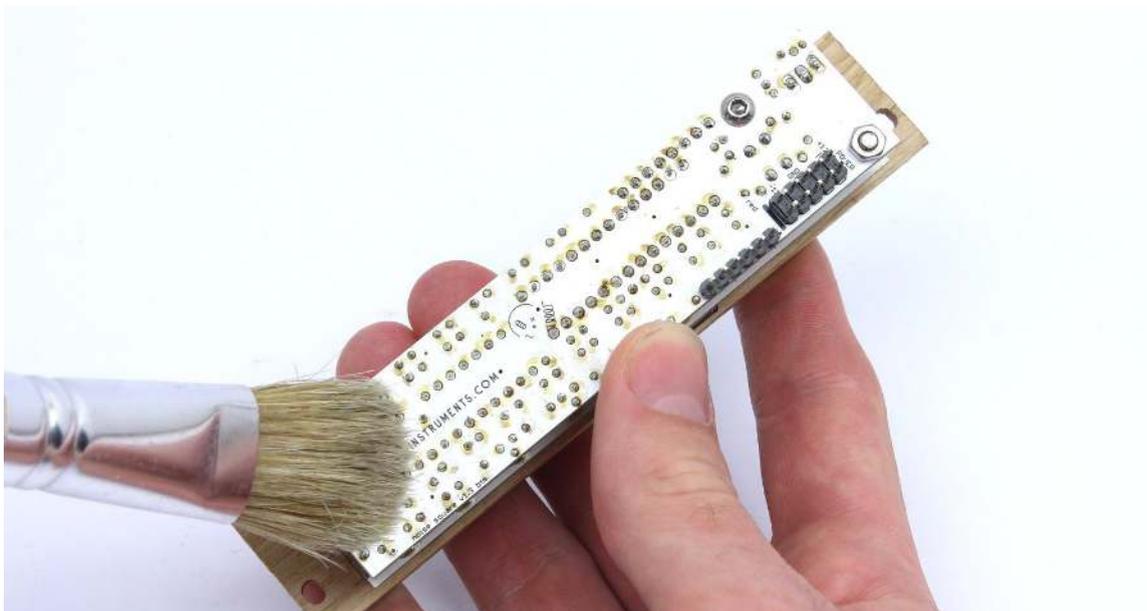


Congratulations! You have made it through. Now just connect the boards, add the knobs and your NOISE SQUARE is ready to go! Before you connect anything, make sure that your system is disconnected from power. Also double check the polarity of the ribbon cable, the red cable should match the -12V rail both on the module and on the bus board!



NOTE FOR CLEANING THE PCBS

You may want to **clean** your PCBs in order to keep the PCBs alive for a longer time and to just look cool. You can use e.g. isopropyl alcohol. Put some of the liquid all over the PCB using the brush, let it act for a while and sweep it off. Then just let it dry. You can repeat these steps until you are satisfied with the result.



TROUBLESHOOTING

First check out the [DIY F.A.Q.](#)

If you are having some more trouble, the best thing is to take a nap! Especially late at night!

If you are still in trouble you can send the detailed description of the problem with enclosed high-resolution photos on diy@bastl-instruments.com.

If you think that you are unable to make the module work on your own, consider our "[Come to Daddy](#)" service.