

B A S T L INSTRUMENTS

NOISE² v1.1 - Assembly Guide

bastl-instruments.com

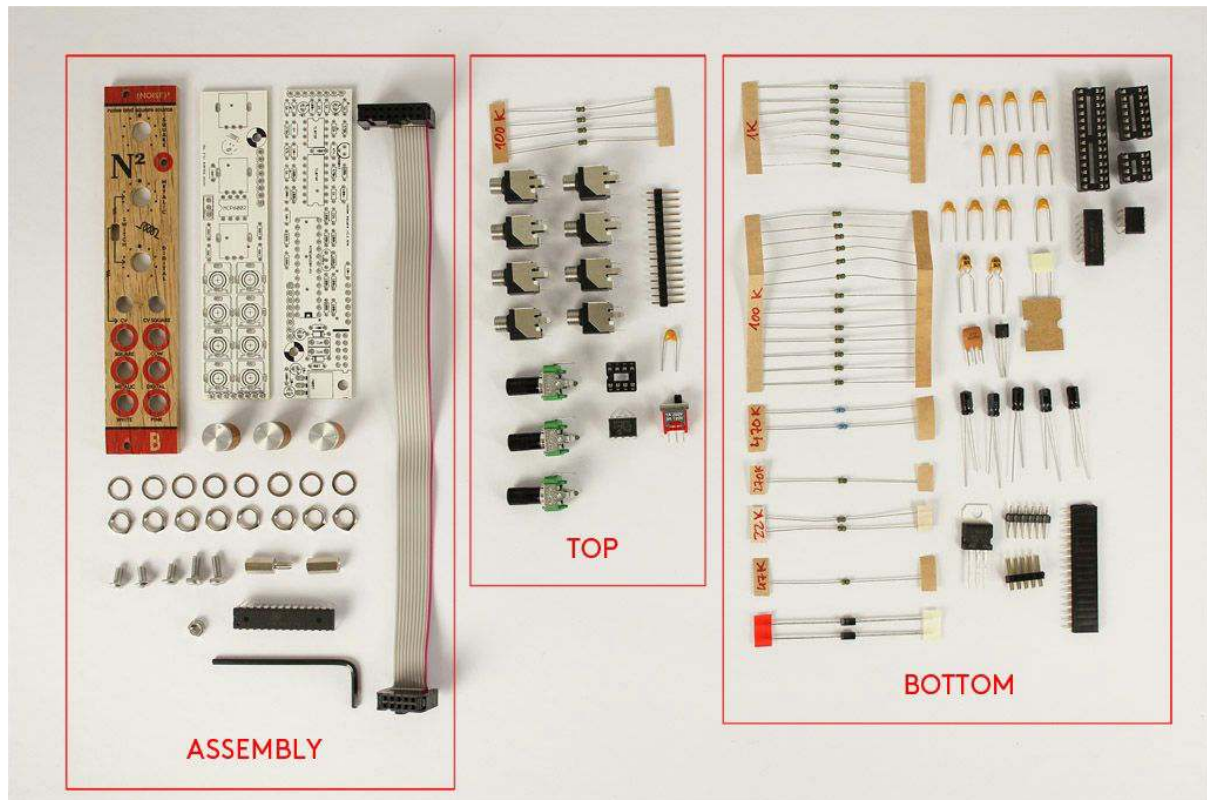


INTRODUCTION

This guide is for building Noise² module from 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](http://www.instructables.com/id/How-to-solder/)¹. We even included some of the best quality solder to help you solder everything faster and better.

¹ <http://www.instructables.com/id/How-to-solder/>

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



Before starting this kit, prepare the following tools:

- Soldering iron (15-20W)
- Multi-meter
- Flush cutters
- n2. hex screwdriver or allen key (enclosed with kit)
- Phillips screwdriver (cross)
- 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.

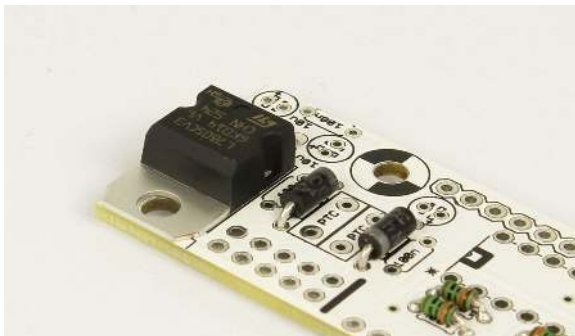
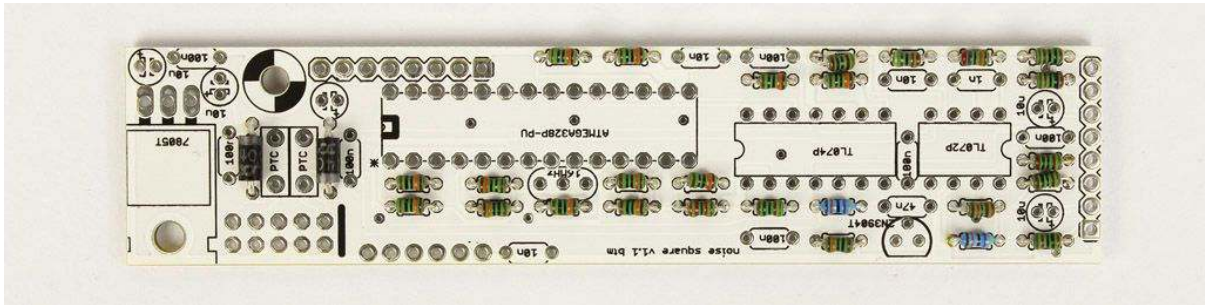
BOTTOM BOARD

Let's start with the bottom board. Before you start soldering, take your time and find all the resistors values [using a multimeter](#)² (or you can check the color codes if you are seasoned enough).

Now insert and solder 26 **resistors** (13x 100k, 7x 1k, 2x 22k, 1x 270k, 1x 47k, 2x 470k). Then snip the leads as close to the PCB as you can (be sure to make this step on all remaining leads in the course of

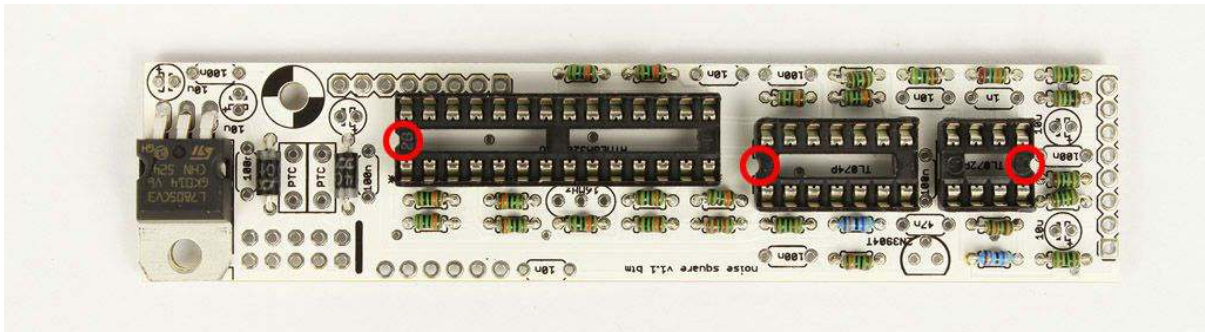
² <https://learn.sparkfun.com/tutorials/how-to-use-a-multimeter/measuring-resistance>

this guide). Then do the same with the two **1N4007 diodes**. **Be careful, diodes are polarized!** Make sure that the marking ring on the diode body matches the marking on the circuit board.

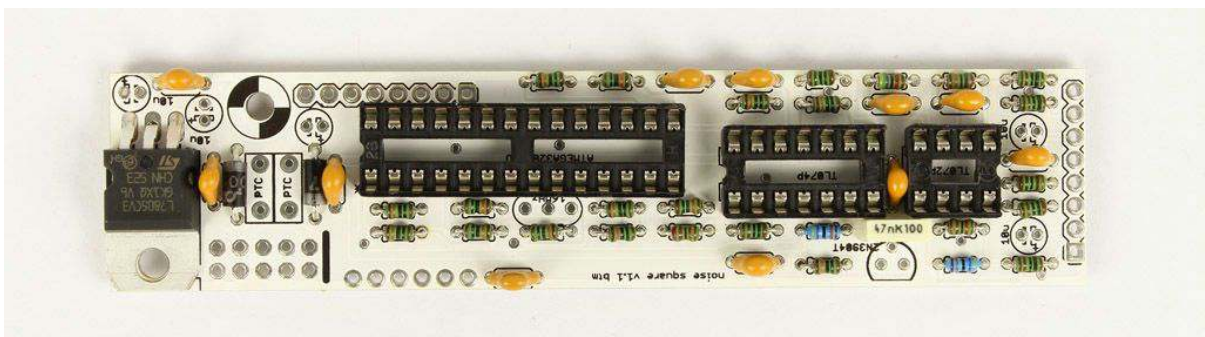


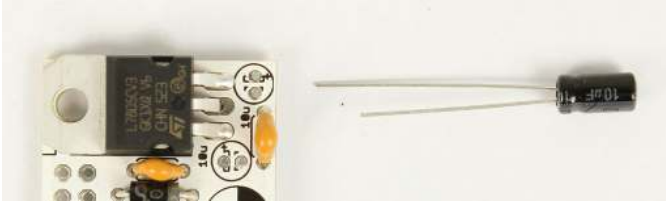
Next solder the **7805 voltage regulator**. Bend its legs as close to the body as possible, at a right angle to make sure that it lies flat on the circuit board.

Then solder three **IC sockets** (1x 8 pin, 1x 14 pin, 1x 28 pin). Make sure that the notch on the socket matches the print on the board.



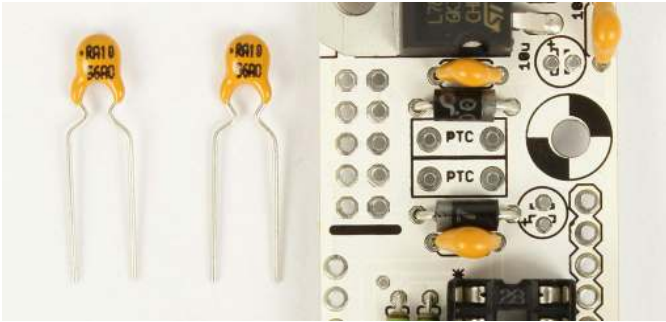
Now let's do the **capacitors**. There are seven **100nF** (marked 104), three **10nF** (103), one **1nF** (102) and one **47nF**. They might be in ceramic or polyester film package. These parts are not polarized so don't worry.





Next solder in:

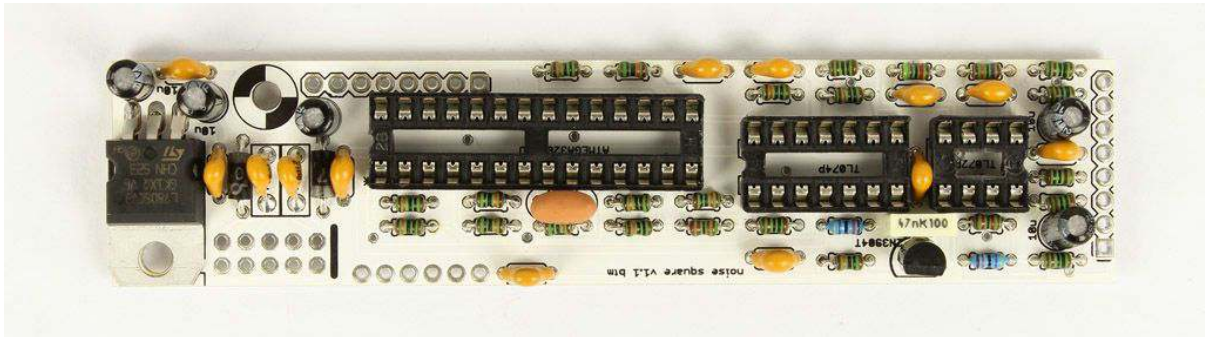
- Five **10µF electrolytic capacitors** (**watch out, these ones are polarized!** There is a plus (+) sign on the PCB that should match the longer lead of the capacitor)



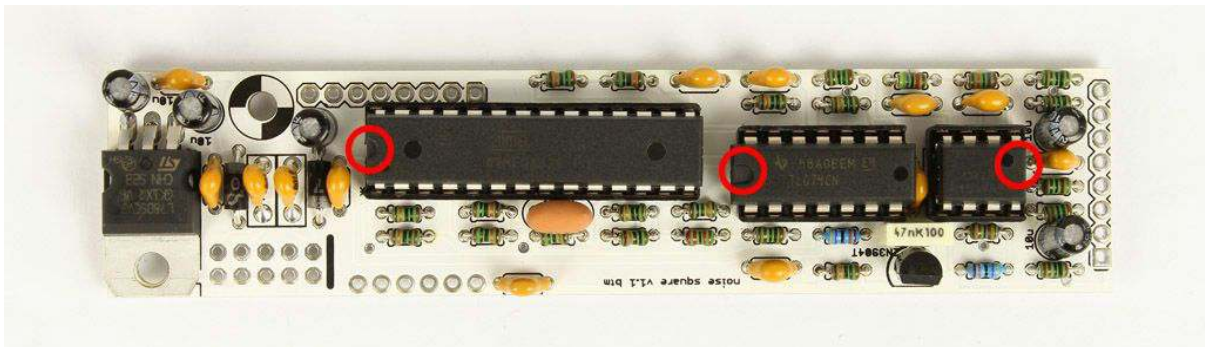
- One **2N3904 transistor** (flat side of transistor must match the outline drawn on the PCB!),

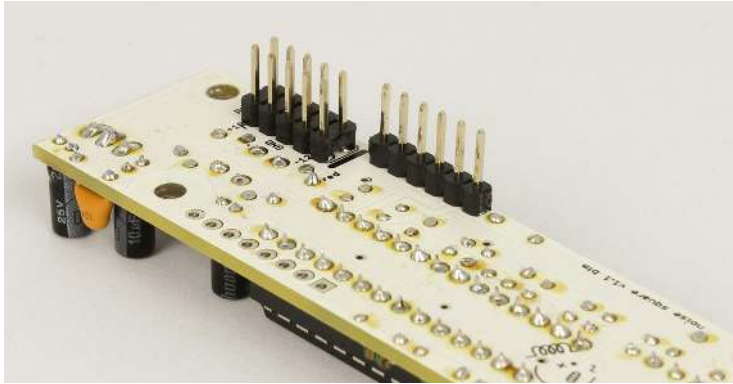
- One **20MHz resonator** (the orange component with 3 leads, the marking on board is actually wrongly signed "16MHz"),

- Two **fuses** (marked "PTC" on board; be careful, **they look quite similar to capacitors**).

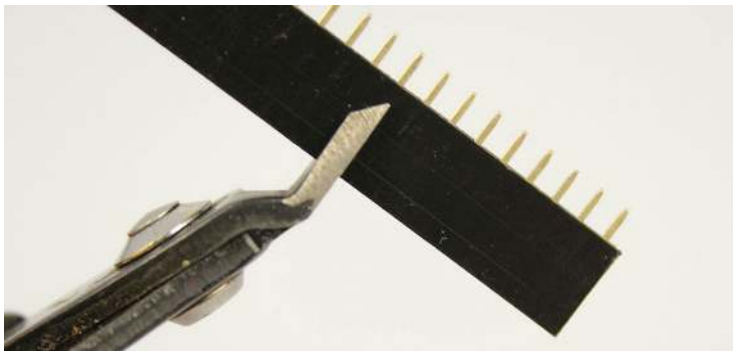


Now insert **ICs** into sockets (1x Atmega, 1x TL074 and 1x TL072). **Watch out for orientation!** There is a notch on Atmega and TL074 that should match with the notch on the sockets. For TL072 is relevant the dot on it.

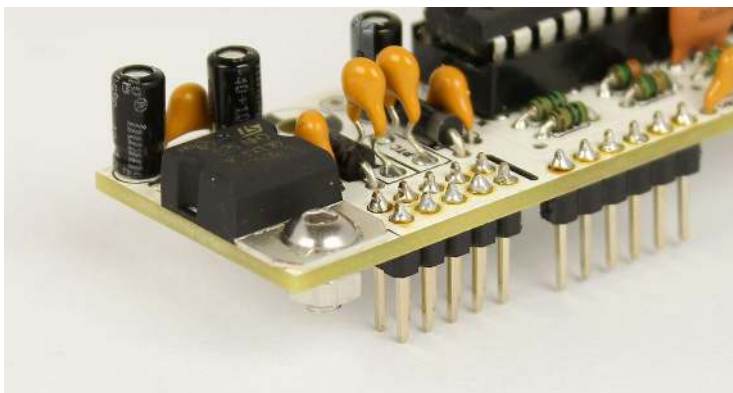




Turn around the PCB and prepare **pinheaders** with your flush cutters if it is necessary. You need **one 1x6 pin** and **one 2x5 pin male**. Be careful to solder the pinheaders straight! You may first solder the middle pin, then take the board in your hand and re-heat that pin while pressing down on the header to align it. Wait for it to cool and solder the rest of the pins.



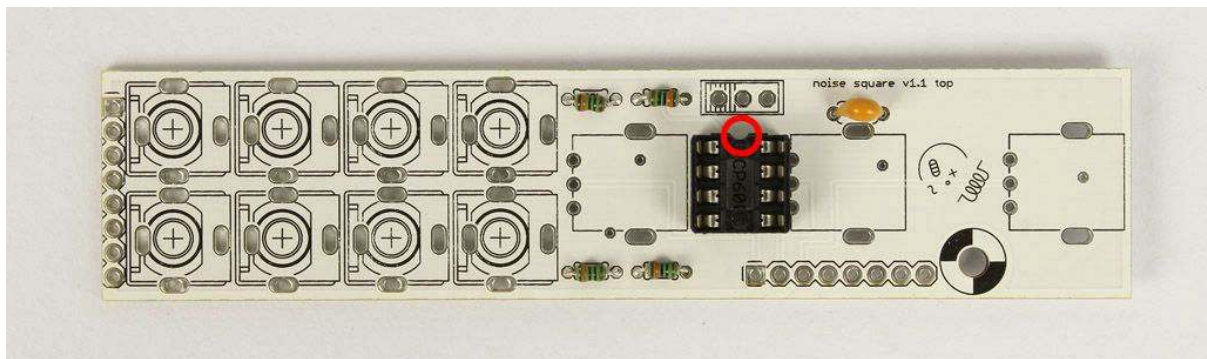
As you can see one **female pinheader** left. Use your flush cutters to get **two 1x8 headers** (you will always lose one pin when cutting the female headers, so be sure to cut it always after the last required pin - check the picture to see where to cut to get 8 pin). Then keep them as you will use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



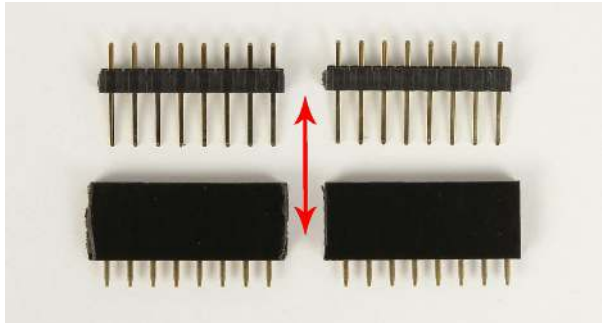
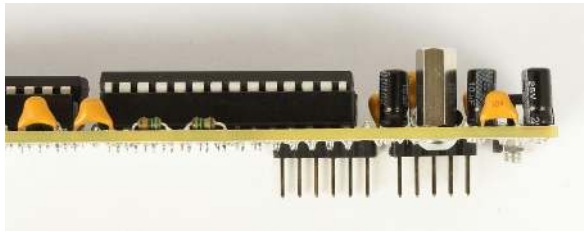
Now you can install the small nut and screw on the voltage regulator.

TOP BOARD

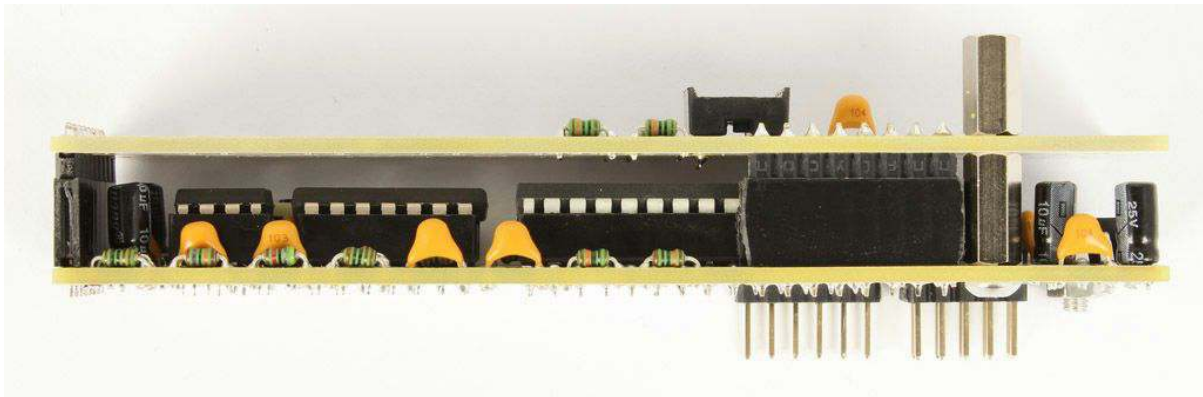
Now populate the top. Again start with the four remaining **resistors** (100k) and solder them in. Then solder the **IC socket** (1x 8 pin) and the **100nF capacitor**. Make sure that the notch on the socket matches the print on the board.



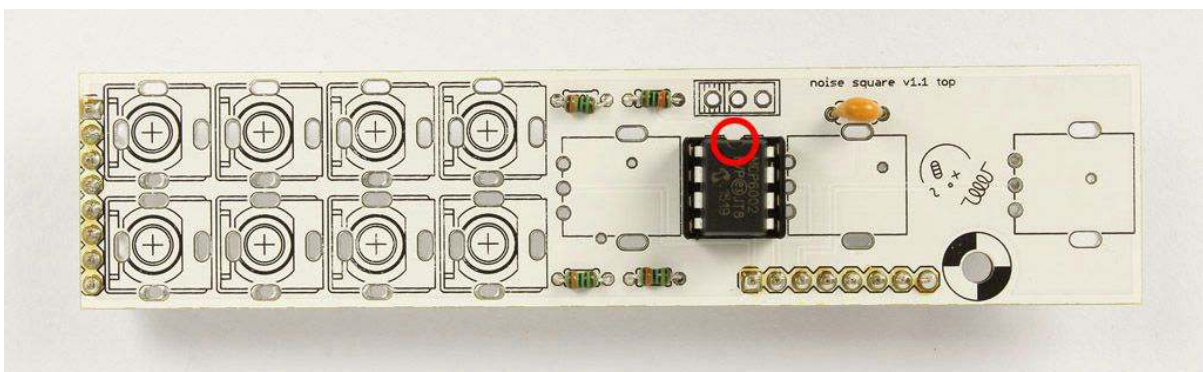
You need **two 1x8 male pinheaders** now. Cut them with your flush cutters. Now to ensure that the headers are properly aligned, **screw the 11 mm spacer** on the bottom board.



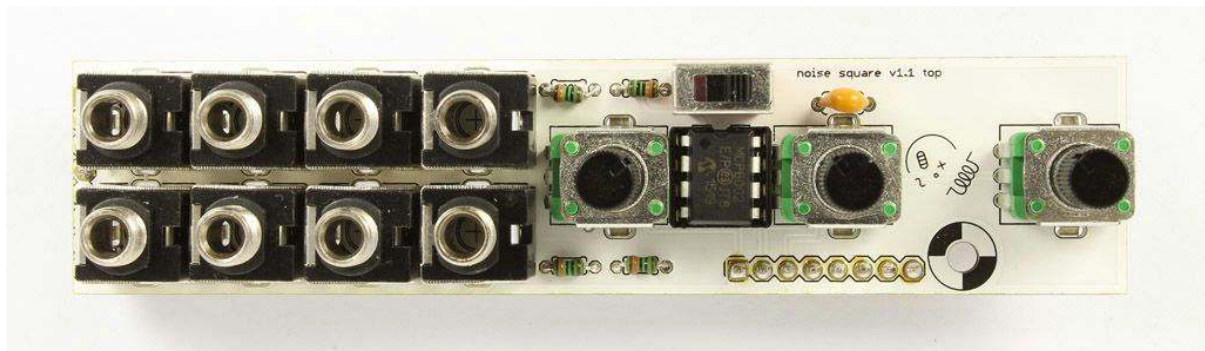
Place the **female headers on the bottom board with the male pins inserted**. **Connect the boards together** and mount them with the other spacer. Now you can **solder the headers** on both boards.



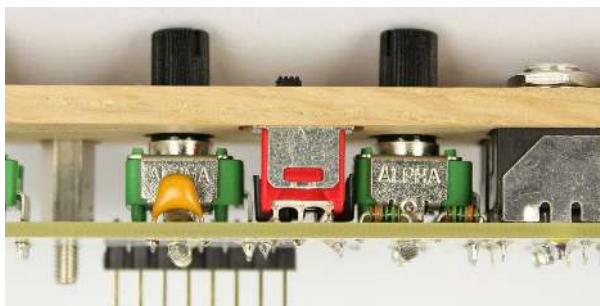
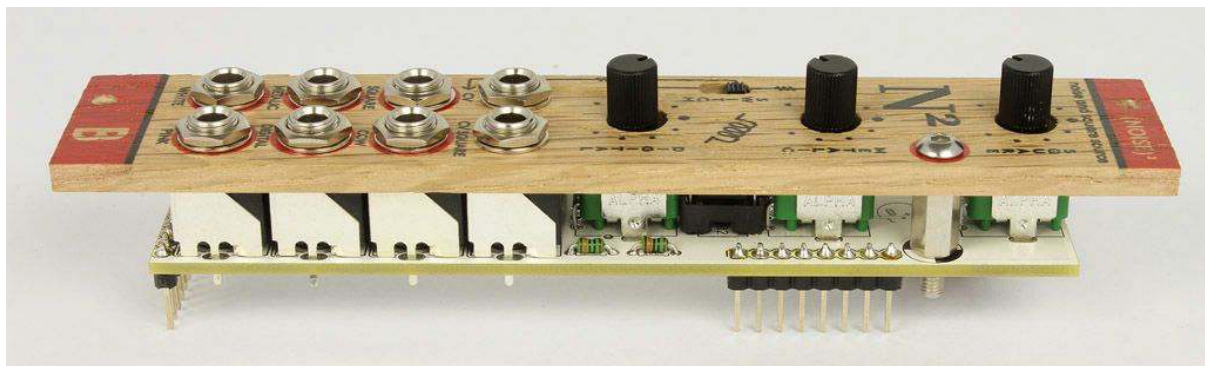
After this place the **IC (MCP6002)** into the socket (**the socket notch should match the IC notch**).



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.**



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 the jacks to the panel with the **washers** and the **nuts** (keep in mind not to tighten the jack nuts too much as you may damage the panel!).



Push the **switches** 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.

