attention | disclaimer

This project is presented as artwork, and is solely intended as such.

build at your own risk

this pdf is for information purposes only

© e-licktronic 2023

What skills are needed?

Meticulous care is the first skill you will need to build one of those kits. One single component that is not in the right place, in the right direction, one single solder joint that is deficient and the whole thing won't work! So, the secret to success is simple: follow the assembly guide down to the letter and **triple check all your moves**.

Good soldering practice is also needed. Your boards will be as weak as your weakest solder joint. So, this is very important. But it is easy to learn. If you have no soldering experience, start practicing with a piece of Veroboard and a handful of resistors. You should be perfectly operational after half an hour. Please have a look on those tutorials:
EEVblog #180 - Soldering Tutorial Part 1 - Tools
EEVblog #183 - Soldering Tutorial Part 2

Basic electrical and electronic understanding. Do you know what is a Volt, an Ohm, an Ampere? Can you use a digital multimeter? These are the questions you must ask yourself. If the answers are no, you might try to find a friend that knows

If you own these 3 skills, your kit will work for the first time!

What tools do you need?

- A good soldering iron, preferably thermally regulated, 60W, with a medium size tip (1.5-2.5mm wide), screwdriver shape.
- Solder. **Only use first grade solder**, 0.7 to 1 mm diameter, "no clean" type. Good quality solder will make the soldering process easier and you will get nice shiny joints.
- Cutting plier. If you plan to go ahead with DIY, buy good quality cutters.
- Flat nose pliers.
- Phillips screwdriver, Slot screwdriver 3mm for trimmers, Nut driver ø5.5mm

- Digital multimeter with thin, insulated test hooks. If you can afford it, choose an RMS voltmeter with a good resolution.

What is included in the M-118a kit?

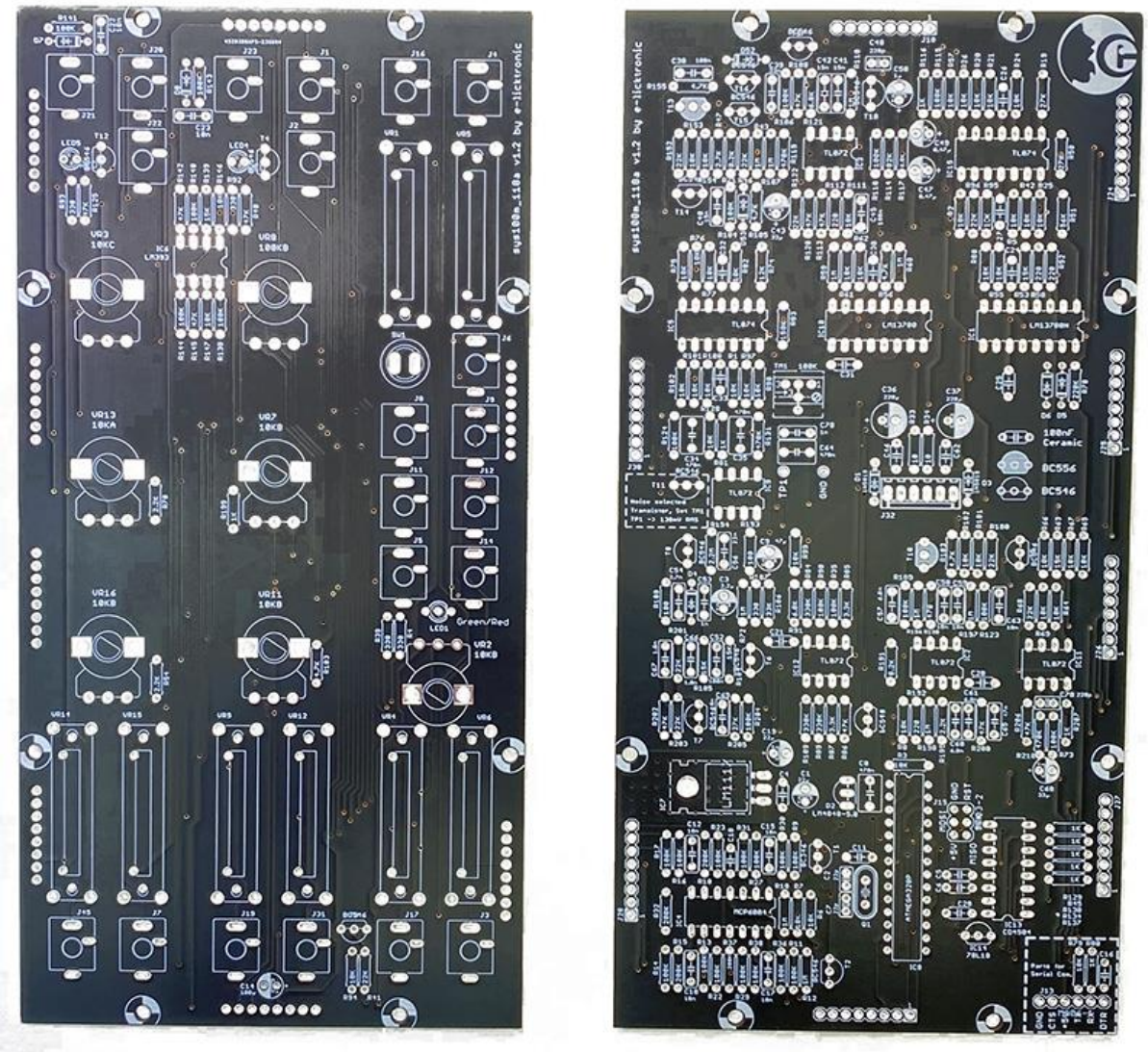
- 1x 5mm aluminum silkscreened faceplate
- 1x Epoxy backplate
- 2x PCBs: One components PCB and one potentiometer PCB
- 21x mono audio jacks 3.5mm
- 1x 8 Pin DIN Female Solder Panel Mount Connector
- 1x push button switch
- 1x 6 pins Molex housing cable with PCB connector
- 6x original Roland system 100M 21mm knob
- 1x original Roland system 100M 13mm knob
- 8x original Roland system 100M slider cap
- 8x 20mm 10KB linear potentiometer
- 1x 10KC potentiometer
- 1x 100KB potentiometer
- 1x 10KA potentiometer
- 4x 10KB potentiometer
- 1x Preprogrammed µC
- 2x LM13700 quad OTA
- 8x 8 pins 2.54mm pitch female headers
- 2x 40 pins 2.54mm pitch male headers
- 2x White nylon T washer
- 2x M3x10 Pan head pozi machine screw with integrated washer
- 10x M3x6mm **Pan head** pozi machine screw
- 8x 11mm black nylon spacer
- 8x 12mm black nylon spacer
- 8x 30mm black nylon spacer
- 1x 30cm Din8 to Din8 cable
- 2x M3 Full nut

SOLDER THE COMPONENTS BOARD

Use the **iBOM** to help you to place the components.

Pictures could show different parts type and color than the one you have.

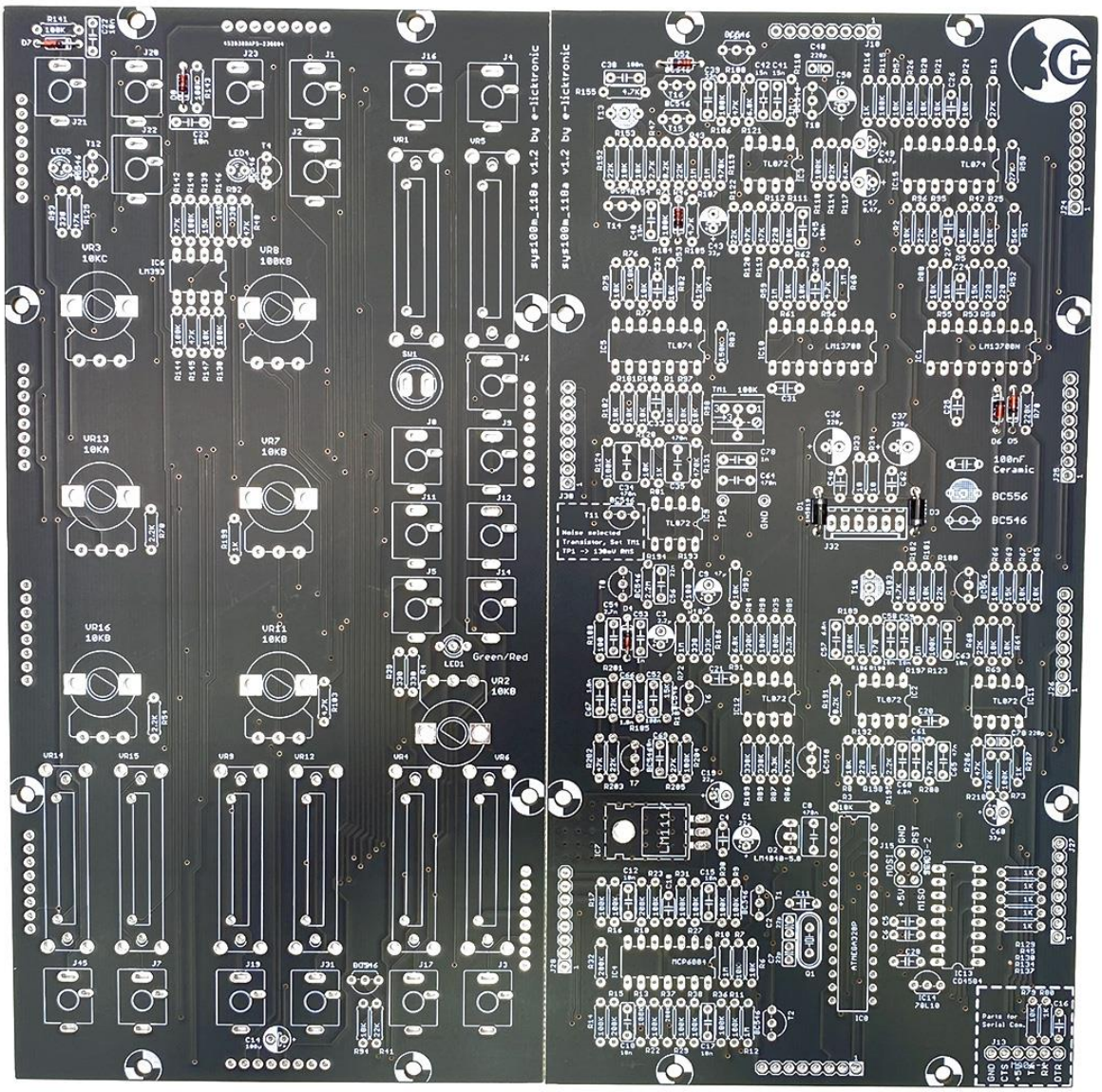
First you need to split both printed circuit board (PCB).
Simply bend the PCB in the middle.



Ok, now we're getting serious! Let's start stuffing the components board, beginning with the diodes. There are two types of diodes, the 1N4148 and 1N5819. Don’t mix it. Once the diodes are insert in the PCB, bend a little bit the legs then turn over the PCB to solder them. Cut the excess part of the leg with the cutting plier. Take your time to solder, **you can’t**

burn the diodes. Check the “placed box” on the diodes line in the **iBOM**.

Warning: Make sure to respect the direction of the diodes which is marked by a ring on the component and a line on the PCB.



After soldering the diodes in place, you can move on resistors.

The best method to select and install the resistors is the following:

-Take a marker and write value on the resistors bag

-Look up the **iBOM**

-Choose the first resistors value coming in the column

Example: the 10Kohm resistors

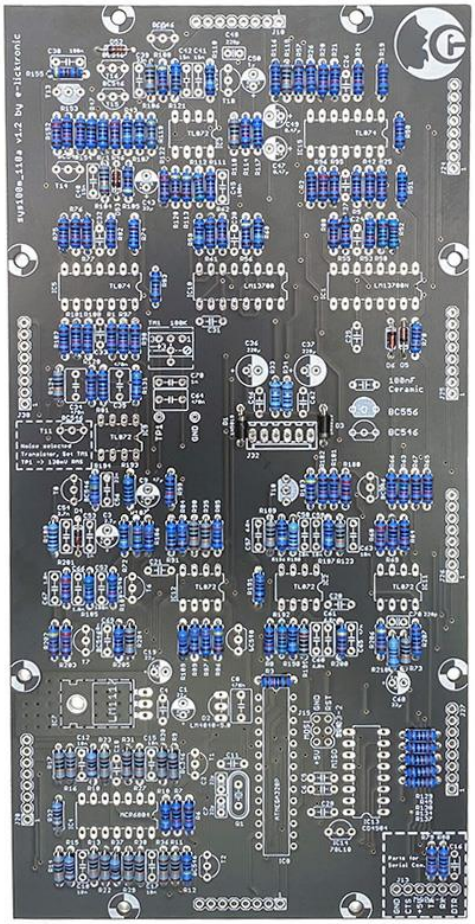
-Take 44x 10Kohm resistors in the bag and solder it on the PCB.

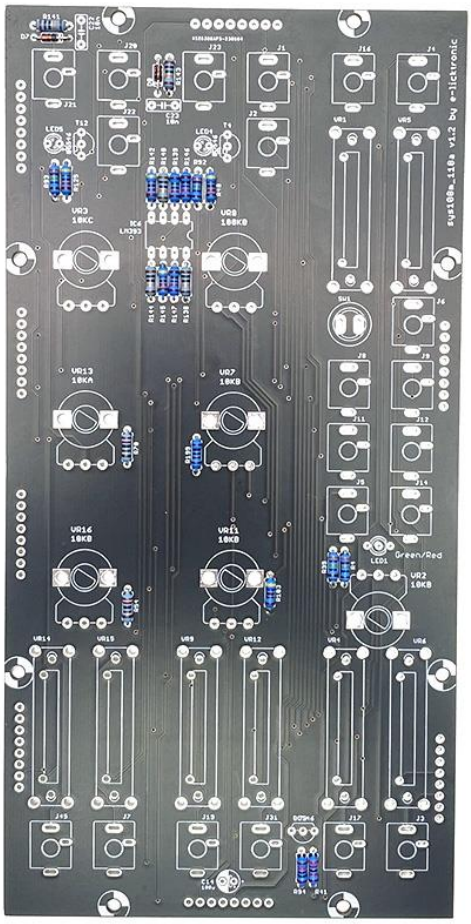
-Check the “placed box” on the resistor line in the **iBOM**

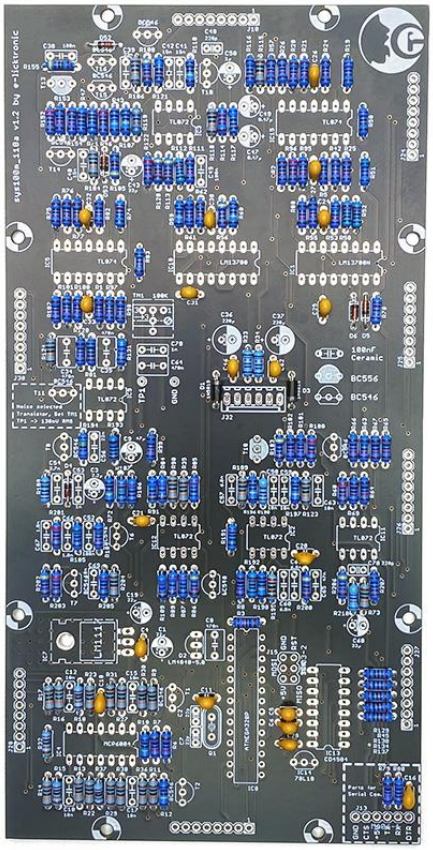
It is one of the best ways to not mix resistor values.

And so on for all resistors. Once resistors are solder, cut the leg on the solder side.

Warning: R33 and R34 has wrong value on the PCB and the iBOM. The correct value is 0 ohm instead of 10 ohms

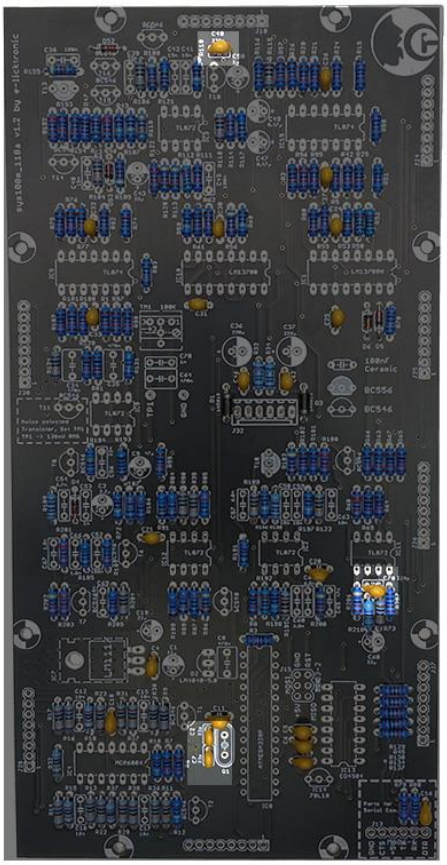


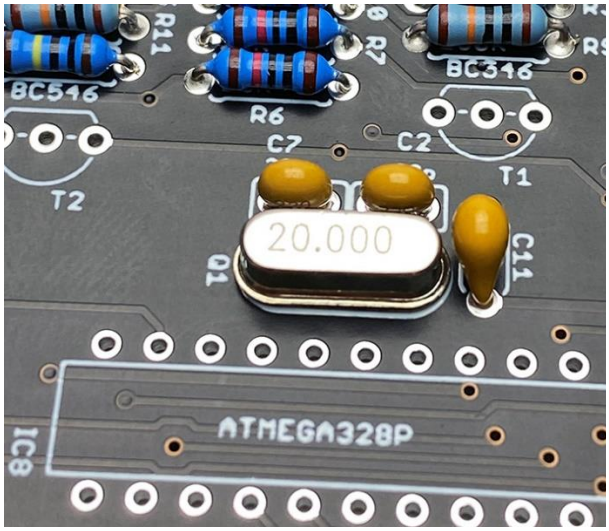




Next solder the 100nFceramic capacitor. Use the same method as resistor for these ceramic capacitors

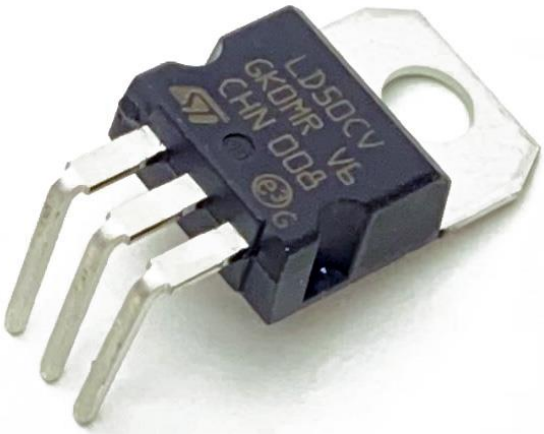
Solder the 22pF and 220pF ceramic capacitor. Use the same method as resistor for these ceramic capacitors

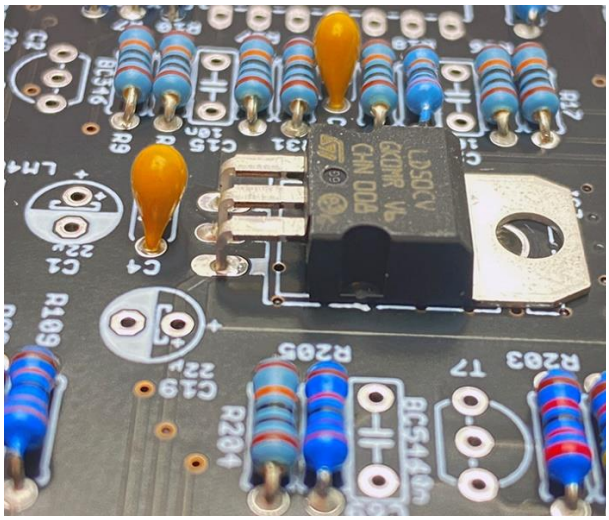




Solder the Q1 the 20MHz crystal. It has to sit flat on the PCB.

Take the LM1117 regulator and bend the pins with the flat nose plier





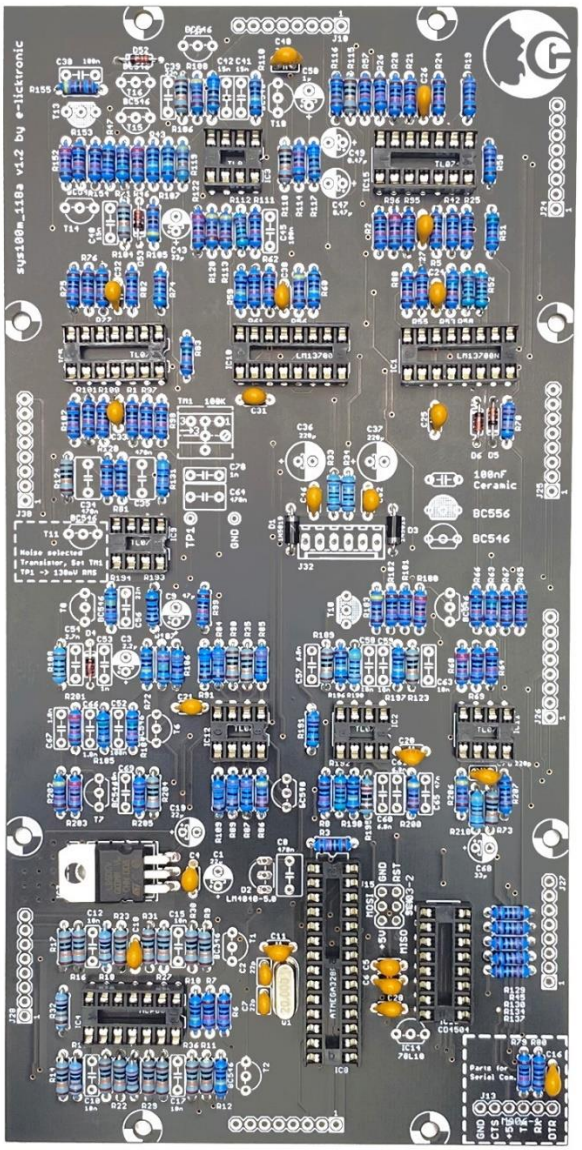
Solder the regulato, it has to sit flat on the PCB

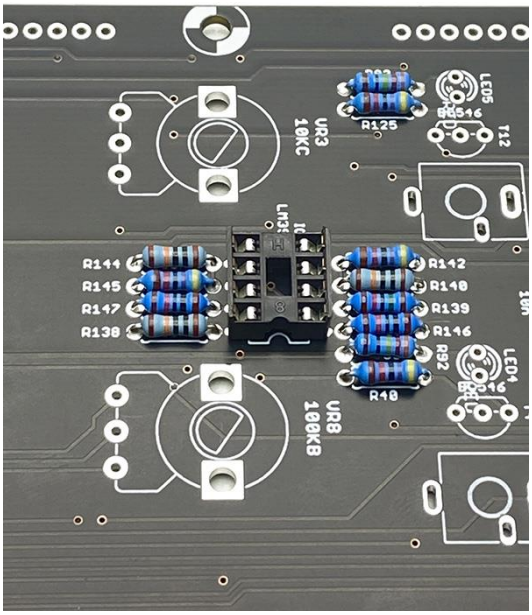
Now, solder the 8 pins, 14pins, 16pins and 28pins integrated circuit sockets.

Warning: Take care of the orientation of the socket, it has to match the PCB silkscreen.

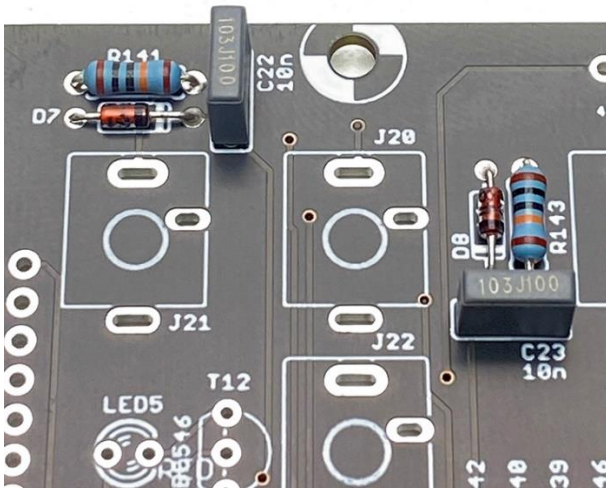
Solder the first pin of the socket only, flip the board then heat the pin and press on the socket to sit tight and square.

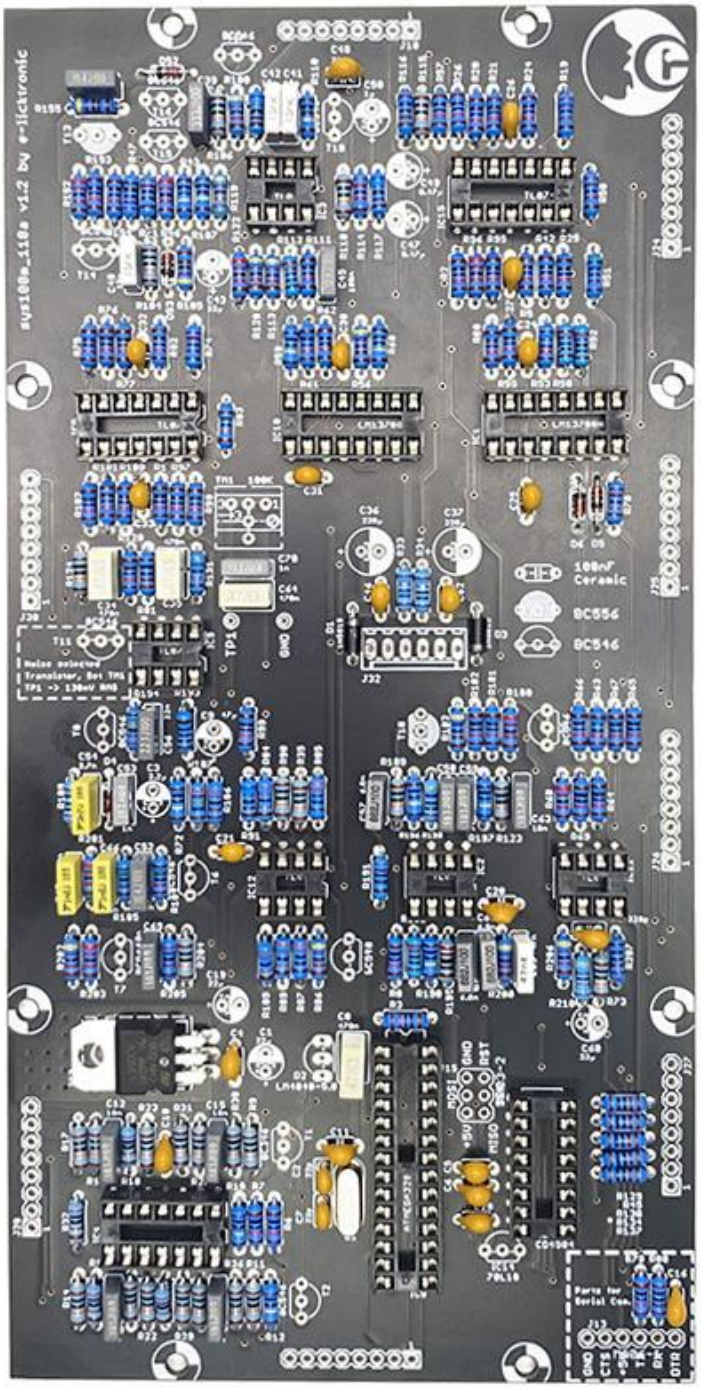
DO NOT MOUNT THE ICs NOW



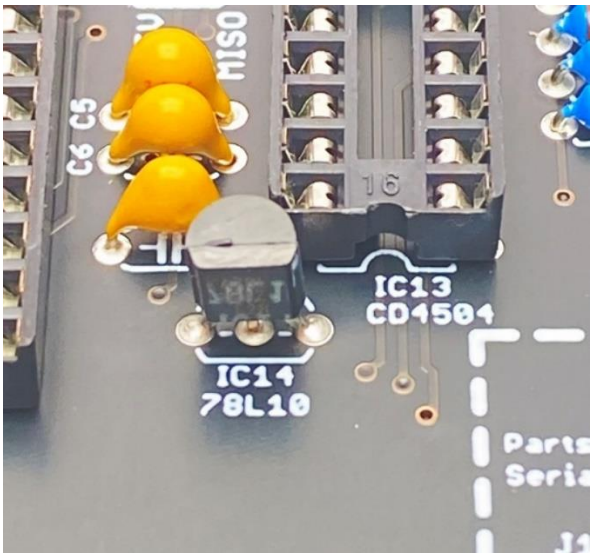


Solder film capacitors. Do not mix value, it is very important.
Still use the **iBOM** and place and solder each value one by one

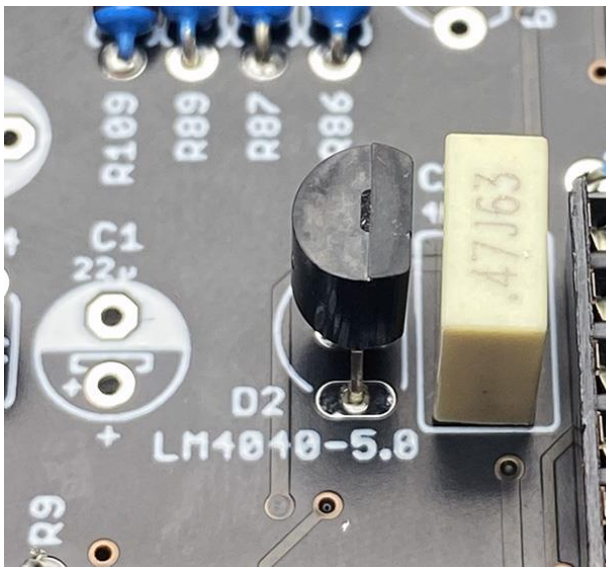




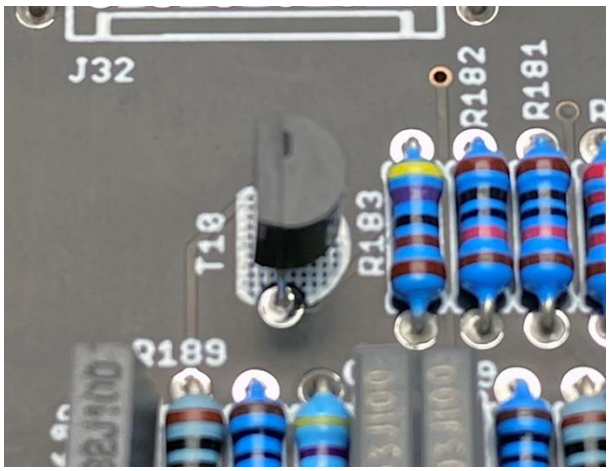
Now solder IC14 78L10 regulator. Do not mix with transistor, it is regulator 10V. Look at the picture, you can see 78L10 print on it
Warning: Do not heat this part to much

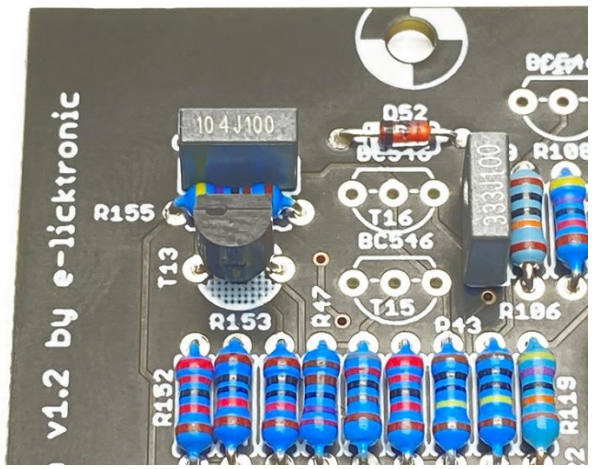


Next, solder D2 the LM4040, it is a precision voltage reference and NOT a transistor
Warning: Do not heat this part to much

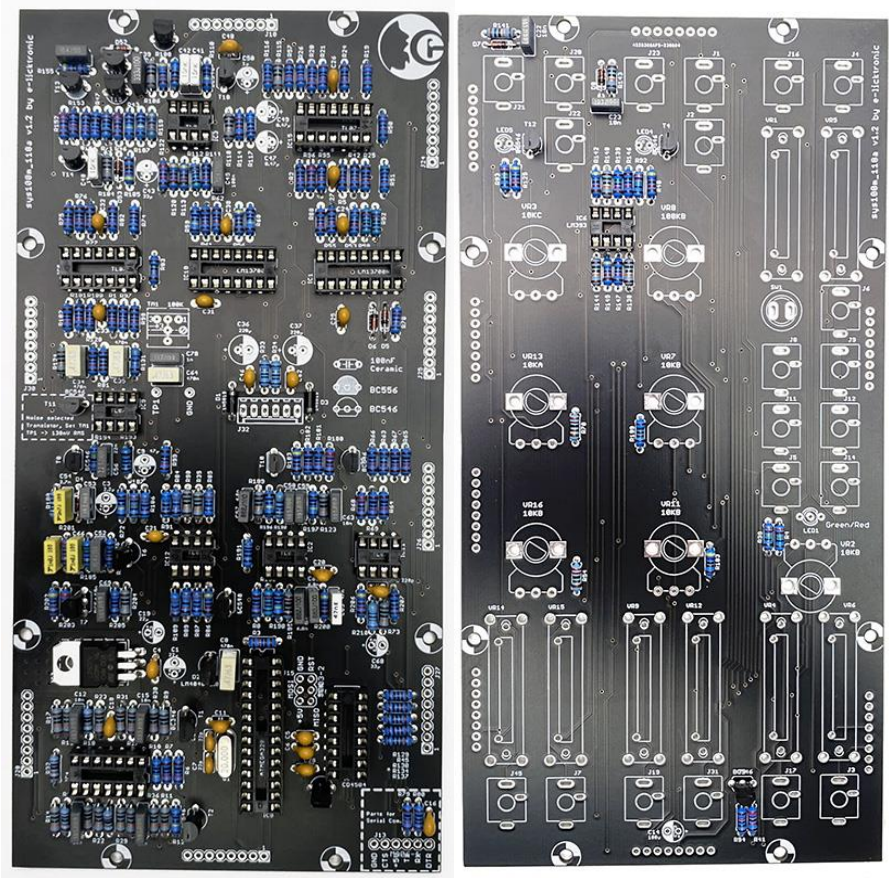


It is time to solder the two BC556 PNP transistor T10 and T13
Warning: Do not heat this part to much too

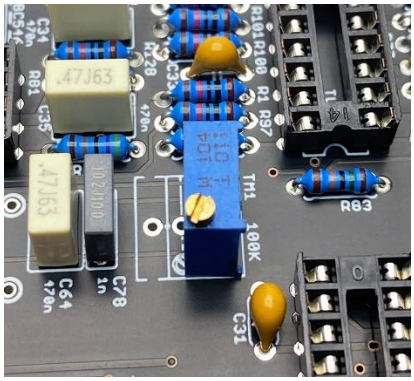




Finally, solder the sixteen BC546 NPN transistor
Warning: Do not heat this part to much

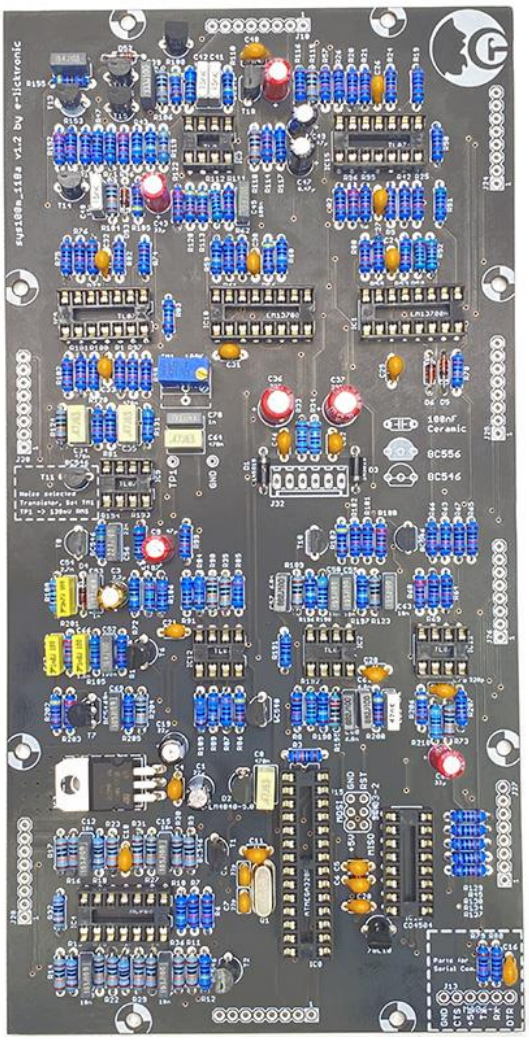


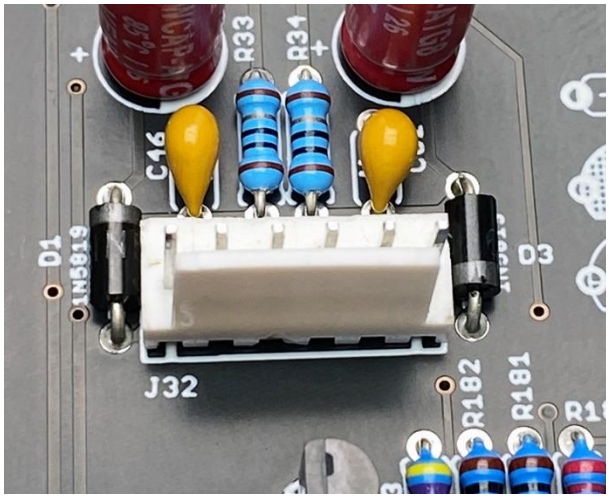
Now, solder TM1 the 100Kohms trimmers. It has to sit flat on the PCB



Next solder the electrolytic capacitors. Take care of the value and direction.

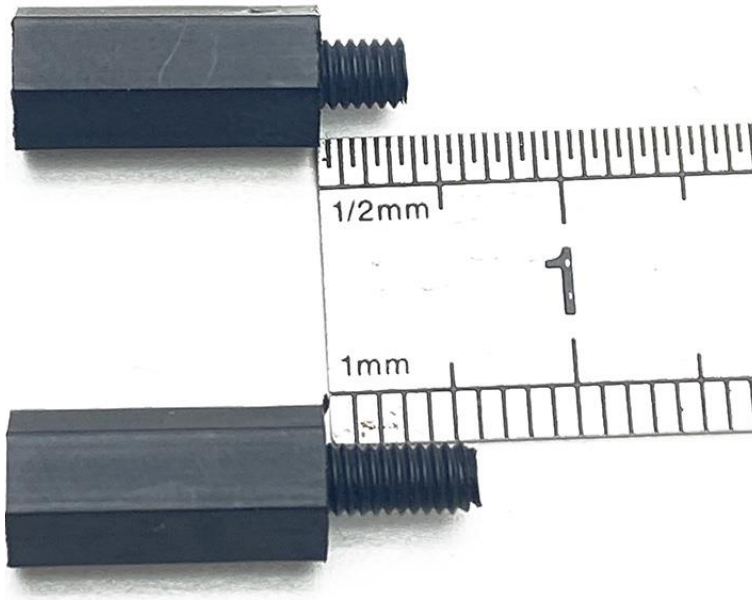
Warning: Make sure to respect the direction of the electrolytic capacitors. The negative side of the capacitor is marked by the white part on the PCB and on the capacitor and the positive side is the long leg of the capacitor. **It has to match**

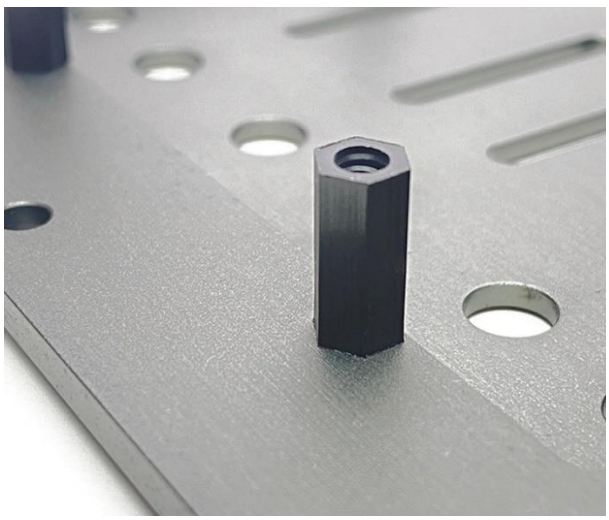




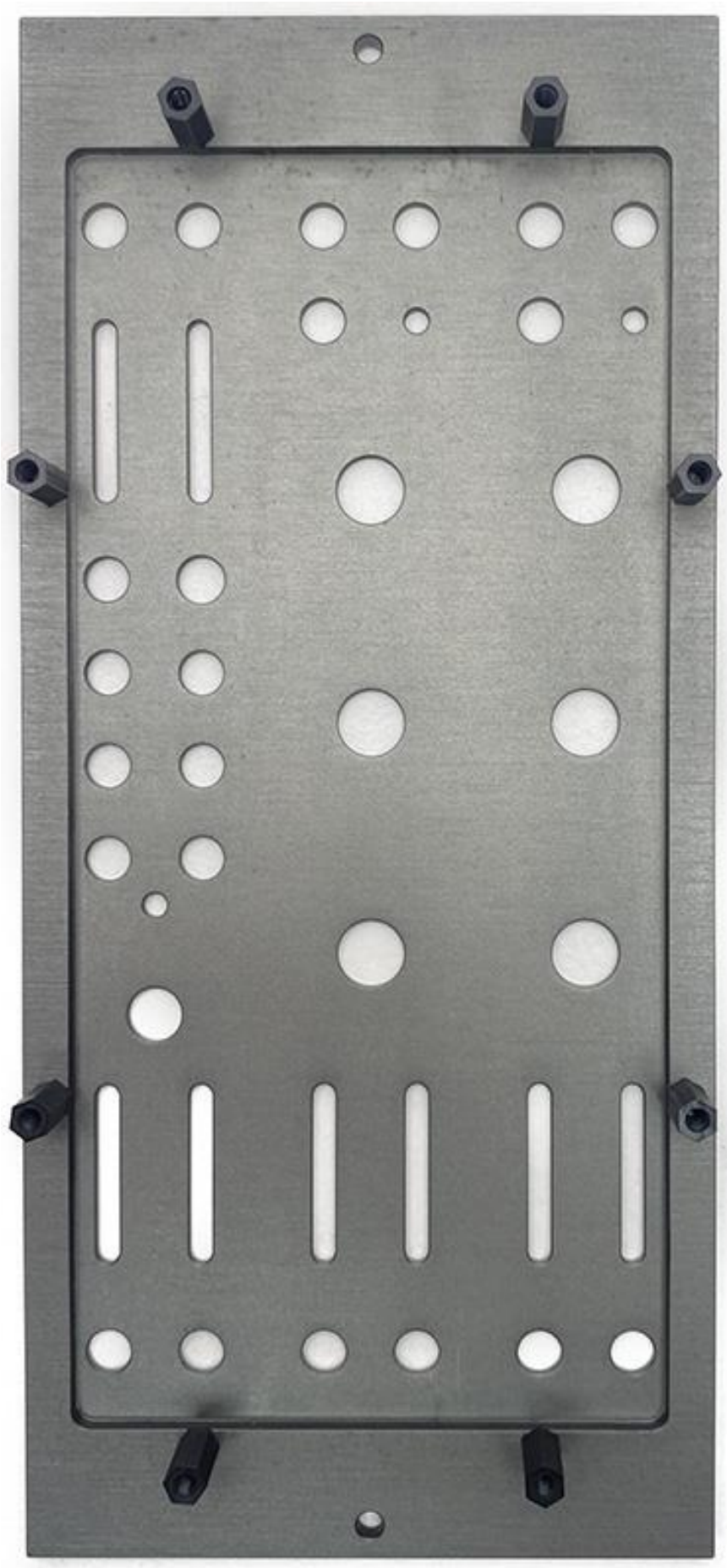
Now solder the connector.
Confirm they sit tight and square.
Solder the first pin only, then heat the pin and press on the connector to sit tight and square.

Time to prepare the faceplate.
Shorten the eight 12mm nylon spacer threaded part to 4mm with a cutter.
Warning: Do not mix 12mm and 11mm nylon spacer

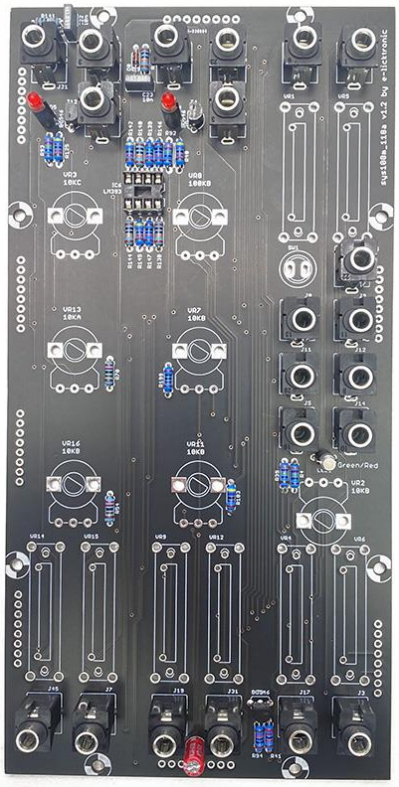




Mount the eight shorten spacers on the faceplate. Has to sit flat

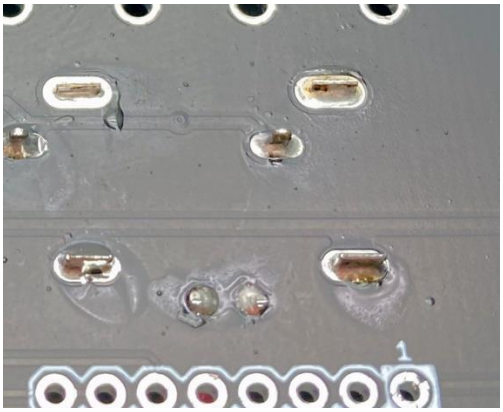


Mount the 21x jack on the faceplate. Flip the board and make it flat as possible on a table then solder only one pin of each jack, the one on the picture.



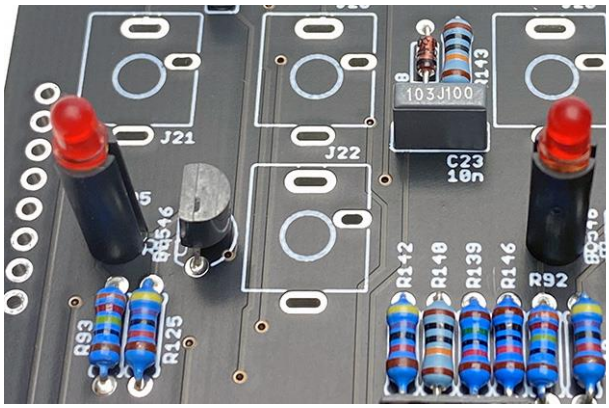
Mount the board on the faceplate using the 30mm spacer as screw (four is enough to hold the pcb) and take your time to **perfectly** align jack with the faceplate.

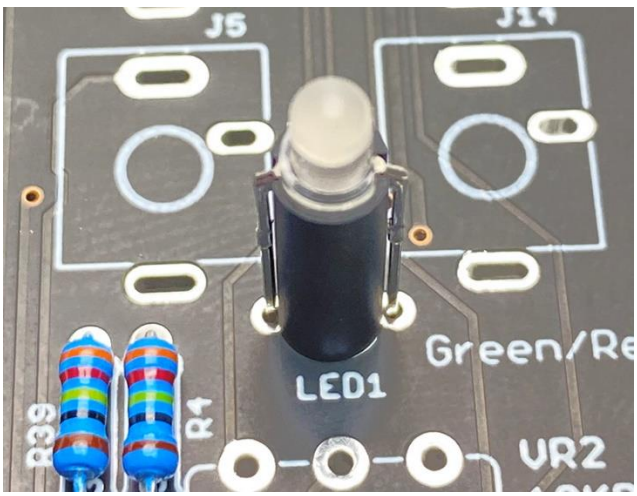
Once it is done, solder the remaining pins of the jack



Next you can install the LEDs. First, mount the LEDs on the nylon spacers

Insert the LEDs on PCB and solder one pin of each LED
Warning: Take care of LEDs orientation. The LED flat part has to match the LED silkscreen



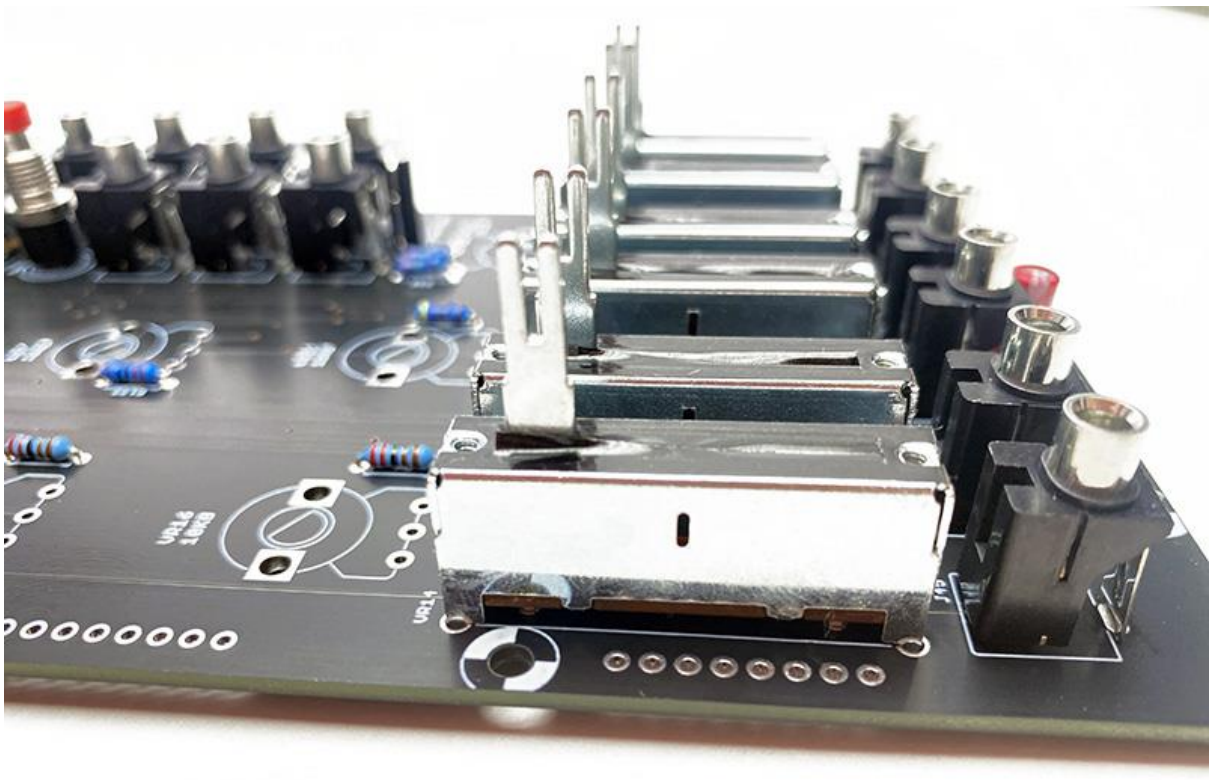


Mount the pcb on the faceplate using 30mm spacer as screw. Check that led is aligned with the faceplate and sit flat on the pcb then solder the remaining pins

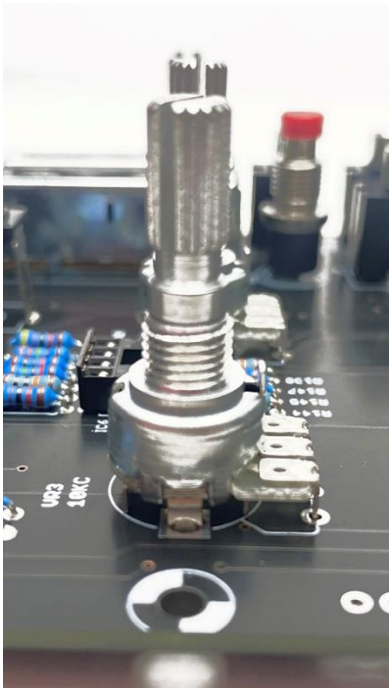
Mount the SW1 push button on the pcb, solder only one pin and check that it sits flat and align on the faceplate. Then solder the second pin

Warning: Do not hot to much the push button pins, it will destroy it

It is time to mount the 10KB 20mm faders on the PCB. Make it sit flat, it is very important. **The leveler has to be square**. solder one pin and check if it matches the faceplate then solder the other pin.

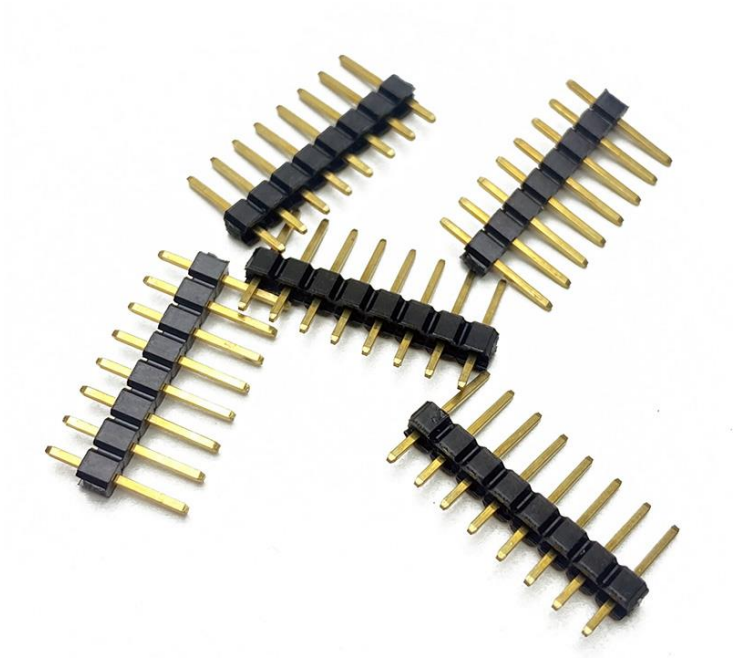


Then solder the potentiometer. **Do not mix values.** The potentiometers value is written on the back of the potentiometer. VR3 is 10KC (**C103**), VR8 is 100KB (**B104**), VR13 is 10KA (**A103**) and the others are 10KB (**B103**). On the picture you can see a 100KB potentiometer

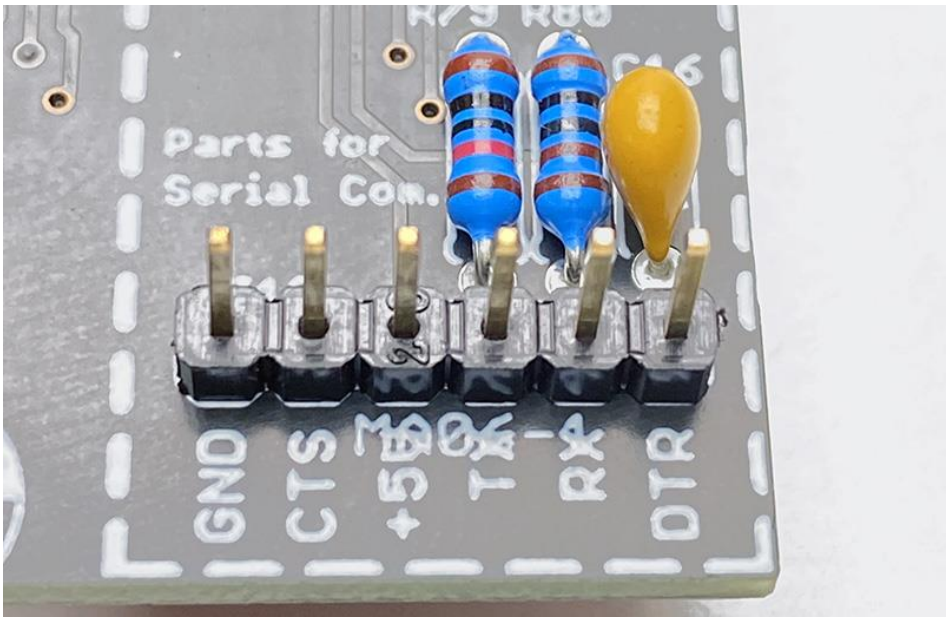


To insert the potentiometer, you must to **press a little bit**. It can help to bend the two big pins. It has to sit flat one the PCB. Solder one pin and check that it is aligned with the faceplate, then solder the remaining pins

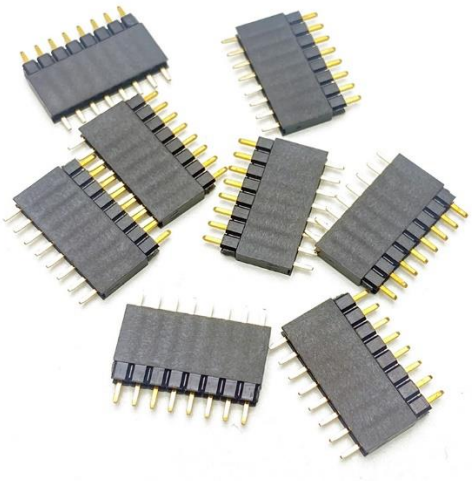
It is time to prepare the “sandwich” 😉.
Cut the both 40pins male header to get 8 pieces of 8 pins and one piece of 6 pins (keep the remaining part)

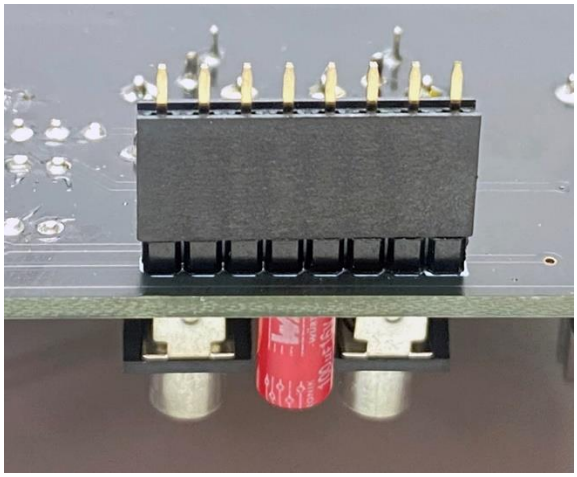


Take the 6 pins male header and solder J13 on the component PCB
This connector is used to program the microprocessor with a USB/COM port converter. It allows you to customize the sequencer of the M-118a Module



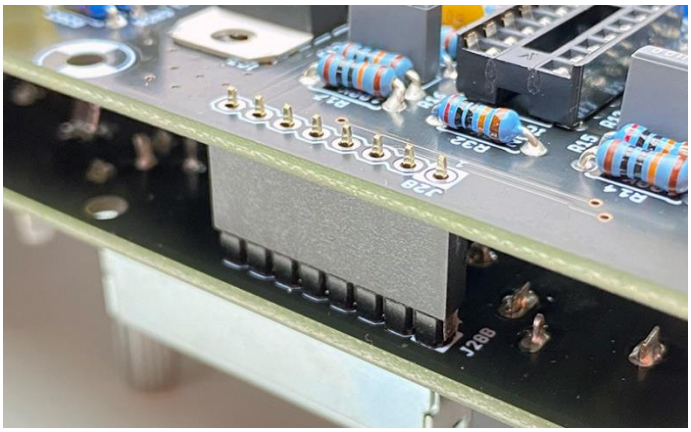
Now, connect the male header with the 8 pins female header

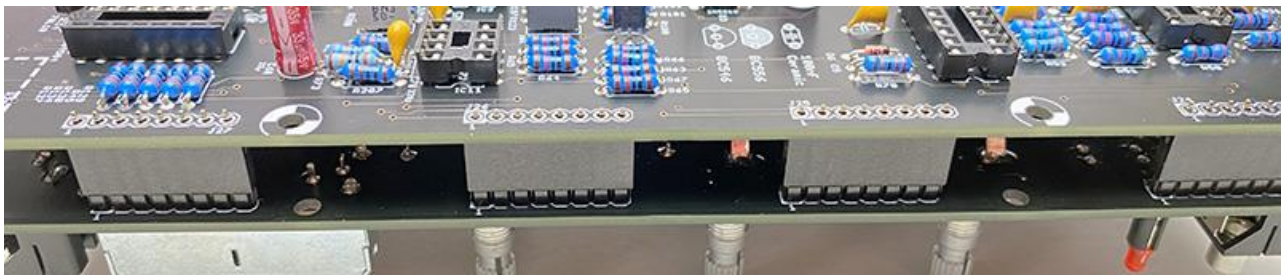




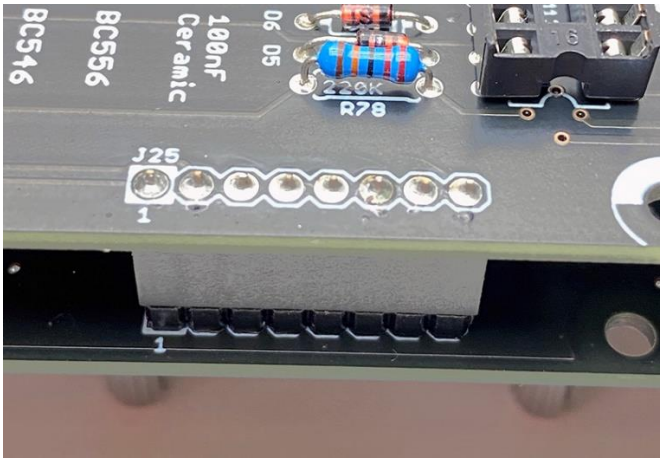
Insert the eight headers on the potentiometer PCB, the male header has to sit the PCB (see picture)

Now, the trickiest part 😉, insert the component PCB on the female headers to make the “sandwich”.
TIPS*:* *Start by the long side where there are four headers*

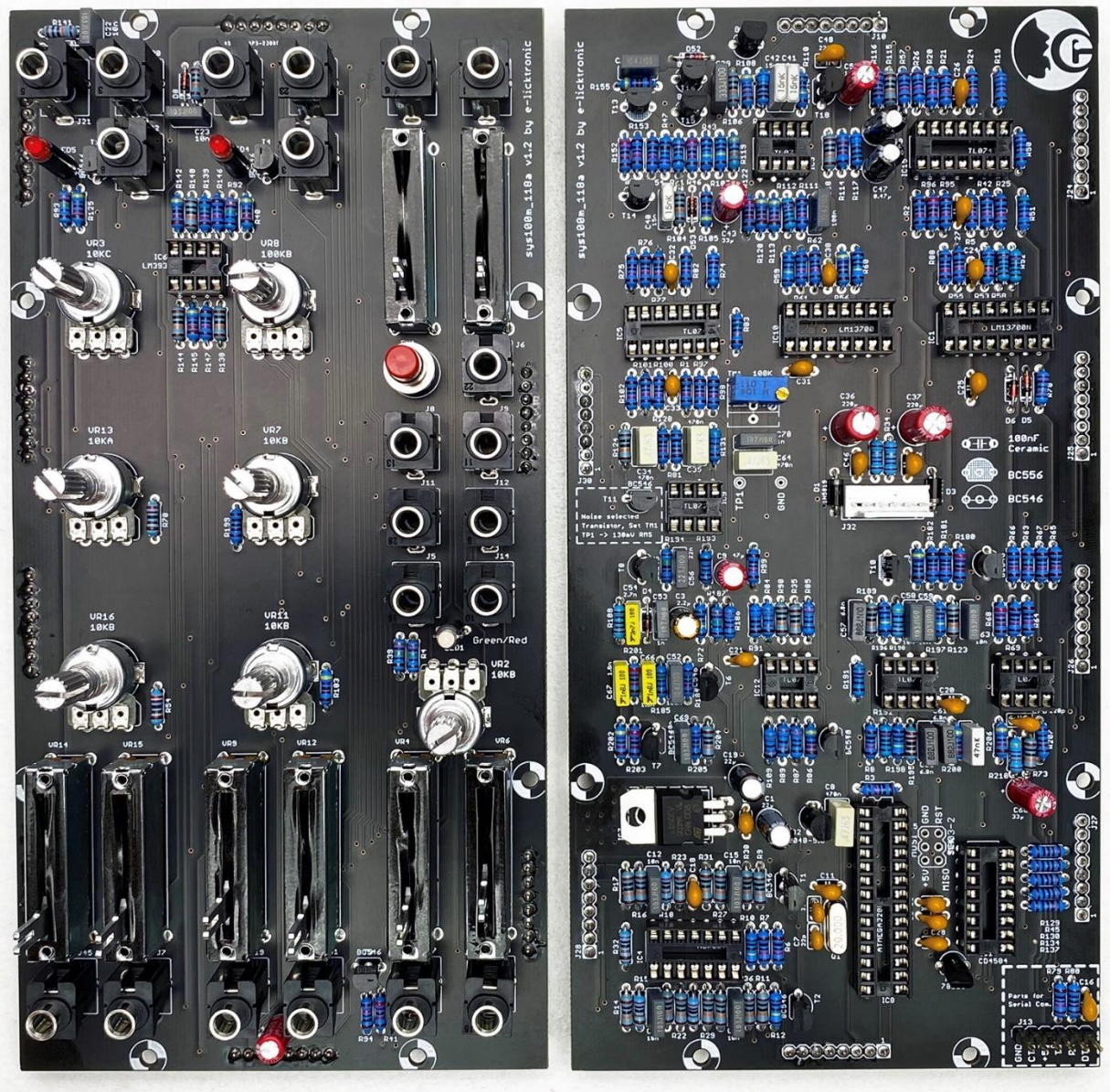




Solder all female headers on the components PCB



Then flip the “sandwich” and solder all male headers on the potentiometer PCB. Now everything is solder on the PCBs



MOUNT DIN CONNECTOR

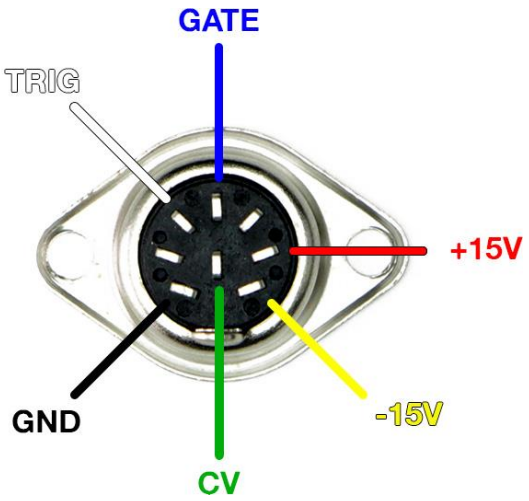
Take the 6 pins cables, then strip off a 5mm piece of insulation on these wires and twist and add solder.

Cut 6 pieces of 10mm heat shrink tubing

Put a piece of shrink tube on the green wire.
Make a hook with the end of the **GREEN** wire.

Solder this hook on the **center pin**.
Warning: Do not heat the pin to much it will burn the isolation between pins.
Put the shrink tube over the pin and heat it.

Tips and tricks: *Glue a piece of double-sided tape on your table and glue the din connector to hold it*



Repeat this operation for each wire. Here is wire color and corresponding pin. **Don’t forget the shrink tube**

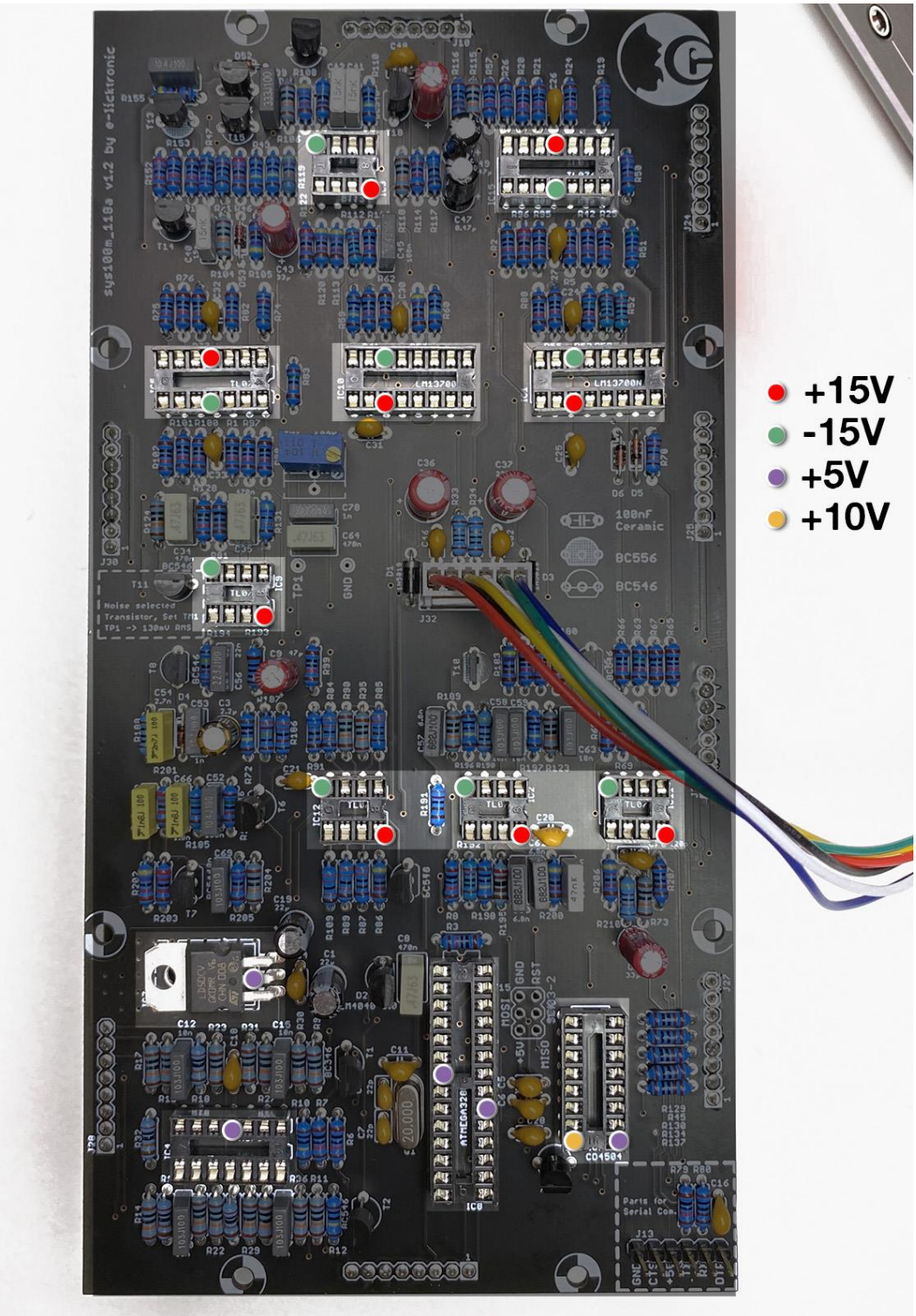
TESTING AND SETTING TM1

Now it is time to test if everything works properly.
Connect the Din8 pin cable to your System 100M case and connect the 6Pins cable to the PCB.

Power up your system 100M case.

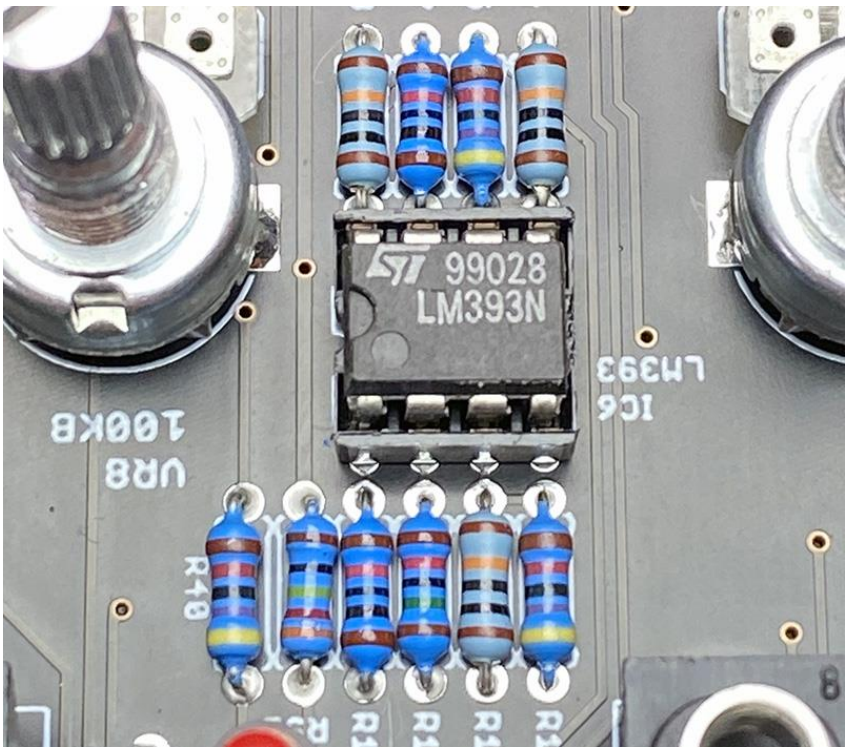
Warning: If “magic smoke” appears, shut down the power immediately

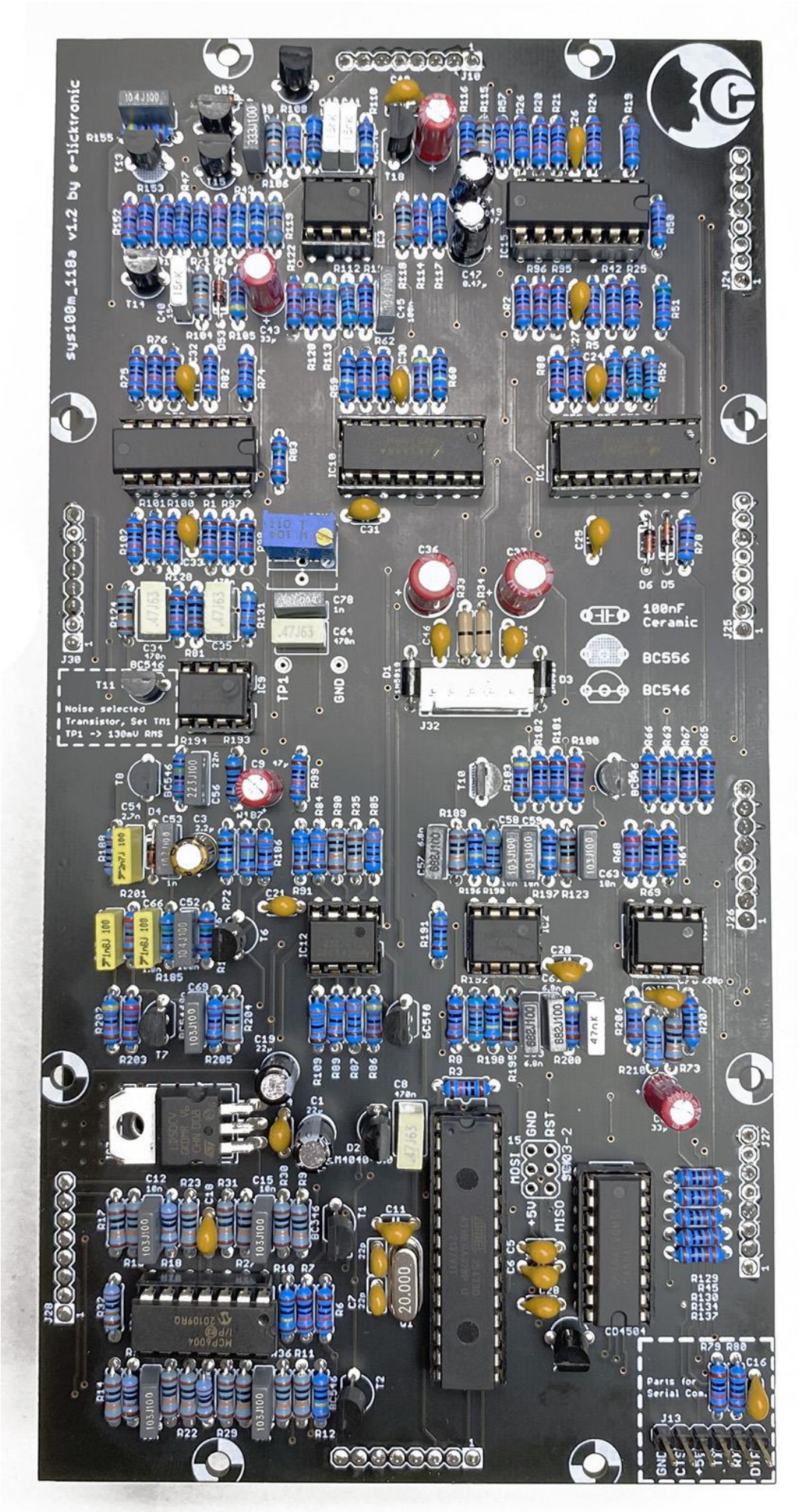
Set your digital multimeter to DC volt on a 20V scale and test the following point. Black probe on J15 GND. You should find +15V more or less 0.5V, -15V more or less 0.5V and +5V more or less 0.1V



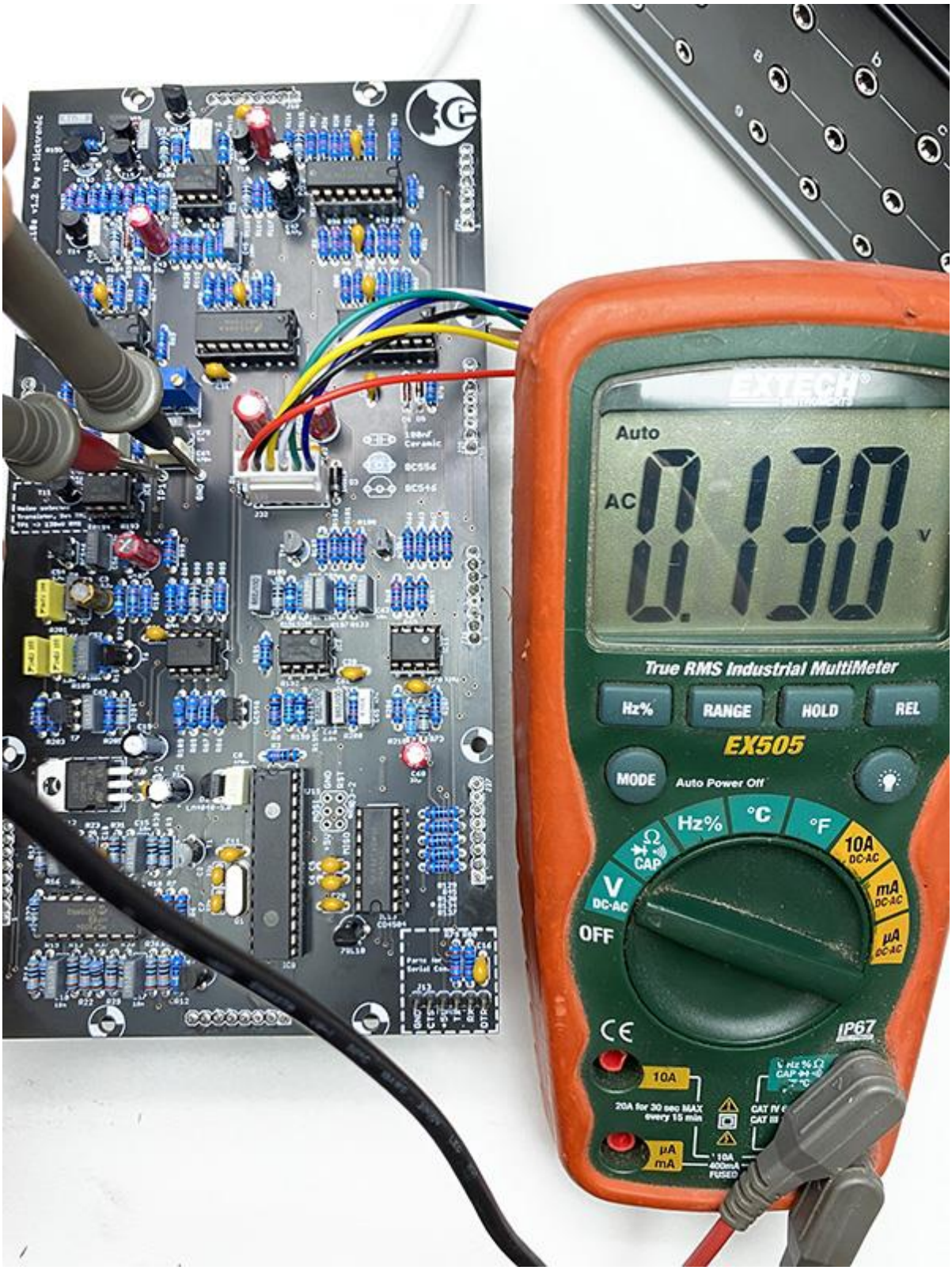
If you don’t find those value, first make a visual check of your solder, then check also part value and orientation.

It is time to mount all the integrated circuits.
If everything thing is ok, you can mount all the integrated circuits in sockets.
Use the **iBOM** to help you to place the ICs
Warning: Make sure to insert the ICs in the correct direction which is identified by a notch




sys100m_118a v1.2 by e-licktronic
100nF Ceramic
BC556
BC546
Noise selected Transistor, Set TM1 TP1 -> 130mV RMS
TP1
GND
MOSI
MISO
+5V
GND
RST
20.000
Parts for Serial Com.

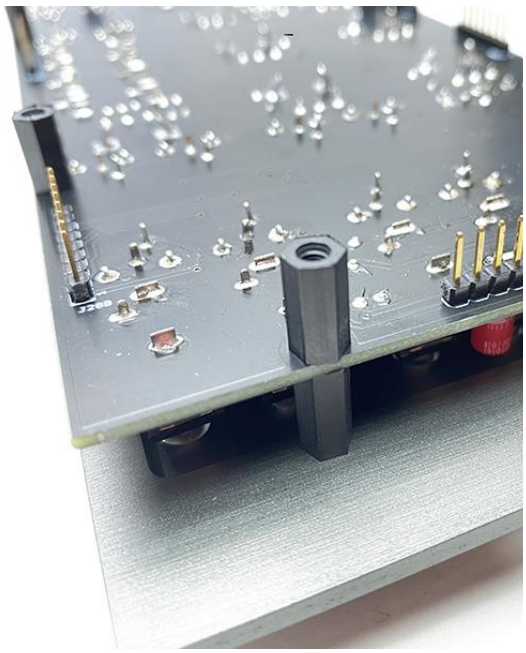
Connect the module to your system 100M case again.
Set your digital multimeter to AC volt on a 1V scale. Black probe on GND and red probe on TP1. Adjust TM1 to get 130mV.



FINISHING

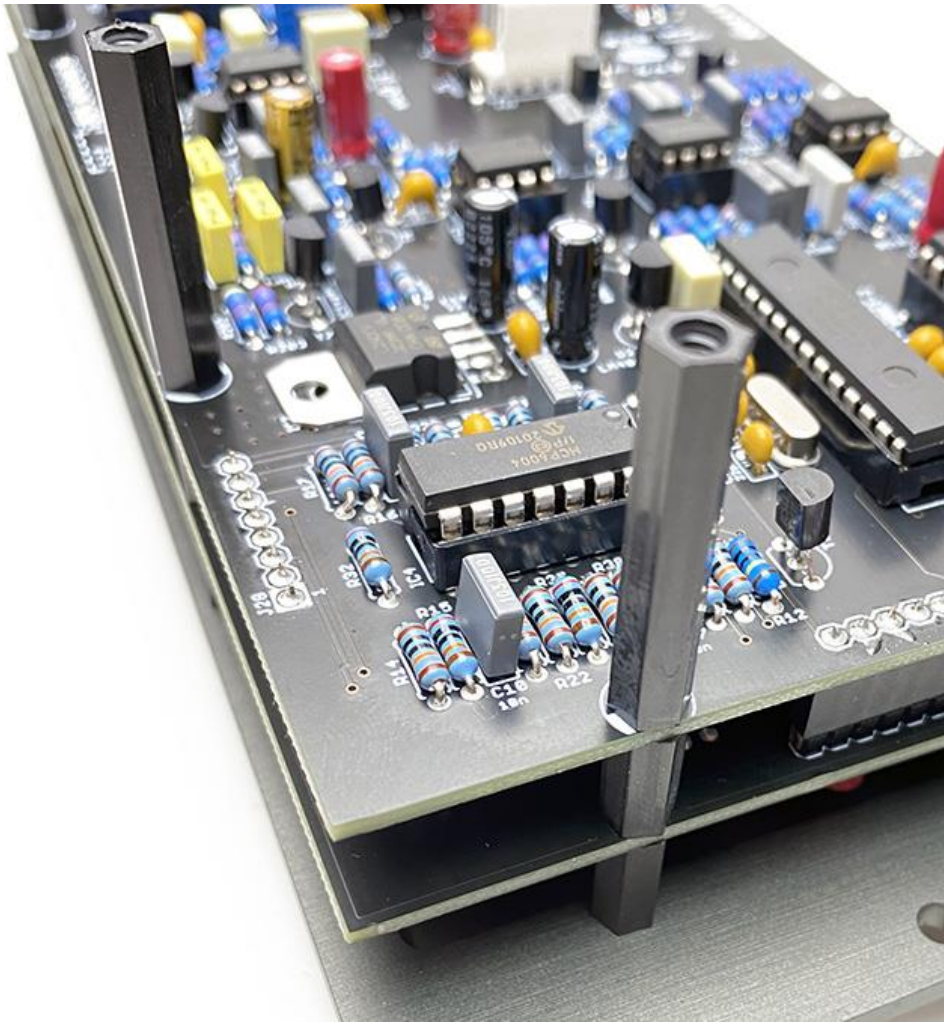
It is time to mount everything together

First, take two M3x6mm black Pan head pozi machine screw with the two M3 nuts and screw the din8 connector to the back panel



Take eight 11mm nylon spacer and screw the potentiometer PCB on the faceplate

Connect both PCB together and screw the second PCB with eight 30mm nylon spacer



Screw the back panel on the 30mm nylon spacer using eight M3x6mm black Pan head pozi machine screw. Don’t forget to connect the 6pins cable to J32

MODEL: M-118a
SER. NO.
REGULATED DC +15V 80 mA MAX
REGULATED DC -15V 40 mA MAX
UNREGULATED DC +22V 0 mA MAX
e-licktronic
MADE IN

Last thing to do, add knobs and slider cap and congratulation!

YOU ARE DONE AND OWNER OF A BRAND NEWS SYSTEM100M 118a MODULE. CONGRATULATION!

(don’t forget to write on the back sticker the country and serial number with a marker)