

MODULE ASSEMBLY APPENDIX

Before we start building, let's take a look at the complete **mki x es.edu Snare Drum** schematics (see next page) that were used for the final module's design and PCB fabrication. Most components on the production schematics have denominations (a name – like R1, C1, VT1, VD1, etc.) and values next to them. Denominations help identify each component on the PCB, which is particularly useful during **calibration**, **modification** or **troubleshooting**.

XS1 is the **Accent input** jack socket; it requires +5V gate signal to initiate the accent. **XS2** is the **Tune CV input** jack socket, **XS3** is the **Snappy CV input**, **XS4** is the **Trigger input** jack socket and **XS5** is the **Audio output** jack socket – these are the very same we've already been using on the breadboard for interfacing with other devices. In our designs, we use euro-rack standard 3,5mm jack sockets (part number WQP-PJ301M-12).

XP1 is a standard eurorack power connector. It's a 2x5 male pin header with a key (the black plastic shroud around the pins) to prevent accidental reverse polarity power supply connection. This is necessary because connecting the power incorrectly will permanently damage the module.

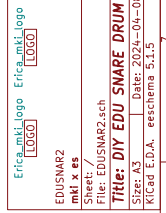
XP1 is a standard eurorack **power connector**. It's a 2x5 male pin header with a key (the black plastic shroud around the pins) to prevent accidental reverse polarity power supply connection. This is necessary because connecting the power incorrectly will permanently damage the module.

VD1 and **VD2** are **schottky diodes** that double-secure the reverse polarity power supply protection. Diodes pass current only in one direction. Because the anode of VD1 is connected to +12 V on our power header, it'll only conduct if the connector is plugged in correctly. If a negative voltage is accidentally applied to the anode of VD2, it closes, and no current passes through. The same goes for VD2, which is connected to -12 V. Because schottky diodes have a low forward voltage drop, they are the most efficient choice for applications like this.

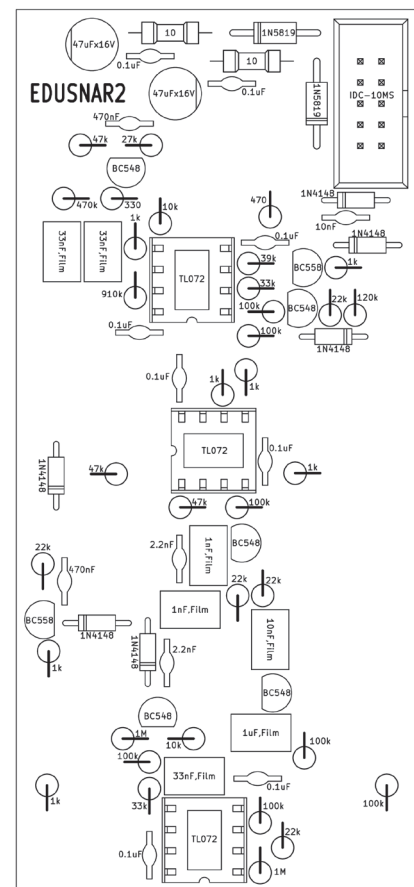
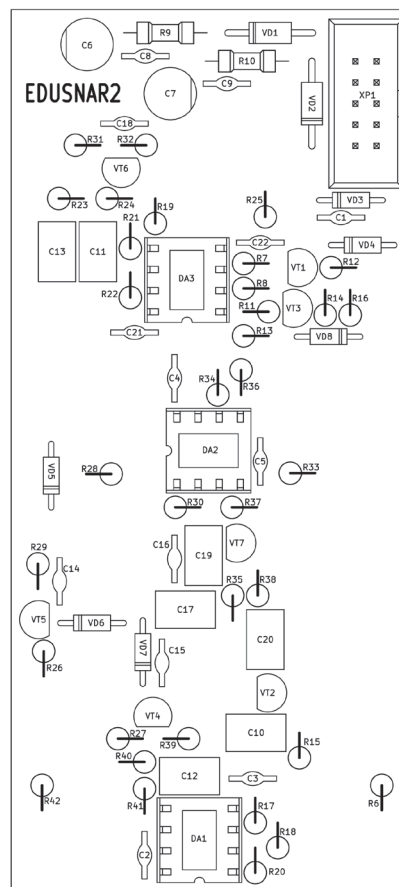
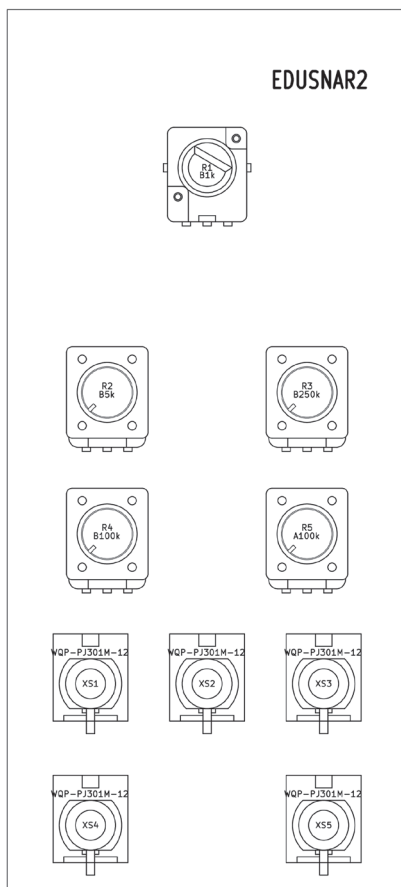
Next, we have two **10 Ohm resistors** (**R9** and **R10**) on the + and – 12 V rails, with **decoupling** (or **bypass**-) capacitors **C6 – C9**. These capacitors serve as energy reservoirs that keep the module's internal supply voltages stable in case there are any fluctuations in the power supply of the entire modular system. In combination with R9 and R10, the large 47 microfarad pair (C8 and C9) compensates for low frequency fluctuations, while C6 and C7 filter out radio frequencies, high frequency spikes from a switching power supplies and quick spikes created by other modules. Often another component – a **ferrite bead** – is used instead of a 10 Ohm resistor and there's no clear consensus among electronic designers which works best, but generally for analogue modules that work mostly in the audio frequency range (as opposed to digital ones that use microcontrollers running at 8 MHz frequencies and above), resistors are considered to be superior.

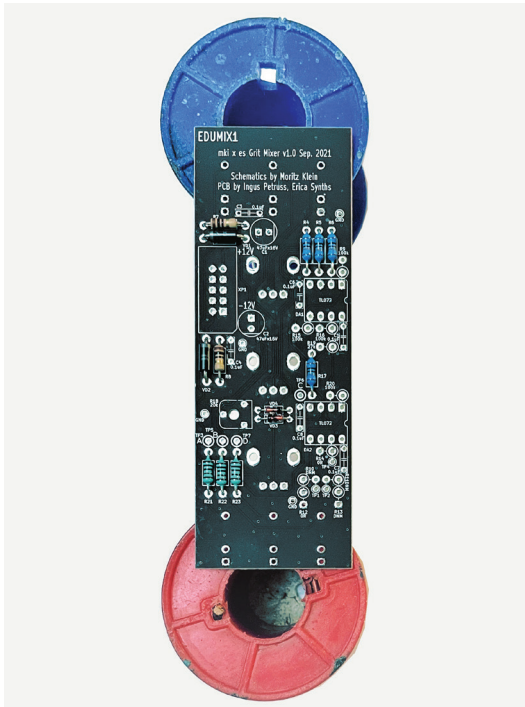
Another advantage of 10 Ohm resistors is that they will act like **slow “fuses”** in case there's an accidental short circuit somewhere on the PCB, or an integrated circuit (IC) is inserted backwards into a DIP socket. The resistor will get hot, begin smoking and finally break the connection. Even though they aren't really fuses, just having them there as fuse substitutes is pretty useful - **you'd rather lose a cent on a destroyed resistor than a few euros on destroyed ICs.**

Capacitors **C2 – C5 and C21, C22** are additional decoupling capacitors. If you inspect the PCB, you'll see that these are placed as close to the power supply pins of the ICs as possible. For well-designed, larger PCBs you will find decoupling capacitors next to each IC. Like the others, their job is to simply compensate for any unwanted noise in the supply rails. If the input voltage drops, then these capacitors will be able to bridge the gap to keep the voltage at the IC stable. And vice-versa - if the voltage increases, then they'll be able to absorb the excess energy trying to flow through to the IC, which again keeps the voltage stable. Typically, 0.1 uF capacitors are used for this purpose.

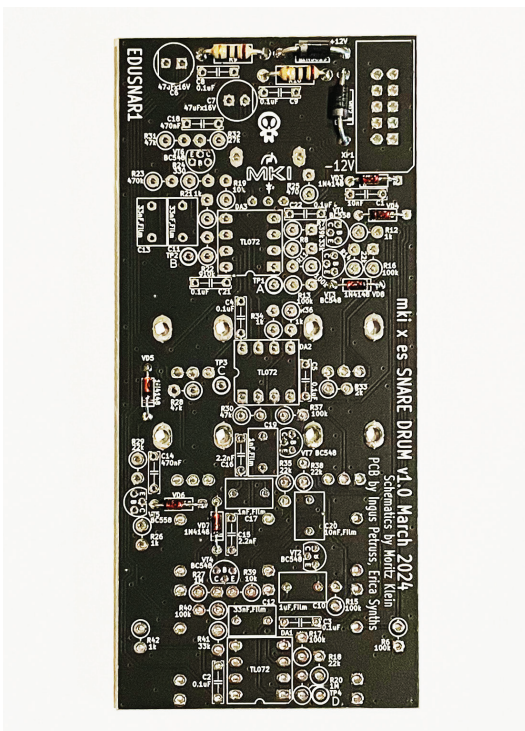


Before you start soldering, we highly recommend printing out the following part placement diagrams with designators and values. Because some of our PCBs are rather densely populated, this will help you to avoid mistakes in the build process.

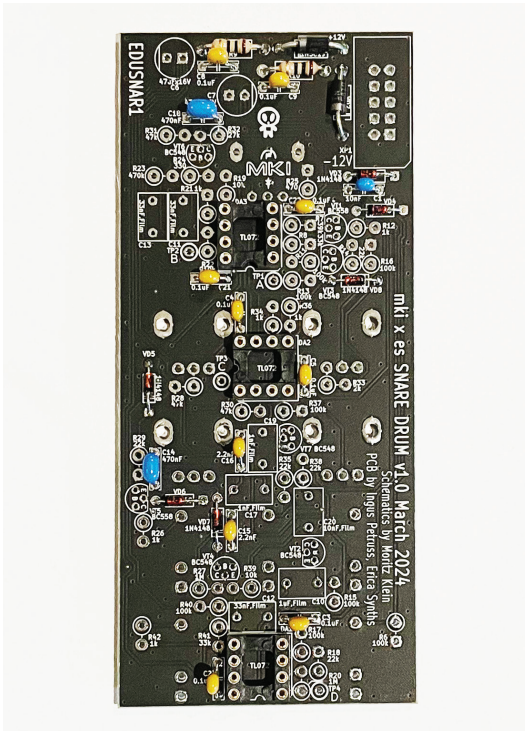




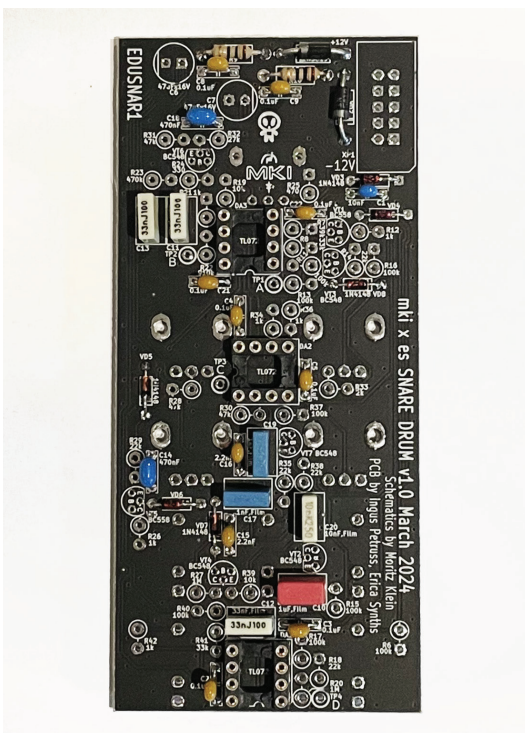
Place the Snare Drum PCB in a PCB holder for soldering or simply on top of some spacers (I use two empty solder wire coils here).



I usually start populating PCBs with lower, horizontally placed components. In this case, these are **10 Ohm resistors**, **switching diodes** and the **power protection diodes**. Bend the component leads and insert them in the relevant places according to the part placement diagram above. All components on the PCB have both their value and denomination printed onto the silkscreen. If you are not sure about a resistor's value, use a multimeter to double-check. Next, insert the diodes. Remember – **when inserting the diodes, orientation is critical!** A thick white stripe on the PCB indicates the cathode of a diode – match it with the stripe on the component. Flip the PCB over and solder all components. Then, use pliers to cut off the excess leads.

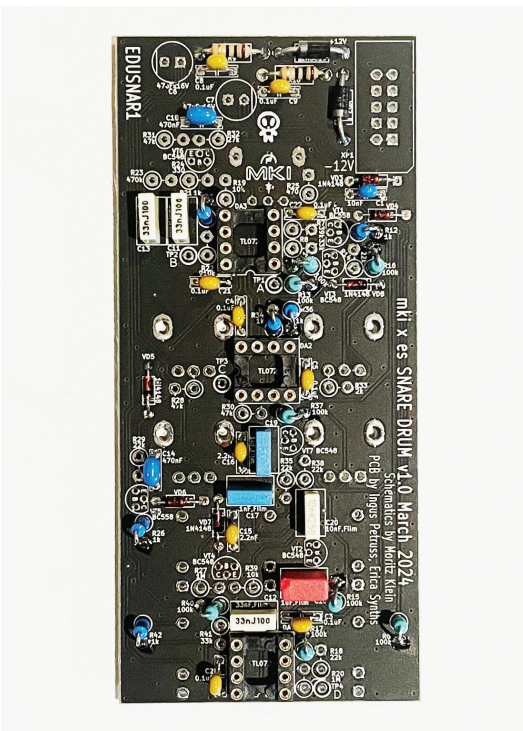
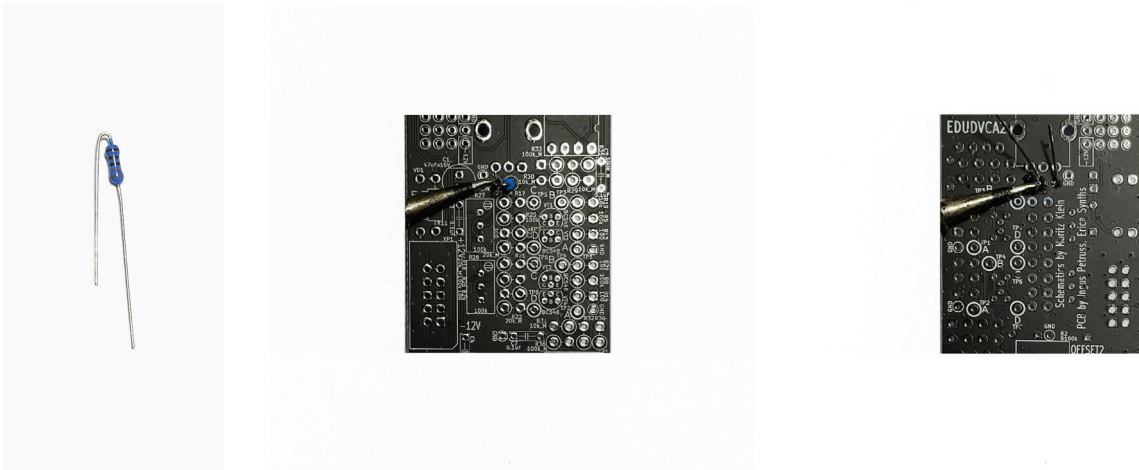


Next, insert the first **DIP socket**, hold it in place and solder one of the pins. Continue with the **next DIP socket**. Make sure the DIP sockets **are oriented correctly** – the notch on the socket should match the notch on the PCB's silkscreen. Now, turn the PCB around and solder all remaining pins of the DIP sockets. Then insert and solder **the ceramic capacitors**.

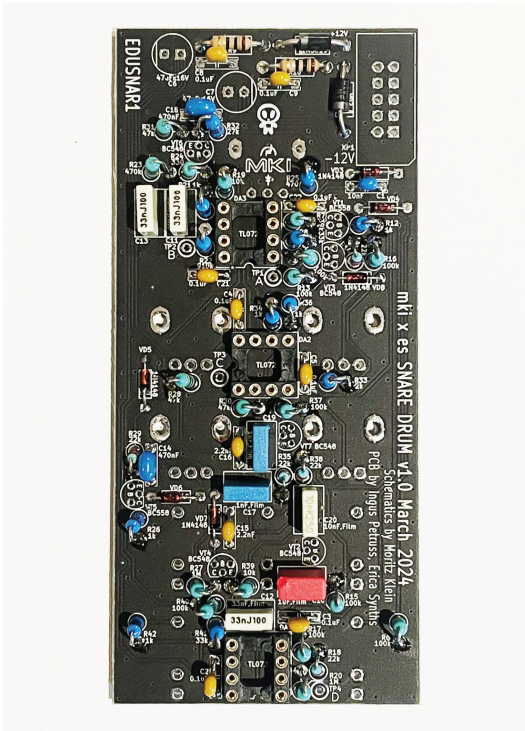


Then proceed with **the film capacitors**. Place the PCB in your PCB holder or on spacers, insert the capacitors and solder them like you did with the resistors & diodes before. Now your PCB should look like this:

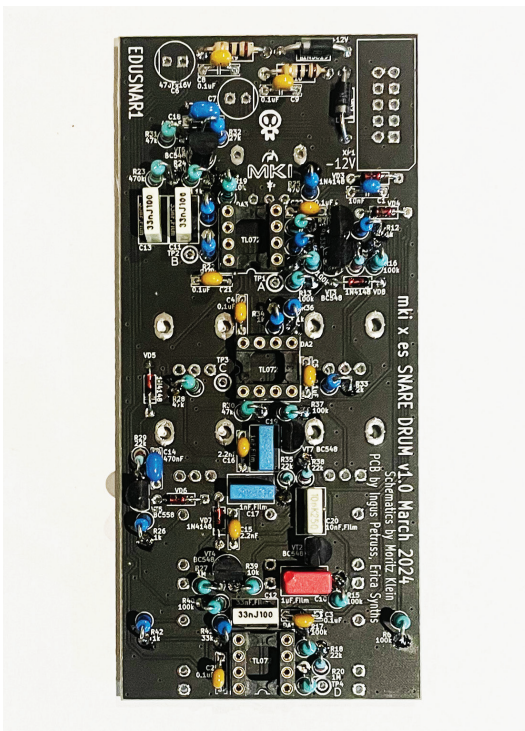
In order to save space on the PCB, some of our projects, including the Snare Drum, have vertically placed resistors. The next step is to place & solder those. Bend a resistor's legs so that its body is aligned with both legs and insert it in its designated spot. Then solder the longer lead from the top side of the PCB to secure it in place, turn the PCB around and solder the other lead from the bottom. You can insert several resistors at once. Once done with soldering, use pliers to cut off excess leads.



Because the Snare drum features quite few resistors, let's start with **100 k resistors** (light blue on the photo to the left) and **1 k resistors** (dark blue on the photo to the left). When those are installed, the PCB should look like this:



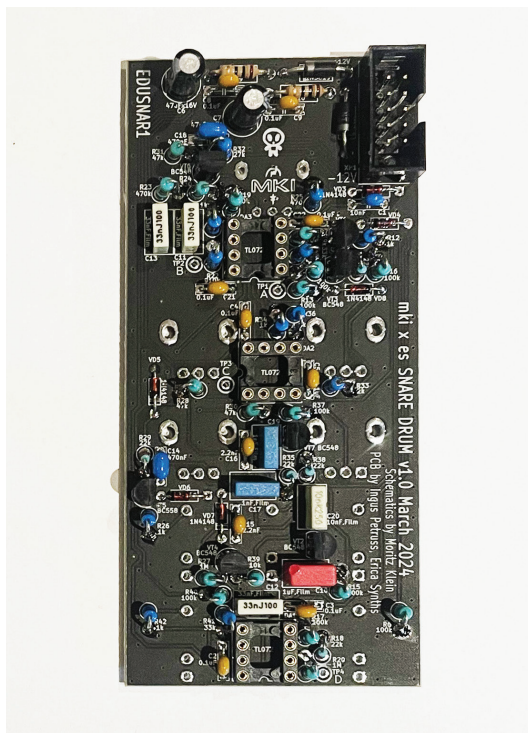
Now, proceed with other vertically placed resistors. If you are not sure about resistor value, use the multimeter to measure resistance of each resistor before soldering them. Once you have completed installing the resistors, your PCB should look like this:



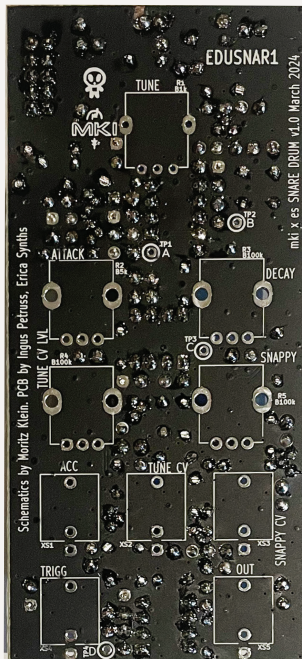
Next, insert and solder **transistors**. There are PNP and NPN transistors in the kit, make sure you install them in correct places and pay attention on the orientation of the transistors – notch on the silkscreen has to match the flat part of the transistor.



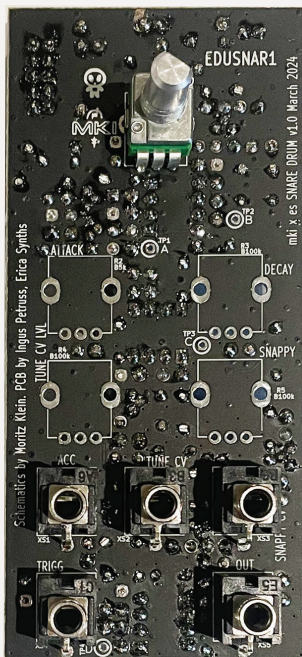
Also, **insert & solder the electrolytic capacitors**. Electrolytic capacitors are bipolar, and you need to mind their orientation. The positive lead of each electrolytic capacitor is longer, and there is a minus stripe on the side of the capacitor's body to indicate the negative lead. On our PCBs, the positive pad for the capacitor has a square shape, and the negative lead should go into the pad next to the notch on the silkscreen.



Then complete the component side of the VCF PCB by soldering the **PSU socket**. Make sure the orientation of the socket is as shown in the picture below – the arrow pointing to the first pin is aligned with a notch on the silkscreen. The key on the socket will be facing outwards the PCB. Now your PCB should look like this:

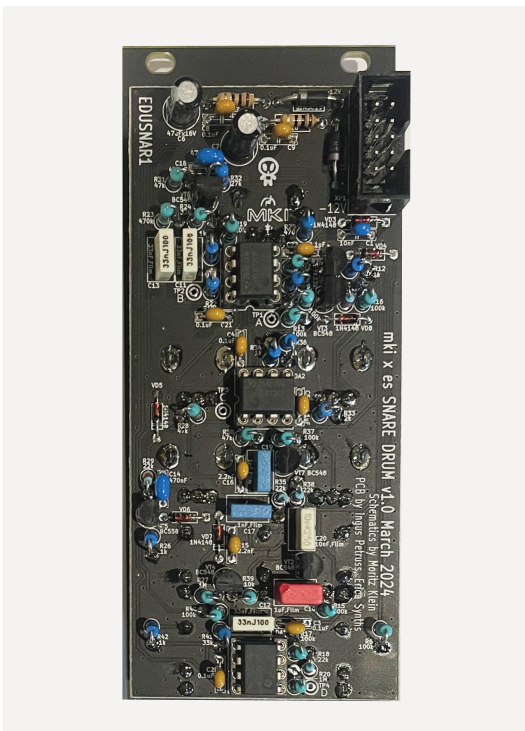
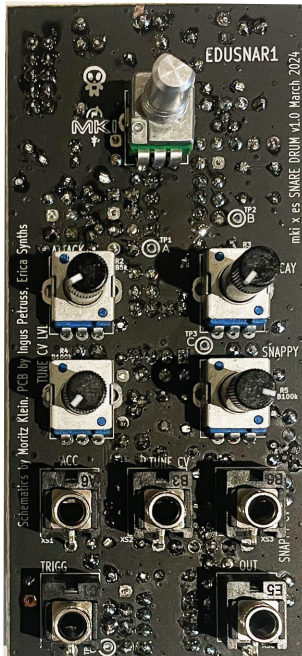


Now, turn the PCB around and inspect your solder joints. **Make sure all components are soldered properly and there are no cold solder joints or accidental shorts.** Clean the PCB to remove extra flux, if necessary.



Insert the top potentiometer and jack sockets and solder them.

Insert other potentiometers, but don't solder them yet! Fit the front panel and make sure that the potentiometer shafts are aligned with the holes in the panel – and that they're able to rotate freely. Now, go ahead and solder the potentiometers.



Now, insert the ICs into their respective **DIP sockets**. Mind the orientation of the ICs – match the notch on each IC with the one on its socket.



Finally fit the Tune potentiometer knob and we are done!

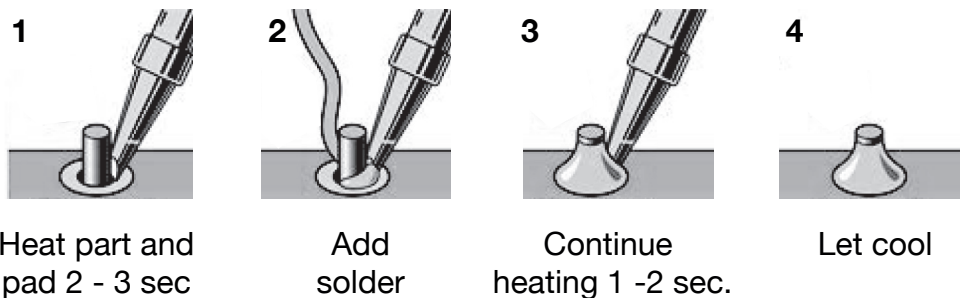
Congratulations! **You have completed the assembly of the mki x es.edu Snare Drum module!** Connect it to your eurorack power supply and switch it on. If there's no "magic smoke", it's a good sign that your build was successful. The module doesn't need any calibration. Patch trigger signal (the gate output of your DIY.EDU Sequencer will be the best choice) to the input of the module and connect the output of the module to a mixer. You should hear the kick drum sound. Turn gates on the sequencer on and off in order to achieve a desired kick drum pattern and tweak some knobs on the module to observe change of the sound.

Enjoy!

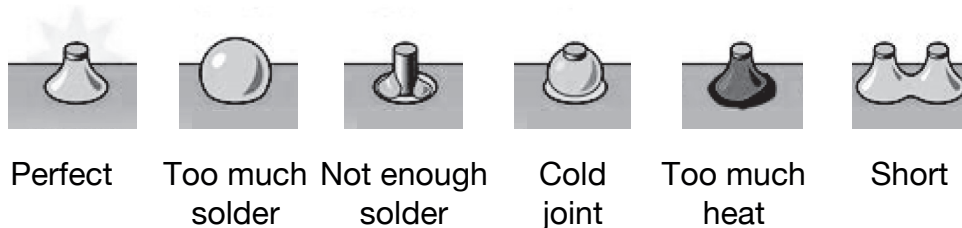
SOLDERING APPENDIX

If you've never soldered before – or if your skills have become rusty – it's probably wise to check out some **THT** (through-hole technology) **soldering tutorials on YouTube**. The main thing you have to remember while soldering is that melted solder will flow towards higher temperature areas. So you need to make sure you apply equal heat to the component you are soldering and the solder pad on the PCB. The pad will typically absorb more heat (especially ground-connected pads which have more thermal mass), so keep your soldering iron closer to the pad on the PCB. It's critically important to dial in the right temperature on your soldering station. I found that about 320 °C is the optimal temperature for most of parts, while for larger elements like potentiometers and sockets, you may want to increase that temperature to **370 °C**.

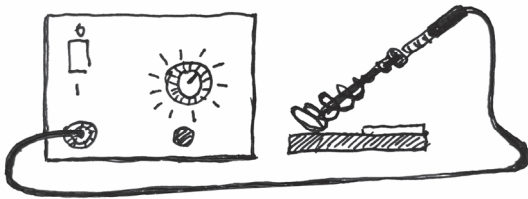
Here's the recommended soldering sequence:



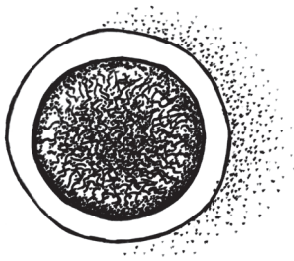
After you have completed soldering, inspect the solder joint:



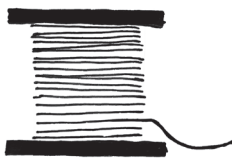
DIY electronics is a great (and quite addictive) hobby, therefore we highly recommend you invest in good tools. In order to really enjoy soldering, you'll need:



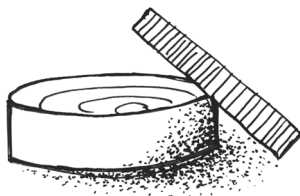
A decent soldering station. Top-of-the-line soldering stations (brands like Weller) will cost 200€ and above, but cheaper alternatives around 50€ are often good enough. Make sure your soldering station of choice comes with multiple differently-sized soldering iron tips. The most useful ones for DIY electronics are flat, 2mm wide tips.



When heated up, the tips of soldering irons tend to oxidize. As a result, solder won't stick to them, so you'll need to clean your tip frequently. Most soldering stations come with a **damp sponge for cleaning the iron tips** – but there are also professional solder tip cleaners with **golden curls** (not really gold, so not as expensive as it sounds). These work much better because they do not cool down the iron.



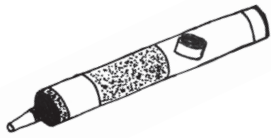
Solder wire with flux. I find 0,7mm solder wire works best for DIY projects.



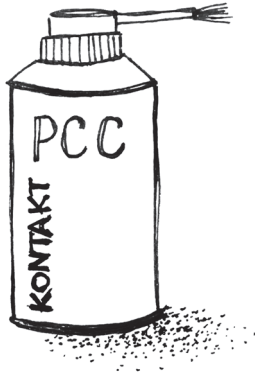
Some **soldering flux** paste or pen will be useful as well.



Cutting pliers. Use them to cut off excess component leads after soldering.



A solder suction pump. No matter how refined your soldering skills are, you will make mistakes. So when you'll inevitably need to de-solder components, you will also need to remove any remaining solder from the solder pads in order to insert new components.



Once you have finished soldering your PCB, it's recommended to remove excess flux from the solder joints. **A PCB cleaner** is the best way to go.

All of these tools can be found on major electronic components retailer websites, like Mouser, Farnell and at your local electronics shops. As you work your way towards more and more advanced projects, you'll need to expand your skillset and your tool belt – but the gratification will be much greater.

“I just love the hypnosis of a single bass drum.”

– Jon Hopkins