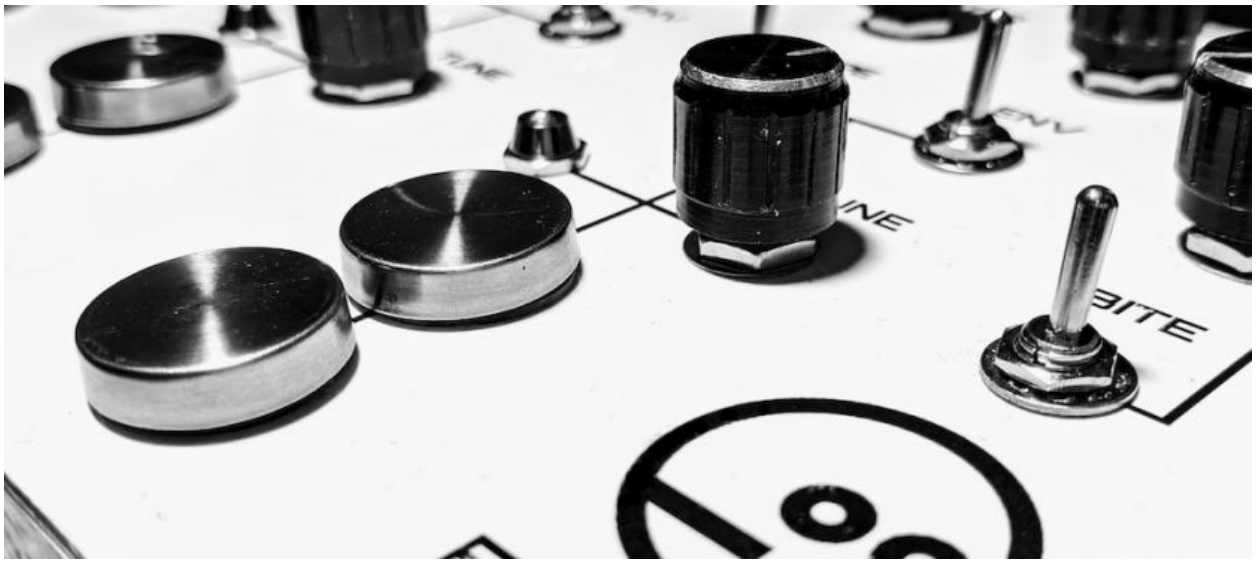
ELMYRA V1.1

BY NEUTRAL LABS



Build Guide

Congratulations on your decision to build the mighty Elmyra. Your life will change forever due to this experience. If nothing else, you'll be in possession of this wonderful machine that you likely did not possess before. Unless, of course, this is not your first build, in which case I need to tell you that you are a great person.

First of all, let's take stock of all the components that go on the PCB (and 2 that go on the panel).

D are diodes, R are resistors, C are capacitors, U is the integrated circuit and its DIP8 socket.

Bill of Materials - PCB

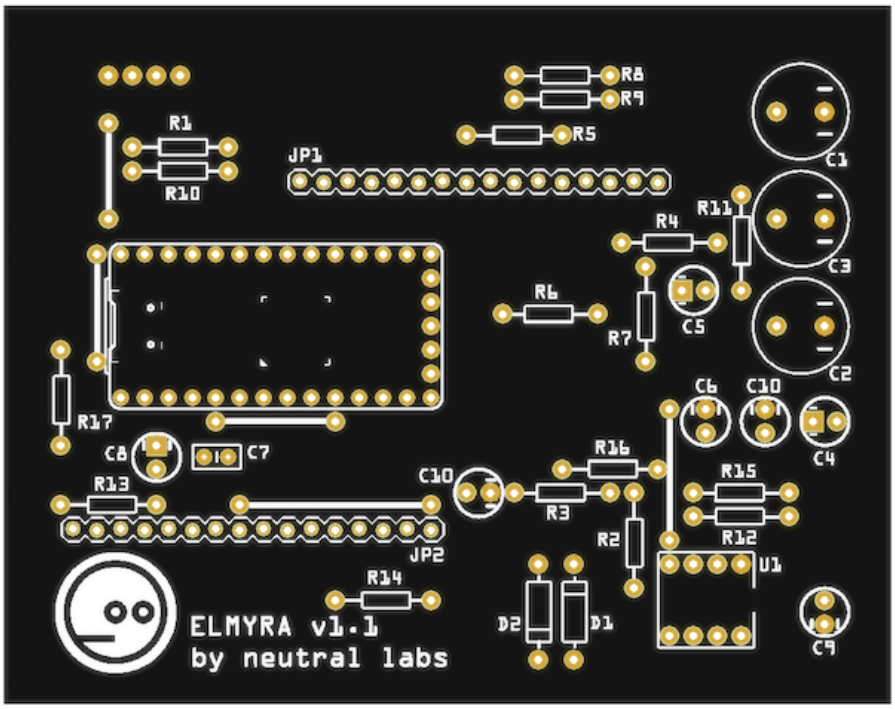
Part	Type	Polarity matters?	Notes
D1, D2	BAT85	yes	Distortion diodes. Can be substituted by BAT54. You may try other Schottky diodes as well, but regular diodes such as 1N4001 won’t have the same edge to it.
R1, R15	1 MΩ	no	
R2, R17	1 kΩ	no	
R3, R5, R7, R9	47 Ω	no	
R4, R6, R8, Rx	4.7 kΩ	no	Rx doesn’t go on the PCB, but on the panel (see below). You can substitute it for a lower value if you like the LED to be brighter. Do not go below 50 Ω.
R10, R11, R12	220 kΩ	no	You could experiment with different resistor values in case you’re using different types of touch plates. Don’t go below 100k or above 1M though.
R13, R14, R16	10 kΩ	no	
C1, C2, C3	1000 µF	yes	
C4, C5, C6	1 µF	yes	
C7	3.3 nF	no	
C8	4.7 µF	yes	
C9	10 µF	yes	

C10 (2)	1 µF	yes	There are two C10 on the PCB by mistake. One should have been C11, but it’s all good, they should both be 1 µF. ;)
Cx	0.1 µF	yes	Cx doesn’t go on the PCB, but on the panel (see below).
JP1	header	no	
JP2	header	no	
U1	MCP6002	yes	Solder the socket without IC first.
MCU	Itsybitsy M0 Express		Solder female headers to the board. Solder male headers to the MCU and place it onto the board.
4-pin header	header	no	This is unused, but you can put a header there and go from the USB power into the USB plug on the microcontroller via this header if you like. Pins 1+2 are connected, as are pins 3+4. It’s not needed though.
black lines (5)		no	These are jumpers. Use short lengths of wire across the PCB.

If polarity matters, it means you need to pay attention which way you solder the components to the PCB or panel. Diodes have a black line on them which needs to match the black line on the PCB. Electrolytic capacitors have a short leg that goes on the minus (“-”) side, and they’re also marked with a printed stripe on this side. The IC has a dot that must match the notch in the socket. And this in turn should match the gap on the PCB.

Populate the PCB

Begin by soldering the jumpers, resistors and diodes. For diodes, polarity matters!



Now solder the IC socket. It helps to solder one pin first and then ease it in place. Pay attention to polarity. Do not place the IC in the socket while soldering! Most ICs are severely allergic to heat.

Next solder all capacitors except for the 3 large ones (C1, C2, C3). Once more, polarity matters, except for C7!

Now solder the headers onto the PCB. Break the long header strip apart as needed by making a small cut on both sides with a sharp knife and then snap it along the edge of your workbench. You might use a file to remove burrs and make everything fit nicely.

Then place C1, C2 and C3 into their place and solder. I need not remind you to check the polarity, right?

Place the microcontroller into the headers and the IC in its socket. Do not use excessive pressure. If it won’t go in smoothly, check the

alignment again. Pay attention to IC polarity! Also, you may need to bend the IC's legs the tiniest bit to make it fit.

Next step: panel

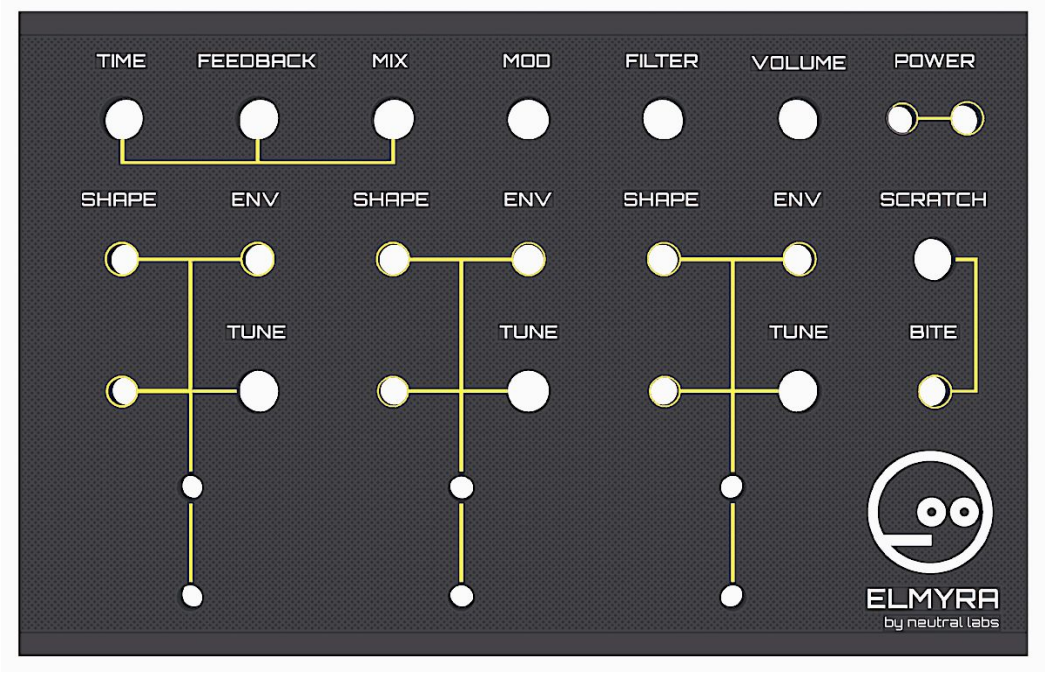
Now to place all components in their respective holes on the panel. You'll need:

Bill of Materials - Panel

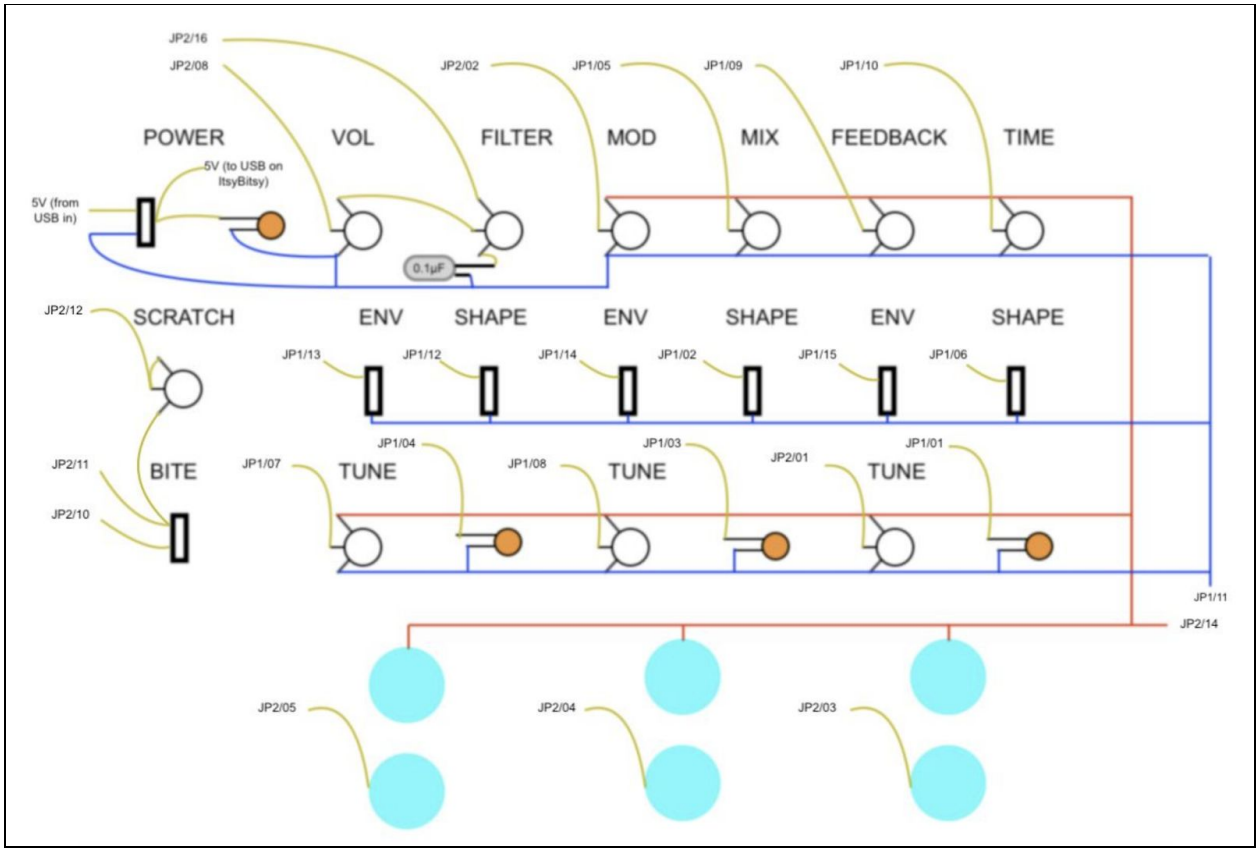
Part	Type	Notes
TIME, MIX, FEEBACK, MOD, TUNE (x3)	10 kΩ B	You may use higher values, like 22 kΩ or 25 kΩ, but not higher than that. Lower than 10 kΩ is not recommended. B means linear.
FILTER, VOLUME, SCRATCH	10 kΩ B	You must use this exact value, although you could use A (logarithmic) if you prefer a different response curve.
Power LED	3 mm blue LED	The hole for the socket is 6 mm.
Env LED (x3)	3 mm orange LED	The holes for the sockets are 6 mm.
Power switch	SPDT	The hole is 6 mm.
ENV switches (x3)	SPST	The holes are 6 mm. You may use SPDT as well.
SHAPE switches (x3)	SPST	The holes are 6 mm. You may use SPDT as well.
BITE switch	SPST	The hole is 6 mm. You may use SPDT as well.
Touch pads (x6)		The holes are 5 mm. Make sure to use plastic washers at least on the front side, so that the pads do not come into contact with the gold plating!

Populate the panel

Now place all components in their respective holes.



See this diagram (view from the back of the panel):

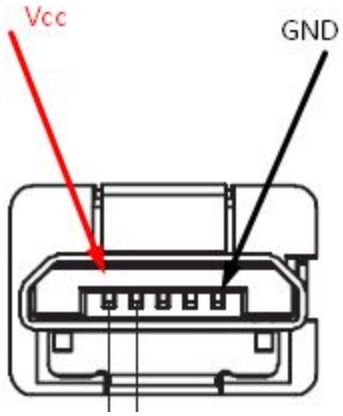


Connect wires as shown. First solder the blue and red strands of wire to their pins. You don't need to run individual wires, just use a single one for each colour per row and remove the insulation right where the solder points are.

For the shape and env switches, if you use SPST, make it so that they're closed in the up position. If you use SPDT, the same, but leave the third pin unattached to any wires. (For SPDT, the middle pin will connect to either top or bottom pin depending on the position. Use a multimeter to check if you're unsure.)

The bite switch gets two individual wires to the PCB. They should be connected if the switch is in the down position. If you use an SPDT here, leave one pin empty.

The power switch must be an SPDT. Its middle pin connects to the power (Vcc) on the microcontroller's USB socket. You can use a short length of micro-USB cable or make your own:



The wires inside USB cables will generally be colour-coded. You only need to use the red and black ones as indicated above, but if you're experieced, you may want to connect the others as well, so you'll be able to update the ItsyBitsy without having to open your case.

The other pins on the power switch connect to ground (off position) or 5V from your USB input (on position). Then solder the supply side of the wire to the on position pin and the other side to the middle pin of the power switch as indicated in the diagram above.

The middle pin also connects to the longer leg of the blue power LED via a 4.7 kΩ resistor (Rx) right behind the panel. Solder it nicely and keep the legs somewhat short to keep it from moving. (The high resistance is used to keep the LED dim, just like the orange ones for the envelopes. A blue LED connected through a more standard 50 Ω resistor is blinding! Of course you can change it if you want, but don't go below 50 Ω).

The pots for time, feedback, mix, mod, and 3x tune are all 10 kΩ linear (although 22 kΩ will work as well) and they connect to 3.3V from the PCB (as seen in the diagram) on the right pin and to ground on the left pin. (All pot legs referred to are as viewed from the panel side, so left means the left one when looking at the printed side of the panel, not at the wiring.) The middle pins all get their individual wire to the headers on the PCB as indicated by the numbers.

The 3 envelope LEDs all connect to ground on the shorter leg and get their individual wire to the PCB on the longer leg. The resistors for the LEDs are on the PCB, so no need to solder any to the panel as you did for the power switch.

For the volume pot you can use 10k linear, although logarithmic is fine too. Left leg to ground, right leg connected to the middle leg of the filter pot, middle leg to audio out jack (via PCB if you like, there is an in and an out available on one of the headers, otherwise you can of course go directly to the jack). The filter pot is also 10k linear, but you could use a logarithmic. Solder the 0.1 µF (same as 100 nF) capacitor (Cx) between the leftmost leg and ground, rightmost leg connected to audio out from the PCB. Be sure to solder the short leg of the capacitor to ground if it's an electrolytic, not the long one!

The scratch pot is 10 kΩ linear. Solder the rightmost leg to the middle leg with a short length of wire. Solder a cable there and another one to the leftmost leg (not to ground!). These go to the PCB as seen in the diagram above.

The top touch pads are all connected among each other and also to 3.3V from the PCB. The bottom pads have individual wires to the PCB as seen in the diagram.

For the touch pads, make sure to use the washers on front and back of the panel, but front is really important as they must not get into contact with the gold plating. Wind a bit of wire (once only) for each pad around the threaded part on the back of the pad, so you pinch it between the nut and the washer when tightening the nut. Use these to connect to other pads (top) and also to the respective header pins on the PCB.

You can collect all wires from the front plate with zip ties and solder or crimp them to connectors that plug into the headers on the PCB.

Connect panel and PCB

Now connect the wires from the panel to the header plugs as shown below. (e.g. JP1/05 goes to the MIX pot and JP2/07 goes to your audio out jack).

For header JP1:

- 01 ENV LED 1
- 02 SHAPE 2
- 03 ENV LED 2
- 04 ENV LED 3
- 05 MIX
- 06 SHAPE 1
- 07 TUNE 3
- 08 TUNE 2
- 09 FEEDBACK
- 10 TIME
- 11 GROUND (to panel)
- 12 SHAPE 3
- 13 ENV 3
- 14 ENV 2
- 15 ENV 1
- 16 unused

For header JP2:

- 01 TUNE 1
- 02 MOD
- 03 TOUCH 1
- 04 TOUCH 2
- 05 TOUCH 3
- 06 ground (to jack)
- 07 audio out (to jack)
- 08 audio in (from volume pot on panel)
- 09 unused
- 10 from BITE switch
- 11 to BITE switch and SCRATCH pot left leg
- 12 from BITE pot middle/right leg
- 13 unused
- 14 3.3V
- 15 unused
- 16 to FILTER on panel

Now connect the PCB to the panel. Pay attention to the orientation of JP1 and JP2. JP1 pin 1 is on the left when seen from above and JP2 pin 1 is on the right!

Power in

Elmyra gets power through the ItsyBitsy microcontroller via USB. Install a micro-USB socket on the back or side of your case (or just a regular USB cable with an A plug – simply cut off the other plug) and connect the power wire to the power switch and also to the ItsyBitsy as explained above.

Audio out

Install a TS audio out jack on the back or side of your case and connect 2 wires to it. Both go to JP2 as shown above. GND must be connected to the outer ring of any plug that is inserted into the jack. Use a multimeter if you are unsure.

Flashing the firmware

If you've purchased a pre-programmed microcontroller, you can omit this step!

Download the firmware here:

https://github.com/neutral-labs/elmyra

Install and open Arduino IDE and install the SAMD21 libraries as described here:

https://learn.adafruit.com/introducing-itsy-bitsy-m0/using-with-arduino-ide

Load the project. Connect your microcontroller via USB and upload the sketch via the arrow button on the top left.

It is done!

If you wired everything correctly, you're now done. Go create some noise!

If you need help or want to share photos, audio and/or video of your creations (please do), send a message to admin@neutral-labs.com

IO pin reference

For reference (e.g. for your own projects based on the same code), these are the IO pins used on the ItsyBitsy M0 Express:

A0 audio out
A1 touch 3
A2 touch 2
A3 touch 1
A4 MOD
A5 TUNE 1
D0 (20) TUNE 2
D1 (21) TUNE 3
D2 SHAPE 3
D3 (23) TIME
D4 (22) FEEDBACK
D7 SHAPE 1
D9 MIX
SCK ENV 1
MOSI ENV 2
MISO ENV 3
D10 LED 3
D11 LED 2
D12 SHAPE 2
D13 LED 1