1U Attenuverter / Attenuator – Build Guide

This practical 6hp 1U utility module that can be built in 3 different configurations

1. Passive attenuator → only use front panel + controls pcb
2. Attenuverter with 5V voltage reference connected to the switch in the input jack. If nothing is plugged in, you will have +/- 5V to use as offset.
3. Attenuverter without voltage reference. If you don't need this functionality you can save yourself ~$0.50 worth of parts.

1U Attenuverter / Attenuator – Build Guide

Intro:

In this build guide I'm going to assume that you have some basic proficiency in building PCB/panel kits. If this is the very first module you're building, I can highly recommend https://learn.adafruit.com/adafruit-guide-excellent-soldering

The build difficulty is of this module as passive attenuator is incredibly easy.

The build difficulty is of this module as attenuverter is average.

- It's all through hole components but there are a few standing resistors.

It can be built using the most basic equipment.

- Soldering Iron – I use and can highly recommend the Hakko FX-888D
- Side cutters
- Needle nose pliers (I personally use an 8mm/10mm pipe wrench for pots and Thonkiconn nuts)
- A Multimeter to verify component values and check for shorts.
- Helping hand (sometimes called third hand) to hold your PCB's – not required, but nice to have.

1U Attenuverter / Attenuator – Build Guide

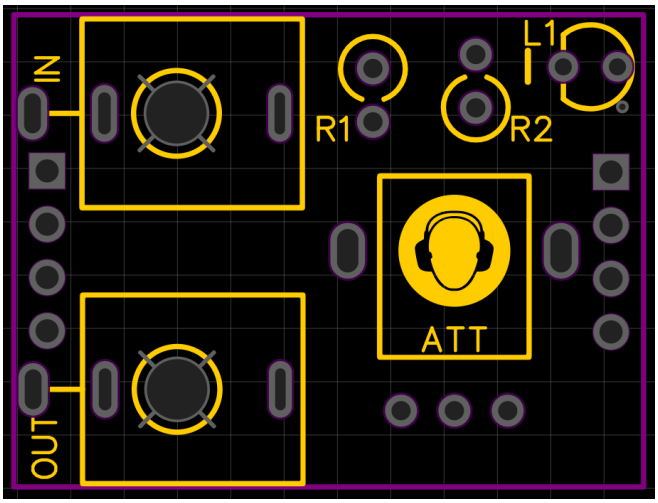
Bill of materials

Name	Designator	Type	Quantity
B100K*	ATT	ALPHA 9MM, Song Huei 'tall trimmer'	1
10uF	C1,C2	Low profile electrolytic, like UPW1E100MDD	2
100nF	C3,C4	MLCC / ceramic 5mm lead spacing	2
Male	H1,H3	40 PIN 2.54MM single row header	2
Female	H2,H4	Tayda A-1303 or similar	2
Thonkiconn	IN,OUT	From Thonk, or any of the alternatives	2
LED 3MM Bipolar R/G	L1	LED 3MM BIPOLAR R/G	1
470R	R1	1/4W 1% resistor	1
1K*	R2	1/4W 1% resistor	1
47K	R3,R4,R5	1/4W 1% resistor	3
100K*	RV1	Trimpot 3296X/T410X - same value as pot	1
LM4040DIZ-5.0/NOPB	U1	TO-92	1
TL072	IC1	DIP8	1
8 pin IC socket	optional	8 pin IC socket. Tayda A-857 or similar	1
10 pin header	U2	boxed header, angled header, or 2x5 pin. Tayda A-2939 or similar	1
Knob		Pick your favorite knob. Davies 1900h or smaller will work.	1

Sources I use:

https://www.thonk.co.uk/	Potentiometers, knobs, Thonkiconn, rare parts
https://www.taydaelectronics.com/	Resistors, generic components, IC sockets
https://www.tme.eu/nl/en/	Wima film caps, trim pots, IC's (cheaper than Mouser)
https://eu.mouser.com/	Has almost everything except the Thonk items.
https://electricdruid.net/	Rare IC's, vactrols, Electric Druid PIC's

1U Attenuverter / Attenuator – Build Guide



Controls PCB – Passive Attenuator

Skip these components:

- R1, R2, R3
- Led1
- Q1

Place the potentiometer, but don't solder yet.

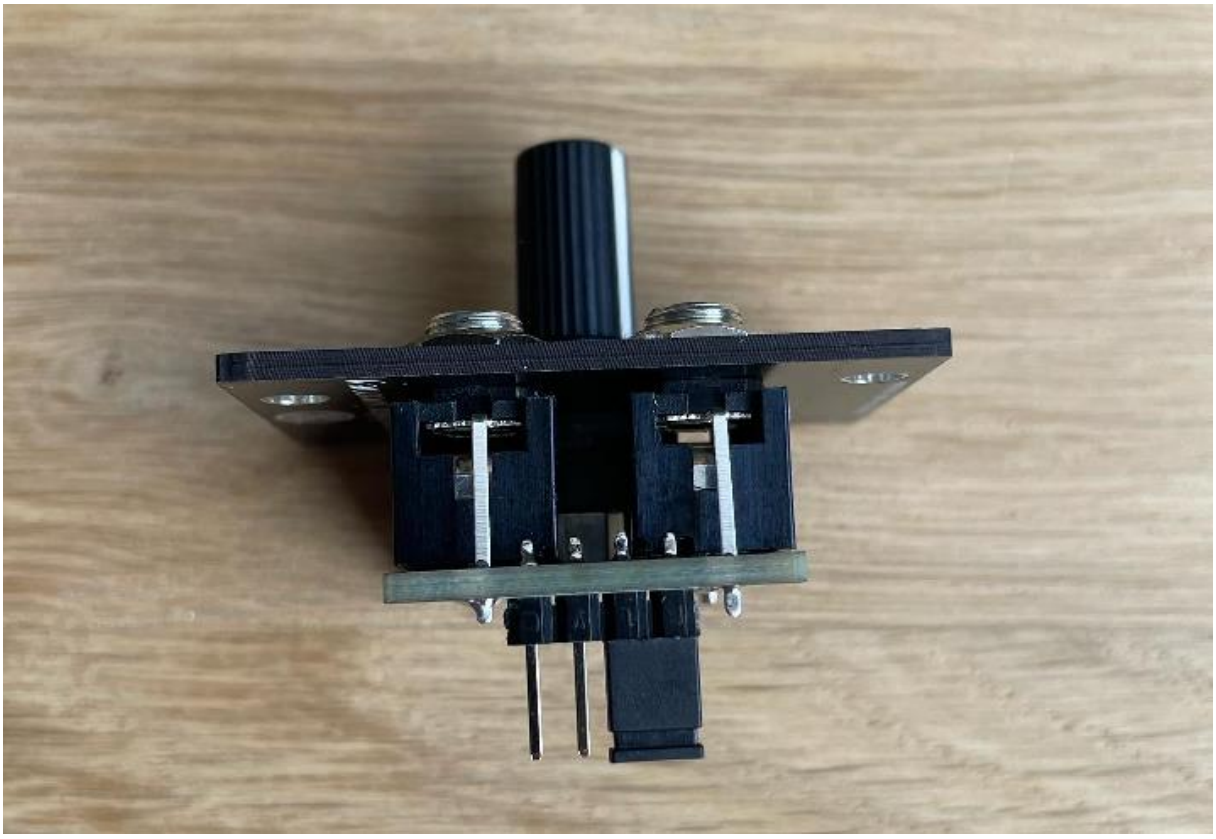
Install the 2 Thonkiconn jacks → but don't solder yet!

Now take the front panel and tighten the nuts on the Thonkiconn and potentiometer to secure the front panel in place. Make sure they don't rotate under the force. Everything should align up nicely without tension.

Flip the whole thing over and make sure everything still lines up. Make sure the LED is in the hole in the front panel.

Solder all controls pcb components in place. → double check if nothing moved when done.

Bridge H1-pin3 with H1-pin4 with a jumper or just a piece of cut off resistor lead.



Since you only wanted a passive attenuator, you're build is finished now. No calibration required.

Skip the rest of the guide and start attenuating things.

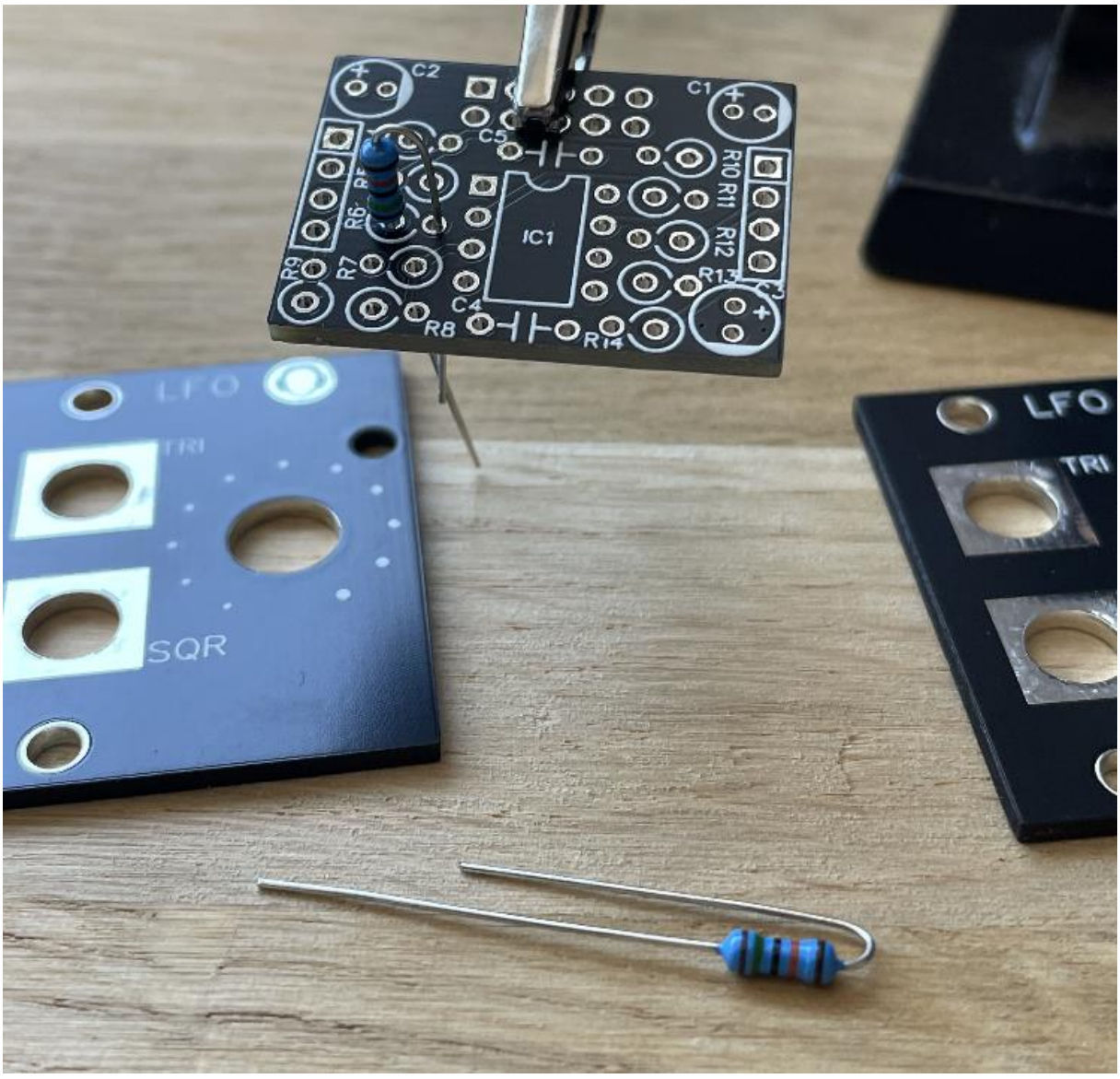
I'm always happy to see if anyone would actually build my modules.

If you finish this and feel like it, tag me on Instagram:

@my_modular_journey or use the hashtag #mymodularjourney so I can see your build.

Soldering standing resistors

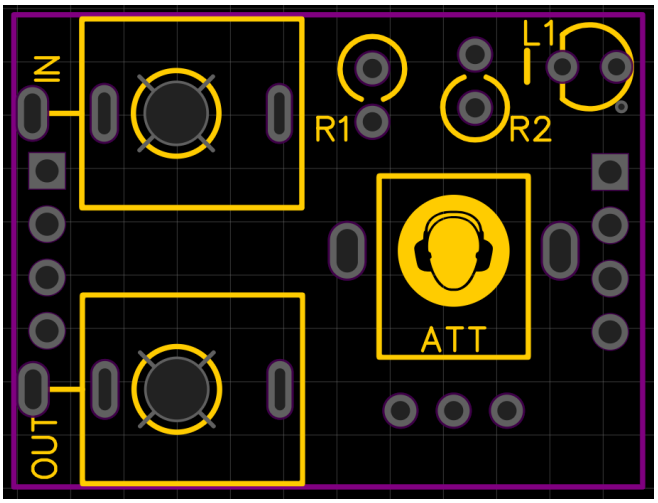
- Bend one of the legs 180 degrees in a smooth curved radius.
- Make sure the body + bent leg don't exceed ~9mm height when standing up to avoid shorts with the controls pcb.
- The silkscreen circle indicates where the resistor body goes. The hole next to the gap in the silkscreen circle will take the bare bent leg.
- Solder the bare leg from the top of the pcb to hold it in place. → make sure to not apply too much pressure with the soldering iron to prevent knocking over the resistor.
- After all standing resistors are placed with one leg soldered, flip the board and solder the other legs.



Quincas - Synth DIY Guy uses this same technique.

If my explanation is not clear, just watch some of his video's on the Erica Synths diy series. Even if you got this, I highly recommend his video's for anyone interested in building DIY modules.

Synth DIY Guy - Erica Synths Bassline



Controls PCB – Attenuverter

Install the standing resistors.

- R1(470R), R2(1K*) *use your favorite LED current limiting resistor value.

Install the LED, mind the orientation. The flat side on the LED should match the flat side on the silkscreen should → but don't solder yet!

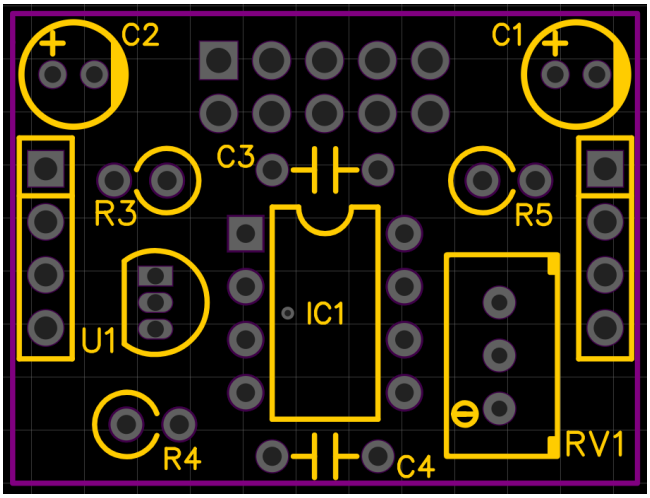
Place the potentiometer, but don't solder yet. → Make sure the potentiometer and the RV1 trimmer are the same resistance!

Install the 2 Thonkiconn jacks → but don't solder yet!

Now take the front panel and tighten the nuts on the Thonkiconn and potentiometer to secure the front panel in place. Make sure they don't rotate under the force. Everything should align up nicely without tension.

Flip the whole thing over and make sure everything still lines up. Make sure the LED is in the hole in the front panel.

Solder all controls pcb components in place. Double check if nothing moved when done.



Main PCB

Install the IC socket → Watch the orientation!

- Use the front panel to hold the socket in place while flipping the PCB. Solder in place.

Now it's time for the standing resistors.

- Install R4, R5 (47K) solder the bare leg from above
- Install R3 (47K*) solder the bare leg from above
  * skip this resistor if you don't want to install the internal 5V offset voltage.
- Flip the board and solder the legs on the body side.
- Trim all the legs with a cutter and touch up your soldering where necessary.

Install the ceramic caps. You can solder these from the top.

- 100nF C3, C4

Install* the 4040 voltage reference.

- * Skip the 4040-5.0 + R3(47K) if offset voltage is not needed.

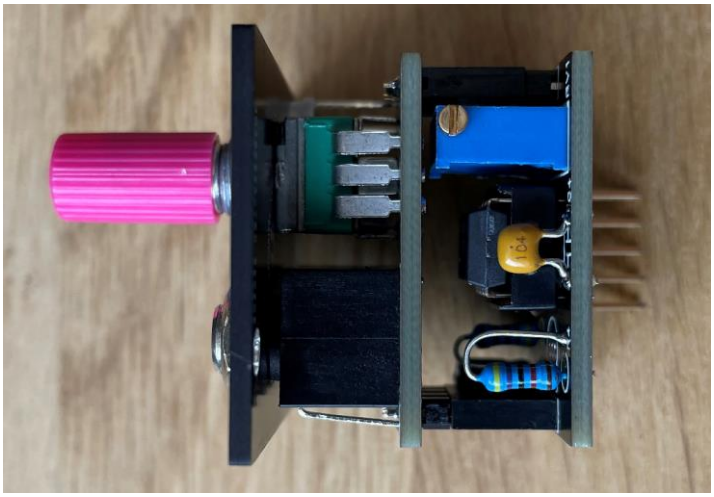
Install the electrolytic caps → Watch the orientation!

- 10uF C1, C2 If you can't find a low profile 10uF, place it on the back of the pcb.

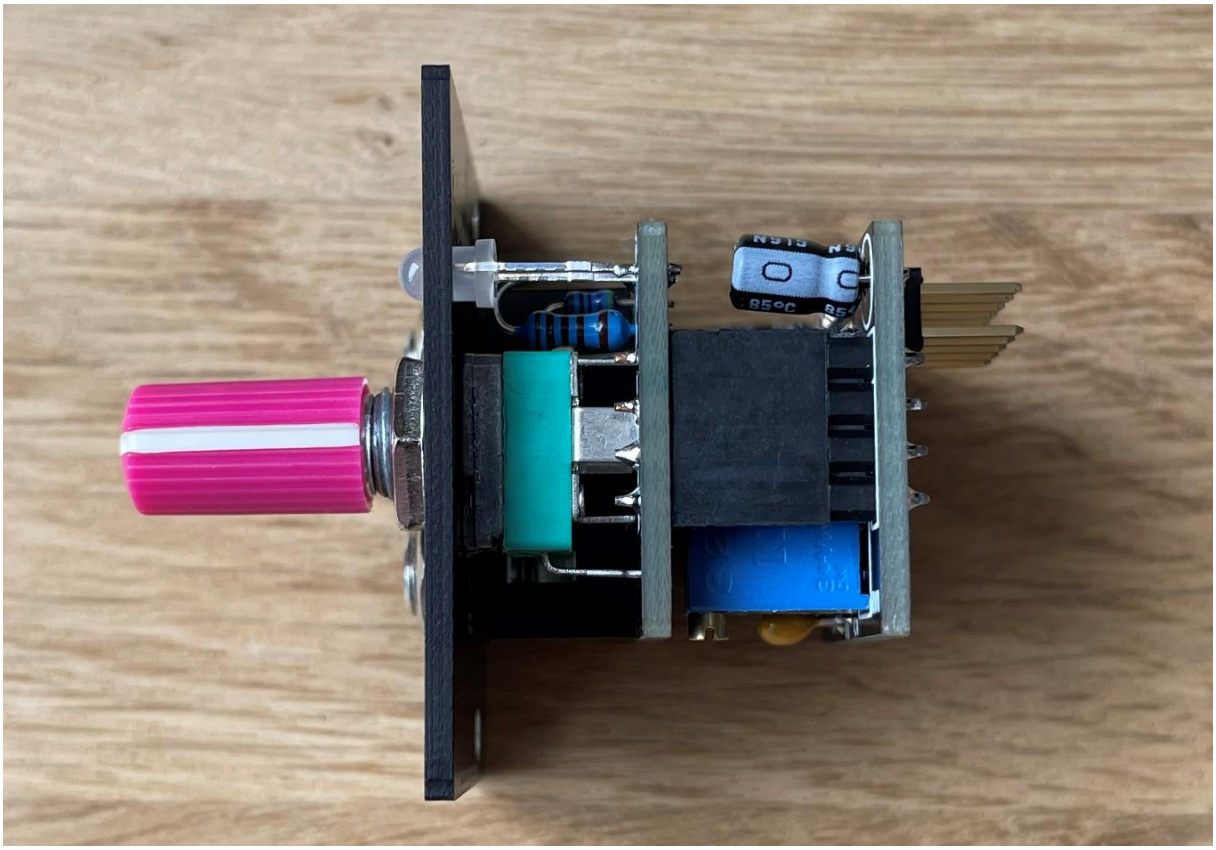
Bend the legs a little outwards to hold them in place while flipping the PCB to solder it from the other side.

Trimpot: → Make sure the potentiometer and the RV1 trimmer are the same resistance!

- Install the trim pot on the topside of the PCB if you use a 3296X or T910X type. (X indicates the screw on the side of the body.
- If you use the more common W-type trimmer, mount it on the back.



Install the IDC 10 pin header on the bottom side of the PCB. → Make 100% sure you do this right. It's very hard to unsolder these without destroying the board.



Building the sandwich

There are multiple ways to do this, but this is my preferred method.

Start with the front panel still flat on the table.

Assemble the pin headers together.

Make sure that when you install the header assemblies, you build in some safety (poka yoke) measure to be able to never accidentally install the boards in the wrong orientation.

- One of the sides should have female header on the bottom
- One of the sides should have male header on the bottom

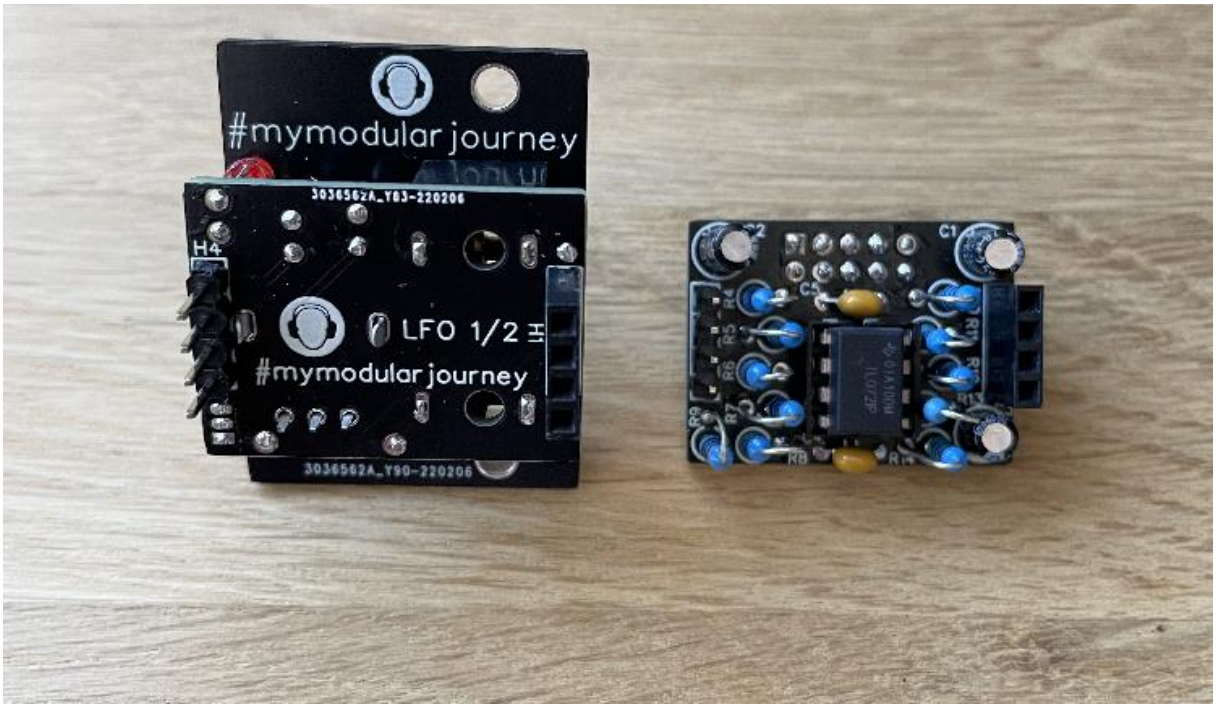
Now place the main board on top with the power header on your side. All text should be oriented the same. If not, one of the boards is placed wrong.

Solder one pin on both headers.

Flip the sandwich on its side, like in the picture. Make sure it's all aligned neatly and flat.

Solder all the pins on the headers on the visible side.

Rotate the board and do the same for that side.



This is an example of Poka Yoke* (loosely translated as idiot-proof) when it comes to the headers.

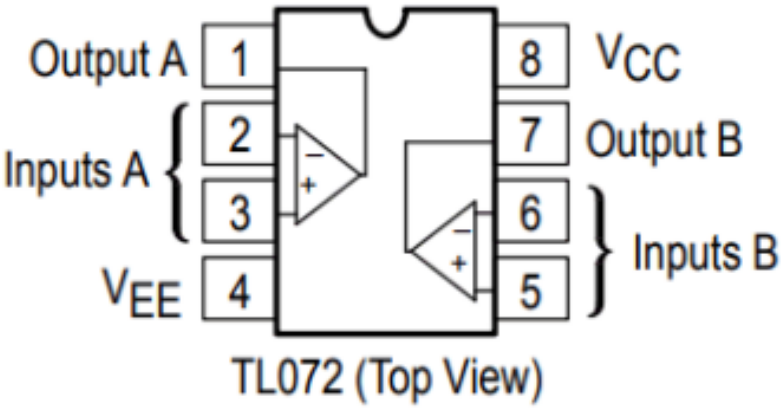
By alternating the header orientations there is no chance to accidentally assembling the boards in the wrong orientation and potentially damaging components or the power supply.

*For true safety by design I should have offset the placement of the headers. This way there would be a real physical barrier to prevent accidental wrong orientation. Even if the builder doesn't follow the instructions.

Power-up & calibration

Before you power up the module, do a continuity test to check for shorts.

- Set your multimeter to resistance.
- Measure the resistance between +12V and ground, -12V and ground, and between the two power inputs. There can be some fluctuations due to the capacitors, but there should never be any shorts. → If you find a short, check your soldering.
- Remove the back PCB
- Connect your power cable → watch the orientation. The red stripe on the cable should match the stripe / -12V label on the pcb.
- Use your multimeter in the volts setting.
- Verify that the pin 8 receives +12V and pin 4 receives -12V.



- Check the 5V voltage reference if you installed that. PCB's from rev1.1 have a mark where to measure.

- Remove the power
- Install the TL072 IC → watch the orientation. The notch should line up with the notch in the socket and silkscreen.
- Reassemble the pcb sandwich.

No calibration required. Your new module is ready to use.

I'm always happy to see if anyone would actually build my modules.

If you finish this and feel like it, tag me on Instagram:

@my_modular_journey or use the hashtag #mymodularjourney so I can see your build.