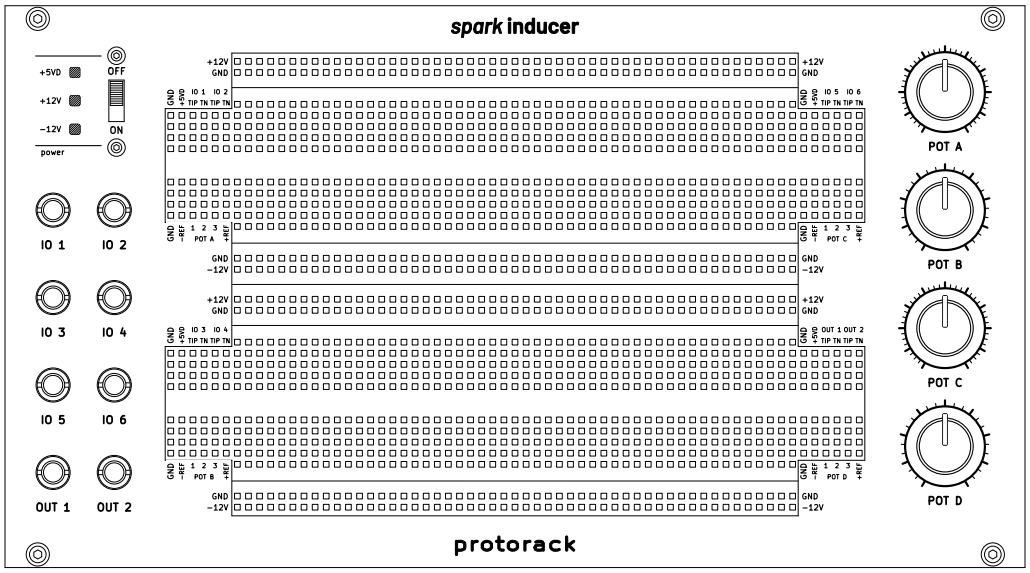
spark inducer - ASSEMBLY MANUAL (Rev.2)



Designed by Max Schlecht

Licensed under CERN-OHL-W-2.0+

Introduction

The *spark* inducer is a development kit for prototyping audio circuits specially designed for creating eurorack modules. It can be used to test your ideas in an isolated environment (like you'd do when breadboarding normally) but with the added benefit of easily connecting to other eurorack compatible devices.

It can also be mounted in a standard eurorack enclosure, so you can quickly test how your creation will perform in an actual patch. Permanent rack installation is also possible, but additional precautions must be taken (see 4.1).

It features:

- 2x 830 point breadboards (BB830/MB-102 type)
- 6x general purpose connections (inputs or outputs) and 2x buffered outputs
- 4x arbitrary value potentiometers (the kit comes with $100k\Omega$ ones)
- preconnected analog power rails, digital +5V power (from the +5V bus)
- up to 4x selectable voltage references (positive and negative)
- power switch, status LEDs and resettable fuses on all power rails
- being powered from a standard 16pin eurorack power connector

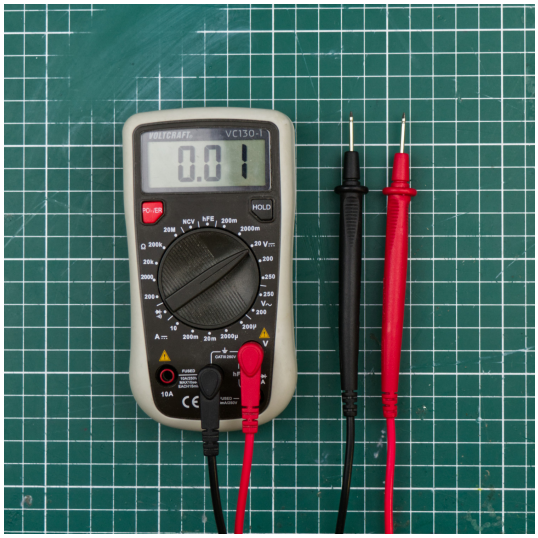
This DIY kit is perfect for intermediate builders who already know how to do basic through-hole soldering but want to dip their feet into SMD soldering, as it mainly consists of relatively easy to solder 0805 and 1206-sized passive components, some SOIC packages and a bunch of 2.54mm headers.

Important:

Even if you are an experienced builder, please make sure to follow the steps in the correct order when building this kit. Some sections are very dense and might be hard (or even impossible) to assemble in the wrong order.

Required Tools

- Side Cutters
- Tweezers
- 1.5mm hex key
- Lead-Free Solder
  We use Stannol Kristall 611 (Sn96.5Ag3Cu0.5, 0.7mm) at 380°C.
- Soldering Iron
  Temperature control is nice, but not strictly required.
- Basic Multimeter
  Anything cheap that can measure Voltage and Continuity will do.

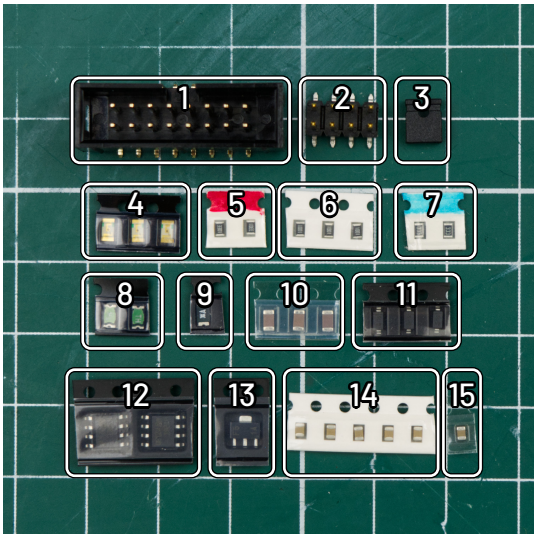


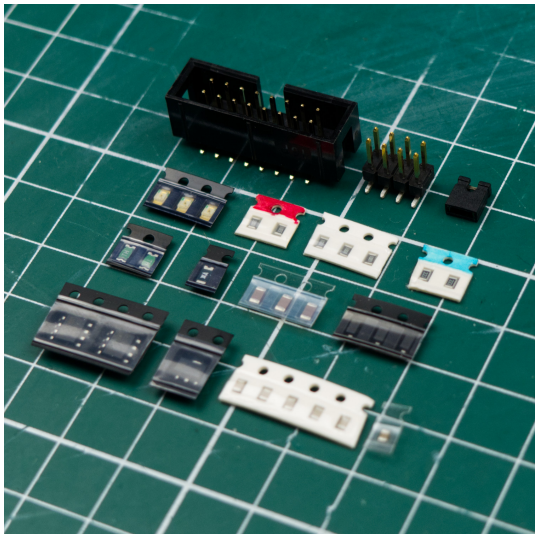
Nice to have

- Solder Wick
  To get rid of superfluous solder. Thinner one (~1.0mm Ø) tends to work best.
- Magnification
  A simple magnifying glass works nicely to inspect your soldering.
- Ruler/Calipers
- Utility Knife

Build Steps

1 SMD Components





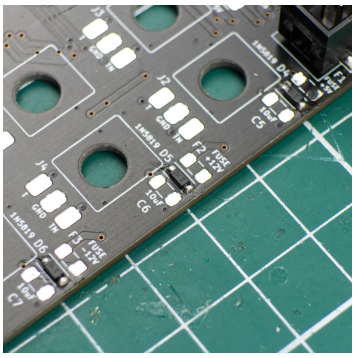
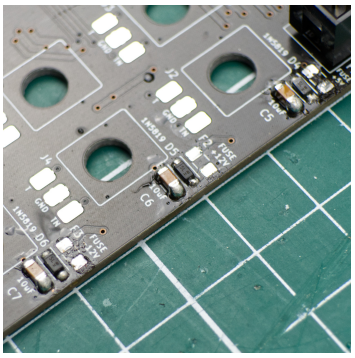
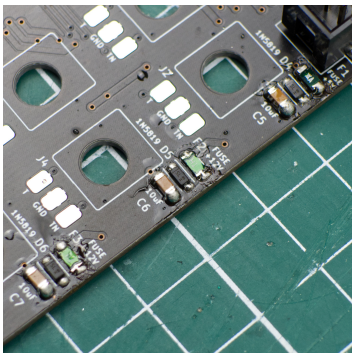
ID	Qty	Description	Code on Part	PCB Identifier
1	1	2×8 power connector	-	J1
2	1	2×4 header	-	J18
3	1	jumper	-	-
4	3	red led, 1206	-	D1, D2, D3
5	2	resistor, 0805, 1.0%, 3.3k	3301	R2, R3
6	3	resistor, 0805, 1.0%, 1.0k	1001	R1, R7, R8
7	2	resistor, 0805, 0.1%, 33.0k	333	R10, R11
8	2	resettable fuse, 100mA	TODO	F2, F3
9	1	resettable fuse, 200mA	TODO	F1
10	3	capacitor, 1206, 10uF	-	C5, C6, C7
11	3	diode, SOD-123, 1N5819	TODO	D4, D5, D6
12	2	opamp, SOIC8, TL072B	TODO	U1, U2
13	1	voltage regulator, SOT-89, L78L05	TODO	U4
14	5	capacitor, 0805, 100nF	-	C1, C2, C3, C4, C10
15	1	capacitor, 0805, 330nF	-	C9

1.1 Power Protection/Decoupling

	ID	Qty	Description	Code on Part	PCB Identifier
	11	3	diode, SOD-123, 1N5819	TODO	D4, D5, D6
	10	3	capacitor, 1206, 10uF	-	C5, C6, C7
	8	2	resettable fuse, 100mA	TODO	F2, F3
	9	1	resettable fuse, 200mA	TODO	F1

Start by soldering the 1N5819 Schottky diodes and 10uF decoupling capacitors for each of the three power rails. **Mind the polarity of the diodes!**

Then, do the same for the resettable PTC fuses. The two green ones are for the +12V and -12V power rails and can hold a maximum current of 100mA indefinitely and will trip at ~250mA. The black one is for the +5V rail and has a hold current of 200mA and will trip at ~400mA. (This should be enough for most circuits, even ones including power-hungry digital microcontrollers.)

(If you plan on building circuits that require more current, you can replace the fuses with ones that support a higher hold current, **making sure your power supply can still reach their rated trip current!**)

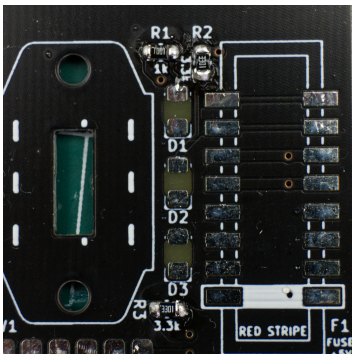
1.2 Power Connector, LEDs

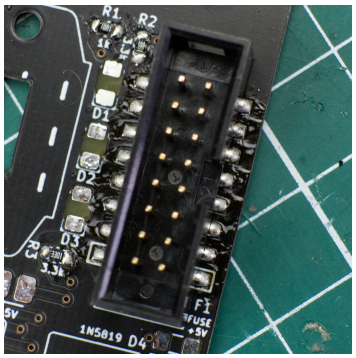
	ID	Qty	Description	Code on Part	PCB Identifier
	5	2	resistor, 0805, 1.0%, 3.3k	3301	R2, R3
	6	1	resistor, 0805, 1.0%, 1.0k	1001	R1
	1	1	2×8 power connector	-	J1
	4	3	red led, 1206	-	D1, D2, D3

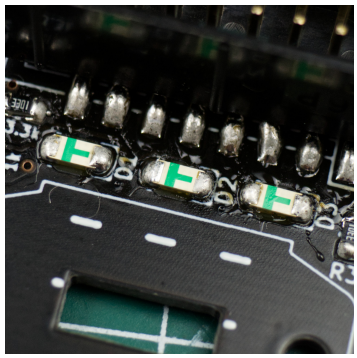
Start by soldering the two 3.3k resistors and the 1k resistor.

Next is the power connector. Align it carefully, **minding its orientation (there's an arrow on one side, which has to align with the white stripe on the PCB)** and solder it in place. Closely inspect your soldering and ensure you don't have any shorts, as this area will be hard to access later on.

Finally, solder the LEDs. These have to be soldered upside-down so that they can shine through the board. There's a T-shaped marking on their back that has to be aligned with the matching symbol on the PCB. **Be careful not to overheat the LEDs! The plastic lenses can be damaged quite easily!**





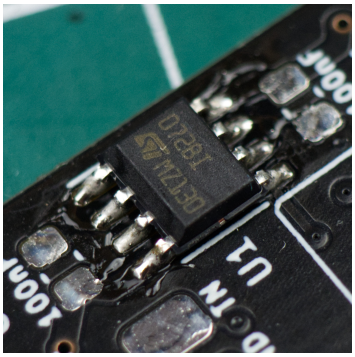


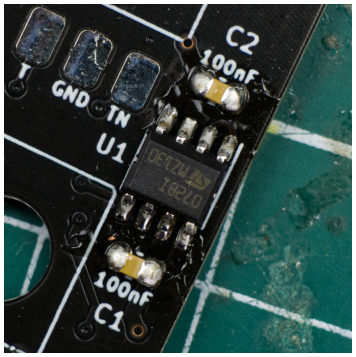
1.3 Buffered Outputs

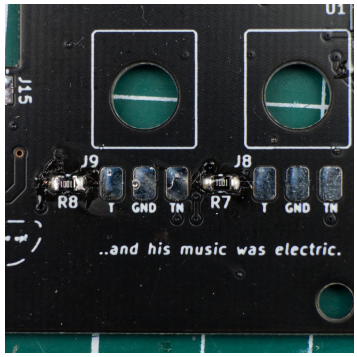
	ID	Qty	Description	Code on Part	PCB Identifier
	12	1	opamp, SOIC8, TL072B	TODO	U1
	14	2	capacitor, 0805, 100nF	-	C1, C2
	6	2	resistor, 0805, 1.0%, 1.0k	1001	R7, R8

First, solder the TL072B. Make sure to line up the little dot that marks Pin 1 with the marker on the PCB.

Then, solder the 100nF decoupling capacitors and the 1k outputs resistors.



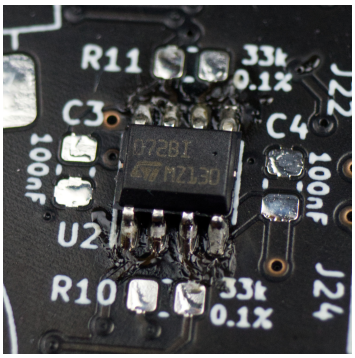


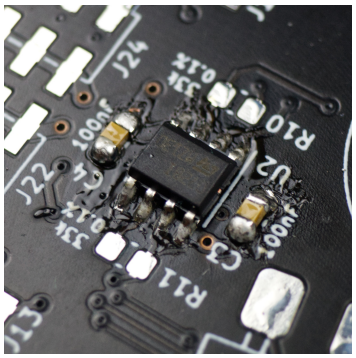


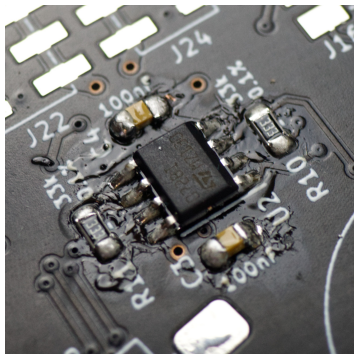
1.4 Reference Voltage Buffer

	ID	Qty	Description	Code on Part	PCB Identifier
	12	1	opamp, SOIC8, TL072B	TODO	U2
	14	2	capacitor, 0805, 100nF	-	C3, C4
	7	2	resistor, 0805, 0.1%, 33.0k	333	R10, R11

Once again, start by soldering the TL072B, then, solder the 100nF decoupling capacitors, as well as the 33.0k precision resistors.







1.5 (default) L78Lxx Voltage Reference

	ID	Qty	Description	Code on Part	PCB Identifier
	13	1	voltage regulator, SOT-89, L78L05	TODO	U4
	14	1	capacitor, 0805, 100nF	-	C10
	15	1	capacitor, 0805, 330nF	-	C9
	2	1	2×4 header	-	J18
	3	1	jumper	-	-

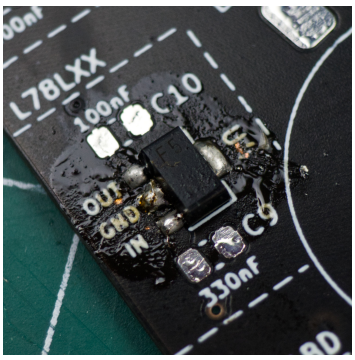
This is the default voltage reference that comes with the DIY kit. It uses a cheap 78xx regulator to produce a stable but not terribly accurate voltage reference. The particular regulator that comes with the kit is a 5V one, but ones for other voltages are also available.

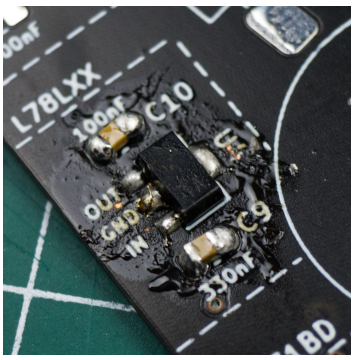
As for all the other reference options, its output voltage gets buffered by one channel of the Reference Voltage Buffer, while the other one inverts it to get the negative reference. The buffering/inverting isn't necessary per se, but for this use case, the opamp offers some level of protection, so that you won't be able to damage the references directly by connecting something in the wrong way. Using a low-offset opamp in combination with 0.1% precision resistors, can also be more cost-effective opposed to adding a separate negative reference.

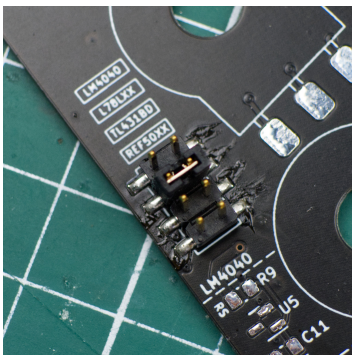
Start with the L78L05 regulator. Solder the outer pins first, then the center pin and the tab (these can take a while to solder because of their large thermal mass).

Next, assemble the 330nF input and 100nF output capacitors.

Finally, solder the 2×4 header and add the jumper to select the L78Lxx reference.







If you're building a kit and don't have any of the parts for the other Voltage References, skip straight to assembling the Breadboard Headers.

1.6 (optional) LM4040 Voltage Reference

TODO

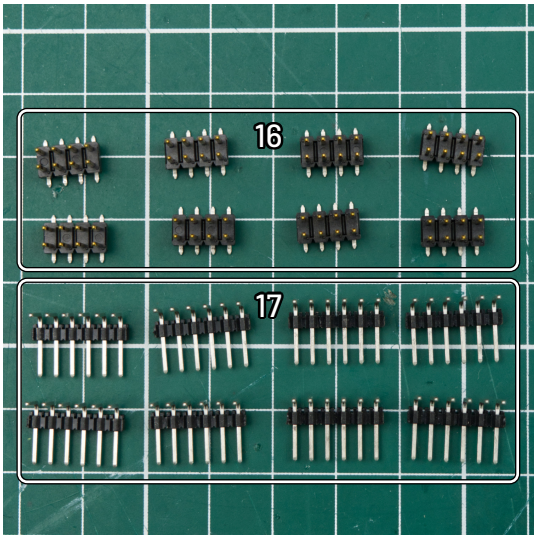
1.7 (optional) TL431BD Voltage Reference

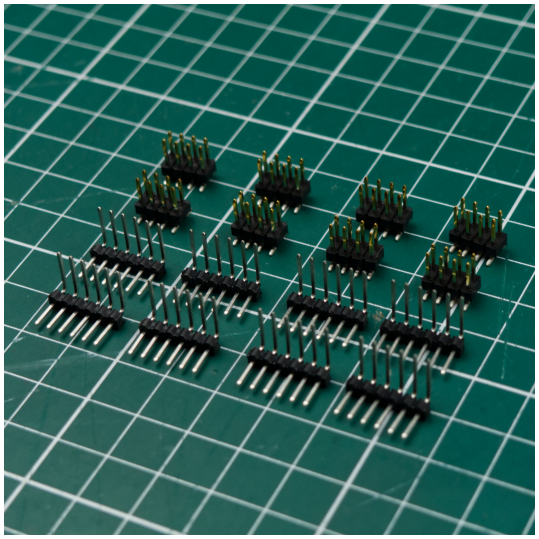
TODO

1.8 (optional) REF50xx Voltage Reference

TODO

2 Breadboard Headers





ID	Qty	Description	PCB Identifier
16	8	2×4 header	J19, J20, J21, J22, J23, J24, J25, J26
17	8	1×6 right angle header	J10, J11, J12, J13, J14, J15, J16, J17

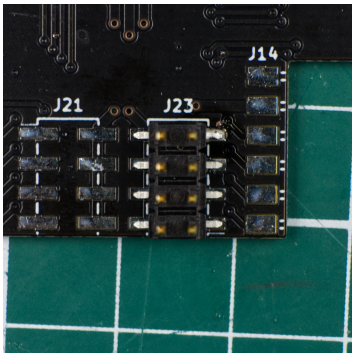
2.1 Power Headers

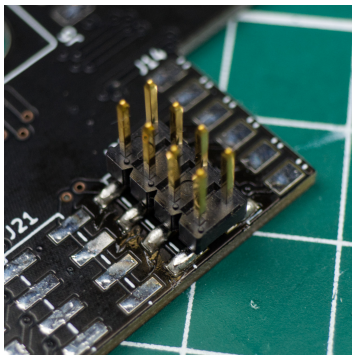
	ID	Qty	Description	PCB Identifier
	16	8	2×4 header	J19, J20, J21, J22, J23, J24, J25, J26

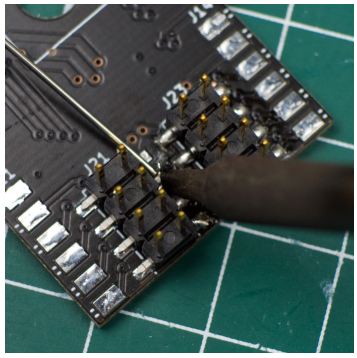
Take your time on soldering these, as they have to be aligned reasonably well, for the breadboard to fit later on. It doesn't have to be perfect, but there shouldn't be any obvious rotations or offsets larger than ~1mm.

To make this easier, tin one pin first (ideally, one that's not a ground pin), then solder the header to it and center it to your best ability. After that, you can carefully push/bend it a bit until the rotation looks good too, and solder the remaining pins.

The pins of the headers in the center can be tricky to solder, but you should still be able to reach them, as shown. Be careful not to melt the plastic of the headers.



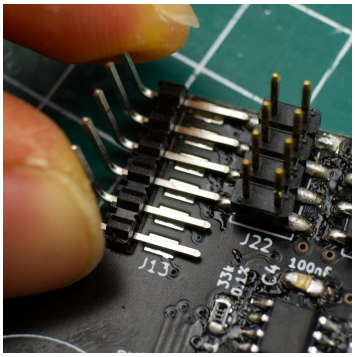


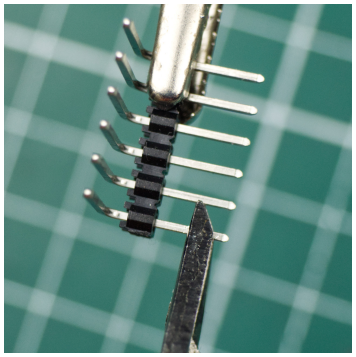
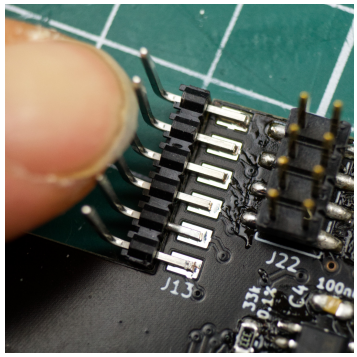


2.2 Right Angle Headers

	ID	Qty	Description	PCB Identifier
	17	8	1×6 right angle header	J10, J11, J12, J13, J14, J15, J16, J17

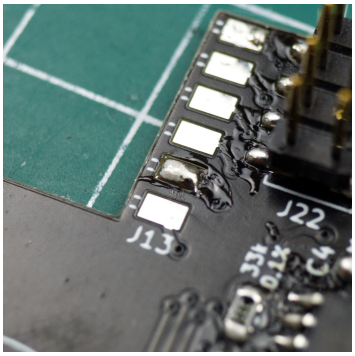
Take your side cutters and cut away about half (~3mm) of the shorter side of the headers, so that when pushed into the corner of the PCB, as seen below, they don't interfere with the power headers.

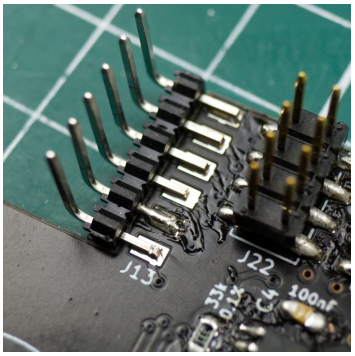


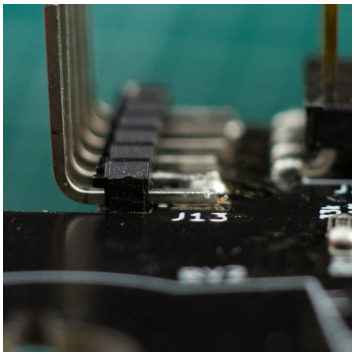



Then, for each header, tin one of the pads, reheat it and carefully push the header into the corner of the PCB. Make sure they aren't crooked and sit flat on the PCB without any vertical offset.

As before, these have to be aligned reasonably well, although it's a little less critical, because alignment is generally easier, and they can also be bent into place later on.

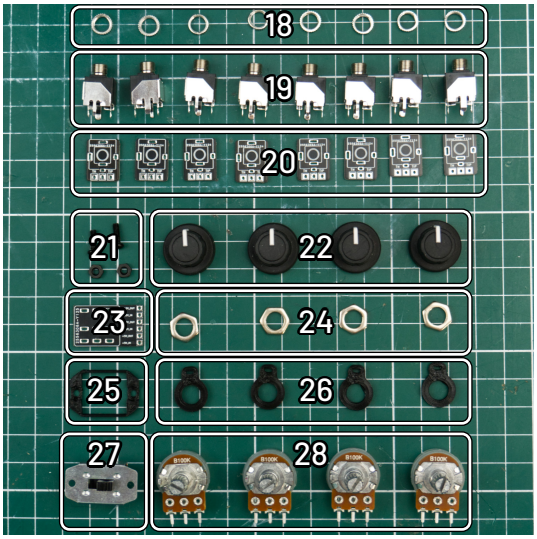


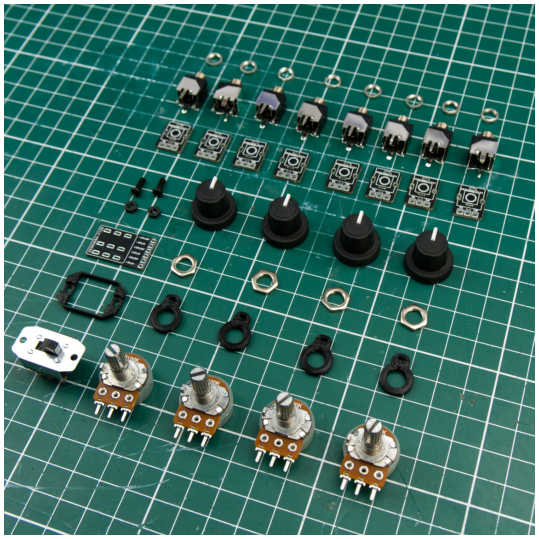




Finally, solder all the remaining pins.

3 Front Panel Components





ID	Qty	Description	PCB Identifier
18	8	knurled nut	-
19	8	WQP518MA-BM audio jack	J2, J3, J4, J5, J6. J7, J8, J9
20	8	audio jack PCB	-
21	2	M2 screw, nut	-
22	4	potentiometer knob	-
23	1	power switch PCB	-
24	4	potentiometer nut	-
25	1	power switch bracket	-
26	4	potentiometer spacer	-
27	1	power switch	SW1
28	4	B100k potentiometer	RV1, RV2, RV3, RV4

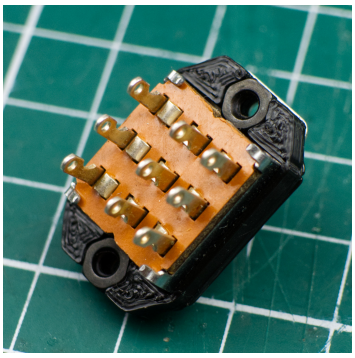
3.1 Power Switch

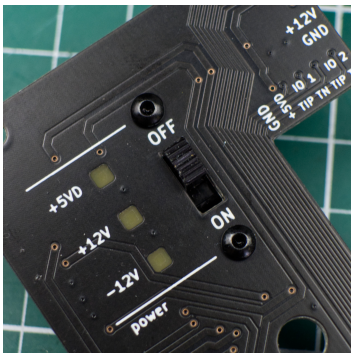
	ID	Qty	Description	PCB Identifier
	21	2	M2 screw, nut	-
	23	1	power switch PCB	-
	25	1	power switch bracket	-
	27	1	power switch	SW1

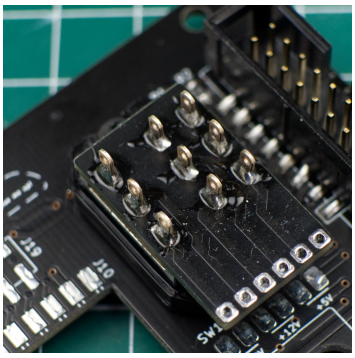
Insert the two M2 nuts into the 3D printed plastic bracket and slip it onto the switch (with the M2 nuts facing away from the switch).

Next, hold the switch against the PCB and screw it in place using the two M2 screws and a 1.5mm hex key. **Be careful to put it on the right way. The two rows of pins on the back of the switch that are closer to each other should be facing towards the power connector.**

Add the switch PCB to the switch (blank side up), push it down and solder down all the pins.







Snip off six **~12mm** pieces of the solid core wire and put each of them through one of the through-holes on the switch PCB. Fix them in place by soldering them to the pad on the main PCB first, then make sure they're straight and solder the pad on the switch PCB.

Finally, use your multimeter in continuity mode to ensure you didn't create any short-circuits. With the switch being in the OFF position, there shouldn't be continuity between any of the wires.

3.2 Testing

TODO

3.3 3.5mm Jacks

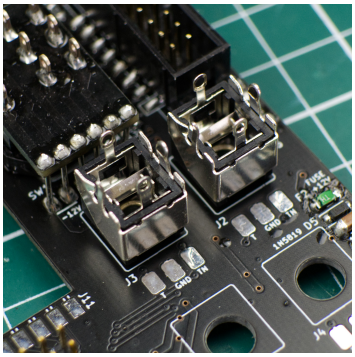
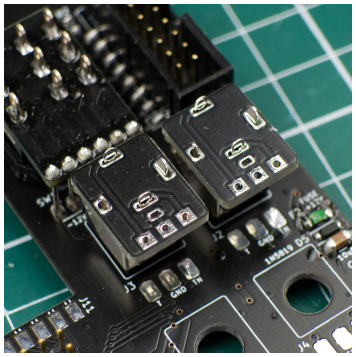
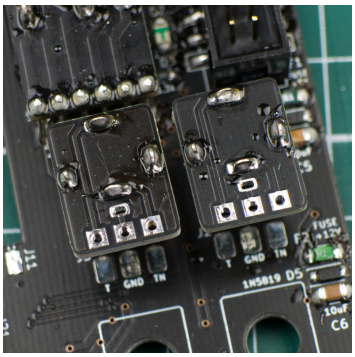
	ID	Qty	Description	PCB Identifier
	18	8	knurled nut	-
	19	8	WQP518MA-BM audio jack	J2, J3, J4, J5, J6. J7, J8, J9
	20	8	audio jack PCB	-

Important:

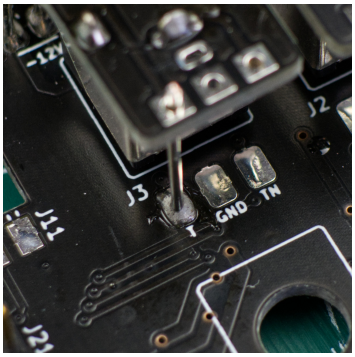
Do each of the following steps for each pair/row of audio jacks, **one pair/row at a time, from top to bottom**. Once a row is assembled, it will be **nearly impossible to reach the one above it**. Make sure your soldering is solid, and there are no shorts.

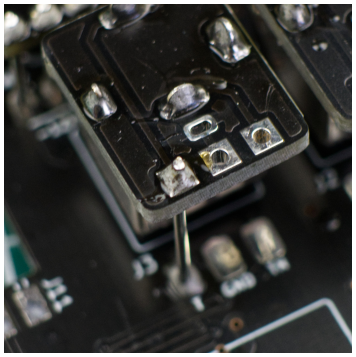
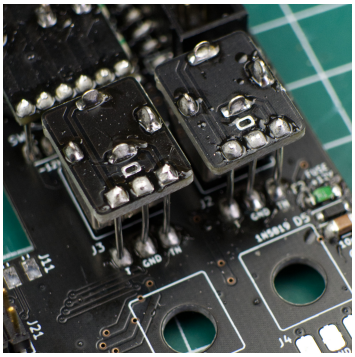
Take two audio jacks and knurled nuts and attach them to the two holes on the first row (from top to bottom) of the PCB. Align them so that they fit into the box marked on the back side of the PCB.

Add two audio jack PCBs to the audio jacks and solder all the pins.

Next, snip off six **~15mm** pieces of the solid core wire and put each of them through one of the through-holes on the audio jack PCBs. Fix them in place by soldering them to the pad on the main PCB first, then make sure they're straight and solder the pad on the audio jack PCB.



Use your multimeter in continuity mode to ensure you didn't create any short-circuits. There shouldn't be any continuity between any of the wires.

Repeat these steps for the remaining three rows of audio jacks.

3.4 Potentiometers

	ID	Qty	Description	PCB Identifier
	22	4	potentiometer knob	-
	24	4	potentiometer nut	-
	26	4	potentiometer spacer	-
	28	4	B100k potentiometer	RV1, RV2, RV3, RV4

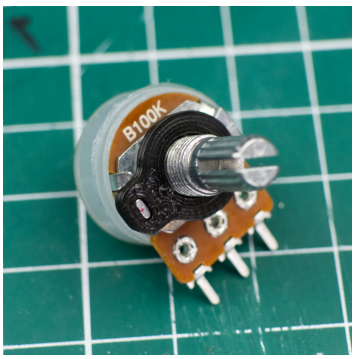
Tip:

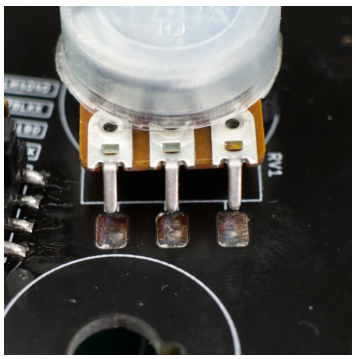
While not as critical as the audio jacks, this part is also easier to do if you follow the steps for each potentiometer, one by one, from top to bottom.

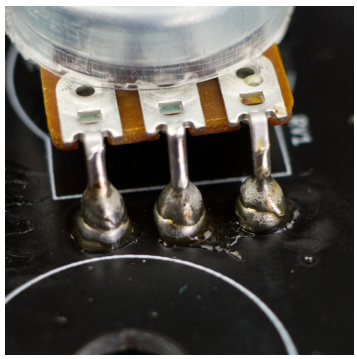
Attach the 3D printed spacers to the potentiometers. There's a little nubbin on one side, which indexes into the PCB and helps to align the rotation of the potentiometer.

Next, use the included 3D printed tool to screw down the potentiometers to the PCB with the hex nuts. Don't overtighten them, and make sure the potentiometer aligns with the pads on the back of the PCB nicely.

Finally, bend down the solder lugs till they touch the PCB and solder them in place. (If you don't have potentiometers with solder lugs but shorter pins, you can stack up some solder to make them reach.)







4 Breadboard

	ID	Qty	Description
	29	4	BB830 type breadboard
	30	5	rubber feet

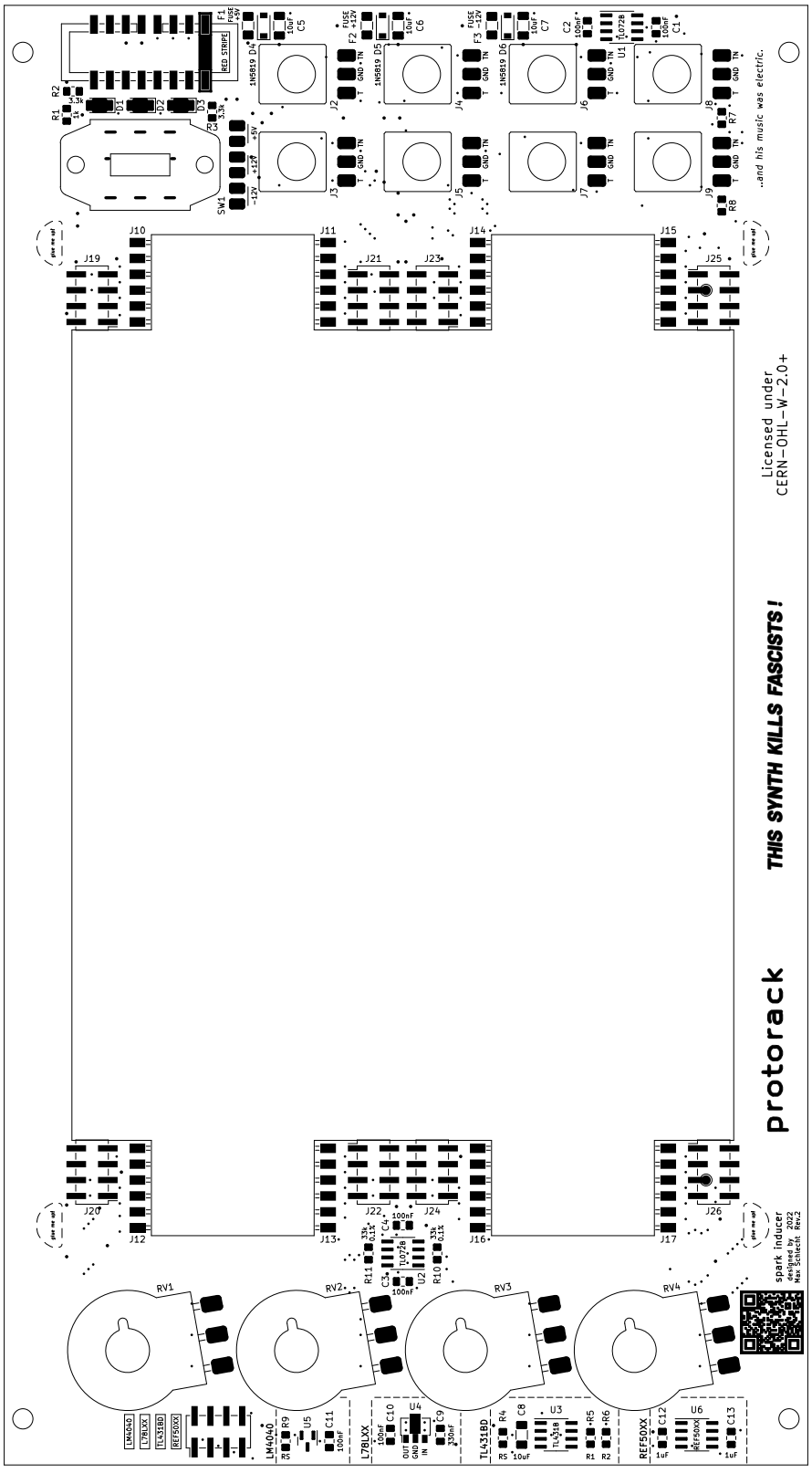
Take the two breadboards and combine them by pushing them together using the nubbins and slots on their sides.

Next, turn them around and start attaching the rubber feet to the bottom. Ideally, they should be directly attached to the adhesive on the breadboards back. For this, use a ruler to measure out five 20×20mm squares (one in the center and one in each corner). Then use a utility knife to cut through the top cover of the adhesive and carefully peel it off. Finally, peel off the cover film from the rubber feet and stick them onto the exposed adhesive.

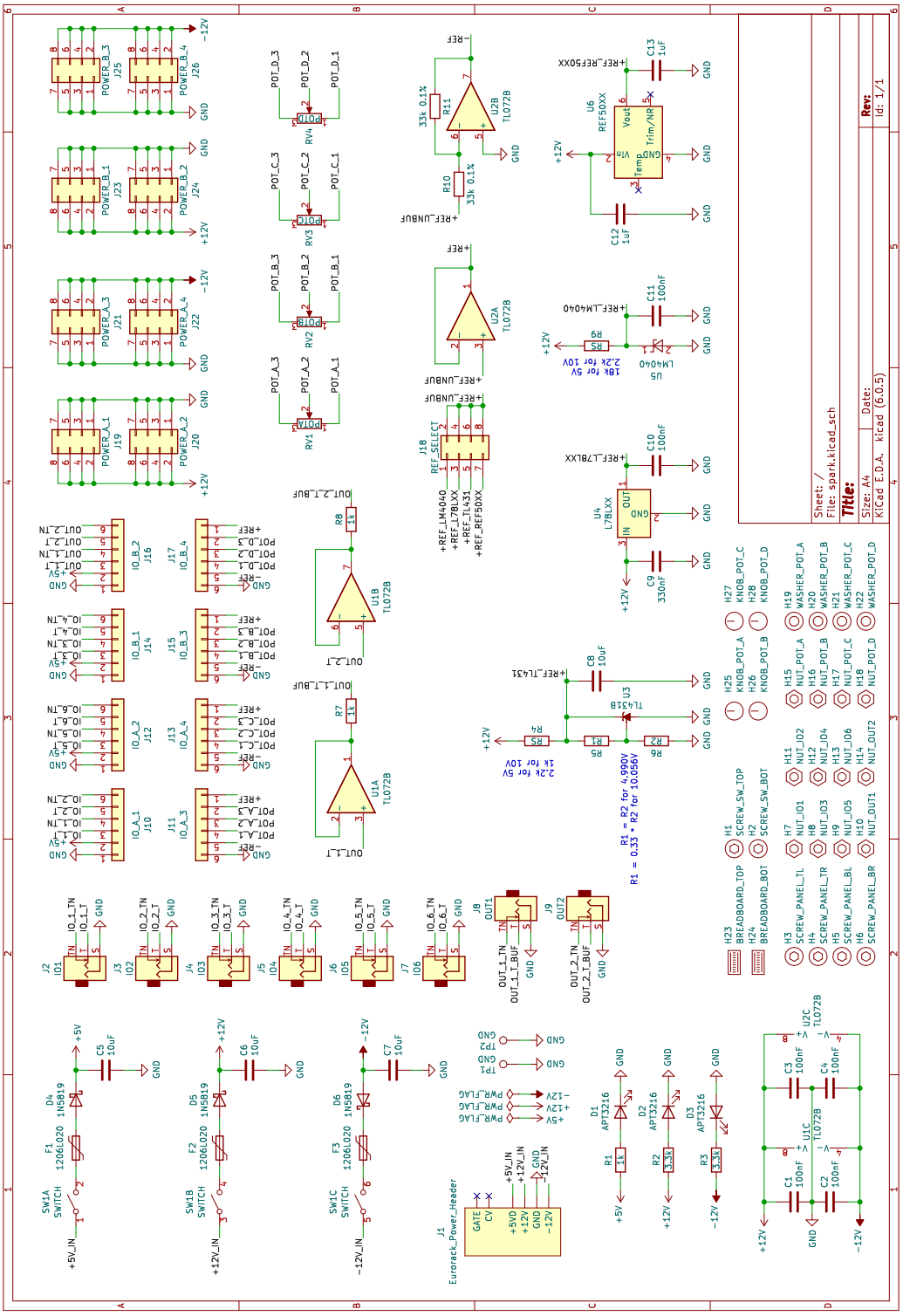
4.1 (optional) Gluing

TODO

Board View


THIS SYNTH KILLS FASCISTS !
protorack
spark inducer
designed by Max Schlecht Rev.2
2022
Licensed under CERN-OHL-W-2.0+
..and his music was electric.
RED STRIPE
RV1
RV2
RV3
RV4

Schematic



Bill of Materials

Qty	Reference	Value	Part Number	Note
5	C1, C2, C3, C4, C10	100nF	CL21B104KACNNNC	>=25V, X5R/X7R
3	C5, C6, C7	10uF	CL31A106KAHNFNE	>=25V, X5R/X7R
1	C9	330nF	CL21B334KBFNNNE	>=25V, X5R/X7R
3	D1, D2, D3	APT3216	APT3216SURCK	Red, ~80mcd (TODO)
3	D4, D5, D6	1N5819	1N5819HW-7-F	-
3	F1, F2, F3	1206L020	1206L020YR	TODO
1	J1	POWER IN	3020-16-0300-00	-
8	J2, J3, J4, J5, J6, J7, J8, J9	IO1, IO2, IO3, IO4, IO5, IO6, OUT1, OUT2	WQP-PJ376M	-
8	J10, J11, J12, J13, J14, J15, J16, J17	IO_A_1, IO_A_3, IO_A_2, IO_A_4, IO_B_1, IO_B_3, IO_B_2, IO_B_4	TSW-106-09-T-S-RA	-
9	J18, J19, J20, J21, J22, J23, J24, J25, J26	REF_SELECT, POWER_A_1, POWER_A_2, POWER_A_3, POWER_A_4, POWER_B_1, POWER_B_2, POWER_B_3, POWER_B_4	54202-S08-04	-
3	R1, R7, R8	1k	RC0805FR-7W1KL	>=1/4W, 1%
2	R2, R3	3.3k	RC0805FR-073K3L	>=1/8W, 1%
2	R10, R11	18k	RC0805FR-0718KL	>=1/8W, 1%
4	RV1, RV2, RV3, RV4	POTA, POTB, POTC, POTD		-
1	SW1	SWITCH	GF-161-3011	-
2	U1, U2	TL072B	TL072BIDT	-
1	U4	L78LXX	L78L05ABUTR	-
1	C11	100nF	CL21B104KACNNNC	>=25V, X5R/X7R
1	C8	10uF	CL31A106KAHNFNE	>=25V, X5R/X7R
2	C12, C13	1uF	CL21B105KAFNNNG	>=25V, X5R/X7R
2	R4, R9	RS	RC0805FR	>=1/8W, 1%
1	R5	R1	RC0805FR	>=1/8W, 1%
1	R6	R2	RC0805FR	>=1/8W, 1%
1	U3	TL431B	TL431BID	-
1	U5	LM4040	LM4040CYM3-5.0-TR	-
1	U6	REF50XX	REF5050AIDR	-