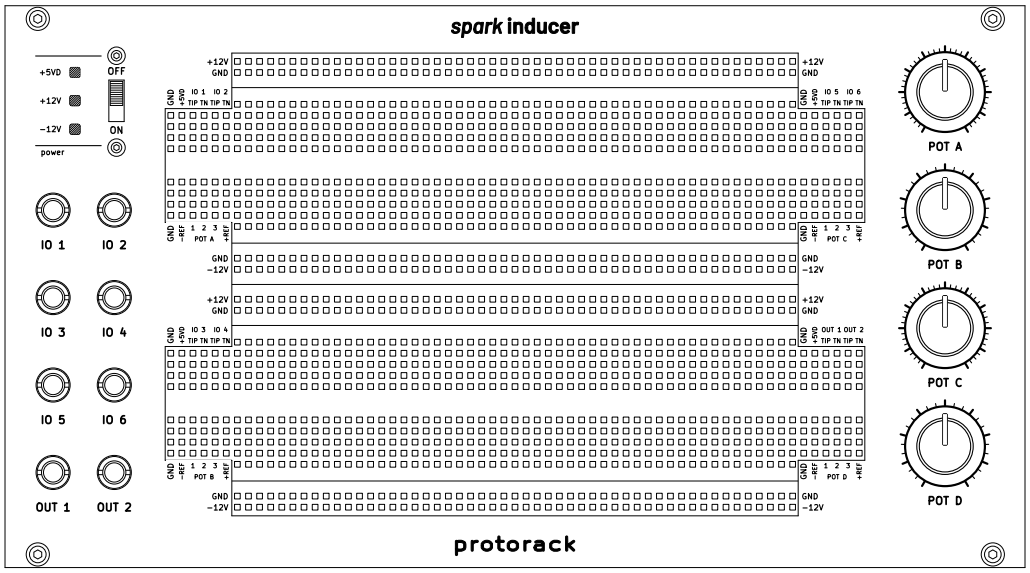
spark inducer - ASSEMBLY MANUAL (Rev.3)



Designed by Max Schlecht

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 would 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.

Warning:

Even if you are an experienced builder, make sure to follow the build steps in the right order when building this kit. Some sections are very dense and will be almost impossible to assemble, if you do them in the wrong order.

If you need any help with the build or have any question/remarks about this guide, feel free to message us via the support page at: https://protorack.de/support

Happy soldering!

Required Tools

- Tweezers
- Side Cutters
- 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.
- Eye Protection

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

0 An Introduction into SMD Soldering - P-02 LFO

The spark inducer full kit comes with a few extra components, as well as two P-02 LFO breadboard helpers. These are meant as a quick (re)introduction/warm-up to do before soldering the main PCB. They are also really useful as a quickly set up modulation source for experimenting with your own circuits.

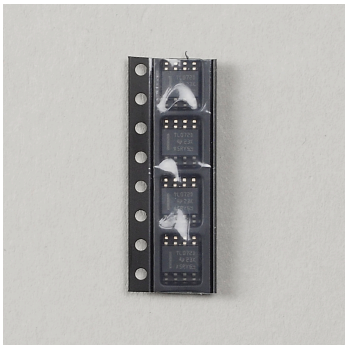
If you are not building a full kit, you can skip this step, although reading through the Basic SMD Soldering Technique section is still recommended, especially if you are new to SMD soldering.

0.1 Handling SMD Components

The best tools to handle SMD components are a pair of tweezers and a steady hand. They are generally very light, so always handle them carefully and only apply the minimum force necessary with your tweezers. Too much force can easily send them flying across your room, which is not exactly dangerous, but good luck finding them again afterwards.

Most SMD components come in either paper or plastic tapes like this:

Paper tape



Plastic tape

Markings on a SMD resistor

They can easily be mixed up, so only take them out of the tape right before using them. To do this, carefully remove the transparent cover tape (you can use your tweezers for this) and drop them onto a clean working surface.

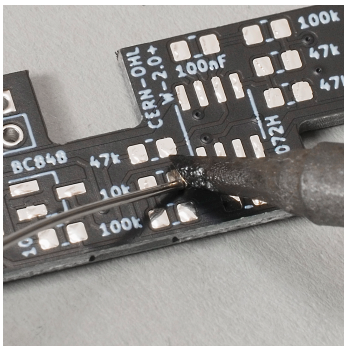
Most (but not all) SMD components also come with some markings on them, see above, which you can use to identify them if required (magnification makes this way easier). For resistors and capacitors (if your multimeter supports capacitance) you can also try to measure their value.

0.2 Basic SMD Soldering Technique

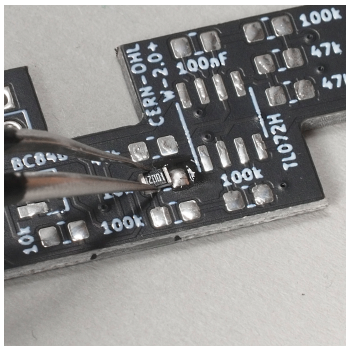
Note: This only one of many possible ways to hand-solder SMD components, but one that does not require any special tools and is reasonably easy to learn and implement.

1. Apply some solder to one of the pads on the PCB.
2. Pick up the component using a pair of tweezers (using your weak hand).
3. Reheat the solder on the PCB and place the component.
4. Retract your soldering iron while holding the component in place, until the solder solidifies again.
5. Check the component placement and repeat 3 - 4, until you are satisfied.
6. Solder all the remaining pads. Optionally reflow (add some fresh solder) the first solder joint as well.
7. Check your work for any visible short-circuits and reflow the affected pads.

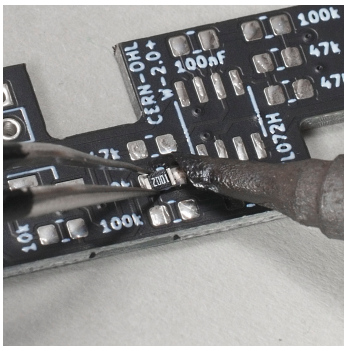
Make sure to heat all pads long enough so that the solder has time to flow into place. If this does not happen, try to increase your soldering temperature or add some fresh solder or flux.



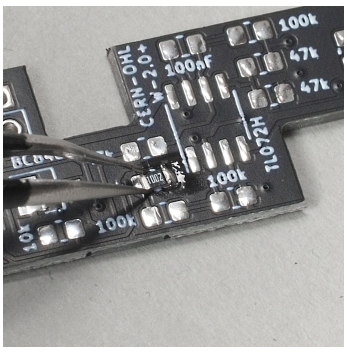
Step 1.



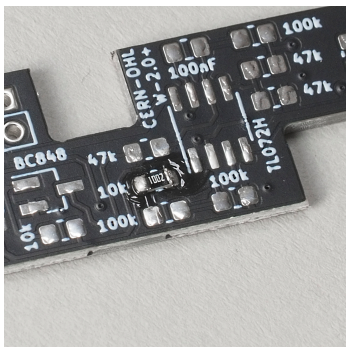
Step 2.



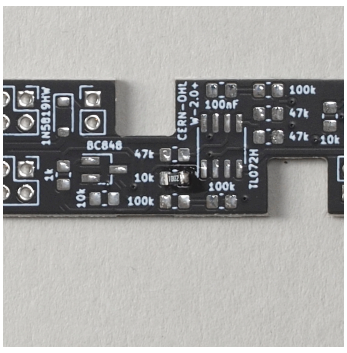
Step 3.



Step 4.



Step 5.

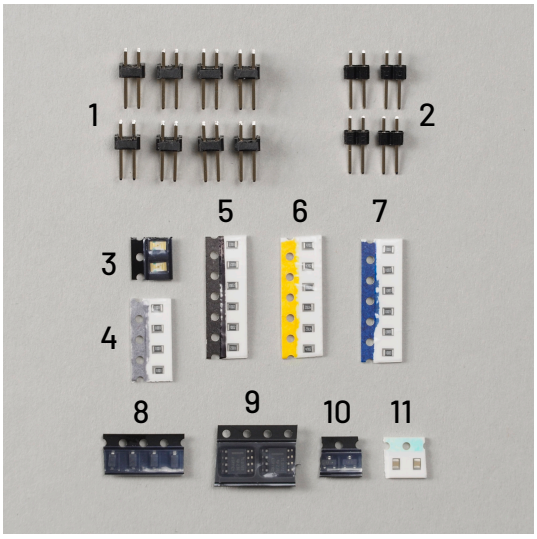


Step 6.

Note: Some pads (especially ones connected to GND) might require more heat / longer heating times to solder properly. This is in most cases, because they are connected to a large copper plane on the PCB, which sucks away the heat from the connection.

0.3 P-02 LFO

Note: If you have a full kit, you will only find the PCBs and headers in the P-02 LFO bag. The remaining SMD components are in the main SMD Components bag.

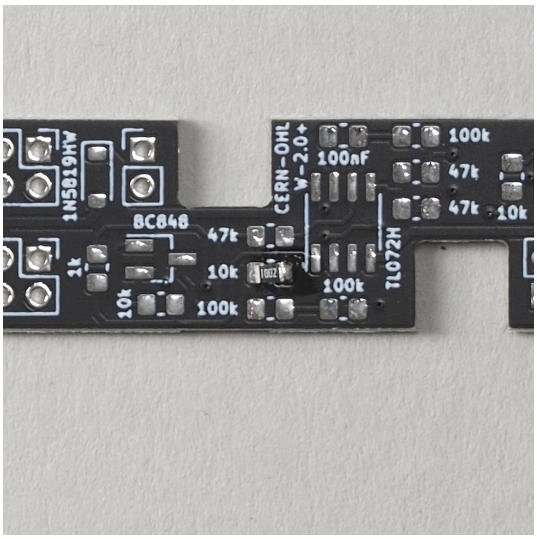


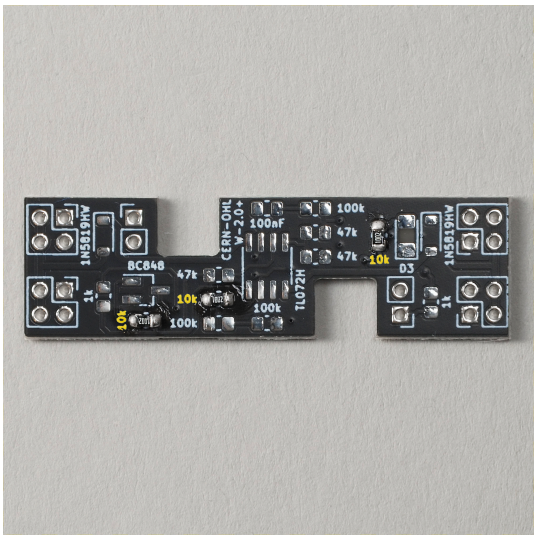
ID	Qty	Description	Code on Part	PCB Identifier	Ref
1	4	2×2 Header	-	-	J1, J2, J3, J4
2	2	2×1 Header	-	-	J5, J6
3	1	Red LED, 1206	-	D3	D3
4	2	1k Resistor, 0805 1% 1/4W	1001	1k	R8, R11
5	3	10k Resistor, 0805 1%	1002	10k	R5, R7, R10
6	3	47k Resistor, 0805 1%	4702	47k	R2, R3, R4
7	3	100k Resistor, 0805 1%	1003	100k	R1, R6, R9
8	2	1N5819HW Schottky Diode, SOD-123	SL	1N5819HW	D1, D2
9	1	TL072H OpAmp, SOIC8	TL072D	TL072H	U1
10	1	BC848 NPN Transistor, SOT-23	1K	BC848	Q1
11	1	100nF Capacitor, 0805	-	100nF	C1

P-02 LFO - SMD Components

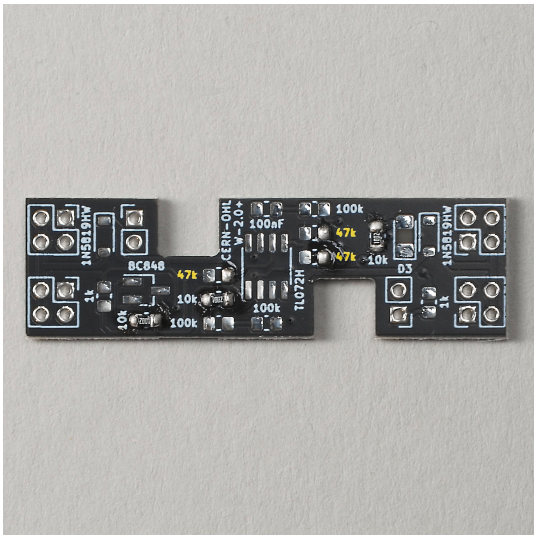
	ID	Qty	Description	Code on Part	PCB Identifier	Ref
	5	3	10k Resistor, 0805 1%	1002	10k	R5, R7, R10
	6	3	47k Resistor, 0805 1%	4702	47k	R2, R3, R4

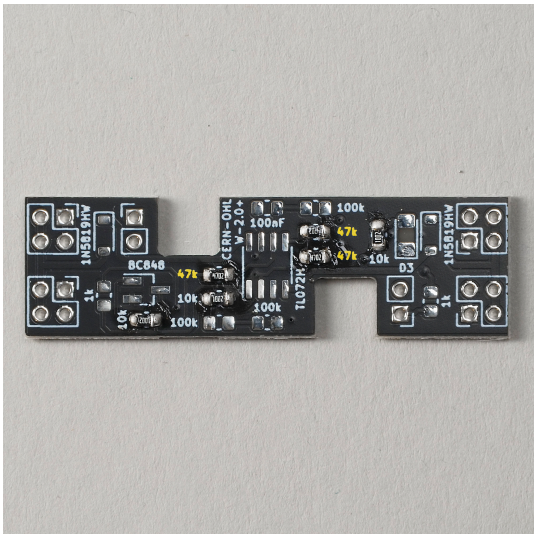
Start by soldering the 10k and 47k resistors. Use the previously described technique and add some solder to one pad, then reheat it and put the resistor in place with your tweezers and solder the second pad. Repeat this for all resistors.





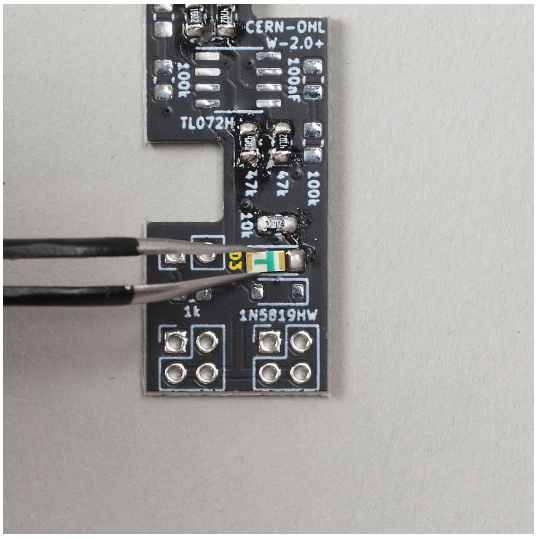
Note: You can streamline this by adding solder to one pad of each of the resistors first, then reheat and drop them into place one after the other and finally solder all the remaining pads. This way, you do not have to switch between solder and tweezers that often.

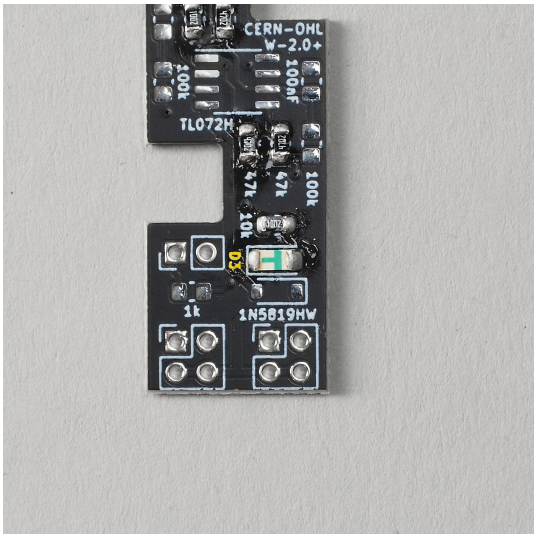




	ID	Qty	Description	Code on Part	PCB Identifier	Ref
	3	1	Red LED, 1206	-	D3	D3
	8	2	1N5819HW Schottky Diode	SL	1N5819HW	D1, D2

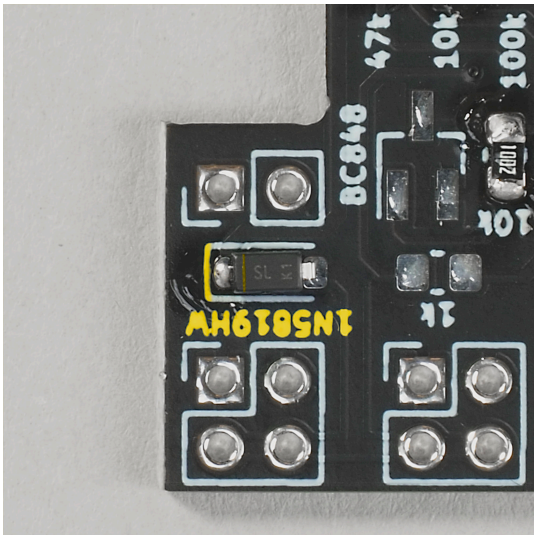
Next up is the indicator LED. This needs to be soldered upside-down, so it can shine through the PCB substrate. Orient it as shown in the picture below, with the green T-shaped marking facing up and solder it in place. **Make sure the solder is properly bridging the gap between the LED and the pads on the PCB.**

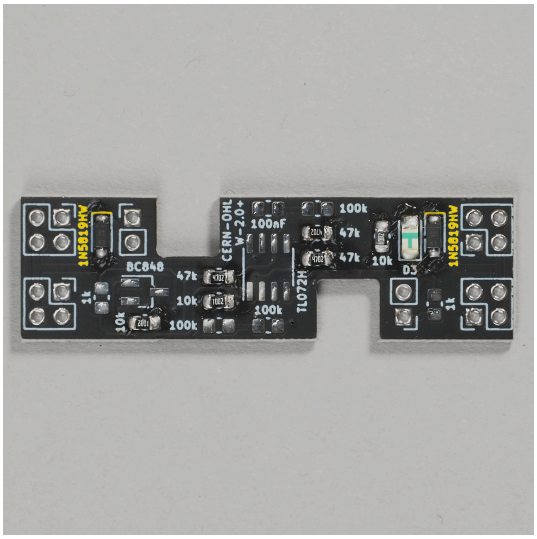




Then, also solder the two Schottky diodes. Orientation matters for these, so you will have to align the laser-etched marking line on them with the white silkscreen on the PCB as shown below.

Note: Try looking at the diodes under a strong light source, with magnification. Changing the angle the light is coming from can also help you to spot it.





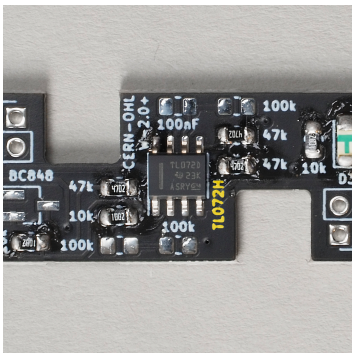
	ID	Qty	Description	Code on Part	PCB Identifier	Ref
	9	1	TL072H OpAmp, SOIC8	TL072D	TL072H	U1
	10	1	BC848 NPN Transistor	1K	BC848	Q1

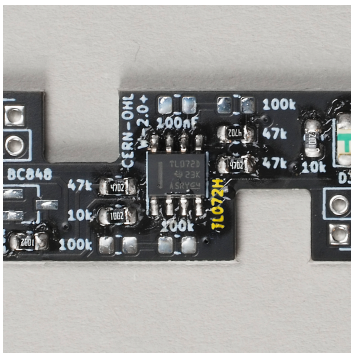
For the TL072H OpAmp orientation also matters. The white line on the Package has to be on the same side as the longer white line on the PCB.

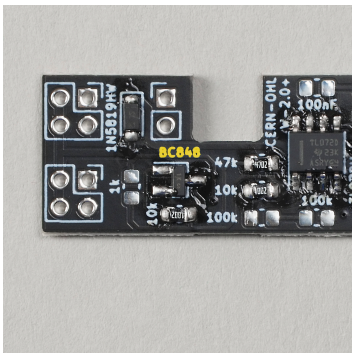
Solder one pin in place and then carefully check the alignment of all the other ones. Reheat the pin and reorient it if necessary. If it is rotated a little, you can also get away with very carefully bending it into place using your tweezers.

Once the alignment looks good, solder the remaining pins, one by one and check that there are no short-circuits between them.

Do the same for the BC848 transistor. Make sure it is centered well.

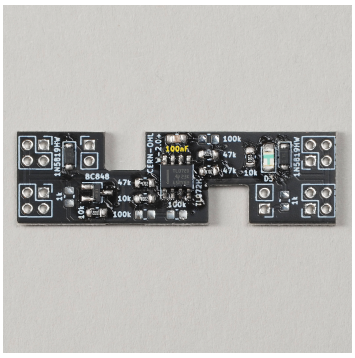
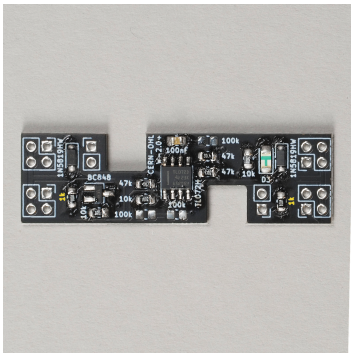
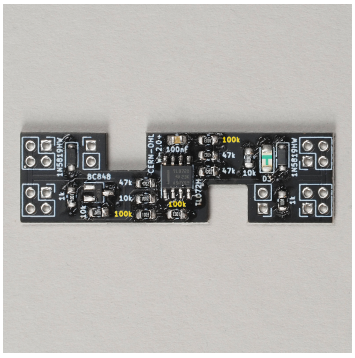






	ID	Qty	Description	Code on Part	PCB Identifier	Ref
	11	1	100nF Capacitor, 0805	-	100nF	C1
	4	2	1k Resistor, 0805 1%	1001	1k	R8, R11
	7	3	100k Resistor, 0805 1%	1003	100k	R1, R6, R9

Finish the SMD assembly by soldering the 100nF capacitors, as well as the remaining 1k and 100k resistors.

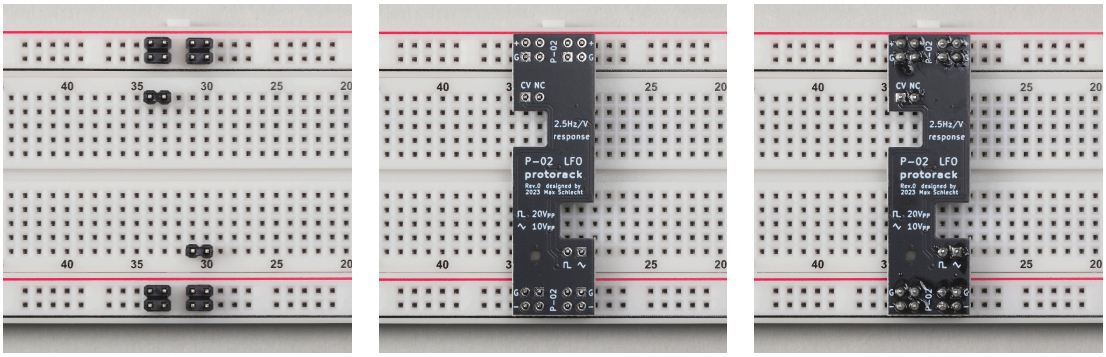




P-02 LFO - Headers

	ID	Qty	Description	Ref
	1	4	2×2 Header	J1, J2, J3, J4
	2	2	2×1 Header	J5, J6

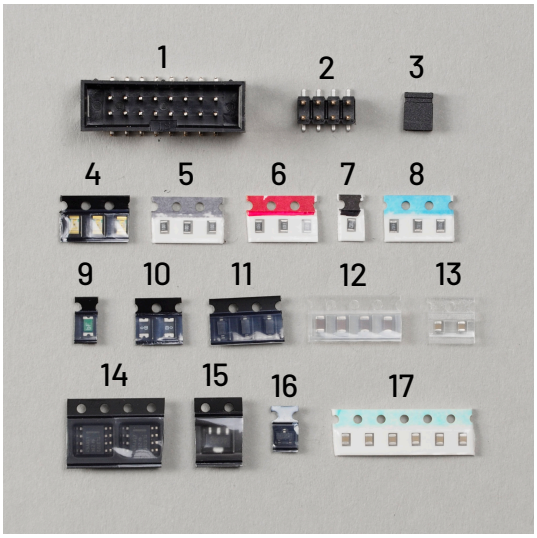
To solder the headers, it is best to use a breadboard to align everything (you can use one of the ones that come with the full kit). Insert all the headers into the breadboard as show below. Pay attention to not use too much force when inserting them into the breadboard, the metal pins can slide through the black plastic part.

Then, put the PCB (with all SMD components assembled) on them, making sure **the component side is facing down**. Finally, solder all the headers in place.



If you have successfully finished the first P-02 LFO board, you can go ahead and follow the next steps to solder the main PCB. After that is done you should be left with a few extra spare parts, which you can then use to build a second LFO board.

1 SMD Components



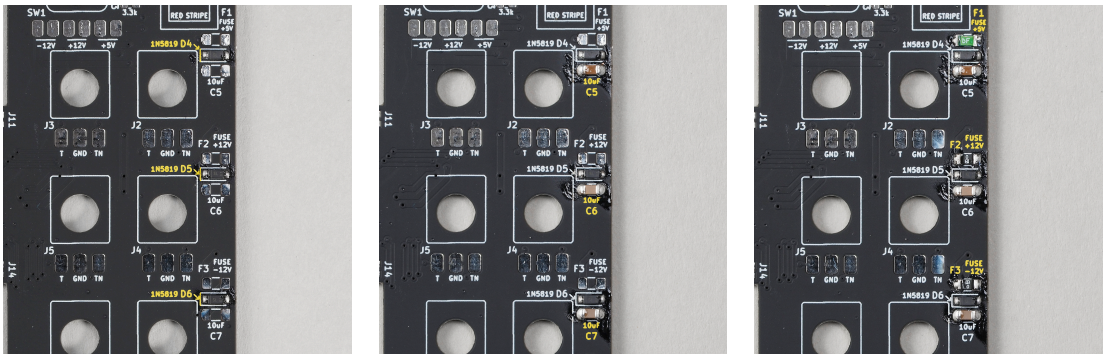
ID	Qty	Description	Code on Part	PCB Identifier / Ref
1	1	2×8 IDC Connector	-	J1
2	1	2×4 Header	-	J18
3	1	Jumper	-	-
4	3	Red LED, 1206	-	D1, D2, D3
5	3	1k Resistor, 0805 1% 1/4W	1001	R1, R7, R8
6	2	3.3k Resistor, 0805 1%	3301	R2, R3
7	1	10k Resistor, 0805 1%	1002	R9
8	2	33k Resistor, 0805 0.1%	3302	R10, R11
9	1	200mA Resettable Fuse, 1206	bF	F1
10	2	100mA Resettable Fuse, 1206	0	F2, F3
11	3	1N5819HW Schottky Diode, SOD-123	SL	D4, D5, D6
12	3	10uF Capacitor, 1206	-	C5, C6, C7
13	1	330nF Capacitor, 0805	-	C9
14	2	TL072H OpAmp, SOIC8	TL072D	U1, U2
15	1	L78L05 Voltage Regulator, SOT-89	8C	U4
16	1	LM4040 Voltage Reference, SOT-23	Y2C	U5
17	6	100nF Capacitor, 0805	-	C1, C2, C3, C4, C10, C11

1.1 Power Protection/Decoupling

	ID	Qty	Description	Code on Part	PCB Identifier
	11	3	1N5819HW Schottky Diode, SOD-123	SL	D4, D5, D6
	12	3	10uF Capacitor, 1206	-	C5, C6, C7
	10	2	100mA Resettable Fuse, 1206	0	F2, F3
	9	1	200mA Resettable Fuse, 1206	bF	F1

Start by soldering the 1N5819HW Schottky diodes and 10uF decoupling capacitors for each of the three power rails. **Mind the polarity of the diodes! The white line on them has to align with the line on the PCB.**

Then, do the same for the resettable PTC fuses. The two black ones are for the +12V and -12V power rails and can hold a maximum current of 100mA indefinitely and will trip at ~250mA. The green 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 supply their rated trip current!**)

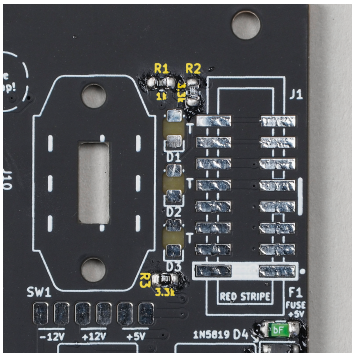
1.2 Power Connector, LEDs

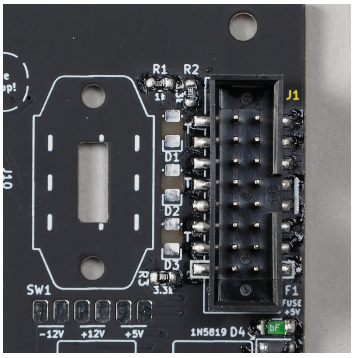
	ID	Qty	Description	Code on Part	PCB Identifier
	6	2	3.3k Resistor, 0805 1%	3301	R2, R3
	5	1	1k Resistor, 0805 1%	1001	R1
	1	1	2×8 IDC Connector	-	J1
	4	3	Red LED, 1206	-	D1, D2, D3

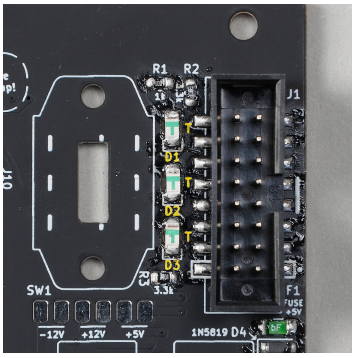
Start by soldering the two 3.3k resistors and the 1k resistor.

Next is the power connector. Align it carefully, **minding its orientation (there is 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 do not 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 is 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!**





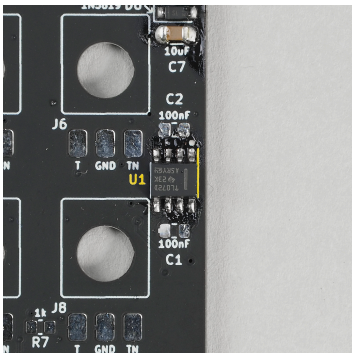


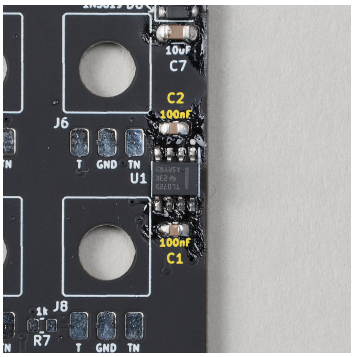
1.3 Buffered Outputs

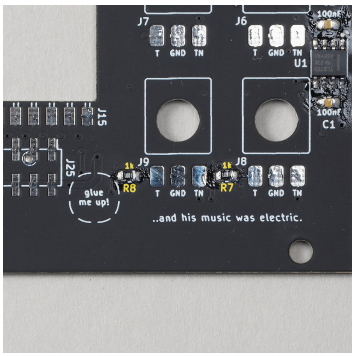
	ID	Qty	Description	Code on Part	PCB Identifier
	14	1	TL072H OpAmp, SOIC8	TL072D	U1
	17	2	100nF Capacitor, 0805	-	C1, C2
	5	2	1k Resistor, 0805 1% 1/4W	1001	R7, R8

First, solder the TL072H. 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.



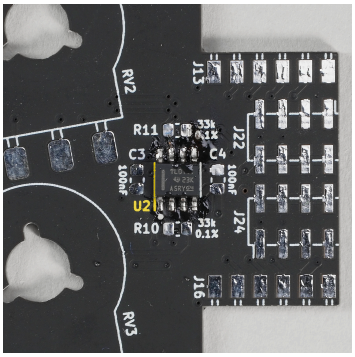


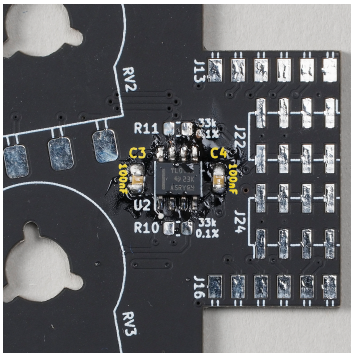


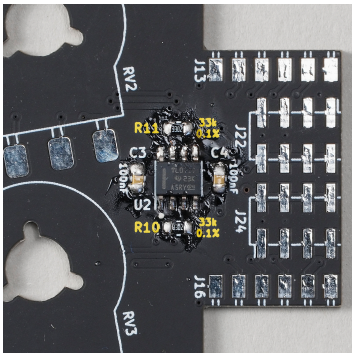
1.4 Reference Voltage Buffer

	ID	Qty	Description	Code on Part	PCB Identifier
	13	1	TL072H OpAmp, SOIC8	TL072D	U2
	17	5	100nF Capacitor, 0805	-	C3, C4
	8	2	33k Resistor, 0805 0.1%	3302	R10, R11

Once again, start by soldering the TL072H, then solder the 100nF decoupling capacitors, as well as the 33.0k precision resistors.







1.5 L78Lxx Voltage Reference

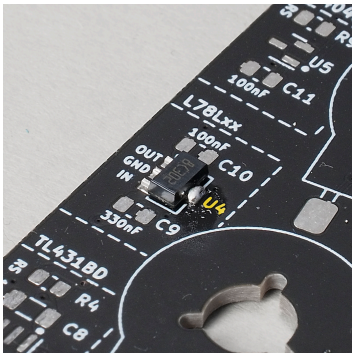
	ID	Qty	Description	Code on Part	PCB Identifier
	15	1	L78L05 Voltage Regulator, SOT-89	8C	U4
	17	1	100nF Capacitor, 0805	-	C10
	13	1	330nF Capacitor, 0805	-	C9

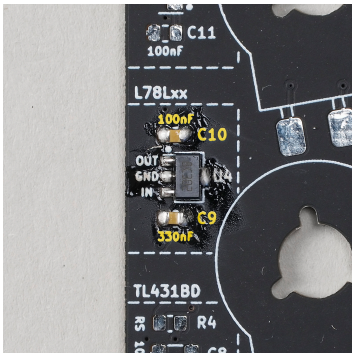
This is the first of two voltage references that come with the full DIY kit. It uses a cheap 78xx regulator to produce a stable but not terribly accurate voltage reference. The particular regulator that comes with the kit is a 5V one, but ones for other voltages are also available.

The output voltage of the voltage reference gets buffered by one channel of the Reference Voltage Buffer, while the other one inverts it to get the negative reference voltage. The buffering/inverting is not necessary per se, but for this use case, the OpAmp offers some protection against damaging the references directly by connecting something in the wrong way. Using a low-offset OpAmp in combination with 0.1% precision resistors, is also almost always more cost-effective than 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).

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





1.6 LM4040 Voltage Reference

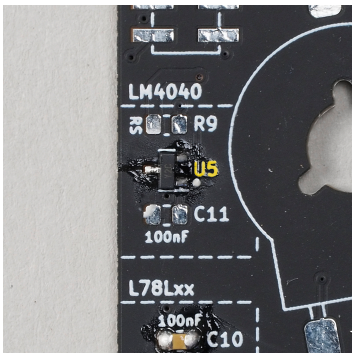
	ID	Qty	Description	Code on Part	PCB Identifier
	16	1	LM4040 Voltage Reference, SOT-23	Y2C	U5
	7	1	10k Resistor, 0805 1%	1002	R9
	17	1	100nF Capacitor, 0805	-	C11
	2	1	2×4 Header	-	J18
	3	1	Jumper	-	-

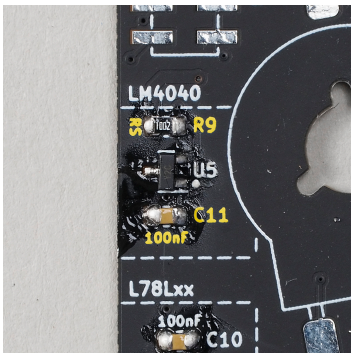
This is the second voltage reference that come with the full DIY kit. It uses a LM4040 Zener reference to produce a very stable and accurate voltage reference. The LM4040 that comes with the kit is a 2.5V, C-Grade variant with 0.5% output Tolerance (other variants are available).

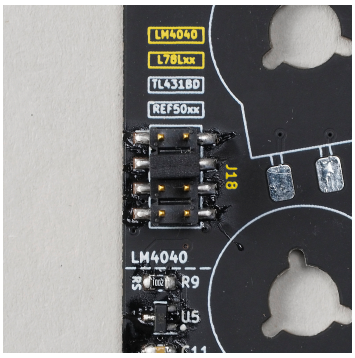
Note: If you are using a LM4040 with a different reference Voltage, make sure to select an appropriate bias resistor, according to Selecting the bias resistor.

Start with the LM4040 reference, then solder the 100nF output capacitor and the 10k bias resistor.

Finally, solder the 2×4 header and add the jumper to select either the L78Lxx or LM4040 reference.







If you are building a kit and do not have any of the parts for the other Voltage References, skip straight to assembling the Breadboard Headers.

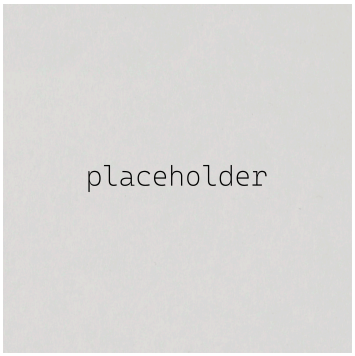
1.7 (extra) TL431B Voltage Reference

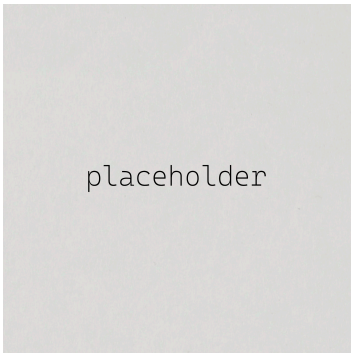
	Qty	Description	Code on Part	PCB Identifier
	1	RS Resistor, 0805	-	R4
	1	R1 Resistor, 0805	-	R5
	1	R2 Resistor, 0805	-	R6
	1	TL431B Programmable Reference, SOIC8	TODO	U3
	1	10uF Capacitor, 1206	-	C8

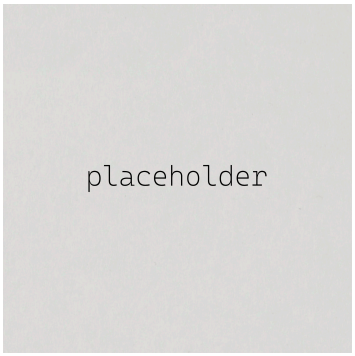
The TL431B is an accurate programmable voltage reference. It is relatively inexpensive and can be setup to produce any output voltage from $2.5V - 36V$. It requires some extra components to work though, namely two (precision) resistors or a trimmer potentiometer to accurately set the desired output voltage, as well as a bias resistor to limit the supplied current and a large output capacitor.

To find suitable values for these, take a look at the Using Alternative Voltage References Section.

Start by soldering the TL431B reference, then solder the output capacitor and the resistors.







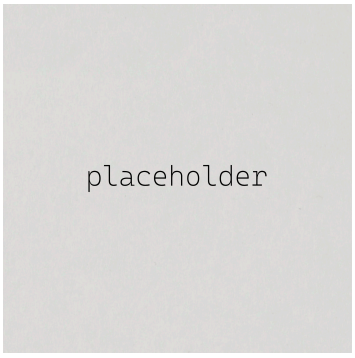
1.8 (extra) REF50xx Voltage Reference

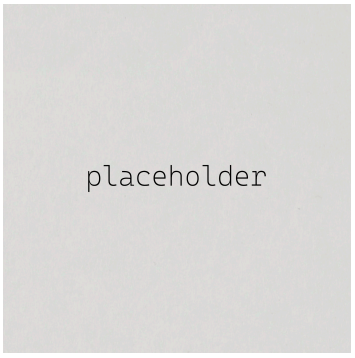
	Qty	Description	Code on Part	PCB Identifier
	1	REF50xx Voltage Reference, SOIC8	TODO	U6
	2	1uF Capacitor, 0805	-	C12, C13

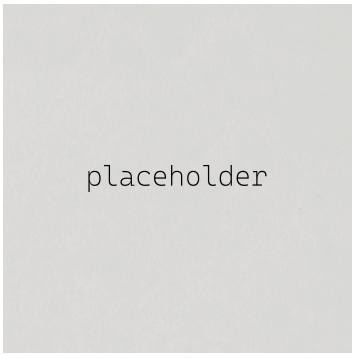
The REF50xx is a fully contained, high precision Voltage Reference. It is the priciest of all the options, but only requires two additional input/output capacitors to function.

Any of the available references from Texas Instruments will work.

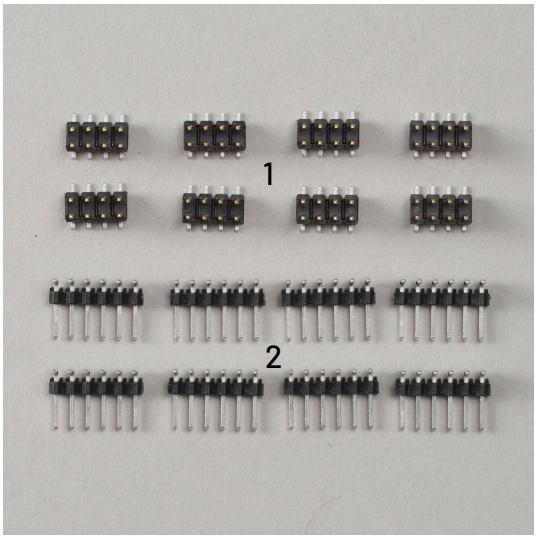
Solder the REF50xx reference IC first, then solder the input and output decoupling capacitors.

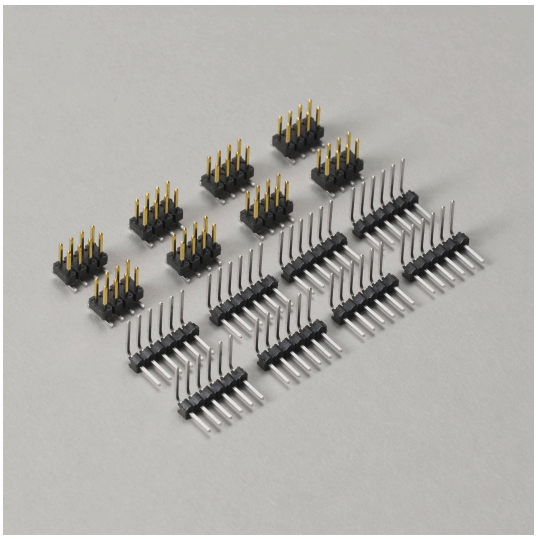






2 Breadboard Headers





ID	Qty	Description	PCB Identifier / Ref
1	8	2×4 Header	J19, J20, J21, J22, J23, J24, J25, J26
2	8	1×6 Right Angle Header	J10, J11, J12, J13, J14, J15, J16, J17

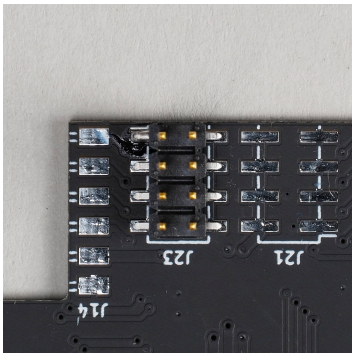
2.1 Power Headers

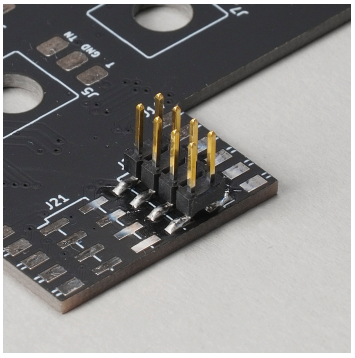
	ID	Qty	Description	PCB Identifier
	1	8	2×4 Header	J19, J20, J21, J22, J23, J24, J25, J26

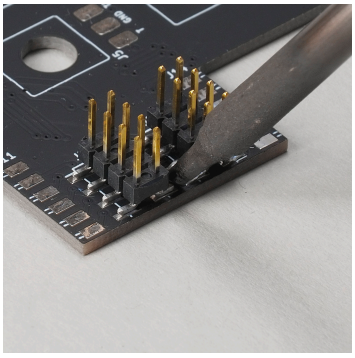
Take your time on soldering these, as they have to be aligned reasonably well, for the breadboard to fit. It does not have to be perfect, but there should not be any obvious rotations or offsets larger than ~1mm.

To make this easier, tin one pin first, 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.



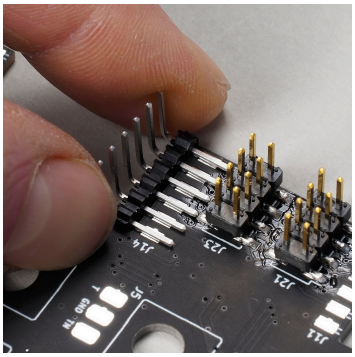
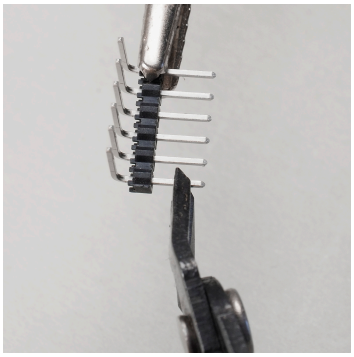
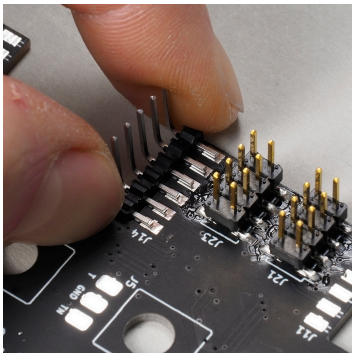




2.2 Right Angle Headers

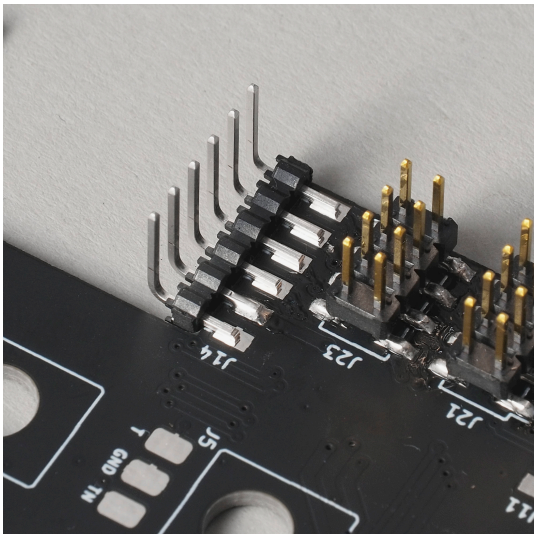
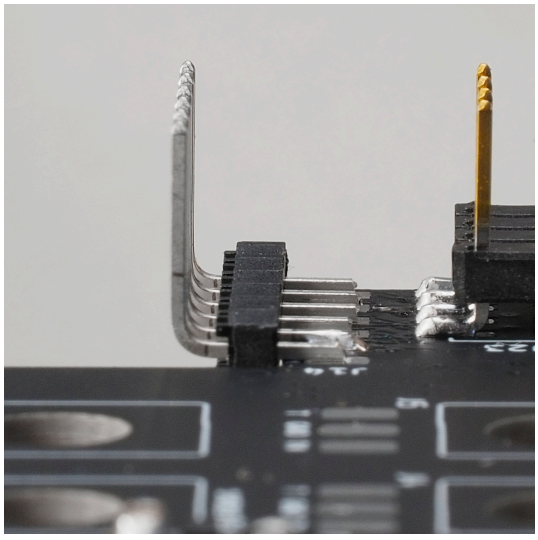
	ID	Qty	Description	PCB Identifier
	2	8	1×6 Right Angle Header	J10, J11, J12, J13, J14, J15, J16, J17

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 do not interfere with the power headers.

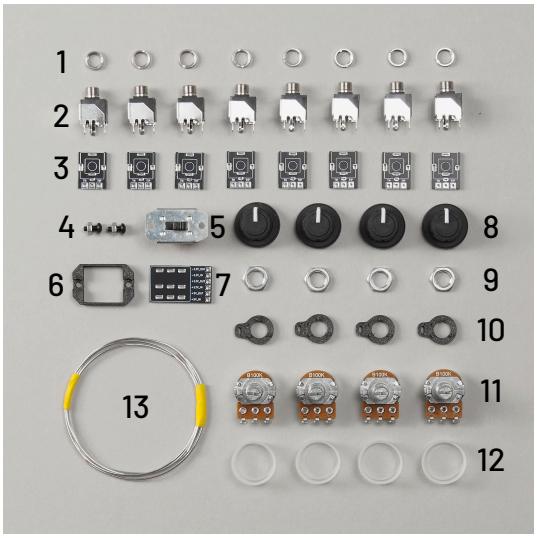
Then, for each header, tin one of the pads, reheat it and carefully push the header into the corner of the PCB. Make sure they are not crooked and sit flat on the PCB without any vertical offset.

As before, these have to be aligned reasonably well, although it is 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 / Ref
1	8	Knurled Nut	-
2	8	WQP518MA-BM Audio Jack	J2, J3, J4, J5, J6. J7, J8, J9
3	8	Audio Jack PCB	-
4	2	M2 Screw, Nut	-
5	1	Power Switch	SW1
6	1	Power Switch Bracket	-
7	1	Power Switch PCB	-
8	4	Potentiometer Knob	-
9	4	Potentiometer Nut	-
10	4	Potentiometer Spacer	-
11	4	100k Potentiometer (linear)	RV1, RV2, RV3, RV4
12	4	Potentiometer Dust Covers	-
13	1	0.8mm Tinned Copper Wire (75cm)	-

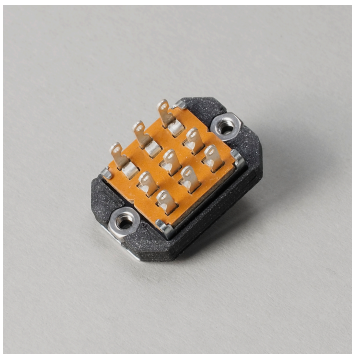
3.1 Power Switch

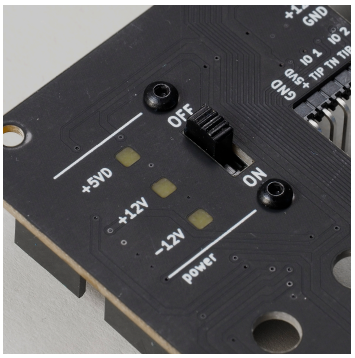
	ID	Qty	Description	PCB Identifier
	4	2	M2 Screw, Nut	-
	6	1	Power Switch Bracket	-
	5	1	Power Switch	SW1
	7	1	Power Switch PCB	-

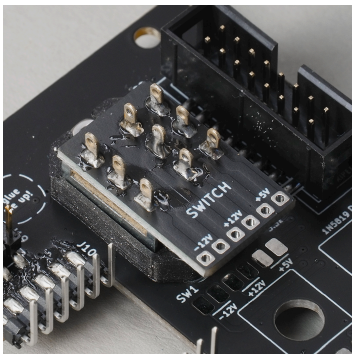
Insert the two M2 nuts into the 3D printed plastic bracket and slip it onto the switch (with the M2 nuts facing away from the switch).

Next, hold the switch against the main 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.**

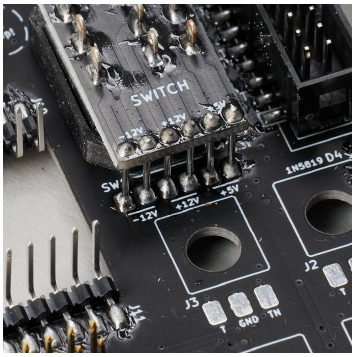
Add the Power Switch PCB to the switch (blank side up), push it down and solder down all the pins.

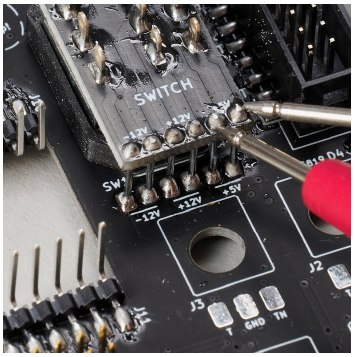


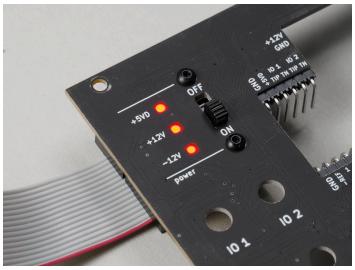




Snip off six **~12mm** pieces of the solid core wire and put each of them through one of the through-holes on the switch PCB. Fix them in place by soldering them to the pad on the main PCB first, then make sure they are straight and solder the pad on the switch PCB.







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

Note: At this point it is also a good idea to perform a quick smoke test to confirm the power input circuit is working. Connect the PCB to an eurorack power supply via the included cable and confirm all three LEDs are lighting up.

3.2 3.5mm Jacks

	ID	Qty	Description	PCB Identifier
	1	8	Knurled Nut	-
	2	8	WQP518MA-BM Audio Jack	J2, J3, J4, J5, J6. J7, J8, J9
	3	8	Audio Jack PCB	-

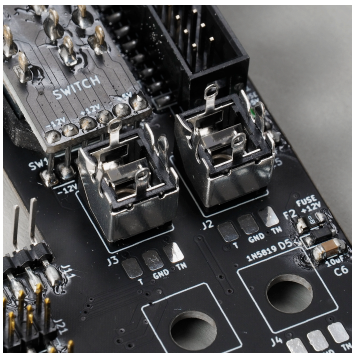
Important:

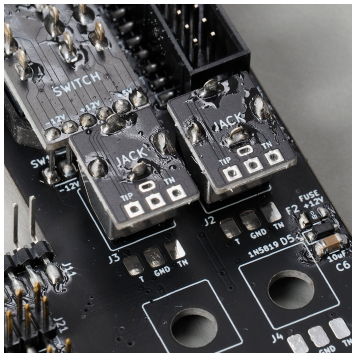
Do each of the following steps for each pair/row of audio jacks, **one pair/row at a time, from top to bottom**. Once a row is assembled, it will be **nearly impossible to reach the one above it**. Make sure your soldering is solid, and there are no shorts.

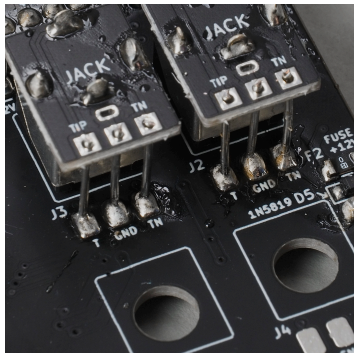
Take two audio jacks and knurled nuts and attach them to the top two holes of the main PCB. Align them so that they fit into the box marked on the back side of the main PCB.

Add two audio jack PCBs (with the side that says JACK facing up) to the audio jacks and solder all the pins.

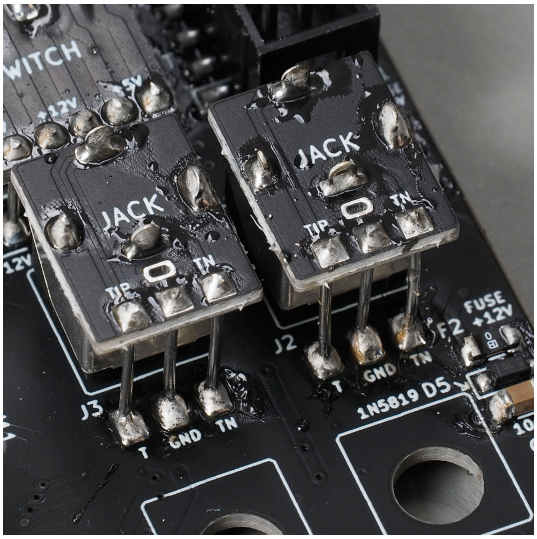
Note: The audio jack PCBs come together on one panel PCB. Break them apart at the V-Scores. To remove the tabs between them, a pair of pliers can be helpful.

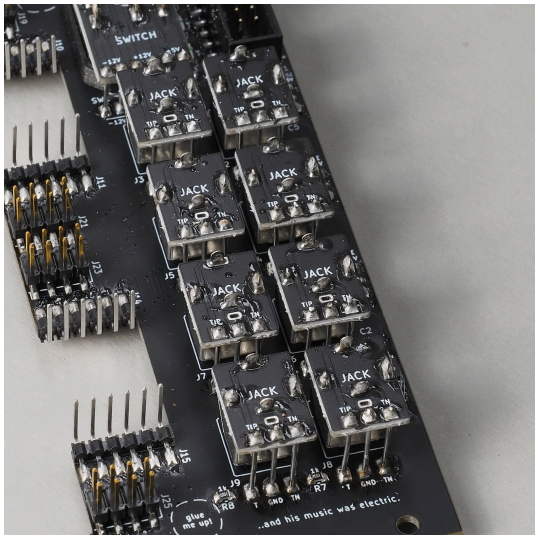






Next, snip off six **~15mm** pieces of the solid core wire and put each of them through one of the through-holes on the audio jack PCBs. Fix them in place by soldering them to the pad on the main PCB first, then make sure they are straight and solder the pad on the audio jack PCB.





Use your multimeter in continuity mode to ensure you did not create any short-circuits. There should not be any continuity between any of the wires.

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

3.3 Potentiometers

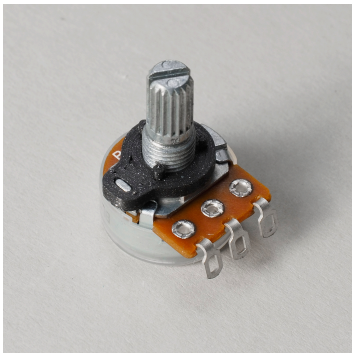
	ID	Qty	Description	PCB Identifier
	8	4	Potentiometer Knob	-
	9	4	Potentiometer Nut	-
	10	4	Potentiometer Spacer	-
	11	4	100k Potentiometer (linear)	RV1, RV2, RV3, RV4
	12	4	Potentiometer Dust Covers	-

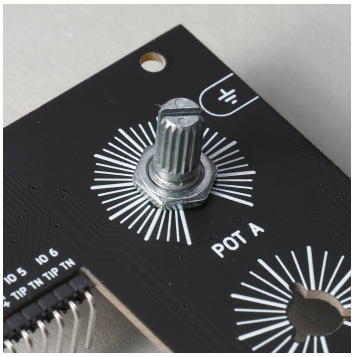
Note: While not as critical as the audio jacks, this part is also easier to do if you follow the steps for each potentiometer, one by one, from top to bottom.

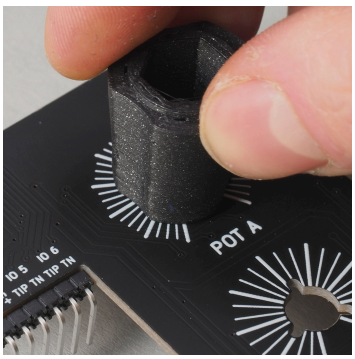
Attach the plastic dust cover and 3D printed spacer to the potentiometer.

Next, attach it to the PCB. There are three little nubbins on the 3D printed spacer, which index into the PCB and help to align the rotation of the potentiometer.

Use the included 3D printed tool to screw down the potentiometer to the PCB with the hex nut. Do not overtighten and make sure the potentiometer aligns with the pads on the back of the PCB nicely.



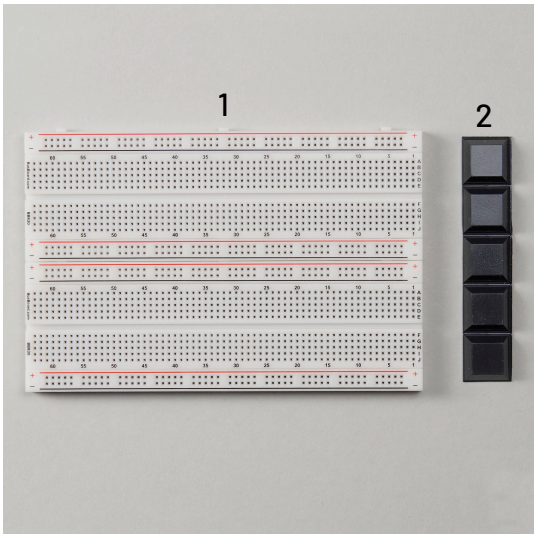


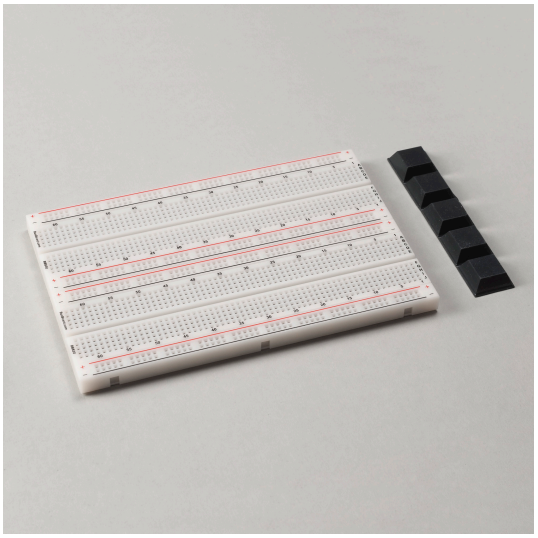


Then, bend down the solder lugs until they touch the PCB and solder them in place.

Finally, turn the potentiometer all the way to the left and attach the knob so that it aligns with the first line on the scale.

4 Breadboard





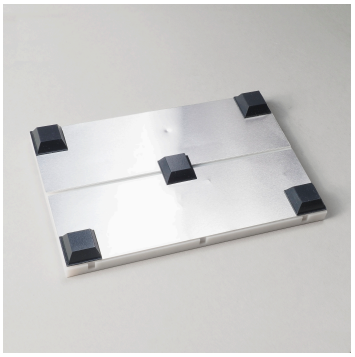
	ID	Qty	Description
	1	2	BB830 type breadboard
	2	5	rubber feet

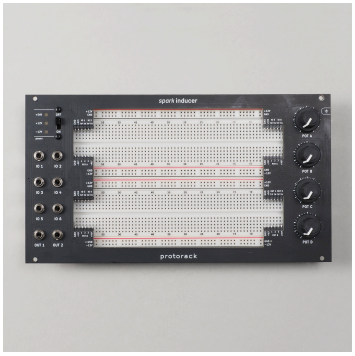
Take the two breadboards and attach the included metal backing plates to the adhesive on their backs. Then, combine them by pushing them together using the nubbins and slots on their sides.

Next, turn them around and attach the rubber feet to the bottom. Put one in each corner and one in the center, connecting both breadboards.

Take the completed PCB and align the headers to the breadboards. Then, carefully press the two together to finish the assembly!

Note: Do not push in all the headers at once. Do all the corners one by one and then finish with the center.

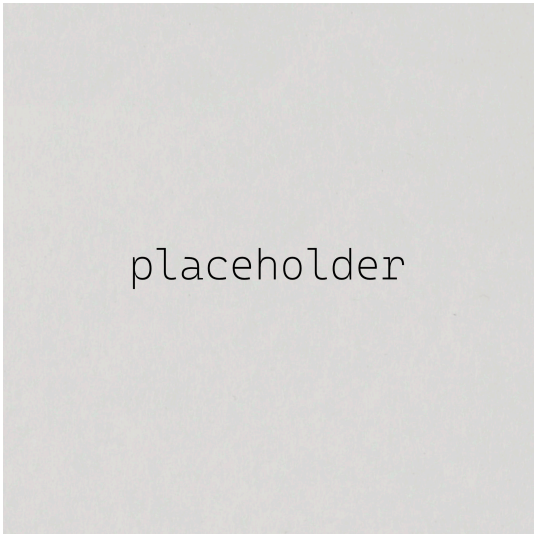


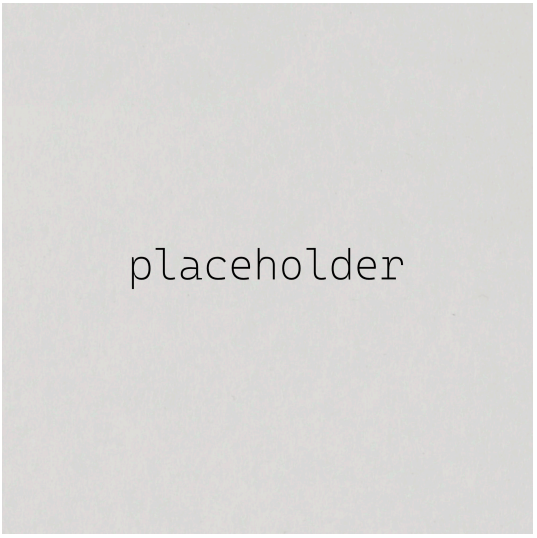


Congratulations! You are done with the main build. Next, checkout the user manual!

4.1 (optional) Gluing

If you want to permanently put the spark inducer into a eurorack case, it is highly recommended that you glue the breadboards to the main PCB. To do this, simply add a couple of blobs of hot-glue to the marked glue spots, as shown below.





Using Alternative Voltage References

L78Lxx

Any pin compatible, SOT-89 package Low-Dropout-Regulator can be used in place of the default L78L05, as long as $V_D < 12V - V_O$ (V_D and V_O being the regulators Dropout and Output-Voltage).

LM4040

Any pin compatible, SOT-23-3 package LM4040 voltage reference can be used in place of the default one, but the bias resistor will have to be adjusted.

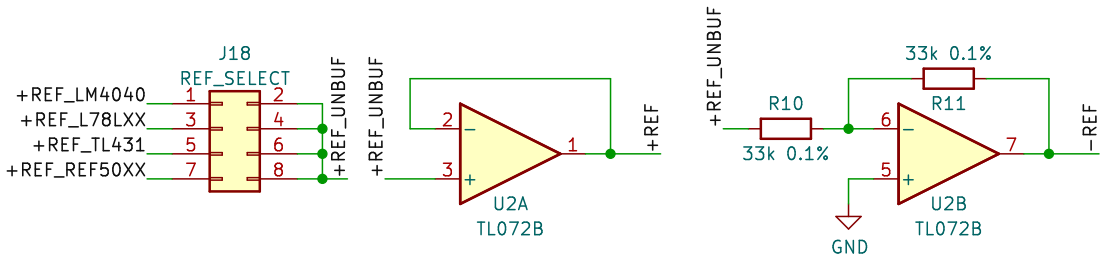
Selecting the bias resistor

The bias resistor connects between the LM4040s cathode and the supply voltage and sets the available output/load current I_L, as well as its bias current I_Z.

The bias current is required to make the reference work and has to be within its operating current range, which is $\sim 100\mu A < I_Z < 15mA$ (depending on the exact reference used). We want to keep the bias current rather low to avoid the reference heating up too much, while staying within the range and keeping some distance to the extremes, so we target:

$$I_Z = 1mA \tag{a}$$

The load current is dependent on what is connected to the output of the reference. On the spark inducer all voltage references are being buffered by two OpAmps:



Thus, ignoring the input leakage currents of the OpAmps, the only load attached to the voltage reference is R10, which is connected to a virtual ground, so the load current is constant and equal to:

$$I_L = \frac{V_Z}{33k\Omega} \tag{b}$$

with V_Z being the LM4040s Output-/Reference-Voltage.

To calculate the value of the bias resistor, the LM4040s datasheet specifies the equation:

$$R_S = \frac{V_S - V_Z}{I_L + I_Z} \tag{1}$$

So for our application, with (a), (b) and $V_S = 12V$ we get:

$$R_S = \frac{12V - V_Z}{\frac{V_Z}{33k\Omega} + 1mA} \tag{2}$$

You can use this equation to calculate the approximate value for the bias resistor of the spark inducers LM4040 voltage reference. Then select the closest E12 resistor value (or the closest resistor value you have on hand) and confirm that the operating current is $\sim 1mA$ with:

$$I_Z = \frac{V_S - V_Z}{R_{Selected}} - I_L \quad \Rightarrow \quad I_Z = \frac{12V - V_Z}{R_{Selected}} - \frac{V_Z}{33k\Omega} \tag{3}$$

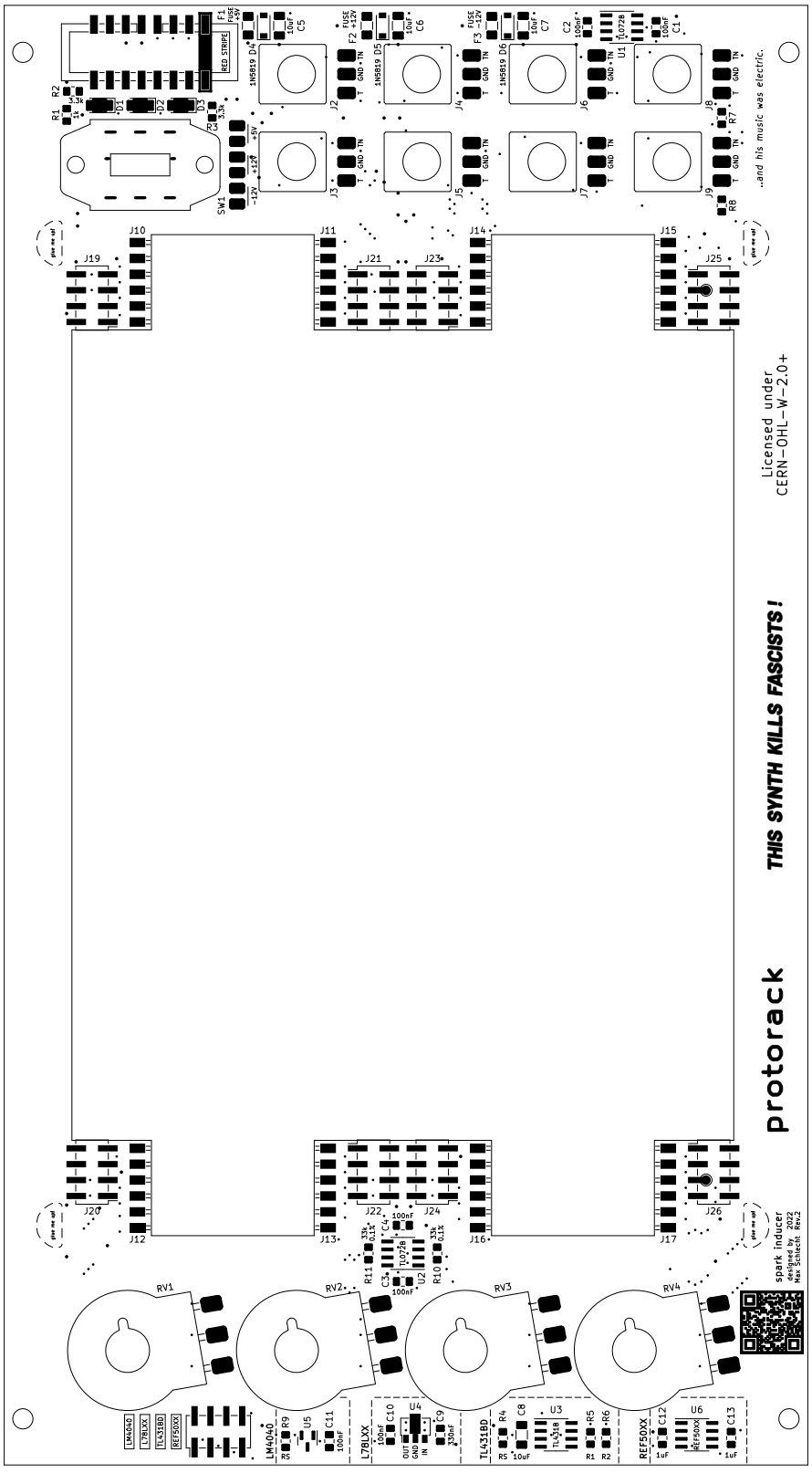
TL431B

TODO

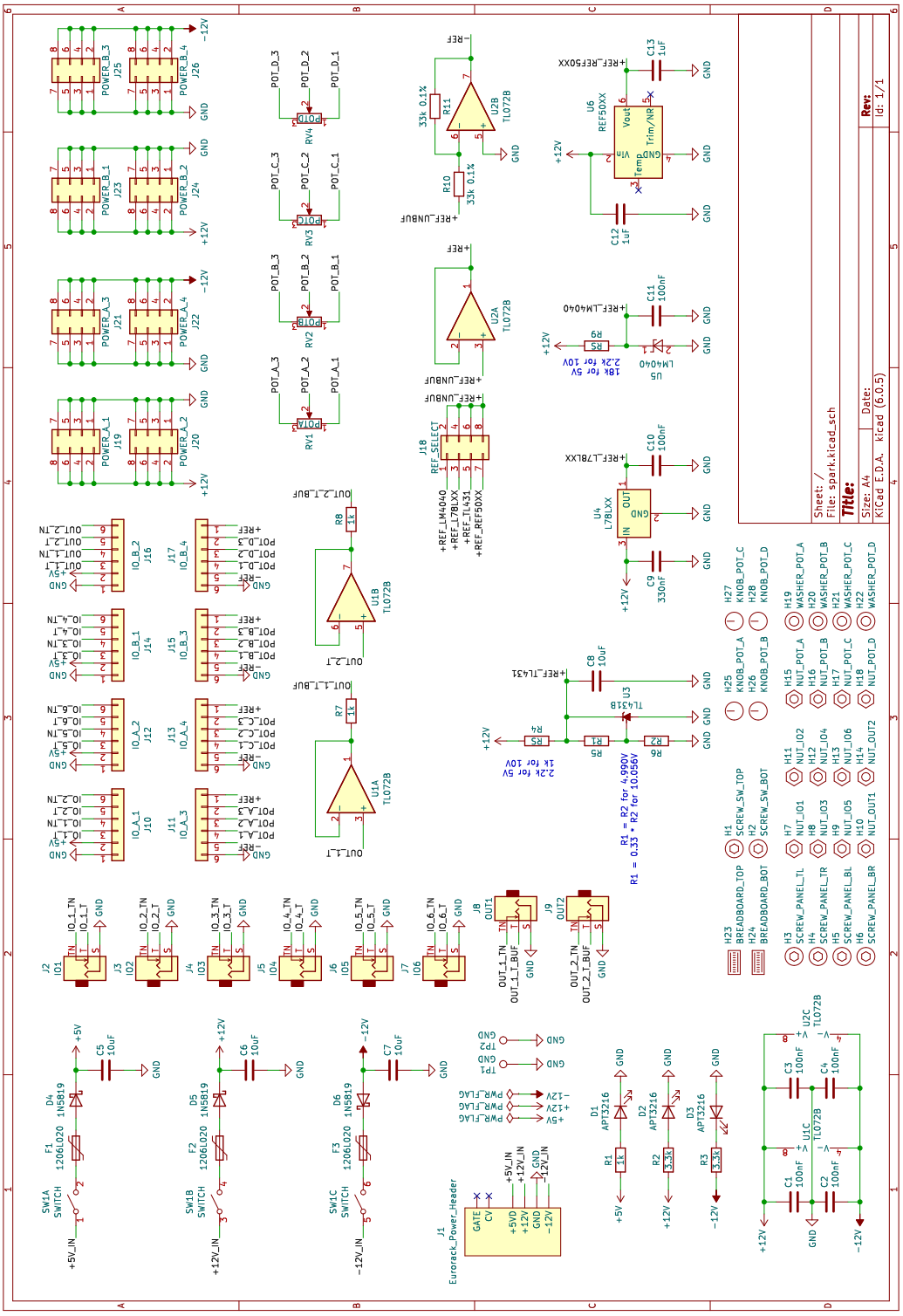
REF50xx

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.

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	-