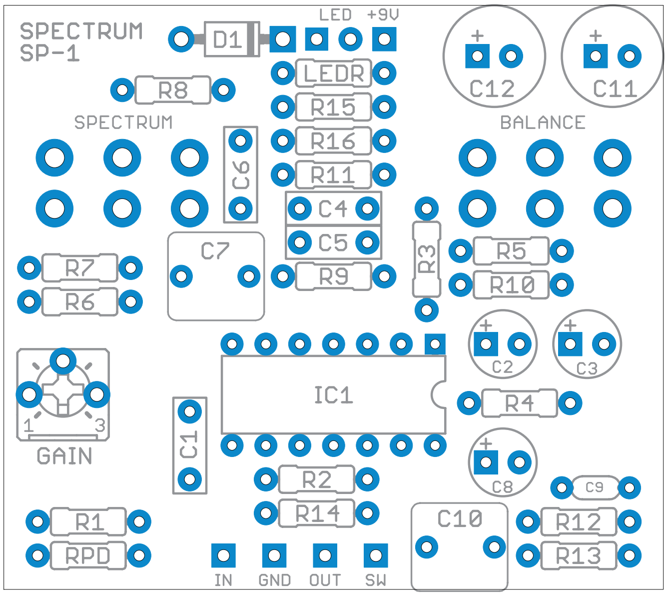
Chroma Equalizer

BOSS SP-1 Spectrum

Overview



The Chroma Equalizer project is a clone of the BOSS SP-1 Spectrum, one of the three original BOSS compact stomp boxes released in 1977. (The other two were the OD-1 OverDrive, which is available as my Corona project, and the PH-1 Phaser.) It was discontinued in 1981, and due to its extreme rarity, it routinely sells for $500 or more on eBay.

The Spectrum is a parametric equalizer with only two knobs. The frequency knob sets the boosted frequency between 500Hz and 5 kHz, and the balance (or mix) knob adjusts the dry-to-wet signal ratio. The gain and Q (frequency width) are fixed. It only boosts frequencies—it doesn't have the ability to cut. Its function is very similar to a fixed wah and you will hear a wah-like effect if you turn the Spectrum knob while a signal is being passed.

Controls & Usage

The SP-1's controls are pretty simple:

- **Balance** adjusts the mix between dry signal and equalized signal.
- **Spectrum** sweeps through the frequencies to boost, from 500Hz on one end to 5KHz on the other.

Making a "Balance" pot

The Balance control requires a dual **10kA/10kC pot**, meaning one wafer is audio taper and the other is reverse audio. These pots are not available for purchase from any suppliers, but it is pretty easy to make your own if you don't mind tearing things apart. See page 3 for full instructions, and read through them before undertaking this project!

Modifications & Experimentation

Not much. The original **RC3403 quad op-amp** is obsolete and pretty hard to find (though if you get one, you might as well buy two of them and build an OD-1 clone as well!) and since it's a clean effect, it's unlikely that the characteristics of this particular op-amp contribute anything desirable to this effect in the same way that it might for an overdrive pedal. Fortunately, the RC3403 is pin-for-pin compatible with most modern quad op-amps such as the **TL074** or **LM837** which may produce a higher fidelity sound, so you might try one of those.

Beyond that, I haven't really tried experimenting with this circuit at all, but it seems that it should be possible to modify the character of the equalization, such as changing the upper and lower ends of the range or modifying the width of the EQ. Most of this takes place near **IC1B** in the schematic, so try swapping out some of these capacitors and see what happens. And let me know if you come across anything interesting!

Parts

Resistors

R1	1k
R2	470k
R3	10k
R4	10k
R5	1k
R6	10k
R7	18k
R8	10k
R9	22k
R10	10k
R11	10k
R12	10k
R13	470R
R14	100k
R15	33k
R16	33k
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	47n
C2	4u7 electro
C3	4u7 electro
C4	100n
C5	3n3
C6	3n3
C7	1uF film
C8	4u7 electro
C9	47pF
C10	1uF film
C11	47uF
C12	47uF

Semiconductors

IC1	RC3403A [1]
D1	1N4002
LED	5mm LED

Potentiometers

Balance	10kA/C dual [2]
Spectrum	100kB dual
Gain	1k trim [3] (3362P)

[1] The original unit uses an RC3403A, but the circuit would probably benefit from a higher-fidelity quad op-amp. The **TL074** and **LM837** are drop-in replacements and will probably sound better.

[2] To create a dual 10kA/10kC pot, you will need to buy both a **dual 10kA pot** and a **single 10kA pot**. I'll explain why, and how to make it, on the next page.

[2] According to the service manual, this trimmer should be set as follows: "Test signal: 5KHz +/- 10% sine, -20dBm, set SPECTRUM, BALANCE at max (FCW), adjust trimmer pot 1K for 2.5dBm output at peaking freq. (5KHz +/-10%)" - but for goodness sakes, just adjust it by ear. I couldn't hear much difference at all on the whole sweep of the pot. The overall effect is very subtle. You might be able to make it more pronounced by raising the value of R7. I haven't tested this, though, so I don't know if it would cause any side effects!

Additional Part Notes

- Capacitors are shown in nanofarads (n or nF) where appropriate. 1000n = 1uF. Many online suppliers do not use nanofarads, so you'll often have to look for 0.047uF instead of 47n, 0.0056uF instead of 5n6, etc.
- The PCB layout assumes the use of film capacitors with 5mm lead spacing for all values 1nF through 470nF. I prefer EPCOS box film or Panasonic ECQ-B/V-series.
- Potentiometers are Alpha 16mm right-angle PCB mount.
- Switches are Taiway (Small Bear) or Mountain Switch (Mouser) brand with solder lugs. I prefer the short-toggle variety, but that's just a matter of aesthetics.
- I recommend using these dust covers / insulators from Small Bear to insulate the back of the pots from the board and prevent shorts. If you don't use these, use some electrical tape or cardboard to act as insulation. The right-angle pots will make direct contact with the solder pads otherwise.

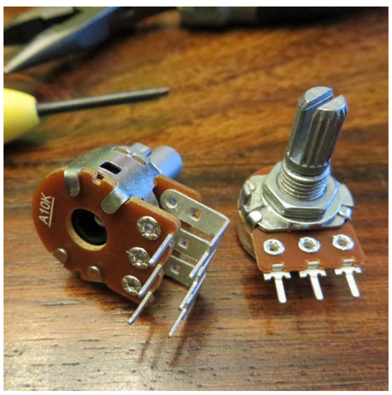
Making a dual 10kA/10kC "Balance" pot

The main reason the SP-1 has never been cloned before is due to the odd 10kA/10kC Balance pot. This pot is unique to the pedal, and to custom-order it from Alpha would mean ordering at least 1,000 minimum. This has historically made the circuit a little prohibitive to DIYers wanting to build their own. Fortunately, dual 10kA pots have recently become available, most notably from Small Bear Electronics in right-angle PCB mount (which this circuit is built to use). It's very easy to switch out the top wafer of these pots, and the top wafer is identical in construction to the wafers used in single pots.

So you would think that by pulling a wafer from a 10kC single pot, you could get what you want, but there's a catch: since the top wafer is upside down, the taper is actually mirrored. The top wafer of a dual 10kA pot is the same as a 10kC. As a result, **you will need to pull a wafer from a single 10kA pot, which will give you the correct "C" taper when used upside-down in a dual pot.** Make sense?

So your shopping list is **one 10kA dual pot** (preferably right-angle PCB mount) and **one 10kA single pot** (preferably straight-pin PCB mount). The other thing I'll mention is that you have to completely destroy the single pot in order to remove the wafer, and there is a possibility that you'll break the wafer in the process if you're not careful. It may be wise to practice destroying a different pot first, or order two 10kA pots to have one as a backup. I've done it a few times and never broken one, but I'd err on the side of caution.

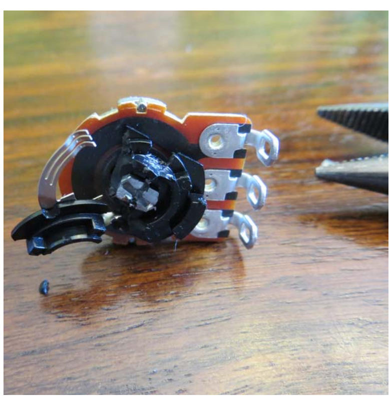
Step by step



1. Start with a 10kA dual pot (right angle PCB mount) and a 10kA single pot (straight-pin PCB mount).

2. On the dual pot, bend out the four tabs with needle-nose pliers to separate the top wafer.

3. On the single pot, bend out the four tabs near the nut to separate the metal cover from the rest.

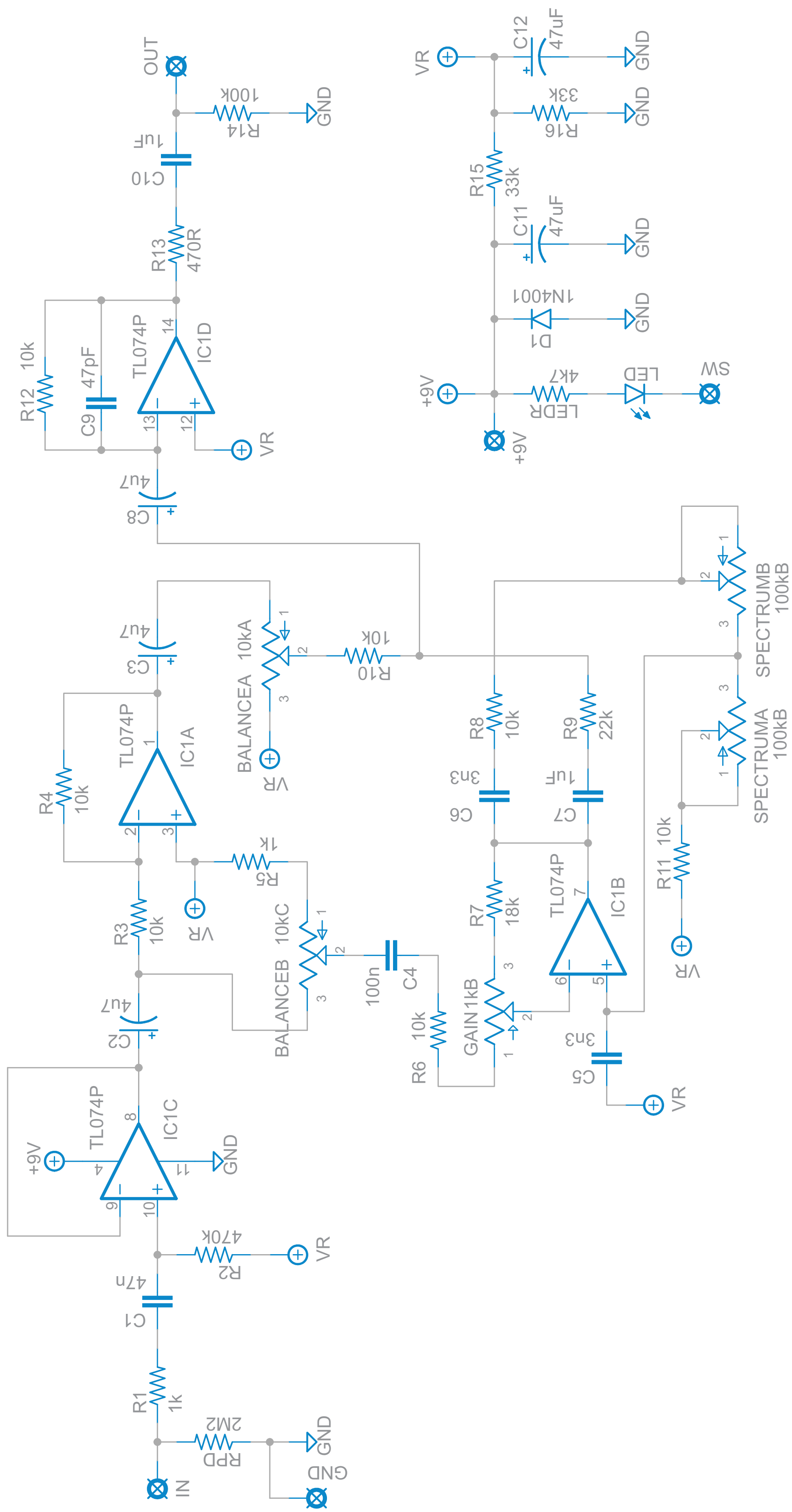


4. On the single pot, break away all the black plastic from the metal shaft, taking care not to scratch the carbon on the wafer itself. (This will take a fair amount of force. You could also use a utility knife or Dremel tool to cut it away.)

5. Once the plastic has been broken away, set aside the wafer and throw the rest away. (Note that this photo shows a solder-lug wafer, which won't fit the PCB without cutting down the lugs. Use a straight-pin pot as shown in step 1.)

6. Place the 10kA wafer on the dual pot as shown, carbon side down, and bend the four tabs inward to secure it in place. Bend the pins straight upward.

Schematic



IN
OUT
RPD
2M2
GND
R1
1k
C1
47n
R2
470K
VR
+9V
TL074P
IC1C
IC1A
IC1B
IC1D
C2
4u7
R3
10k
R4
10k
R5
1K
C3
4u7
BALANCEA
10kA
BALANCEB
10kC
R10
10K
C4
100n
R6
10k
GAIN
1kB
R7
18k
R8
10k
R9
22k
C5
3n3
C6
3n3
C7
1uF
R11
10k
SPECTRUMA
100kB
SPECTRUMB
100kB
C8
4u7
C9
47pF
R12
10k
R13
470R
C10
1uF
R14
100K
C11
47uF
C12
47uF
R15
33k
R16
33K
D1
1N4001
LEDR
4K7
LED
SW

General Build Instructions

These are general guidelines and explanations for all Aion Electronics DIY projects, so be aware that not everything described below may apply to this particular project.

Build Order

When putting together the PCB, it's recommended that you do not yet solder any of the enclosure-mounted control components (pots and switches) to the board. Instead, follow this build order:

1. Attach the **audio jacks**, **DC jack** and **footswitch** to the enclosure.
2. Firmly attach the **pots** and **switches** to the enclosure, taking care that they are aligned and straight.
3. Push the **LED**[1] into the hole in the enclosure with the leads sticking straight up, ensuring that the flat side is oriented according to the silkscreen on the PCB.
4. Fit the **PCB** onto all the control components, including the leads of the LED. If it doesn't fit, or if you need to bend things more than you think you should, double-check the alignment of the pots and switches.
5. Once you feel good about everything, **solder them from the top**[2] as the last step before wiring. This way there is no stress on the solder joints from slight misalignments that do not fit the drilled holes. You can still take it out easily if the build needs to be debugged, but now the PCB is "custom-fit" to that particular enclosure.
6. Wire everything according to the wiring diagram on the last page.

[1] **For the LED**: You can use a bezel if you'd like, but generally it's easier just to drill the proper size of hole and push the LED through so it fits snugly. If you solder it directly to the PCB, it'll stay put even if the hole is slightly too big. Make absolutely sure the LED is oriented correctly (the flat side matches the silk screen) before soldering, as it'll be a pain to fix later! After it's soldered, clip off the excess length of the leads.

[2] **Note on soldering the toggle switch(es):** It will require a good amount of solder to fill the pads. Try to be as quick as possible to avoid melting the lugs, and be prepared to feed a lot of solder as soon as the solder starts to melt. I recommend waiting 20-30 seconds between soldering each lug to give it time to cool down.

"RPD" and "LEDR" resistors

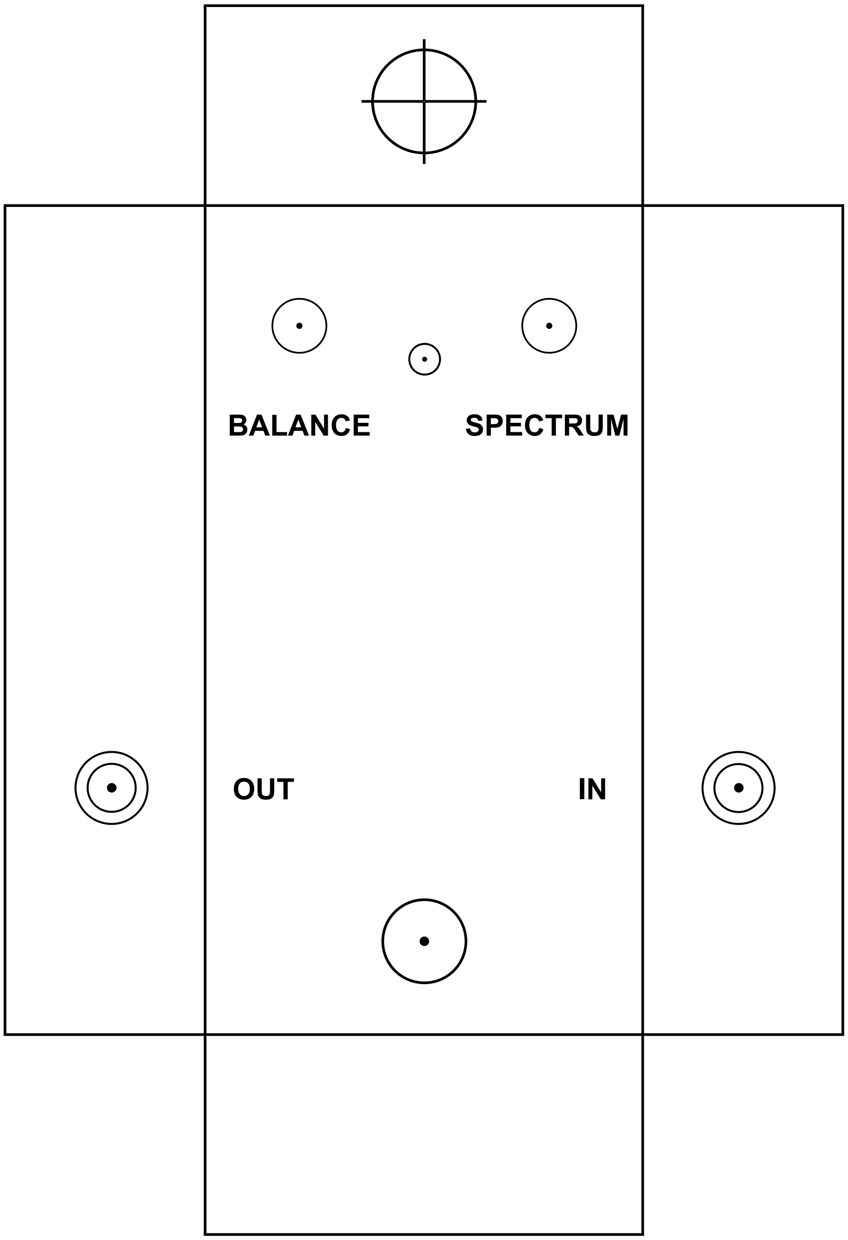
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

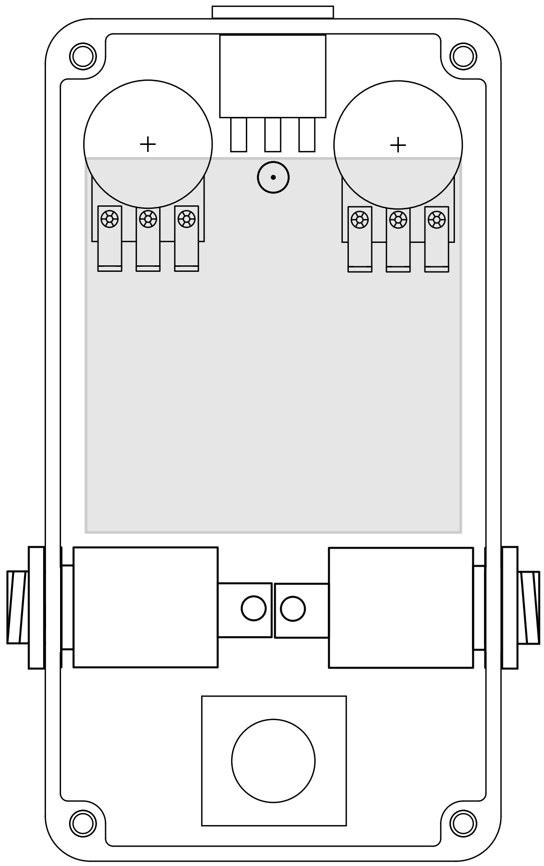
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

Drilling & Placement

Print this page and have an adult cut out the drilling template below for you. Tape it to the enclosure to secure it while drilling. Note that the holes are shown slightly smaller than they need to be, so drill out the holes as shown and then step up until they are the correct size for the components.



Hammond 1590B
(bottom/inside view)



Parts Used

- Switchcraft #111A enclosed jacks
- Kobiconn-style DC jack with internal nut

Standard Wiring Diagram

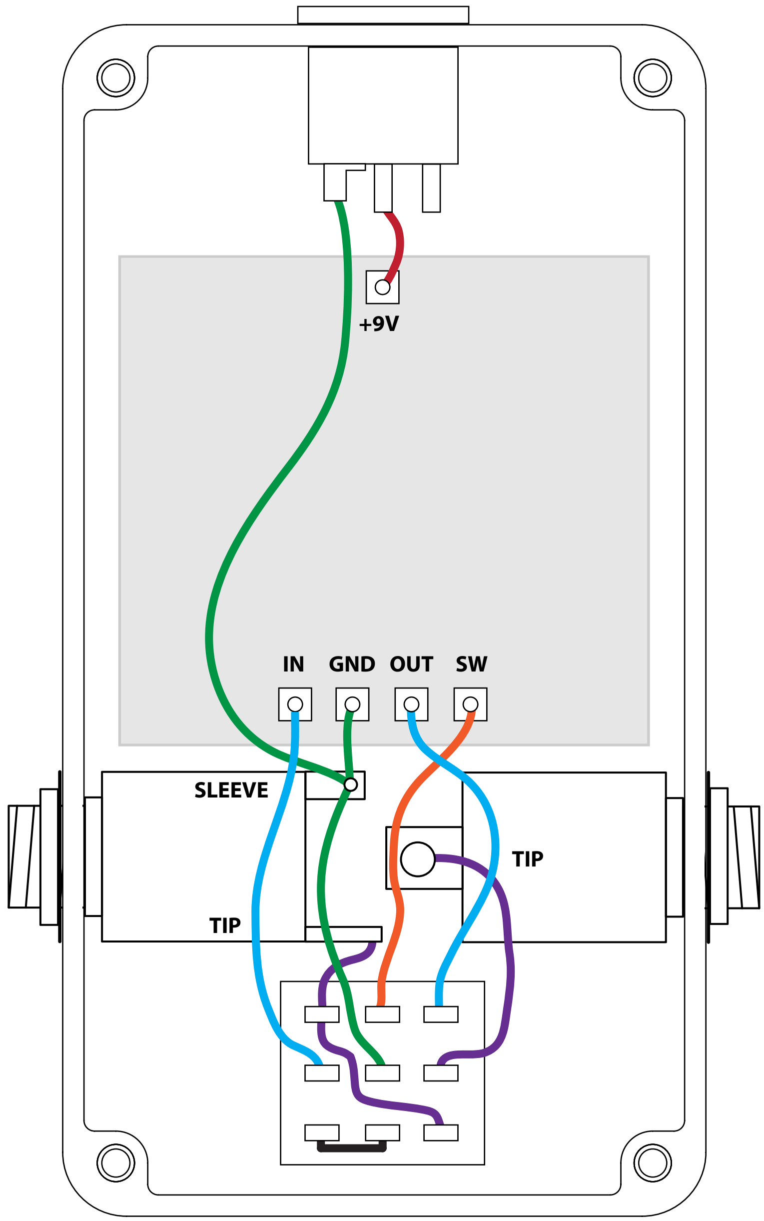
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

The **SW** pad is the cathode connection for the LED. This will connect to ground to turn it on when the switch is on. Usage of the on-board LED connection is not required if you have specific placement needs for your enclosure, but's incredibly convenient.

The wiring diagram also makes use of **star grounding** principles where all of the grounds connect to a single ground point (in this case the sleeve of the input jack). This is best practice to avoid added noise caused by improper grounding. The sleeve of the output jack is unconnected.

If using a painted or powdercoated enclosure, **make sure both jacks have solid contact with bare aluminum** for grounding purposes. You may need to sand off some of the paint or powdercoat on the inside in order to make this happen.

Make sure to double-check the markings of the pads on the PCB for your particular project – they are not always in the order shown here!



License / Usage

No direct support is offered for these PCBs beyond the provided documentation. It is assumed that you have at least some experience building pedals before starting one of these. Replacements and refunds will not be offered unless it can be shown that the circuit or documentation are in error. I have in good faith tested all of these circuits. However, I have not necessarily tested every listed modification or variation. These are offered only as suggestions based on the experience and opinions of others.

Projects may be used for commercial endeavors in any quantity unless specifically noted. No bulk pricing or discounting is offered. No attribution is necessary, though a link back is always greatly appreciated. The only usage restrictions are that **(1) you cannot resell the PCB as part of a kit**, and **(2) you cannot "goop" the circuit, scratch off the screenprint, or otherwise obfuscate the circuit to disguise its source**. (In other words: you don't have to go out of your way to advertise the fact that you use these PCBs, but please don't go out of your way to hide it. The guitar effects pedal industry needs more transparency, not less!)