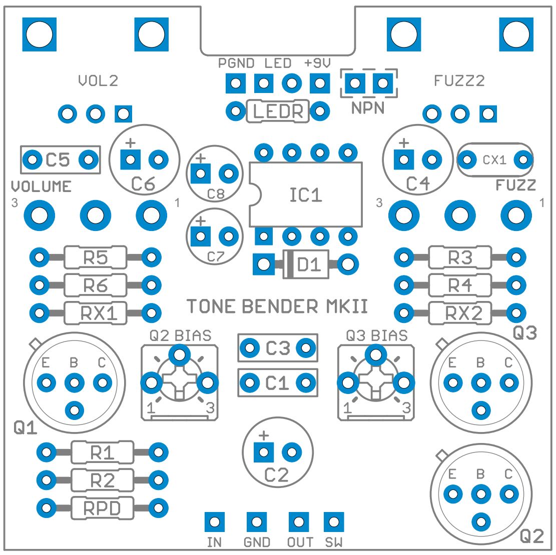
Deimos Fuzz

Tone Bender Professional Mark 2

Overview

Deimos Project Link



The Deimos Fuzz is a clone of the Tone Bender Mark 2 (two-knob). While the first version of the Tonebender was essentially a slightly tweaked Fuzz Face, the Mark 2 version added a single-transistor boost in front of it. But even though the topology is similar to a Fuzz Face, the biasing scheme is very different, and its optimum transistors are not the same as a Fuzz Face set.

The Deimos project is a faithful reproduction of the Tonebender Professional Mk 2 with one big enhancement: a voltage inverter has been added which allows you to power the effect with a standard center-negative adapter while maintaining the positive-ground operation of the original. The PCB also includes space for biasing trim pots so you can dial in a perfect bias without having to worry about swapping out resistors.

Controls & Usage

- **Fuzz** changes the bias of the Q3 transistor, acting as a gain control.
- **Volume** is the output level of the effect.

Modifications

I hunted far and wide for possible Tonebender Mk II modifications to incorporate into this circuit, but there really weren't any that were viable. The typical germanium fuzz mods (input cap blend, treble control, etc.) just don't really have as much of an impact in this circuit, and the general consensus is that it's pretty much perfect as-is and doesn't need tweaking.

So, this is my only PCB project with two knobs. This created a new problem to solve: I don't like the idea of mounting the PCB to the pots along a single axis (the two 16mm pots) since it is not as structurally sound—the PCB can flex along that axis and it could be more prone to broken solder joints as a result of stress. My solution was to use 9mm pots, the kind that mount to the PCB with additional pins for support.

This creates a very sturdy mount, and I would highly recommend using these 9mm pots if you can get them. However, knowing that they may not be as readily available as the more common 16mm pots, I've included pads for the 16mms as well, and they are positioned so that the drill template is the same regardless of which type of pot you use. (Just don't mix them since they mount at different heights.)

The **voltage inverter** is optional, but recommended. Pads have been provided so you can easily bypass it if you'd like to either power it with an isolated (non-daisy-chained) power supply, or use NPN transistors for a standard negative-ground circuit.

Parts

Capacitors

C1	10n
C2	4μ7 electro
C3	100n film
C4	4μ7 electro
C5	10n film
C6	47μF electro
C7	10μF electro [1]
C8	10μF electro [1]
CX1	10pF MLCC [2]

Resistors

R1	10k [3]
R2	10k
R3	47k
R4	100k
R5	470R
R6	1k
RX1	(omit) [4]
RX2	(omit) [4]
RPD	2M2
LEDR	4k7

Semiconductors

Q1–Q3	PNP Ge [5]
IC1	TC1044S [1]
D1	1N4742 [1]
LED	5mm LED

Potentiometers

Fuzz	1kC 9mm [6, 7]
Volume	100kA 9mm [7]
Q2BIAS	50k trim (3362P) [8]
Q3BIAS	10k trim (3362P) [8]

Build Notes

[1] **Part of the voltage inverter circuit.** Omit these and use a standard 1N4001 for D1 if you want to either (1) use NPN transistors, or (2) power the circuit with an isolated supply and reverse the DC jack wiring. Both of these are outside the scope of this documentation—in other words: if you don't know how, don't try!—but in either case you will want to jumper the pads marked **NPN** up near the top of the PCB.

[2] **Optional:** This is a common modification not found in the original circuit. It helps smooth out the fuzz a little bit and most people prefer it. At the very least, socket it and see if you notice a difference.

[3] Some schematics list this as **100k**. Original units had both values—it depends on the transistor being used for Q1, particularly the leakage. The original OC81D transistor worked best with a 100k resistor, but these are almost impossible to come by, and most other transistors work better with a 10k in this position.

[4] **Leakage simulation resistors:** These resistors can be used to simulate extra leakage for optimal performance if your transistors are not leaky enough. A good starting point is **1.5M** and adjust it up to 2M or down to 1M as needed. (If you don't know what these are for then don't bother with them!)

[3] **Germanium transistors:** For this circuit, as with other vintage fuzzes, it's not so much the part number of the germanium transistor as it is the properties (gain and leakage). See next page for more information.

[6] **Taper:** The original uses a linear (B) taper for the Fuzz control. However, a reverse-audio (C) taper will offer much more usable control over the amount of fuzz so use it if you can. (The range is the same either way.)

[7] **9mm or 16mm**. See page 1 for notes on why 9mm is preferred.

[8] **Bias trimmers:** These two trimmers allow for fine biasing of the transistors to get the correct voltages without having to swap out resistors. See next page for more information.

Additional Part Notes

- Capacitors are shown in nanofarads (n or nF) where appropriate. 1000n = 1μF. Many online suppliers do not use nanofarads, so you'll often have to look for 0.047μF instead of 47n, 0.0056μF 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 brand, either 9mm PCB mount (preferred) or 16mm right-angle PCB mount.
- If using 16mm pots, 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 pots will make direct contact with the solder pads otherwise.

Transistor Selection & Biasing

A person could write a book, or at least a pretty lengthy essay, about germanium transistors and their use in vintage fuzz circuits. I don't have the level of expertise to do this, so what follows is far from the full story, but it's what you need to know to get a great-sounding fuzz that's as close to the original units as possible.

Sourcing

First off: **you can save yourself a lot of time by buying a matched set of TB Pro Mk II transistors.** They aren't terribly cheap, but it's a bargain compared to buying a bunch of old-stock transistors and testing and sorting them yourself. I very strongly recommend going that route if you can. Here are a few great sources:

Small Bear Electronics (USA) - International shipping can be a little pricey, but you know you're getting a set that was matched by people who know what to look for. Look for sets marked "TB Pro Mk II".

LIC Pedals (USA) - This eBay seller has great prices on a number of different types of sets.

Musikding (Germany) - For European DIYers, Musikding has a great selection of NOS transistors. They don't sell sets, though—you have to buy them individually based on desired gain range. (See next section.)

Characteristics

What follows is some very general information about what makes an ideal Tone Bender Mk II transistor, since some online stores only sell transistors individually by gain range. These are only the ideal characteristics—it'll be fine if yours are a few HFe outside the range. That's what the bias trimmers are for.

Q1: 60-70 HFe, high leakage
Q2: 60-70 HFe, low to medium leakage
Q3: 100-120HFe, medium leakage

Leakage (measured in microamps or μA) is important. Unlike most other transistor-based pedals, in this case some leakage is required for it to function properly. You're looking for somewhere between 100μA and 400μA of leakage—any less than that and you might not get any signal past Q1. (Note that **RX1** and **RX2** have been provided to simulate leakage on Q1 and Q3 if your transistors measure lower than the optimal range.)

Biasing

Part of the characteristic sound of the Tone Bender Mk II is the **uneven bias**. Many people assume that -4.5v is the target bias point, as in a Fuzz Face, but vintage TB units actually have biases at around **-8.5V** on the collector. It seems strange, but if it sounds good, I don't question it!

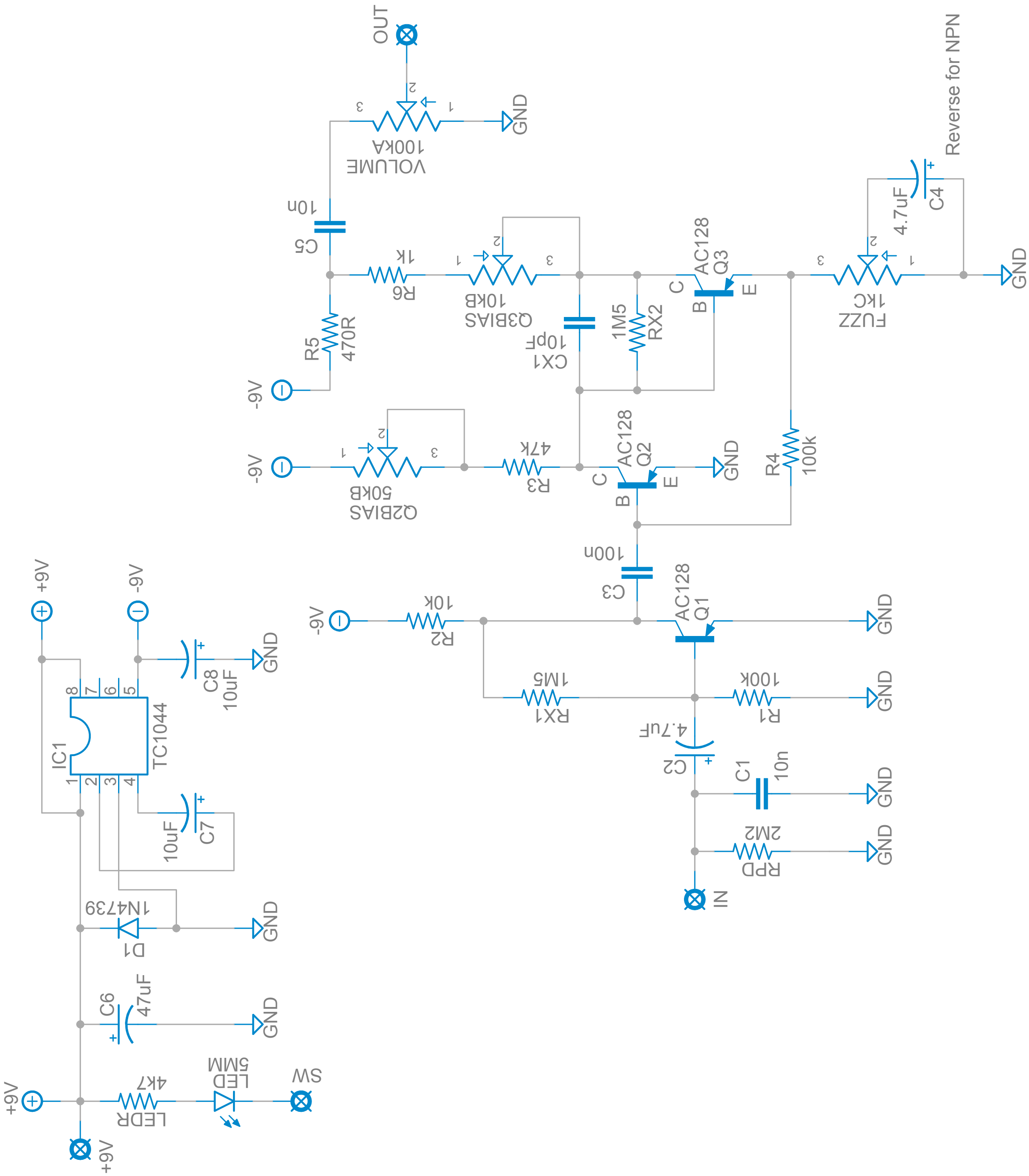
The Deimos Fuzz project is set up to allow for **easy biasing of the Q2 and Q3 transistors via trim pots** without having to swap out resistors. One thing to keep in mind is that the Q2 bias trimmer actually adjusts the bias of Q3 as well, while the Q3 pot only adjusts Q3. So, **always bias Q2 first** and then only use Q3's bias adjustment if you need to. The Q2 trimmer is going to be far more useful.

As a starting point, set the Q3 bias trimmer to around 75% rotation (approximately **8.2k total resistance** when combined with R6). Turn the Q2 bias trimmer all the way down. Then, with a multimeter, touch the black (common) lead to ground and touch the red lead to the far-right leg of Q2. You're looking for something near these voltages.

Q1: Collector -8.5v, Base -0.05v, Emitter 0v
Q2: Collector -0.15v, Base -0.07v, Emitter 0v
Q3: Collector -8.5v, Base -0.15v, Emitter -0.1v

Since Q2's voltages are so low, it's best to **test Q3's collector as you adjust the Q2 trimmer** and stop when Q3 is where you want it. The Q3 bias trimmer should only be used if Q2's voltage is way off once this is done.

Schematic


OUT
VOLUME
100KA
GND
10n
C5
1K
R6
Q3BIAS
10KB
AC128
Q3
C
B
E
Reverse for NPN
4.7uF
C4
FUZZ
1KC
R5
470R
-9V
CX1
10pF
1M5
RX2
Q2BIAS
50KB
47K
R3
AC128
Q2
R4
100k
100n
C3
10K
R2
AC128
Q1
1M5
RX1
100K
R1
4.7uF
C2
C1
10n
2M2
RPD
IN
+9V
IC1
TC1044
C8
10uF
10uF
C7
1N4739
D1
C6
47uF
4K7
LEDR
LED
5MM
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

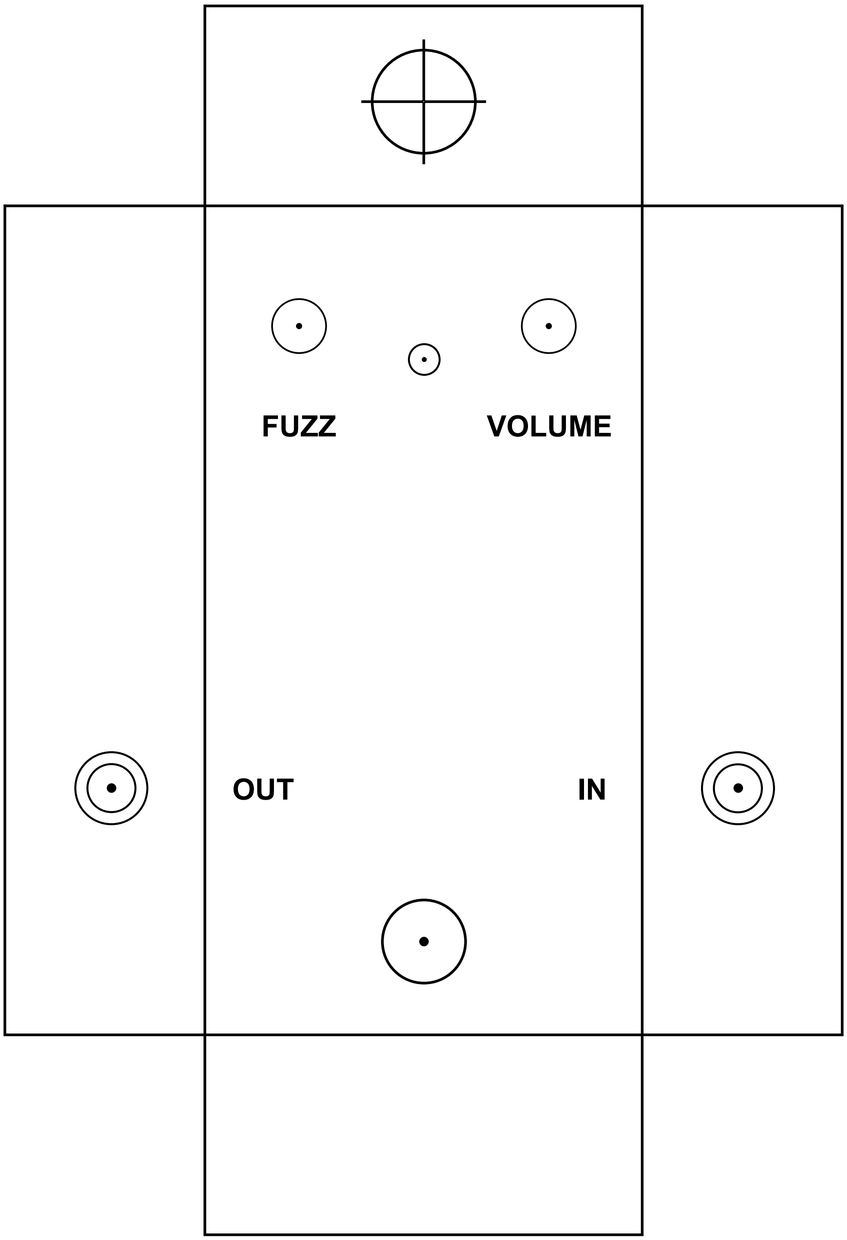
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

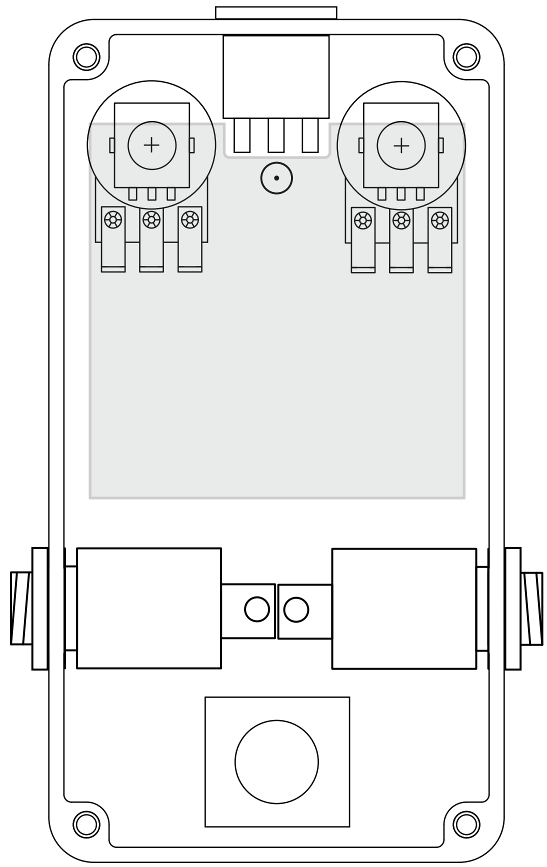
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

Drilling & Placement

Print this page and cut out the drilling template below. Fold the flaps down the sides of the enclosure and use tape 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 111X enclosed jacks
- Kobiconn-style DC jack with internal nut

Standard Wiring Diagram

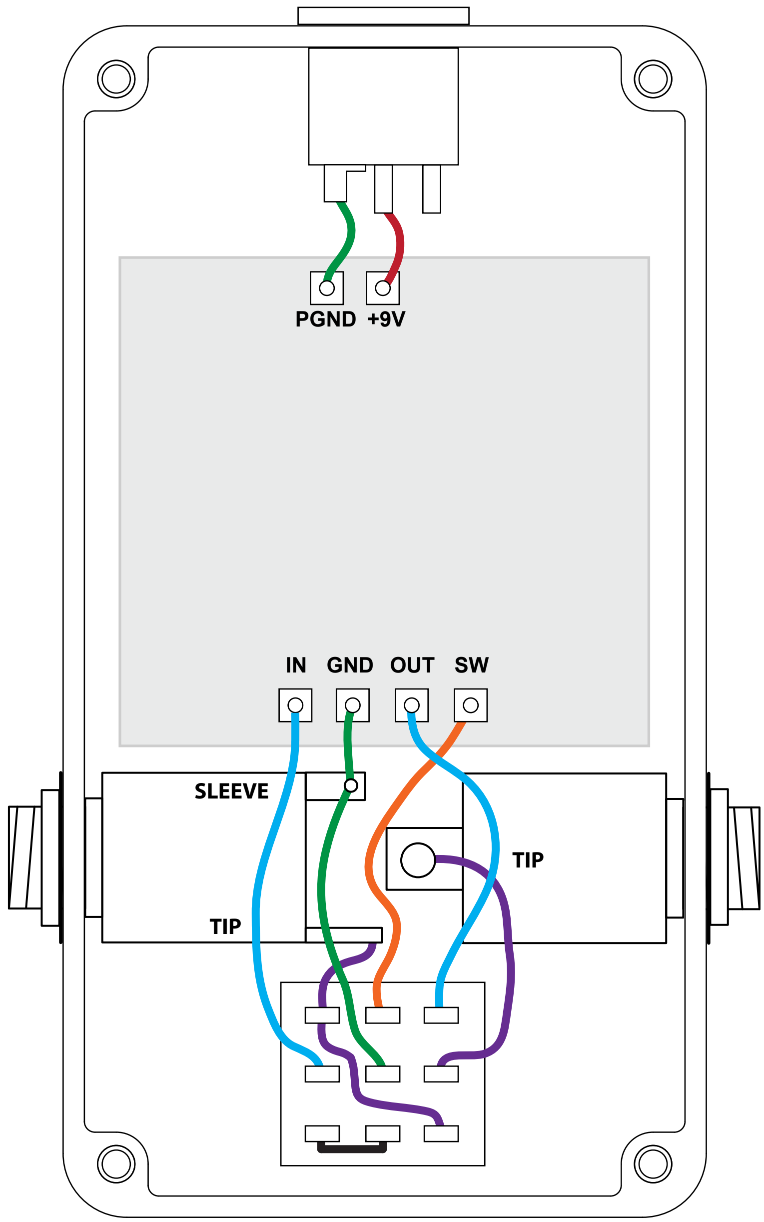
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)