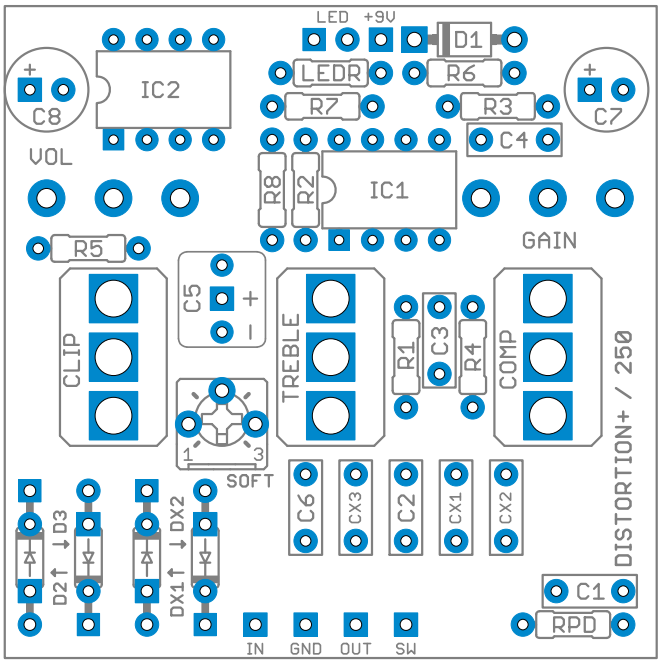
Aphelion Distortion

MXR Distortion+ / DOD 250 Overdrive

Overview

Distortion+ PCB project page



The Aphelion Distortion project is a clone of the MXR Distortion+ and DOD 250+ Overdrive, which are essentially the same circuit with a few different values. The Distortion+ first came out in 1973 and the DOD 250 followed in 1976. Other later variants include the MXR MicroAmp and the DOD YJM308.

This was one of the first circuits to use hard diode-to-ground clipping, and while it's a fairly simple circuit, it serves as an ancestor to other pedals like the Boss DS-1 and Maxon/Ibanez SD-9. It features no tone control; rather, the gain control also changes the tone as you turn it up. It's capable of a wide range of tones, from slight breakup to gritty distortion.

This project will allow you to build any historical variant of the Dist+/250. I included three switch mods in this project as well, but you can always leave them off and build a stock circuit.

Controls & Usage

The Distortion+ controls are pretty basic:

- **Gain** controls the amount of gain from the op amp that is fed into the diode clipping stage. This control works differently than a standard op-amp gain control and actually cuts bass from the signal as you turn up the knob, so in some ways it acts as a tone control as well.
- **Volume** controls the overall output.

Modifications & Experimentation

The **Clipping** switch mod allows you to set up a second set of diodes to toggle back and forth from stock. Extra pads have been provided so you can stack two diodes in a row if desired. (The middle two pads are connected in each diode.) The second position of the switch has a **Softness** trimmer as well, which you can experiment with if you want to try for some different textures, or turn it all the way down to remove it from the circuit. If you use a SPDT center-off switch, the middle position becomes a diode lift mode.

The **Comp** switch changes the value of the input capacitor, which in addition to affecting the bass can also have a slight compression effect on the circuit.

The **Treble** switch changes the amount of treble that is cut at the end of the circuit. One side of the switch is stock, the other side is either more or less treble (depending on how you want to set it up), and if you use a center-off switch, the middle position removes the treble-cut capacitor altogether.

IC2 is for experimentation with dual op-amps. Do not use both IC1 and IC2 at the same time!

Parts (Distortion+ / ‘79 Gray 250)

Resistors

R1	10k / 10k
R2	1M / 470k
R3	1M / 1M
R4	4k7 / 4k7
R5	10k / 10k
R6	Jumper / 100R
R7	1M / 22k
R8	1M / 22k
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	omit / 1n
C2	10n / 10n
C3	47n
C4	omit / omit
C5	4u7 / 1u tantalum
C6	1n / 1n
C7	omit / omit [2]
C8	10u / 1u [2]
CX1	10n
CX2	47n
CX3	2n2

Semiconductors

IC1	LM741 [1]
IC2	JRC4558 [1]
D1	1N4002
D2, D3	1N270 / 1N914
DX1, DX2	3mm LED [3]
LED	5mm LED

Potentiometers

Softness	10k trim (3362P)
Distortion	500kC [4]
Volume	100kA

Other

CLIP	SPDT center off [5]
COMP	SPDT center off
TREBLE	SPDT center off [5]

[1] **Use only one of these.** The original units use an LM741, which is a single op amp (compatible with the TL071 and most others singles). For experimentation purposes, a pad layout for a dual op amp is provided as IC2 so you can use half of a more modern dual op amp such as a JRC4558 or OPA2604.

[2] **Filtering:** Vintage stompboxes did not tend to have much power filtering since they ran primarily off of batteries, so the now-common practice of a large filter capacitor on V+ and a smaller filter capacitor on Vref was not common back then. No vintage units of either the Distortion+ or 250 utilize C7, but it’s recommended to use a 47uF capacitor here. For C8, some use 1u and some use 10u, but it’s recommended to use 10u for all versions.

[3] **Clipping diodes:** You can use whatever diodes you’d like for the second set, but it’s recommend to use some with high clipping thresholds, such as **LEDs or two 1N914s in series** on each side. This will increase the output volume too.

[4] Many Distortion+ schematics show a **1MC** pot for the Distortion control. The two vintage units I’ve seen (mid-70’s and 1980) had a 500kC pot, so I’m not sure whether the 1M value was used for any production units.

[5] If you choose to exclude either of these switches, you will have to **jumper pads 2 & 3 together** (the bottom two pads) for normal operation. The Comp switch does not need to be jumpered if it’s left off.

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.

Variations

This circuit has had many different variations during its 40-year history. The following table owes a debt of gratitude to research done by Steve Mavronis on the history of the DOD 250, but includes my own additions for MXR's versions.

Part	'77 Gray 250	'78 Gray 250	'79 Gray 250	'82 Yellow 250	'97 Yellow 250	'02 YJM 308	Distortion+	Micro Amp
IC	LM741	LM741	LM741	LF351N	KA4558 [1]	KA4558 [1]	LM741	LM741
R1	10k	10k	10k	10k	10k	10k	10k	10k
R2	510k	470k	470k	470k	470k	470k	1M	10M
R3	1M	1M	1M	1M	1M	1M	1M	56k
R4	4k7	4k7	4k7	4k7	4k7	4k7	4k7	2k7
R5	10k	10k	10k	10k	100k	10k	10k	470R
R6	jumper [2]	jumper [2]	100R	100R	100R	100R	jumper [2]	jumper [2]
R7	20k	22k	22k	22k	22k	22k	1M	100k
R8	20k	22k	22k	22k	22k	22k	1M	100k
C1	2n2	(omit)	(omit)	(omit)	(omit)	(omit)	1n	(omit)
C2	10n	10n	10n	10n	10n	10n	10n	100n
C3	47n	47n	47n	47n	56n	47n	47n	4u7
C4	(omit)	(omit)	(omit)	25pF	25pF	25pF	(omit)	47pF
C5	4.7uF	4.7uF	4.7uF	4.7uF	4.7uF	4.7uF	1uF	15uF
C6	1n	1n	1n	1n	1n	1n	1n	10k resistor [3]
C7	(omit) [4]	(omit) [4]	(omit) [4]	(omit) [4]	(omit) [4]	(omit) [4]	(omit) [4]	(omit) [4]
C8	(omit) [4]	10uF	10uF	10uF	10uF	10uF	1uF	1uF
D2/D3	1N4148	1N4148	1N4148	1N4148	1N4148	1N4148	1N270 (Ge)	(omit)
Gain	500kC	500kC	500kC	500kC	500kC	500kC	500kC	500kC
Volume	100kA	100kA	100kA	100kA	100kA	100kA	50kA	(omit) [3]

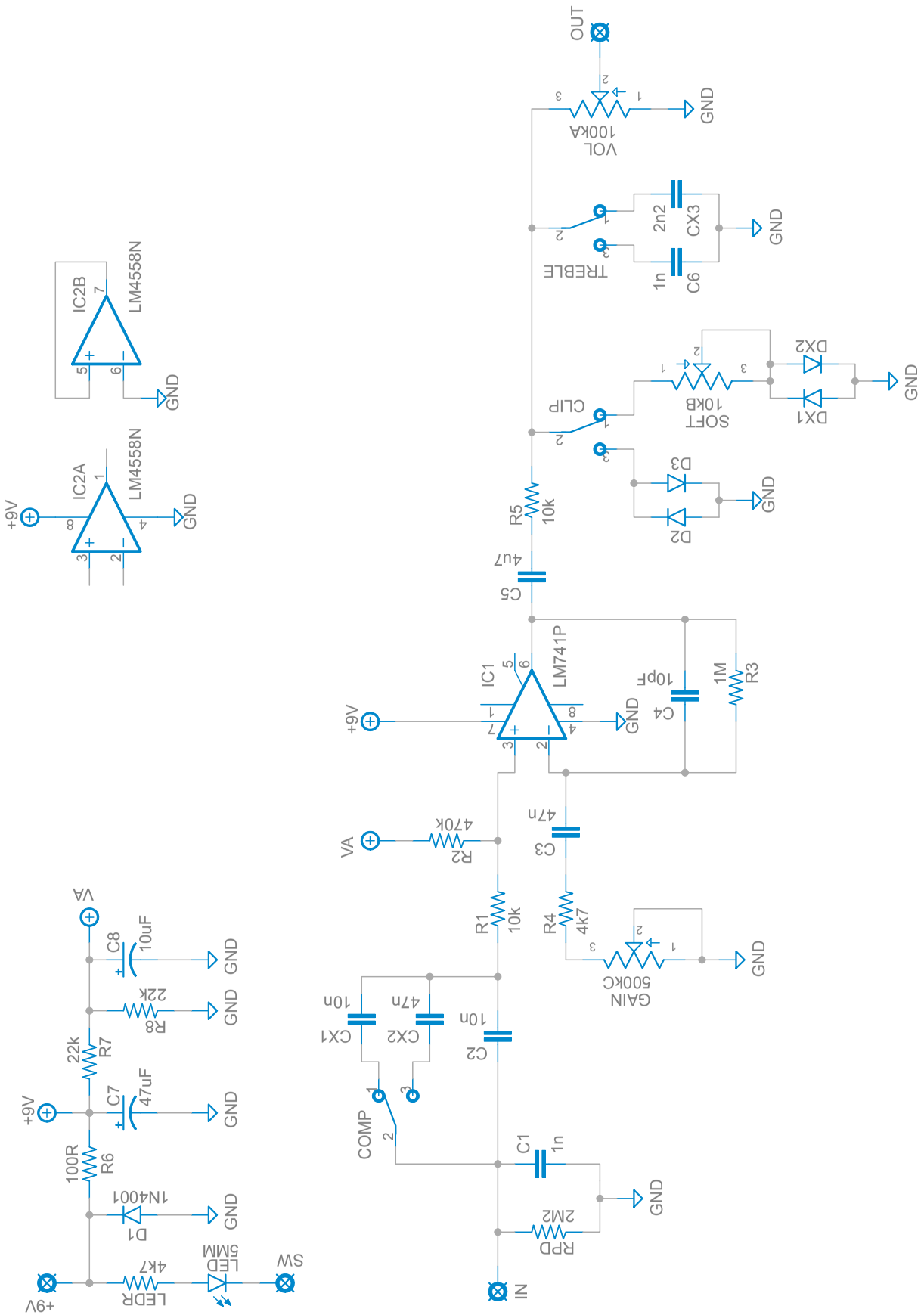
[1] Since the 4558 is a dual op-amp, you'll need to use IC2 for this instead of IC1. Any 4558 should be fine—there's nothing special about the "KA".

[2] This 100R resistor has no effect on the tone and may help reduce noise, so there's really no reason not to include it regardless of the version.

[3] The Micro Amp leaves off the output volume control. You might as well include it, though, since it can allow for some additional tone-shaping. If using the output volume control, you can leave off the 10k resistor in the C6 position.

[4] None of the variants have filtering directly on the power supply, but since this is a common convention I've allowed a space for it. I recommend putting a 47uF capacitor in C7 and a 10uF capacitor in C8 regardless of the variant.

Schematic (Distortion+ values)



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

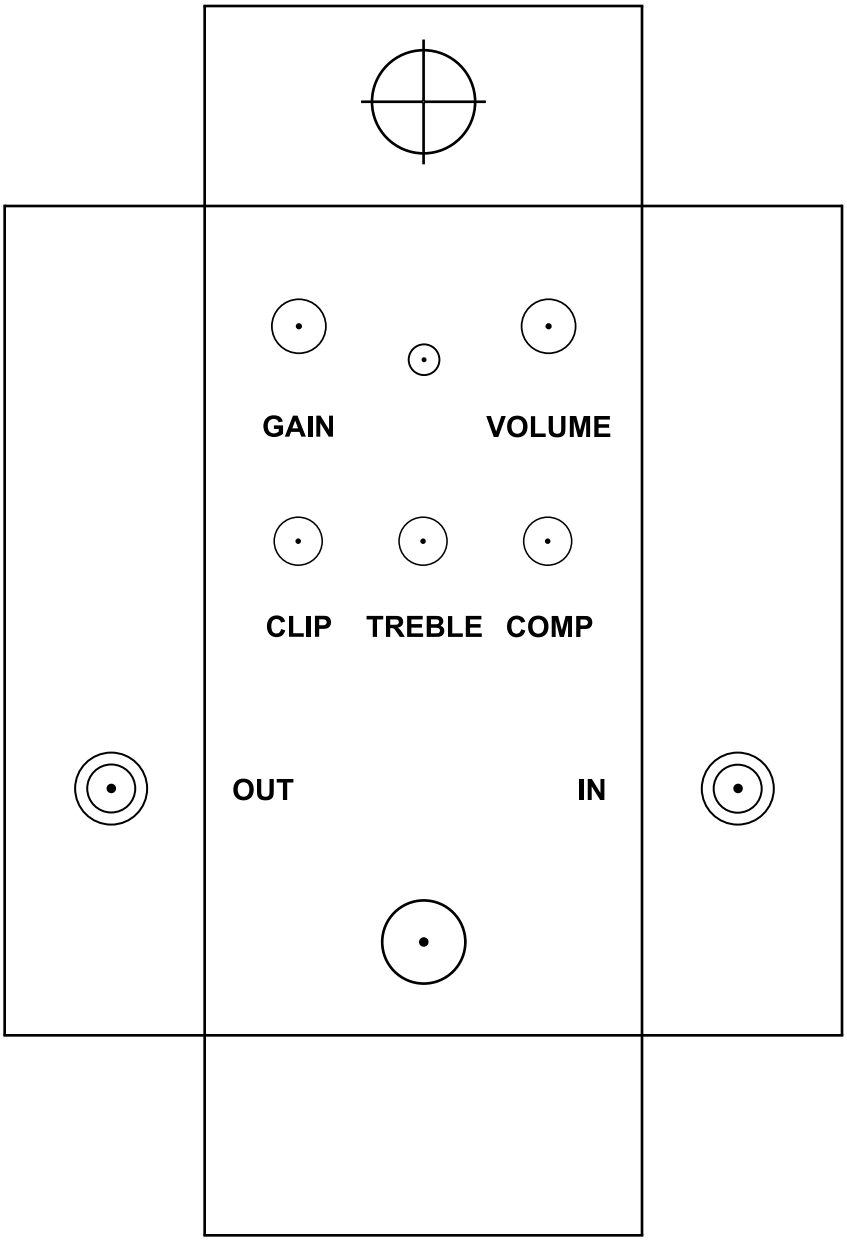
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

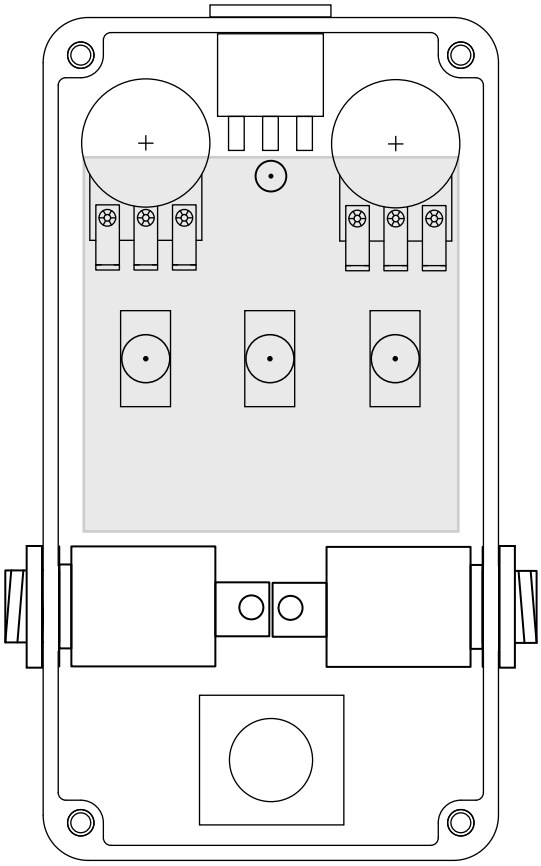
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

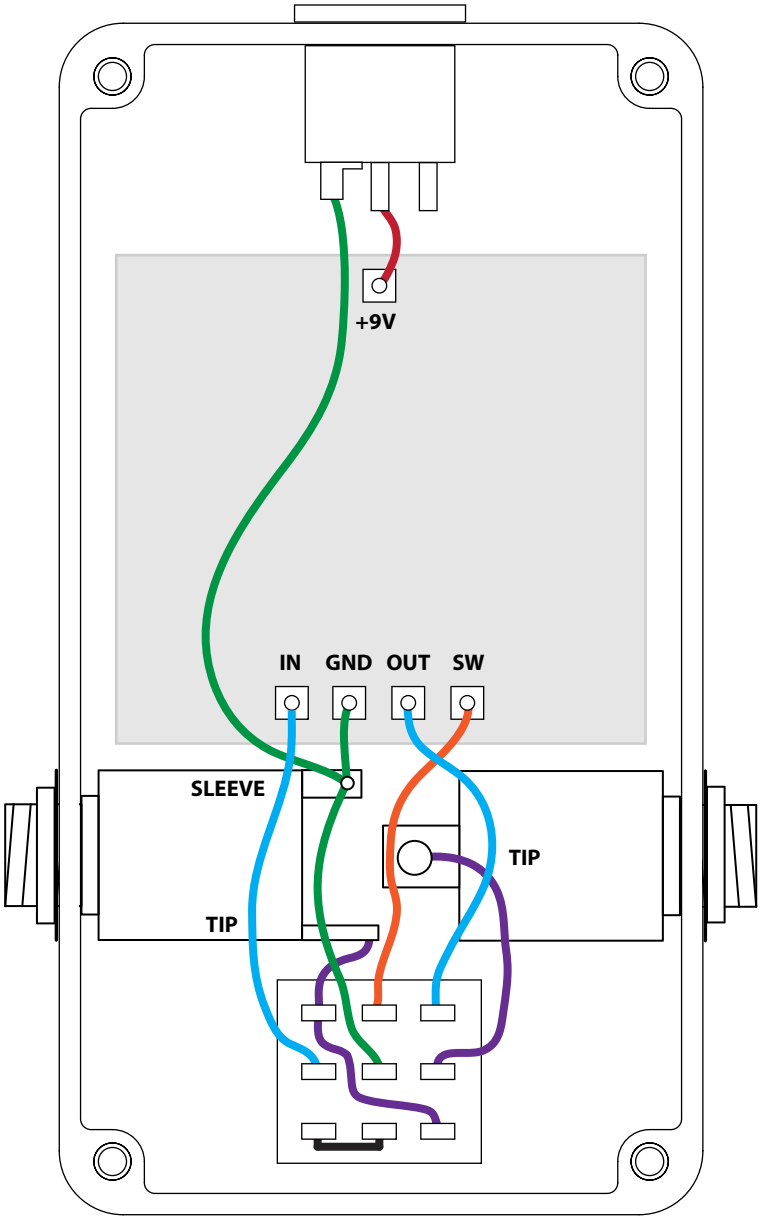
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)