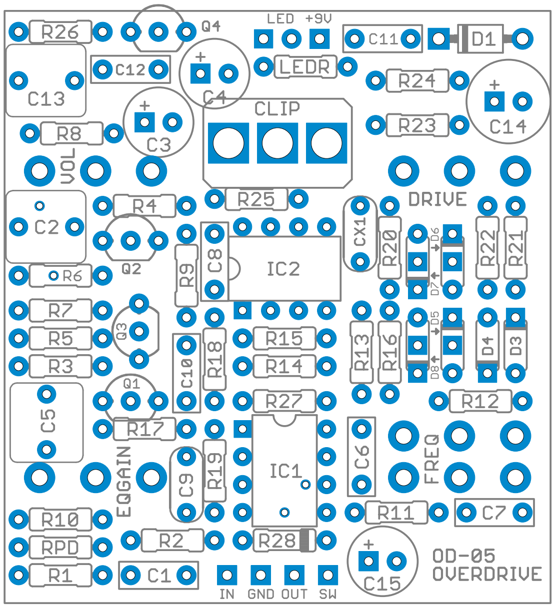
Fractal Overdrive

Pearl OD-05 Overdrive

aion electronics

Overview



The Fractal Overdrive is a clone of the Pearl OD-05 Overdrive, a strangely obscure effect from the early 80's when Pearl (yep, the drum company) briefly dipped into the effects pedal business. The result was a line of very well-engineered pedals that unfortunately never got the attention they deserved.

The interesting thing is, even 30 years later this pedal still is relatively unknown. There are no commercial clones of this pedal and it's relatively unexplored in the DIY scene (with the exception of BYOC's Parametric Overdrive project). But those who have played through one universally praise it for its tonal flexibility.

The OD-05 has a standard feedback-diode clipping stage like a Tube Screamer, but the magic is what comes in front of it: a parametric equalizer that allows either a cut or boost of up to 15dB in a range from 100 Hz to 4 KHz. With this flexible pre-clipping EQ, you can get some incredible sounds out of it. You can crank the mids for a cocked-wah sound, or tune the circuit to enhance the resonant peak of your specific guitar and amp. You can even get a scooped-mids sound if that's what you're shooting for, something that's pretty rare in an overdrive.

The circuit has a lot of familiar things in common with others you've seen, but in this implementation they don't always behave how you expect. This is not quite a Tube Screamer where you can tweak just about any part and get something good out of it! The original designers balanced the stock circuit very well, but if you're modifying it, you may find that if you change one thing on one end you may need to change something clear over on the other end to make the first thing sound good.

Specific modification ideas can be found on page 3. If you're the type to socket every component on a PCB and experiment by swapping parts in and out, this circuit will keep you busy for a long time and you will undoubtedly be rewarded with something you love. If you do end up with something interesting, make sure to send in a build report!

Controls & Usage

- **Drive** controls the amount of gain from the op amp that is fed through the feedback clipping diodes.
- **Level** is the output level of the effect.
- **Frequency** sweeps through the midrange from 100Hz to 4kHz, setting the frequency that is either boosted or cut by the EQ Gain control.
- **EQ Gain** is the boost or cut of the frequency set with the Frequency knob. It provides up to +/-15dB gain. At the center position, it's a flat response and the Frequency knob has no effect.
- **Clip** is a clipping diode switch to allow you to use different combinations of diodes.

Parts

Resistors

R1	1k
R2	1M
R3	10k
R4	680k
R5	270k
R6	22k
R7	22k
R8	22k
R9	470R
R10	3k3
R11	2k2
R12	2k2
R13	22k
R14	22k
R15	22k
R16	22k
R17	10k
R18	10k
R19	10k
R20	1M
R21	1k [1]
R22	1k [1]
R23	10k
R24	1k

Resistors (cont.)

R25	10k
R26	10k
R27	10k [2]
R28	10k [2]
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	33n
C2	1uF film
C3	10uF electro
C4	47uF electro [3]
C5	1uF film
C6	4n7
C7	68n
C8	100n
C9	470pF MLCC
C10	100n
C11	220n
C12	15n
C13	1uF film
C14	100uF electro
C15	47uF electro
CX1	47pF MLCC [4]

Semiconductors

Q1–Q4	2N5088
IC1–IC2	JRC4558D
D1	1N4002
D3, D4	1N914 [1]
D5–8	1N914 [1]
LED	5mm LED

Potentiometers

Drive	100kB
Volume	10kB
EQ Gain	10kB [5]
Frequency	100kC dual [6]

Other

Clip	SPDT [1]

Build Notes

[1] The stock clipping diode setup is nonstandard—it uses two 1N914s, each in series with a 1k resistor (R21 and R22). This will give slightly softer clipping than just the diodes. **D5–D8** are there so you can experiment with alternate diode arrangements. This circuit is sensitive to high clipping thresholds, so you probably won't want to go much higher than 1.2V (two 1N914s in series)—LEDs are not recommended. For the same reasons, I would not recommend using a center-off switch for the diode selector.

[2] In the original unit, **R27** was a **3k3** resistor and **R28** was actually a **5.1v zener** diode. This is because the whole line of pedals shared the same schematic block for the power supply, and it was developed for the phaser, flanger and delay which needed a fixed bias point that was independent of the battery's drifting voltage. However, for an overdrive that needs higher headroom, this is a very bad idea because it means that the reference voltage is unbalanced and half of the waveform can hit its rail more quickly than the other half. Accordingly, this circuit has been updated to use a more standard voltage divider. There's a stripe on the silkscreen of **R28** indicating the original diode's orientation, though, if you want it to be 100% accurate.

[3] The original uses **100uF** here, which seems like overkill to me. **47uF** should be more than adequate.

[4] This is not present in the original, but it's in nearly every other feedback diode clipping circuit (such as the Tubescreamer). I'd recommend including it since its function is to tame harshness as the gain is increased.

Build Notes (cont.)

[5] Since the **EQ Gain** knob boosts and cuts, with the center position being zero, I prefer using a **center-detent pot** here, and I've linked to one from Small Bear. This is purely cosmetic, and the downside is that they are only available in solder-lug and so you have to run the wires to the pads. **Any 10kB pot** will work fine here.

[6] The **100kC dual pot** is pretty easy to find since it's the same value used for the classic Uni-Vibe's speed control. However, Build Your Own Clone is the only place that sells it in PCB-mount—I've linked to them above. I put in a request with Small Bear Electronics to stock the right-angle version, but it probably won't be available until mid-2015. The solder-lug version will work just fine for now, it's just a lot of extra wiring.

Modifications & Experimentation

Input capacitor: The stock circuit is very well-balanced, but if you're doing any experimentation with the clipping section then you may want to reduce the input cap value to tighten up the bass a little. Try **22n** for **C1**.

Q (frequency width) adjustment: C6 and C7 set the **Q** (width) of the parametric frequency band. By changing the values of these capacitors in relation to each other, you can create either a narrower or a wider Q than the stock circuit. For narrower Q, try **C6** = **2n2** and **C7** = **33n**. For wider Q, try **C6** = **6n8** and **C7** = **100n**.

Gain/EQ adjustment: By changing **C11** and **R24**, you can adjust the overall gain of the circuit as well as the EQ. In the stock circuit, these values give the same corner frequency roll-off as a Tube Screamer (723 Hz). You can increase the gain by reducing the value of **R24**—just increase the value of **C11** by the same proportion to keep the same EQ. If you use more common Tube Screamer values, you may need a different drive pot value to compensate: **C11** = **47n**, **R24** = **4k7**, and a **500kA** drive pot.

Brightness: R25 and **C12** form a low-pass filter with a corner frequency of 1061 Hz. You can raise **C12** to **18n** or **22n** to dampen the high-end, or you could lower it to **12n** or **10n** to increase the brightness.

Minimum gain: Drop **R23** down to **4k7** or even **1k** to reduce the minimum gain of the circuit. I'd recommend this even if you're going for an otherwise-stock build.

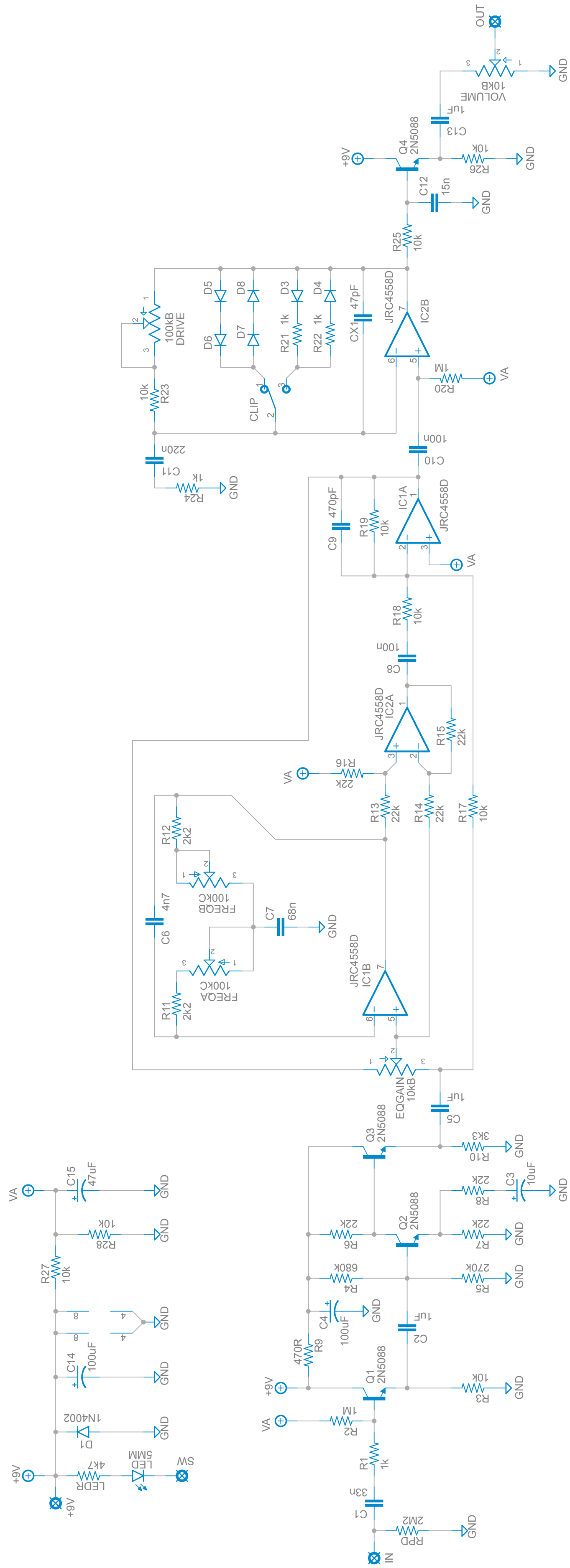
Volume: The stock circuit uses a **10kB** volume pot. You'll get a better taper adjustment by switching to a more common **100kA** pot as seen in most Tube Screamer variants.

Op amp experimentation: I designed the op-amp layout so that one primarily controls the parametric EQ section and one primarily handles the feedback clipping. This way, if you want to try a higher-fidelity op amp for the EQ, you can still get the 4558 "mojo" in the clipping section. Try a **TL072** or an **OPA2134** for **IC1**.

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.

Schematic



OUT
VOLUME
10kB
C13
1uF
Q4
2N5088
R26
10K
GND
+9V
C12
15n
R25
10k
JRC4558D
IC2B
R20
1M
VA
CX1
47pF
D3
D4
D5
D6
D7
D8
R21
1k
R22
1k
100kB
DRIVE
R23
10k
CLIP
C11
220n
R24
1K
C10
100n
IC1A
C9
470pF
R19
10k
R18
10k
C8
100n
IC2A
R15
22k
R16
22K
R13
22k
R14
22k
R17
10k
R12
2k2
FREQB
100KC
C6
4n7
C7
68n
FREQA
100KC
R11
2k2
IC1B
EQGAIN
10kB
C5
1uF
Q3
2N5088
R10
3K3
C3
10uF
R8
22K
Q2
2N5088
R6
22K
R7
22K
R4
680K
R5
270K
C4
100uF
C2
1uF
R9
470R
Q1
2N5088
R3
10K
R2
1M
R1
1k
C1
33n
RPD
2M2
IN
C15
47uF
R28
10K
R27
10k
C14
100uF
D1
1N4002
LEDR
4K7
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

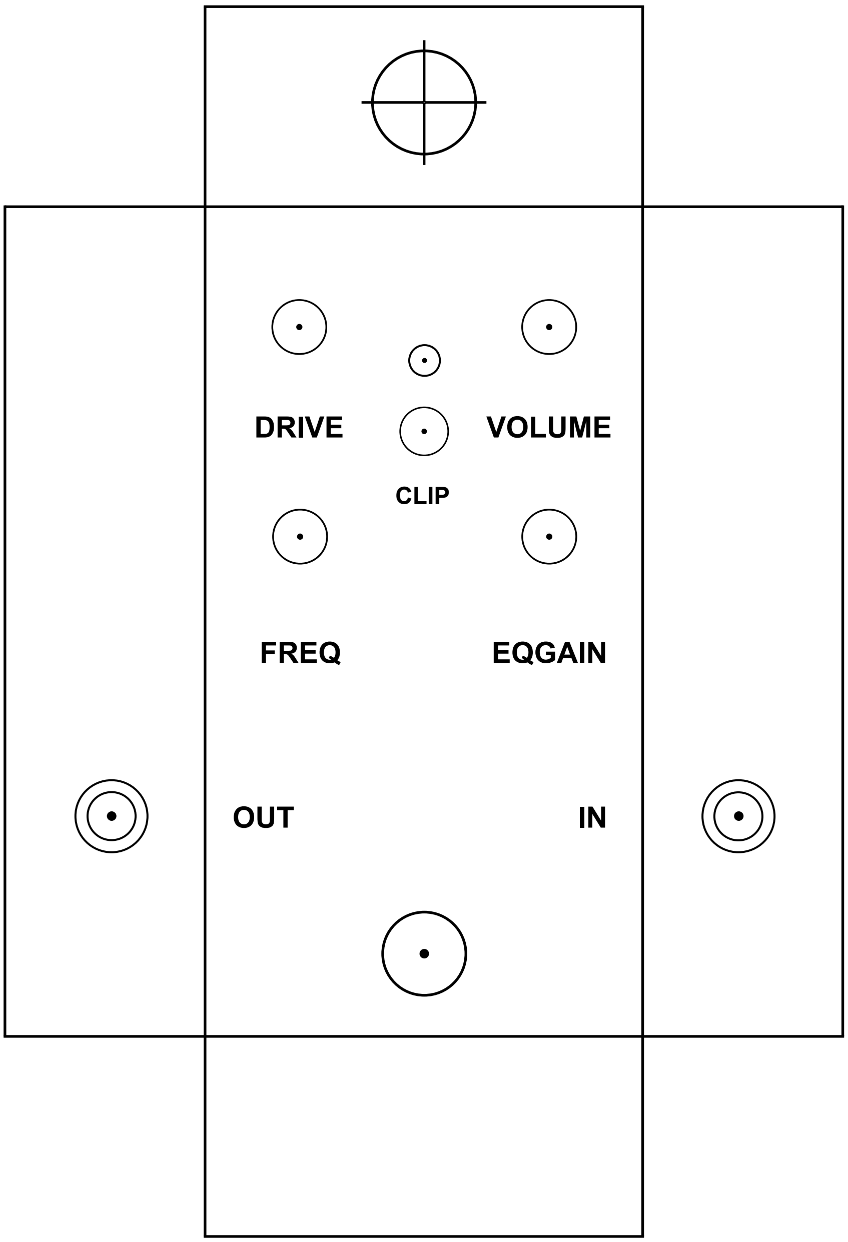
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

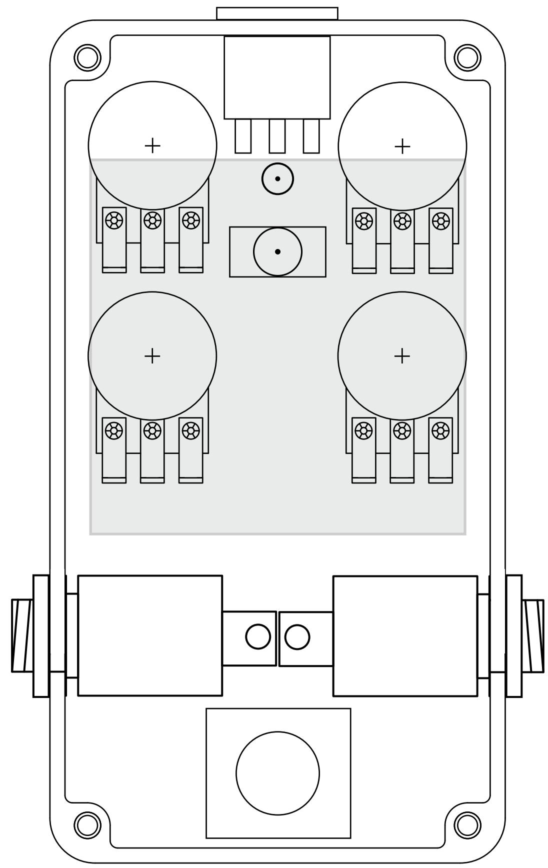
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

Drilling & Placement

Print this page and 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 111X enclosed jacks
- Kobiconn-style DC jack with internal nut

Standard Wiring Diagram

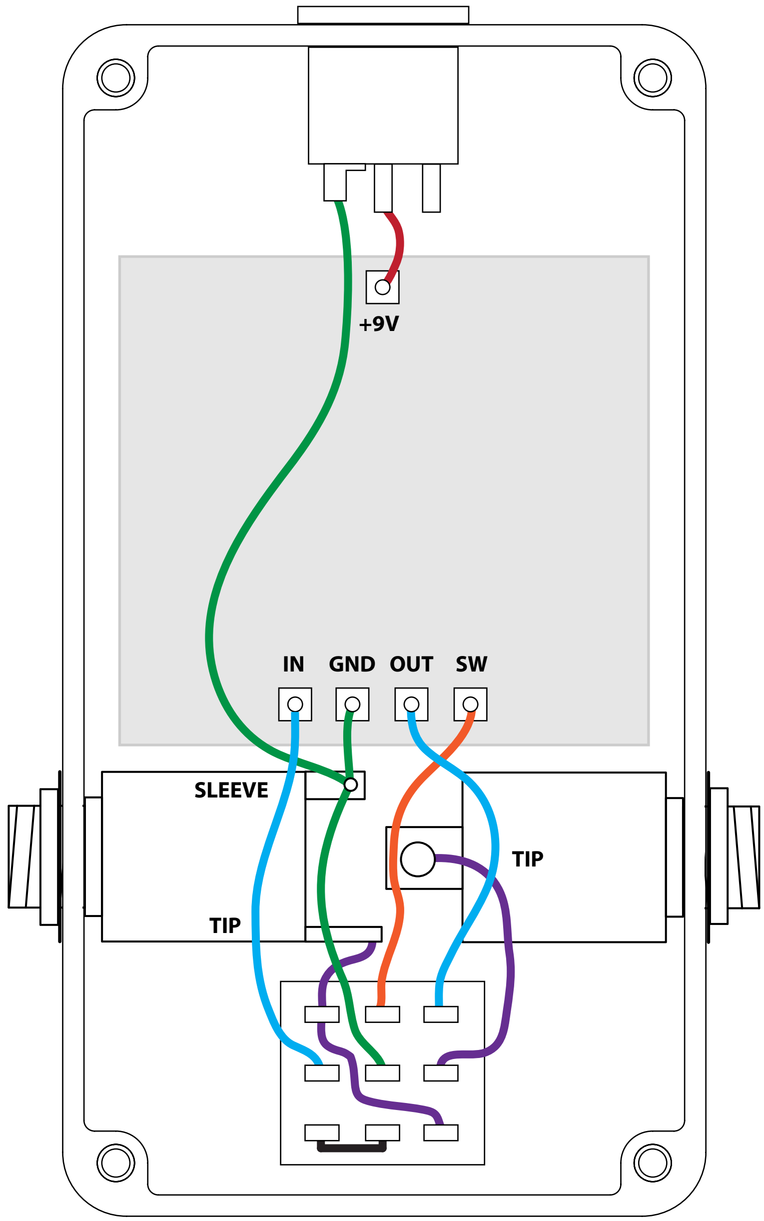
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)