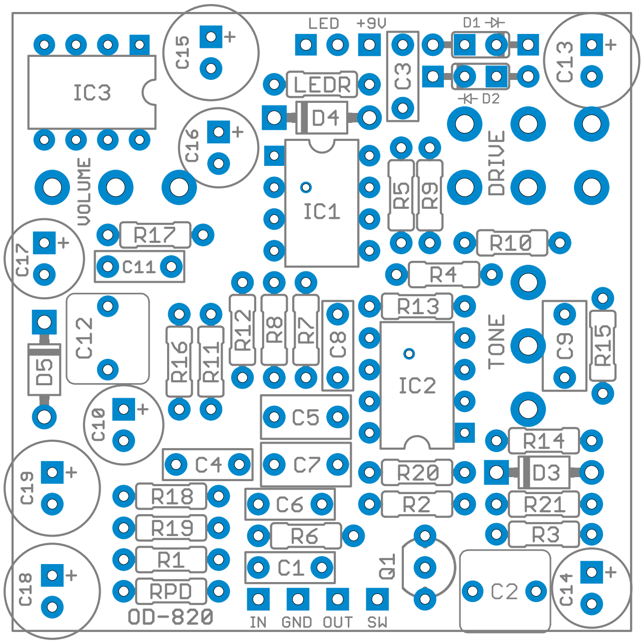
Nimbus Overdrive

Maxon OD-820 Overdrive Pro

aion electronics

Overview

Nimbus Project Link



The Nimbus Overdrive project is a straight clone of the Maxon OD-820 Overdrive Pro, also released as the VOP-9 Vintage Overdrive Pro. This pedal was intended to be an update to the legendary Tube Screamer, but it never really caught on. However, those who have the opportunity to play it are invariably impressed. It's the favored overdrive of Nigel Hendroff of Hillsong.

The OD-820 has drawn both technical and tonal comparisons to the Klon Centaur Professional Overdrive, which it was undoubtedly inspired by. Both of them use a charge pump to run at 18v (technically the OD-820 is +/-9v bipolar), and both of them use a dual potentiometer for the Drive control to simultaneously increase the drive and blend it with the clean signal. The similarities end there, but it's a little too close to be a coincidence.

Controls & Usage

The controls of the OD-820 are identical to that of a Tube Screamer as well as most other overdrive pedals.

- **Drive** controls the overall amount of drive, but with this pedal it also controls the balance between clean and overdriven. This produces a very unique tonal shift as you turn up the Drive control.
- **Tone** controls the treble response in the circuit. The tone control is taken directly from a Tube Screamer and has the same function: it cuts treble for the first half of the rotation, it's flat in the middle, and it boosts treble for the second half.
- **Volume** controls the overall output.

Build Notes

The original circuit calls for a great deal of power filtering, far more than any other overdrive pedal out there. I've included space for all the original filtering capacitors, but you can probably do without some of them (particularly one of the parallel 220uFs) and you won't notice any difference.

The circuit calls for a dual 250k linear pot for the Drive control. The circuit was designed to use right-angle PCB mount pots, and Steve at Small Bear Electronics was kind enough to stock these at my request. As far as I know, he is the only one who carries them.

You can find straight-pin versions elsewhere, such as at Tayda Electronics, but these will require a little patience and manual dexterity to get them to fit. See page 3 for more information on how to accomplish this.

Parts

Resistors

R1	10k
R2	1M
R3	10k
R4	560R
R5	62k
R6	2k2
R7	1k
R8	56k
R9	100k
R10	7k5
R11	20k
R12	56k
R13	1k
R14	1k
R15	220R
R16	47k
R17	100k
R18	100k
R19	1k
R20	10k
R21	10k
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	100n
C2	1uF film
C3	100pF [1]
C4	100n
C5	220n
C6	10n
C7	220n
C8	10pF [1]
C9	220n
C10	10uF [2]
C11	20pF [1]
C12	1uF film
C13	100uF 25v
C14	10uF
C15	100uF 25v
C16	10uF
C17	10uF
C18	220uF 16v
C19	220uF 16v

Semiconductors

Q1	2SK246
IC1 - IC2	JRC4558D
IC3	TC1044S
D1 - D2	1N914 [3]
D3 - D5	1N4001
LED	5MM

Potentiometers

Tone	20kW
Drive	250kB dual [4]
Volume	100kA

[1] **Can be ceramic or film.** I prefer to use multilayer ceramic capacitors (MLCC) for these small values.

[2] **Can be electrolytic or tantalum.** C10 is in the signal path.

[3] **Extra pads have been provided** in case you want to stack diodes (e.g. 2x 1N914s in either direction).

[4] **Get these from Small Bear Electronics if you can.** Non-right-angle versions can be found at places such as Tayda Electronics. If using one of these, see page 3 for more details on how to make these fit the PCB.

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.
- 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.

Wiring the 250k Drive pot

The original circuit calls for a **dual 250k linear pot** for the **Drive** control. Steve at Small Bear Electronics began stocking right-angle versions of these in early 2014, which is what I built this circuit around. However, non-right-angle versions are also available from places such as Tayda Electronics. Here are instructions on how to craft a right-angle pot from straight-pin if you are not able to source them from Small Bear.

To make these work for the OD-820, you'll first want to **bend each of the pins back toward the pot** to make a sort of eyelet (small needle-nose pliers work well for this). I recommend bending the pins of both rows inward toward each other.

From here, note that you do not have to follow the method below. You can just use these eyelets like normal solder lugs and run wires to the PCB pads. The PCB will still be held securely by the other two pots. But since I have a deep hatred for pot wiring, I prefer the method outlined here, although it may take a little longer.

Using thick wire (such as solid-core home electrical wiring, like from a hardware store), **loop the wire through the eyelets you made** and bend it so each wire sticks straight out. (You'll want to make the bottom row significantly longer so you have enough length to bend them upwards.)

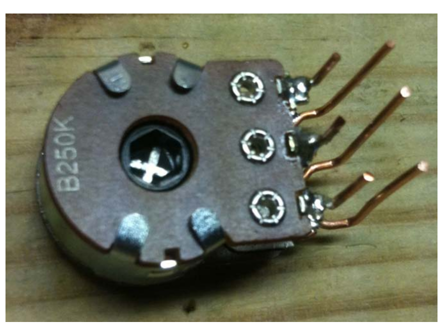
Squeeze the eyelets down to better hold the wire in place. Then, once they're all in position, solder them.

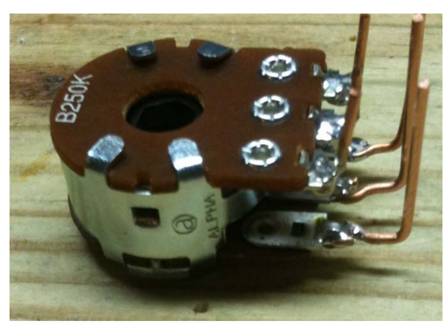
Bend the wires upward. The top row should go as nearly straight upward as possible. The bottom row should go out about a quarter inch and then up. The bottom row doesn't need to be terribly accurate since there's room to bend the wires outward or inward slightly to match the PCB pads.

Follow the build order in **General Build Instructions** below for wiring up the PCB, but read and take note of the following before you start:

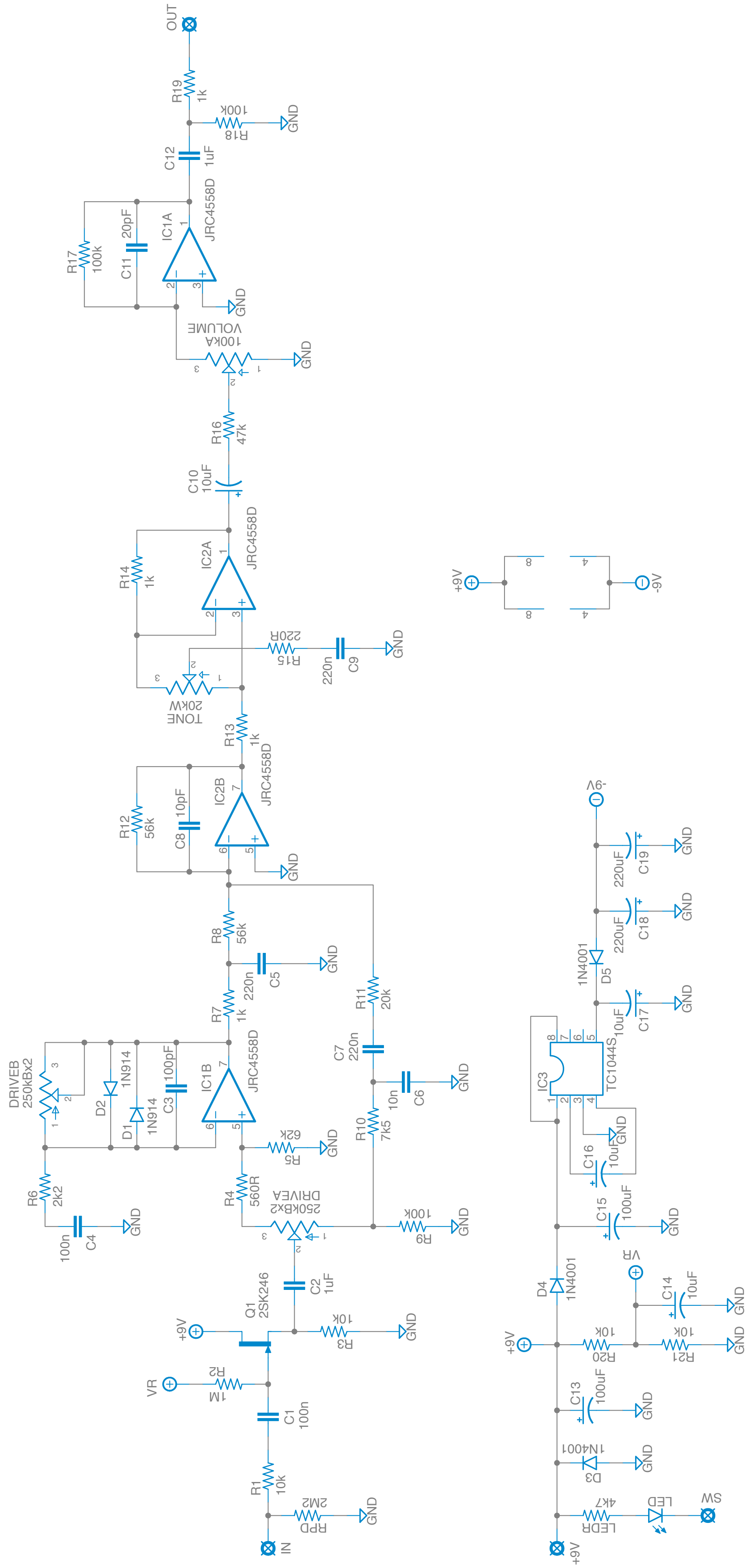
- When fitting the PCB onto the pots, you'll probably have to bend the wires around a bit more in order to get it to fit.
- **The PCB will not lay perfectly flat** since the top of a dual pot is slightly taller than the top of a standard pot. However, once it's soldered into place it'll be just fine.
- My recommended Alpha 16mm dust covers will not work with dual pots, so **you'll need to insulate the backs** in some other way. I use cardboard or pieces of a manila folder. Make it a little wider than it needs to be and bend it down the sides to help keep it in place. You don't want it shifting slightly and putting any of the tabs or rivets in contact with the solder joints on the PCB.

Here are some pictures of a Tayda pot that has been "retrofitted" to be right-angle PCB mount:





Schematic


OUT
R19 1k
R18 100k
GND
C12 1uF
IC1A JRC4558D
R17 100k
C11 20pF
VOLUME 100KA
R16 47k
C10 10uF
IC2A JRC4558D
R14 1k
R15 220R
C9 220n
TONE 20KW
R13 1k
IC2B JRC4558D
R12 56k
C8 10pF
+9V
-9V
R8 56k
C5 220n
R7 1k
R11 20k
C7 220n
C6 10n
R10 7k5
IC1B JRC4558D
C3 100pF
D1 1N914
D2 1N914
DRIVEB 250kBx2
R6 2k2
C4 100n
R5 62k
R4 560R
DRIVEA 250kBx2
R9 100k
C2 1uF
Q1 2SK246
R3 10k
R2 1M
VR
C1 100n
R1 10k
RPD 2M2
IN
IC3 TC1044S
C16 10uF
C15 100uF
C17 10uF
D5 1N4001
C18 220uF
C19 220uF
D4 1N4001
C14 10uF
R20 10k
R21 10k
C13 100uF
D3 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

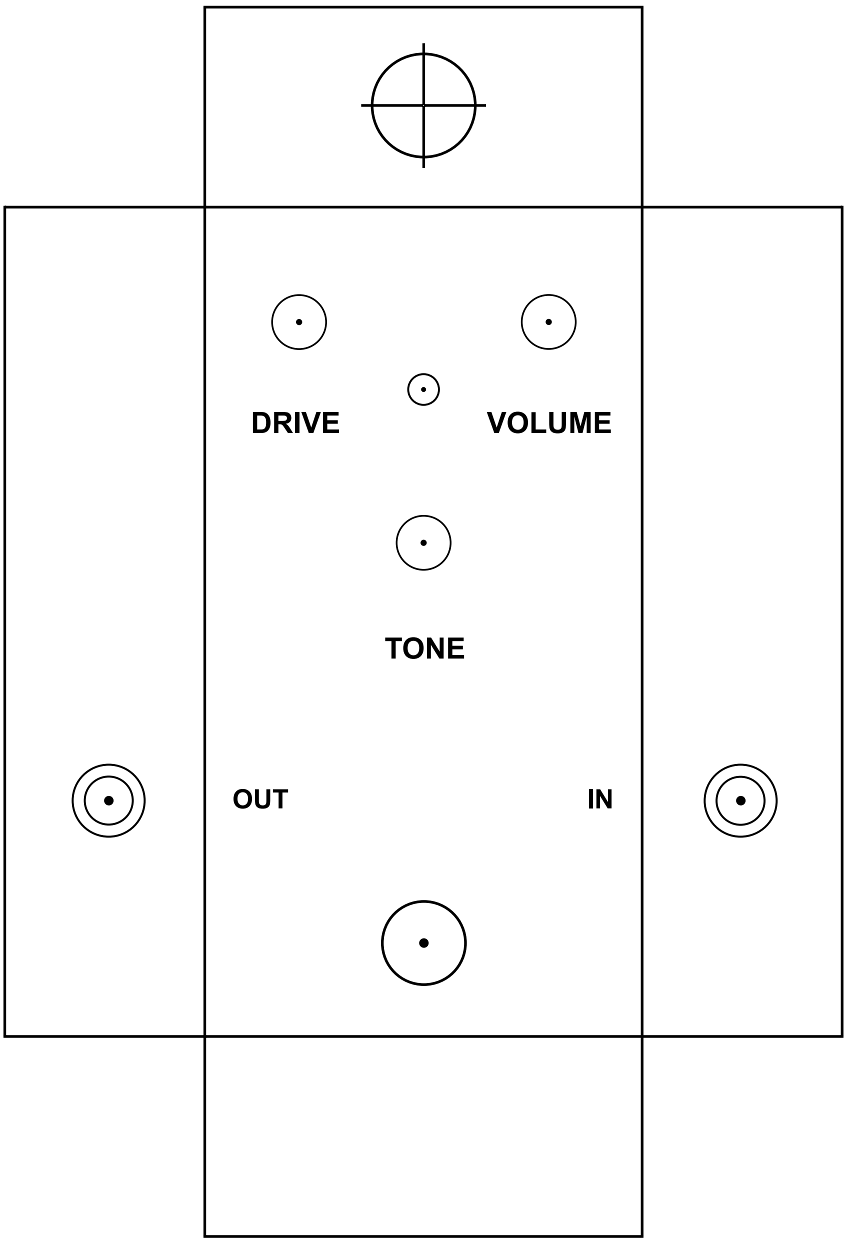
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

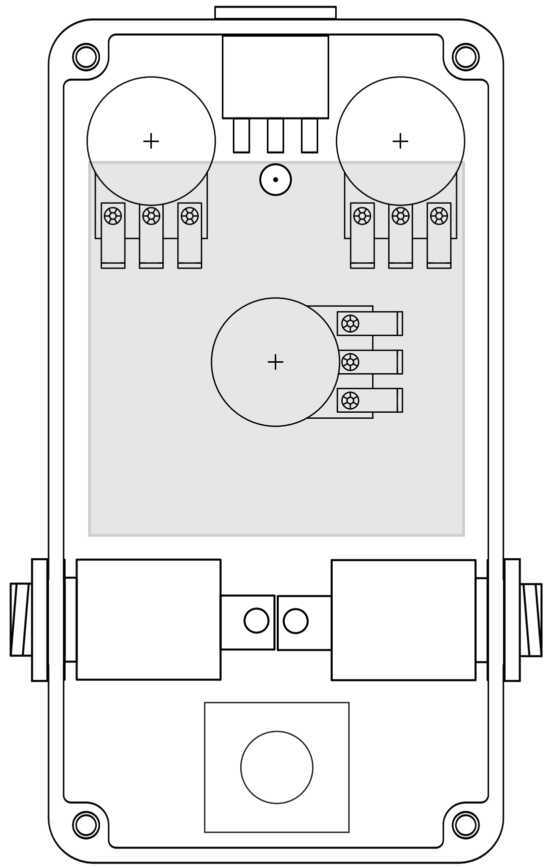
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

Standard Wiring Diagram

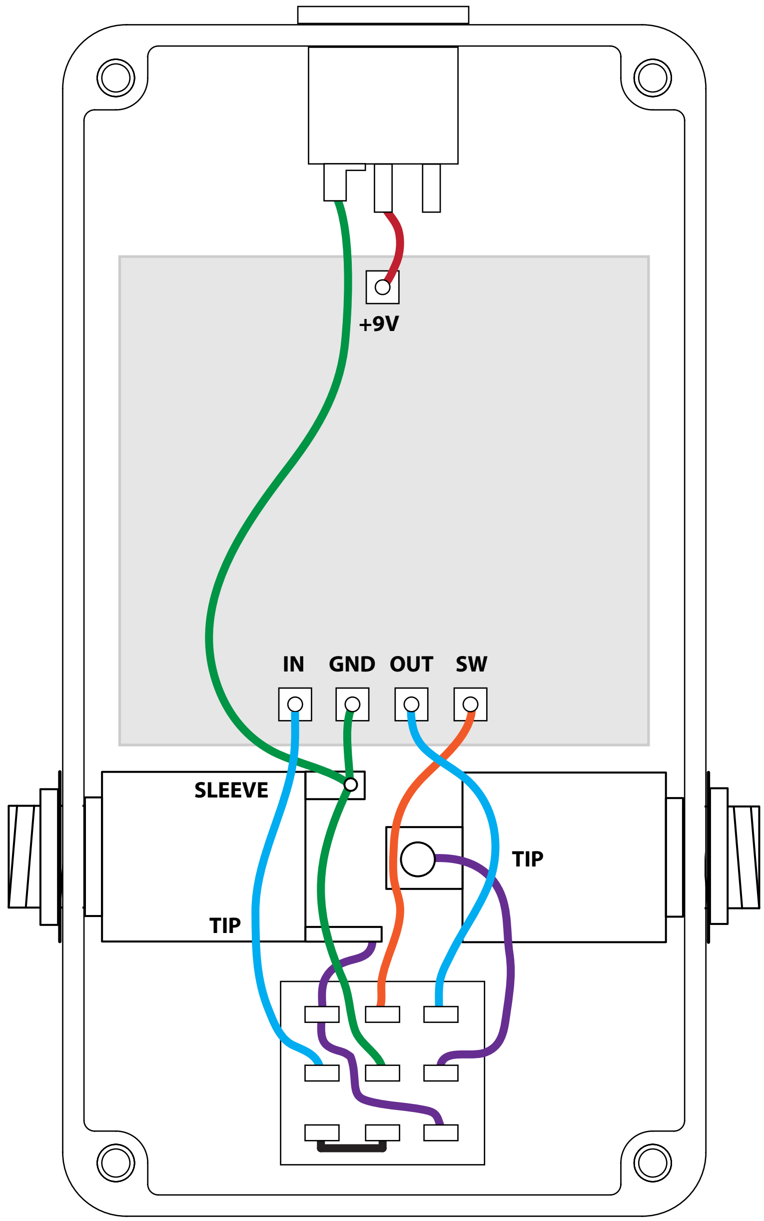
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)