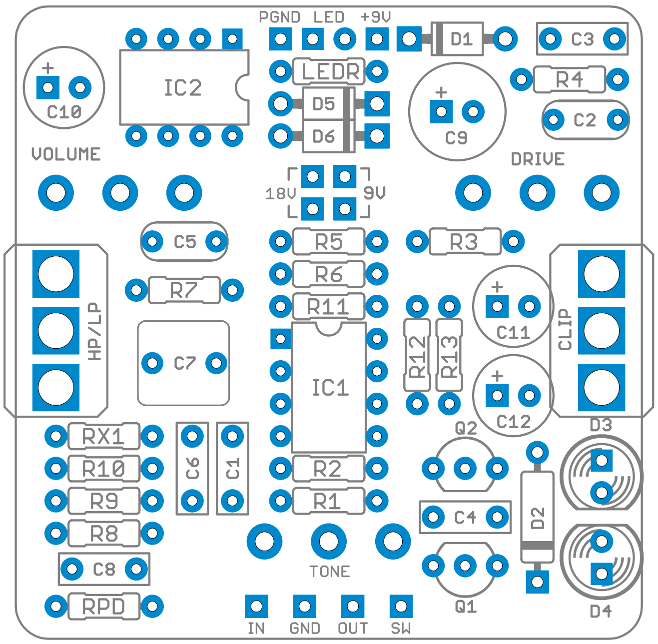
Titan Overdrive

Modified Voodoo Lab® Overdrive

Overview

Titan Project Link



The Titan Overdrive is a modified version of the Voodoo Lab® Overdrive pedal. There have been a few commercial pedals that were originally developed as modified VLODs that have an enormous following.

The Voodoo Lab® Overdrive was originally released in 1995 and was heavily inspired by the Distortion+/250 circuit. Other pedalbuilders later added a tone control and a different diode-clipping setup and tweaked two or three other values to produce modified versions.

While these modified versions were originally designed for 9V operation, it's very common for people to run them at 18 volts, so this PCB has an optional charge pump to get 18V operation from a standard 9V supply.

This project includes the HP/LP switch from one notable VLOD-inspired circuit, but adds a third option midway between the two. It also includes a diode clipping switch so you can experiment with other options.

Controls & Usage

- **Drive** controls the gain of the op-amp stage which increases the amount of signal clipping.
- **Tone** is a simple variable low-pass filter to control the amount of treble cut at the end, very similar to the tone pot on a guitar.
- **Volume** controls the overall output level of the effect.
- **HP/LP** (High Peak / Low Peak) controls the frequency response of the tone control.
- **Clipping** allows you to select a second set of clipping diodes or bypass the diodes altogether.

Modifications & Experimentation

A charge pump has been included for built-in 18V operation. Use the jumpers on board to select either 9V or 18V mode. You can even put this on an external switch if you want, although you might need to use a bigger enclosure or mount it to the side. **You must use the jumpers to select either 9V or 18V mode or the effect will not work!**

The **HP/LP** switch controls a series resistor right before the output. It's often misunderstood to be a boost switch of some sort. While it does affect the overall signal level, its primary purpose is actually to form an R-C filter in series with C8, the tone capacitor, and so it affects the frequency of the tone that gets rolled off. The tone control itself then affects how much of this treble-bleed is allowed to actually pass through to ground.

The original unit has **33k** of total resistance in LP mode and **13.2k** resistance in HP mode (33k and 22k in parallel). I added a third position to this switch, a 10k resistor (RX1), that drops the total resistance down to 7.6k, which could be considered "Extra High Peak" mode. However, you could also set RX1 higher to get a setting midway between High Peak and Low Peak—for example, a **56k** resistor for around **21k** total resistance.

Parts

Resistors

R1	10k
R2	470k
R3	18k
R4	2k2
R5	10k
R6	10k
R7	150k
R8	39k
R9	33k
R10	22k
R11	100R
R12	10k
R13	10k
RX1	10k or 56k
RX2	jumper [1]
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	22n
C2	220pF MLCC
C3	68n
C4	1n
C5	220pF MLCC
C6	100n
C7	1uF film [2]
C8	47n
C9	47uF electro
C10	10uF electro [3]
C11	10uF electro
C12	10uF electro

Semiconductors

IC1	TL082
IC2	TC1044 [3]
Q1, Q2	2N7000
D1	1N4742 zener
D2	1N34A [4]
D3, D4	LED [5]
D5, D6	1N4002 [3]
LED	5MM

Potentiometers

Drive	500kA [6]
Tone	10kB [7]
Volume	500kA [8]

Other

Clip, HP/LP	SPDT center off

[1] **Tone control minimum:** The tone control can get pretty dark at its minimum settings. RX2 has been added to counteract this, allowing you to raise the minimum position of the control. **1k** is a good place to start for this.

[2] **Capacitor type:** C7 was a tantalum in the original, but I included space for the larger film caps. Polarity markings have been added to the silkscreen in case you want to use tantalum.

[3] **Part of the voltage doubler circuit.** You can omit these parts if building a straight 9V version.

[4] **Germanium diode:** Any germanium diode should work fine here—the exact part number isn't important. This diode was added in V4, so it can be jumpered if you're building a lower version number.

[5] **LEDs:** I recommend using red diffused 5mm LEDs for the second clipping option.

[6] **Drive pot value:** One commercial version of the modified VLOD switched to a 1MA pot for the Drive control. General consensus is that this was not an improvement and the pedal loses its character above 50%. I recommend sticking with 500kA here.

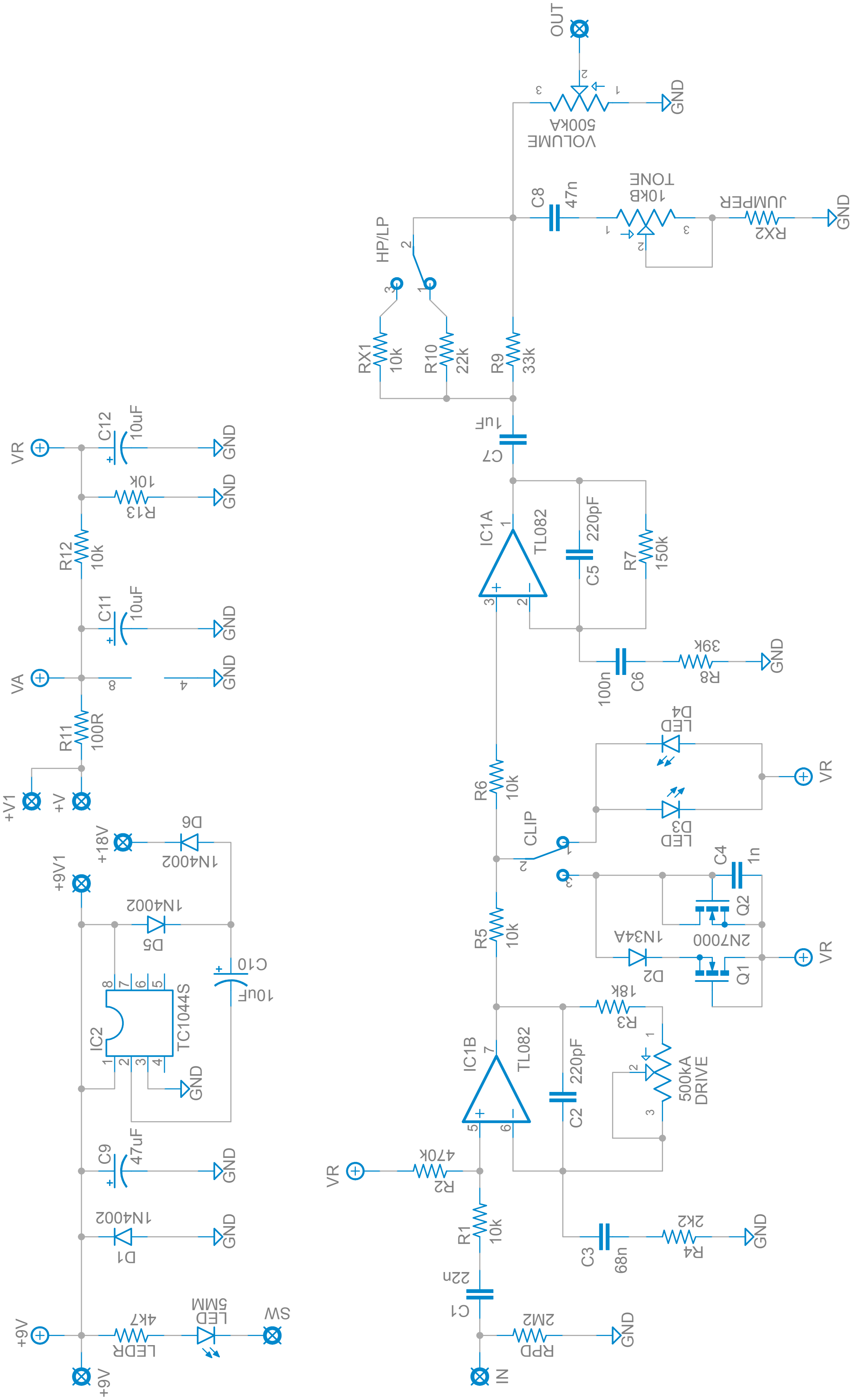
[7] **Tone pot value:** Early versions of the modified VLOD used **25kA** for the tone control and **100n** for C8.

[8] **Volume pot value:** The first version of the modified VLOD used **100kA** for the output volume control. The 500k gives a lot more output volume with the downside that it's a little harder to dial in at lower volumes.

Additional Part Notes

- Capacitors are shown in nanofarads (n or nF) where appropriate. 1000n = 1uF.
- 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 insulate.

Schematic


OUT
VOLUME
500KA
GND
TONE
10kB
JUMPER
RX2
C8
47n
HP/LP
RX1
10k
R10
22k
R9
33k
C7
1uF
IC1A
TL082
C5
220pF
R7
150k
C6
100n
R8
39K
D4
LED
D3
CLIP
VR
R6
10k
R5
C4
1n
Q2
2N7000
Q1
D2
1N34A
R3
18K
IC1B
C2
500kA
DRIVE
R2
470K
R1
C1
22n
C3
68n
R4
2k2
RPD
2M2
IN
VA
R11
100R
R12
R13
10K
C11
10uF
C12
+V1
+V
+18V
D6
1N4002
+9V1
D5
IC2
TC1044S
C10
C9
47uF
D1
+9V
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

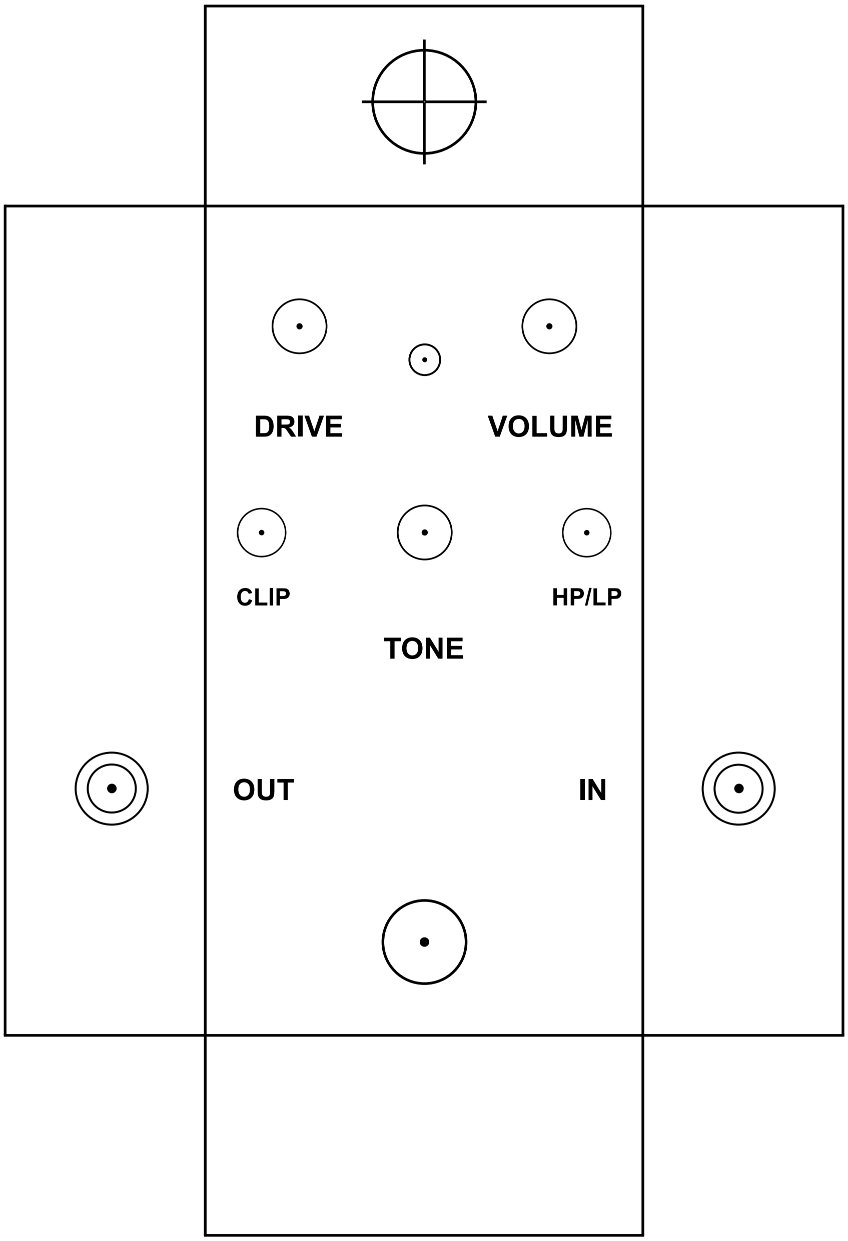
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

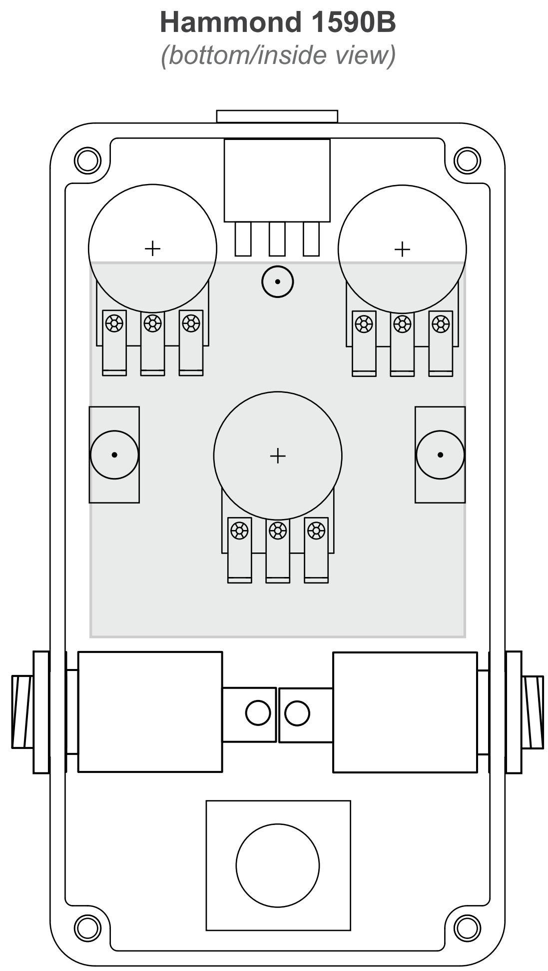
Sockets

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 cut out the drilling template below. 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.





Parts Used

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

Standard Wiring Diagram

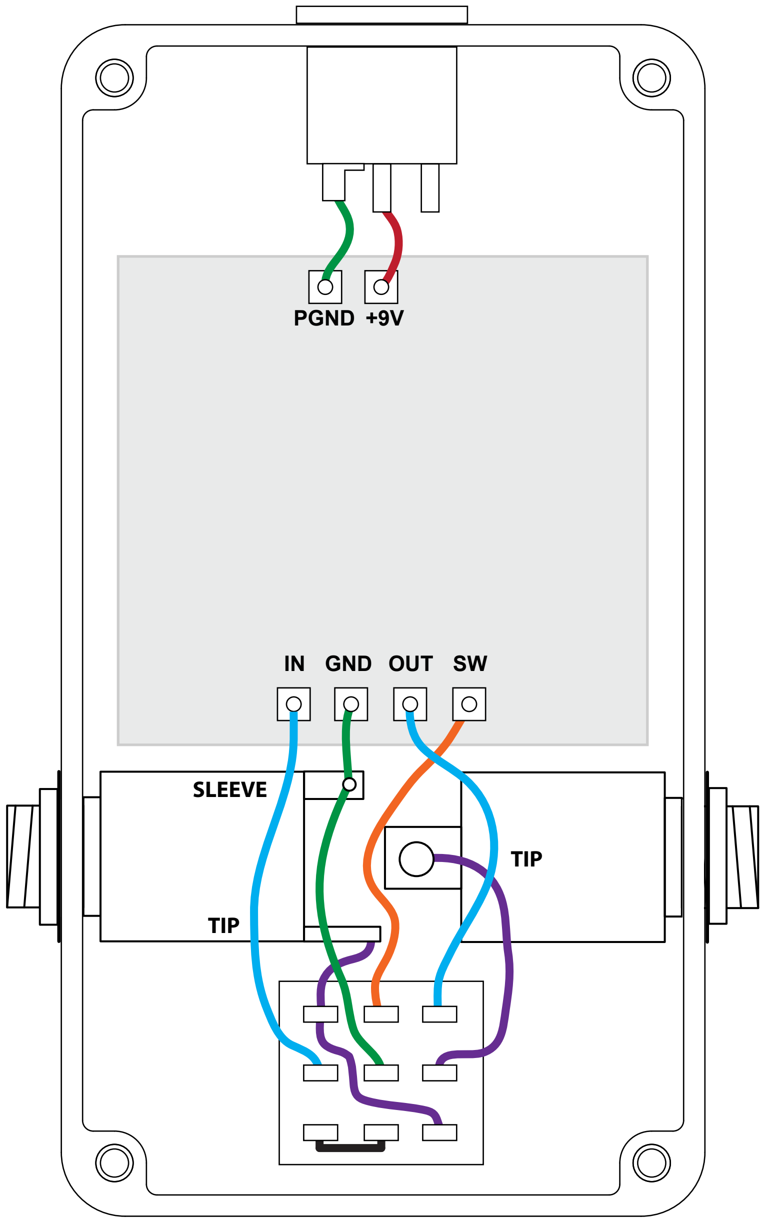
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)