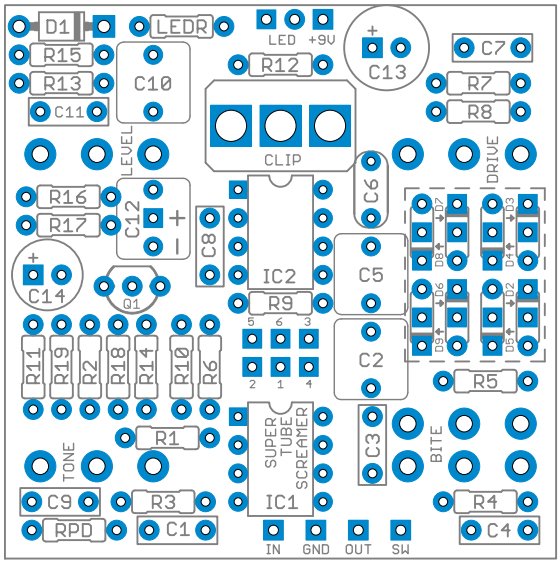
Cirrus Overdrive

Ibanez ST-9 Super Tube Screamer

Overview

ST-9 Super Tube Screamer PCB project page



The CIrrus Overdrive is a clone of the Ibanez ST-9 Super Tube Screamer, a very rare pedal that to my knowledge was only released in Europe and Japan around 1984, and these days routinely sells for $600 or more. It has never been commercially cloned and it's not been well-traveled in the DIY scene either.

The ST-9 is identical to a standard Tube Screamer except with the addition of a pre-clipping midrange boost with selectable frequency, allowing for a great deal more tonal flexibility.

There was another ST-9 variant called the STL, which is slightly different: the midrange EQ and clipping stages have been flipped around, but otherwise the circuits are the same. The order of these stages makes a significant tonal difference, and to that end, this PCB project has been designed so you can build either the ST-9 or STL variants by setting a few jumpers. You can even omit the mid boost and build a stock Tube Screamer.

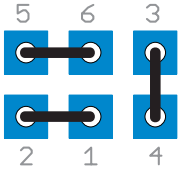
Controls & Usage

- **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.
- **Tone** is an active boost/cut of the treble in the circuit, identical in function to a standard Tube Screamer.
- **Mid Boost** is a fixed 8 dB boost with sweepable frequency.
- **Clipping switch** allows you to use an alternate set of diodes, with a "diode lift" mode in the middle position if you use a center-off switch.

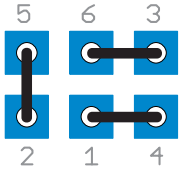
Setting the jumpers

The placement of the mid boost control is configurable so that you can build a ST-9 or STL, or even a stock Tube Screamer. There are six pads located between IC1 and IC2, and the jumpers can be set as follows:

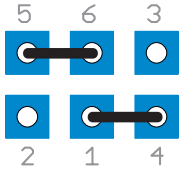
ST-9
(Mid EQ before clipping)



STL
(Mid EQ after clipping)



TS-9
(No midrange EQ)



You must choose one of them or the effect will not work!

Modifications

Any standard Tube Screamer modifications will work for this circuit. See the Stratus PCB project for more detailed information on modifications—just note that the part numbers are different here.

Parts

Resistors

R1	1k
R2	510k
R3	220k
R4	7k5 [1]
R5	7k5 [1]
R6	10k
R7	51k
R8	4k7
R9	1k
R10	10k
R11	220R
R12	1k
R13	1k
R14	510k
R15	10k
R16	10k
R17	470R
R18	10k
R19	10k
LEDR	4k7
RPD	2M2

Capacitors

C1	47n
C2	1uF film
C3	5n6 [1]
C4	12n or 15n [1]
C5	1uF film
C6	51pF MLCC
C7	47n
C8	100n
C9	220n
C10	1uF film
C11	100n
C12	1uF film [2]
C13	100uF electro
C14	47uF electro

Semiconductors

Q1	2N5088
IC1–IC2	JRC4558D
D1	1N4002
D2, D4	1N914
D3, D5	jumper
D6–D9	[3]
LED	5mm LED

Potentiometers

Drive	500kA
Level	100kB
Tone	20kW
Mid Boost	100kC dual [1]

Other

Clip	SPDT center off [3]

Build Notes

[1] The original ST-9 and STL circuit uses a **20kC dual pot** for the Mid Boost control. Since these are not available without special-ordering a minimum quantity of 2000 from Alpha, the resistors and capacitors in the mid-boost section have been changed by a factor of 5 in order to accommodate the more readily available 100kC dual pots while preserving the same frequency adjustment. The original unit uses **27n** for C3 and **68n** for C4, and **1k5** for R4 and R5; if by some miracle you do have a 20kC pot on hand, use these original values. In my opinion, the midrange boost is not as drastic as it could be. See the project page for modifications.

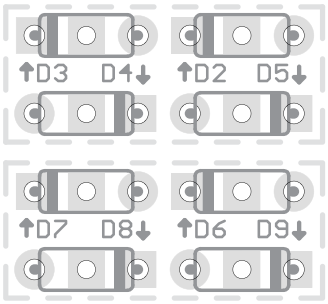
[2] **Value change:** The original uses a **10uF electrolytic capacitor** here. If you do want to use a 10uF electro here, the positive and negative pads have been marked on the silkscreen.

[3] **Your choice.** This is the second set of clipping diodes. See next page for some ideas.

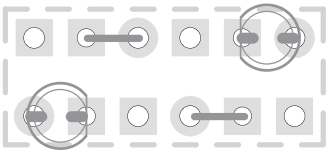
Additional Part Notes

- 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.

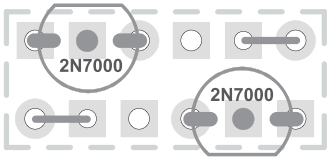
Diode Clipping Options



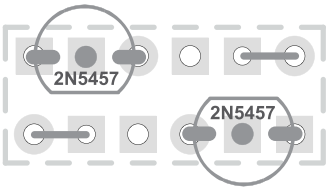
Close-up of silkscreen. (Square pad pairs are always connected.)



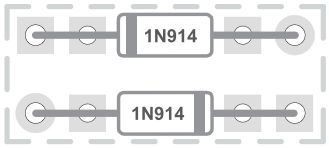
LED clipping



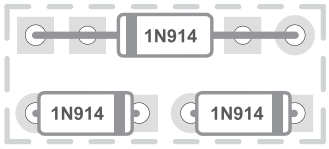
MOSFET clipping



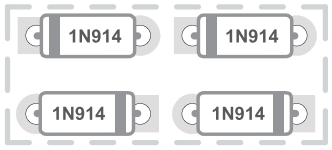
JFET clipping



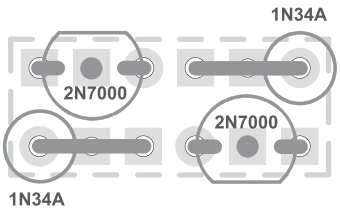
Stock TS-9 (symmetrical clipping)



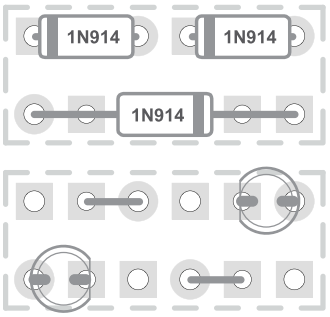
Boss SD-1 (asymmetrical clipping)



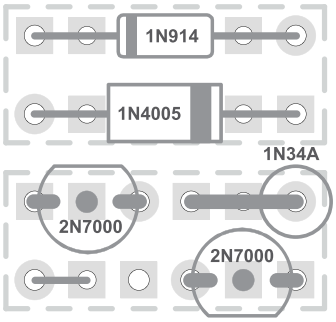
Lovepedal Eternity, Marshall Bluesbreaker



Zendrive

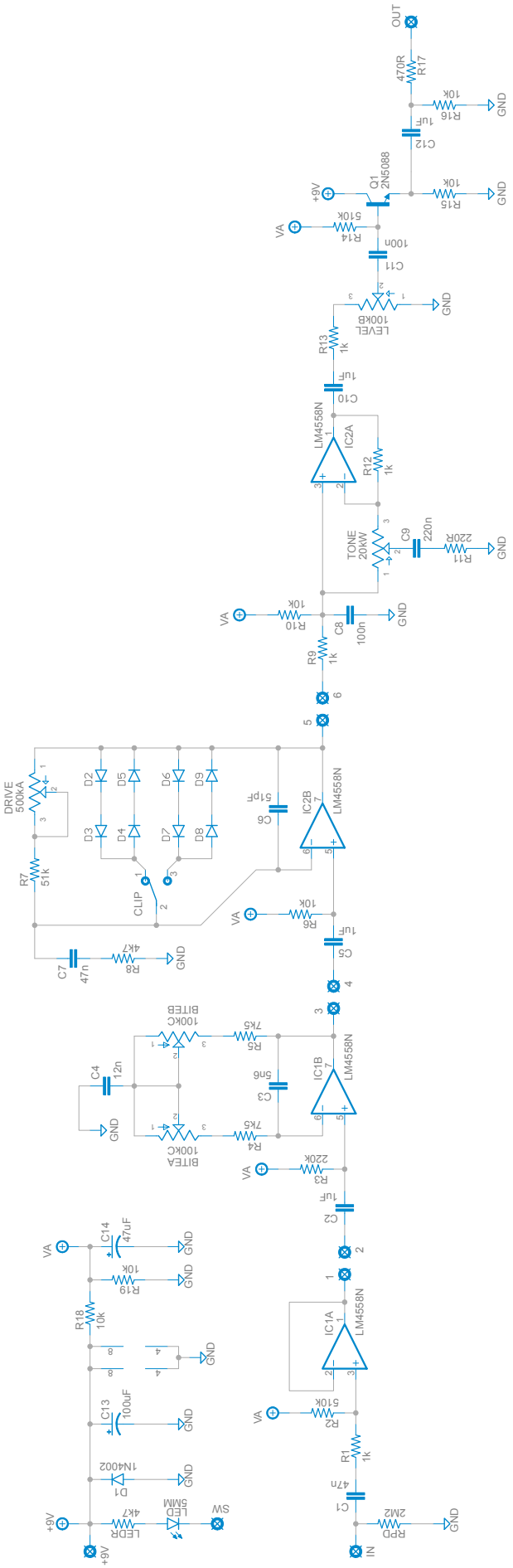


Landgraff Dynamic Overdrive (asymmetrical silicon on top, LED on bottom)



Fulldrive 2 (Silicon top, MOSFET bottom)

Schematic


IN
OUT
+9V
VA
GND
LM4558N
IC1A
IC1B
IC2A
IC2B
Q1
2N5088
1N4002
LED
SW
CLIP
DRIVE
TONE
LEVEL
BITEA
BITEB
R1
R2
R3
R4
R5
R6
R7
R8
R9
R10
R11
R12
R13
R14
R15
R16
R17
R18
R19
C1
C2
C3
C4
C5
C6
C7
C8
C9
C10
C11
C12
C13
C14
D1
D2
D3
D4
D5
D6
D7
D8
D9

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

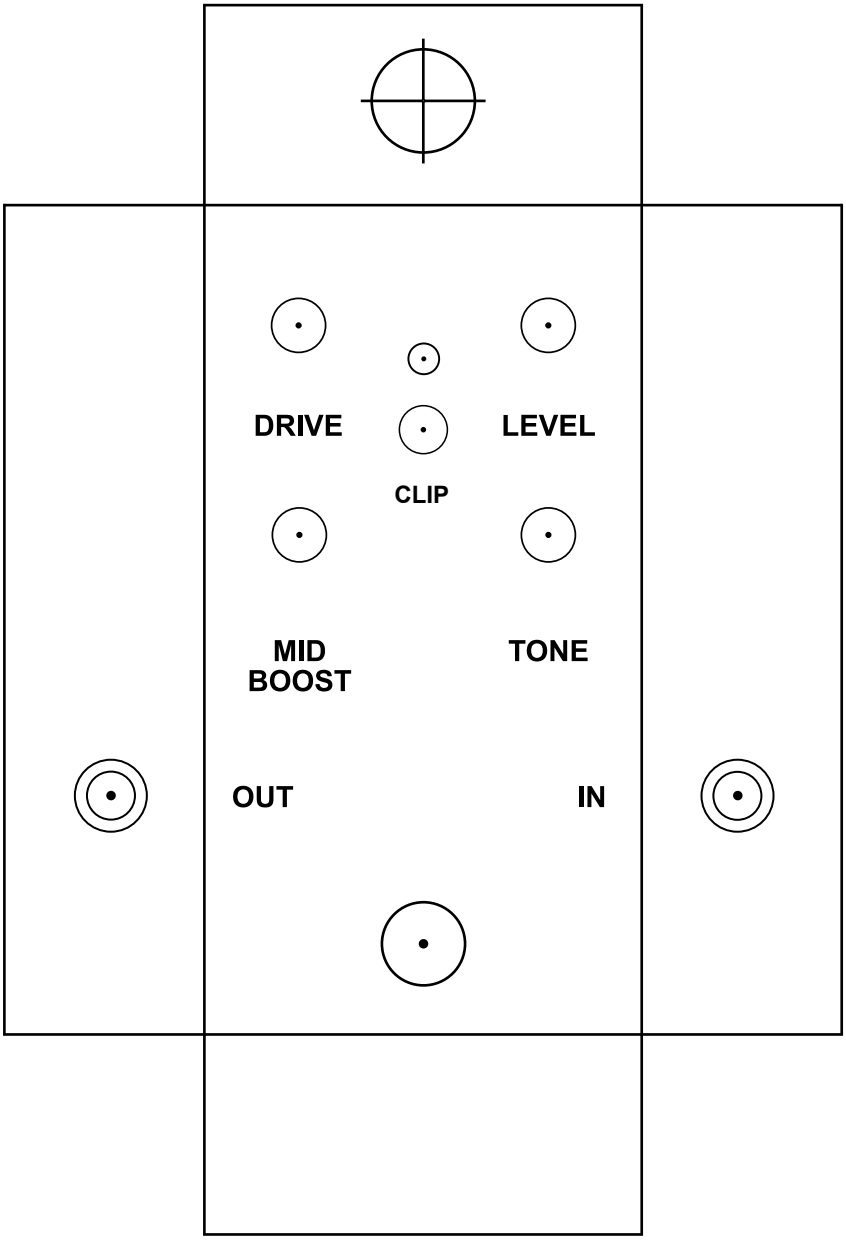
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

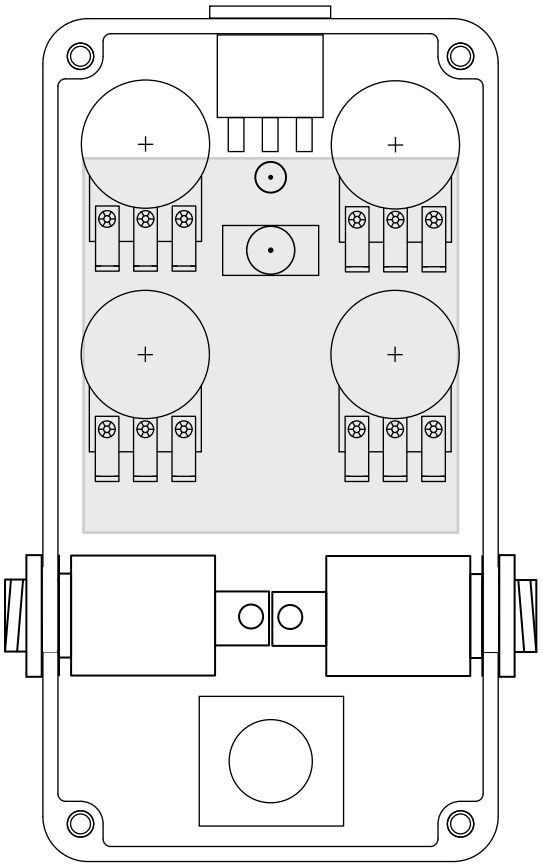
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

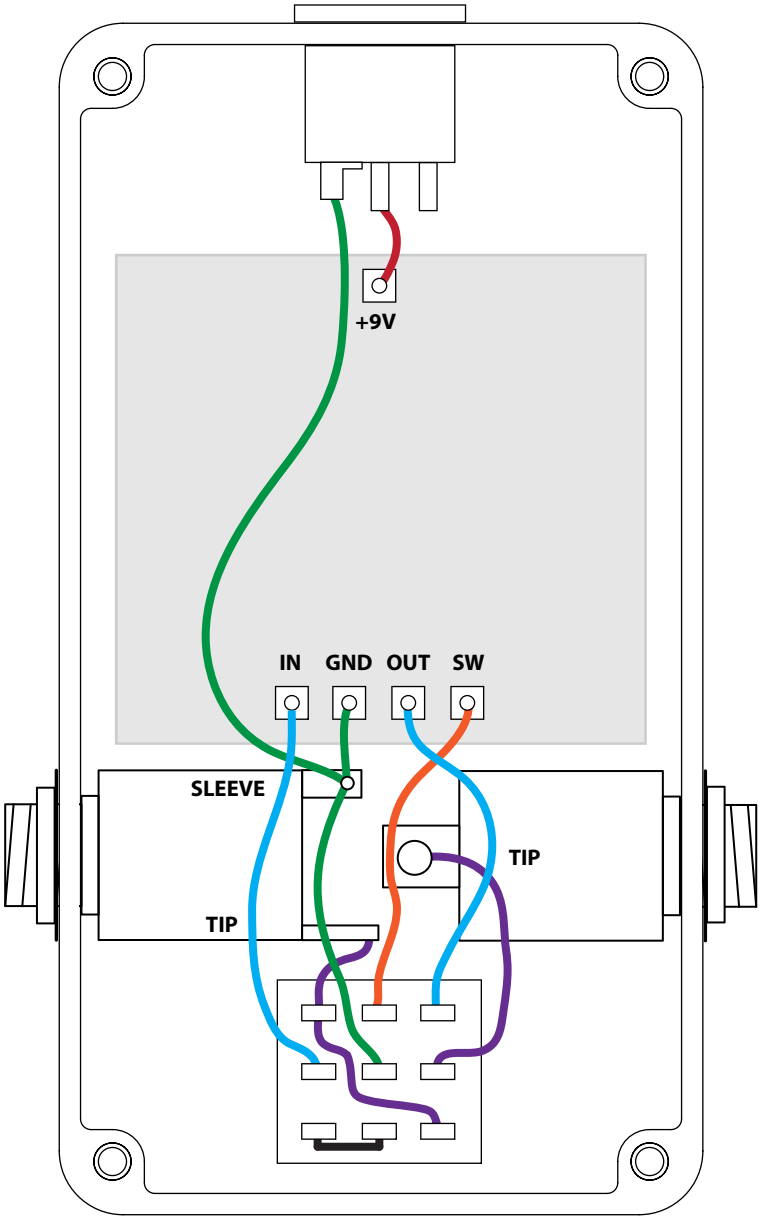
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)