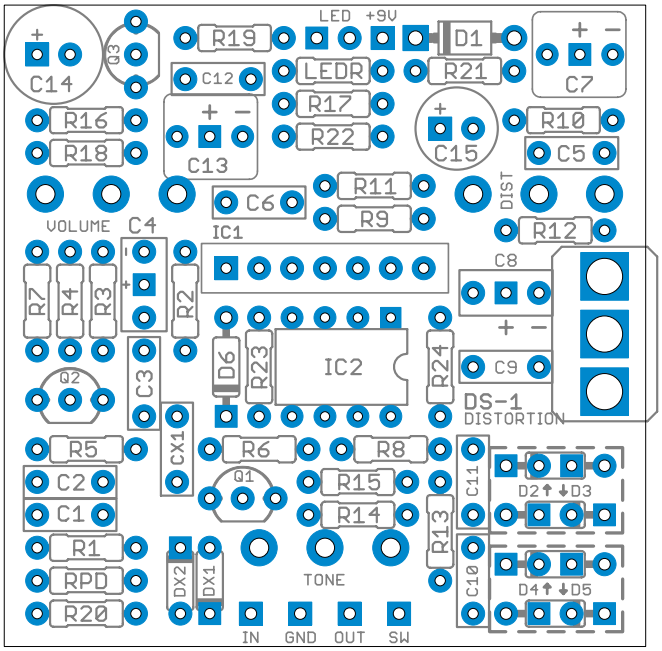
Comet Distortion

BOSS DS-1 Distortion

Overview

Comet Project Link



The Comet Distortion project is a clone of the classic orange BOSS DS-1 Distortion, in continuous production since 1978. The circuit has a few different things going on—the pregain and tone sections are very similar to the Big Muff, while the diode-to-ground hard clipping is similar to circuits like the Distortion+.

The DS-1 had a major circuit revision in 1994 that dropped the original single op-amp in favor of a dual, along with some other changes to accompany the new op-amp. The original “Made in Japan” version is very well-regarded, while the revised “post-1994” version is not as much. However, it is one of the most frequently-modified pedals out there, with great mods available from Keeley, Analogman and Monte Allums. The circuit itself is a great one—the BOSS pedal just suffers from cheap components.

The Comet Distortion will allow you to build either the pre- or post-1994 versions of the circuit, as well as incorporating most of the popular modifications.

Controls & Usage

The DS-1’s controls are standard drive/distortion controls:

- **Drive** controls the amount of gain from the op amp that is fed into the diode clipping stage.
- **Tone** is similar to the Big Muff tone control, panning between a bass and treble emphasis.
- **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.) If you use a SPDT center-off switch, the middle position becomes a diode lift mode, but you can also use a regular SPDT if you don’t care about this.

The pre-1994 version uses a **TA7136P** single-in-line op amp. The revised post-1994 circuit uses a poor-quality dual op amp, and one of the most common modifications is to switch this out with a better op amp such as an **OPA2134** or **JRC4558**. However, this is a tricky mod and requires an awkward breakout adapter, since the dual op amp is also a single-in-line style. I elected to build the layout around a DIP8 dual op amp to accommodate. (Ironically, then, the only version of the DS-1 that you *can’t* build with this PCB is the one you can buy for $40 from any music store today.)

Due to the similarity between the DS-1’s pregain stage and the Big Muff’s clipping gain stages, this PCB has space for a few extra diodes and capacitors to convert this stage to Big Muff specs. (Thanks to Build Your Own Clone for this mod!) This will tighten up the bass and change the distortion character.

Parts (vintage / modern values)

Resistors

R1	1k
R2	470k
R3	10k
R4	100k
R5	470k
R6	10k
R7	22R
R8	100k
R9	100k
R10	4k7
R11	27k / (omit)
R12	2k2
R13	6k8
R14	2k2
R15	6k8
R16	10k
R17	1M
R18	10k
R19	1k
R20	100k
R21	10k
R22	10k
R23	(omit) / 47k
R24	(omit) / 1k
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	47n
C2	42n [1]
C3	250pF
C4	470n / 68n
C5	250pF / 100pF
C6	150pF / (omit)
C7	1uF / 470n [2]
C8	470n
C9	10n
C10	22n
C11	100n
C12	47n
C13	1uF [2]
C14	100uF
C15	47uF
CX1	(omit)

Semiconductors

IC1 *(vintage)*	TA7136P [3]
IC2 *(modern)*	TL072 [3]
Q1-Q3	2N5088
D1	1N4002
D2, D3	1N914 [4]
D4, D5	3mm LED [5]
D6	1N914 *(modern only)*
DX1, DX2	(omit)
LED	5MM

Potentiometers

Distortion	100kB
Tone	25kB
Volume	100kB

Other

CLIP	SPDT center off

See following page for footnotes.

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.

Build Notes

[1] With the FET switching in the original, there are two capacitors in series here, 470n and 47n. A series calculation puts the “consolidated” value for this at **42n**. However, since 42n is a pretty rare value for a capacitor, the more common **47n** can be used instead and won’t be noticeably different.

[2] **Electrolytic or film.** The original uses electrolytics as coupling capacitors, so for “vintage tone” the lower fidelity of electrolytic capacitors may be desired. The layout has room for full-sized film capacitors, but the polarity is marked if you want to use electrolytics.

[3] **Don’t use both of these.** Hopefully this goes without saying, but you only want to use one of these op amps depending on which version you want to build. Bad things might happen if you were to use both of them.

[4] **Stacking diodes:** The original diodes in the MIJ version are **1S2473**, which are long out of production and very difficult to find. There is not anything special about these diodes to warrant tracking them down and paying high prices for them. Their notable characteristic is that they clip between 1 and 1.2 volts, which is far higher than the standard 1N914s used in the modern DS-1 and will give a more open and less compressed sound which is critical for a MIJ clone. I recommend combining a **1N914** (0.6V) and a **BAT41** (0.4V) in series on each side to get an equivalent clipping threshold, which will require standing them on end. You could also try a **BA243**, **BA278** or **BA282** here, which all clip at around 0.9v and are easy to come by.

[5] **Your choice.** LEDs are very commonly used as alternate clipping diodes in a DS-1 mod.

Modifications

The BOSS DS-1 is one of the most frequently modded pedals out there, and on a personal note, was both the first guitar pedal I owned and the first one I ever modded. Despite its popularity, the off-the-shelf $40 version is shrill, brittle and uneven. Fortunately it doesn’t take much to turn it into a very good distortion pedal!

The IC is the first thing upgraded in most modifications, and the single thing that makes the biggest difference. Since this layout was made for a DIP8 op amp rather than a single-in-line like the original post-’94 version, you have an abundance of excellent options available, but a couple of good choices are the **TL072**, **JRC4558**, and **Burr-Brown OPA2134** (more expensive but much higher fidelity).

Beyond that, here are a few value changes inspired by mods from Keeley, Monte Allums and Analogman.

Mod 1

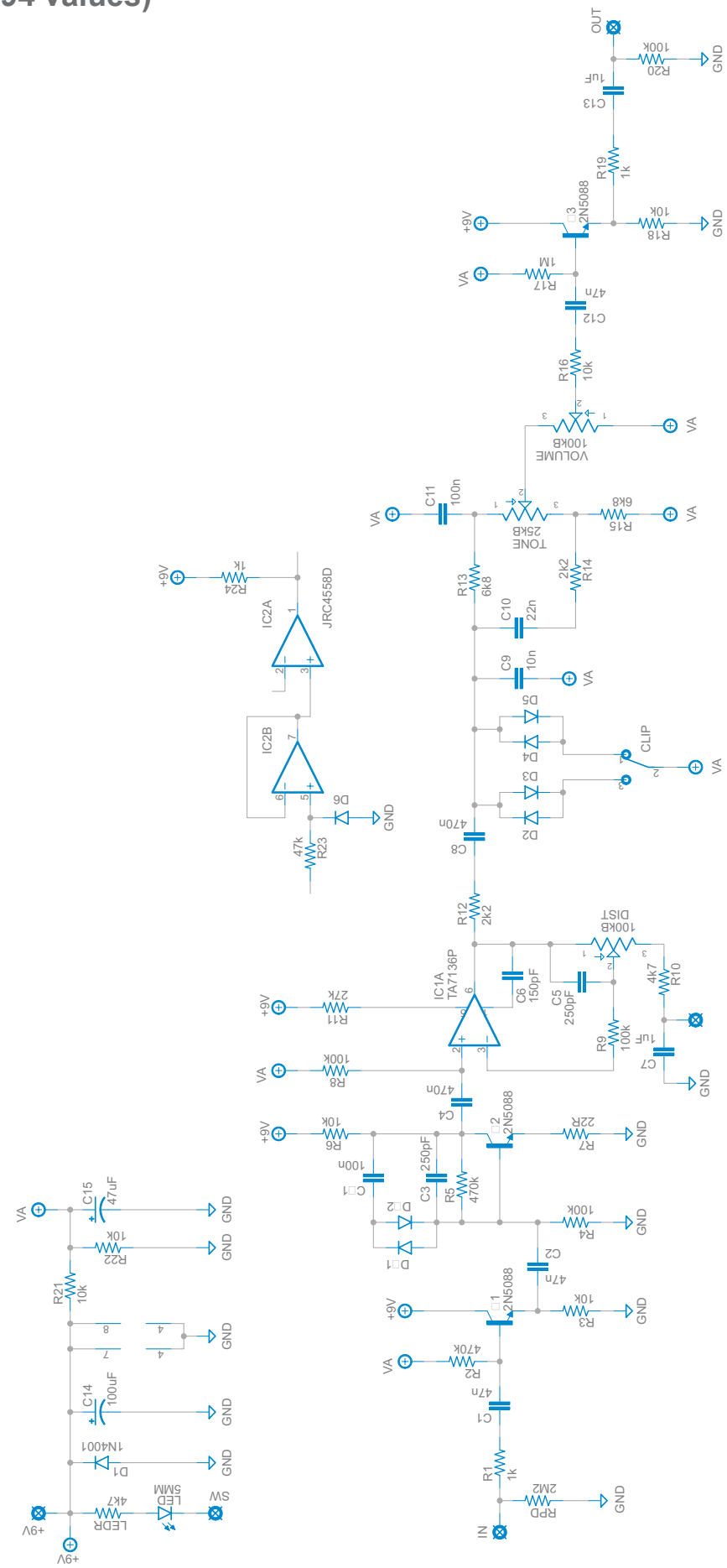
IC1	OPA2134
C1	100n
C2	100n
C7	1uF
C10	47n
C12	100n

Mod 2

All changes from Mod 1, plus...	
C3	470pF
C4	100n
C5	220pF
R10	2k4
R12	1k5
R23	20k

Big Muff mod: The DS-1 has a couple of building blocks in common with the Big Muff, notably the tone control and the gain stage before the op amp. Some have experimented with adding a couple of parts to the DS-1’s gain stage to make it identical to a Big Muff clipping stage. To do this, put a 100n cap in **CX1** and two 1N914s for **DX1** and **DX2**. This will clip the signal before it gets amplified by the op amp.

Schematic (pre-1994 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

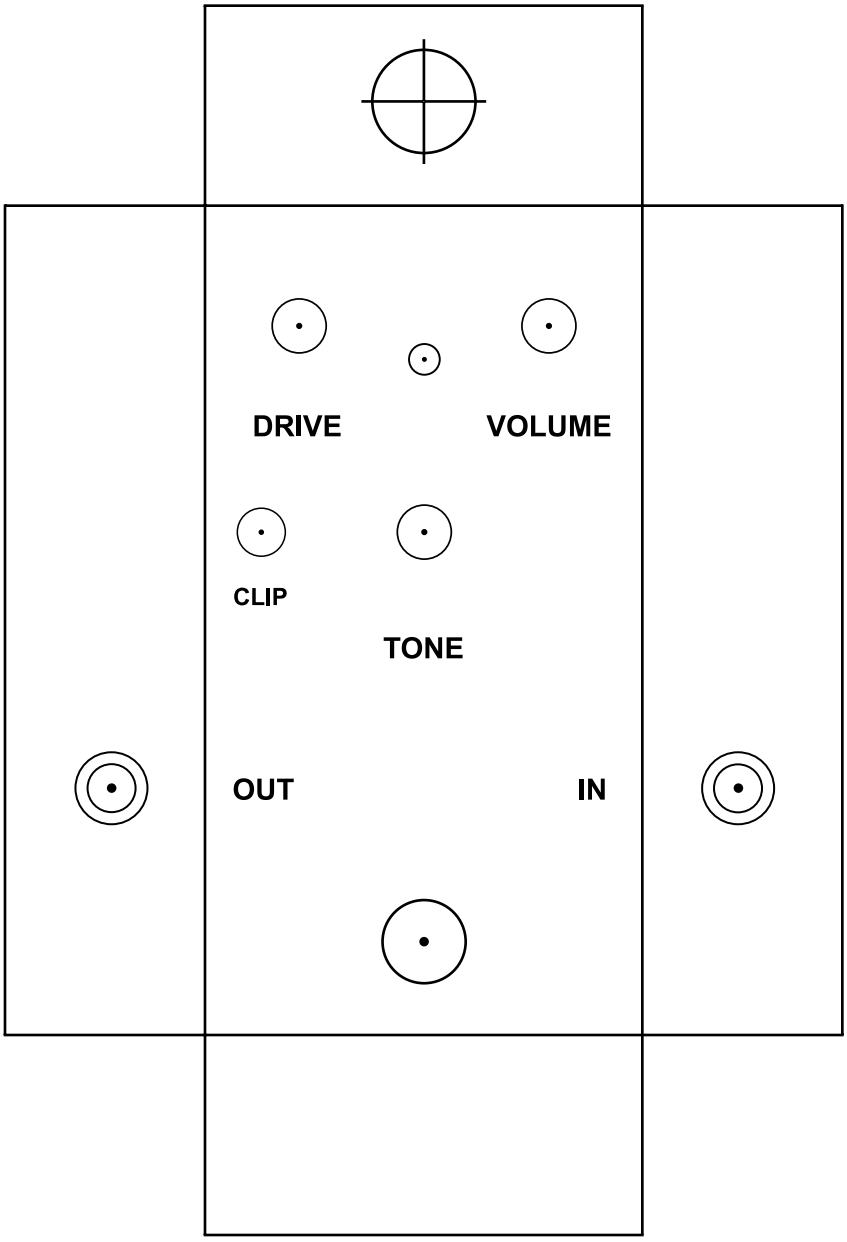
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

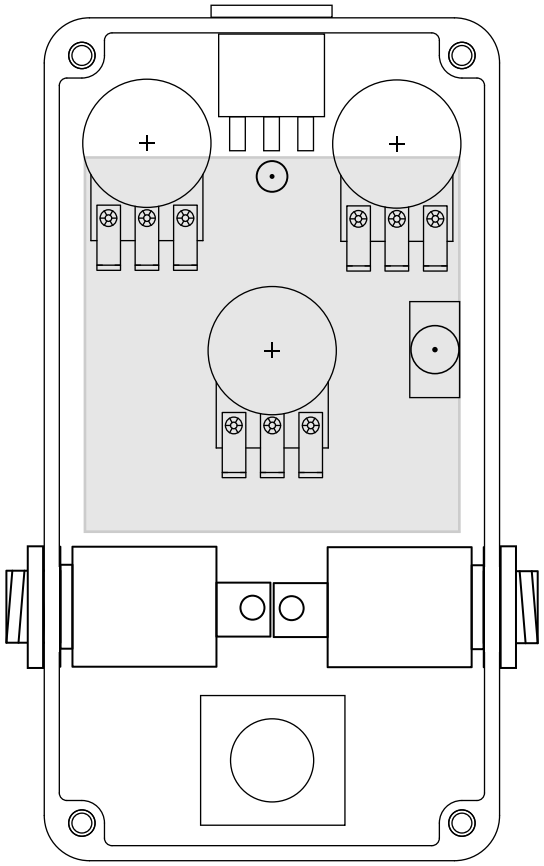
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

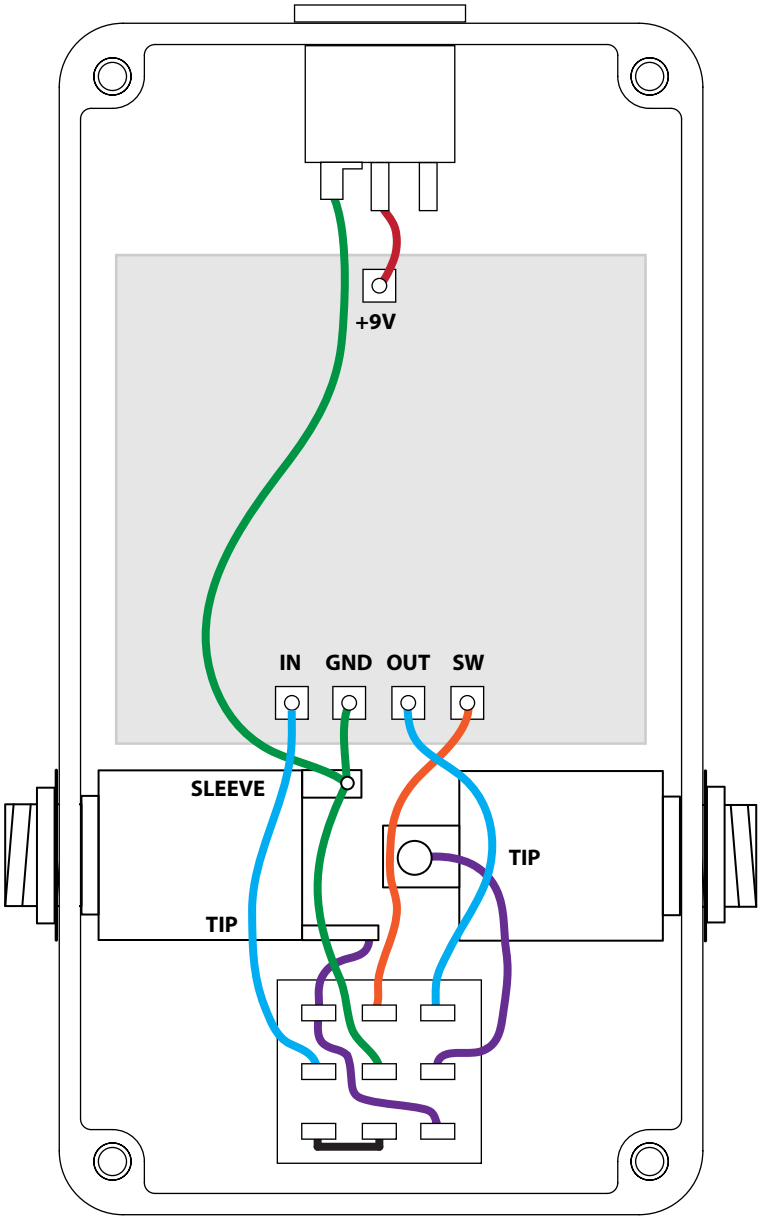
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)