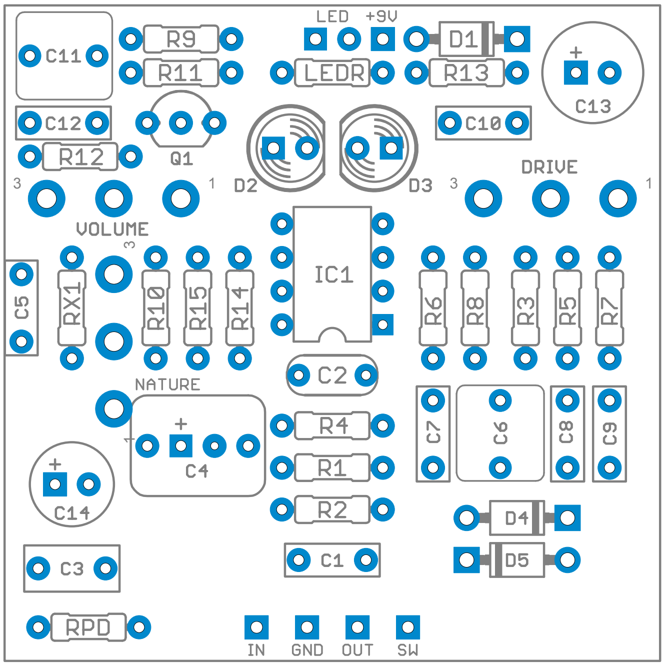
Procyon Overdrive

BJFE/Bearfoot Honey Bee Overdrive

Overview

Procyon Project Link



The Procyon Overdrive project is a clone of the Honey Bee Overdrive, a circuit first released in 2002 that is credited for starting the "low gain overdrive" trend. These pedals were handmade in Sweden and were extremely expensive ($300-400 USD), and so in 2011 an agreement was made with BearfootFX to produce them in the USA for a much more reasonable cost.

Bearfoot has also released a number of variants of this basic circuit such as the Model H, Model G, Honey Beest, Uber Bee, and Sparkling Yellow Overdrive. While each of these circuits are similar, the Procyon project only supports the standard Honey Bee (both BJFe and Bearfoot versions).

The circuit itself is a rather unique overdrive design, incorporating both soft and hard clipping and utilizing frequency-dependent negative feedback from different points in the circuit, for a result that is often called "amp-like" in tone.

Controls & Usage

The Honey Bee has the same control layout as most overdrive or distortion effects:

- **Drive** controls the gain of the op-amp stage which increases the amount of signal clipping.
- **Nature** is a very strange tone control, doing two things at once. On one half of the rotation it affects the amount of bass and changes the gain structure of the circuit. On the other half of the rotation, it blends in a frequency bypass capacitor on the gain-recovery FET, causing the highs to be emphasized more. The end result is that you have more bass on one side and more treble on the other, but the manner of accomplishing this is very different than something like the Big Muff.
- **Volume** controls the overall output level of the effect.

Modifications & Experimentation

The topology of this circuit has changed very little since it was first designed, and the standard overdrive mods such as the clipping or corner frequency switches don't work as well here due to the diode combination and the Nature control. So, this is not exactly a tweaker's paradise like the Tube Screamer or Big Muff.

That said, there are a few tweaks that can be made. The original Honey Bee has been described as boxy, dark, or thin. The Bearfoot version goes a long way toward correcting this by making the Nature control much more effective, but even building this version, you may still find that it's not enough. There's a lot of filtering that takes place right after D4 and D5, and a few tweaks here will have a big impact on the tone. Try dropping the value of **C9** to **15n** or **10n**, and raising the value of **C10** to **10n** or **22n**.

Parts

Resistors

R1	360k
R2	15k
R3	3k
R4	1k
R5	1k
R6	27k
R7	10k
R8	150k [1]
R9	1M
R10	2k2
R11	5k6
R12	47k
R13	51R
R14	31k6 [2]
R15	31k6 [2]
RX1	(jumper) [3]
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	47n
C2	100pF MLCC
C3	220n
C4	2u2 electro/film [4]
C5	4n7 [3]
C6	1uF film
C7	22n
C8	4n7
C9	22n [5]
C10	4n7 [5]
C11	1uF film
C12	4n7
C13	100uF electro
C14	22uF electro

Semiconductors

Q1	2N5952 [6]
IC1	CA3130
D1	1N4002
D2, D3	LED (diffused) [7]
D4, D5	1N4007 [8]
LED	5MM

Potentiometers

Drive	500kA
Nature	50kA
Volume	50kA [9]

[1] **Alternate value:** Some versions of the schematic use **47k** for R8. General consensus is that 150k is correct.

[2] **Substitution:** This is just a simple voltage divider, so **27k** or **33k** would work exactly the same.

[3] **The Bearfoot version** changes the Nature control so it's more useful. Use **220n** for C5 and **4k7** for RX1.

[4] **Capacitor type:** I made space for the huge ECQ-V film cap from Panasonic to fold flat against the board.

[5] **Experimentation:** To open up the tone a bit more, reduce C9 to **15n** or **10n** and/or increase C10 to **22n**. It's worth socketing these along with C5 and RX1 to see which combination you like best.

[6] **Substitution:** A **2N5457** should sound identical here, but note that the pinout is reversed from the silkscreen.

[7] **Clipping LEDs:** The original used green LEDs here, but I prefer red ones. Don't use the water-clear types.

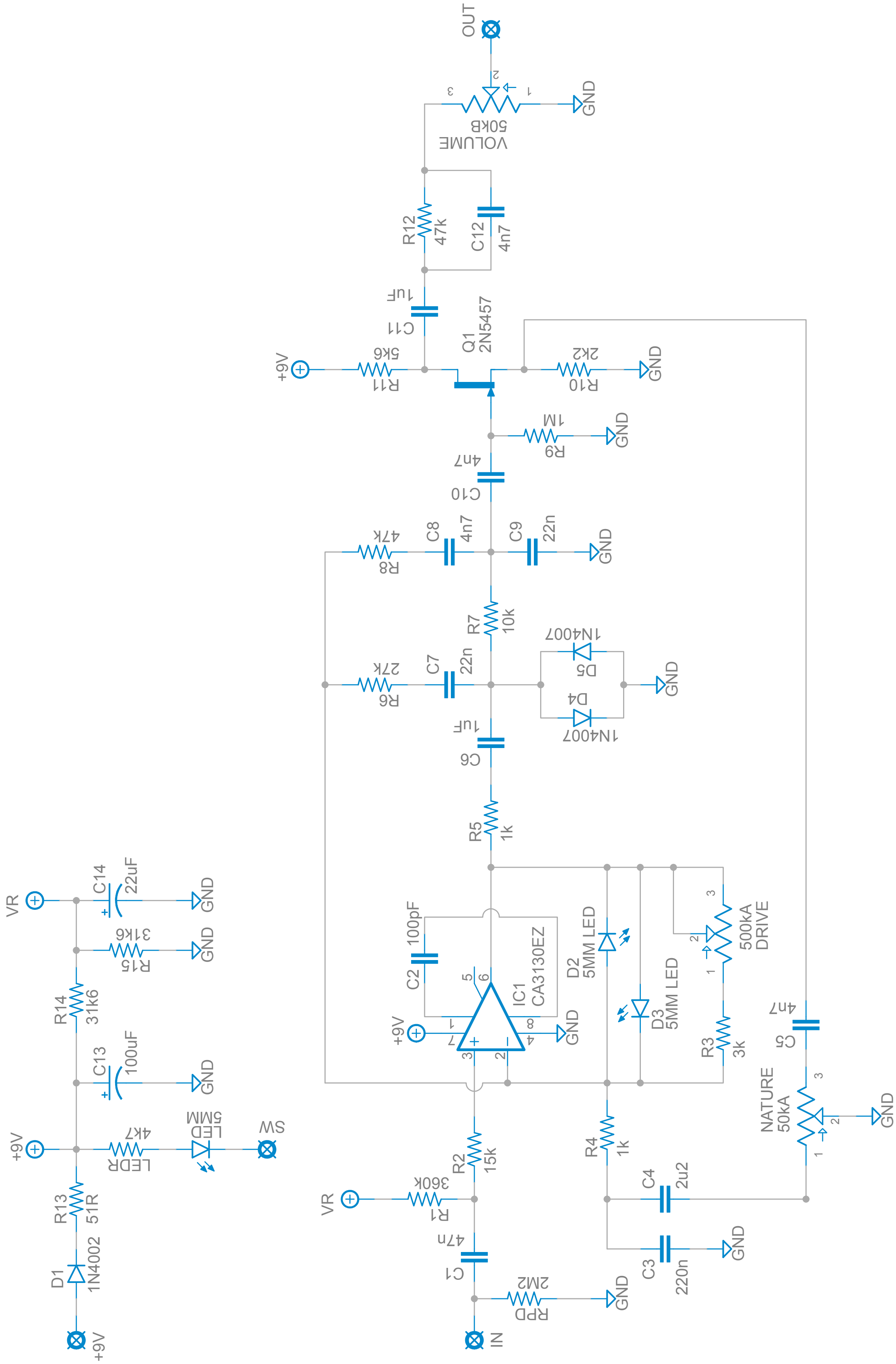
[8] **Clipping diodes:** The original uses 1N4007s here, but any "1N400X" will perform identically.

[9] **Volume pot value:** The original uses **50kB** (linear taper) but I recommend using a 50kA (audio taper).

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.

Schematic


OUT
VOLUME
50kB
GND
R12
47k
C12
4n7
C11
1uF
+9V
R11
5k6
Q1
2N5457
R10
2k2
R9
1M
C10
4n7
R8
47K
C8
4n7
C9
22n
R7
10k
D5
1N4007
D4
1N4007
R6
27K
C7
22n
C6
1uF
R5
1k
C2
100pF
IC1
CA3130EZ
D2
5MM LED
D3
5MM LED
500kA
DRIVE
R3
3k
C5
4n7
NATURE
50kA
R4
1k
C4
2u2
C3
220n
R2
15k
R1
360K
VR
C1
47n
RPD
2M2
IN
C14
22uF
R15
31k6
R14
31k6
C13
100uF
LEDR
4K7
LED
5MM
SW
R13
51R
D1
1N4002

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

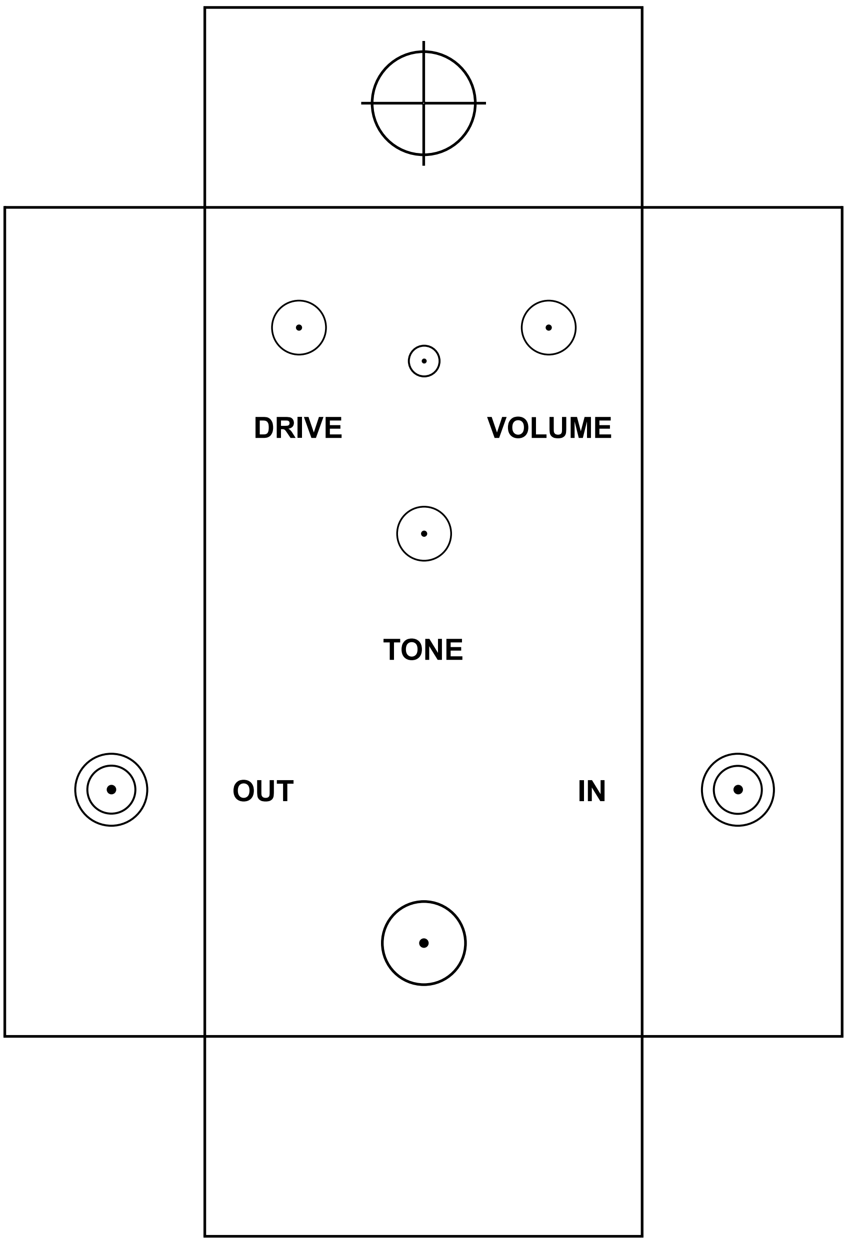
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

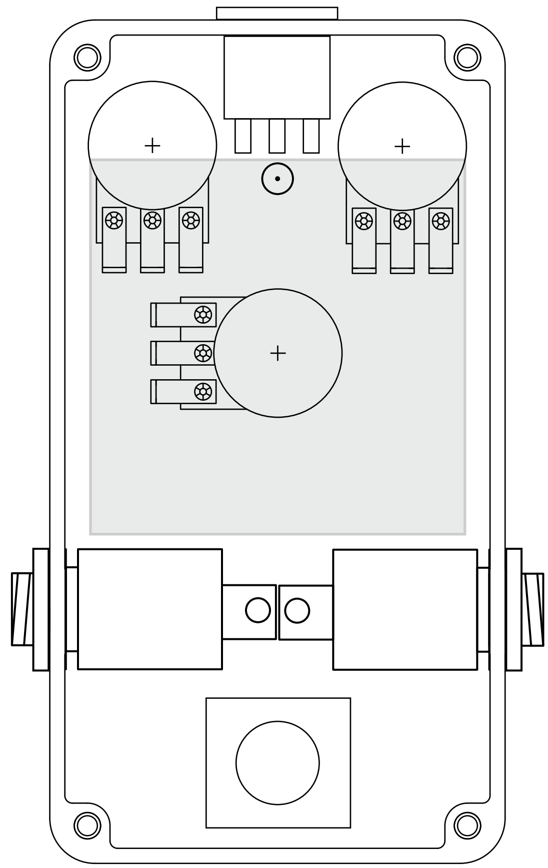
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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.



Hammond 1590B
(bottom/inside view)



Parts Used

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

Standard Wiring Diagram

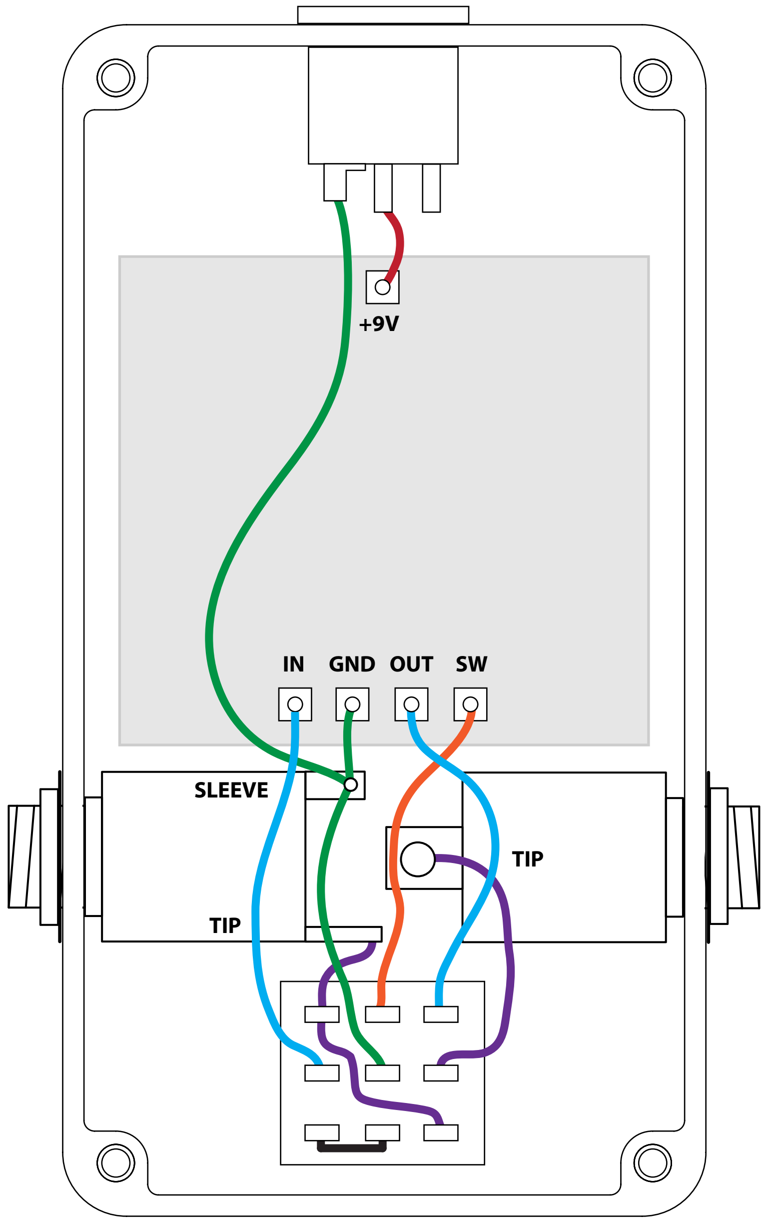
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)