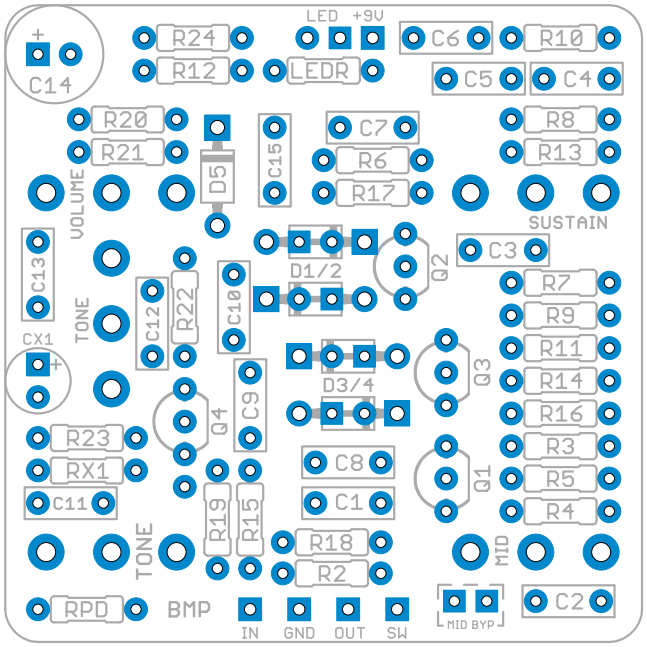
Halo Fuzz

Electro-Harmonix Big Muff Pi

Overview

Halo Project Link



The Halo Fuzz project is a DIY version of the Big Muff Pi, one of the most popular (and copied) fuzz pedals next to the Fuzz Face. There are plenty of DIY Big Muff projects already, but in order to justify its existence I've tried to make mine the most flexible one out there.

I researched a lot of different BMP variants and created a layout that can accommodate any of them, but the real fun is in coming up with your own version. This circuit is endlessly tweakable, so load up a PCB with sockets and start experimenting!

I have included parts lists for a few of the more well-known variants, but it's not exhaustive and there are many other types of BMPs that can be built on this board besides the ones listed.

Minor update, 10/2015: The pot positioning (and resulting drill template) has been revised slightly to make it consistent with most other Aion projects.

Controls & Usage

The Big Muff is a thick distortion/fuzz pedal with a ton of sustain. The controls are as follows:

- **Tone** controls allow for EQ adjustments
- **Mid / Presence** control (optional) sets the frequency of the tone control
- **Sustain** controls the amount of distortion
- **Volume** sets the overall output

Modifications & Experimentation

The AMZ Mid / Presence control has been included as an optional modification. To preserve "symmetry" of the control layout whether you use it or not, the Tone pot has two possible orientations: middle (for a 3-knob triangular layout) or to the side (for a 4-knob square layout with the Mid control on the other side). **To bypass the midrange knob**, jumper the pads marked **MID BYP** in the lower-right side of the PCB.

The diodes in both gain stages have extra pads in case you want to stack them. The middle two pads of each diode are connected, so you can fit two in a row in a stand-up configuration. If you only use one diode in each direction then just skip the two middle pads and solder it to the two outer pads as in the silkscreen.

Another option is the usage of a JFET gain recovery stage (as in the BJF Fire Red Fuzz) instead of the standard bipolar transistor gain stage. I don't know how big of a difference this makes in the overall sound, but it can be fun to play around with and can be added to any variant. To use this, include C101 and R101 and use a JFET such as 2N5485 in Q4. Since JFETs normally have different pinouts than BJTs, an extra pad has been provided to keep you from having to twist the legs. Assuming the pinout is D-S-G, just shift the whole thing down so it's still facing the same direction as the silkscreen but offset by one pad.

Parts

Variant: Triangle Big Muff

This is the first version of the Big Muff Pi, released in 1969.

Capacitors	
C1	100n
C2	(omit)
C3	100n
C4	100n
C5	560pF
C6	50n
C7	100n
C8	560pF
C9	50n
C10	4n
C11	10n
C12	100n
C13	100n
C14	100uF
C15	100n

Resistors	
R2	3k3
R3	82k
R4	390k
R5	820R
R6	22k
R7	1k
R8	8k2
R9	(omit)
R10	390k
R11	150R
R12	12k
R13	8k2
R14	82k
R15	390k
R16	820R
R17	22k

Resistors, cont.	
R18	39k
R19	39k
R20	390k
R21	100k
R22	12k
R23	2k7
R24	100R

Semiconductors	
Q1 - Q4	2N5088
D1 - D4	1N914

Variant: Violet Ram's Head

This is the second version of the Big Muff Pi, released in 1973.

Capacitors	
C1	100n
C2	470pF
C3	100n
C4	100n
C5	470pF
C6	100n
C7	100n
C8	470pF
C9	100n
C10	4n
C11	10n
C12	100n
C13	100n
C14	100uF
C15	100n

Resistors	
R2	39k
R3	100k
R4	470k
R5	100R
R6	15k
R7	1k
R8	8k2
R9	100k
R10	470k
R11	100R
R12	10k
R13	8k2
R14	100k
R15	470k
R16	100R
R17	15k

Resistors, cont.	
R18	39k
R19	39k
R20	390k
R21	100k
R22	10k
R23	2k7
R24	100R

Semiconductors	
Q1 - Q4	2N5088
D1 - D4	1N914

Variant: Red Army

This is the seventh major version of the Big Muff, released in 1991 after production had moved to Russia. For the "Green Russian" version, use 470pF or 500pF capacitors for C2, C5 and C8.

Capacitors	
C1	100n
C2	430pF
C3	100n
C4	100n
C5	430pF
C6	47n
C7	100n
C8	430pF
C9	47n
C10	3n9
C11	10n
C12	100n
C13	100n
C14	100uF
C15	100n

Resistors	
R2	39k
R3	100k
R4	470k
R5	390R
R6	12k
R7	1k
R8	10k
R9	100k
R10	470k
R11	390R
R12	12k
R13	10k
R14	100k
R15	470k
R16	390R
R17	12k

Resistors, cont.	
R18	22k
R19	20k
R20	470k
R21	100k
R22	10k
R23	2k7
R24	100R

Semiconductors	
Q1 - Q4	2N5088
D1 - D4	1N914

Variant: Creamy Dreamer

A boutique BMP from the late 1990s that claims to replicate the Smashing Pumpkins "Siamese Dream" tone. Part values taken from Kit Rae's Big Muff Pi site. Ignore the connection from Q3's emitter (Q2 on Kit Rae's schematic) - the 100R resistor that separates +9V from the filtering and polarity diode makes no difference.

Capacitors	
C1	1uF
C2	470pF
C3	47n
C4	1uF
C5	470pF
C6	1uF
C7	1uF
C8	470pF
C9	1uF
C10	4n7
C11	10n
C12	100n
C13	100n
C14	100uF
C15	100n

Resistors	
R2	39k
R3	100k
R4	470k
R5	(jumper)
R6	15k
R7	1k
R8	8k2
R9	100k
R10	470k
R11	(jumper)
R12	15k
R13	8k2
R14	100k
R15	470k
R16	(jumper)
R17	15k

Resistors, cont.	
R18	47k
R19	47k
R20	390k
R21	100k
R22	10k
R23	2k2
R24	(jumper)

Semiconductors	
Q1 - Q3	2N5089
Q4	2N5088
D1 - D4	1N914

Variant: Fire Red Fuzz

A boutique BMP clone from BJFE in Sweden. It's somewhat close to a Violet Ram's Head, but take note of the alternate JFET gain recovery stage.

Capacitors	
C1	1uF
C2	(omit)
C3	1uF
C4	100n
C5	470pF
C6	100n
C7	100n
C8	(omit)
C9	1uF
C10	3n9
C11	10n
C12	220n
C13	1uF
C14	100uF
C15	100n
C101	10uF

Resistors	
R2	33k
R3	100k
R4	470k
R5	100R
R6	15k
R7	470R
R8	8k2
R9	100k
R10	470k
R11	100R
R12	10k
R13	8k2
R14	100k
R15	470k
R16	150R
R17	15k

Resistors, cont.	
R18	22k
R19	22k
R20	(omit)
R21	68k
R22	14k7
R23	6k8
R24	100R
R101	2k

Semiconductors	
Q1 - Q3	BC550C
Q4	2N5485*
D1 - D2	2N3819**
D3 - D4	1N914

*** For the JFET gain recovery stage**, omit R20 and add C101 and R101. JFETs have different pinouts than bipolar transistors, so an extra pad has been provided to keep you from having to twist the legs around. Assuming your JFET's pinout is D-S-G (typical for 2N5485), keep the transistor facing the same direction as the silkscreen but shift it down by one pad. (The two outer pads are connected.)

**** For the JFETs in the first clipping stage**, tie together the source and drain. However, make sure to check the datasheet for your brand of JFET. 2N3819s tend to be D-G-S, different from a 2N5485, but sometimes different brands have different pinouts. For a D-G-S pinout you will need to tie together the two outer pins which takes some dexterity. Leave plenty of space around the middle pin so nothing shorts against it!

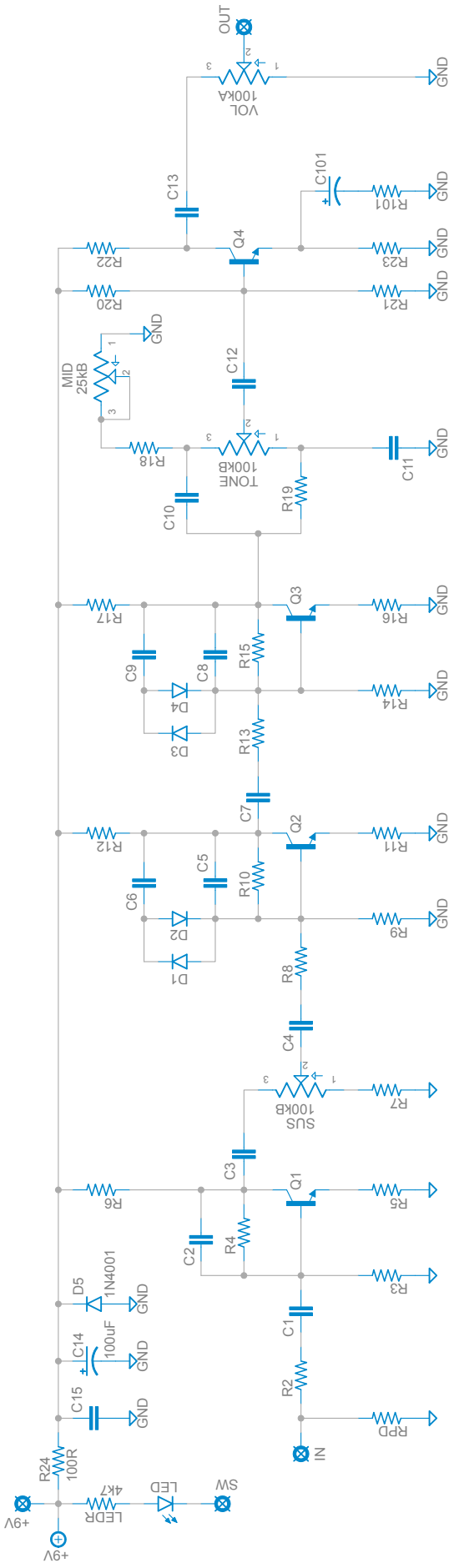
Additional Part Notes

- **For all variants listed above,** the **Sustain** and **Tone** pots are 100kB and **Volume** is 100kA.
- In some of the configurations, capacitors such as C4 and C7 are 1uF. I left space around these so you can use Panasonic ECQ-V film capacitors. Some others such as WIMA box film will fit as well.
- **Values above are assuming the midrange mod is not used**. If you do use this mod, you'll want R18 to be 2.2k or 3.3k and C10 to be 12n. This may not always sound exactly like the original since many of the variants get their distinctive sound from the tone section, so it's advisable to socket R18, R19 and C10 and experiment.
- 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

Besides the pots, no part values are shown on this schematic as every variant is vastly different.

See variant parts lists above for 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.

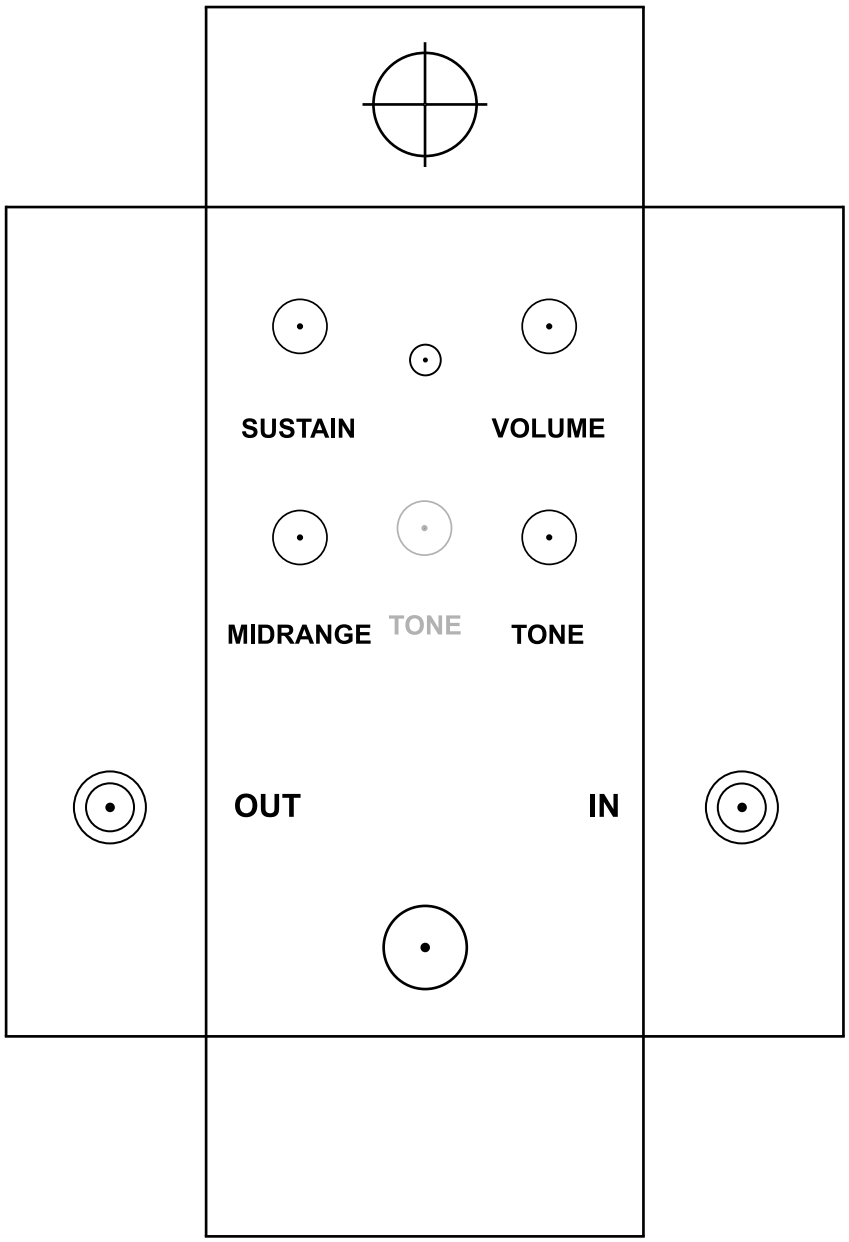
Sockets

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.

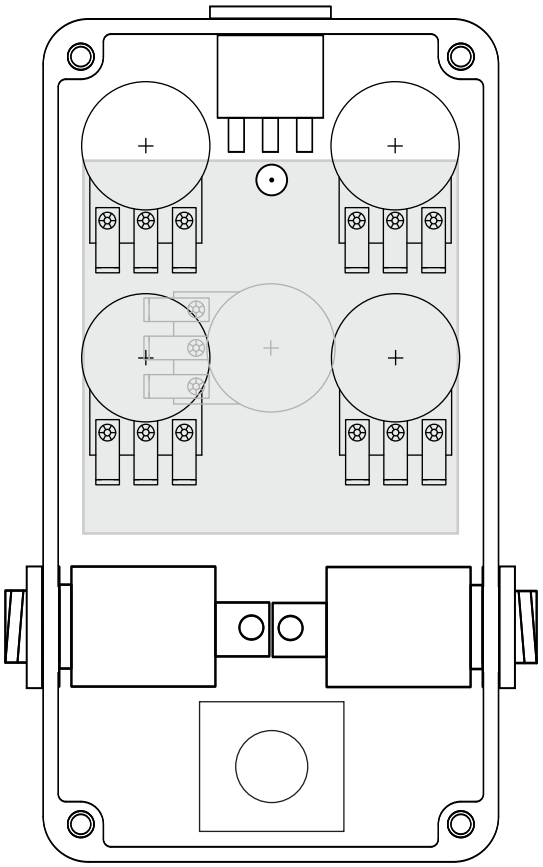
Drilling & Placement

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.

Note that the TONE pot has two possible orientations depending on whether you use the Midrange pot or not. The center or triangle-configuration orientation is shown in gray. If you're using the four-pot configuration, do not drill out the gray hole.



Hammond 1590B
(bottom/inside view)



Parts Used

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

Standard Wiring Diagram

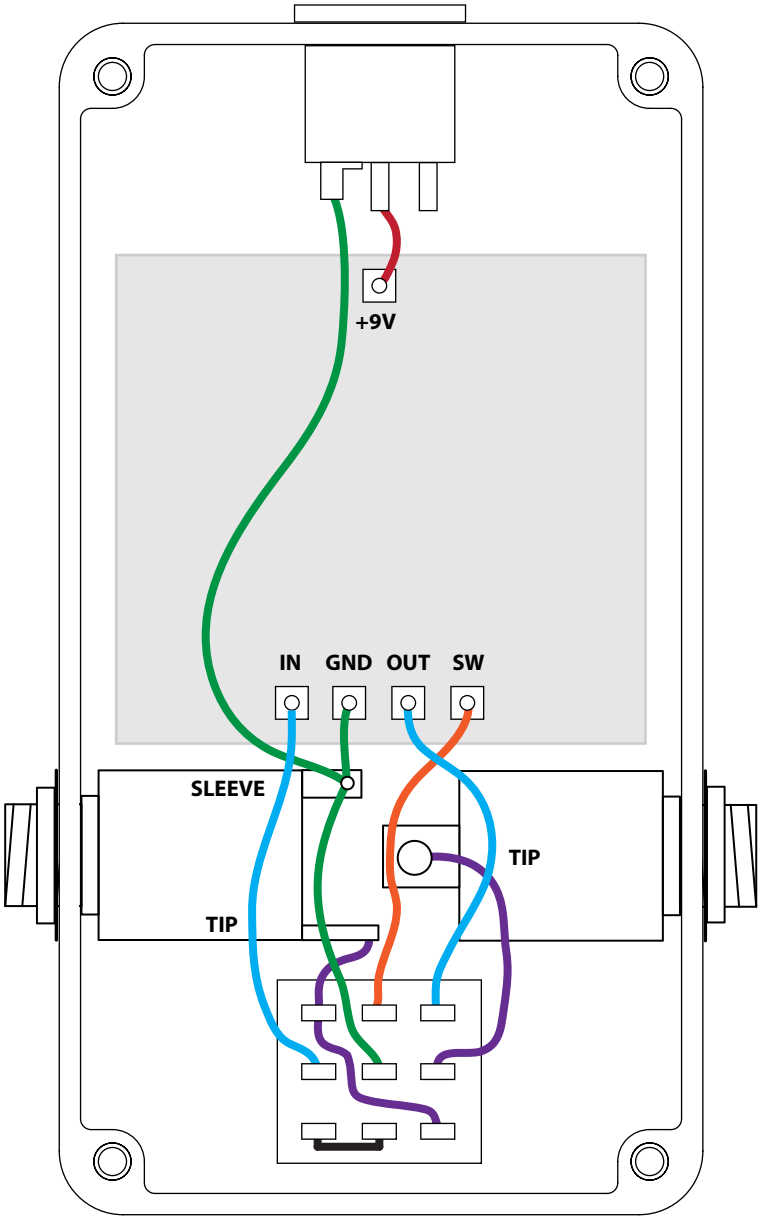
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)