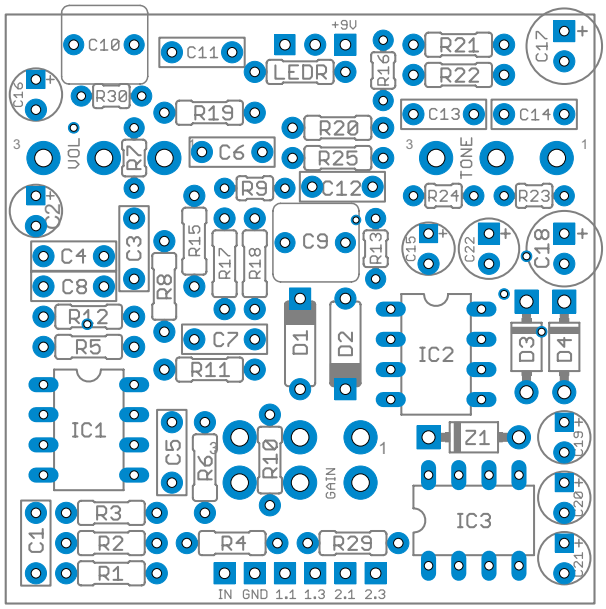
Refractor Overdrive

Klon Centaur Professional Overdrive

Overview



The Refractor is a part-for-part replica of the legendary Klon Centaur Professional Overdrive. The Centaur is a really unique overdrive pedal, an original design not based on a Tube Screamer or Big Muff like a lot of other “innovations” in the past 20 years. It is currently out of production.

Interestingly, the Centaur shares a couple things in common with the Ibanez OD-820 (available as my **Nimbus** PCB project) especially with the dual “gain/blend” control which to my knowledge is exclusive to these two pedals.

This project was designed to fit into a 1590B, so it’s a great deal smaller even than the KTR (the “miniaturized” reissue of the Centaur).

The Centaur’s switch wiring is pretty cumbersome. Check out the Refractor Bypass PCB for an easy solution.

Controls & Usage

Standard overdrive controls on this one.

- **Treble** control allows for EQ adjustment of the top end.
- **Output** is the overall effect volume.
- **Gain** simultaneously increases the gain and blends from clean signal to dirty using a dual pot.

Modifications & Experimentation

No built-in mods for this one, but there are a few part substitutions you can experiment with.

Tone mod: A very common and well-regarded modification is to change C14 to either **6n8** or **8n2**. It fixes a complaint of Centaur clones that the treble is a little thin, and nearly everyone who’s done it says it’s a huge improvement.

Socket the diodes. The original uses germanium diodes with a forward voltage of around 0.35v, but others have reported good sounds from **BAT41**, **OA126** or even the plain old **1N914**.

Socket the op amps as well. The original uses TL072s, but many people have reported good things from mixing these up, such as an **LF353** or **JRC4558D** for IC1 and a **NE5532** for IC2.

Bass guitar mod: Use a **390n** film capacitor for C1 if you’re going to use it as an overdrive for bass.

Parts

Resistors

R1	10k
R2	1M
R3	100k
R4	560R
R5	5k1
R6	10k
R7	1k5 *1/8w*
R8	1k5
R9	1k *1/8w*
R10	2k
R11	15k
R12	422k
R13	1k *1/8w*
R15	22k
R16	47k *1/8w*
R17	27k
R18	12k
R19	15k
R20	392k
R21	1k8
R22	100k
R23	4k7 *1/8w*
R24	100k *1/8w*
R25	560R
R26	68k *1/8w* [1]
R27	68k *1/8w* [1]
R28	100k [1]
R29	27k
R30	27k *1/8w*
LEDR	4k7

Capacitors

C1	100n
C2	4u7 electro
C3	100n
C4	68n
C5	68n
C6	390n
C7	82n
C8	390pF MLCC
C9	1uF film
C10	1uF film
C11	2n2
C12	27n
C13	820pF MLCC
C14	3n9
C15	4u7 electro
C16	1uF tantalum
C17	47uF electro
C18	47uF electro
C19	1uF electro
C20	1uF electro
C21	1uF electro
C22	1uF electro

Semiconductors

IC1	TL072
IC2	TL072
IC3	TC1044SCPA[2]
D1, D2	Ge[3]
D3, D4	1N4002
Z1	1N4742[2]
LED	5MM

Potentiometers

Treble	10kB
Output	10kB
Gain	100kB dual

See next page for footnotes and part notes.

Build Notes

Due to space constraints, a number of resistors are listed above as 1/8w. This is only because of the available space on the board—you can still use 1/4w resistors if you stand them up on end.

[1] These resistors are wired off-board and are not found on the PCB itself. See wiring diagram further down. Note that these three resistors are **not** shown in the schematic.

[2] Can use a MAX1044 as well, but I recommend the TC due to the higher maximum voltage. If using a MAX1044, use a **1N4739** for **Z1** to keep the voltage below the MAX1044's max voltage.

[3] The original uses germaniums of an unknown type. But germanium diodes have such wild ranges that even if the part number was known, it wouldn't necessarily mean that another of the same name would sound the same. What is known is that the Centaur's diodes have been measured at around 0.35v forward voltage. I've had pretty good luck getting this value with the 1N270 from Tayda Electronics, but you may want to order 5 or 10 and then measure them to find the ones that are closest to 0.35v.

Keith Vonderhulls from Build Your Own Clone did some extensive testing of different diodes and found that Russian **D9E** diodes were an exact sonic match. (D9B, D9D, D9J, D9K and D9L are equivalents and can usually be found cheaper.) Many people have tried this diode since then and agreed that it's very, very good. It's unlikely that the real Centaur unit uses this exact diode, but it's the closest that the DIY scene has gotten.

However, a lot of people have used other types of diodes and found that they preferred them over standard germaniums. One good one that is often tried is the **BAT41**. Others have reported really good results with the **OA1160**, which are obscure but available from pedalhackerelectronics.com.

Silver Pony specs

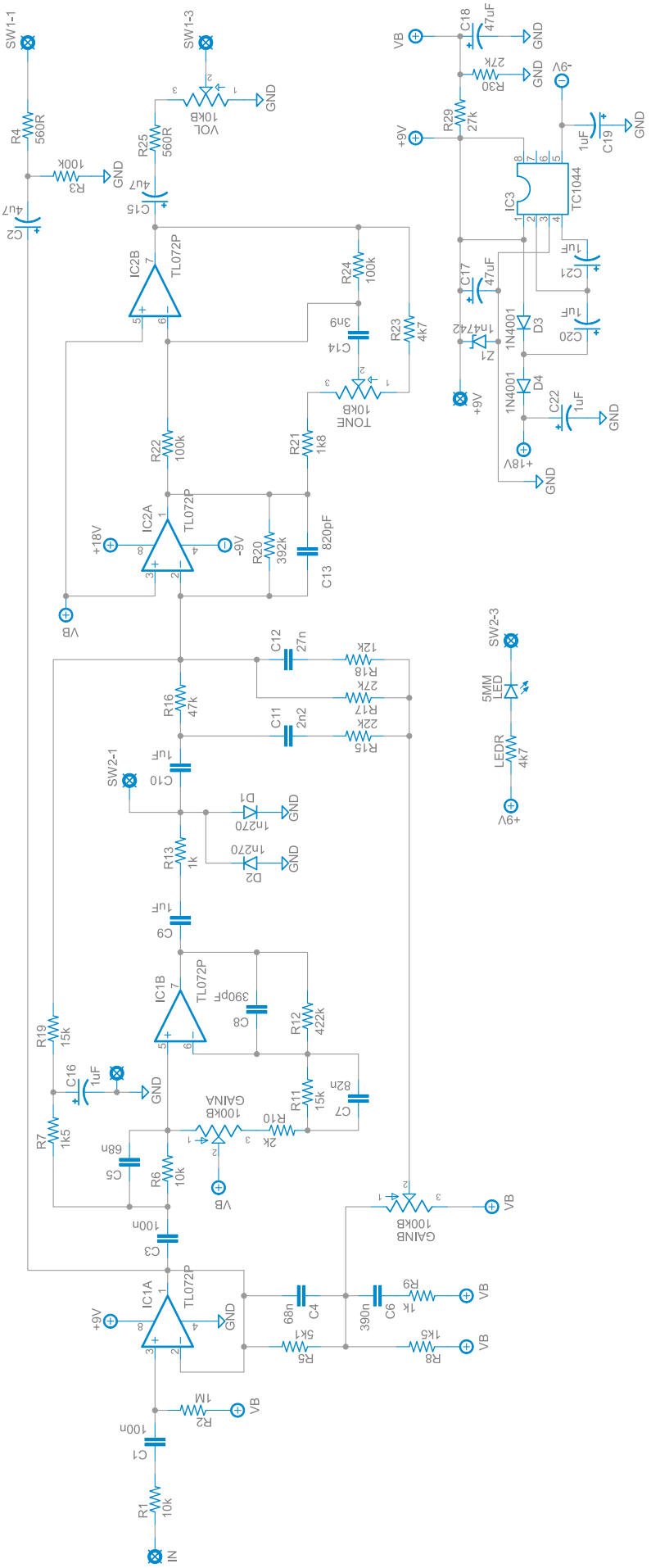
In July 2014, Build Your Own Clone reverse-engineered a silver Centaur, serial S2207, and found a number of discrepancies. It was later determined that this was a unique unit and not representative of a normal Centaur, but it represents a curiosity that some may be interested in building. The "Silver Pony" changes are as follows:

Part	Original	Silver Pony	Effect
R10	2k	47R	More gain
R17	27k	10k	Increased volume & treble for the dry signal
R18	12k	4k7	(along with R17 change) Better wet/dry ratio
R21, R23	1k8 / 4k7	4k7 / 1k8	Slight change in range of Tone knob
C13	820pF	560pF	Slight increase in overall brightness

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.

Schematic


SW1-1
SW1-3
VOL
10KB
GND
R4
560R
R25
560R
R3
100K
C2
4u7
C15
4u7
IC2B
TL072P
R24
100k
3n9
C14
R23
4k7
TONE
10KB
R22
100k
R21
1k8
IC2A
TL072P
+18V
-9V
R20
392k
C13
820pF
VB
C12
27n
R18
12k
R17
27k
R16
47k
C11
2n2
R15
22k
SW2-1
C10
1uF
D1
1n270
GND
R13
1k
D2
1n270
C9
1uF
IC1B
TL072P
390pF
C8
R12
422k
R19
15k
C16
1uF
R11
15k
82n
C7
GAINA
100KB
R10
2k
R7
1k5
68n
C5
R6
10k
100n
C3
GAINB
100KB
IC1A
TL072P
+9V
GND
68n
C4
5k1
R5
390n
C6
R9
1k
R8
1k5
1M
R2
100n
C1
R1
10k
IN
C18
47uF
R30
27k
-9V
R29
27k
1uF
C19
IC3
TC1044
C17
47uF
1uF
C21
1uF
C20
1N4001
D3
1n4742
Z1
1N4001
D4
C22
1uF
+18V
SW2-3
5MM
LED
LEDR
4k7
+9V

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

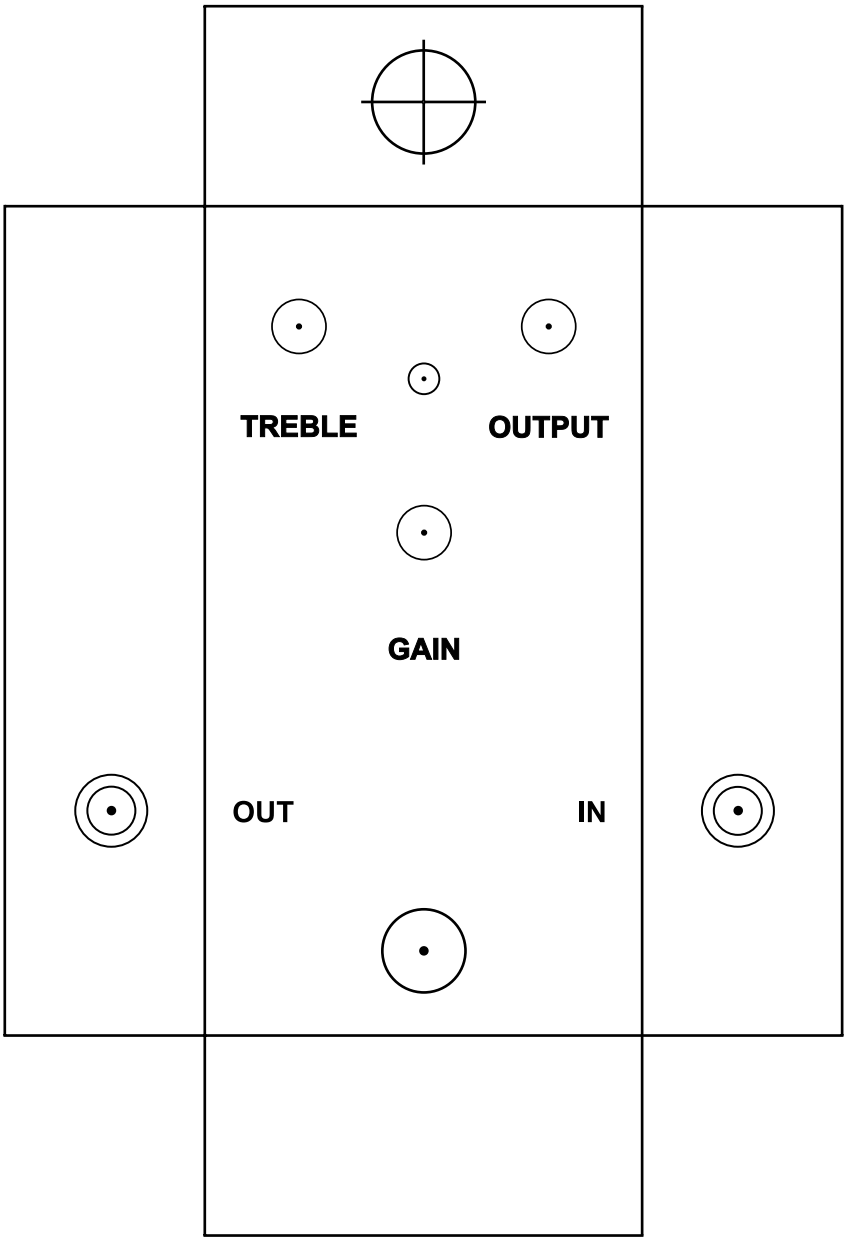
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

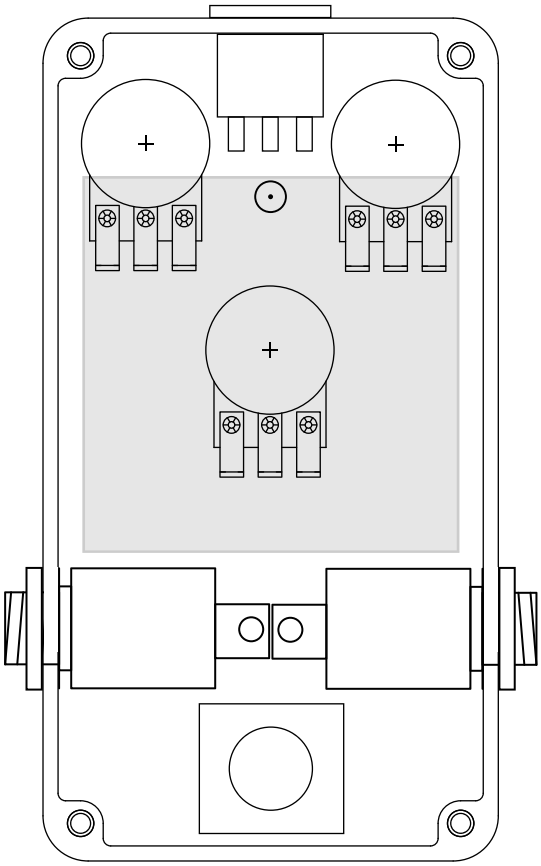
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

Additional Notes & Wiring Diagram

The wiring for this project is very non-standard since the circuit uses buffered bypass and not true bypass. Though it is most likely possible to make this circuit true bypass, that version is not supported or covered in this build document. The original circuit was designed around the buffer, so I'd recommend trying that first and only attempt a true bypass version if you absolutely need to.

The wiring diagram 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.

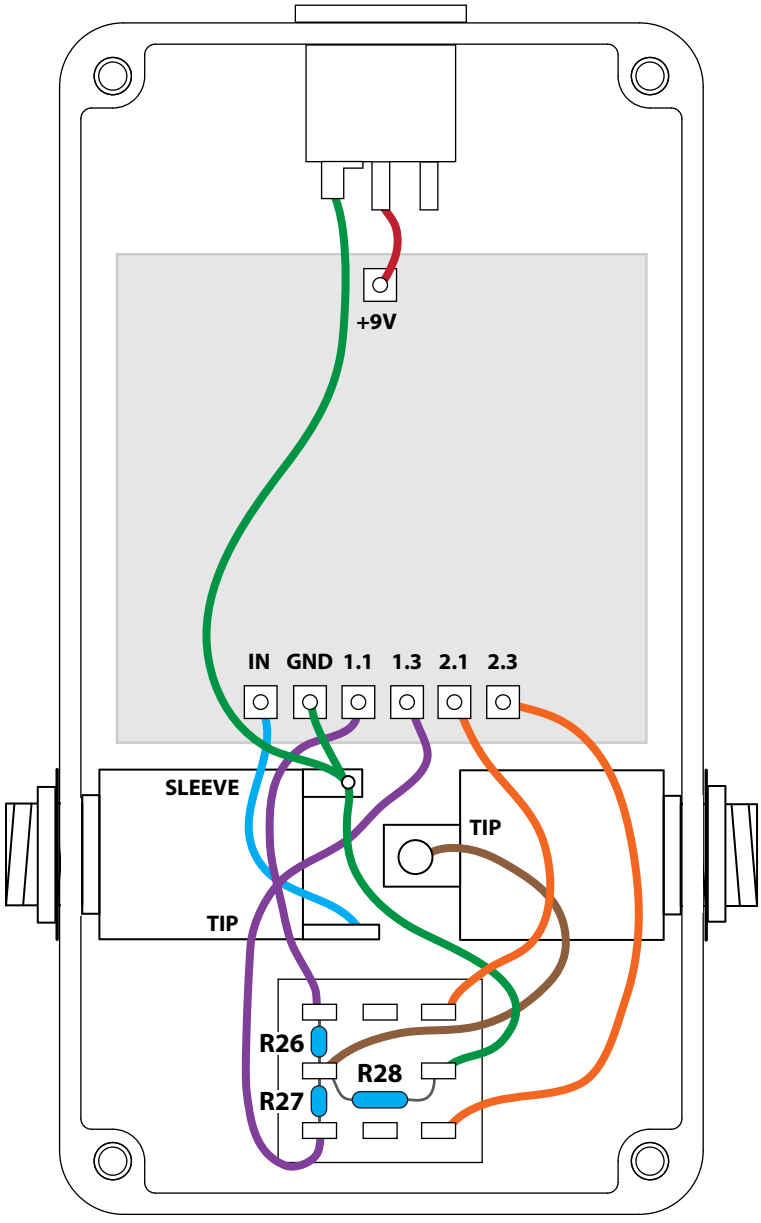
If using a painted or powdercoated enclosure, **make sure both jacks have solid contact with bare aluminum** for grounding purposes. You may need to sand off some of the paint or powdercoat on the inside in order to make this happen.

I've tried to make the diagram big enough that it's easy to read, but if you're having trouble then try zooming the PDF document. The diagram is in vector format so it will scale with the zoom.

Note the positions of **R26** and **R27**, the 68k resistors. These will get pretty tight, especially the middle lug which has four leads crammed into one eyelet. I've found that it helps to make a loop with one of the resistor leads in the center lug and use it for the wire. I strongly recommend using 1/8W resistors here instead of 1/4W—they're much easier to fit!

R28 is a 100k resistor from output to ground. You do not need to put it here, and in fact it may be easier to run it from the output jack's lug to the star ground on the input jack. As long as it's connected to ground you'll be fine.

A 3PDT footswitch is shown below with only the outer two poles being used. You can use a DPDT here as well—depending on the type of switch, it may even allow a little more room for the resistors. My only advice is to avoid the Alpha "X-wing" style switches, as those are reported to have reliability issues.



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.

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 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!)