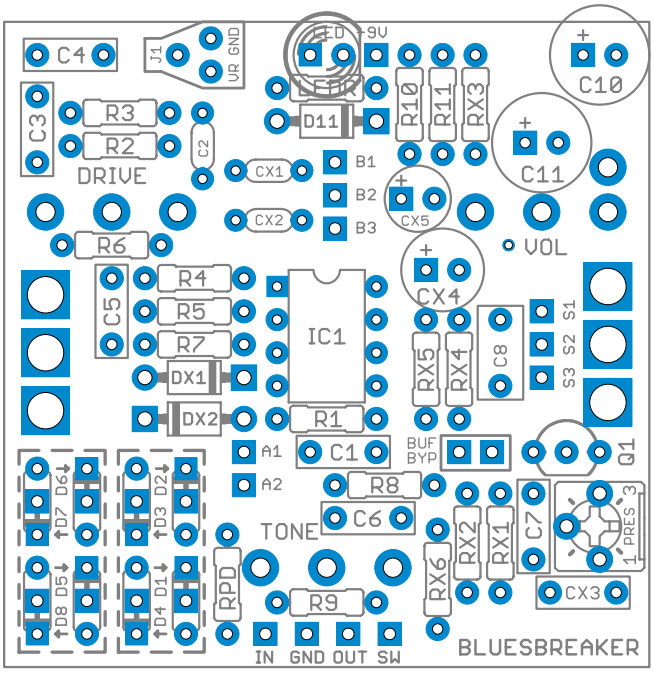
Cerulean Overdrive

Marshall Bluesbreaker

Overview



The Cerulean Overdrive project is a clone of the Marshall Bluesbreaker. The Bluesbreaker itself never achieved legendary status like the Tubescreamer, but a number of well-known boutique pedals such as the King of Tone and Morning Glory are based off of it. With this project, you can build an original Bluesbreaker or any of the modded versions.

There are two switches available: one for clipping diodes (with enough pads available for any diode combination you can dream of, as well as “comp cut” or diode lift in the middle), and a second one that can be used for one of two other mods using jumper wires. More details below.

The Bluesbreaker is similar to the Tube Screamer in that it uses feedback loop diode clipping. As a result, you’ll notice that some parts of the documentation are identical between the two projects, such as the diode-clipping mods and the drilling template.

Controls & Usage

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

- **Tone** controls the treble response of the effect.
- **Drive** controls the amount of gain in the op amp feedback diode clipping stage.
- **Volume** controls the overall output.
- An internal **Presence** control allows you to tweak the overall response of the tone knob.

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. See further down for detailed diagrams on a few different diode configurations.

The second switch is a peculiarity. I didn’t want to pick just one mod, so I made it so you can choose from two. By running jumper wires from different places on the board to the pads by the switch, you can “assign” a mod.

- The **Bright Cut** mod takes out a little bit of sparkle in the feedback/clipping section. This comes before the tone stack and provides a different effect than just turning down the tone control.
- The **Hard Clipping** mod is taken from the King of Tone and allows you to add hard clipping after the standard diode clipping section. In combination with the other diode switch, you’re able to select feedback clipping, hard clipping, both, or neither.

Parts (Stock Circuit)

Resistors

R1	1M
R2	3k3 [1]
R3	4k7 [1]
R4	10k
R5	220k
R6	6k8
R7	1k
R8	6k8
R9	1M
R10	47k
R11	47k
RX1	(omit)
RX2	(omit)
RX3	(omit)
RX4	(omit)
RX5	(omit)
RX6	(omit)
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	10n
C2	47pF
C3	10n
C4	10n
C5	220n
C6	10n
C7	10n
C8	100n
C10	100uF
C11	100uF
CX1	100pF [2]
CX2	470pF [2]
CX3	(omit)
CX4	(omit)
CX5	(omit)

Semiconductors

Q1	(omit)
IC1	TL072
D11	1N4002
D1-4	1N914 [3]
D5-8	[3]
DX1-2	[2]
LED	5MM

Potentiometers

Tone	25kB
Drive	100kB
Volume	100kA
Presence	50k trim (3362P)

Other

SW1 - SW2	SPDT center off
BUF BYP	Jumper [4]
J1	Jumper to VR [4]

[1] The very first version actually uses **27k** for R2 and **33k** for R3. This gives the unit lower drive and changes the EQ a bit, which some people prefer to the later versions. These values are used in the King of Tone.

[2] **Part of a switch mod.** All the parts with an “X” in the number (e.g. CX2) are modifications. See next page.

[3] **Your choice.** The stock Bluesbreaker circuit is shown here in D1-4, but you can do what you want with the other side. See the page further down for a variety of ideas. I recommend using sockets so you can experiment.

[4] **Important**: Don’t forget these jumpers or the effect won’t work at all.

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.

Modifications & Experimentation

Bright Cut Mod

Allows you to switch a capacitor value to change the response of the tone stack. (This is taken from the Morning Glory, but the feedback capacitor is a common part of many other circuits including the Tube Screamer that was just omitted in the original Bluesbreaker.) Include **CX1** and **CX2** and connect **B1 to S1**, **B2 to S2** and **B3 to S3**. The value given for CX1 is arbitrary (the Morning Glory only uses CX2)—the idea is just that it's somewhat less than CX2 to provide a less drastic effect. I chose 100pF because it's the stock Tube Screamer value, but you could also do 220pF which is approximately half the value of the Morning Glory. Higher values provide more treble cut, lower values provide less.

Hard Clipping Mod

Taken from the King of Tone, this adds a second diode clipping stage right after the first. The switches are configured in such a way that with this mod, you can select the standard feedback clipping, hard clipping, both, or neither. If you use this mod, you'll want to choose diodes that have a lower clipping threshold than the ones you used for the feedback stage, otherwise they won't have any effect—they can't clip a signal that's already been clipped below their threshold. (For instance, you probably don't want to use LEDs in this section since they have a much higher threshold than silicon or germanium.)

Anyway, for this mod, include **DX1** and **DX2** and connect **A1** to **S1** and **A2** to **S2**.

Output Gain Stage Mod

The Morning Glory adds a JFET gain stage to the end of the Bluesbreaker, which increases the available volume and changes the output impedance. To use this, you'll want to include **RX1** through **RX6**, **CX3** and **CX4**, and **Q1**, and omit the F1-F2 jumper.

When the gain stage mod is used, **leg 1 of the volume control needs to be tied to ground** instead of VR. A second pad has been provided just to the top of the standard pad. If using right-angle pots, you'll need to bend the leg backward so it fits in the top pad instead of the regular one.

Presence Control

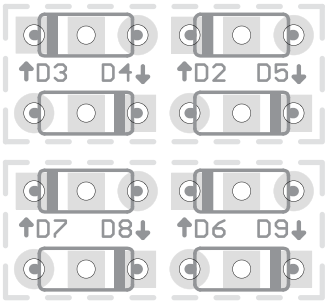
This internal trimmer is taken from the King of Tone. It's a modification to the original circuit, but there's no reason to leave it off no matter what version you're building since by turning the trimmer all the way down (zero resistance) you're left with the original circuit. By turning this up, you'll brighten up the tone in a different way than the Tone control.

High Gain Mod

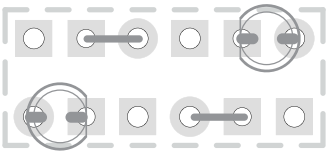
To increase the available gain, you can change the Gain pot to 250kB (from 100kB). The full stock range is still available on the first half of the rotation, but you now have increased range on the second half.

Feedback Diode Clipping Options

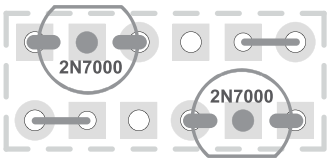
Most of the following diode configurations are taken from Tube Screamer clones. Since the Bluesbreaker uses the same style of clipping, if it works for a Tube Screamer then it's worth a try on a Bluesbreaker. Just remember that the Bluesbreaker's stock circuit has a high clipping threshold, so it may sound best with the stacked-diode configurations below such as the Zendrive and Fulldrive, or with LEDs.



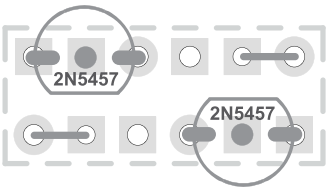
Close-up of silkscreen. (Square pad pairs are always connected.)



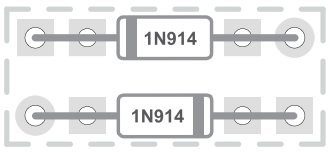
LED clipping



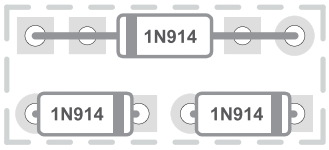
MOSFET clipping



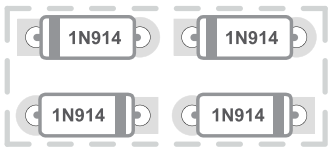
JFET clipping



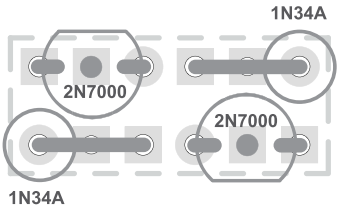
Stock TS-9 (symmetrical clipping)



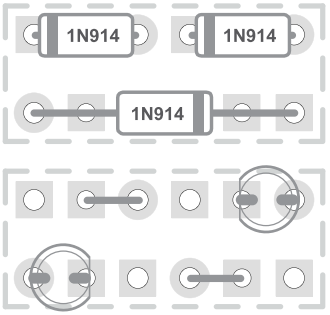
Boss SD-1 (asymmetrical clipping)



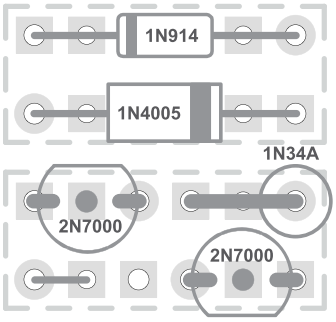
Lovepedal Eternity, Marshall Bluesbreaker



Zendrive

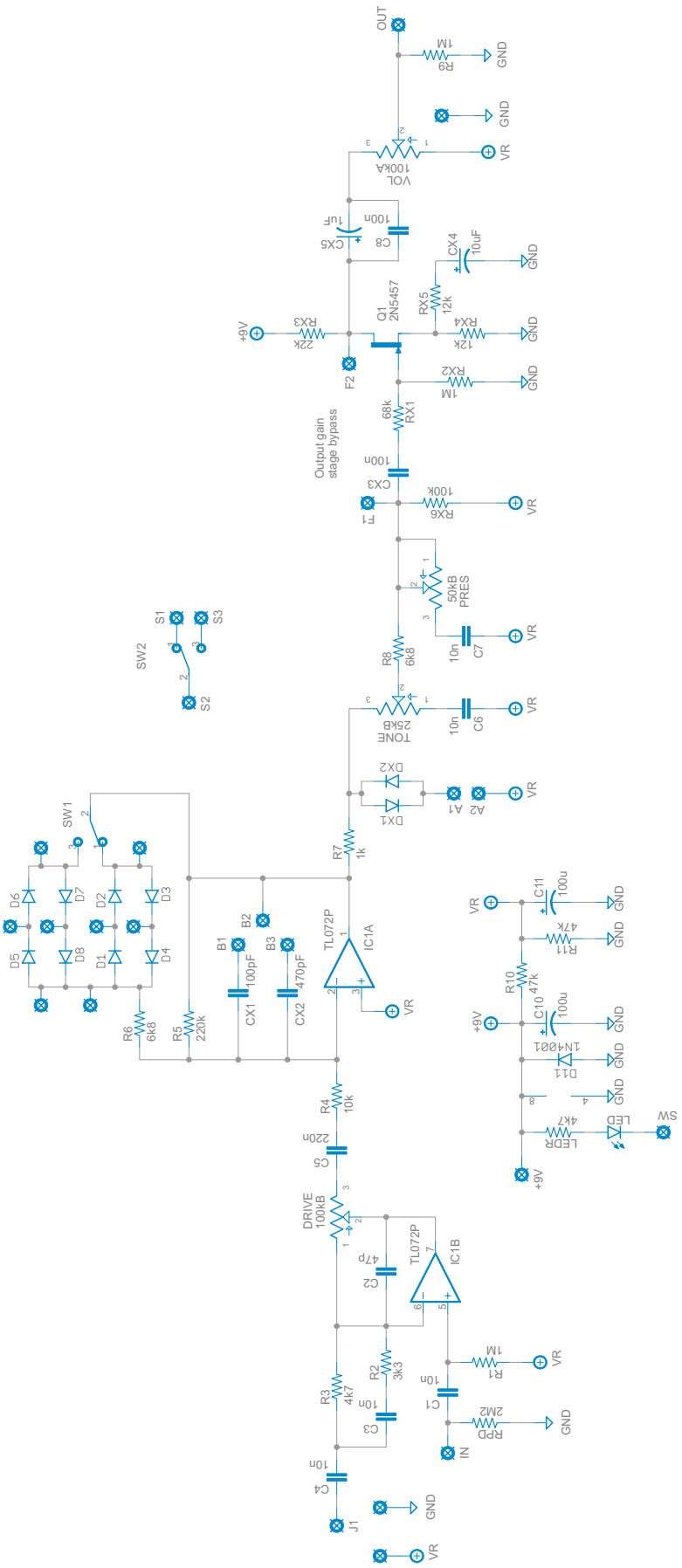


Landgraff Dynamic Overdrive (asymmetrical silicon on top, LED on bottom)



Fulldrive 2 (Silicon top, MOSFET bottom)

Schematic (Stock Circuit Values)



Variations

King of Tone, single-channel version

Resistors	
R1	1M
R2	33k
R3	27k
R4	10k
R5	220k
R6	6k8
R7	1k
R8	6k8
R9	1M
R10	47k
R11	47k
RX1	(omit)
RX2	(omit)
RX3	(omit)
RX4	(omit)
RX5	(omit)
RX6	(omit)
RPD	1M to 2M2
LEDR	4k7

Capacitors	
C1	10n
C2	100pF
C3	10n
C4	10n
C5	100n
C6	10n
C7	10n
C8	1uF film [5]
C10	100uF
C11	100uF
CX1	(omit)
CX2	(omit)
CX3	(omit)
CX4	(omit)
CX5	1uF electro [5]

Semiconductors	
Q1	(omit)
IC1	JRC4580D
D11	1N4002
D1-4	MA856 [1]
D5-8	[2]
DX1-2	1S1588 [3]
LED	5MM

Potentiometers	
Tone	25kB
Drive	100kB
Volume	100kA
Presence	50k trim (3362P)

Other	
SW1 - SW2	SPDT center off
BUF BYP	Jumper [4]
J1	Jumper to GND [4]

Build Notes

The original King of Tone uses two Bluesbreaker circuits in one box and allows them to be stacked. You can build two of these and make an exact two-channel clone, but that is outside the scope of this documentation.

[1] The original uses **MA856 and 1S1588** diodes, but these are very difficult to find. Try a BA278, BA242 or BA282 instead of the MA856, and 1N4001/2/4 for the 1S1588. These diodes are very obscure but not by any means unique, so you will not be missing out on any “mojo” by using substitutes.

[2] **Your choice.** The original doesn’t have a second set of diodes, but you might as well put something else here to have it as an option. If nothing else, build it with three diodes instead of four (see Boss SD-1 on pg. 4) so it’s asymmetrical.

[3] To be exactly like the original, you’ll want to use the Mod switch for the **Hard Clipping** mod: include **DX1** & **DX2** and connect **A1 to S1** and **A2 to S2**.

[4] Don’t forget these jumpers or it won’t work. Note that the King of Tone jumpers J1 to GND, while the stock Bluesbreaker and the Morning Glory both jumper to VR.

[5] The KoT uses parallel film (**C8**) and electrolytic (**CX5**) capacitors in the output to get 2uF. The space on the board is a little tight for the 1uF film capacitor, but you can probably fit it on the underside of the board. Otherwise, keep the leads longer and fold it over to one side. I’d recommend fitting it last just to make sure there’s room for everything else.

Morning Glory

Resistors

R1	1M
R2	3k3
R3	4k7
R4	10k
R5	220k
R6	6k8
R7	1k
R8	6k8
R9	1M
R10	47k
R11	47k
RX1	68k
RX2	1M
RX3	22k
RX4	12k
RX5	12k
RX6	100k
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	47n
C2	47pF
C3	10n
C4	10n
C5	100n
C6	10n
C7	10n
C8	(omit)
C10	100uF
C11	100uF
CX1	100pF [1]
CX2	470pF [1]
CX3	100n
CX4	10uF electro
CX5	2.2uF electro

Semiconductors

Q1	2N5457
IC1	LM833N
D11	1N4002
D1-4	1N4148
D5-8	[2]
DX1-2	(omit)
LED	5MM

Potentiometers

Tone	25kB
Drive	100kB
Volume	100kA [3]
Presence	50k trim (3362P)

Other

SW1 - SW2	SPDT center off
BUF BYP	(omit jumper)
J1	Jumper to VR [4]

Build Notes

[1] To be exactly like the original, you'll want to use the Mod switch for the **Bright Cut** mod: include **CX1** & **CX2** and connect **B1 to S1** and **B2 to S2**. Note that the value of CX1 is arbitrary—the Morning Glory only uses CX2 and does not give a second choice. I'd recommend either 100pF (the Tube Screamer value) or 220pF (half of the Morning Glory's stock value).

[2] **Your choice.** The original doesn't have a second set of diodes, but you might as well put something else here to have it as an option. If nothing else, build it with three diodes instead of four (see Boss SD-1 on pg. 4) so it's asymmetrical.

[3] **Important:** When the output gain stage is used, leg 1 of the volume control needs to be tied to ground instead of VREF. A second pad has been provided just to the top of the standard pad. If using right-angle pots, you'll need to bend the leg back so it fits in the top pad instead of the regular one.

[4] Don't forget this jumper or it won't work!

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 from the top.

[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. If there are two switches on the board, alternate between them one lug at a time.

"RPD" and "LEDR" resistors

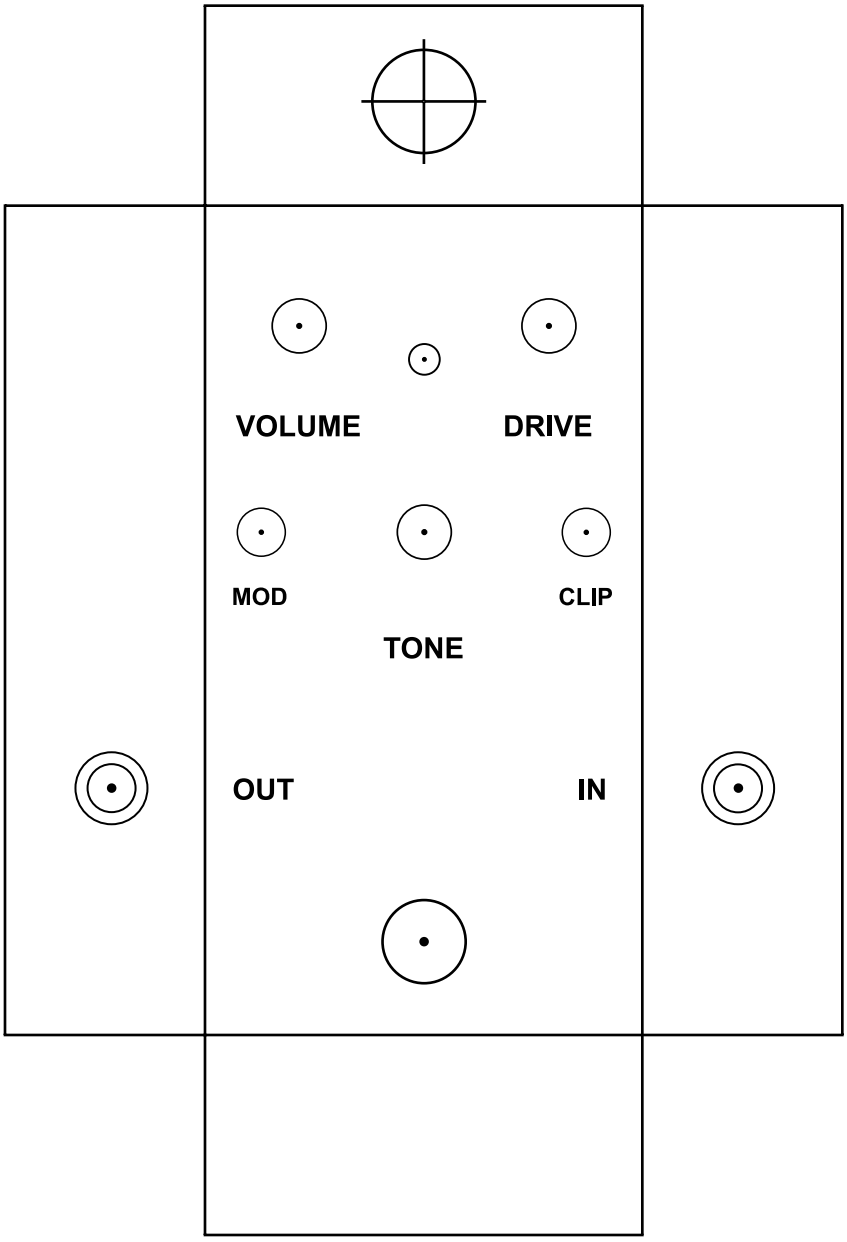
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

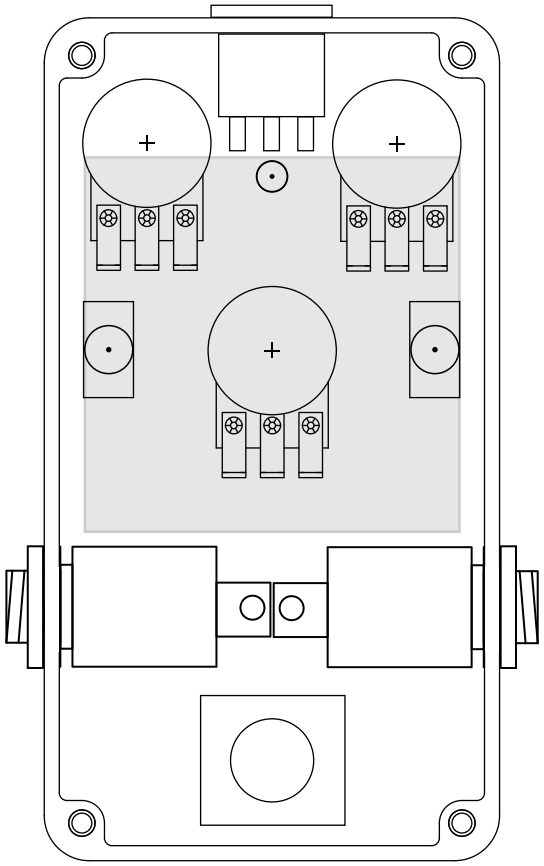
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

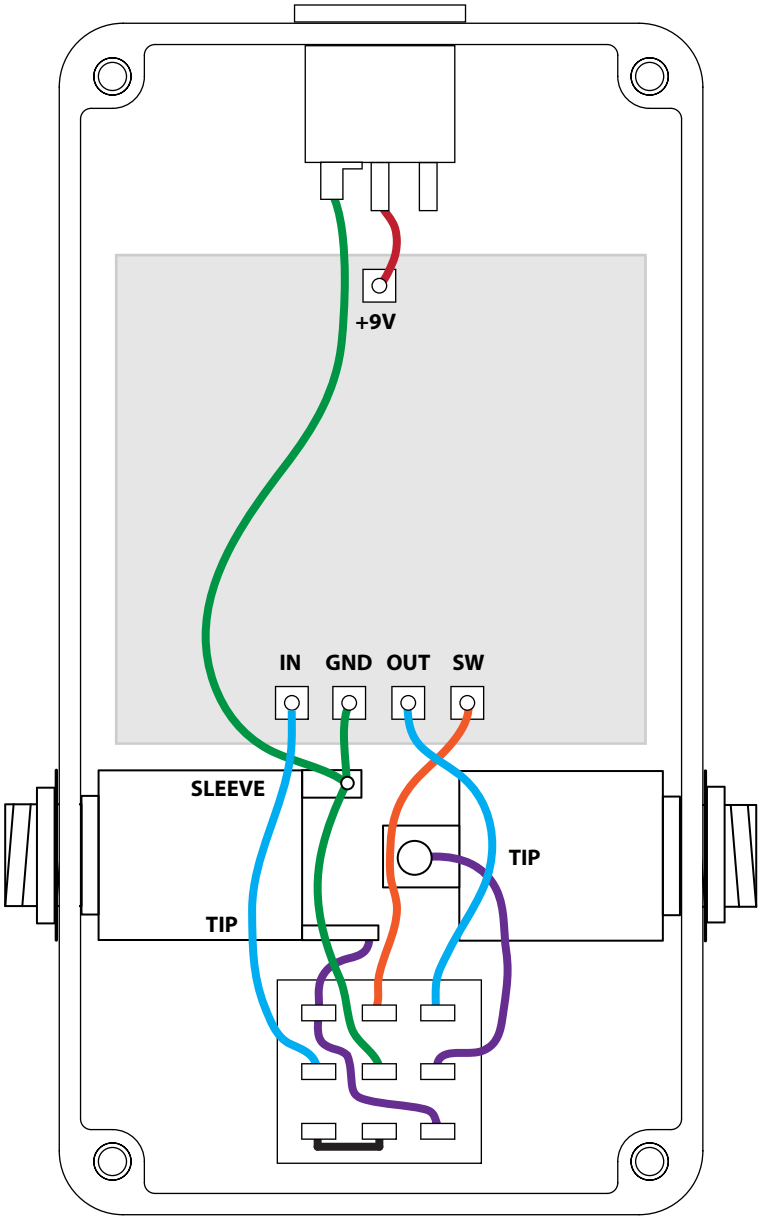
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)