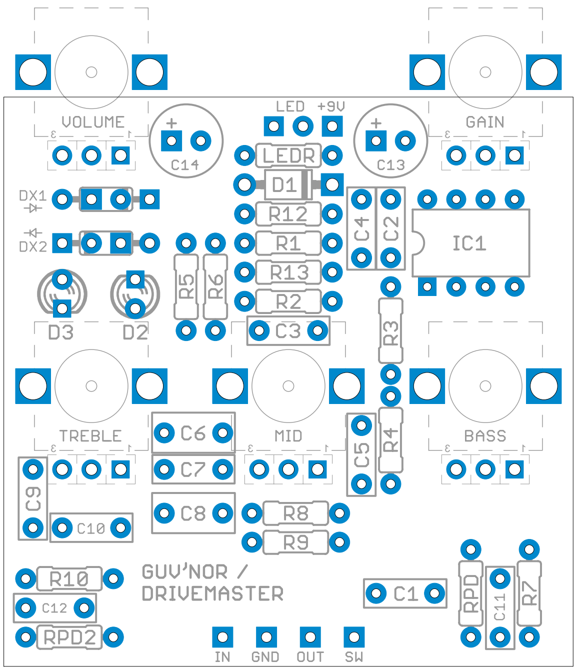
Equinox Overdrive

Marshall Guv'nor / Drivemaster

Overview



The Equinox Overdrive project is a clone of the Marshall Guv'nor and Drivemaster. (They have different names, but they are identical circuits in every way except that the Guv'nor has an effects loop which has been excluded from this project).

First released in 1988, the Guv'nor was replaced in 1991 by the Drivemaster, the same time the Shredmaster and Bluesbreaker pedals were released. It was a very popular pedal at the time and remains well-regarded.

The Guv'nor was among the first overdrive pedals to use a 3-band tone control, allowing for more amp-like toneshaping via bass, midrange and treble knobs. Because of this, you can get a very wide range of tones out of it and it is suitable for many different styles of playing.

Be aware that due to the 5-knob layout, this project uses 9mm PCB-mounted pots (widely available from Small Bear, Tayda, etc). If you use standard 16mm pots, you will have to wire them off-board and it may be really tight.

Controls & Usage

The Guv'nor controls are pretty fancy for an overdrive:

- **Gain** controls the amount of gain from the op amp that is fed into the diode clipping stage.
- **Bass**, **Mid** and **Treble** act as a 3-band tone control, allowing for very flexible tone shaping.
- **Volume** controls the overall output.

Modifications & Experimentation

This is a pretty tight control layout already, so no switch mods for this one. The original uses LEDs for clipping diodes, but pads have been provided for other diodes—you just won't be able to switch between them.

Parts

Resistors

R1	1M
R2	2k2
R3	10k
R4	680k
R5	1k
R6	1k5
R7	680R
R8	680R
R9	100R
R10	22k
R12	47k
R13	47k
RPD	1M to 2M2
RPD2	1M [1]
LEDR	4k7

Capacitors

C1	10n
C2	120pF
C3	100n
C4	68n
C5	220pF
C6	220n
C7	100n
C8	220n
C9	4n7
C10	10n
C11	68n
C12	470pF
C13	47uF
C14	10uF

Semiconductors

IC1	TL072
D1	1N4002
D2, D3	5mm red LED
DX1, DX2	[2]
LED	5mm LED

Potentiometers [3]

Gain	100kB 9mm
Bass	10kA 9mm
Mid	10kA 9mm
Treble	10kA 9mm
Volume	100kB 9mm

[1] **Optional.** This is a pulldown resistor on the output. It's used in the original and may help with switch popping, but does not affect tone. Most effect pedals do not have an output pulldown so you can probably leave it off.

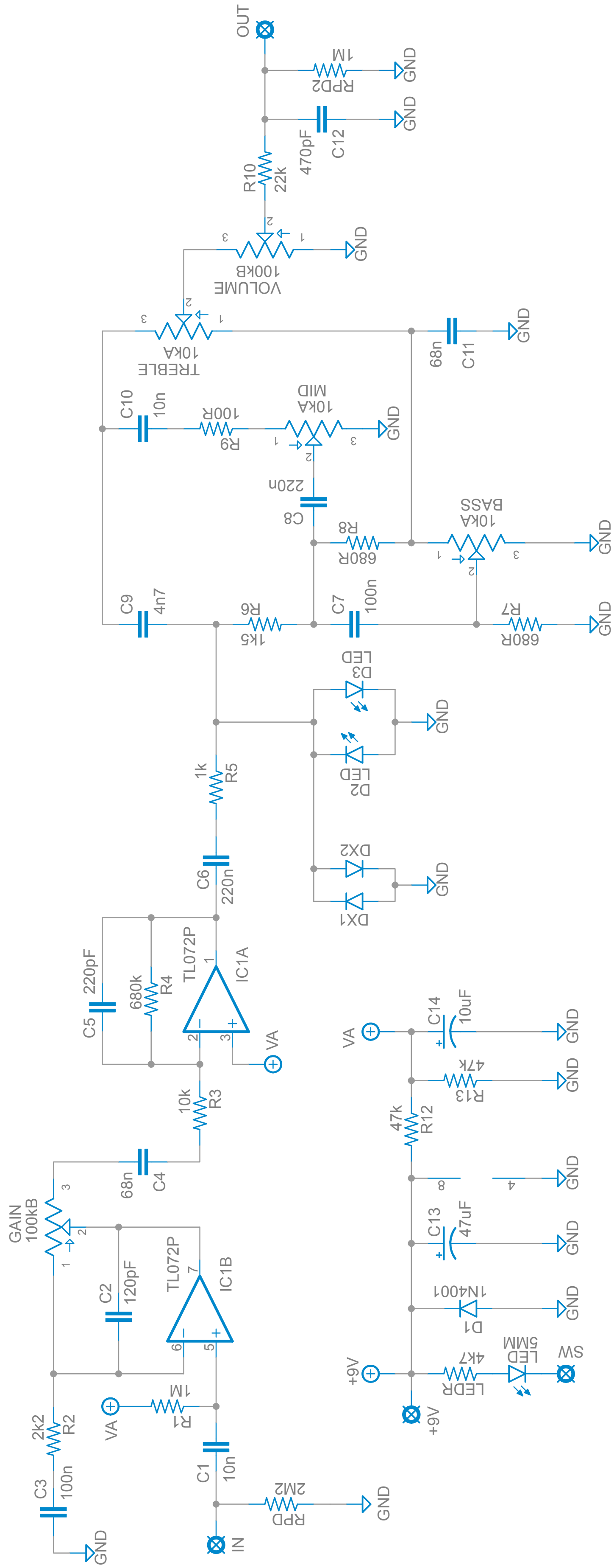
[2] Extra pads have been provided to use alternate diodes if you want, but you'll only want to use one set of them (D2/D3 or DX1/DX2).

[3] This project uses **Alpha 9mm PCB-mount pots**, widely available from Small Bear, Mouser, Tayda, Mammoth and many other places. The standard 16mm pots will not fit easily!

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.

Schematic



OUT
RPD2
1M
GND
C12
470pF
R10
22k
VOLUME
100kB
TREBLE
10kA
C11
68n
C10
10n
R9
100R
MID
10kA
C8
220n
R8
680R
BASS
10kA
C9
4n7
R6
1k5
C7
100n
R7
680R
D3
LED
D2
LED
R5
1k
DX2
DX1
C6
220n
C5
220pF
R4
680k
TL072P
IC1A
VA
C14
10uF
R13
47k
R3
10k
R12
47k
C4
68n
GAIN
100kB
C13
47uF
C2
120pF
IC1B
D1
1N4001
LEDR
4k7
LED
5MM
SW
+9V
R2
2k2
R1
1M
C3
100n
C1
10n
RPD
2M2
IN

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

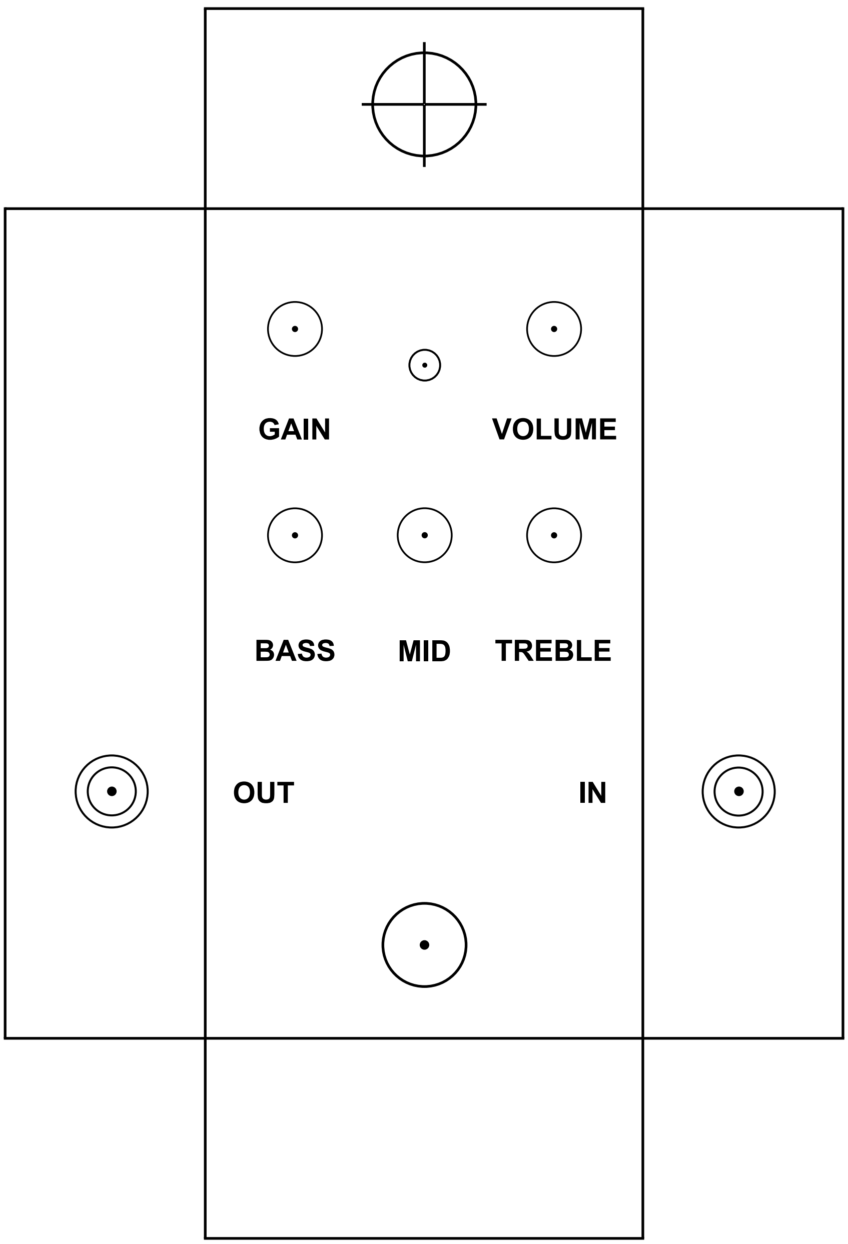
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

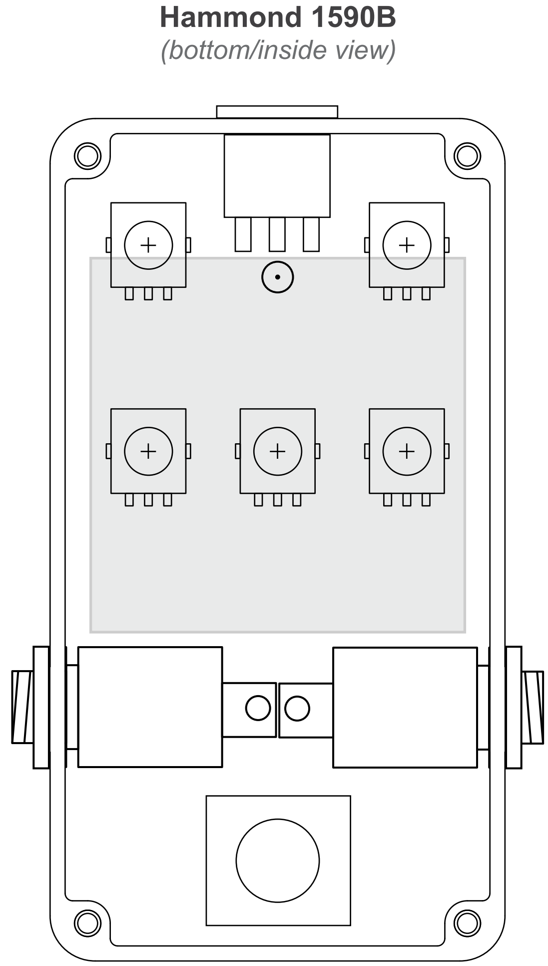
Sockets

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.





Parts Used

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

Standard Wiring Diagram

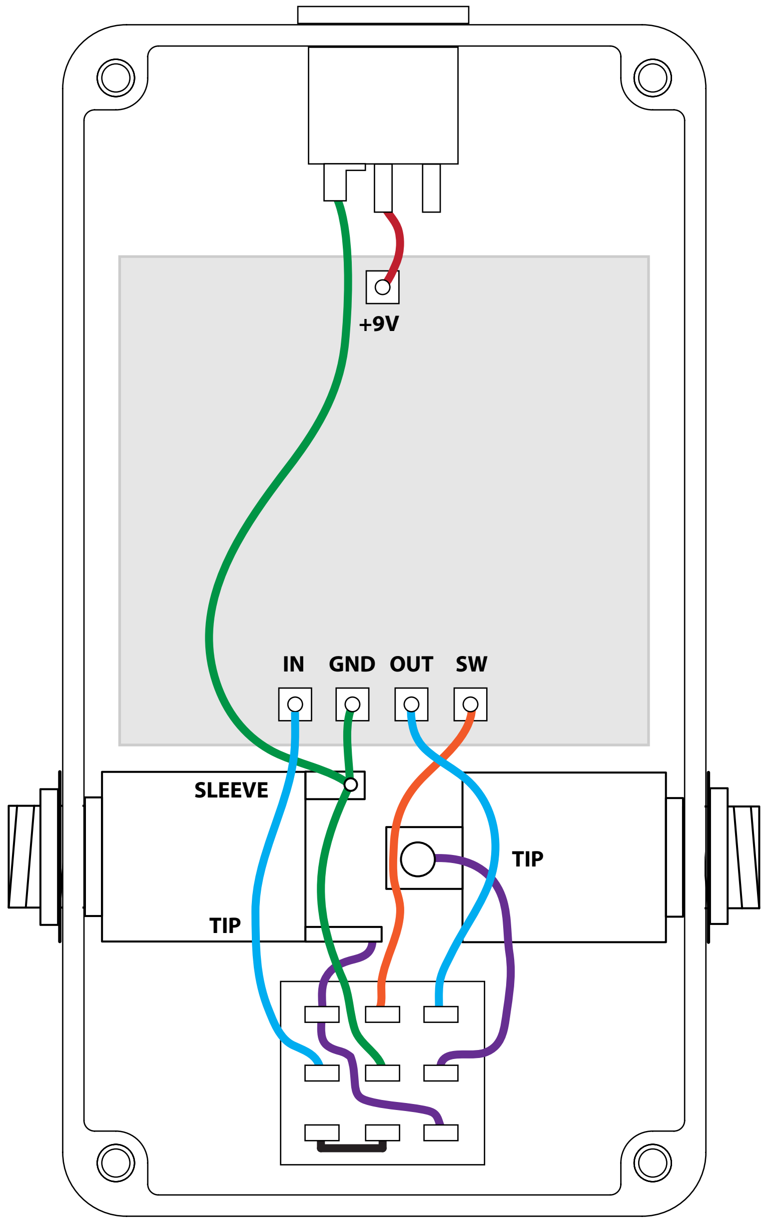
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)