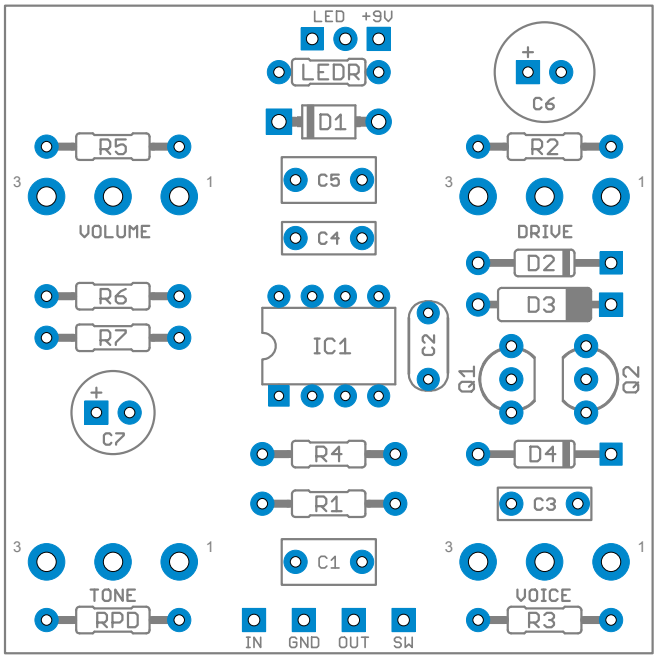
Azimuth Overdrive

Hermida/Lovepedal Zendrive

Overview

Azimuth Project Link



The Azimuth Overdrive project is a clone of the Hermida / Lovepedal Zendrive. It was first released ... and in 2013 production was taken over by Lovepedal. The circuit has changed very little since its inception, with the only major change being the op-amp once Lovepedal took over.

The circuit is very basic, with a standard TS-style clipping section followed by a variable R-C treble cut and a buffer. The bass is adjustable via the “Voice” control, which varies the corner frequency of the filter coming off the op-amp. This also has the side-effect that the available gain increases as the Voice control is turned up.

Despite the legendary stature of the Zendrive, the circuit has the lowest parts count of any of my PCB projects and is a great project for beginners.

Controls & Usage

The Zendrive has the following controls:

- **Drive** controls the amount of gain in the op amp feedback diode clipping stage.
- **Tone** controls the treble response of the effect.
- **Voice** controls the amount of bass while also changing the gain structure. As you turn it up, the available gain range also increases.
- **Volume** controls the overall output.

Modifications & Experimentation

This one won’t be a whole lot of fun for a tweaker. The Zendrive is what it is, and with such a low parts count, if you start changing values around then it quickly stops being a Zendrive.

Parts

Resistors

R1	470k
R2	1k
R3	1k
R4	10k
R5	1k
R6	10k
R7	10k
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	470n film
C2	100pF MLCC
C3	100n
C4	3n3
C5	470n film
C6	100uF electro
C7	47uF electro

Semiconductors

IC1	AD712 [1]
D1	1N4002
D2, D4	BAT41
D3	1N34A Ge [2]
Q1, Q2	2N7000
LED	5MM

Potentiometers

Drive	500kA [3]
Tone	50kB
Voice	10kB
Volume	100kA [4]

[1] **Op-amp selection:** In the original, the back of the op-amp was sanded down, so while it's not known for sure, some expert DIY detectives found that it was almost certainly an AD712. The Lovepedal version uses an unsanded **NE5532**. Due to the high diode-clipping threshold, this circuit has a particularly strong dependence on the type of op-amp used, so it's worth experimenting with a few. Other DIY favorites include the OPA2134, OPA2604, TC2272, MC1458 and LM833.

[2] **Clipping diode:** Some versions of the Zendrive use another BAT41 here instead of a germanium. It can be any type of germanium diode, not just 1N34A.

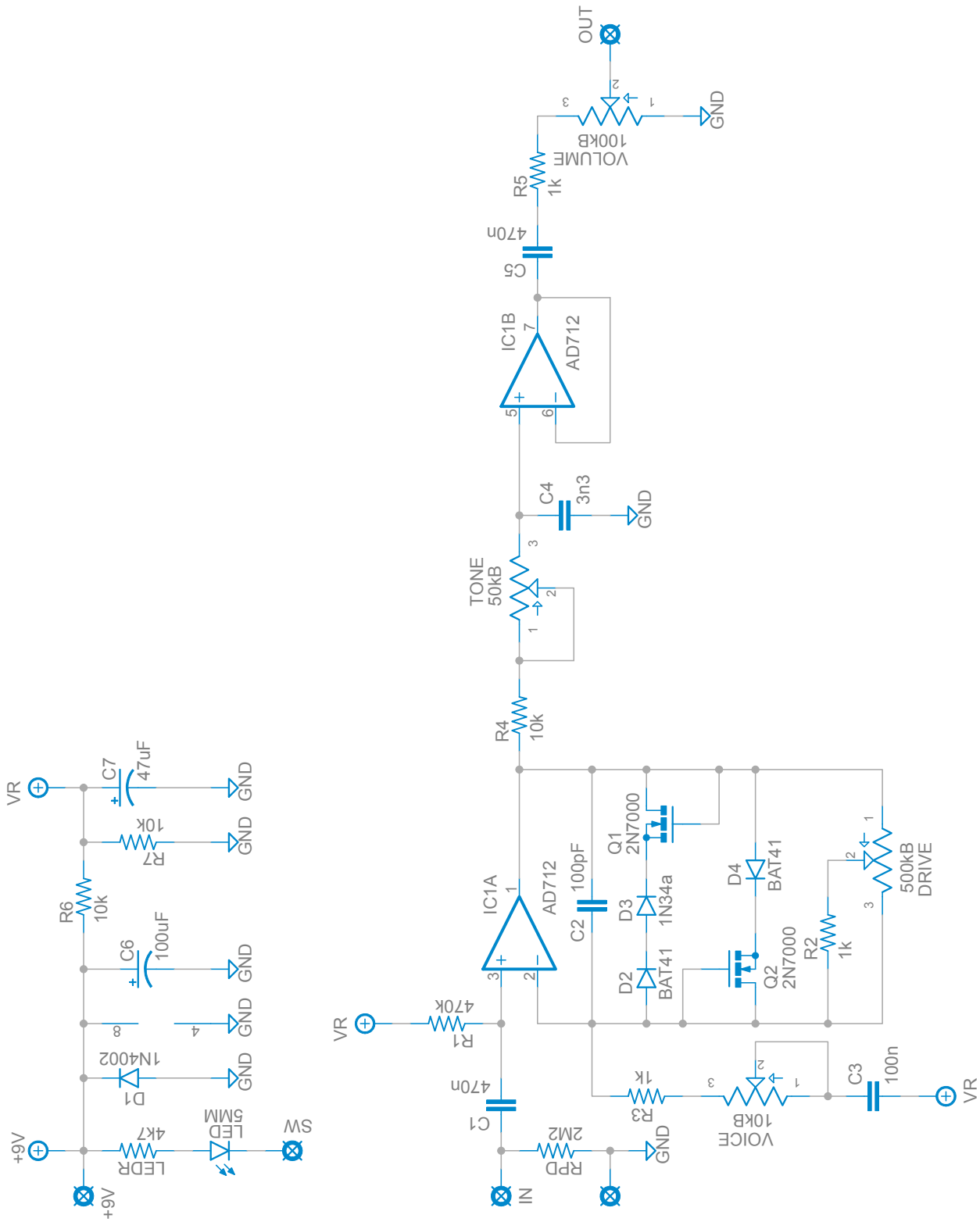
[3] **Drive pot value:** The original uses **500kB** (linear taper) but I recommend using a 500kA (audio taper).

[4] **Volume pot value:** The original uses **100kB** (linear taper) but I recommend using a 100kA (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
GND
VOLUME
100KB
R5
1k
470n
C5
IC1B
AD712
C4
3n3
GND
TONE
50kB
R4
10k
IC1A
AD712
C2
100pF
Q1
2N7000
D3
1N34a
D2
BAT41
D4
BAT41
Q2
2N7000
R2
1k
500kB
DRIVE
VR
470k
R1
470n
C1
IN
RPD
2M2
GND
R3
1k
VOICE
10KB
C3
100n
VR
VR
C7
47uF
GND
10k
R7
GND
R6
10k
C6
100uF
GND
GND
1N4002
D1
GND
+9V
4k7
LEDR
LED
5MM
SW
+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

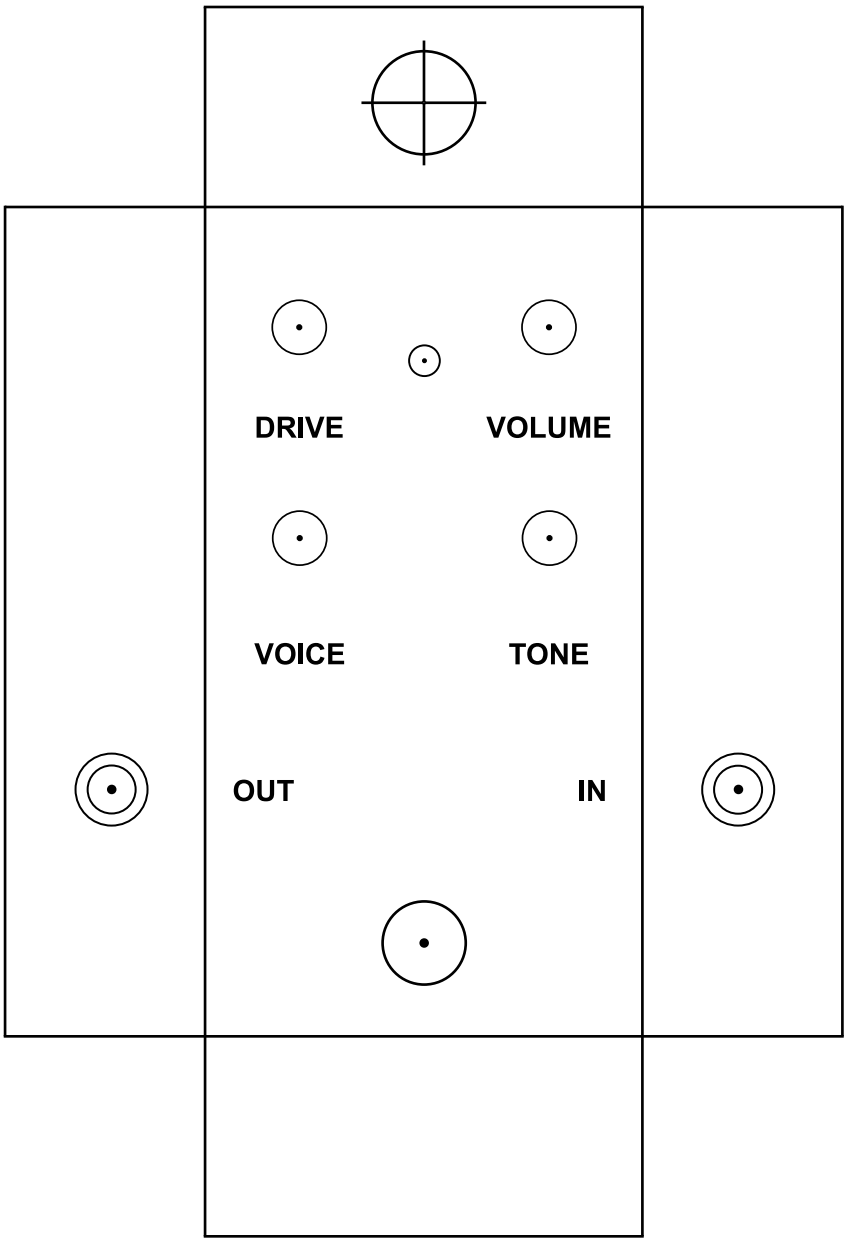
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

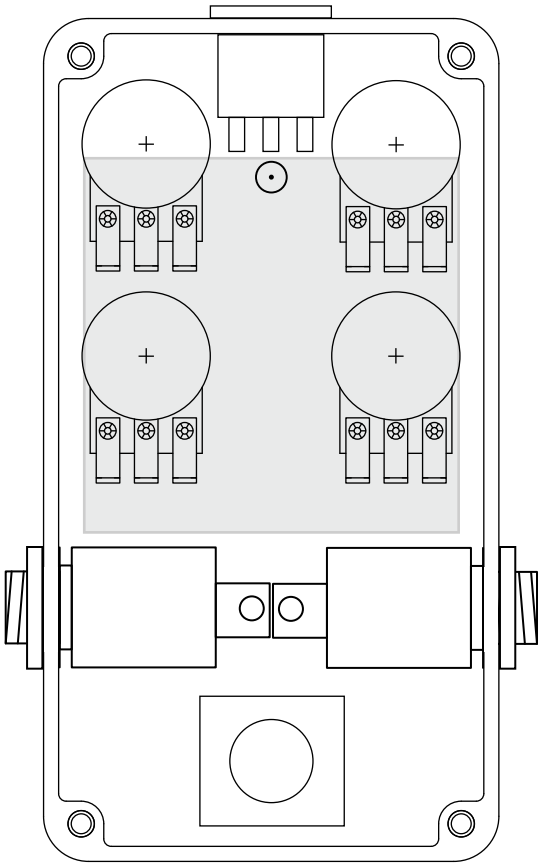
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

Drilling & Placement

Print this page and 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

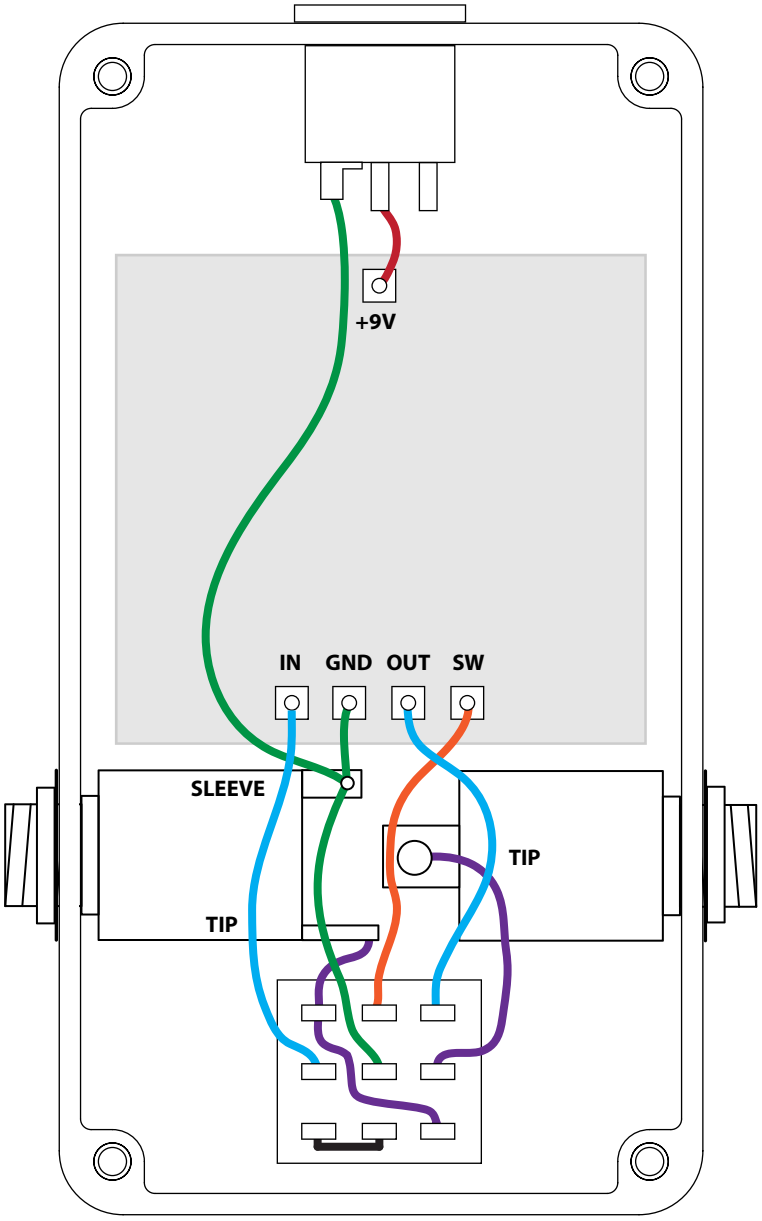
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)