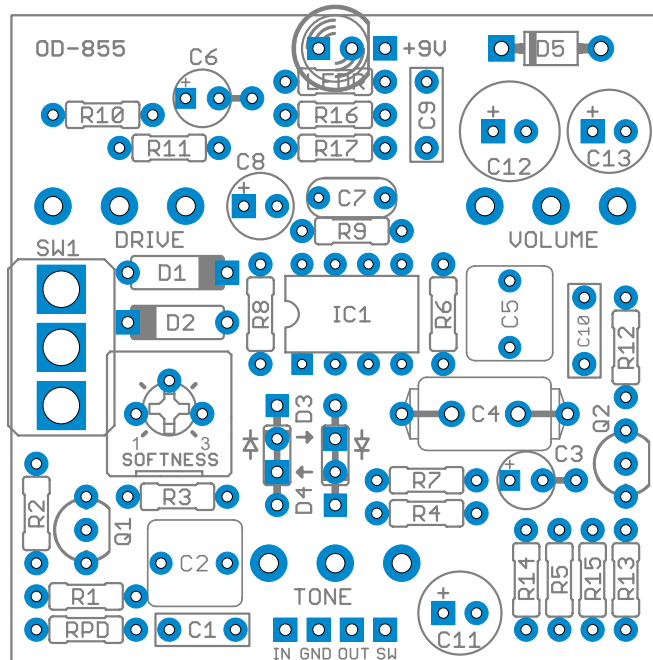


# Cumulus Overdrive

Ibanez Overdrive II / OD-855



## Overview



The Cumulus Overdrive project is a clone of the Ibanez Overdrive II / OD-855. This pedal was first released in 1977 and was a precursor to the Tube Screamer. However, it actually has more in common with an MXR Distortion+ or DOD 250.

This project is an exact copy of the original, but allows for an optional diode switch mod. While the original uses germanium clipping diodes, this board includes space for a switch and second set of clipping diodes if you'd like to experiment. There's also an optional "Softness" trimmer to adjust the clipping threshold of the mod side of the switch, and if you use a center-off toggle switch you can get a third "diode lift" or "comp-cut" mode.

## Controls & Usage

The OD-855 has the same control layout as most overdrive or distortion effects:

- **Tone** controls the treble response of the effect. (Interestingly, in this circuit the tone control comes before the clipping stage instead of after. This is the only effect I know of that does it this way.)
- **Drive** controls the amount of gain in the op amp stage right before the diode-to-ground clipping.
- **Volume** controls the overall output.

## 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.

The second clipping section includes the **Softness** trimmer to control the clipping threshold. This can produce interesting sounds and is worth experimenting with. If you don't like it, you can turn it all the way down (zero resistance) to take it out of the circuit—or, to bypass it entirely, run a jumper between pads 1 and 3.

## Parts

### Resistors

R1	10k
R2	510k
R3	3k
R4	3k
R5	680R
R6	100k
R7	3k
R8	10k
R9	100k
R10	470R
R11	10k
R12	510k
R13	10k
R14	100R
R15	10k
R16	10k
R17	10k
RPD	1M to 2M2
LEDR	4k7

### Capacitors

C1	22n
C2	1uF film
C3	100n/0.1uF <sup>1</sup>
C4	470pF <sup>2</sup>
C5	1uF film
C6	470n/0.47uF <sup>1</sup>
C7	20pF
C8	4u7
C9	2n2
C10	100n
C11	10uF
C12	100uF 25v
C13	47uF 25v

### Semiconductors

Q1 - Q2	2N5088
IC1	JRC4558D
D1 - D2	1N60P <sup>3</sup>
D3 - D4	<sup>4</sup>
D5	1N4002
LED	5MM

### Potentiometers

Tone	20kW
Distortion	50kC
Volume	10kB
Softness	10k trim (3362P)

### Other

SW1	SPDT center off
-----	-----------------

<sup>1</sup> **Can be film or tantalum.** C3 and C6 are tantalum in the original circuit, but the PCB has extra pads to accommodate a 5mm film capacitor as well.

<sup>2</sup> **Can be film, ceramic, silver mica or polystyrene.** I included space for the huge [Xicon polystyrene](#) caps which I like a lot, but the pads in the middle have been provided for other caps with 5mm lead spacing.

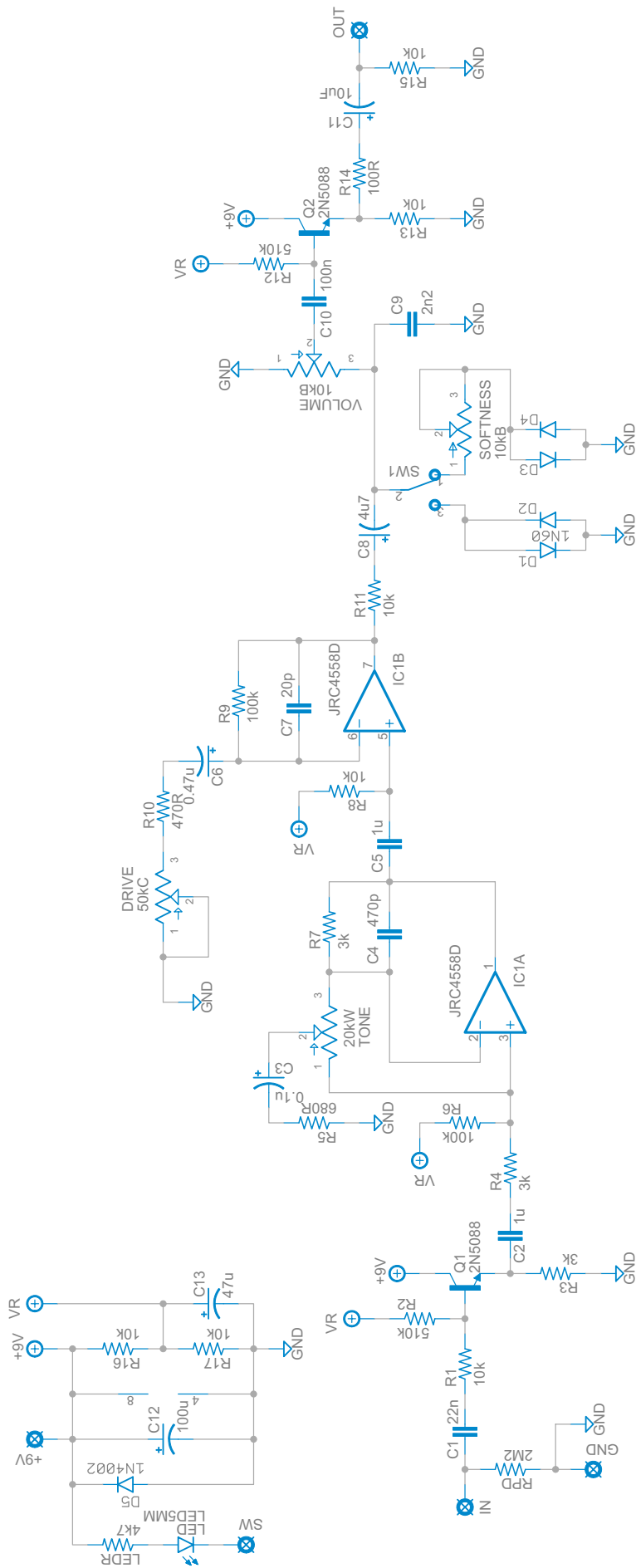
<sup>3</sup> **The exact part number is unimportant.** Germaniums generally have very wide ranges of forward voltage even among the same part number. The 1N60P has about a 0.3v voltage drop, so any germanium with similar specs will sound the same.

<sup>4</sup> **Your choice.** Socket these and have fun! Try silicon (1N914) for a different flavor from the germaniums, or go big with LEDs for a more open, crunchier sound.

## 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



## 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**<sup>1</sup> 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**<sup>2</sup> 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.

<sup>1</sup> **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.

<sup>2</sup> **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

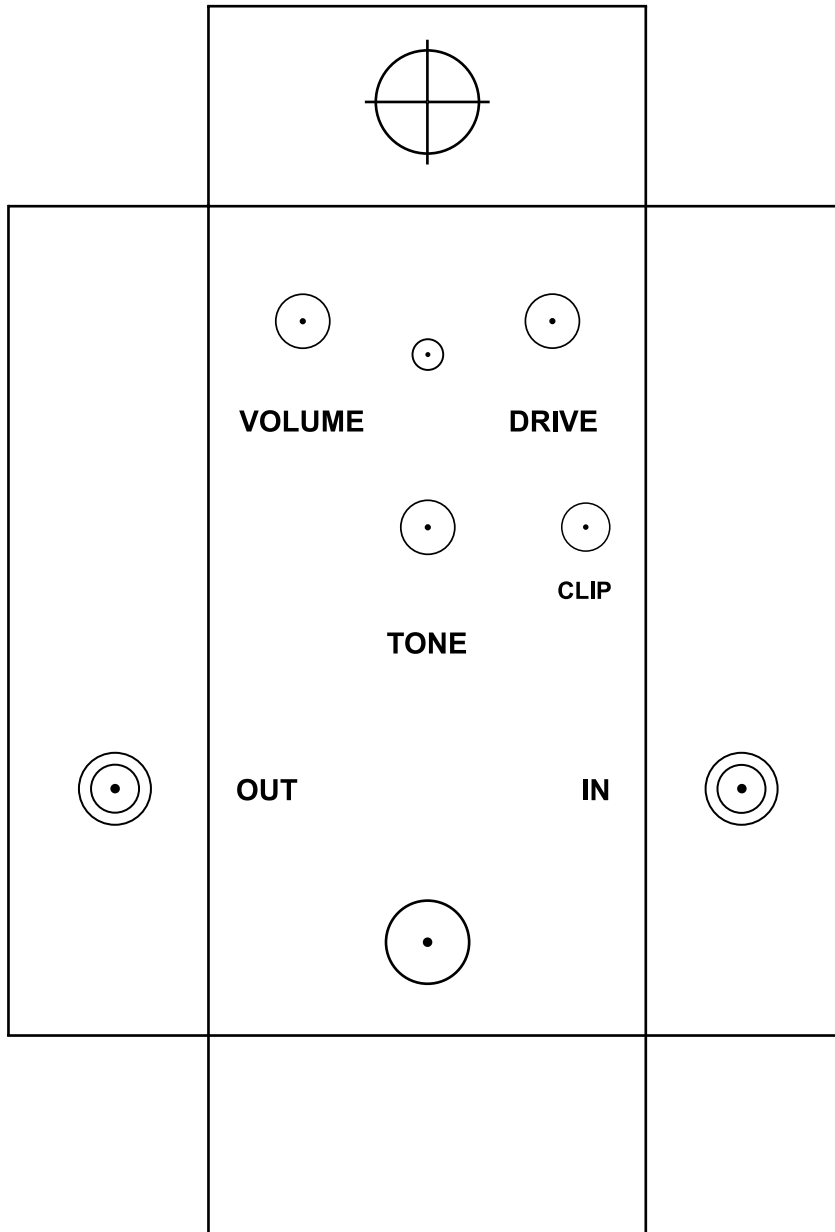
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

### Sockets

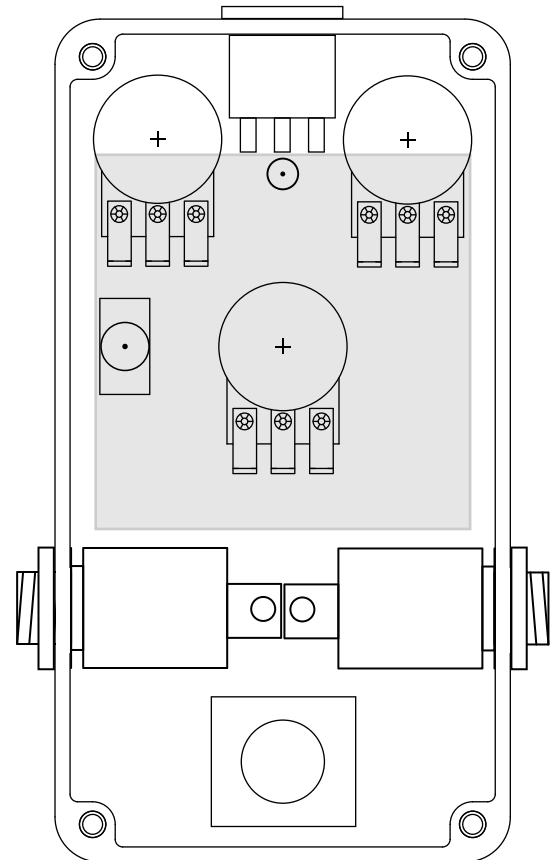
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

## 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

## Standard Wiring Diagram

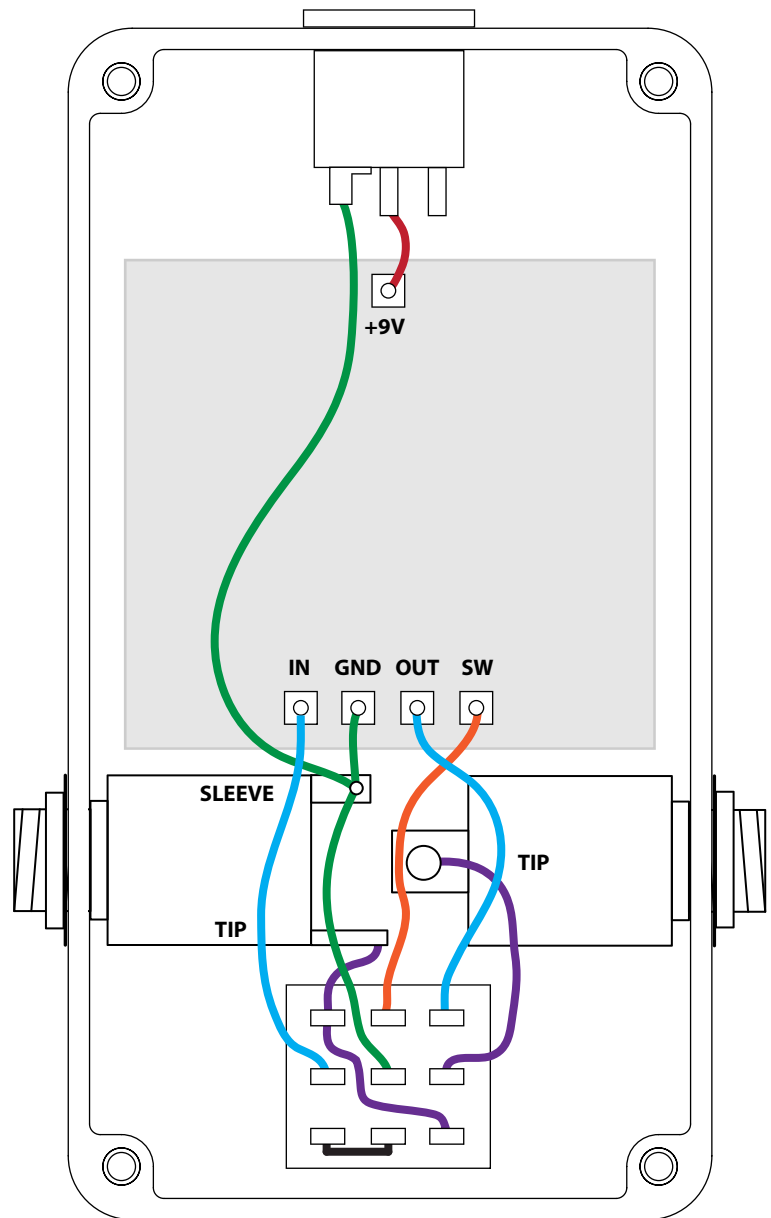
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)