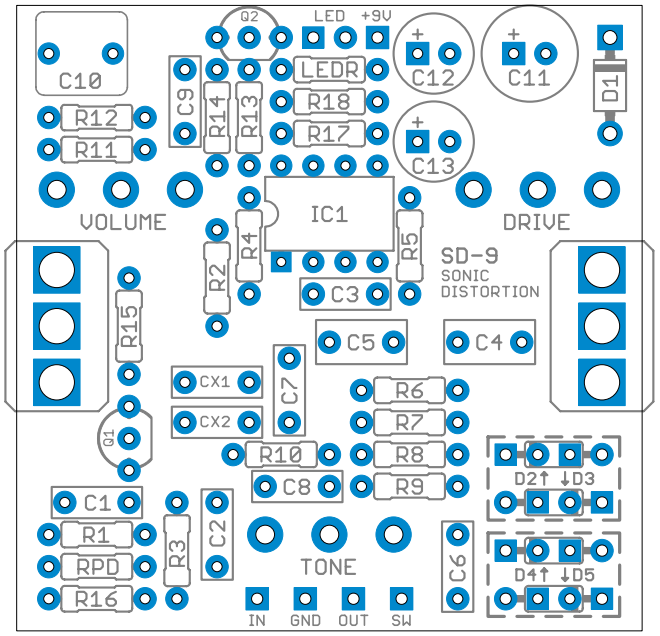
Meteor Distortion

Maxon/Ibanez SD-9 Sonic Distortion

Overview



The Meteor Distortion project is a clone of the Maxon/Ibanez Sonic Distortion, which first hit stores in around 1983. Advertised as a heavier Tube Screamer, in reality it shares nothing at all in common with the classic TS-9 circuit. It has more similarities to hard-clipping circuits such as the BOSS DS-1 or MXR Distortion+.

A “modded reissue” of the SD-9 was released by Maxon in 2012 as the SD-9M. While on the outside it appears to be just an SD-9 with two switch mods, the inside is more than a little different, with a couple of additional op-amp gain stages not found in the original.

This project is for the original SD-9, but features one of the two mods from the SD-9M, the “Mids” switch. (The second SD-9M mod, the gain switch, is not backwards-compatible with this circuit.)

The second switch on this PCB allows you to switch between different sets of clipping diodes.

Controls & Usage

The SD-9’s controls are standard drive/distortion controls:

- **Drive** controls the amount of gain from the op amp that is fed into the diode clipping stage.
- **Tone** is similar to the Big Muff tone control, panning between a bass and treble emphasis (in this case, with a pretty significant mid scoop).
- **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.

Note that the original SD-9 suffers from a fairly low output volume, which is a result of the low clipping threshold of the diodes plus the lack of a gain recovery stage. By using diodes with a higher clipping threshold such as LEDs or two 1N914s in series, the output volume is increased.

The **Mids** switch allows you to switch between a couple of different capacitors on the “bass” side of the tone section to change the frequencies and give more emphasis to the midrange. In the default configuration, the mids are scooped.

Parts

Resistors

R1	1k
R2	510k
R3	10k
R4	100k
R5	33k
R6	470R
R7	2k2
R8	6k8
R9	2k2
R10	6k8
R11	1k [1]
R12	10k
R13	1M
R14	10k
R15	470R
R16	100k
R17	22k
R18	22k
RPD	1M to 2M2
LEDR	4k7

Capacitors

C1	47n
C2	47n
C3	1n
C4	220n
C5	470n
C6	10n
C7	33n [2]
C8	27n
C9	100n
C10	1uF
C11	100uF
C12	47uF
C13	10uF
CX1	33n [2]
CX2	68n [2]

Semiconductors

IC1	JRC4558D
Q1, Q2	2N5088
D1	1N4002
D2, D3	1N914
D4, D5	3mm LED [3]
LED	5mm LED

Potentiometers

Drive	250kB
Tone	25kB
Volume	100kB

Other

CLIP	SPDT center off
MIDS	SPDT center off

[1] This resistor sets the minimum volume for the volume control, preventing it from going entirely to zero. This is very uncommon in guitar pedals and **should be jumpered** unless you are going for a 100% original build.

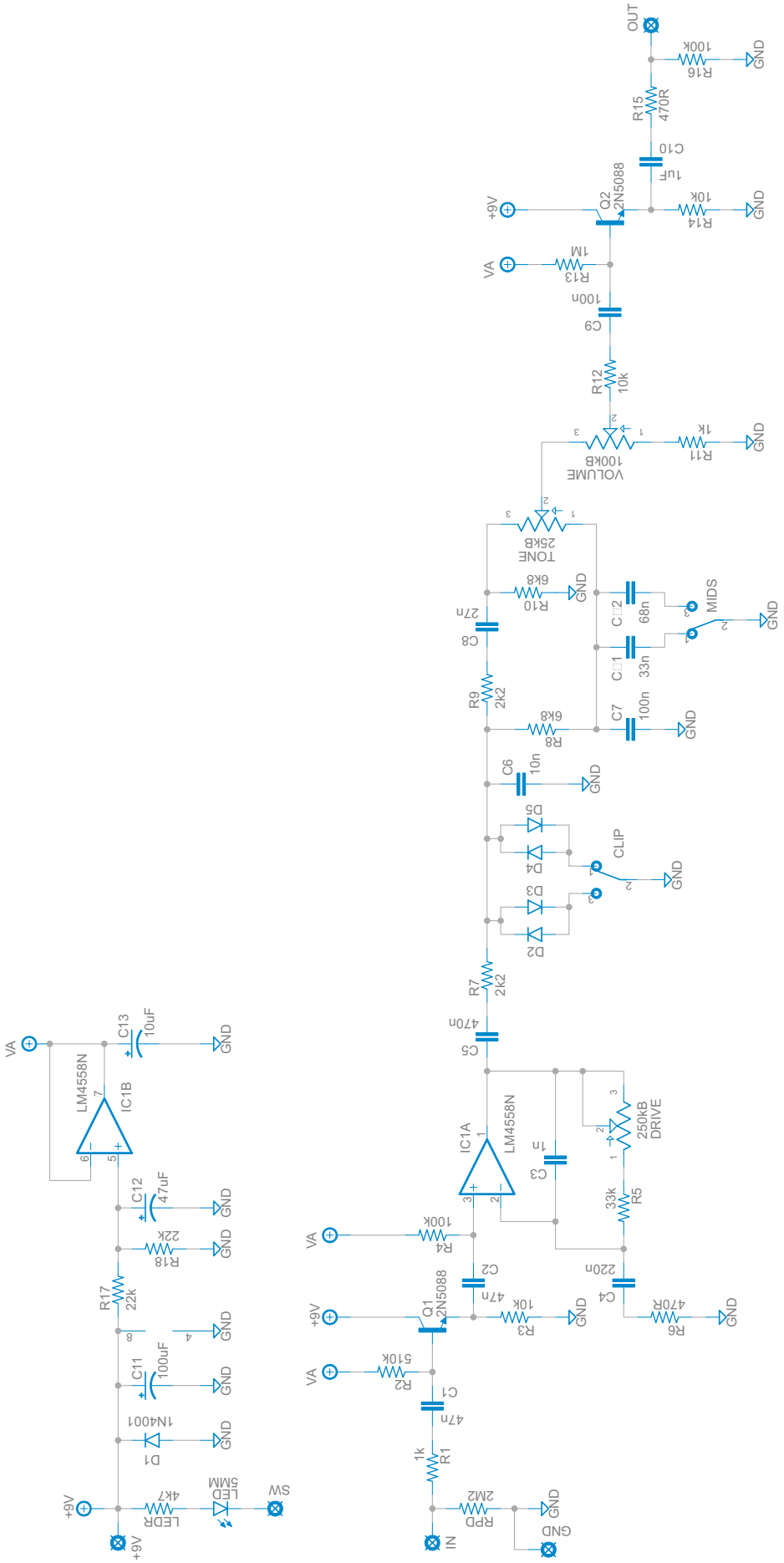
[2] The original value for this is **100n**. If you exclude the Mids switch (and CX1/2), use the original value. Otherwise, a smaller value of 33n should be used, which in combination with CX1/2 will provide values of **33n** (center/off position), **66n** (with CX1) and **101n** (with CX2).

[2] You can use whatever diodes you'd like for the second set, but I would recommend using some with high clipping thresholds, such as **LEDs** or **two 1N914s in series** on each side. This will increase the output volume of the effect, which is fairly low when using the stock diodes.

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


OUT
100K
R16
GND
R15
470R
C10
1uF
+9V
Q2
2N5088
10K
R14
VA
1M
R13
100n
C9
R12
10k
1K
R11
VOLUME
100KB
TONE
25KB
6K8
R10
27n
C8
68n
MIDS
33n
R9
2k2
6K8
R8
100n
C7
C6
10n
D5
D4
CLIP
D3
D2
R7
2k2
470n
C5
IC1A
LM4558N
1n
C3
250kB
DRIVE
33k
R5
220n
C4
470R
R6
100K
R4
47n
C2
Q1
2N5088
10K
R3
510K
R2
C1
47n
1k
R1
2M2
RPD
IN
LM4558N
IC1B
C13
10uF
C12
47uF
22K
R18
R17
22k
C11
100uF
1N4001
D1
LEDR
4K7
LED
5MM
SW

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

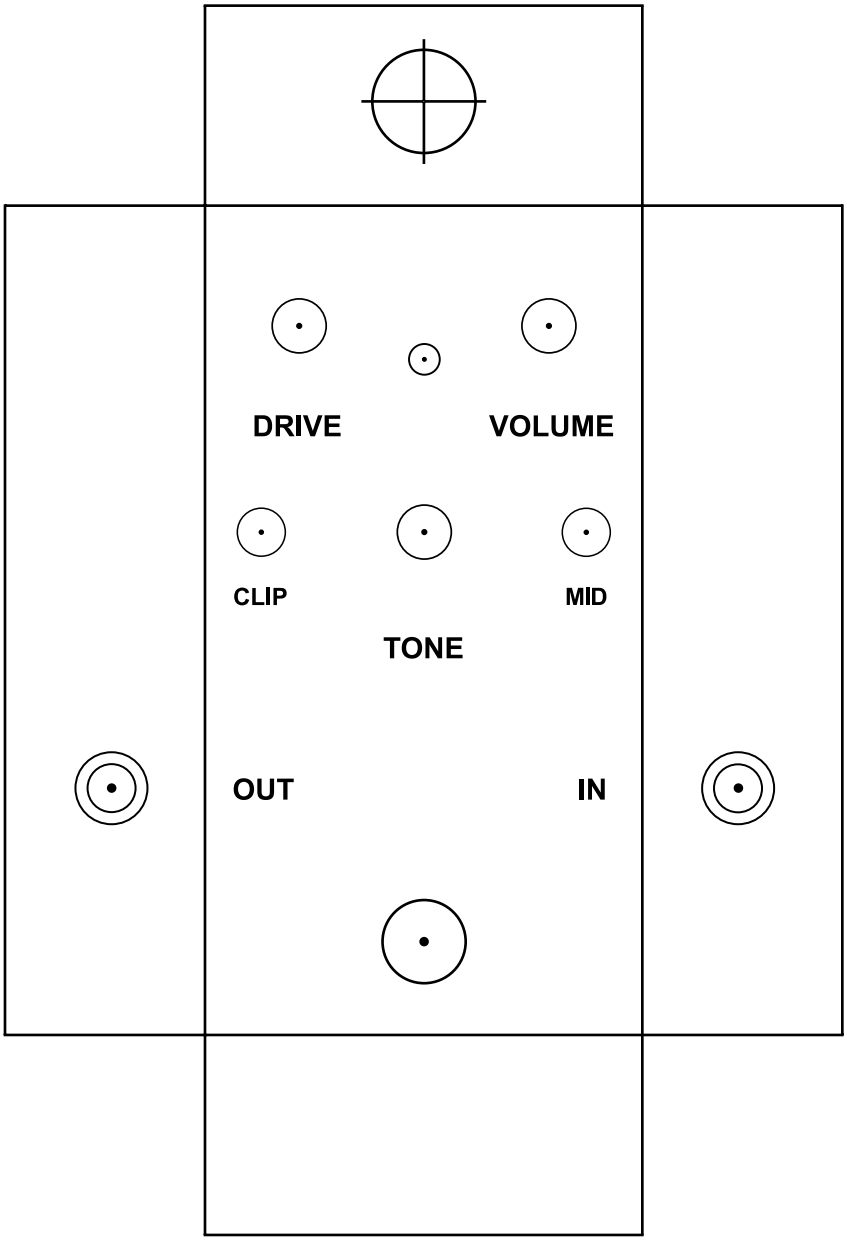
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

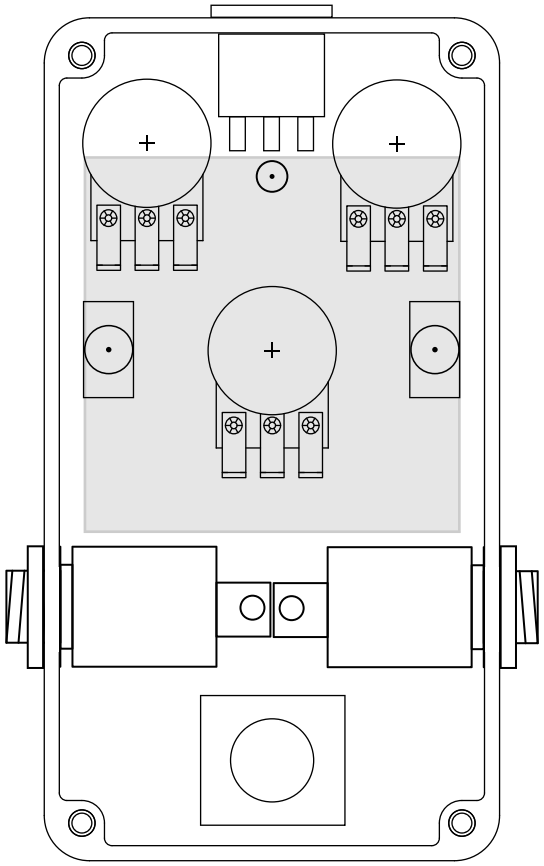
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

Standard Wiring Diagram

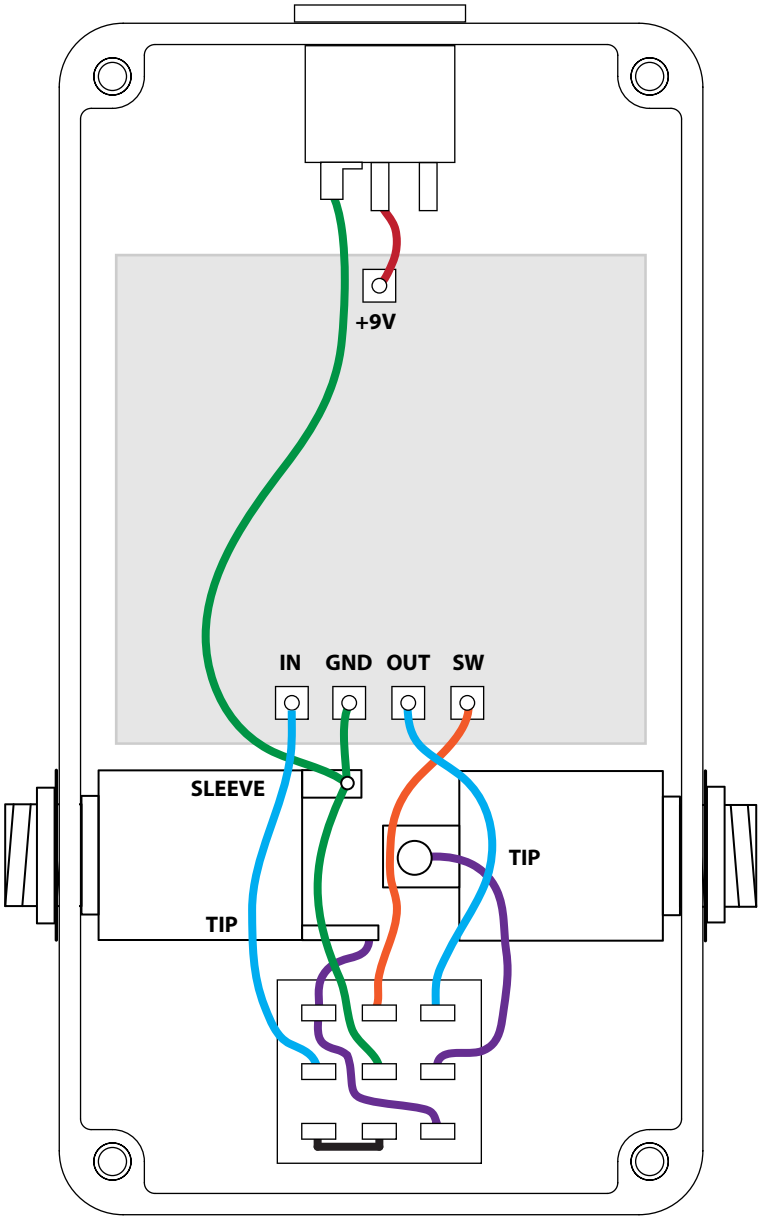
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)