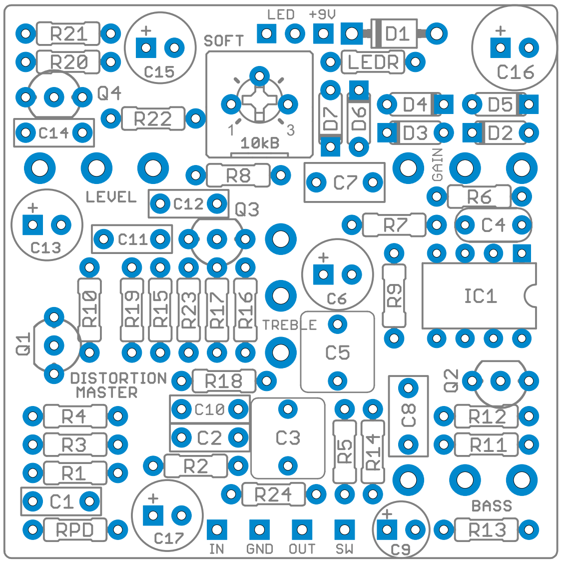
Nebula

Maxon DS-830 Distortion Master

Overview

Nebula Project Link



The Nebula Distortion is a clone of the Maxon DS-380, a relatively new addition to Maxon's line of pedals but one that maintains the spirit of their innovations in the 1980s.

Despite being a relative late-comer, the circuit is a very worthy addition to Maxon's product line and retains the spirit and character of their classic designs: a JRC4558D op-amp with feedback-diode clipping like the Tube Screamer, plus some interesting twist that makes it unique. In this case, it's a two-band gyrator tonestack (not dissimilar from a Boss HM-2 Heavy Metal) that offers a great deal more flexibility and imparts a very different tonal character than the typical amp-like passive tonestack.

Controls & Usage

- **Gain** controls the amount of gain from the op amp that is fed through the feedback clipping diodes.
- **Level** is the output level of the effect.
- **Bass** allows you to adjust the low-end response of the circuit via a gyrator-based tone control.
- **Treble** allows you to adjust the high-end response of the circuit, also via a gyrator tone control.

Modifications

This is a tight layout, so you won't find any of the "standard" Aion modifications such as a clipping diode switch. However, there are extra pads for the soft clipping diodes clipping diodes (D2 through D5) in case you want to stack two diodes in series—for instance, **two 1N914s** or **one 1N914 and one BAT41** on each side. The stock unit only uses one in each direction.

For the hard-clipping diodes, I've included the "Soft" trimmer which is on most of my other hard-clipping circuits. This allows you to put some resistance in series with the hard-clipping diodes which reduces their effect and softens the hard-clipping somewhat. When turned all the way down, it's fully removed from the circuit, so I recommend including it.

Any other standard-pinout dual op-amps will work in this circuit as well. Try a **TL072** or **OPA2104**.

Like a Tube Screamer, you can adjust **R7** + **C5** to change the gain structure and the low-end rolloff frequency. The stock corner frequency is 408 Hz. Just keep in mind you've got a bass control further along in the circuit so you can dial out whatever you don't like.

Parts

Capacitors

C1	100n
C2	3n9 [4]
C3	1uF film
C4	10pF
C5	1uF film
C6	10uF electro
C7	470n
C8	220n
C9	3.3uF electro
C10	47n
C11	100n
C12	3n3
C13	10uF electro
C14	100n
C15	10uF electro
C16	100uF electro
C17	47uF electro

Resistors

R1	10k
R2	1M
R3	10k

Resistors, cont.

R4	10k
R5	100k
R6	2k
R7	390R
R8	1k
R9	10k
R10	100k
R11	3k9
R12	220R
R13	10k
R14	10k
R15	560R
R16	10k
R17	22k
R18	3k
R19	510k
R20	10k
R21	100k
R22	470R
R23	10k
R24	10k
RPD	2M2
LEDR	4k7

Semiconductors

Q1	2N5457
Q2–Q4	2N5088
IC1	JRC4558D
D1	1N4002
D2, D5	1N914
D3–D4	jumper [1]
D6–D7	1N914
LED	5mm LED

Potentiometers

Gain	250kA [2]
Level	100kA
Bass	25kB [4]
Treble	10kB
Soft	10k trim (3362P) [3]

Build Notes

[1] Extra pads have been provided in case you want to double up on diodes in the feedback clipping section. For a fully stock unit, jumper D3 and D4.

[2] The original uses a linear (B) taper for the Gain control, but I prefer an audio (logarithmic) taper here.

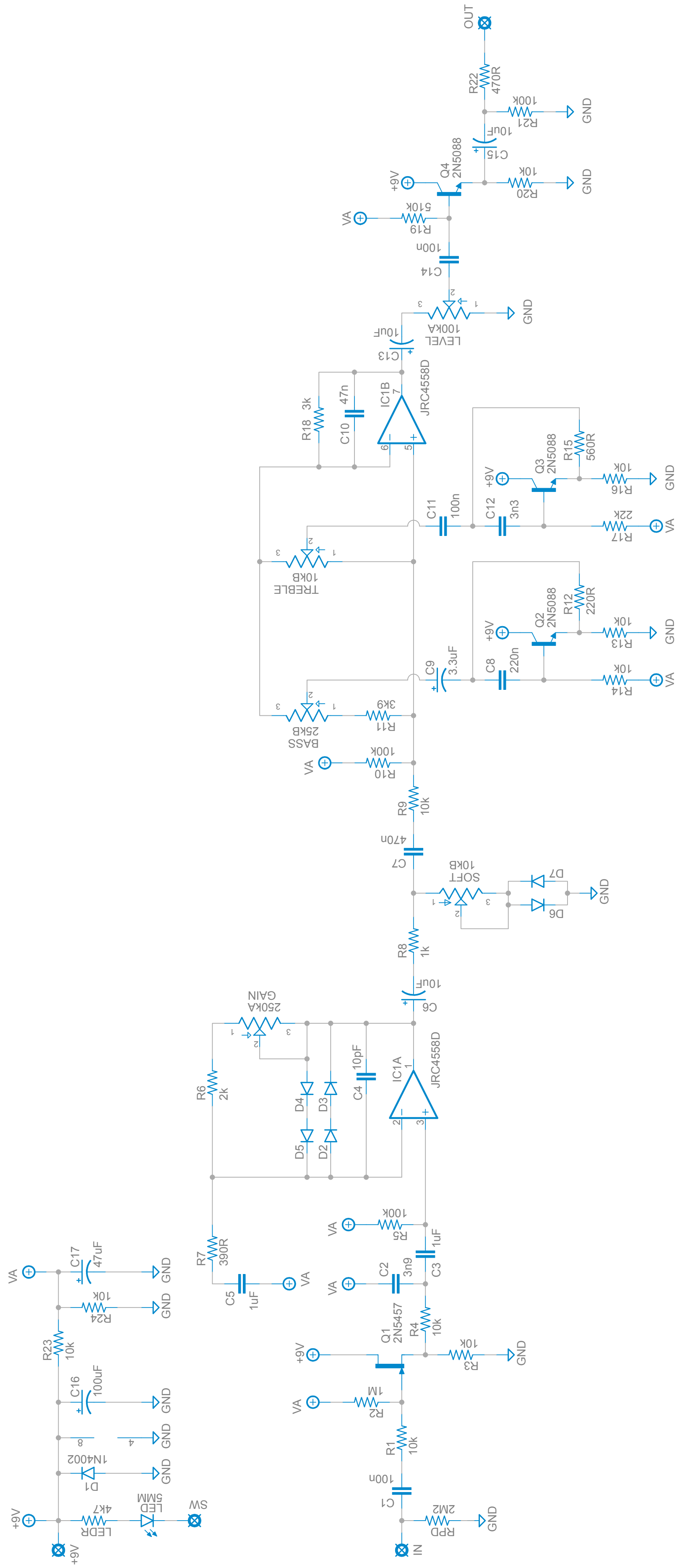
[3] If you want to exclude this control, jumper pads 1 & 2. I recommend including it because it's out of the circuit if turned all the way down. Extra space has been provided in case you want to use the larger 3386 trimmer.

[2] Prior to February 2017, all versions of the DS-830 schematic had C2 at 39n and the Bass pot at 5kB. C2 should instead be **3n9** and the Bass pot should be **25kB**.

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.
- 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
R22 470R
R21 100K
GND
C15 10uF
Q4 2N5088
+9V
R20 10K
VA
R19 510K
C14 100n
LEVEL 100KA
C13 10uF
IC1B JRC4558D
R18 3k
C10 47n
R15 560R
Q3 2N5088
R16 10K
C11 100n
C12 3n3
R17 22K
TREBLE 10KB
R12 220R
Q2 2N5088
R13 10K
C9 3.3uF
C8 220n
R14 10K
BASS 25KB
R11 3K9
R10 100K
R9 10k
C7 470n
SOFT 10KB
D7
D6
R8 1k
C6 10uF
GAIN 250KA
R6 2k
D4
D3
D5
D2
C4 10pF
IC1A JRC4558D
R5 100K
C3 1uF
R7 390R
C5 1uF
C2 3n9
R4 10k
Q1 2N5457
R3 10K
R2 1M
R1 10k
C1 100n
RPD 2M2
IN
C17 47uF
R24 10K
R23 10k
C16 100uF
D1 1N4002
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

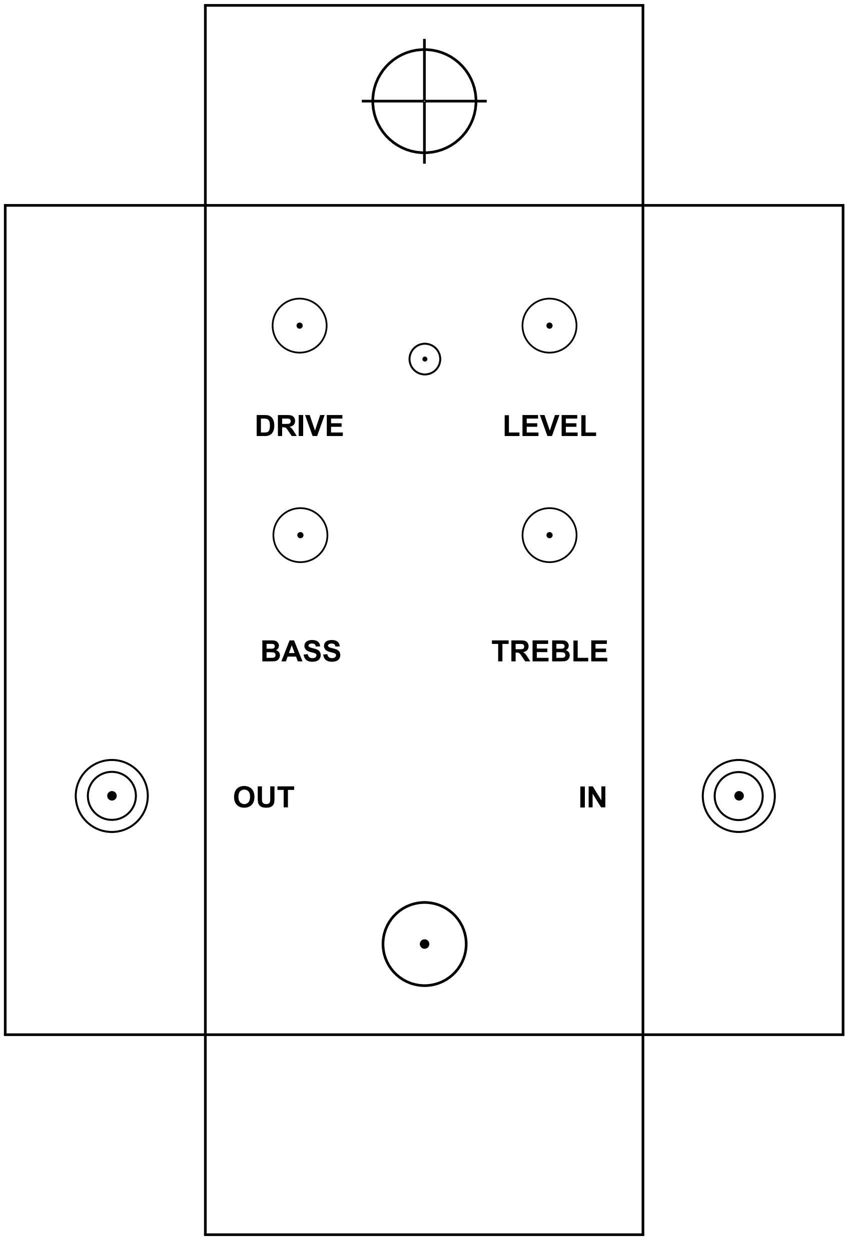
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

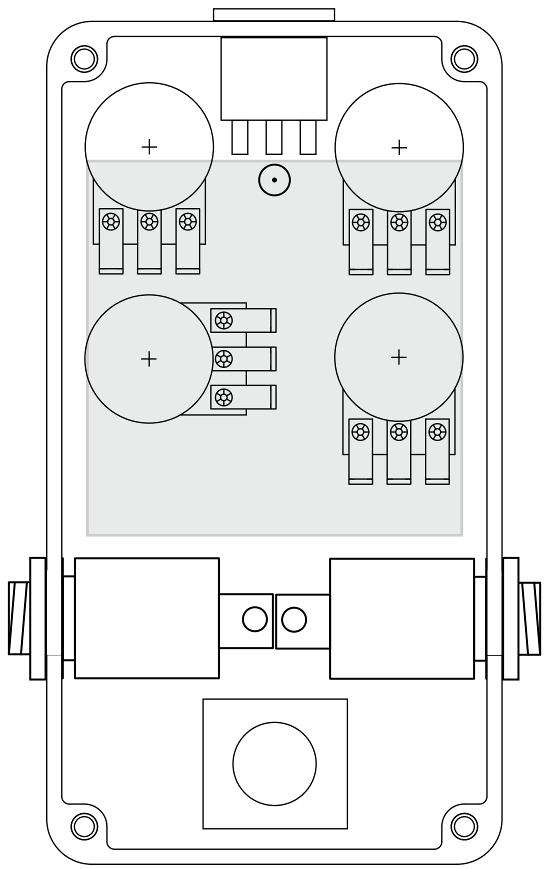
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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 111X enclosed jacks
- Kobiconn-style DC jack with internal nut

Standard Wiring Diagram

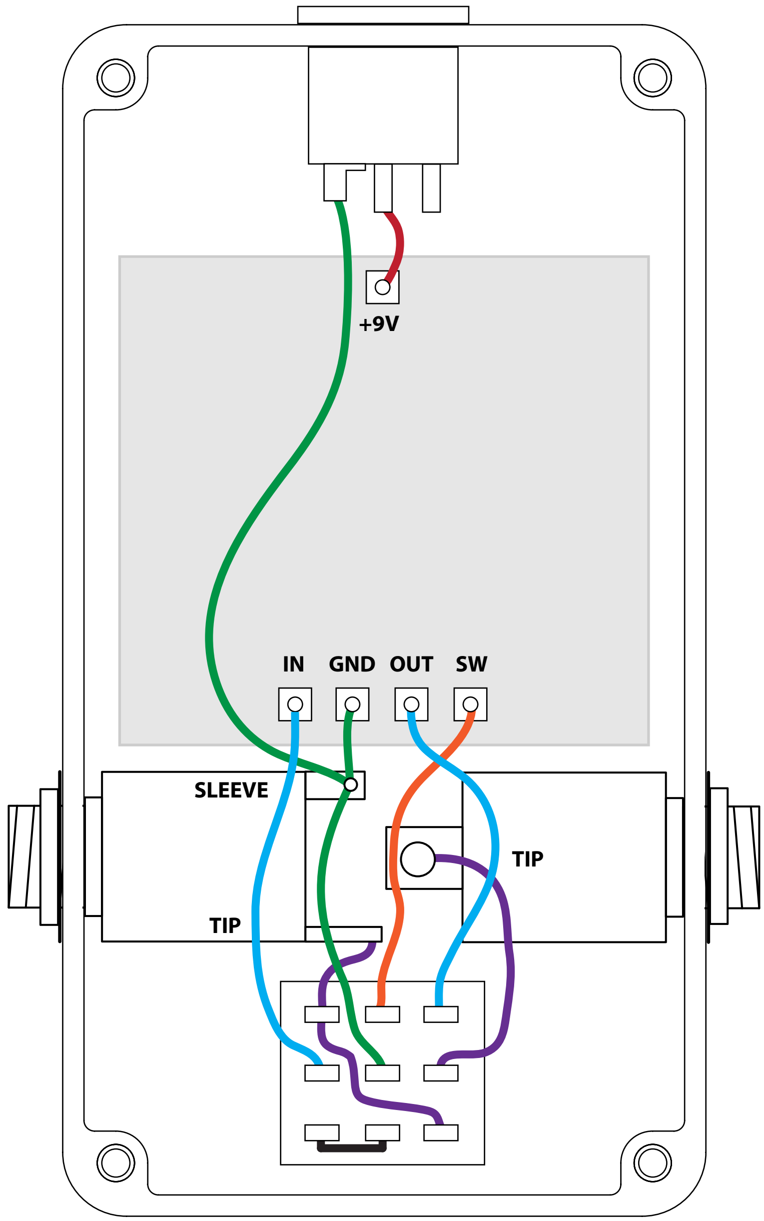
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)