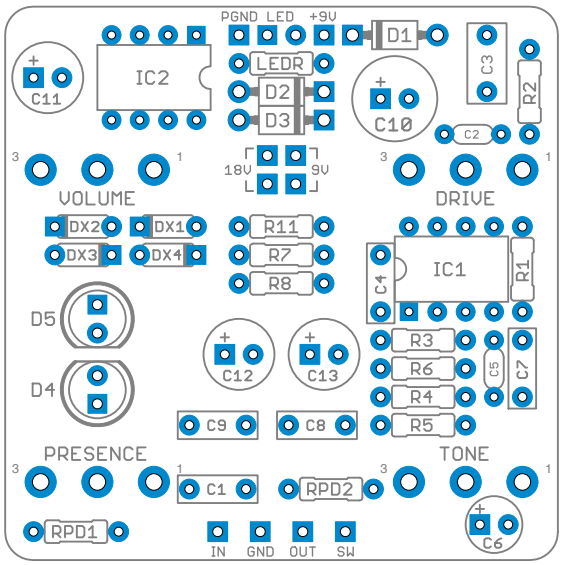
Crescent

Based on the MI Audio Crunch Box

Overview

Crescent Project Link



The Crescent Distortion circuit is inspired by the MI Audio Crunch Box, itself a heavily tweaked variant of the Marshall Bluesbreaker with some inspiration from the Fulltone® OCD®. It's solidly in the family of Marshall-like pedals and is lauded for its high-gain "stack of amps" tone. It was originally released in 2006 and has had four major versions since that time, with the most recent being named the Super Crunch Box.

The Crescent is not a direct clone or copy of any of these versions. It is most similar to a version 3, but with an added option for 18V operation using a charge pump as well as an external "Presence" control.

The Crunch Box was the inspiration for another well-known high gain pedal, the Suhr® Riot Distortion. An Aion counterpart to the Riot is available called the Fusion.

Controls & Usage

- **Drive** controls the amount of gain from the op amp that is fed through the clipping stage.
- **Volume** is the output level of the effect.
- **Tone** cuts treble.
- **Presence** affects the overall 'shape' of the tone of the pedal. It interacts with the treble control.

Modifications

The biggest modification is the addition of a charge pump circuit for 18V operation from a 9V supply. Use the jumpers on board to select either 9V or 18V mode. You can even put this on an external switch if you want, although you might need to use a bigger enclosure or mount it to the side. **You must use the jumpers to select either 9V or 18V mode or the effect will not work!**

Space on the board is provided if you want to use stacked diodes (e.g. 1N914s) instead of LEDs for clipping. There is no switch to go between them. Note that if you put both the diodes and LEDs on the board, the LEDs will have no effect on the sound because the diodes have a lower clipping threshold and will take precendence.

Parts

Resistors

R1	1M
R2	1k
R3	10k
R4	1M
R5	470R
R6	1k
R7	22k
R8	22k
R11	100R
RPD1	2M2
RPD2	1M
LEDR	4k7

Capacitors

C1	22n
C2	100pF MLCC
C3	220n
C4	100n
C5	100pF MLCC
C6	2u2 electro
C7	1n
C8	22n
C9	22n
C10	100uF electro
C11	10uF electro [1]
C12	10uF electro
C13	10uF electro

Semiconductors

IC1	RC4558P
IC2	TC1044SCPA [1]
D1	1N4002
D2-D3	1N4002 [1]
D4–D5	5mm LED diffused
DX1-DX4	1N914 [2]
LED	5mm LED

Potentiometers

Drive	100kB
Volume	100kA
Tone	10kC
Presence	25kB

Build Notes

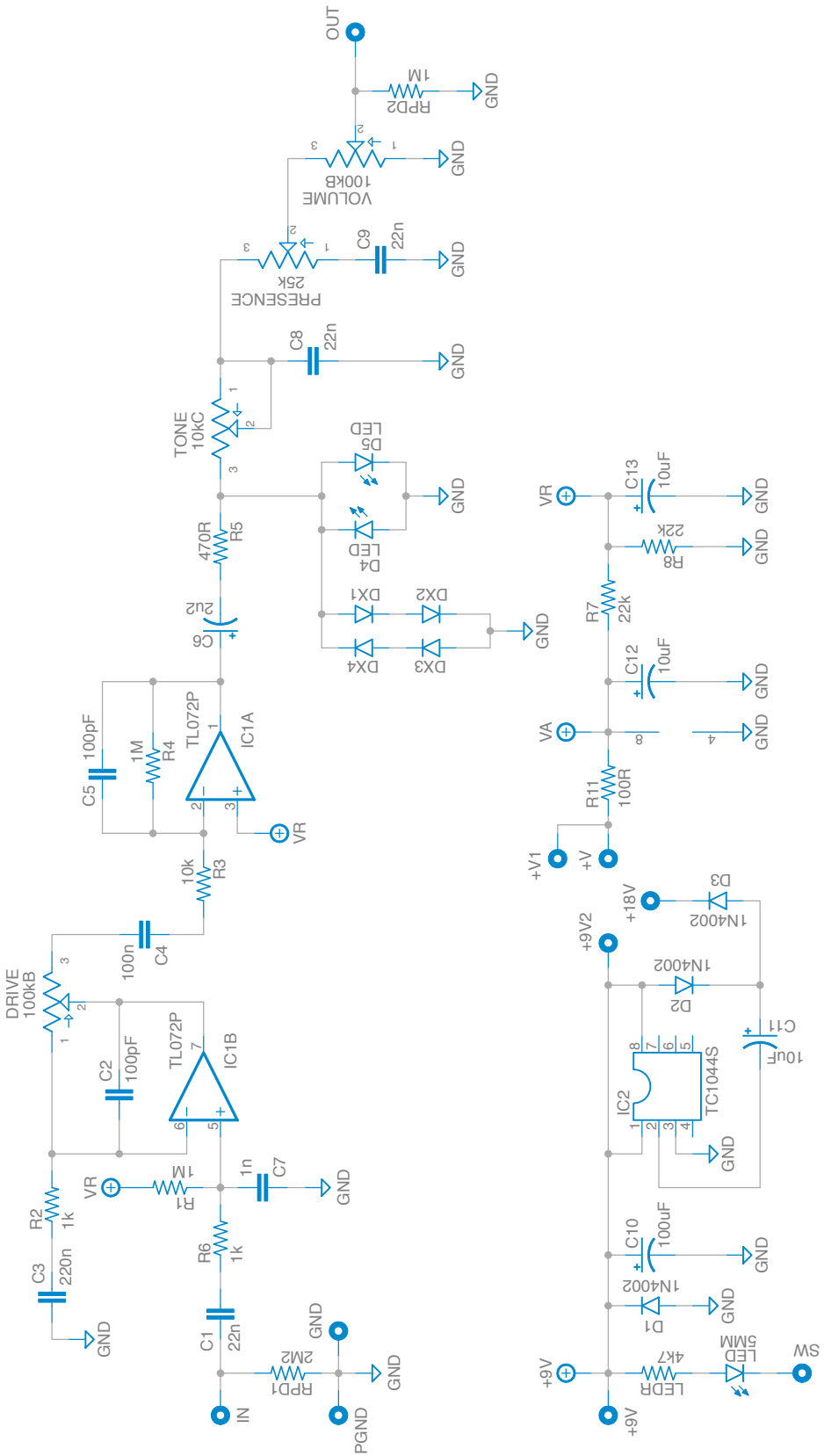
[1] **Charge pump:** If you don't want to have 18V mode available, leave off IC2, D2-D3, and C11 and jumper the "9V" pads. (You can still choose to select 9V mode via the jumpers if the charge pump is installed.)

[2] **Alternate clipping diodes:** By default, this pedal uses LEDs for a high clipping threshold. If you want a lower clipping threshold (e.g. more compressed and gainy), you can use 1N914 diodes in DX1-DX4. Note that if you use both sets of diodes, the LEDs will have no effect on the sound because the 1N914s clip the signal at a much lower threshold.

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
RPD2
1M
GND
VOLUME
100KB
PRESENCE
25K
C9
22n
C8
22n
TONE
10kC
D5
LED
D4
LED
470R
R5
C6
2u2
DX1
DX2
DX3
DX4
100pF
C5
1M
R4
TL072P
IC1A
VR
10k
R3
100n
C4
DRIVE
100kB
C2
100pF
IC1B
R2
1k
C3
220n
R1
1M
1n
C7
R6
1k
C1
22n
RPD1
2M2
IN
PGND
C13
10uF
R8
22k
R7
22k
C12
10uF
VA
R11
100R
+V1
+V
+18V
D3
1N4002
+9V2
D2
1N4002
C11
10UF
IC2
TC1044S
C10
100uF
D1
1N4002
+9V
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

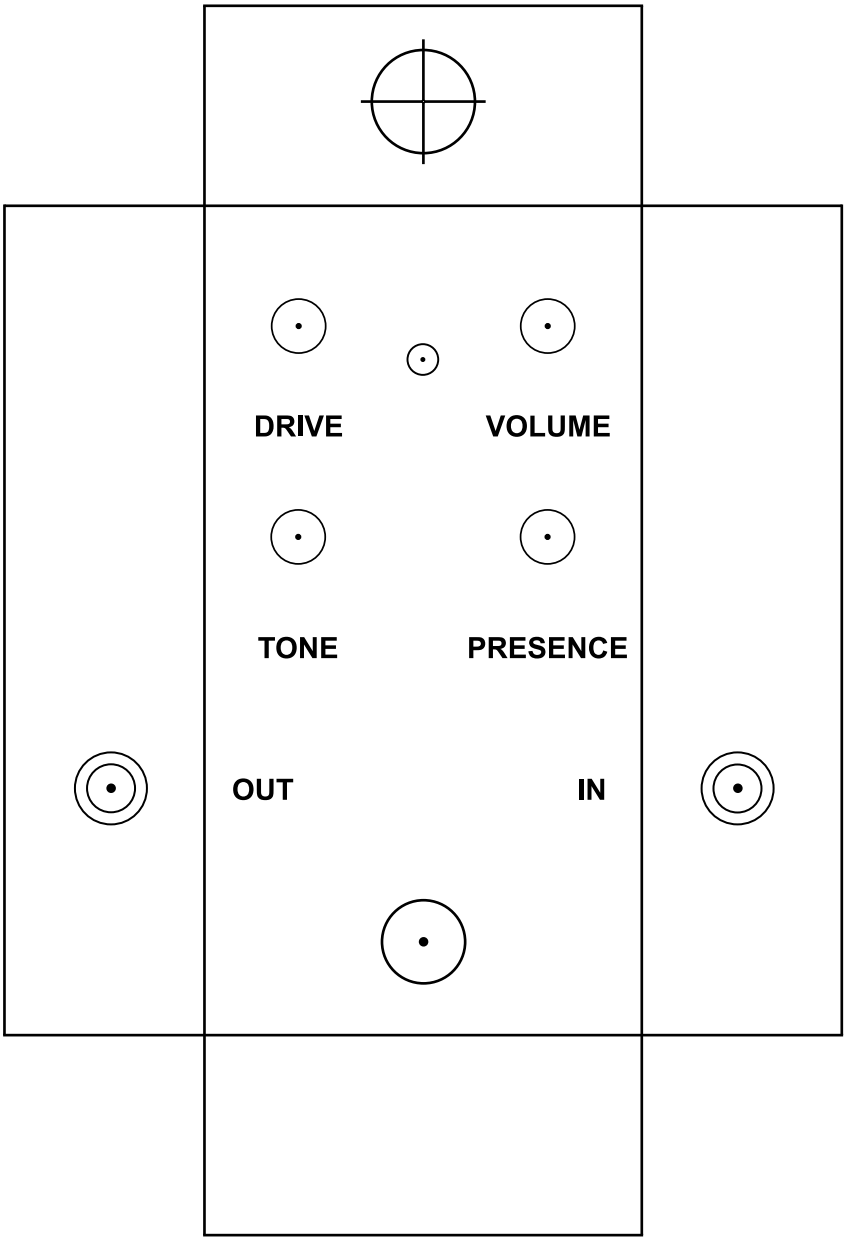
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

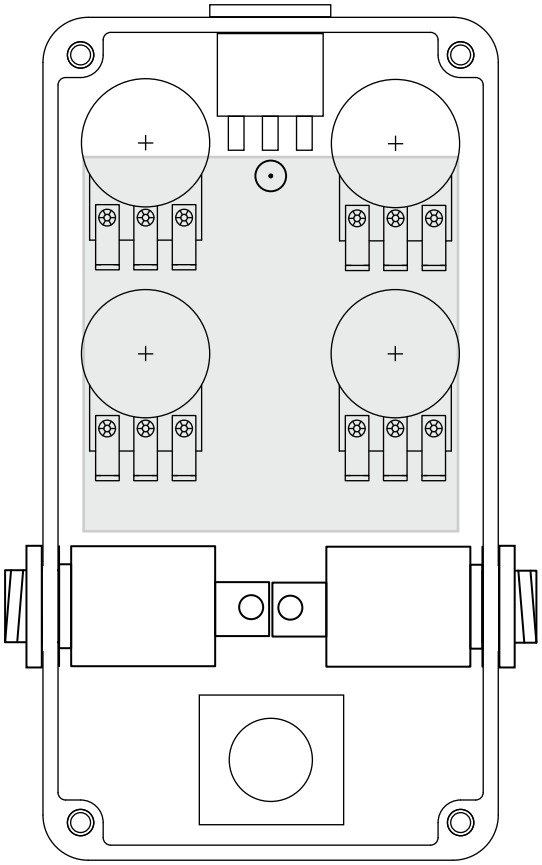
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

Drilling & Placement

Print this page and 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

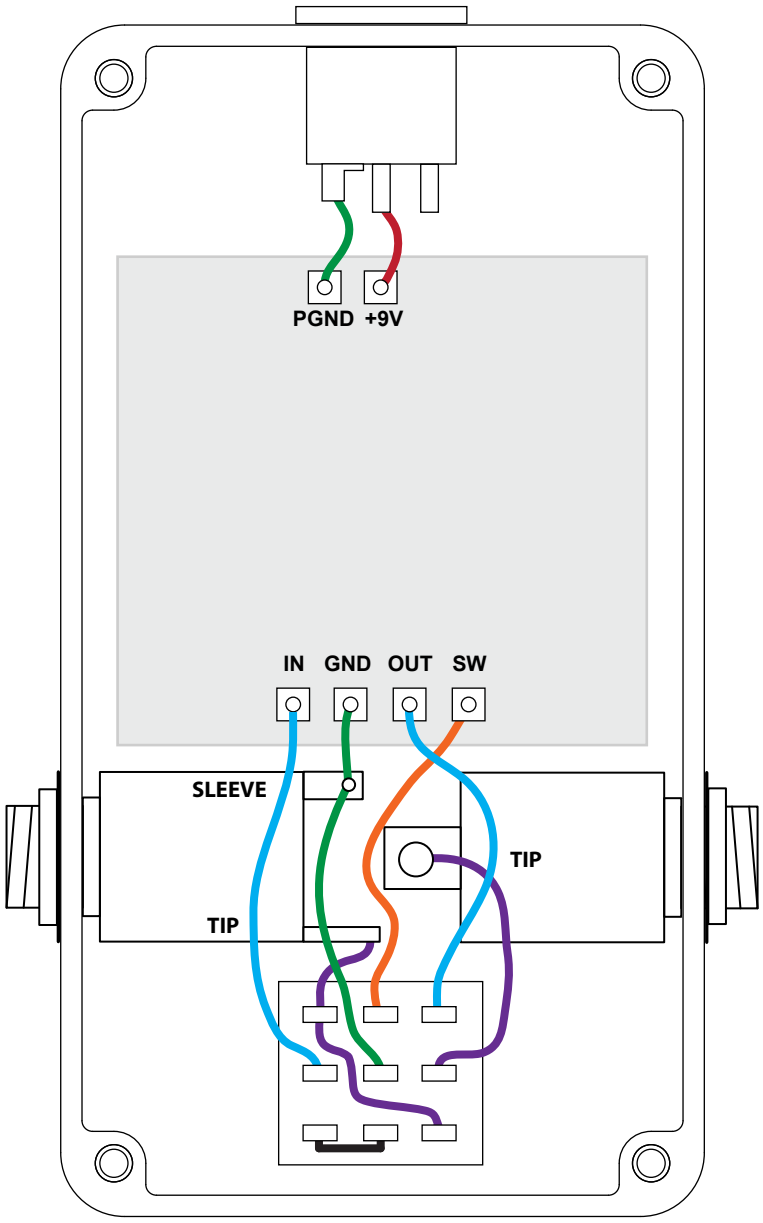
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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 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!)