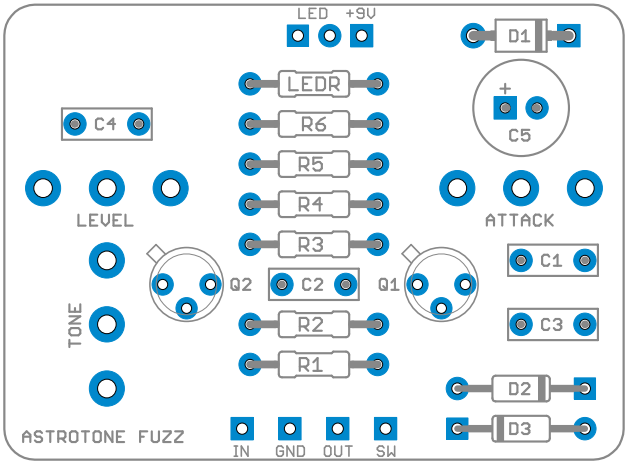
Astra

Based on the Astrotone / Sam Ash Fuzz

Overview

Astra Project Link



The Astra is a clone of the Astrotone Fuzz, produced by the Universal Amplifier Company in New York City from 1966 to 1968. It was also rebranded as the Sam Ash Fuzzz Boxx (yes, three Z's and two X's!) for sale in Sam Ash retail stores. The two units are identical.

The Astrotone circuit is one of the first silicon fuzz units, developed in an era when germanium was still the industry standard. Interestingly, it was also one of the first pedals to feature a tone knob. To be fair, it's almost universally considered to be a poor tone control design, but they get points for trying anyway. The tone control can be made better with some parts substitutions which are discussed later.

Controls & Usage

The controls for the Astrotone are straightforward:

- **Attack** sets the overall fuzz level of the circuit. It is wired as an input volume control, similar to the Superfuzz—essentially duplicating the function of the guitar's volume knob while the actual fuzz level (transistor gain) is fixed in place.
- **Tone** allows you to cut the highs of the effect.
- **Level** is the output level of the effect.

Modifications & Experimentation

Analogman released his version of the Astrotone, conveniently also called the Astrotone since the original trademark had expired. It has the exact same topology, but has several component values changed out to change the sound. So it's not an exact clone, but it's still very highly regarded. A parts list for the Analogman version is provided alongside the original version.

Even if you stick with the orginal vintage circuit, you may find it worthwhile to use Analogman's tone control modification, which has you reduce the R6 resistor to **470R**. All the tones of the original unit are still present except at the very extreme end of the range.

Parts List

Mouser Parts List (Spreadsheet)

Resistors

R1	1M [1]
R2	22k
R3	470k
R4	1M [1]
R5	1k8
R6	1k8 [1, 2]
LEDR	4k7

Capacitors

C1	47n [1]
C2	47n
C3	47n
C4	47n [1]
C5	100uF electro

Semiconductors

Q1-Q2	2N3565 [3]
D1	1N5817 [4]
D2-D3	1N914 [5]
LED	5mm LED

Potentiometers

ATTACK	100kB
TONE	10kB
LEVEL	10kB

Build Notes

[1] **Analogman values:** The Analogman version of the Astrotone uses **1M2** for R1 and R4, **470R** for R6, and **470n** for C1 and C4.

[2] **Tone control resistor value:** The original uses 1.8k here, but a near-universal complaint about the Astrotone is that the tone control barely works. If you reduce the resistor to 470R as in the Analogman version, the tone control is much more useful, whether or not you use the other Analogman values which do alter the tone.

[3] **Transistors:** The original unit used 2N3565 or 2N3566 transistors. These are still available NOS from places like Small Bear Electronics, but there is nothing particularly special about them—the 2N3904 is in the same gain range (120-180) but has better noise specifications. You can also use the 2N2222 if you want one with a more vintage appearance that is still available new from places like Mouser.

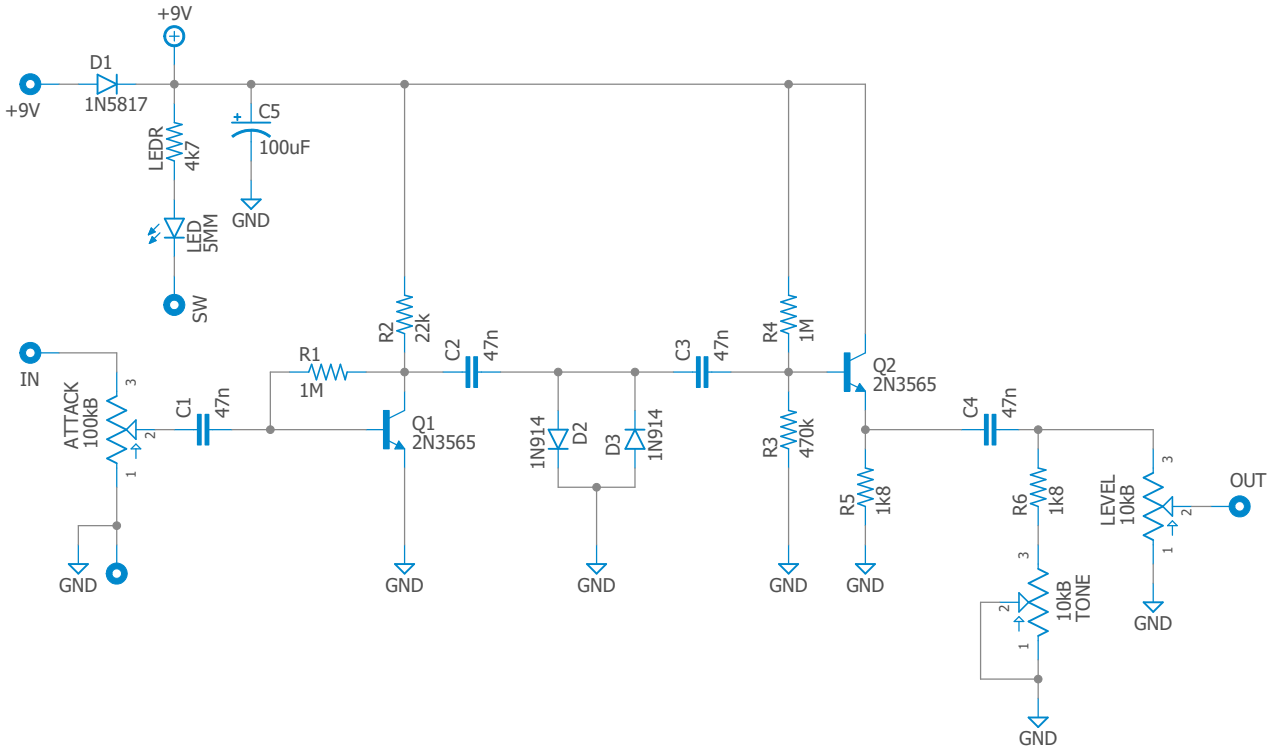
[4] **Polarity diode:** The original unit used a generic silicon diode (e.g. 1N914) for polarity protection. However, the 1N914 isn't great at power handling, so I recommend using a 1N5817 (low voltage drop) or something in the 1N400X series as a replacement.

[5] **Clipping diodes:** The original unit used unmarked diodes in glass DO-07 cases, which these days are almost exclusively associated with germanium diodes. Based on accounts from people who have traced it, though, they were actually silicon. 1N914 is the best modern replacement.

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 1uF. I prefer WIMA box film: the FKS2 series for 1n to 10n and the MKS2 series for 10n to 1uF.
- 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


+9V
D1
1N5817
+9V
LEDR
4k7
C5
100uF
GND
LED
5MM
SW
IN
ATTACK
100kB
C1
47n
R1
1M
R2
22k
C2
47n
Q1
2N3565
1N914
D2
D3
1N914
C3
47n
R4
1M
R3
470k
Q2
2N3565
R5
1k8
C4
47n
R6
1k8
10kB
TONE
LEVEL
10kB
OUT
GND

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

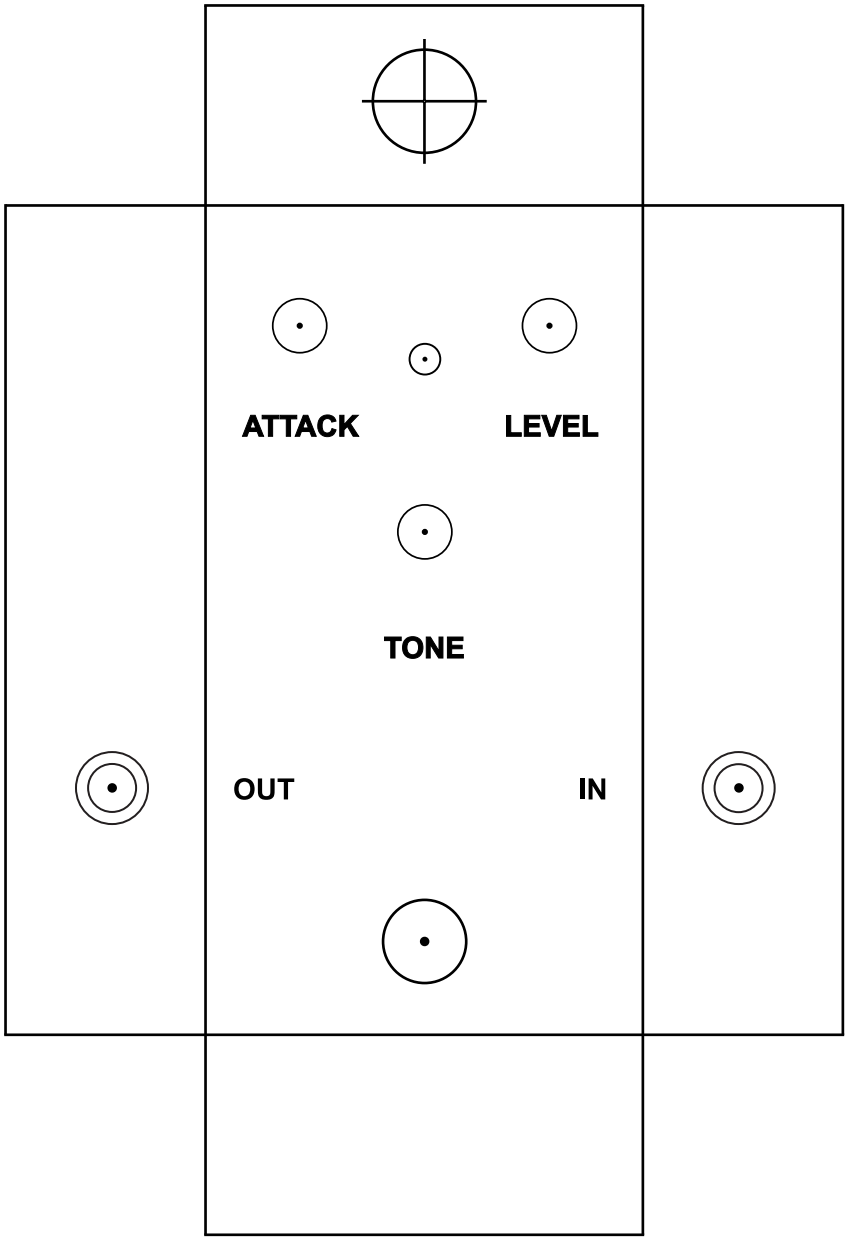
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

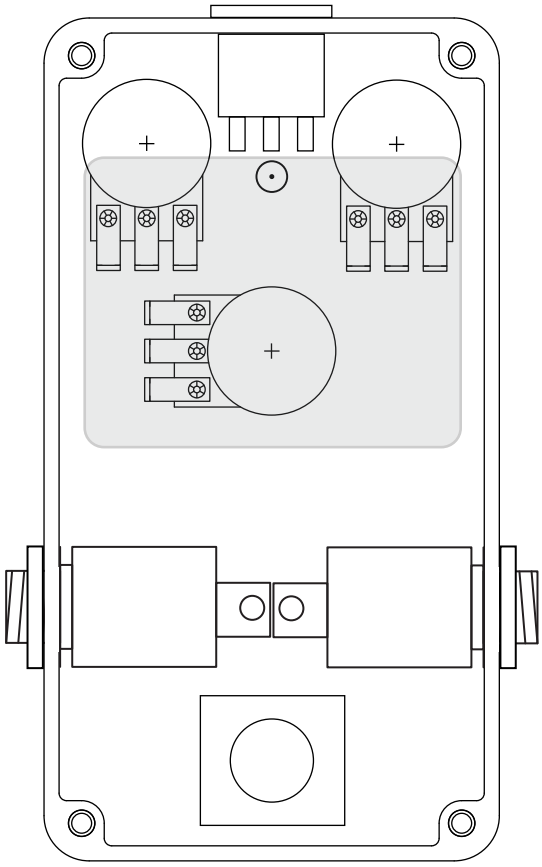
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

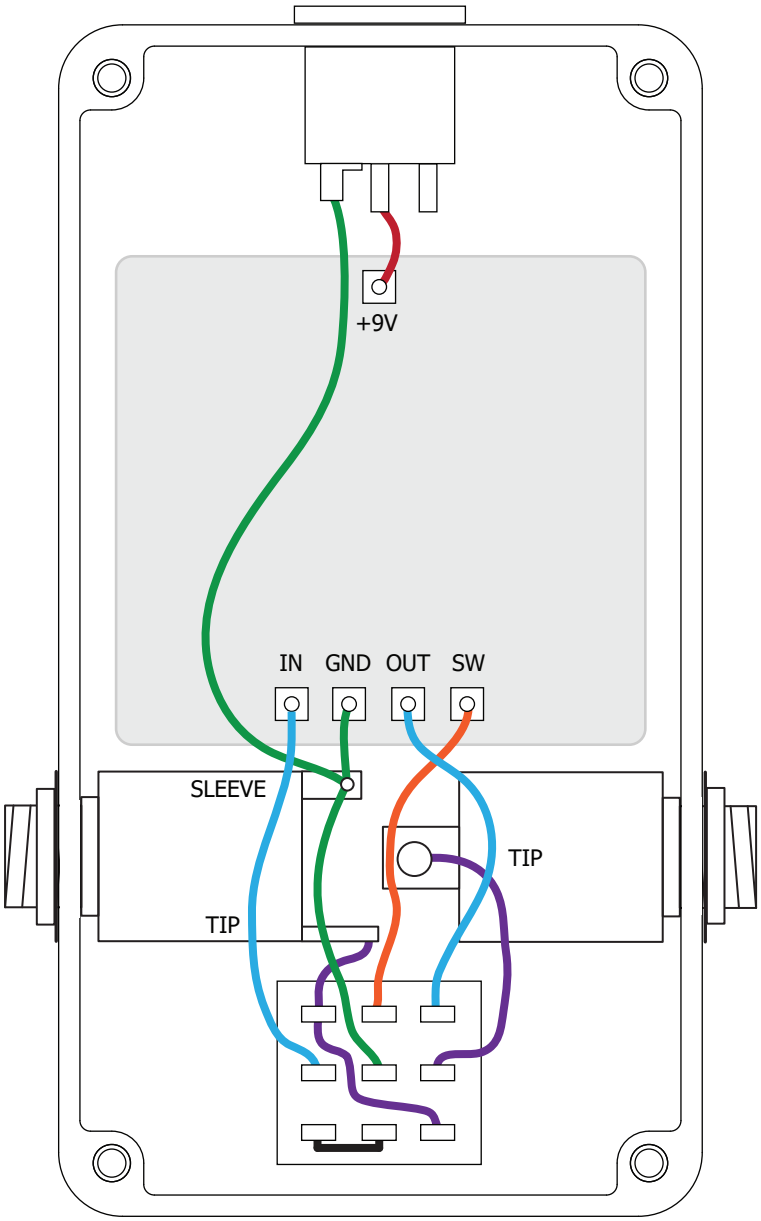
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)