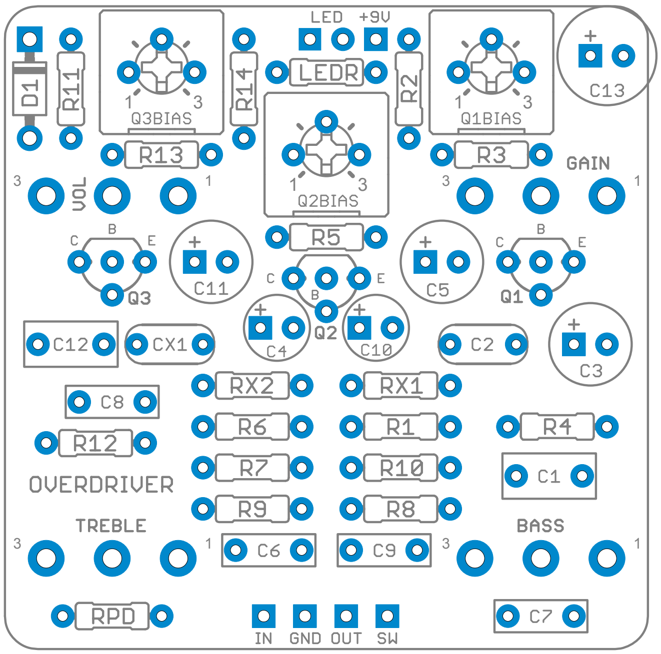
Plasma Drive

Colorsound Overdriver & Power Boost

Overview

Plasma Project Link



The Plasma Drive project is the definitive clone of the Colorsound Overdriver & Power Boost, a pair of really early silicon-based boost/drive pedals. The Power Boost was released around 1969-1970 while the Overdriver followed in 1971.

The Power Boost used two batteries for 18v operation, and while it got extremely loud, it did not have much of an overdrive effect on its own and was primarily a booster. The Overdriver dropped down to one battery for 9v operation, and along with a few other circuit changes, the lower headroom caused it to distort, but without the use of op-amps or diodes like later overdrives. Its sound is probably closer to a fuzz or a Big Muff, but with discrete Bass and Treble controls in a configuration known as a Baxandall tone stack.

This PCB will allow you to build either version, although there is not a charge pump onboard, so you'll have to provide your own 18v supply for the Power Boost variant.

Controls & Usage

The Overdriver's controls are as follows:

- **Gain** controls the amount of gain from the first transistor stage.
- **Treble** controls the high-end response of the circuit.
- **Bass** controls the low-end response of the circuit.
- **Volume** controls the overall output of the effect.

Versions

This PCB will allow you to build either a Power Boost or an Overdriver, which are nearly identical in topology yet with a couple of frustrating differences for a unified layout. **Make sure to pay attention to the build notes** for the version you're building, since in both versions you'll need to jumper some resistors and omit others.

Sound

This circuit is well-known for having a "fizzy" decay in the medium-gain settings. This is just what discrete transistors sound like when they distort, and it's one of the characteristics of this circuit, having no clipping diodes. Most people will use it as either a tone-shaper on low gain settings or an all-out fuzz near the top end of the gain range.

Biasing

See page 4 for biasing information.

Overdriver Parts List

Capacitors

C1	220n film
C2	220pF MLCC
C3	22µF electro [1]
C4	10µF electro
C5	22µF electro
C6	10n film
C7	100n film
C8	100n film
C9	10n film
C10	10µF electro
C11	22µF electro
C12	220n film
C13	47µF electro
CX1	100pF MLCC [2]

Resistors

R1	150k
R2	120k
R3	3k3
R4	470R
R5	1k
R6	12k
R7	4k7
R8	39k
R9	5k6
R10	4k7
R11	150k
R12	33k
R13	1k
R14	470R
RX1	(omit)
RX2	jumper
RPD	1M to 2M2
LEDR	4k7

Semiconductors

Q1-Q3	BC109 [3]
D1	1N4002
LED	5MM

Potentiometers

Gain	1kC [4]
Volume	100kA [5]
Treble	100kB
Bass	100kB
Q1 Bias	10kB trim (3362P) [6]
Q2 Bias	10kB trim (3362P) [6]
Q3 Bias	10kB trim (3362P) [6]

[1] **Alternate value:** Some original Overdriver units used **10µF** for C3. There's no difference in sound.

[2] **Optional:** This feedback capacitor is not in the original unit, but it helps smooth out some of the harsh edges and can also cut down on high-frequency hiss. Anything from 100pF to 470pF would be effective here.

[3] **Transistors:** The original Colorsound unit came with BC169C, BC184C and BC109 transistors. Any silicon transistor should work fine here since the gain is limited via feedback resistance—try a 2N5088 or 2N3904.

[4] **Alternate value:** The original unit used a **10kB** pot here, but the listed **1kC** (reverse audio) taper will give a much more usable control range. From 0 to 9 on the original, there was almost no change, with all of the gain change taking place in the last 10% of the rotation. With 1kC, you're essentially controlling just that last 10%.

[5] **Volume control:** The original unit did not have a volume control on the output, relying only on the gain control to determine the output level. As a result, they were legendary for being extremely loud. Do yourself a favor and build it with the volume control—but if you do want to leave it off, just jumper **pads 2 and 3** of the pot.

[6] **Trimmers:** You can use either the 1/4" 3362P or the larger 3/8" 3386P trimmers here. The screenprint shows the larger size, but the pin spacing is the same for both. **See page 4 for biasing information.**

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.

Power Boost Parts List

Capacitors

C1	220n film
C2	220pF MLCC
C3	22µF electro
C4	4µ7 electro [1]
C5	22µF electro
C6	10n film
C7	100n film
C8	100n film
C9	10n film
C10	22µF electro
C11	4µ7 electro [2]
C12	4µ7 electro [1]
C13	47µF electro
CX1	100pF MLCC [3]

Resistors

R1	150k
R2	120k
R3	4k7
R4	1k2
R5	1k8
R6	omit
R7	4k7
R8	39k
R9	5k6
R10	4k7
R11	180k
R12	33k
R13	3k9
R14	1k
RX1	12k
RX2	470R
RPD	1M to 2M2
LEDR	4k7

Semiconductors

Q1-Q3	BC109 [4]
D1	1N4002
LED	5MM

Potentiometers

Gain	1kC [5]
Volume	100kA [6]
Treble	100kB
Bass	100kB
Q1 Bias	5kB trim (3362P) [7]
Q2 Bias	10kB trim (3362P) [7]
Q3 Bias	10kB trim (3362P) [7]

[1] **Alternate value:** The first version of the Power Boost used **4µ7** for both C4 and C12, while later versions used **10µF** for C4 and **220n** for C12—the same values as the Overdriver.

[2] **Alternate value:** The first version of the Power Boost used **4µ7** for C11, while later versions used **10µF**.

[3] **Optional:** This feedback capacitor is not in the original unit, but it helps smooth out some of the harsh edges and can also cut down on high-frequency hiss. Anything from 100pF to 470pF would be effective here.

[4] **Transistors:** The original Colorsound unit came with BC169C, BC184C and BC109 transistors. Any silicon transistor should work fine here since the gain is limited via feedback resistance—try a 2N5088 or 2N3904.

[5] **Alternate value:** The original unit used a **10kB** pot here, but the listed **1kC** (reverse audio) taper will give a much more usable control range. From 0 to 9 on the original, there was almost no change, with all of the gain change taking place in the last 10% of the rotation. With 1kC, you're essentially controlling just that last 10%.

[6] **Volume control:** The original unit did not have a volume control on the output, relying only on the gain control to determine the output level. As a result, they were legendary for being extremely loud. Do yourself a favor and build it with the volume control—but if you do want to leave it off, just jumper **pads 2 and 3** of the pot.

[7] **Trimmers:** You can use either the 1/4" 3362P, or the larger 3/8" 3386P trimmers here. The screenprint shows the larger size, but the pin spacing is the same for both. **See page 4 for biasing information**.

Note: The Power Boost was designed to run on 18 volts, with the original using two batteries in series. You'll have to provide your own 18v supply for this build. Make sure all of your electrolytic capacitors are rated for at least 25v. You may also need to use higher value for the LED resistor since it will be brighter at 18v.

Additional Part Notes

(See previous page.)

Biasing

All three transistors are biased individually with their own trim pots. To start, turn the trimmers all the way down and measure the far-left pad of the transistor (marked "C" for collector) with a multimeter, using the red probe with the black probe touching ground. Turn the trimmer up until the collector voltage reaches your target.

Q1

Q1 is an interesting one: in standard 9V operation it's between 2V and 3V on the collector, and the bias trimmer doesn't actually change the collector voltage. I can't really give you a good target voltage, but just know that the original unit had 6.8k total resistance here, and the Throbak biases at around 5k. So, somewhere between 4k and 8k is going to be the sweet spot. I recommend using a **3.3k** resistor for R3 to set the minimum resistance, and then using a 5k pot for the Q1 bias. If a 5k pot is turned halfway up, then, the total resistance would be 3.3k + 2.5k = 5.8k.

Q2 & Q3

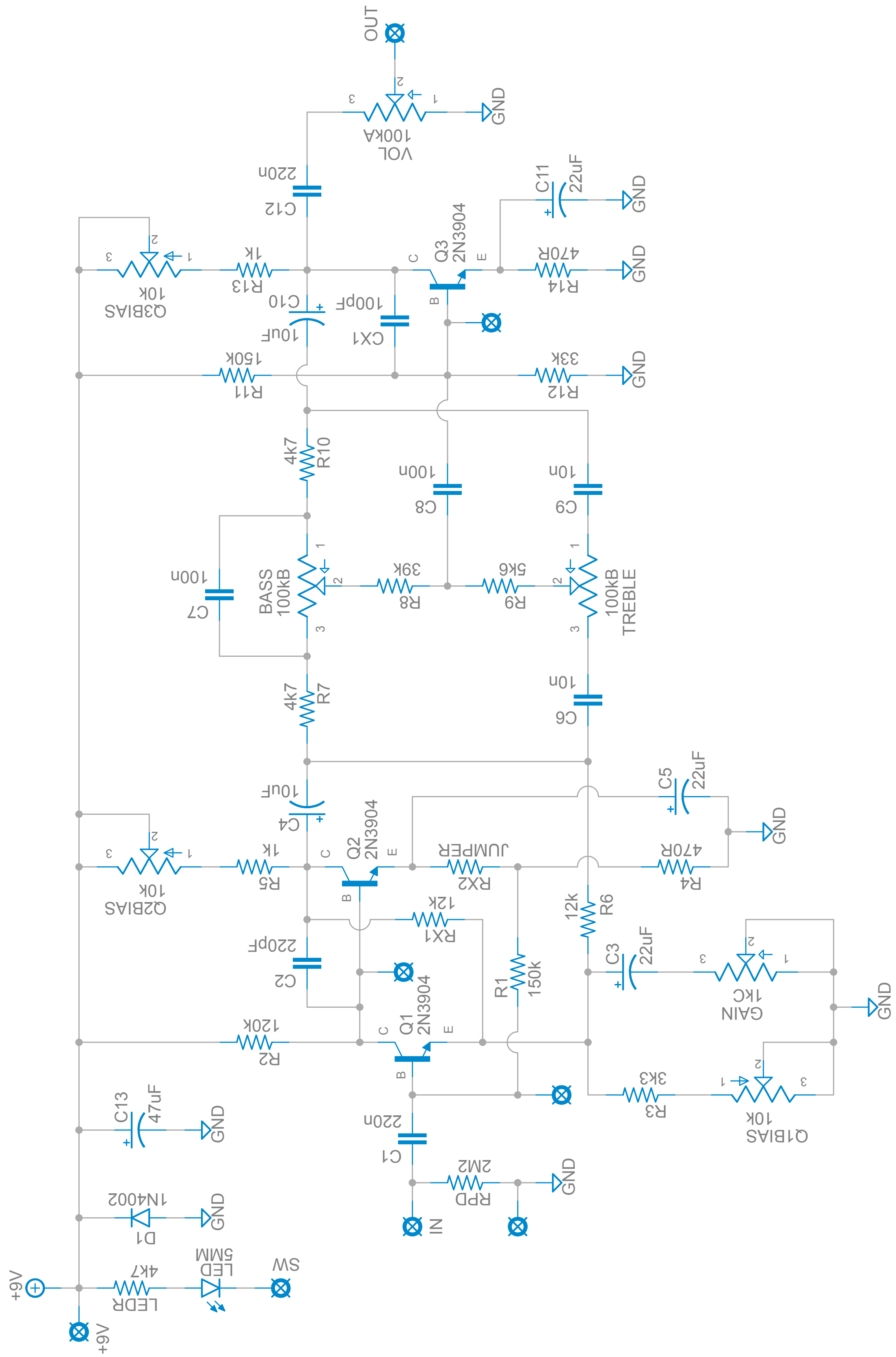
For 9-volt operation, the optimum collector voltage for Q2 and Q3 is right around **5V**, close to half the supply voltage (around 9.6V if using a typical wall wart adapter). This is the value listed on the original Colorsound schematic. I should mention that one commercial clone, the Throbak Overdrive Boost, intentionally sets the Q3 voltage to around 6.5V, and it's possible some of the Overdrivers had biases this high—factory schematics are not always accurate to the real thing. So, you might try both 5V and 6.5V on Q3 and see if you like one better. I recommend using **1k** resistors for R5 and R13 to set the minimum bias resistance.

For the 18V Power Boost, just double the recommended voltages: around 5V for Q1 and 10V for Q2 and Q3. Don't worry about getting it exact; it's not as sensitive as vintage fuzzes like Fuzz Faces and Tonebenders.

Omitting the bias trimmers

If you don't want to use the bias trimmers, the original unit uses different resistor values than I listed. For the Overdriver, use **6k8** for R3, and **1k8** for R5 and R13. For the Power Boost use **4k7** for R3, **1k8** for R5 and **3k9** for R13. You'll also want to **jumper pins 1 & 2** of each of the trimmers in order to bypass them.

Schematic


OUT
VOL
100KA
GND
C12
220n
C11
22uF
Q3
2N3904
R14
470R
R13
1K
Q3BIAS
10K
C10
10uF
CX1
100pF
R11
150K
R12
33K
R10
4k7
C8
100n
C9
10n
C7
100n
BASS
100kB
R8
39K
R9
5K6
TREBLE
100kB
R7
4k7
C6
10n
C4
10uF
C5
22uF
Q2
2N3904
Q2BIAS
10K
R5
1K
RX2
JUMPER
R4
470R
R6
12k
RX1
12K
C2
220pF
C3
22uF
GAIN
1KC
R1
150k
Q1
2N3904
R2
120K
R3
3K3
Q1BIAS
10K
C13
47uF
C1
220n
RPD
2M2
IN
D1
1N4002
LEDR
4K7
LED
5MM
SW
+9V

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

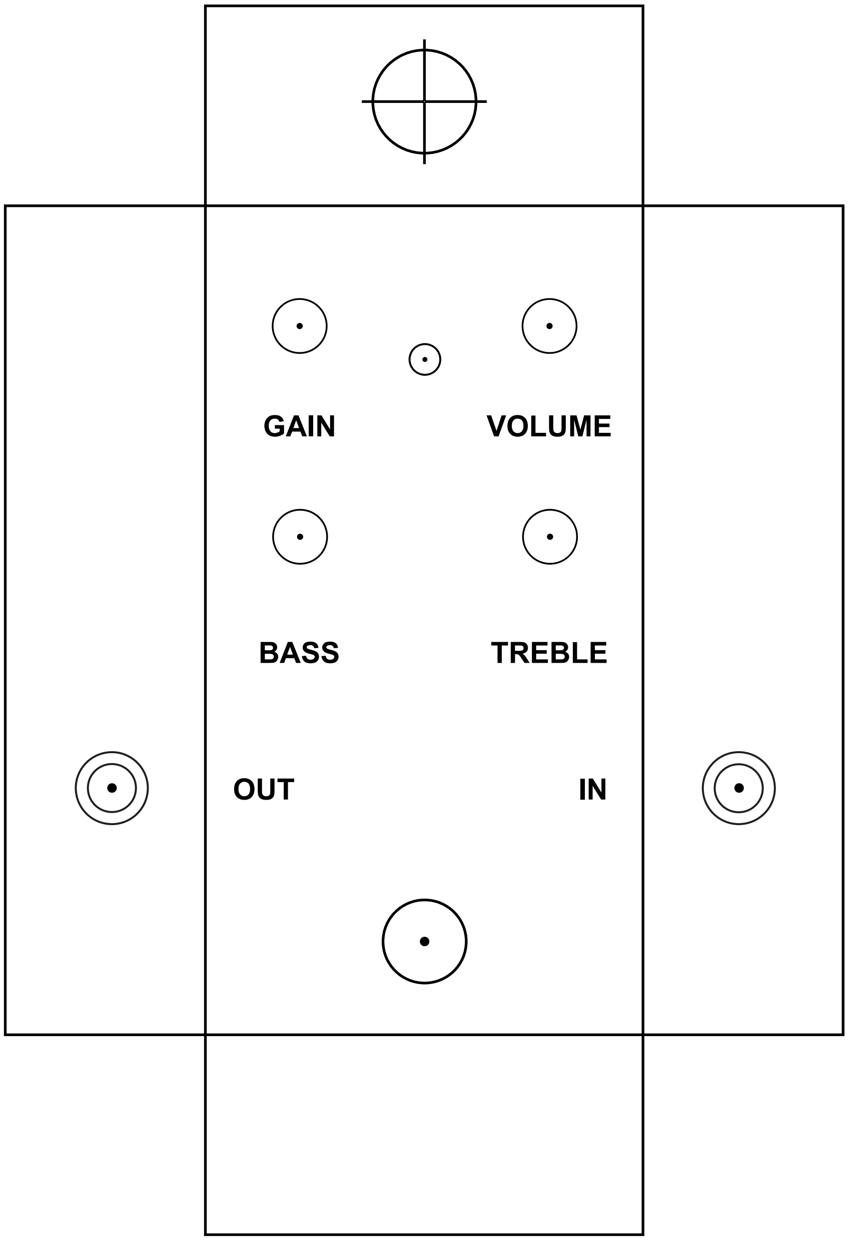
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

Sockets

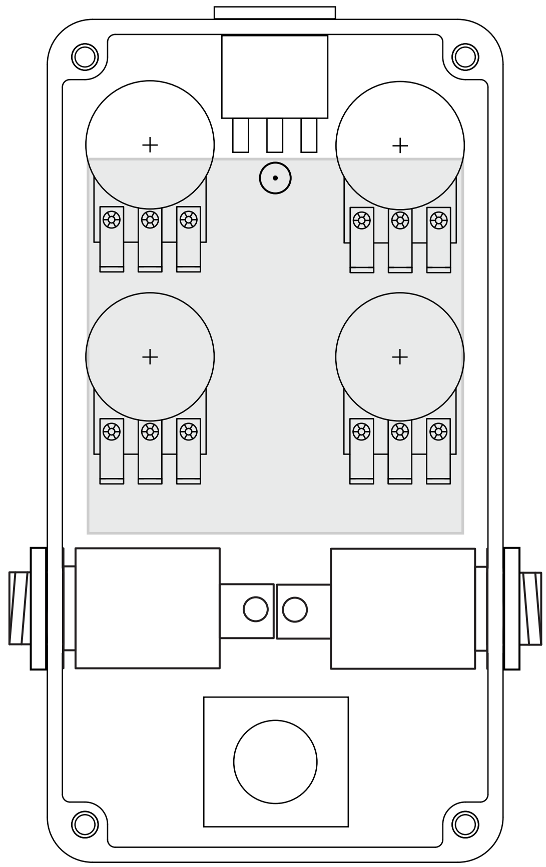
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

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

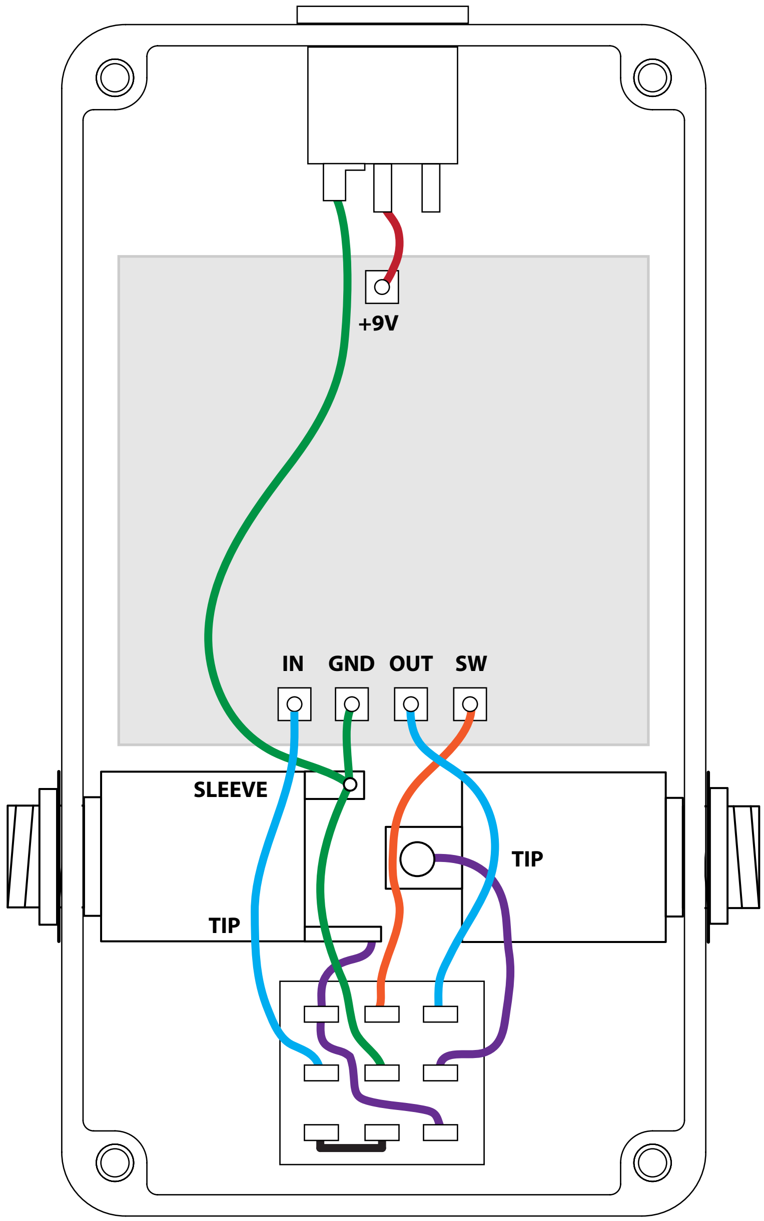
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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 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!)