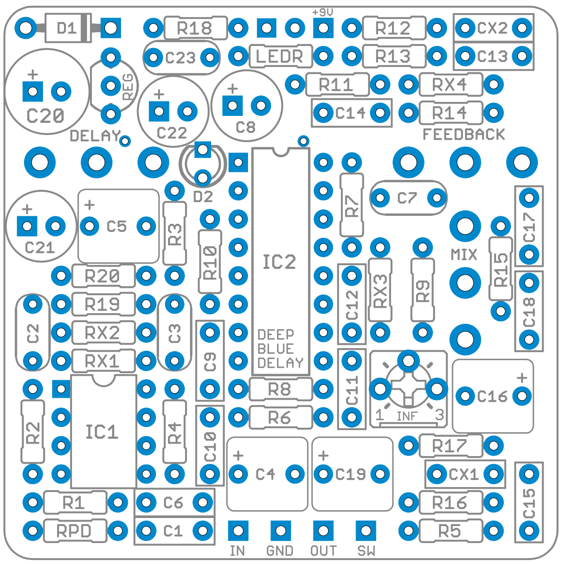
# Vector

*Based on the Deep Blue Delay*

aion electronics

## Overview

Vector Project Link




The Vector Delay is an easy delay project based on the Mad Professor Deep Blue Delay as well as the Tonepad Rebote 2.5, with a few modifications and improvements added. You can build either version using this PCB and parts lists are supplied for both.

The circuit is based around the PT2399 delay chip. While the PT2399 is technically digital, it doesn't have the sterile sound characteristics associated with modern digital delays, but is instead much warmer and analog-voiced.

Your ears can be the final judge, but even most analog purists will give high praise to this circuit. With the overall complexity being much lower and the cost of parts being far cheaper than a BBD-based delay, you owe it to yourself to try this one before jumping straight into a true analog delay!

## Modifications & Experimentation

This circuit has a few additions to allow either the Deep Blue Delay or Rebote 2.5 circuit to be built on a single layout. On pages 2 and 3 of this document, you can find the parts lists for either version.

An internal **infinite repeats trimmer** is included to allow you to set the upper end of the feedback range, either allowing infinite repeats at the Feedback knob's highest position or stopping just short of it, depending on your preference. To calibrate it, turn the Feedback knob all the way up and then adjust the trimmer until you have the range you want. The stock Deep Blue Delay circuit has the equivalent of having this trimmer turned all the way CCW (5.1k total resistance), while the Rebote 2.5 circuit has the equivalent of this trimmer turned all the way CW (15k total resistance). For infinite repeats without getting into "runaway" territory, you'll want the trimmer about halfway up (10-12k total resistance). I recommend that position as a starting point.

An optional **current-control LED** is included to prevent the PT2399 from going into clipping with large input signals (around 20dB or higher), as discussed in this thread on DIYStompboxes. For D2, you'll want to use a water-clear green LED with a forward voltage of 2V to 2.3V for best results—the goal being to get as high of a clipping threshold as you can get without reaching the point where the PT2399 will clip, which is harsh and un-musical.

## Parts (Deep Blue Delay variant)

**Mouser Parts List (Spreadsheet)**

### Resistors

| | |
|---|---|
| R1 | 180k |
| R2 | 360k |
| R3 | 22k |
| R4 | 12k |
| R5 | 1k |
| R6 | 100k |
| R7 | 10k |
| R8 | 10k |
| R9 | 10k |
| R10 | 2k7 |
| R11 | 10k |
| R12 | 10k |
| R13 | 20k |
| R14 | 1k |
| R15 | 2k |
| R16 | 5k1 |
| R17 | 20k |
| R18 | 33R |
| R19 | 10k |
| R20 | 10k |
| RPD | 2M2 |
| RX1 | JUMPER |
| RX2 | JUMPER |
| RX3 | JUMPER |
| RX4 | JUMPER |
| LEDR | 4k7 |

### Capacitors

| | |
|---|---|
| C1 | 22n |
| C2 | 47pF MLCC |
| C3 | 100pF MLCC |
| C4 | 1uF |
| C5 | 1uF |
| C6 | 4n7 |
| C7 | 2n2 |
| C8 | 47uF |
| C9 | 100n MLCC |
| C10 | 100n MLCC |
| C11 | 100n |
| C12 | 100n |
| C13 | 15n |
| C14 | 2n2 |
| C15 | 10n |
| C16 | 1uF |
| C17 | 47n |
| C18 | 22n |
| C19 | 1uF |
| C20 | 100uF |
| C21 | 47uF |
| C22 | 47uF |
| C23 | 100n |
| CX1 | OMIT |
| CX2 | OMIT |

### Semiconductors

| | |
|---|---|
| IC1 | TL072 [1] |
| IC2 | PT2399 [2] |
| D1 | 1N4002 |
| D2 | 3mm LED [3] |
| REG | 78L05 (5V) |
| LED | 5mm LED |

### Potentiometers

| | |
|---|---|
| Delay | 50kB |
| Feedback | 50kB |
| Mix | 50kB |
| Infinite | 10kB trim (3362P) [4] |

## Parts (Rebote 2.5 variant)

### Resistors

| | |
|---|---|
| R1 | 510k |
| R2 | 510k |
| R3 | 24k |
| R4 | 24k |
| R5 | JUMPER |
| R6 | 100k |
| R7 | 12k |
| R8 | 12k |
| R9 | 47k |
| R10 | 1k |
| R11 | 12k |
| R12 | 12k |
| R13 | 24k |
| R14 | 2k7 |
| R15 | JUMPER |
| R16 | 5k1 |
| R17 | 33k |
| R18 | OMIT |
| R19 | 12k |
| R20 | 12k |
| RPD | 1M |
| RX1 | 240k |
| RX2 | 1k |
| RX3 | 12k |
| RX4 | 12k |
| LEDR | 4k7 |

### Capacitors

| | |
|---|---|
| C1 | 82n |
| C2 | 4.7pF MLCC [5] |
| C3 | 47pF MLCC [6] |
| C4 | 1uF |
| C5 | 1uF |
| C6 | 10n |
| C7 | 560pF MLCC |
| C8 | 47uF |
| C9 | 100n MLCC |
| C10 | 100n MLCC |
| C11 | 82n |
| C12 | 82n |
| C13 | 10n |
| C14 | 1n |
| C15 | 10n |
| C16 | 1uF |
| C17 | OMIT |
| C18 | 100n |
| C19 | 1uF |
| C20 | 100uF |
| C21 | 47uF |
| C22 | 47uF |
| C23 | 100n |
| CX1 | 27n |
| CX2 | 10n |

### Semiconductors

| | |
|---|---|
| IC1 | TL072 [1] |
| IC2 | PT2399 [2] |
| D1 | 1N4002 |
| D2 | 3mm LED [3] |
| REG | 78L05 (5V) |
| LED | 5mm LED |

### Potentiometers

| | |
|---|---|
| Delay | 50kB |
| Feedback | 25kB |
| Mix | 100kB |
| Infinite | 10kB trim (3362P) [4] |

## Build Notes

[1] **Op-amp:** You can substitute any 8-pin dual op-amp for the TL072. A higher-fidelity op-amp such as the **OPA2134** would be a good choice.

[2] **PT2399 chip:** Make sure you get it from a reputable source such as Small Bear Electronics. You can find them from places like Tayda and eBay for much cheaper, but more often than not, these are fakes or factory rejects. As a result, they may behave oddly, have higher noise, or just not work at all.

[3] **Current-control LED:** This should be a **green water-clear LED** with around 2-2.3V forward voltage. It is optional, but recommended. It prevents the PT2399 chip from going into clipping when large input signals are present (around 20dB and higher).

[4] **Optional trimmer:** This trimmer allows you to set the maximum range of the Feedback control, allowing infinite repeats at the maximum setting. It should be calibrated by ear if used. (See page 1 for more information on calibrating it.) If you want to omit the trimmer, jumper pins 1 and 2. For the Rebote circuit, if the trimmer is omitted, use a **15k** resistor for R16 instead of the listed 5k1.

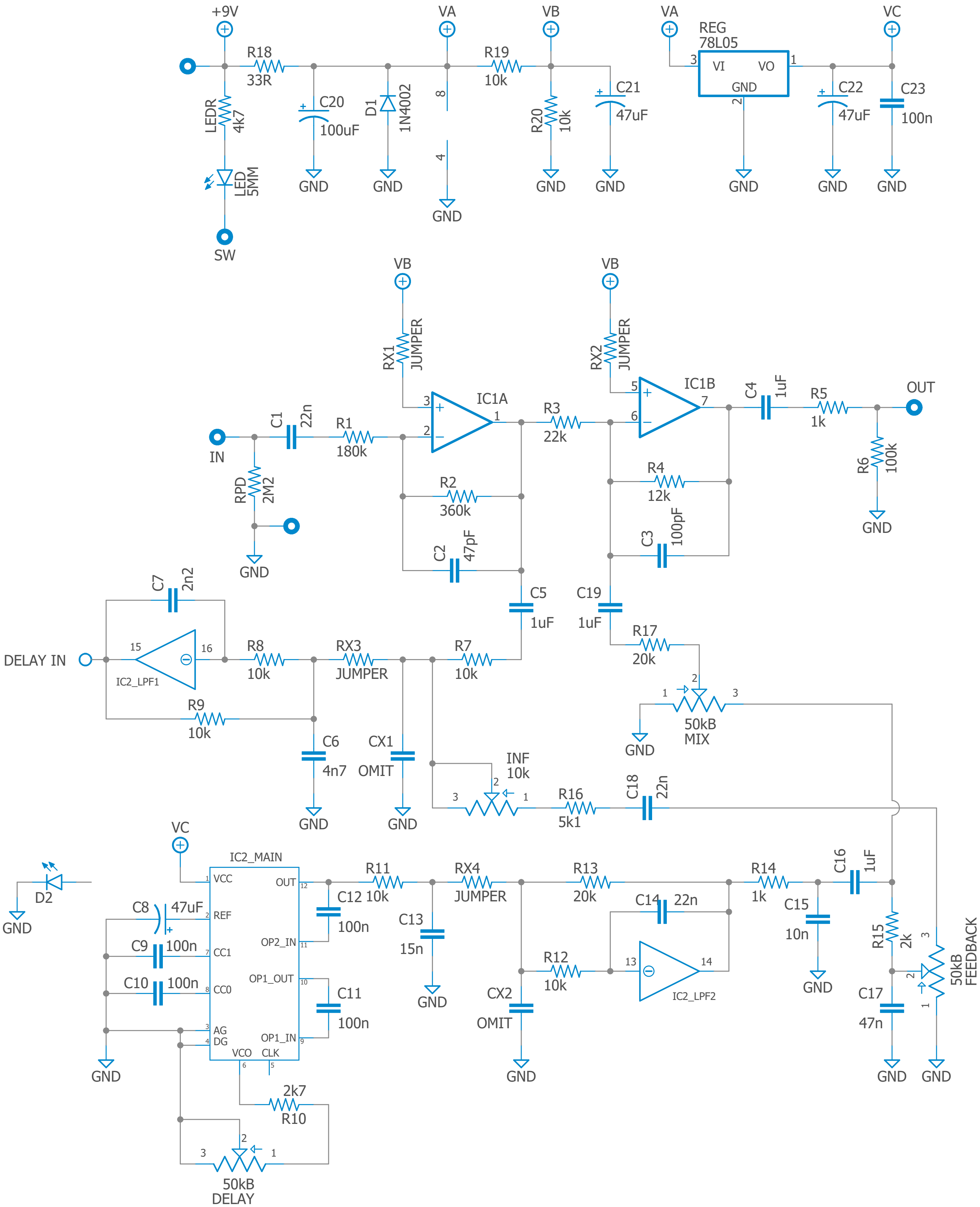
[5] **Substitute value:** The original Rebote 2.5 lists a **5pF** capacitor here, but 4.7pF is much easier to find.

[6] **Substitute value:** The original Rebote 2.5 lists a **51pF** capacitor here, but 47pF is much easier to find.

## 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
R18
33R
LEDR
4k7
LED
5MM
SW
C20
100uF
D1
1N4002
VA
8
4
R19
10k
VB
R20
10k
C21
47uF
REG
78L05
VI
VO
GND
C22
47uF
VC
C23
100n
GND
RX1
JUMPER
RX2
JUMPER
IN
C1
22n
R1
180k
RPD
2M2
IC1A
R2
360k
C2
47pF
R3
22k
IC1B
C4
1uF
R5
1k
OUT
R6
100k
R4
12k
C3
100pF
C5
1uF
C19
1uF
R17
20k
50kB
MIX
DELAY IN
IC2_LPF1
C7
2n2
R8
10k
RX3
JUMPER
R7
10k
R9
10k
C6
4n7
CX1
OMIT
INF
10k
R16
5k1
C18
22n
D2
IC2_MAIN
VCC
REF
CC1
CC0
AG
DG
VCO
CLK
OUT
OP2_IN
OP1_OUT
OP1_IN
C8
47uF
C9
100n
C10
100n
R11
10k
C12
100n
C11
100n
C13
15n
RX4
JUMPER
R13
20k
CX2
OMIT
R12
10k
C14
22n
IC2_LPF2
R14
1k
C15
10n
C16
1uF
R15
2k
C17
47n
50kB
FEEDBACK
2k7
R10
50kB
DELAY

## 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

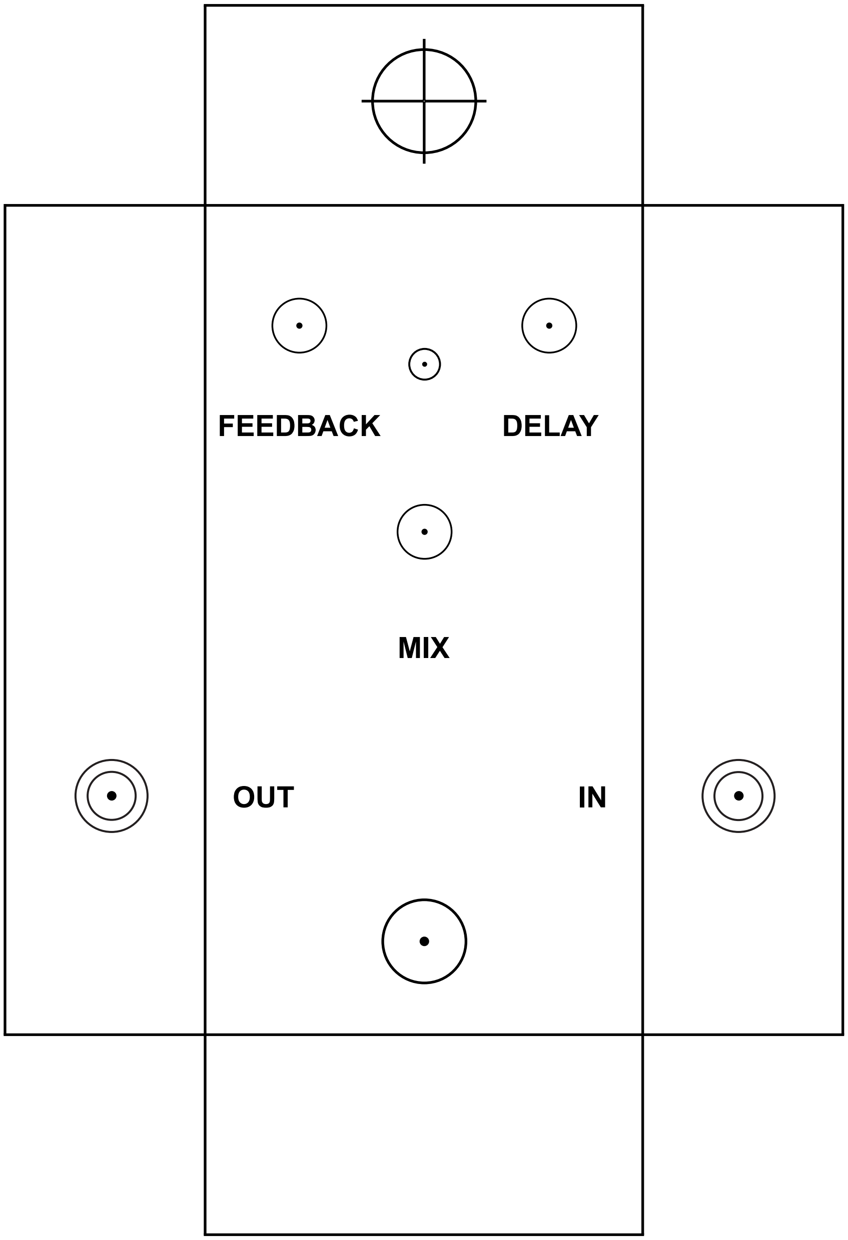
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

### Sockets

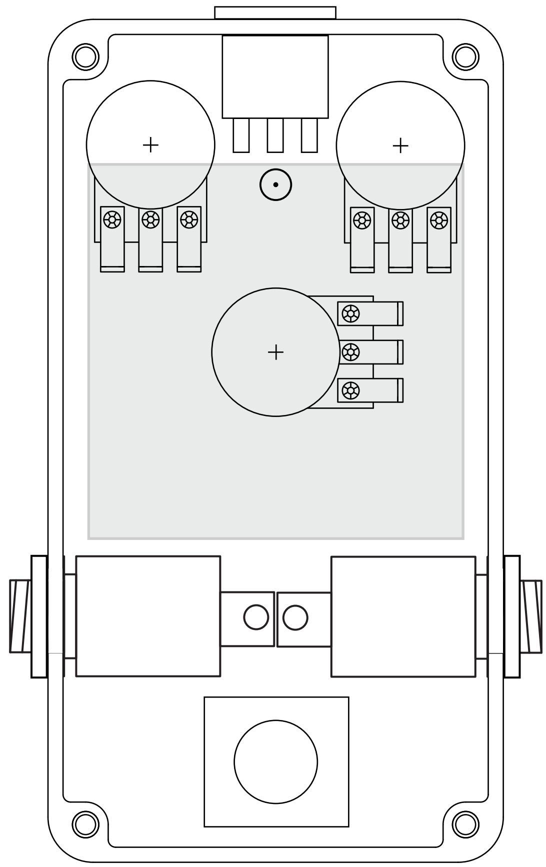
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

## 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.



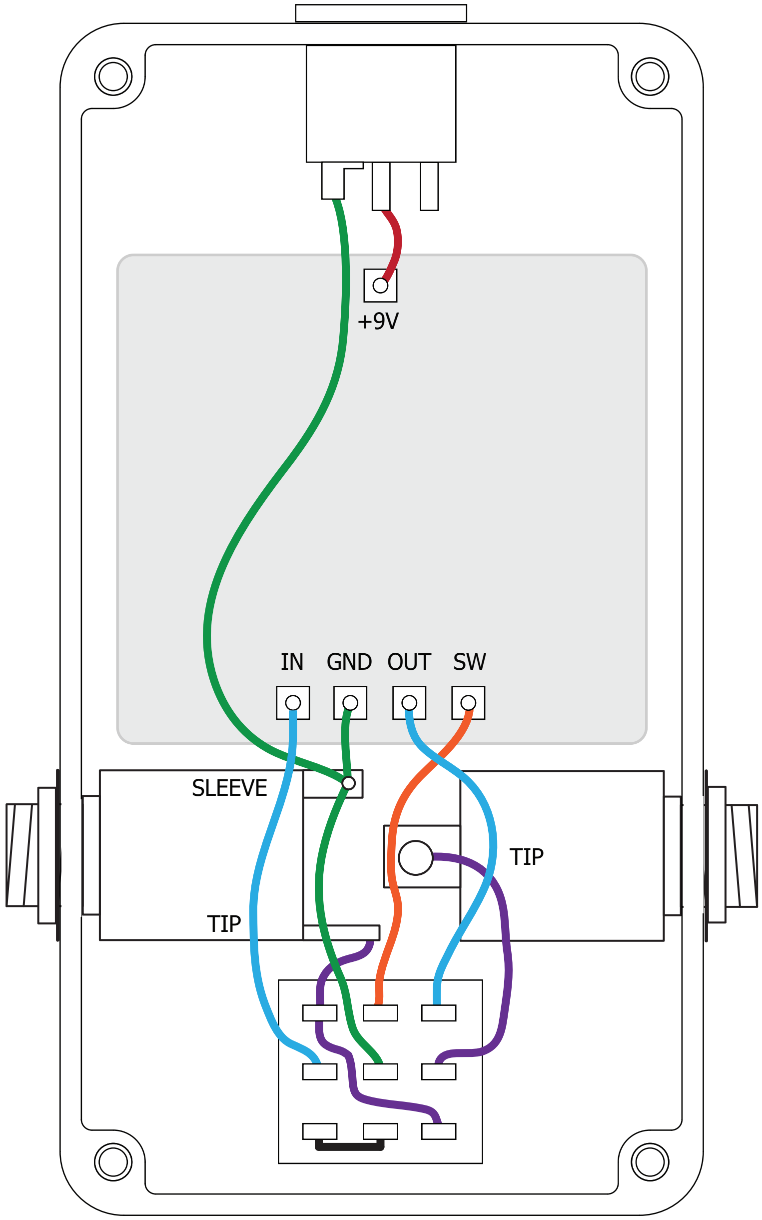

**Hammond 1590B**
*(bottom/inside view)*



## 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!)