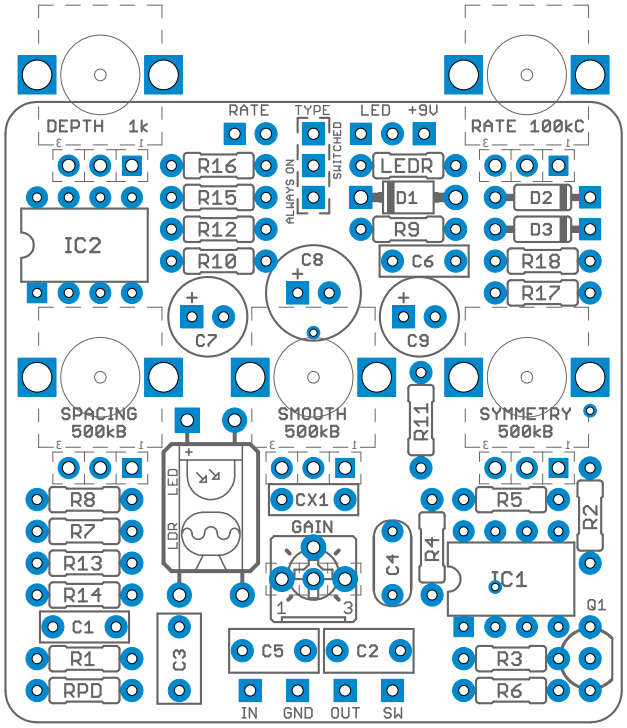
# Luna

*Based on the 4ms Tremulus Lune*

## Overview

**Luna Project Link**




The Luna Tremolo is a modified version of the 4ms Tremulus Lune, a classic DIY optical tremolo circuit that allows fine-grained control over the LFO shape.

Several of the Tremulus Lune circuit's optional modifications are included standard in this PCB, including the Symmetry control as well as an external gain adjustment control so you can achieve unity volume with any setting.

The Luna adds a few improvements to the original design, including an input buffer to reduce noise and some added filtering to counteract ticking from the LFO. All of the knobs are located on the front panel in an efficient layout that easily fits in a 1590B enclosure.

## Controls

- **Rate** is the speed at which the tremolo modulates.
- **Depth** sets the volume of the tremolo at the lowest peak of the waveform. You can turn it all the way up for a choppy sound, all the way down for a clean boost with no modulation, or anything in between.
- **Symmetry** allows you to adjust the position of the peak of the LFO in the cycle. On one end, there will be a fast rise and a slow fall, like the attack of a piano. On the other end there will be a slow rise and a fast decay, like a swell effect. The middle position is a normal tremolo sound with equal rise and fall times.
- **Smooth** lets you set the shape of the LFO. On one end of the knob's range, it's a triangle wave. In the middle, it's a sine wave, and on the far end it's a square wave.
- **Spacing** allows you to space out the LFO cycles. The width of the LFO pulse is the same, but there is a delay between cycles—creating a unique effect that is very uncommon in tremolo designs.
- **Gain** sets the volume of the tremolo at the highest peak of the waveform.

## Modifications & Experimentation

One major addition to this circuit was the **input buffer**. The first op-amp stage is in an inverting configuration to complement the 2nd inverting op-amp and make the output in phase with the input. However, if you set the first gain stage to have too low of an input impedance, it is more affected by what comes before it in the chain, and will behave very differently in different situations.

Some have fixed this by using 470k or 1M resistors for R4 and R5—but if you raise the input impedance, you also raise the noise floor due to R4's series resistance. My solution was to add a standard emitter-follower buffer to the front of the circuit, which prevents any interaction with effects earlier in the chain.

## Parts List

Mouser Parts List (Spreadsheet)

### Resistors

| | |
|---|---|
| R1 | 1k |
| R2 | 470k |
| R3 | 1k |
| R4 | 47k |
| R5 | 47k |
| R6 | 1M |
| R7 | 220k |
| R8 | 150k [1] |
| R9 | 220k |
| R10 | 220k |
| R11 | 2k7 |
| R12 | 330R |
| R13 | 1k |
| R14 | 1k |
| R15 | 1k |
| R16 | 1k [1] |
| R17 | 100k |
| R18 | 100k |
| RPD | 2M2 |
| LEDR | 4k7 |

### Capacitors

| | |
|---|---|
| C1 | 47n |
| C2 | 1uF film |
| C3 | 1uF film |
| C4 | 330pF |
| C5 | 1uF film |
| C6 | 10n |
| C7 | 10uF electro |
| C8 | 100uF electro |
| C9 | 47uF electro |
| CX1 | 33n [2] |

### Semiconductors

| | |
|---|---|
| IC1 | TL072 |
| IC2 | TL022 |
| Q1 | 2N5088 |
| D1 | 1N4002 |
| D2-D3 | 1N914 |
| VACTROL | VTL5C2 [6] |
| LED | 5mm LED |
| RATE LED | 5mm LED [3] |

### Potentiometers

| | |
|---|---|
| DEPTH | 1kB |
| GAIN | 25kB [4] |
| RATE | 100kC [5] |
| SMOOTH | 500kB |
| SPACING | 500kB [1] |
| SYMMETRY | 500kB |

[1] The Spacing pot is wired up a little differently than the original Tremulus Lune circuit to allow better control over the range. For stock wiring, jumper pins 1 & 2 of the Spacing pot and use 470k for R8.

[2] This capacitor is optional, but can help cut down LFO ticking by very slightly smoothing out the LFO's edges.

[3] See next page for more information about the Rate LED. You must set one of the "TYPE" jumpers.

[4] You can use either a trimmer or an external pot for the gain control. I think the external control is most useful, but if you want to use a trimmer, use the Bourns 3362P style mounted to the component side of the board. You can drill a small hole in the bottom of the enclosure to let you adjust it from outside with a small screwdriver.

[5] The original Tremulus Lune uses two pots for the rate: a 100k linear pot for the main speed setting and then a 10k linear pot for fine adjustment. I've collapsed this down into a single pot with a reverse-audio taper, which should allow a similar level of sensitivity without the hassle and confusion of using two pots.

[6] The VTL5C2 is out of production and difficult to find. The NSL32 is a good alternative, but due to the very low "ON" resistance, you need to jumper C2 (the 1uF capacitor) to avoid potentially creating a steep high-pass filter and cutting bass.

## Additional Part Notes

- Capacitors are shown in nanofarads (n or nF) where appropriate. 1000n = 1uF.
- The layout assumes the use of film capacitors with 5mm lead spacing for all values 1nF through 1uF.
- Potentiometers are Alpha 9mm right-angle PCB mount.

## Setting up the rate LED

The Rate LED is a useful feature, but people will have differing opinions on how it should work. Accordingly, I made it as flexible as possible to accommodate any preference.

Here are the decisions you need to make when deciding how to wire up the Rate LED:

1. Do you want an LED that flashes with the rate of the tremolo?

    - If **yes**, include the Rate LED and the resistor marked R16.
    - If **no**, leave these two components unpopulated and skip the rest of this section.

2. Should the Rate LED be separate from the main on/off indicator LED, or should it be the same?

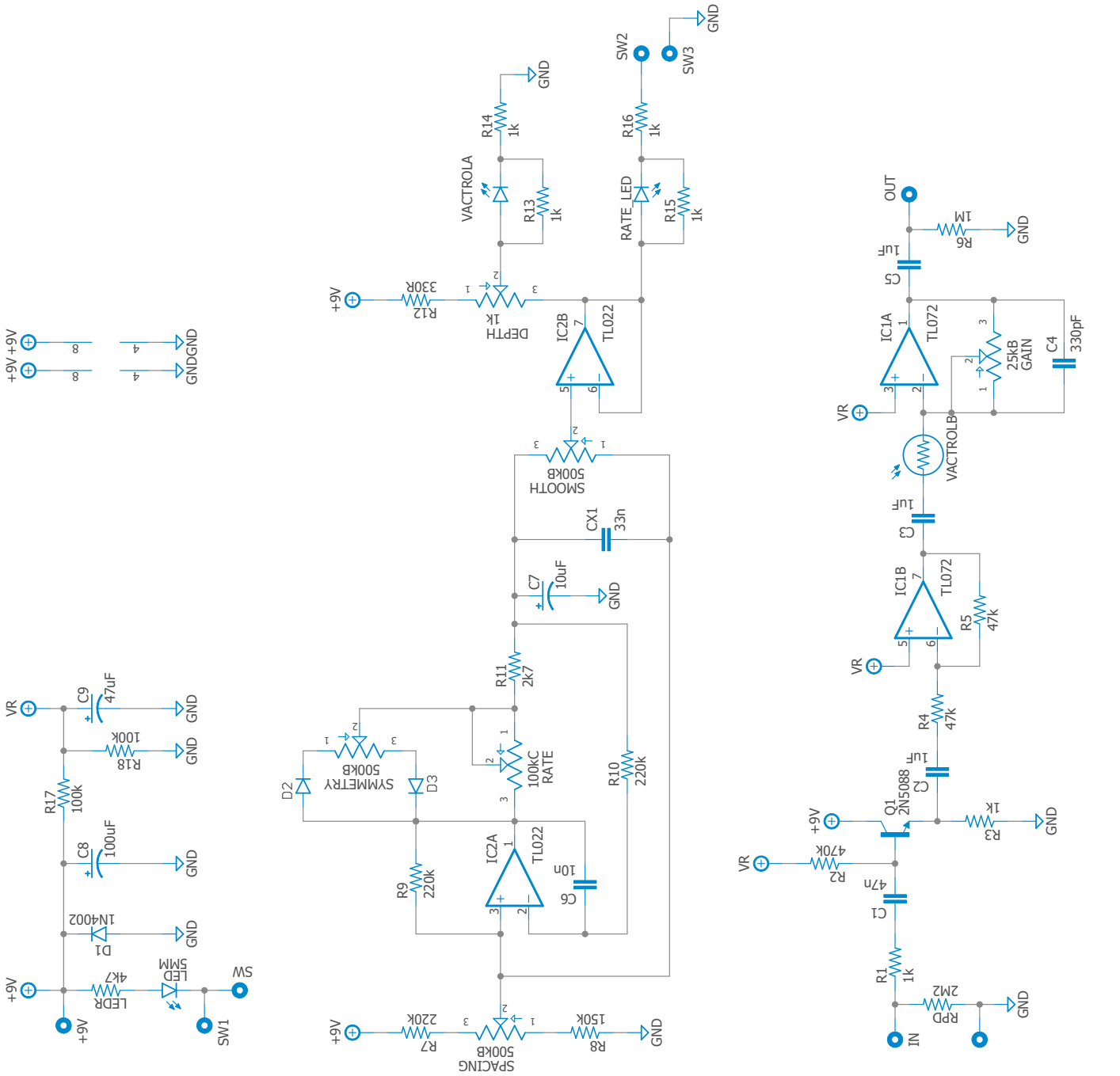
    - If it should be **separate**, include the main indicator LED (the one on the right side of the board) and the resistor marked LEDR.
    - If it should be **the same**, leave off the main indicator LED and the resistor marked LEDR.

3. Should the rate LED turn on and off with the footswitch, or should it always be flashing even when the pedal is bypassed?

    - If it should **turn on and off** with the footswitch, jumper the top two “TYPE” pads between the LEDs, marked “SWITCHED”. This will tie the LED to the footswitch.
    - If it should **always flash** regardless of bypass status, jumper the bottom two “TYPE” pads between the LEDs, marked “ALWAYS ON”.

However you decide to use them, note that if you only use one of the LEDs, I recommend drilling the hole in the center of the enclosure and then bending the legs of the LED to the left or to the right to fit the pads on the PCB so that the front panel layout is symmetrical. All three LED positions are marked on the drill template.

# Schematic


+9V +9V
GND GND
VR
C9 47uF
R18 100K
R17 100k
C8 100uF
D1 1N4002
+9V
LEDR 4K7
LED SMM
SW
+9V
SW1
VACTROLA
R13 1k
R14 1k
GND
RATE_LED
R15 1k
R16 1k
SW2
SW3
GND
+9V
R12 330R
DEPTH 1K
IC2B TL022
SMOOTH 500KB
CX1 33n
C7 10uF
GND
R11 2k7
D2
SYMMETRY 500KB
D3
100kC RATE
R10 220k
IC2A TL022
R9 220k
C6 10n
+9V
R7 220K
SPACING 500KB
R8 150K
GND
OUT
R6 1M
GND
C5 1uF
IC1A TL072
25kB GAIN
C4 330pF
VR
VACTROLB
C3 1uF
IC1B TL072
R5 47k
VR
R4 47k
C2 1uF
Q1 2N5088
+9V
R3 1K
GND
VR
R2 470K
C1 47n
R1 1k
IN
RPD 2M2
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

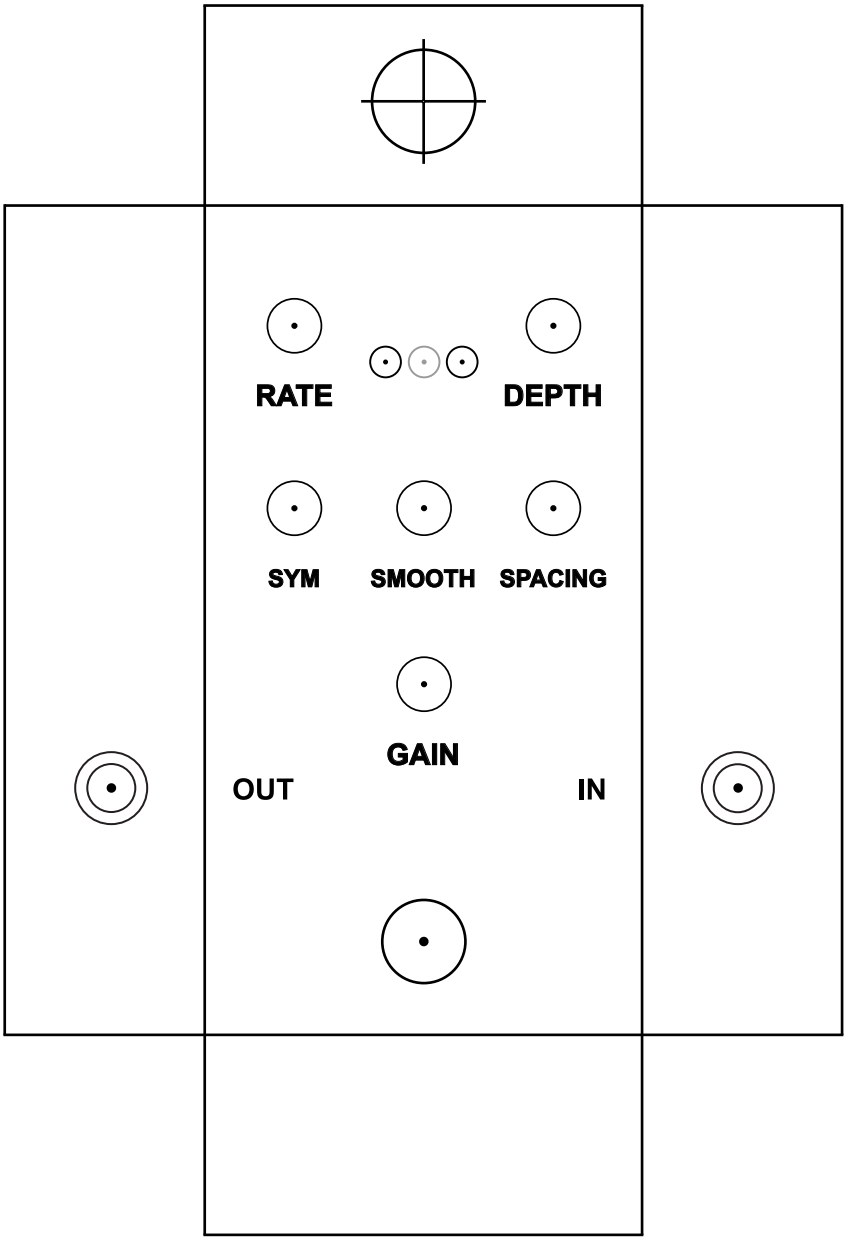
The resistors marked "RPD" and "LEDR" are generally not original to the circuit and can be adjusted to preference. "RPD" is the pulldown resistor to help tame true-bypass popping, while "LEDR" controls the brightness of the LED. I generally use 2.2M for the pulldown resistor and 4.7k for the LED resistor.

### Sockets

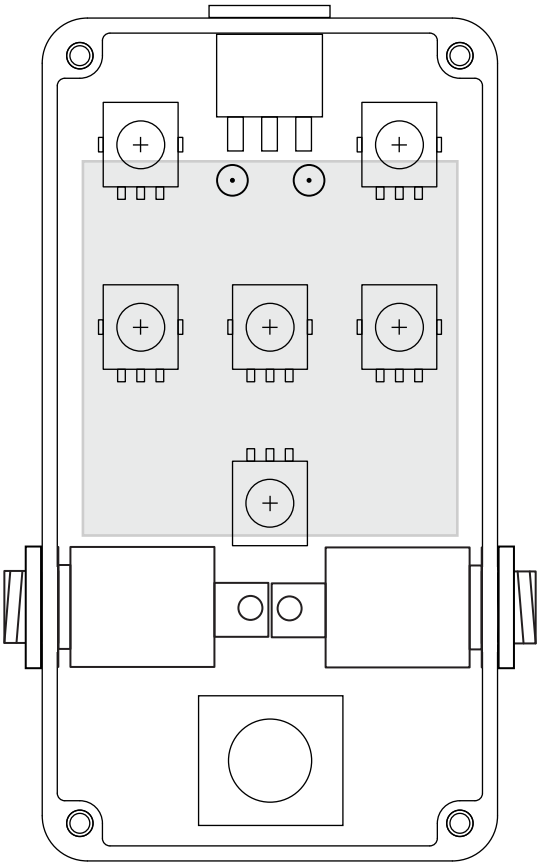
Since double-sided boards can be very frustrating to desolder, especially components with more than 2 leads, it is recommended to use sockets for all transistors and ICs. It may save you a lot of headaches later on.

## 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

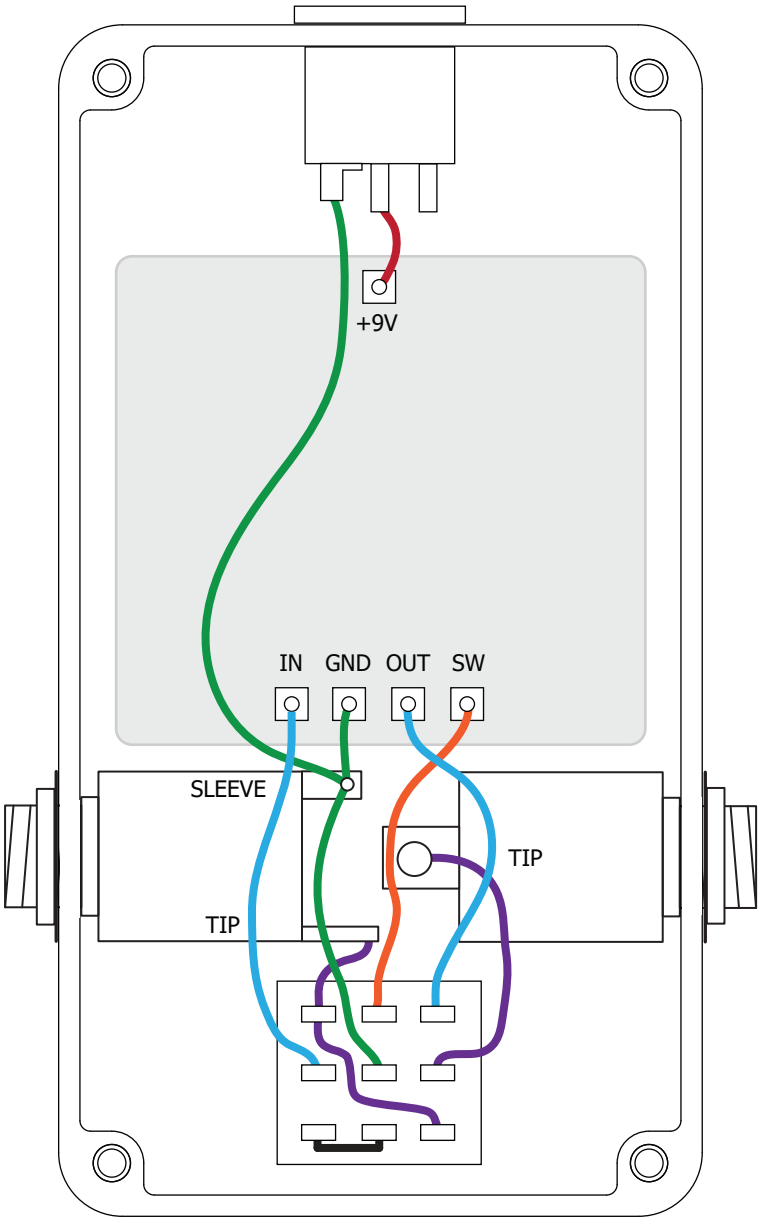
This diagram shows standard true-bypass wiring with a 3PDT switch. When the switch is off, the input of the circuit is grounded and the input jack is connected directly to the output jack.

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!)