Contents

www.MonarchPedals.com / hello@monarchpedals.com / #MonarchPedals

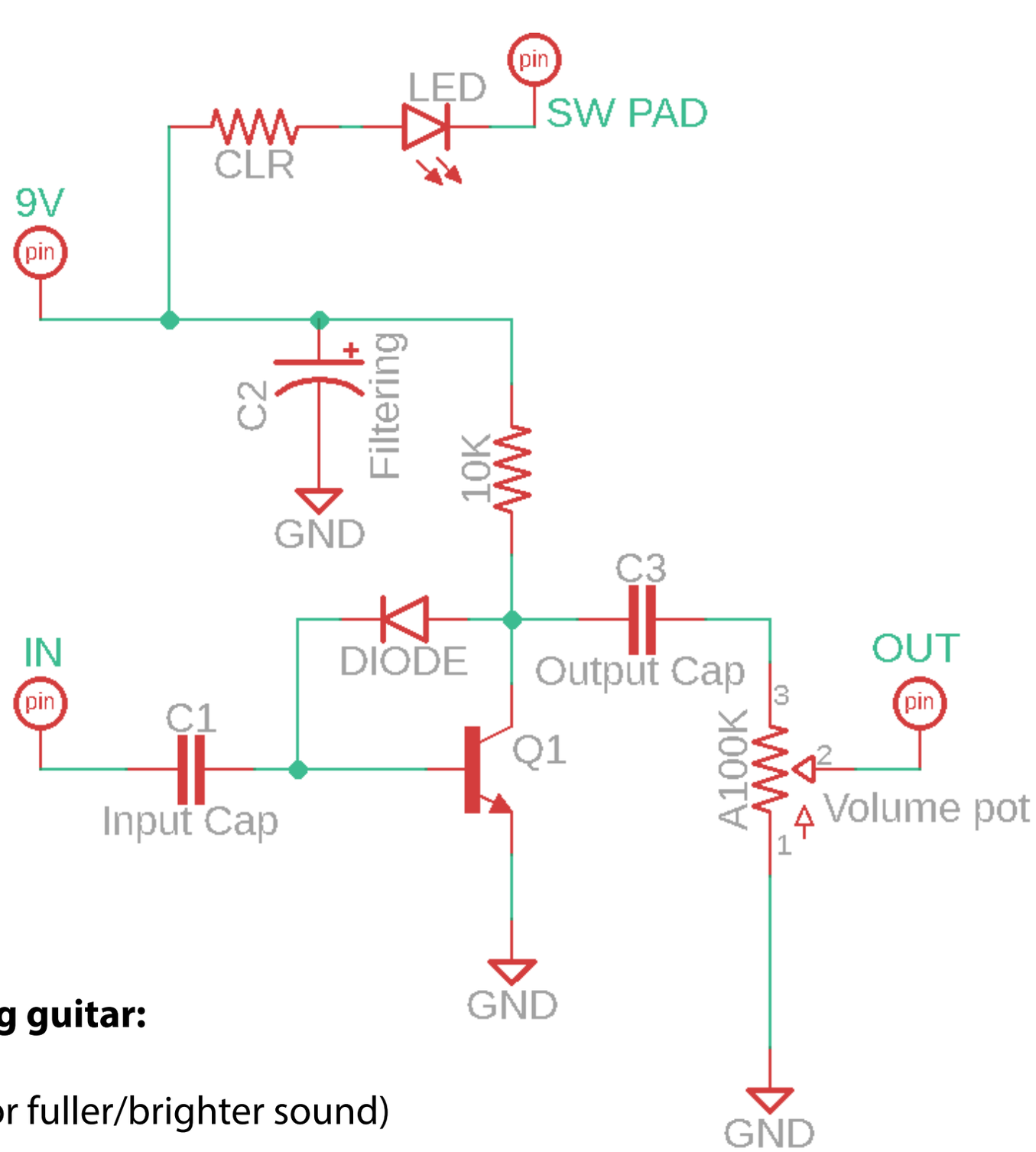
Special thanks to the members of NZGuitars, CodaEffects, ElectroSmash, TaydaElectronics, RunOffGroove, Elams1894, DIYStompBoxes, home-wrecker.com, r/DIYpedals, Chuck D. Bones, PedalPCB.

1. Schematic for the Bazz Fuss.

This circuit was originally designed for bass guitar (hence the name) using a 4.7uf input cap and 220nf output cap for a fat bass response. The original circuit used a 2N5088 transistor and 1N914 diode which provided a moderate amount of boost. There was no volume control, instead relying on the instrument's volume control to adjust the signal strength and clarity. The circuit works like this:

The signal comes in on the left, ahead of C1. Then the signal goes through the circuitry, being amplified by the transistor (Q1) and diode in the feedback loop, before heading out through C3 where any DC current is removed and out through a 100k potentiometer wired as a volume control. As the pot is turned 'up' it sends less of the signal to ground resulting in a louder signal.

The 9v power supply is filtered through a 100uf bypass capacitor which is responsible for removing any unwanted ripple noise from the power supply. The CLR resistor in series with the LED is there to protect the LED from receiving more current than it can handle.



Suggested values for 6 string guitar:

C1 = 1uf (increase/decrease for fuller/brighter sound)
C2 = 100uf
C3 = 100nf (increase/decrease for fuller/brighter sound)
R1 = 10k
CLR = 4.7k-10k for regular LED. Use 10k-22k if using a superbright LED.
DIODE = BAT41 (lower vf = more gain, if you'd like a mellower tone try a 1N4001, 4148, or even a red LED)
Q1 = MPSA13 (higher Hfe = more gain. For lower gain try a 2N2222 or socket different ones, they're cheap!)

2. Introduction & PCB Overview

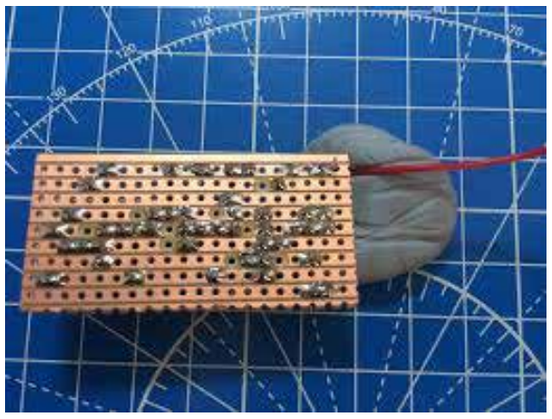
Congratulations on being here. We're going to be building the Bazz Fuss pedal, a classic design that, although not overly complicated, is capable of making a fun and usable tone using just a single control and a handful of readily available bits. Let's get started.

This is the PCB from the top down. You'll know you're looking at the right side because you'll be able to read the silkscreen text.

The holes are surrounded by a piece of copper, these are called pads. You take the component, such as a resistor, bend the legs at 90 degrees or so, thread those legs through the pads while pulling the component through from the other side so it sits snug and flush on the PCB's surface. Then squeeze the component's legs together into a Y shape so the component doesn't fall out on you when you flip the PCB over to solder.

If using sockets then clip off the amount you need with pliers or nippers one at a time (except for the transistor where a row of 3 sockets fits perfectly), insert the socket into the pad, press some BluTack on top to secure them in place, then flip over the board and solder the sockets in. To insert a component into a socket more easily you might need to clip the legs shorter (this is especially relevant to resistors where the legs can be a bit bendy and annoying).

I use a big blob of BluTack on the desk. I push the PCB face down into this blob so it is held securely in place while soldering occurs. I set my iron to 352c, because it's 666F which imbues your circuits with unholy power. But seriously, I stay around 350c and will go up to about 375c for jack and pot lugs which are thicker pieces of metal so need more heat.



BluTack being used to hold veroboard in place during pedal making.

These 6 pads connect to the in and out mono jacks and the DC socket.

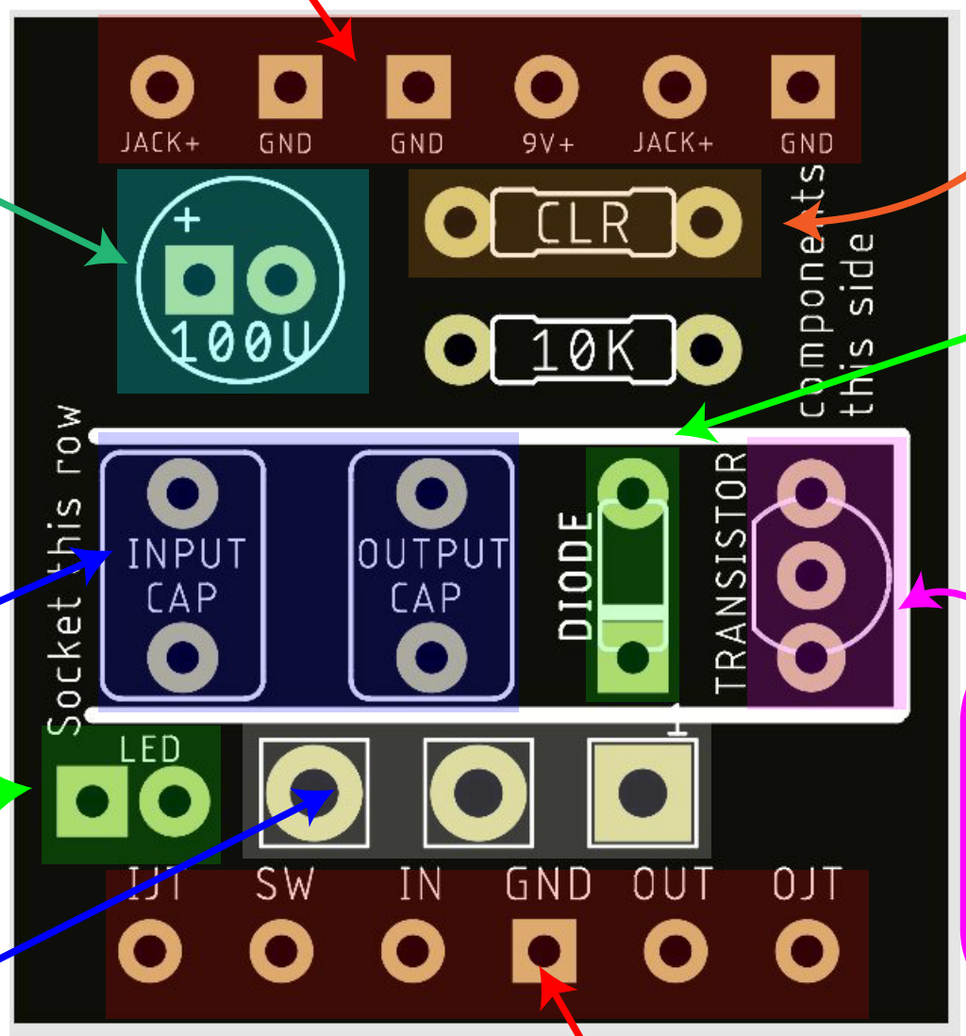
Electrolytic cap for power filtering. The leg by the stripe side goes in the CIRCLE hole.

Stripe

Long leg into SQUARE

Short leg into CIRCLE

CLR = Current Limiting Resistor. This resistor determines how bright the indicator LED will be when the pedal is engaged. The lower the value, the brighter the LED. I would not suggest using below 1k to protect the diode.



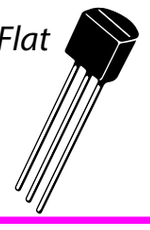
The diode goes here, the stripe on the diode itself corresponds to the stripe on the PCB.

A 1N4001 diode for reference. See the silver stripe?

The input and output caps can be put in any orientation as long as they are not polarised electrolytic.

The transistor (Q1) goes here. Note one side is rounded and one side is flat. Make sure you insert the transistor as shown on the PCB.

Flat *Rounded*



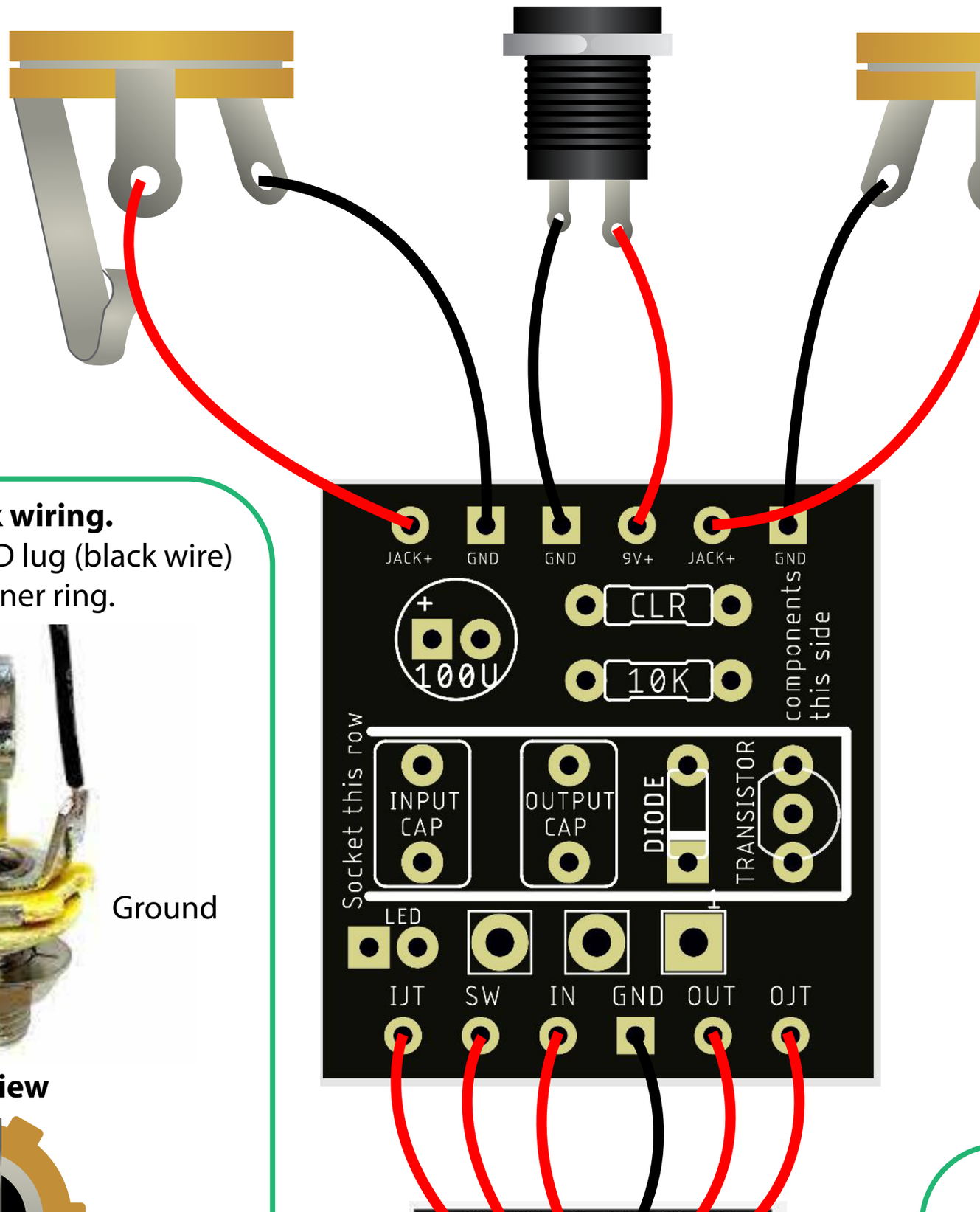
The LED is soldered on the same side as the volume pot.
LONG LEG into the SQUARE pad.

Pot lugs, install this pot from the other side.
Lug 1 always goes in the SQUARE pad.

These 6 pads connect to the corresponding pads on the footswitch PCB.

3.Jacks and Switch Wiring

This is the wiring diagram for the PCB. You'll need 12 wires (2 for each 6.35mm jack, 2 for the DC jack, and 6 for the footswitch PCB), the ends of which you'll need to strip and if using stranded wire you'll need to tin these beforehand as well. I use solid core but you use whatever you like, it's your pedal. The colours here are shown as examples, with the usual convention of GROUND being BLACK and HOT being RED but anything will do, as long as you know what's going where it doesn't matter.



Mono jack wiring.
Notice how the GND lug (black wire) is on the inner ring.

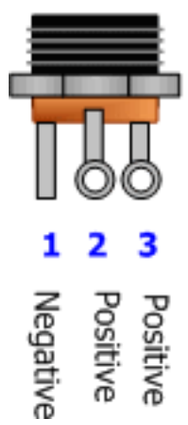
Back view

Note:
Wire the DC jack last after all else is soldered. It's nut threads from the inside.

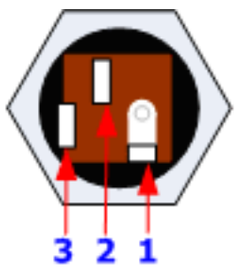
Thinline DC jack:
GND is short leg
9V+ is long leg

or

Boss style DC jack wiring



Side View

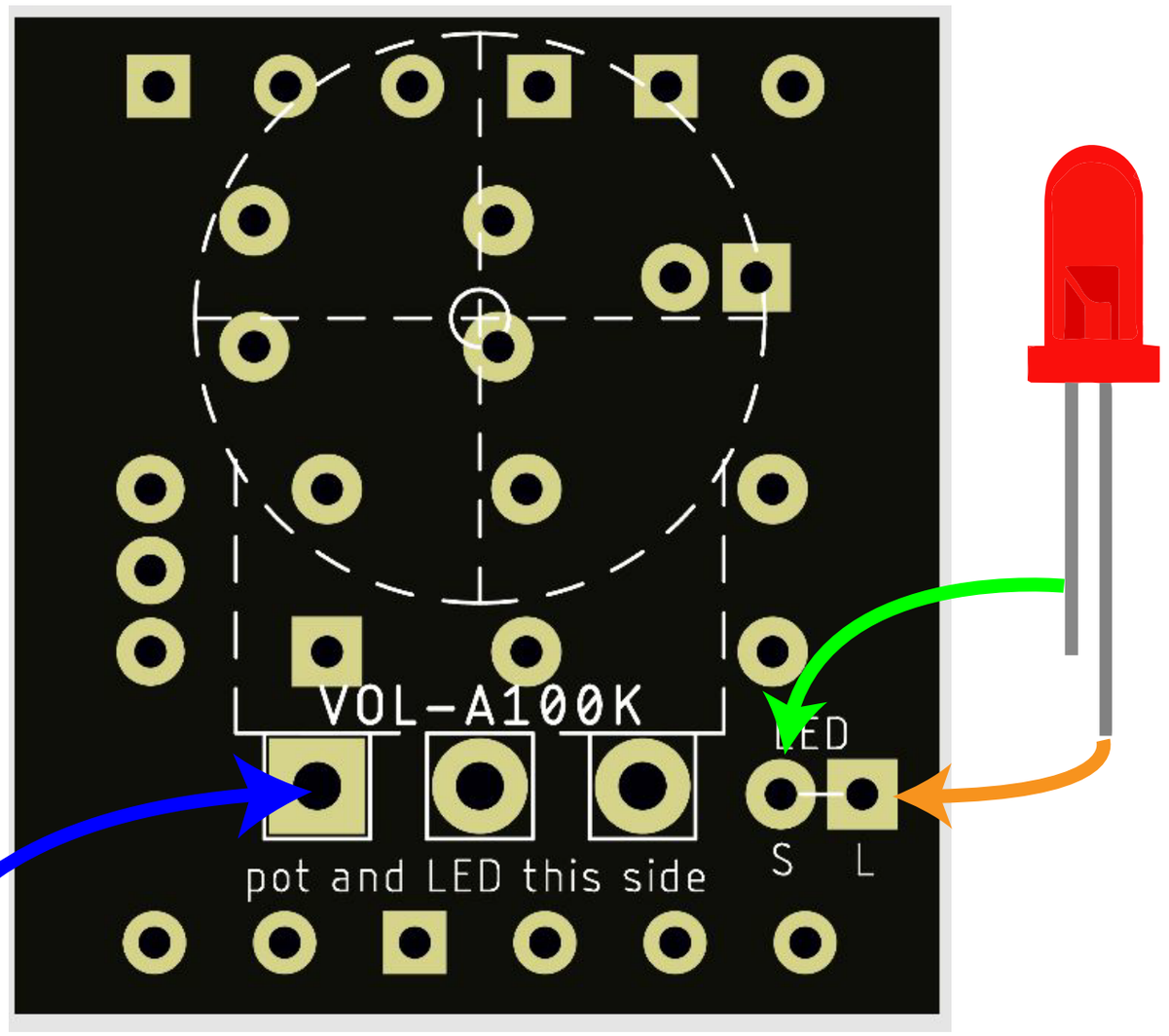


Back View

Leave lug 2 empty if not using a battery

4. LED and pot wiring

This is the back of the board. Here is where we solder the indicator LED and volume pot to the PCB.



LED Wiring

The LED will have a long leg and a short leg.

The long leg goes in the square pad (marked L for LONG). The shorter leg goes in the circle pad (marked S for SHORT).

Leave a little bit more leg than you think, just in case. You can always bend legs to fit later on, just make sure they don't touch anything else.

Wiring the pot

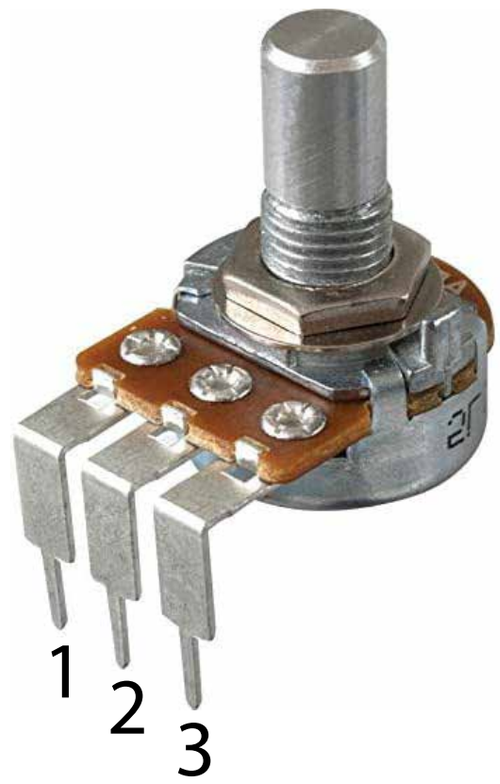
Pot lugs are named 1, 2, and 3. The orientation of which is shown to the right. It helps to mark lug 1 with a black marker.

The number 1 lug always goes in the SQUARE pad on the PCB. Use the placement guide to help out but don't worry if it doesn't match the outline on the board, we might be using different size pots.

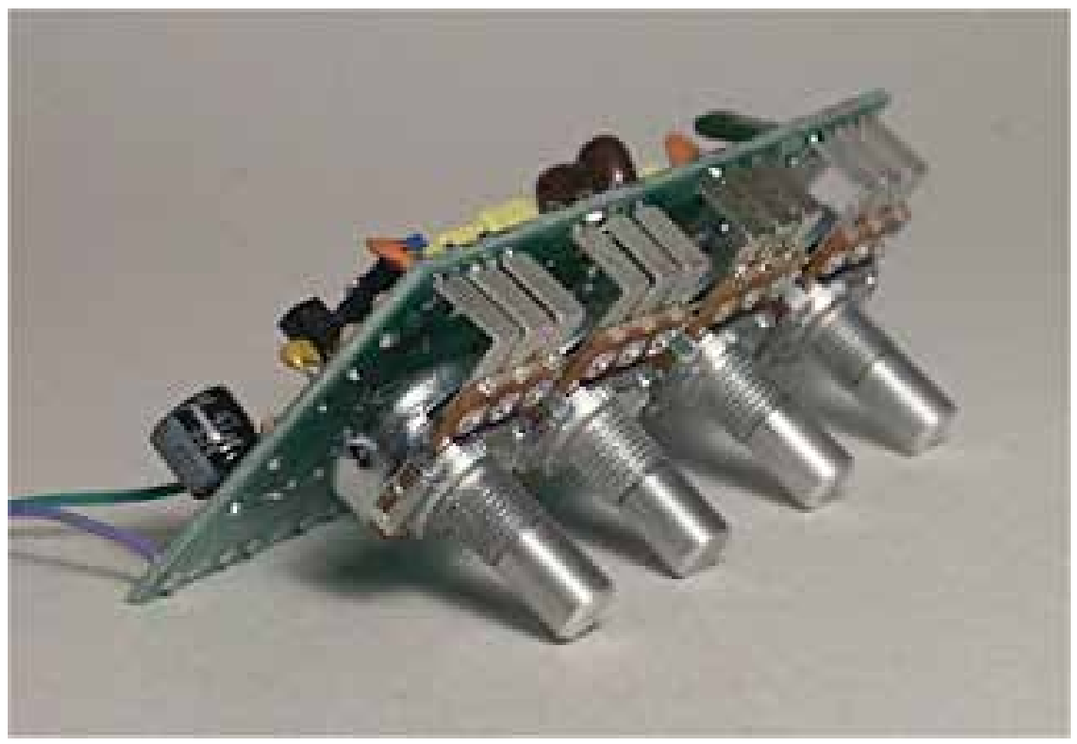
Insert the pot legs into the pads, ensuring LUG 1 is inserted into the SQUARE pad.

Solder the middle leg in place making sure the pot is at a good angle relative to the board. It should lay reasonably flush across the solder blobs. Once you're happy with where it is, solder legs 1 and 3, then re-flow some fresh solder to lug 2 if you think it needs it.

Once the solder has cooled flip over the board and inspect it from the component side. The joints should be shiny pyramids. You might need some more solder if you're not convinced.



Make sure you've clipped the component legs cleanly, any pokey bits may pierce the dust shield on the pot and cause a short circuit meaning no sound :(



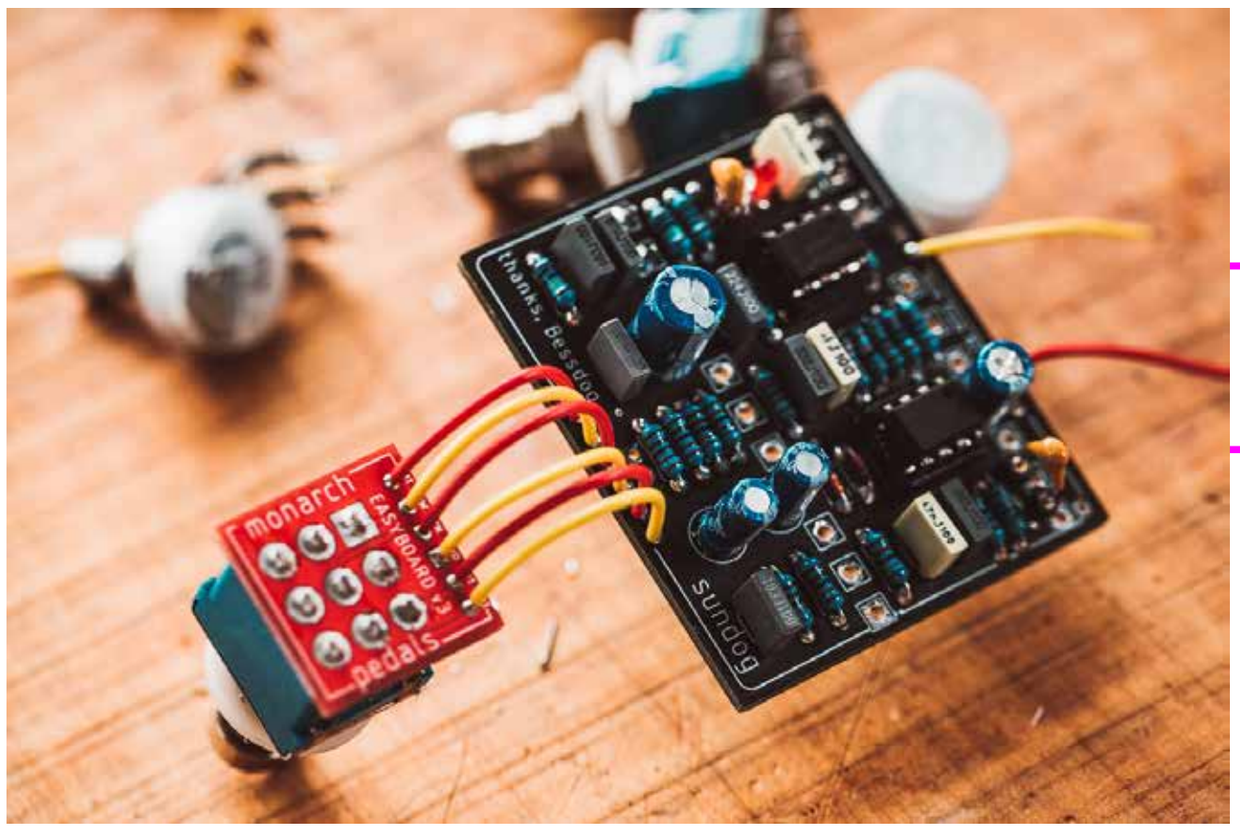
Example. Notice how the components are mounted on the opposite side to the pots

5. The Switch PCB

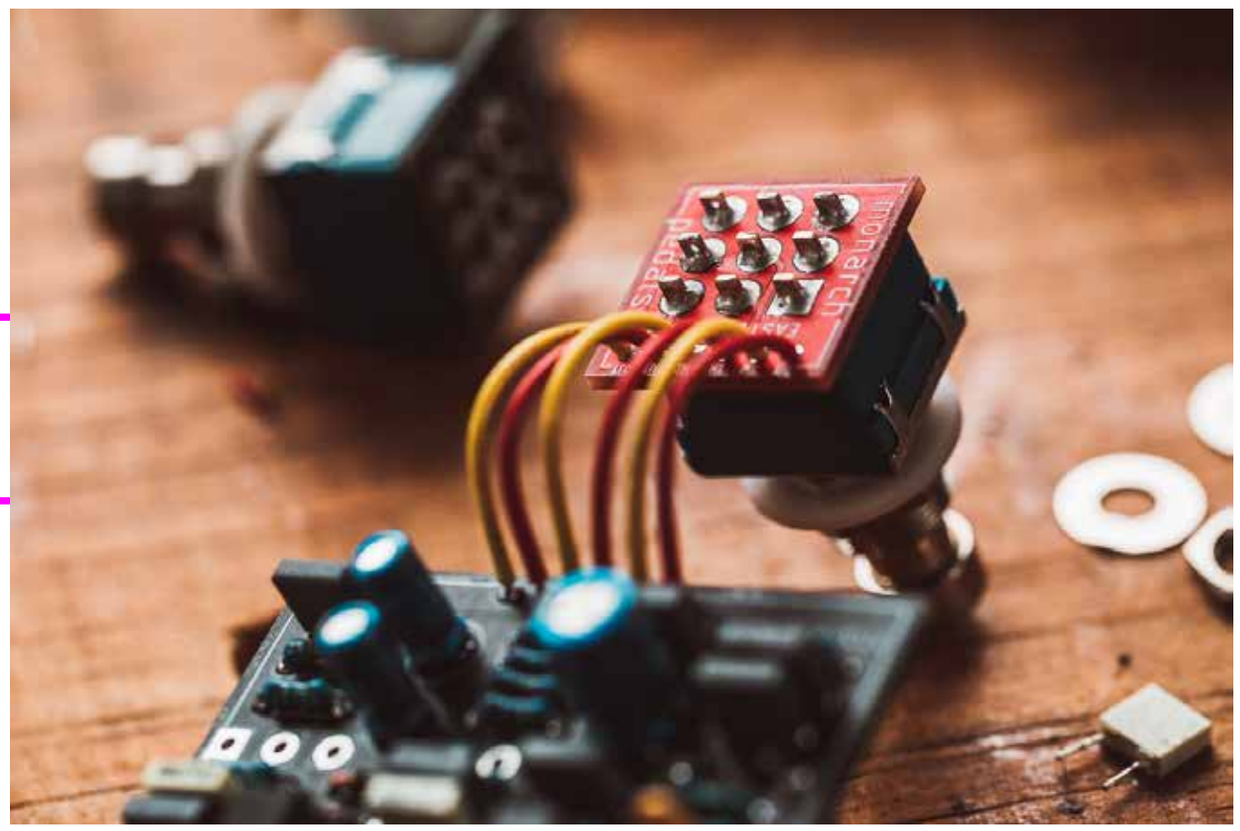
I use a little PCB that sits on top of the switch. This is already wired for true bypass and LED switching. The 6 pads along the top of the switch PCB correspond to the 6 pads along the bottom of the effect PCB. The correct orientation of the switch PCB is shown below. The text reading EASYBOARD V3 should be on the same side as the components on the PCB and be facing the effect PCB.

To solder the switch PCB to the switch I rest the 3PDT upside down so the lugs are facing up. I've drilled a 11.5mm hole into a piece of wood so the switch stands up in this hole, but you can use the BluTack blob from before or a set of helping hands, or whatever works in your setup. Then rest the footswitch PCB atop the switch, making sure you can see the EASYBOARD silkscreen and the lugs are poking through the pads. Check the photos for the correct way around as they only work one way. Solder the middle lug into the middle hole, you might have to heat it for a couple of seconds before the solder will flow. You don't need to fill the pads like I have, just as long as there's a mechanical connection between pad and lug.

Give each lug 5-10 seconds to cool before moving on to another, otherwise you run the risk of cooking the switch and/or melting the epoxy that holds the lugs into place.



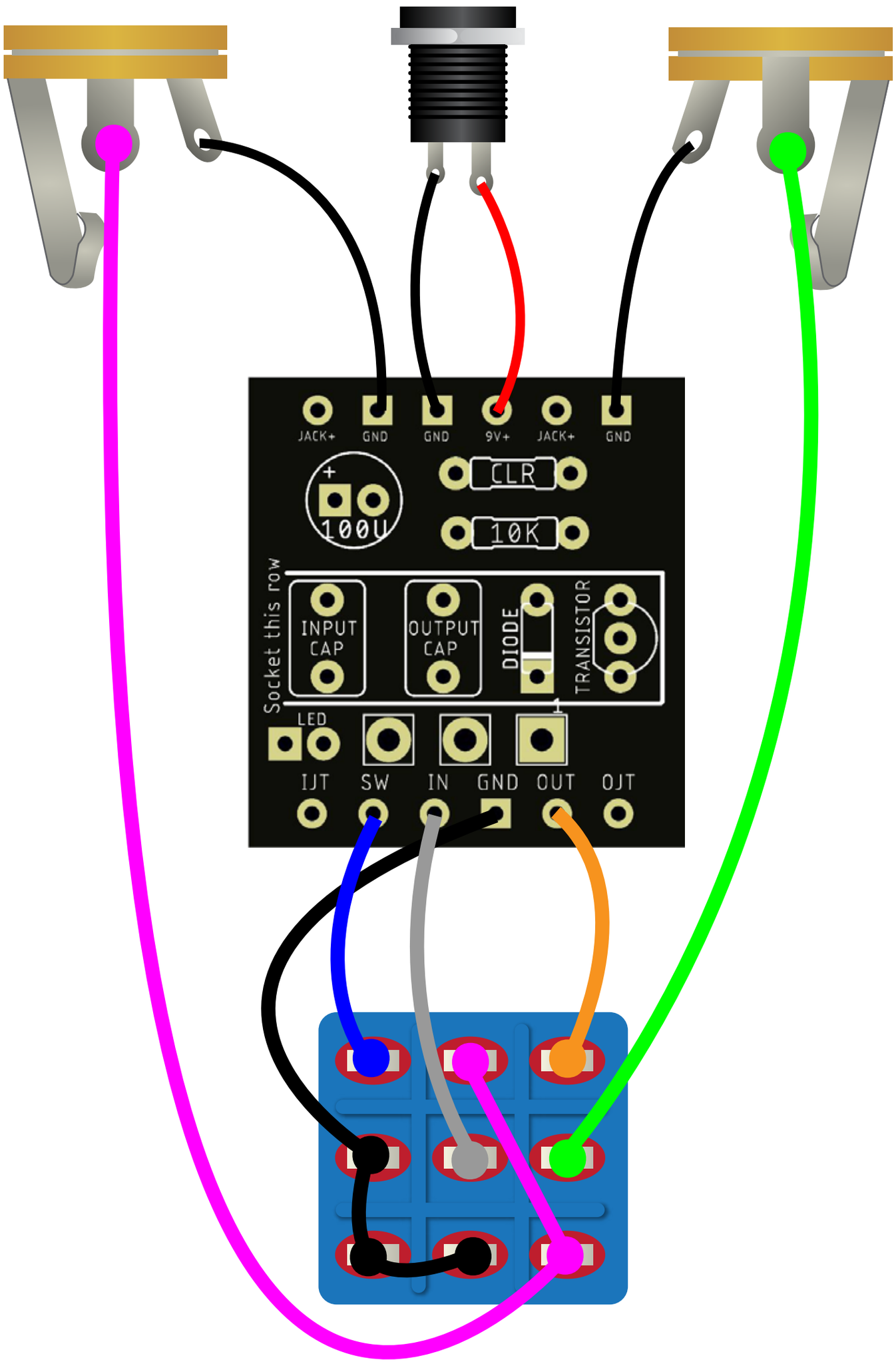
The proper orientation is shown here. The EASYBOARD V3 writing should be closest to the effect PCB.



Reverse view of the same pedal

6. Manual wiring diagram:

Here's the connections for manually wiring the 3PDT but still keeping the LED on the board. If you'd like the LED elsewhere just solder wires to the legs for some extra reach.



7. Bill of Materials

6.35mm mono jack (you'll need 2):
https://www.taydaelectronics.com/6-35mm-1-4-mono-chassis-socket-jack.html

3PDT footswitch:
https://www.taydaelectronics.com/3pdt-stomp-foot-pedal-switch.html

DC Power jack:
https://www.taydaelectronics.com/dc-power-jack-2-1mm-round-type-panel-mount-1.html

100uf Electrolytic Capacitor
https://www.taydaelectronics.com/100uf-50v-105c-jrb-radial-electrolytic-capacitor-8x12mm.html

Resistors (1/4w, 1%, Metal Film):
https://www.taydaelectronics.com/resistors/1-4w-metal-film-resistors/test-group-2.html
The price is per 10. You'll need 10k and your preferred CLR value.
Select the amount (in multiples of 10 and 10 being the minimum order) then click Add to Cart.

Input & output capacitors:
https://www.taydaelectronics.com/capacitors/polyester-film-box-type-capacitors.html?p=1

If using input or output capacitors above 1uf:
https://www.taydaelectronics.com/capacitors/monolithic-ceramic-capacitor.html?p=2

LED (I use 5mm round):
https://www.taydaelectronics.com/leds/round-leds.html

Bare 1590b size enclosure:
https://www.taydaelectronics.com/hardware/enclosures/1590b-style/1590b-style-aluminum-diecast-enclosure.html

Transistors (NPN BJT is good. Hfe data can be found in the datasheet):
https://www.taydaelectronics.com/t-transistors/2n-series.html
https://www.taydaelectronics.com/t-transistors/bc-series.html
https://www.taydaelectronics.com/catalogsearch/result/?q=MPSA

Don't forget the knob for the pot!

Diodes.
https://www.taydaelectronics.com/diodes.html
The forward voltage rating (vf in the datasheets) will determine the amount of gain available, lower = more .
Lower vf = earlier clipping = more gain and compression at the expense of volume and openness.
Examples of low vf = BAT42, 1N34A, Schottky diodes. Examples of higher vf = Red/Green LEDs.

Volume pot (since May 2021 the dust covers are sold separately, link is in this URL):
https://www.taydaelectronics.com/p-100k-ohm-logarithmic-taper-potentiometer-round-shaft-pc-mount-5707.html

Sockets:
https://www.taydaelectronics.com/30-pin-dip-sip-ic-sockets-adaptor-solder-type.html