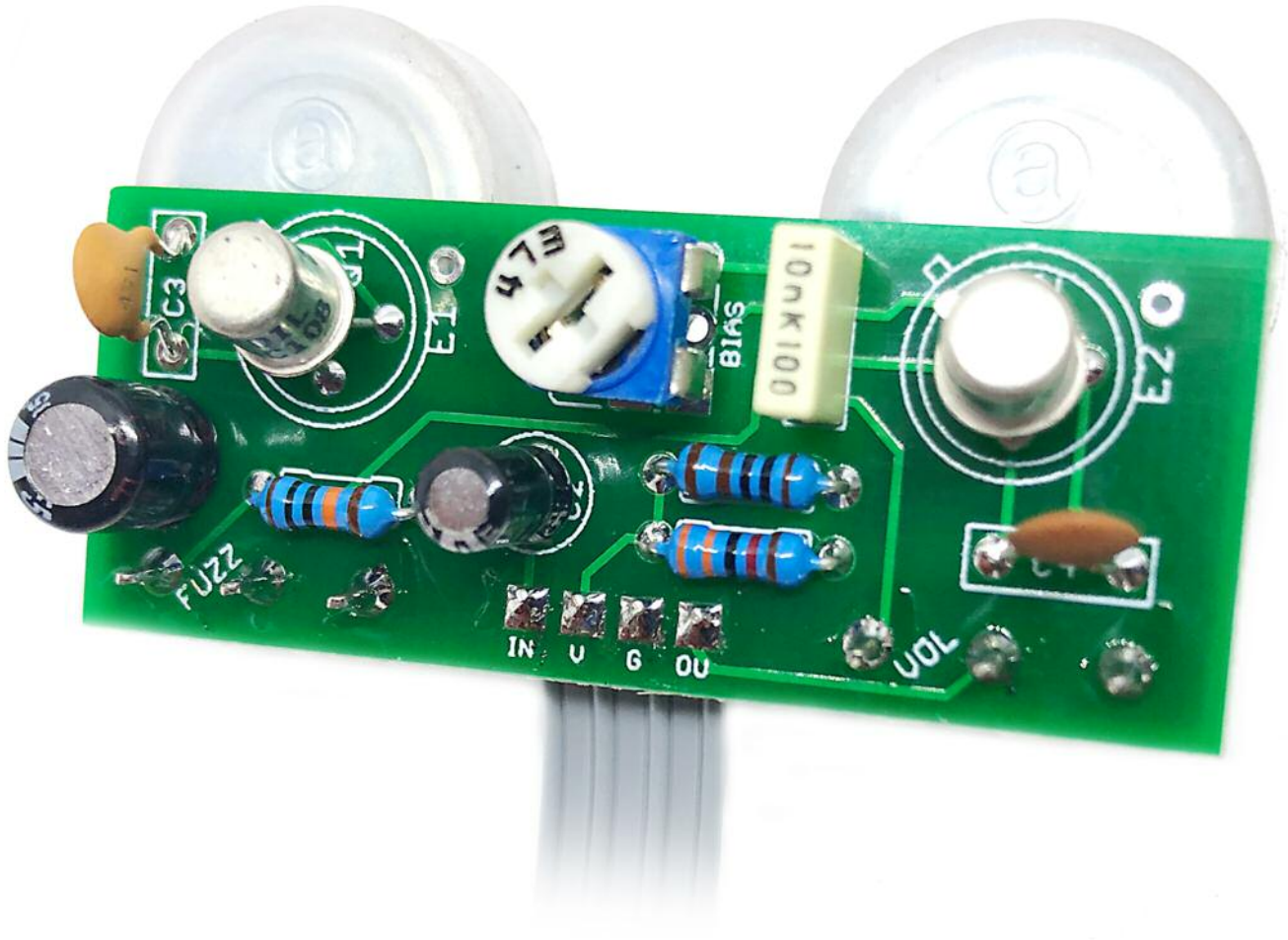


Fuzz Face (BASIC)

No mess, no fuss...
just fuzz

Important notes

If you're using any of our footswitch daughterboards, DOWNLOAD THE DAUGHTERBOARD DOCUMENT

- Download and read the appropriate build document for the daughterboard as well as this one BEFORE you start.
- DO NOT solder the supplied Current Limiting Resistor (CLR) to the main circuit board even if there is a place for it. This should be soldered to the footswitch daughterboard.

POWER SUPPLY

Unless otherwise stated in this document this circuit is designed to be powered with 9V DC.

COMPONENT SPECS

Unless otherwise stated in this document:

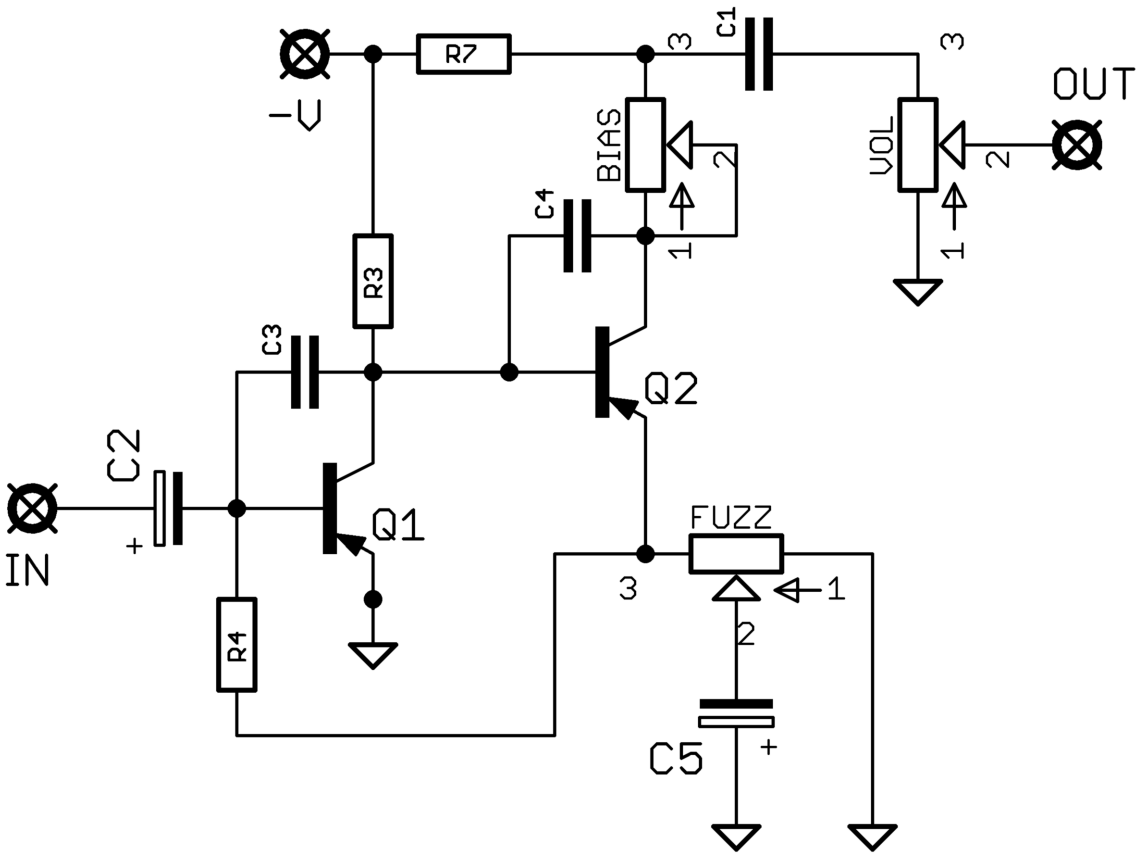
- Resistors should be 0.25W. You can use those with higher ratings but check the physical size of them.
- Electrolytics caps should be at least 25V for 9V circuits, 35V for 18V circuits. Again, check physical size if using higher ratings.

LAYOUT CONVENTIONS

Unless otherwise stated in this document, the following are used:

- **Electrolytic capacitors:**
  Long leg (anode) to square pad.
- **Diodes/LEDs:**
  Striped leg (cathode) to square pad. Short leg to square pad for LEDs.
- **ICs:**
  Square pad indicates pin 1.

Schematic - PNP (posi ground) Ge



R3	33K	C1	10n	Q1	PNP Germanium <>70hFE
R4	100K	C2	2u2 elec	Q2	PNP Germanium <>120hFE
R7	470R (1K)	C3	Empty		
		C4	Empty	FUZZ	1KB (2KB)
BIAS	47K trim	C5	22u elec	VOL	500KA

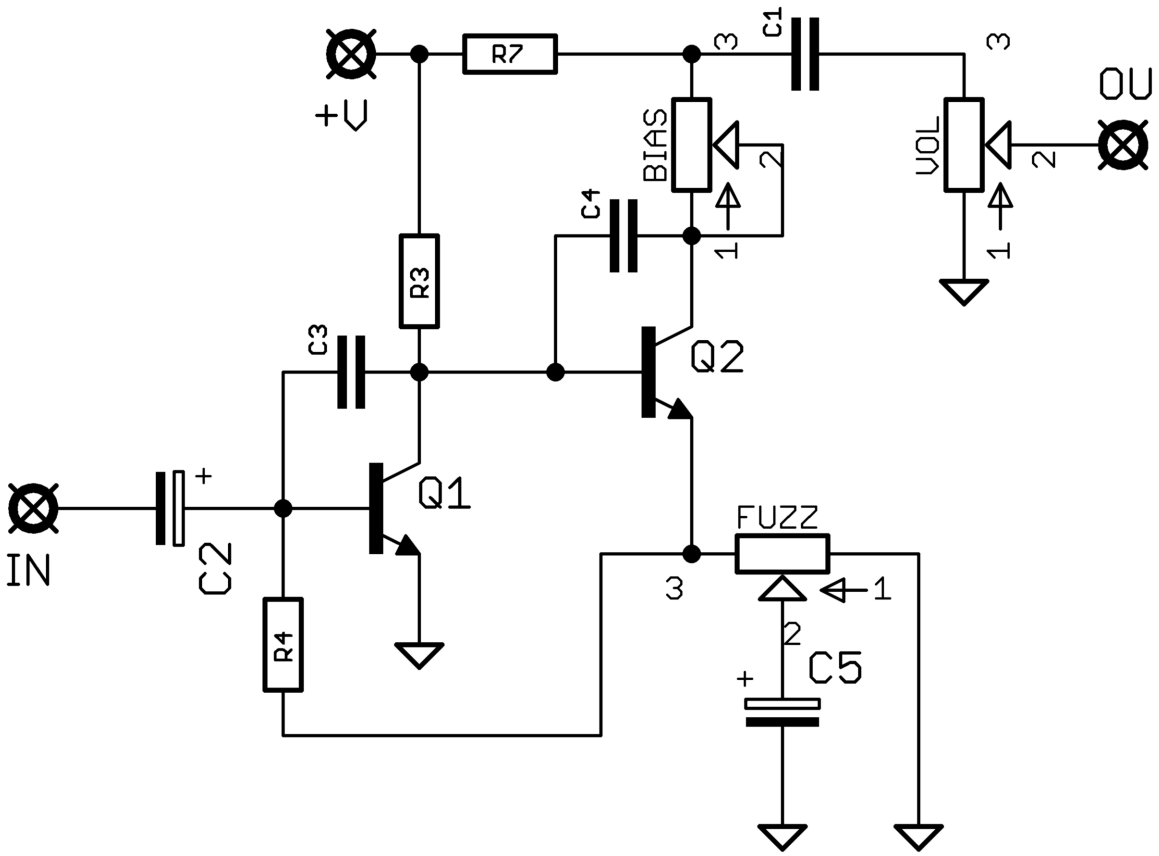
This is the original Fuzz Face design, along with a couple of value swaps (brackets) for the Mayer modified version which is a bit louder and has a better range on the Fuzz control.

This requires positive ground wiring, so can't be daisy chained with standard polarity effects unless you use a voltage inverter.

The parts on the PCB are laid out for this version.

Example transistors: AC128, 2N404

Schematic - NPN (standard ground) Ge



R3	33K	C1	10n	Q1	NPN Germanium <>70hFE
R4	100K	C2	2u2 elec*	Q2	NPN Germanium <>120hFE
R7	470R (1K)	C3	Empty		
		C4	Empty	FUZZ	1KB (2KB)
BIAS	47K trim	C5	22u elec*	VOL	500KA

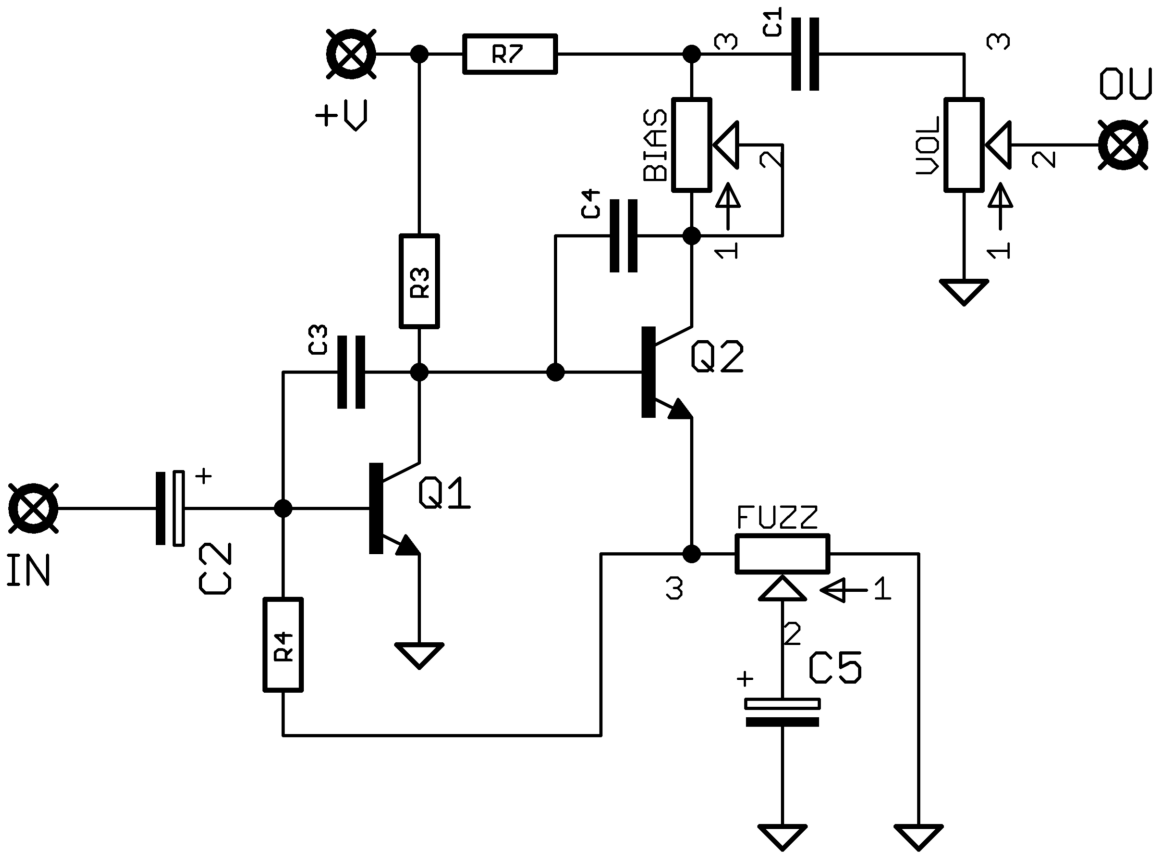
A more pedalboard friendly version, along with a couple of value swaps (brackets) for the Mayer modified version which is a bit louder and has a better range on the Fuzz control.

This requires standard negative ground, so can be daisy chained with no issues.

*You must reverse C2 and C5 for this build, i.e. positive leg into round pad.

Example transistors: AC176, CV7351

Schematic - NPN (standard ground) Si



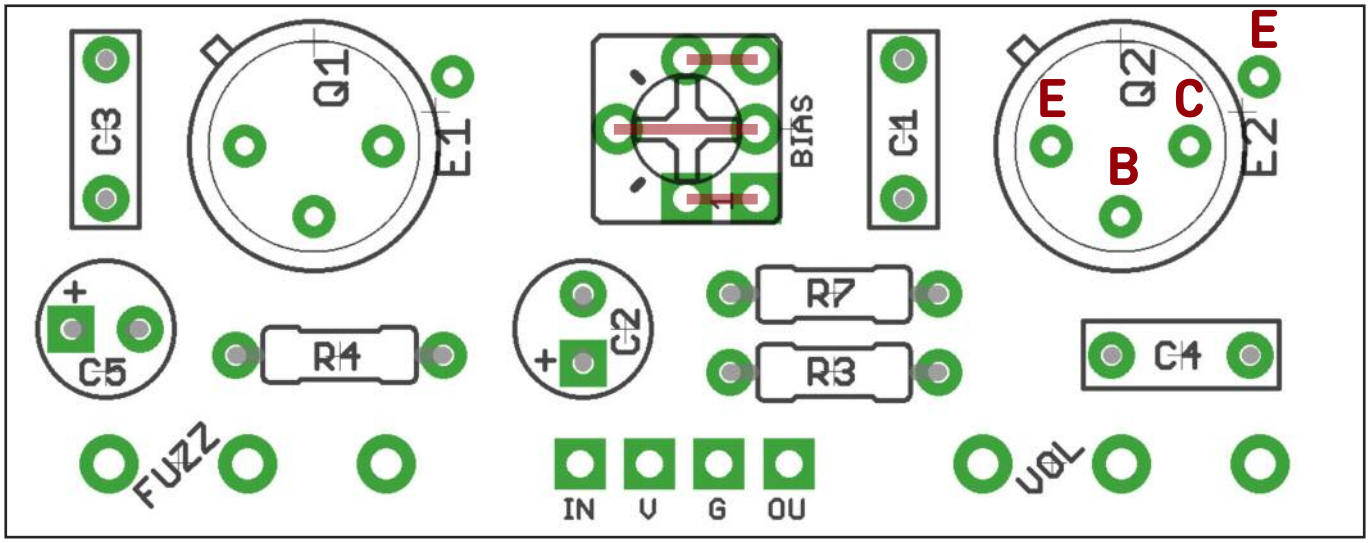
R3	33K	C1	10n	Q1	NPN Silicon
R4	100K	C2	2u2 elec*	Q2	NPN Silicon
R7	470R (1K)	C3	47-470pf**		
		C4	47-470pf**	FUZZ	1KB (2KB)
BIAS	47K trim	C5	22u elec*	VOL	500KA

Silicon version, along with a couple of value swaps (brackets) for the Mayer modified version which is a bit louder and has a better range on the Fuzz control.

This requires standard negative ground, so can be daisy chained with no issues.

Any low-medium gain BJTs can be experimented with. BC108 and 2N2222 are FDHQ favourites.

*You must reverse C2 and C5 for this build, i.e. positive leg into round pad.
**C3-C4 remove some high frequency from each gain stage in the form of negative feedback. The higher the value, the more noticeable it'll be as lower frequencies are added to the range.



The power and signal pads on the PCB conform to the FuzzDog Direct Connection format, so can be paired with the appropriate daughterboard for quick and easy offboard wiring. Check the separate daughterboard document for details.

Be very careful when soldering the transistor. They're very sensitive to heat. You should use some kind of heat sink (crocodile clip or reverse action tweezers) on each leg as you solder them. Keep exposure to heat to a minimum (under 2 seconds).

Snap the small metal tag off the pots so they can be mounted flush in the box.

You should solder all other board-mounted components before you solder the pots. Once they're in place you'll have no access to much of the board. Make sure your pots all line up nicely.

The best way to do that is to solder a single pin of each pot in place then melt and adjust if necessary before soldering in the other two pins. If your pots don't have protective plastic jackets ensure you leave a decent gap between the pot body and the PCB otherwise you risk shorting out the circuit.

BIAS TRIMMER

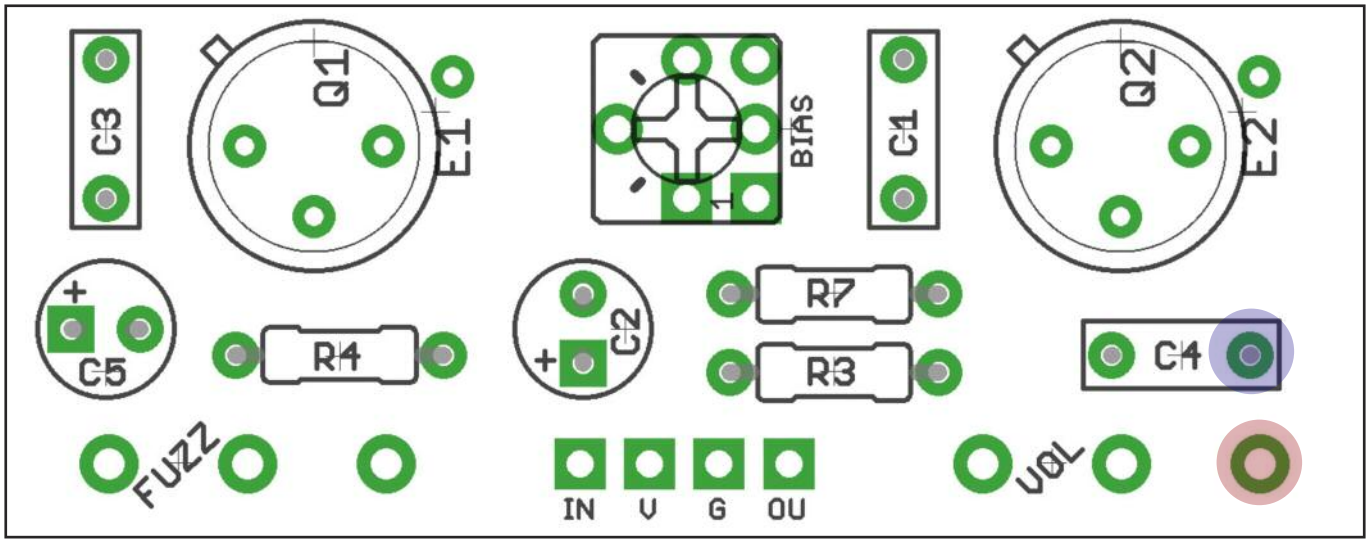
You'll notice there are six pads on the board for the trimmer, but you only have three legs. The extras are to allow different formats to be used. The pads are connected within the PCB as shown above in red. You need one leg in one pad of each of the connected pairs.

TRANSISTOR PINOUTS

We've added an extra pad for each transistor to make it easier to use cans without the standard EBC pinout, like some Russians models.

So, for instance... a standard pinout germanium 2N404 or silicon BC108 would sit neatly within the transistor outline shown on the silkscreen, utilising the triangular formation of EBC.
This can be seen on the cover image.

A Russian GT308B has the pinout BCE, so would mount into the holes in a diagonal line, like this:



BIASING

Once everything's soldered into place you need to adjust the voltage going to the collector of Q2, which sets the bias - the base voltage point from which the transistor will operate. Setting this correctly will give you the best balance of fuzz vs gating. You should set this to around 4.5V, or half your supply voltage.

To do this, set your multimeter to DC voltage with the range (if not auto-ranging) to the nearest voltage above 9 that is available, normally 20V.

Place your Common probe on any ground point, the most accessible being pin 1 of the Volume pot marked in red above.

The + probe should go onto the collector of Q2. If this isn't easily accessible you can use the empty pad, or the leg if you have a cap in there, of C4 marked in blue.

The above applies to NPN circuits. If you're building PNP reverse the probes, or you can just read the voltage as negative. It's all good.

NUMBERS AREN'T EVERYTHING...

Finding the correct bias by measuring is fine, but you may find you prefer the voltage adjusted a little one way or the other. Tweak by ear until you're happy. If you aren't, just get the meter back on and measure it again.

Test the board!

Check the relevant daughterboard document for more info before you undertake this stage.

UNDER NO CIRCUMSTANCES will troubleshooting help be offered if you have skipped this stage. No exceptions.

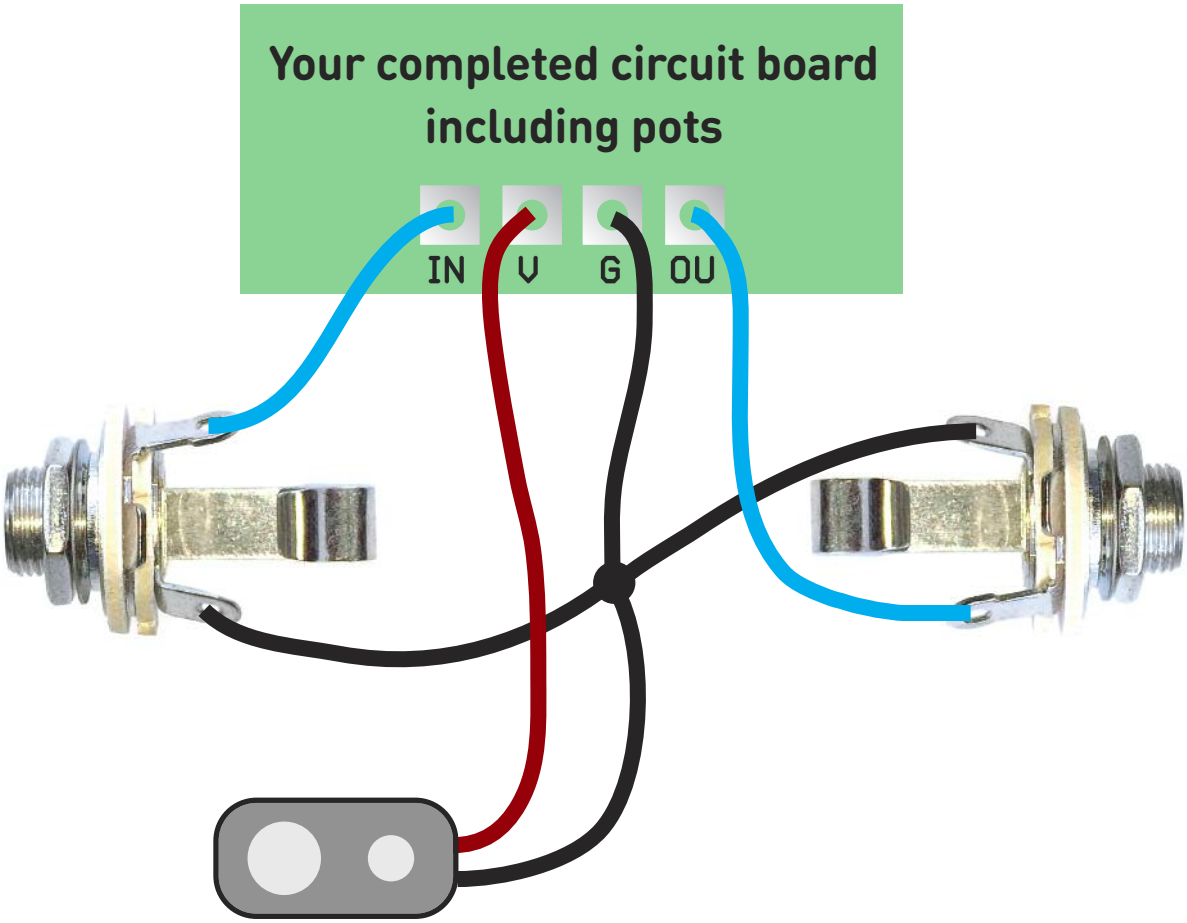
Once you've finished the circuit it makes sense to test is before starting on the switch and LED wiring. It'll cut down troubleshooting time in the long run. If the circuit works at this stage, but it doesn't once you wire up the switch - guess what? You've probably made a mistake with the switch.

Solder some nice, long lengths of wire to the board connections for 9V, GND, IN and OUT. Connect IN and OUT to the jacks as shown. Connect all the GNDs together (twist them up and add a small amount of solder to tack it). Connect the battery + lead to the 9V wire, same method. Plug in. Go!

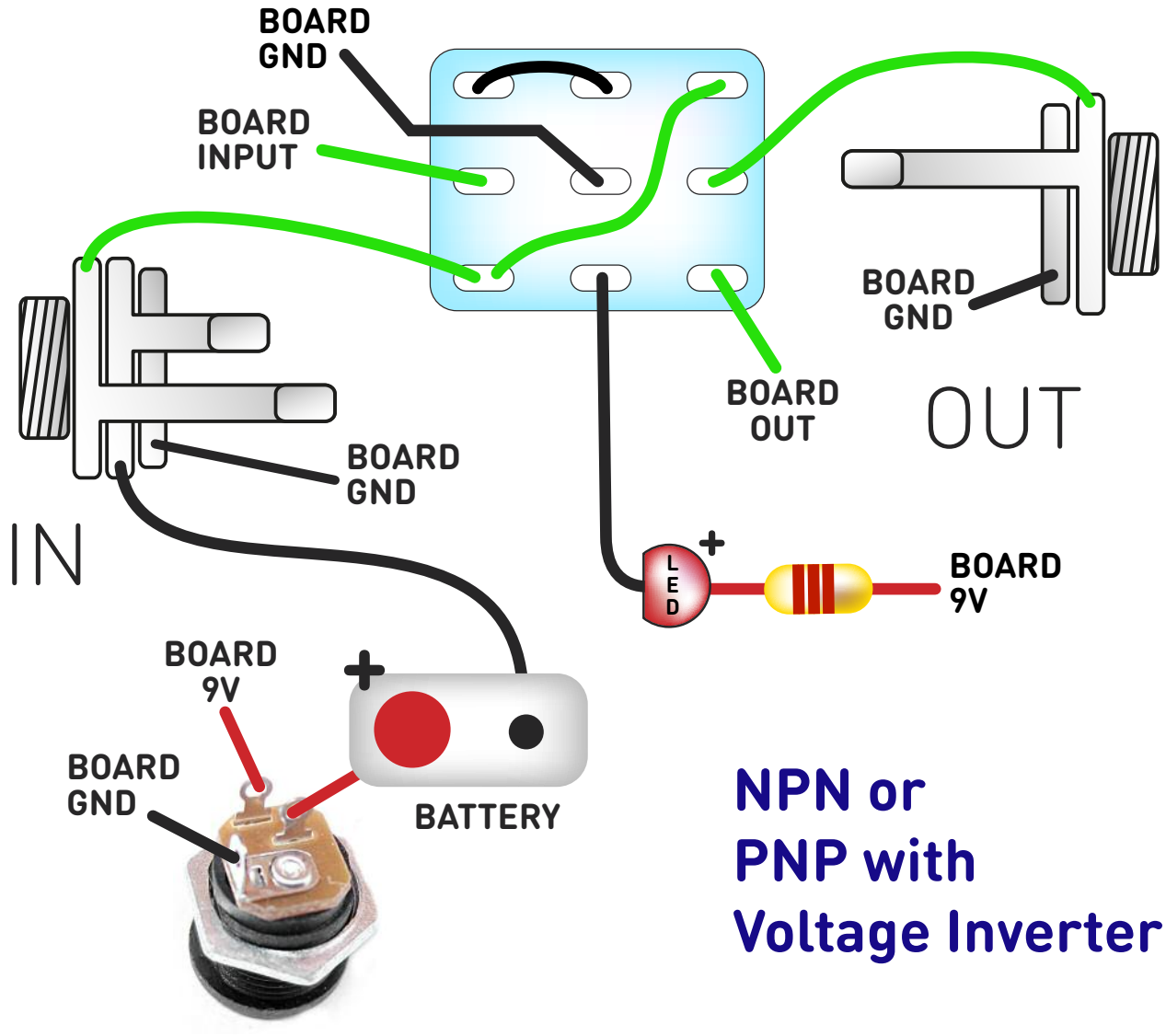
If you're using a ribbon cable you can tack the wires to the ends of that. It's a lot easier to take them off there than it is do desolder wires from the PCB pads.

If it works, carry on and do your switch wiring. If not... aw man. At least you know the problem is with the circuit. Find out why, get it working, THEN worry about the switch etc.

The wiring shown is for NPN/Negative ground builds. For PNP/Positive-ground builds switch the battery wires around. i.e. Red attaches to the jack GND lugs, black goes to the V pad.



Wire it up (if using a daughterboard please refer to the relevant document)



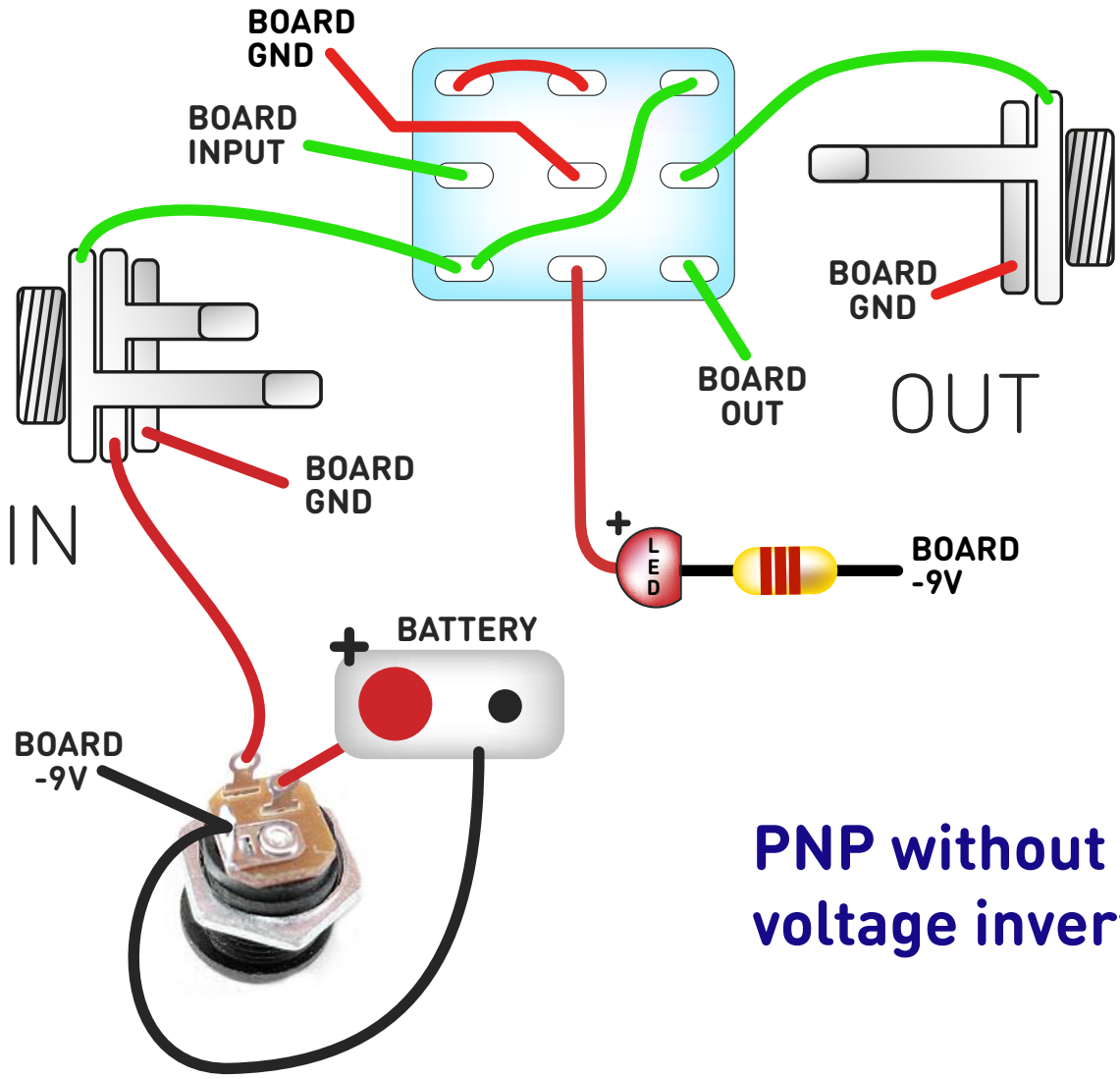
Wiring shown above will disconnect the battery when you remove the jack plug from the input, and also when a DC plug is inserted.

The Board GND connections don't all have to directly attach to the board. You can run a couple of wires from the DC connector, one to the board, another to the IN jack, then daisy chain that over to the OUT jack.

It doesn't matter how they all connect, as long as they do.

This circuit is standard, Negative GND. Your power supply should be Tip Negative / Sleeve Positive. That's the same as your standard pedals (Boss etc), and you can safely daisy-chain your supply to this pedal.

Wire it up (if using a daughterboard please refer to the relevant document)



PNP without voltage inverter

Wiring shown above will disconnect the battery when you remove the jack plug from the input, and also when a DC plug is inserted.

The Board GND connections don't all have to directly attach to the board. You can run a couple of wires from the DC connector, one to the board, another to the IN jack, then daisy chain that over to the OUT jack.

It doesn't matter how they all connect, as long as they do.

This circuit is positive GND. Your power supply should be Tip Negative / Sleeve Positive, but it cannot be daisy-chained with standard negative GND pedals.

Drilling template

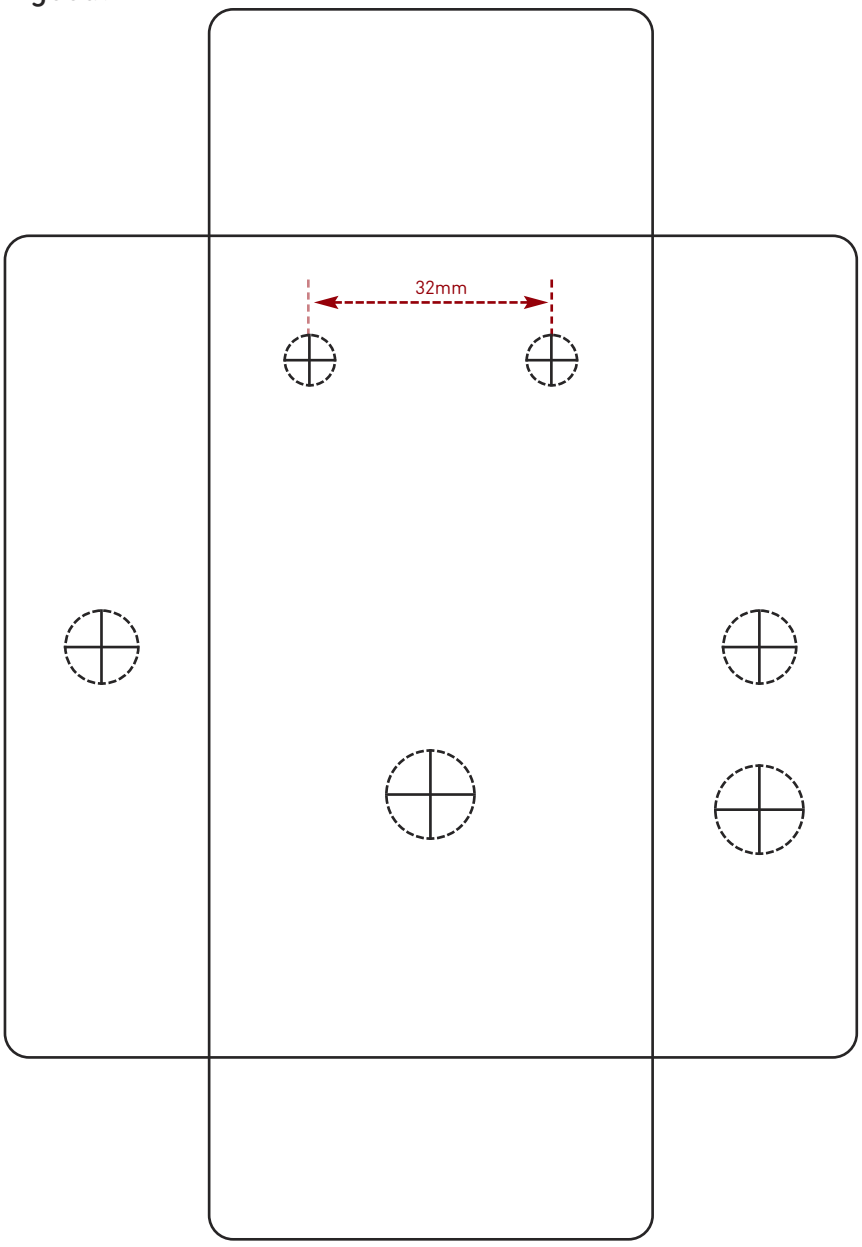
Hammond 1590B

60 x 111 x 31mm

It's a good idea to drill the pot and toggle switch holes 1mm bigger if you're board-mounting them.
Wiggle room = good!

Recommended drill sizes:

Pots	7mm
Jacks	10mm
Footswitch	12mm
DC Socket	12mm
Toggle switches	6mm



This template is a rough guide only. You should ensure correct marking of your enclosure before drilling. You use this template at your own risk.
Pedal Parts Ltd can accept no responsibility for incorrect drilling of enclosures.

FuzzDog.co.uk