

Kismet Stitch

Harmonic Percolator with
extra bits to fiddle on with



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:

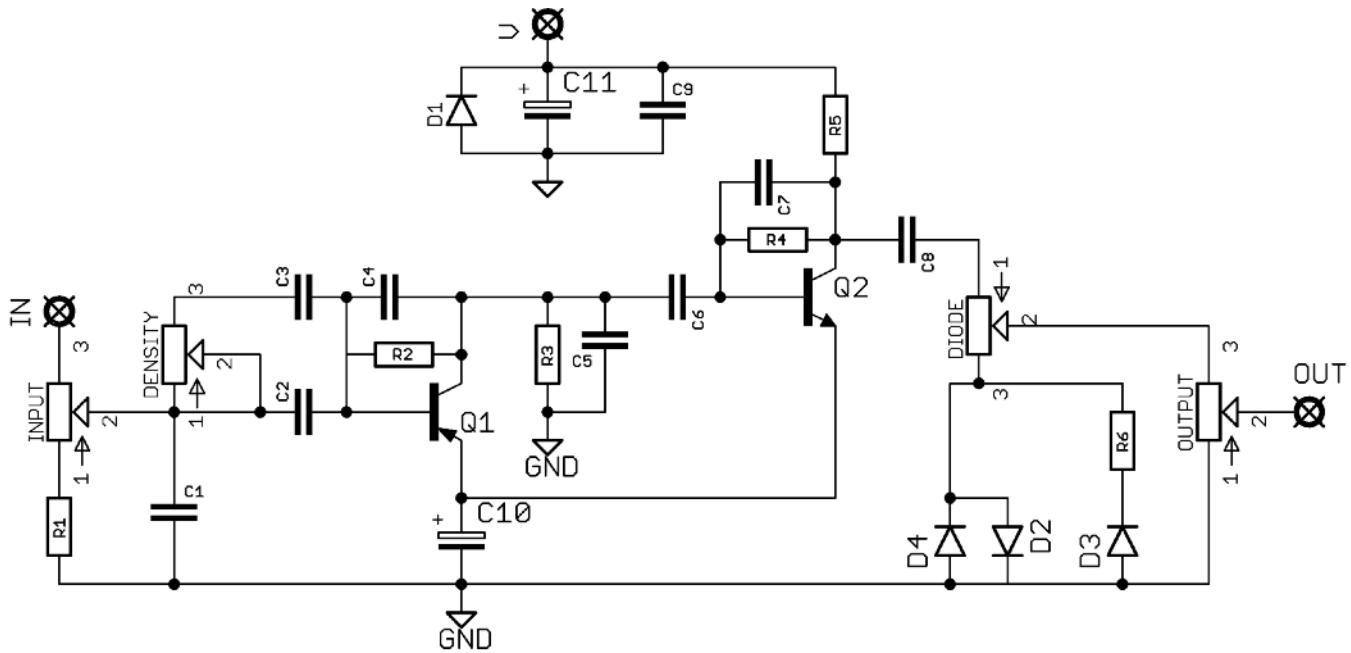
- Resistors should be 0.25W. You can use those with higher ratings but check the physical size of them.
- Electrolytics caps should be at least 25V for 9V circuits, 35V for 18V circuits. Again, check physical size if using higher ratings.

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 + BOM



Germanium Version

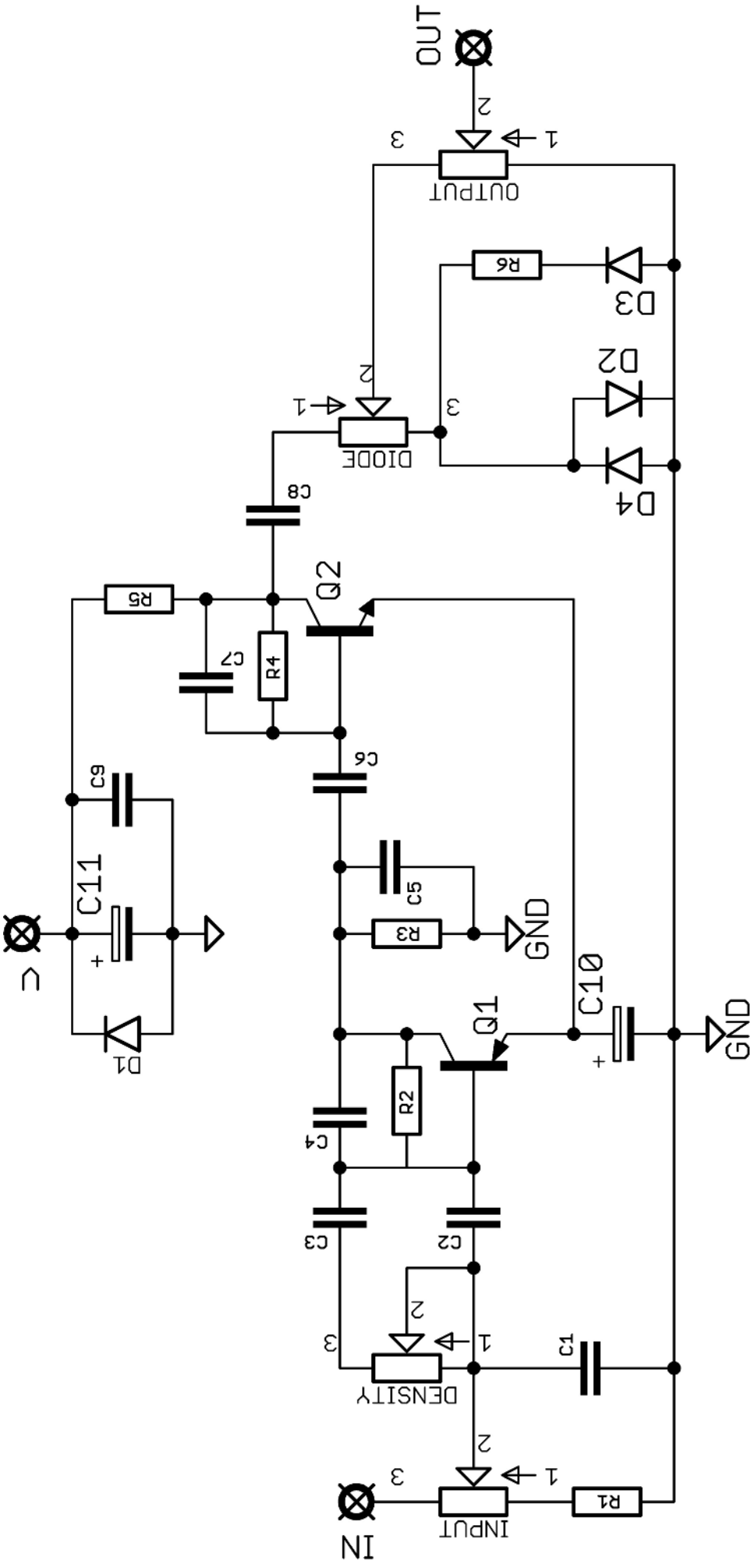
R1	4K7	C1	220p	C8	47n	D1	1N4001
R2	220K	C2	4n7	C9	100n	D2	1N4148
R3	33K	C3	220n	C10	47u elec	D3	Empty
R4	750K	C4	47p	C11	100u elec	D4	Germ**
R5	91K	C5	1n	Q1	1T308A*	INP	100KB
R6	Empty	C6	100n	Q2	PN2222A	DENS	500KC
		C7	330p			DIOD	50KB
						OUT	100KA

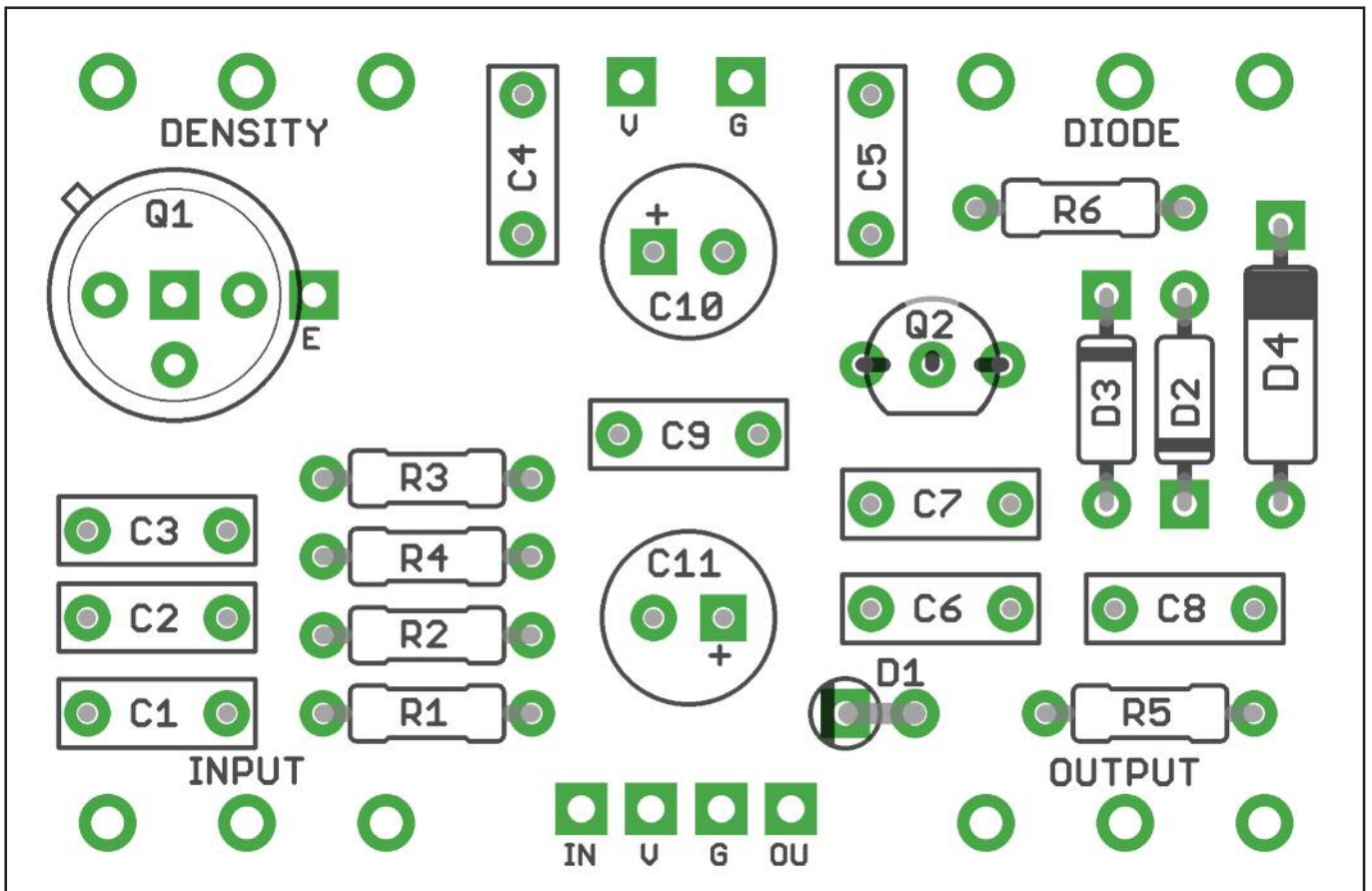
Silicon Version

R1	4K7	C1	47p	C8	47n	D1	1N4001
R2	150K	C2	4n7	C9	100n	D2	1N4148
R3	33K	C3	1u	C10	47u elec	D3	1N4148
R4	470K	C4	47p	C11	100u elec	D4	Empty
R5	91K	C5	1n	Q1	2N3906	INP	100KB
R6	5K6	C6	100n	Q2	PN2222A	DENS	500KC
		C7	100p			DIOD	50KB
						OUT	100KA

*Other transistors can use, and there are extra pads on the PCB to make it easier to do so. See page 6 for more info.

**1N34A, 1N60, Russian diodes such as D9E etc.





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.

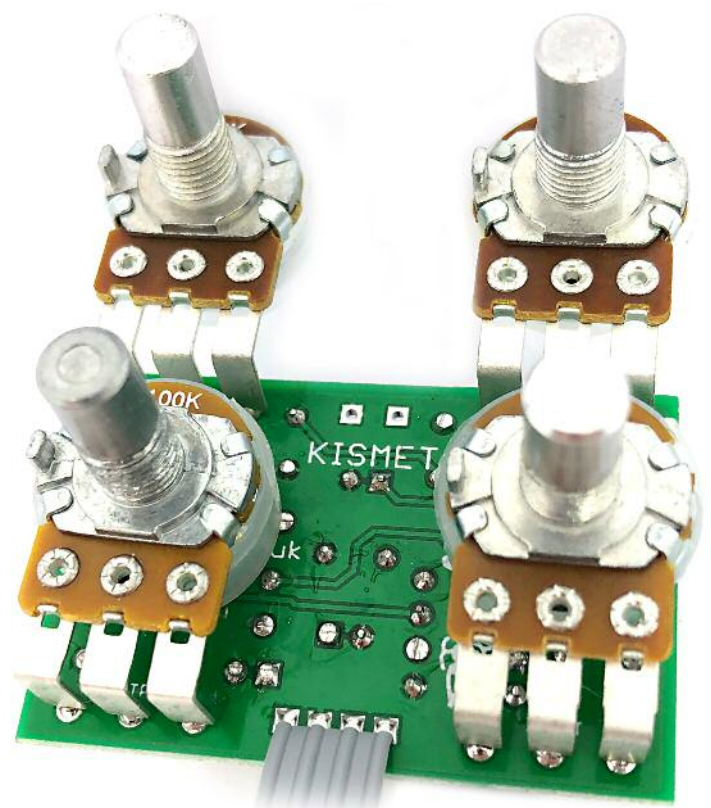
Be very careful when soldering the diodes, transistors and LED. They're very sensitive to heat. You should use some kind of heat sink (crocodile clip or reverse action tweezers) on each leg as you solder them. Keep exposure to heat to a minimum (under 2 seconds).

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.

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

The extra V and G pads at the top of the PCB can be used to connect your DC socket instead of those on the daughterboard if you want it top-mounted on the enclosure.



Transistor placement

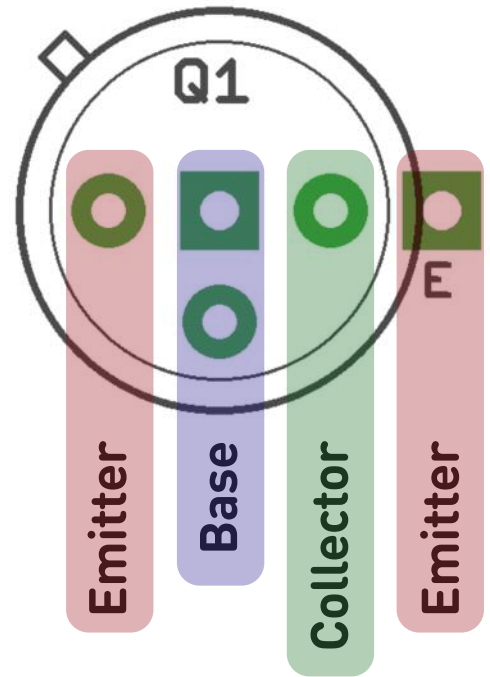
You'll notice extra pads on Q1. These make it easier to use different types of transistor in this position.

This shows what each pad is for >>>>

Your transistor legs will go into one pad for each of Emitter, Base and Collector. We've designed the part so that legs of different transistors will fit naturally into one of several patterns.

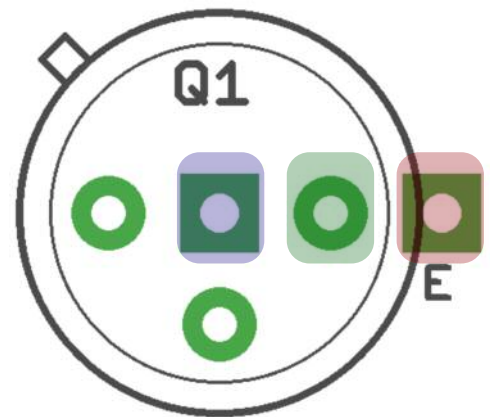
You don't have to use the stated transistors for Q1. Any PNP germanium can be tried in the germanium version - the original Harmonic Percolator used 2N404 and this works well in this version too.

Same goes for the 2N3906. Any PNP BJT should work.



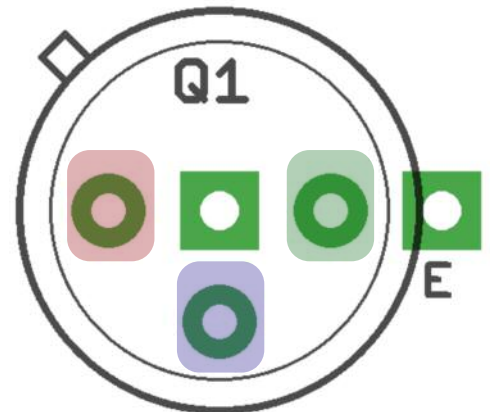
1T308A

This will fit into these three pads. The emitter (E) is marked with a dot on top of the casing.



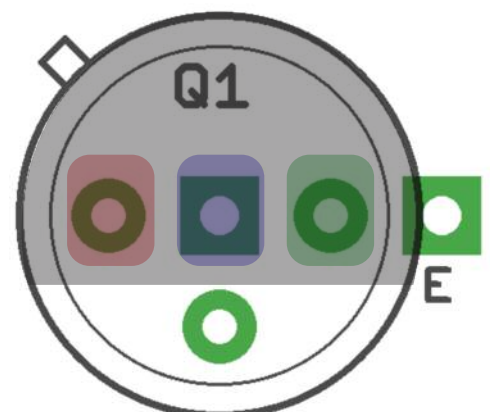
2N404, AC128 etc

This will fit into these three pads. The emitter is usually indicated with a tag on the casing the same as shown on the screen print. If there's not tag the collector will often be indicated with a painted dot.



2N3906

This will fit these pads with the flat side of the casing facing the bottom.



Test the board!

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

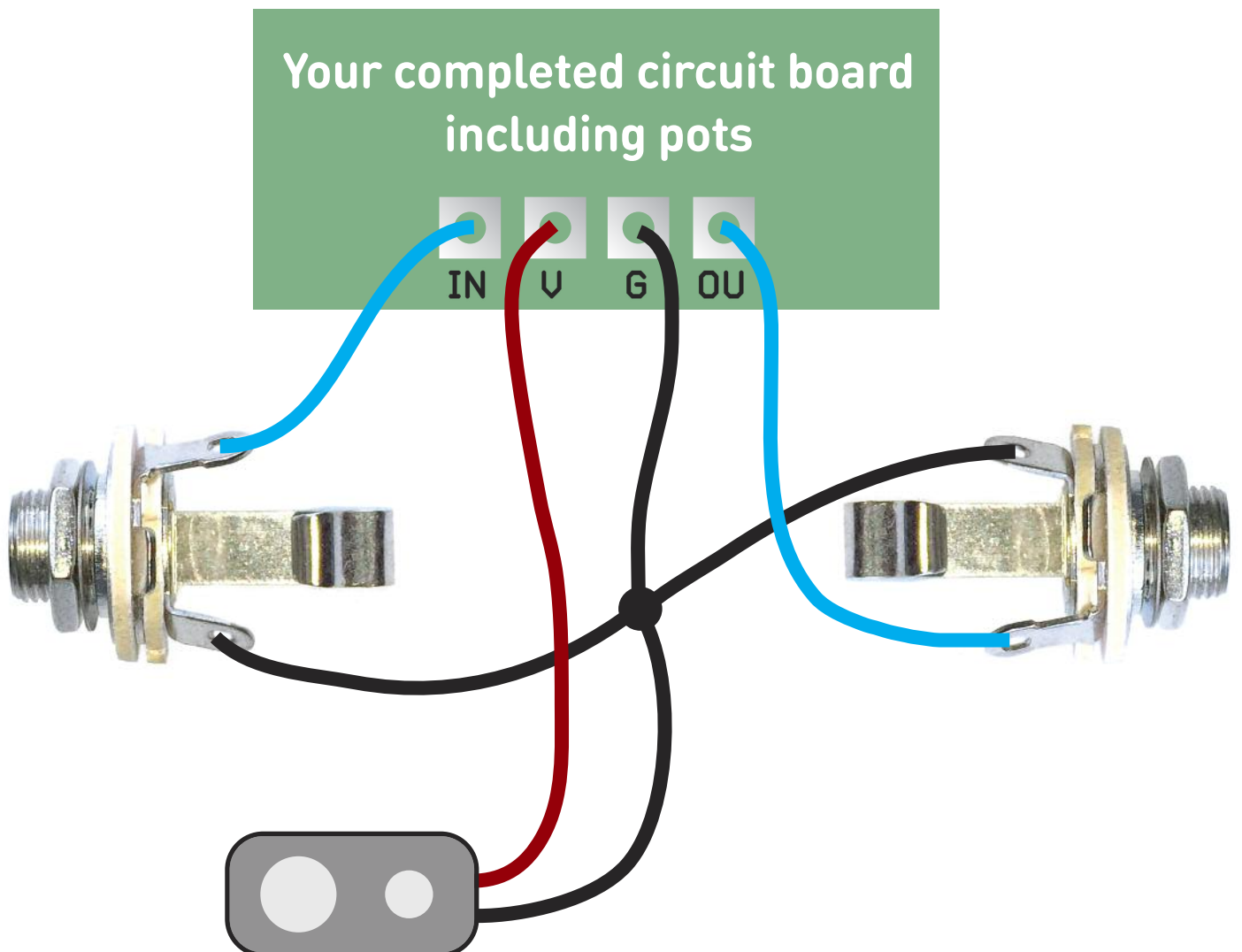
UNDER NO CIRCUMSTANCES will troubleshooting help be offered if you have skipped this stage. No exceptions.

Once you've finished the circuit it makes sense to test it 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.

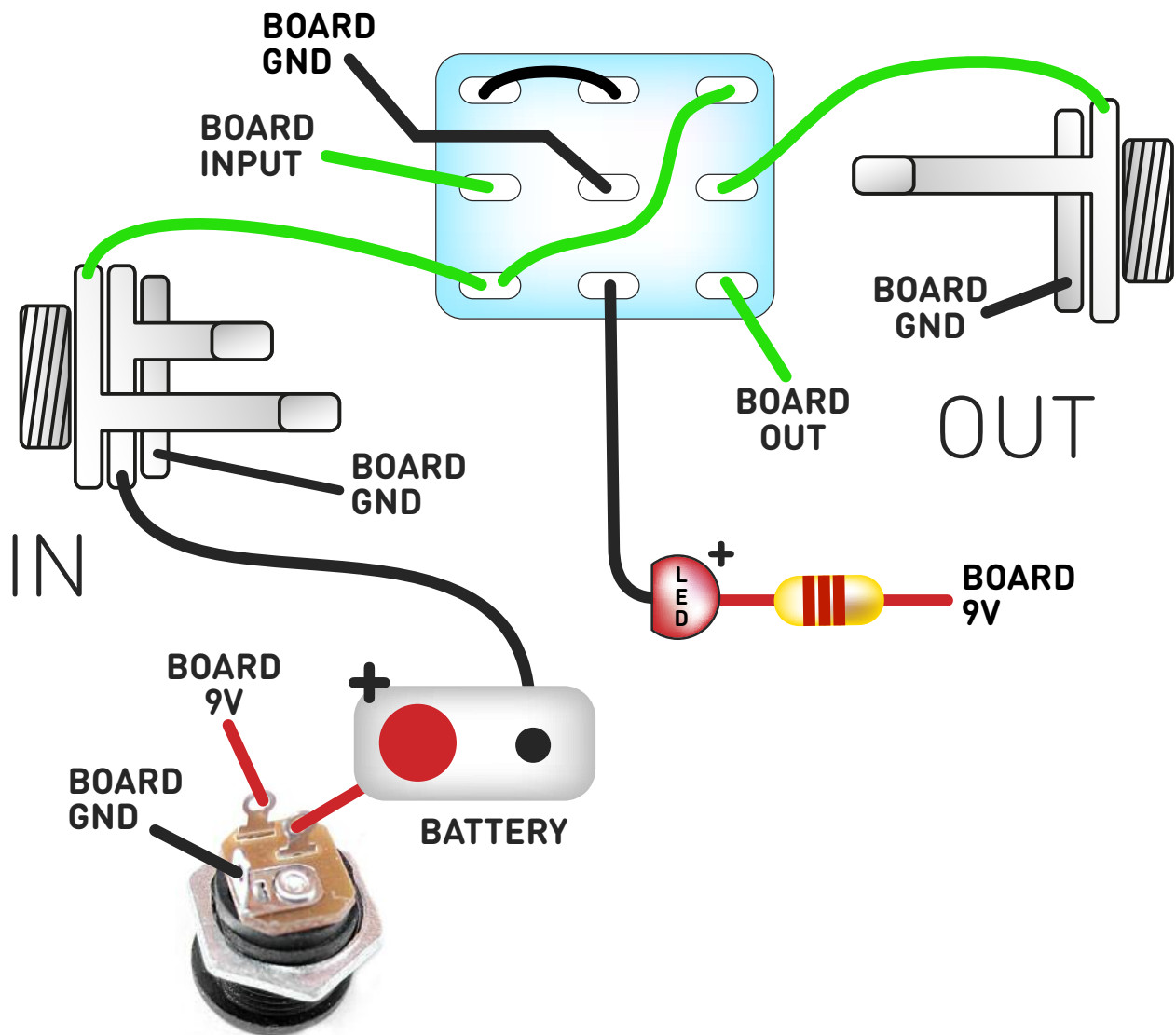
Solder some nice, long lengths of wire to the board connections for 9V, GND, IN and OUT. Connect IN and OUT to the jacks as shown. Connect all the GNDs together (twist them up and add a small amount of solder to tack it). Connect the battery + lead to the 9V wire, same method. Plug in. Go!

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 to 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.



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.

Drilling template

Hammond 1590B

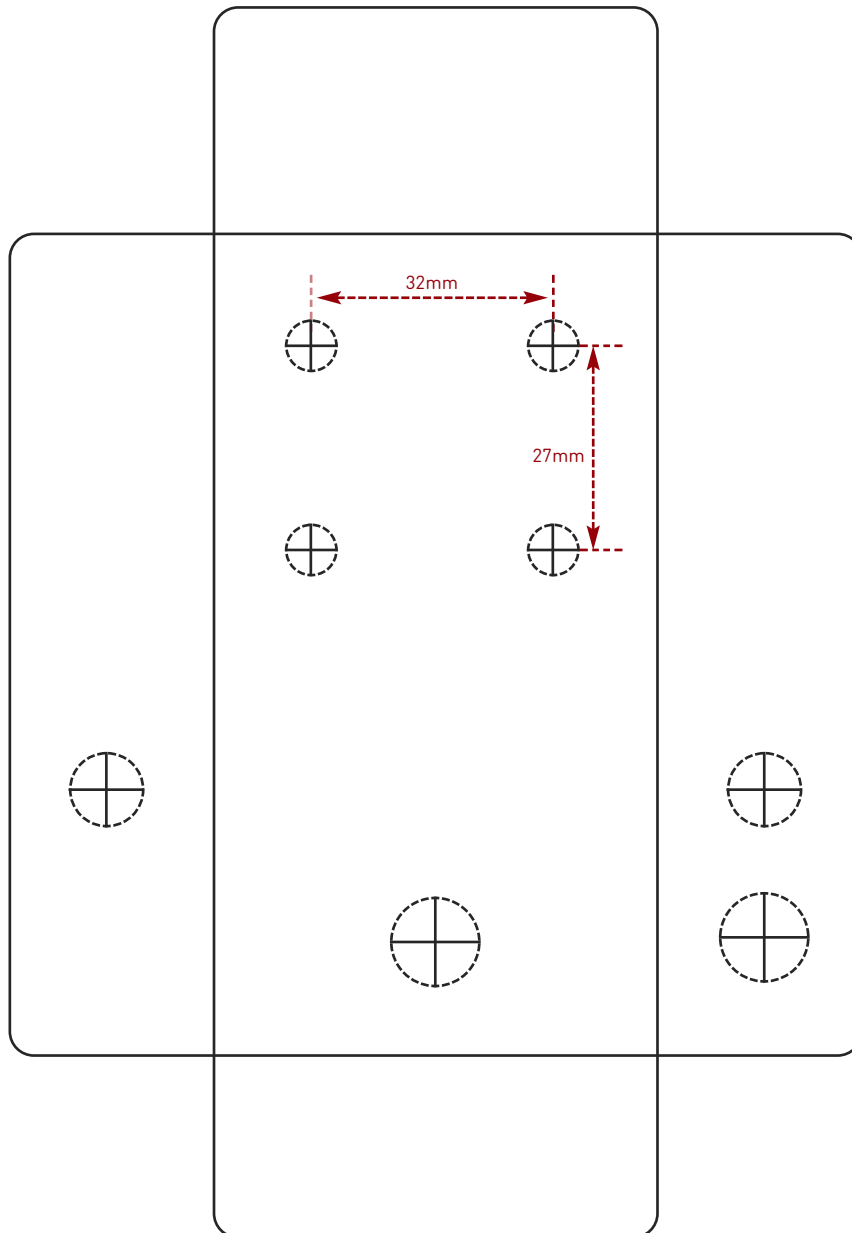
60 x 111 x 31mm

Recommended drill sizes:

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

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

Wiggle room = good!



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