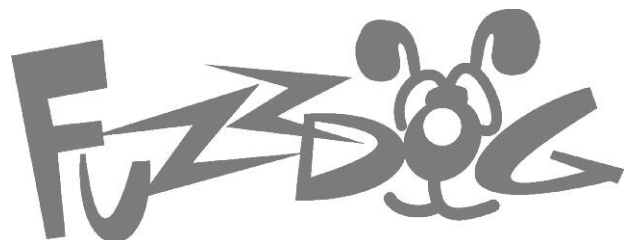


Sacred Utterance

Mystical Muff-based Drive



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:

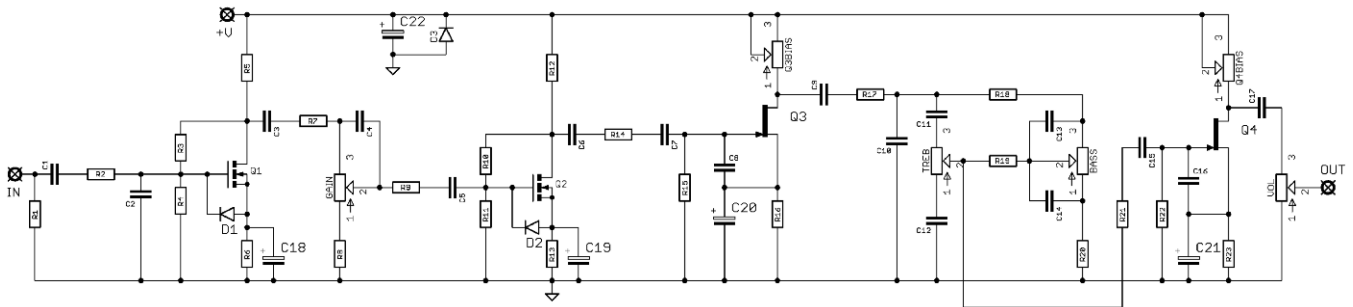
- Resistors should be 0.25W. You can use those with higher ratings but check the physical size of them.
- Electrolytics caps should be at least 25V for 9V circuits, 35V for 18V circuits. Again, check physical size if using higher ratings.

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.

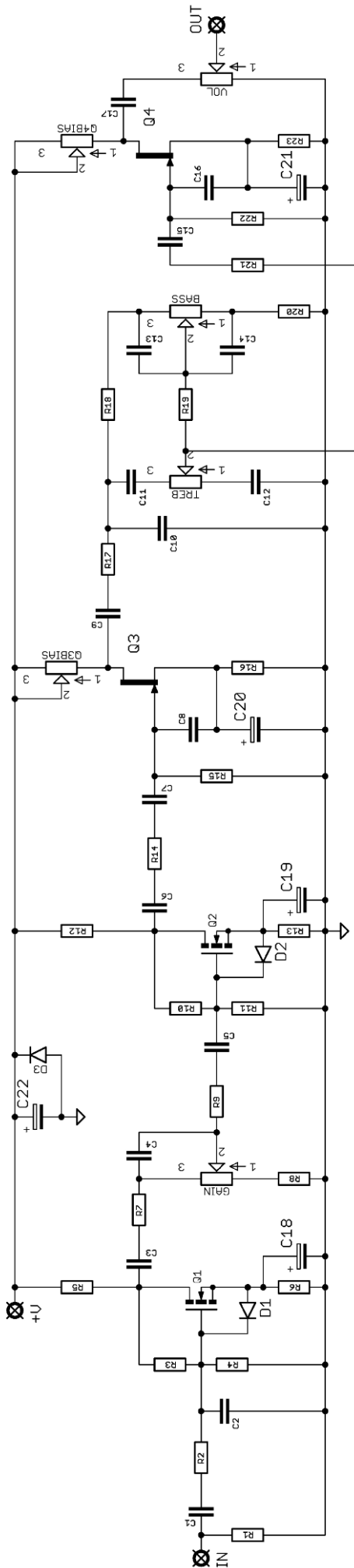
Schematic + BOM

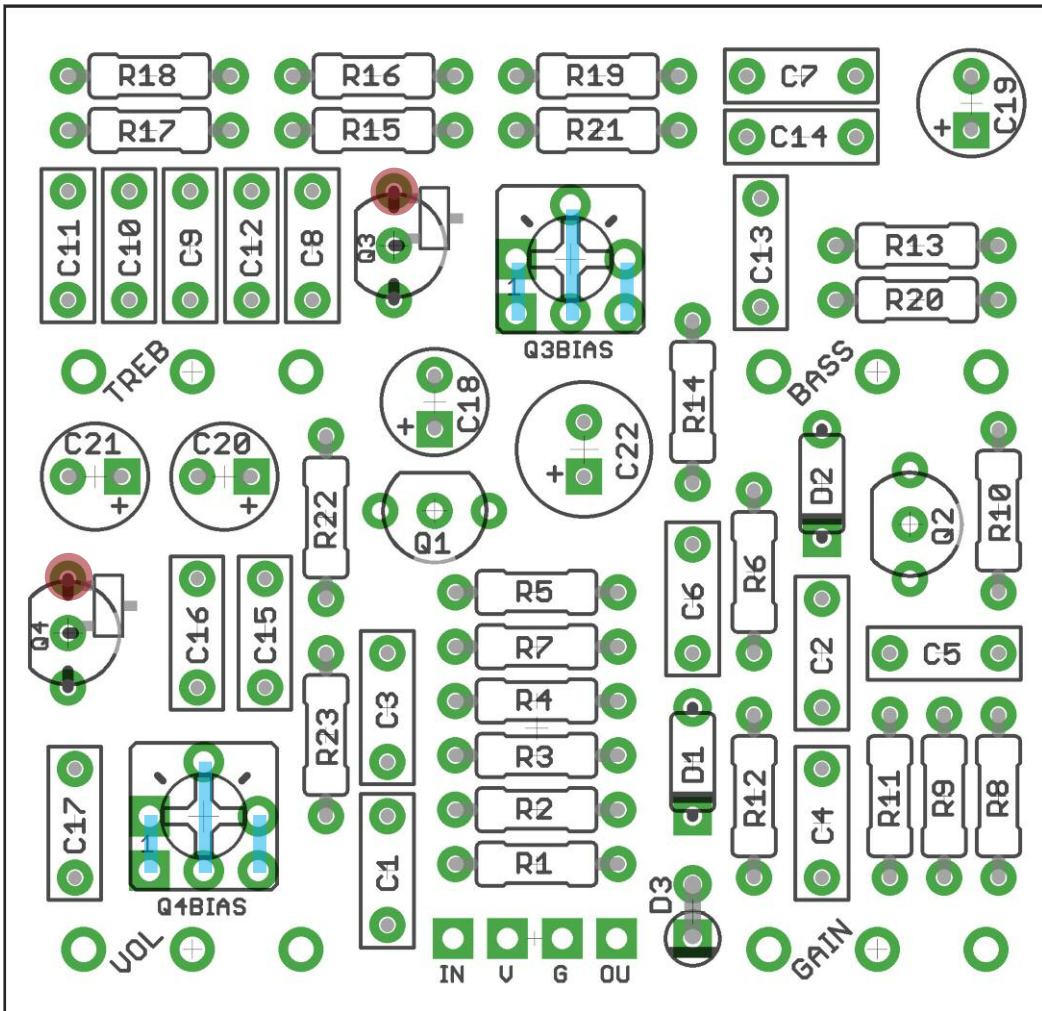


R1	1M	C1	10n	D1-2	9V1 zener
R2	10K	C2	47p	D3	1N4001
R3	2M2	C3	100n	Q1-2	2N7000
R4	2M2	C4	470p	Q3-4	J201*
R5	10K	C5	10n	BASS	100KA
R6	10K	C6	100n	TREB	100KA
R7	100K	C7	10n	GAIN	100KA
R8	10K	C8	47p	VOL	100KA
R9	390K	C9	100n	TRIMS	150-250K**
R10	2M2	C10	47p		
R11	2M2	C11	680p		
R12	10K	C12	3n3		
R13	10K	C13	1n		
R14	470K	C14	3n3		
R15	2M2	C15	10n		
R16	10K	C16	47p		
R17	470K	C17	100n		
R18	100K	C18	10u elec		
R19	100K	C19	10u elec		
R20	10K	C20	10u elec		
R21	470K	C21	22u elec		
R22	2M2	C22	100u elec		
R23	10K				

*Pads are included for through-hole J201 or their SMT equivalent MMBFJ201 which will be easier to source (though harder to solder)

**150K will be enough, but 220K or 250K will be easier to get hold of.





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 FETS and diodes. 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).

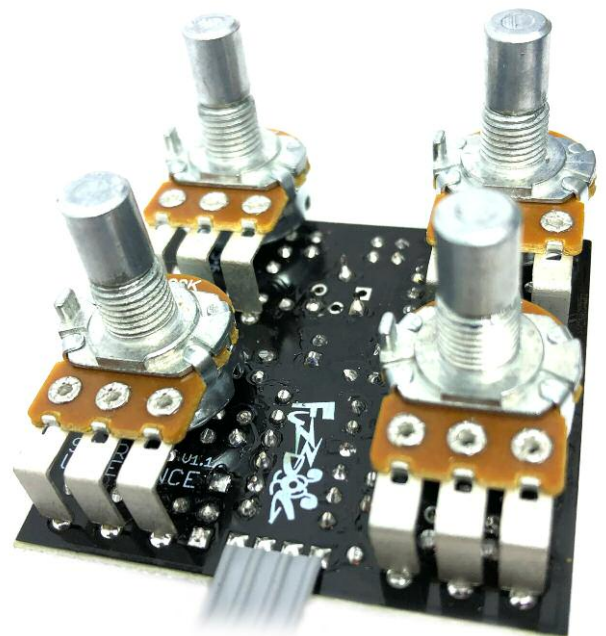
Snap the small metal tag off the pots so they can be mounted flush in the box.

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.

There are extra pads on trimmers to allow different package formats to be used. Pads are connected via PCB traces as shown above, so just fit your trimmer into whichever holes it fits naturally into. As long as you have one pin each in the left, centre and right sections. No jumpers are required.

Use the trimmers to adjust the bias of Q3-4 to be approx 6V at the points marked in red.



Test the board!

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

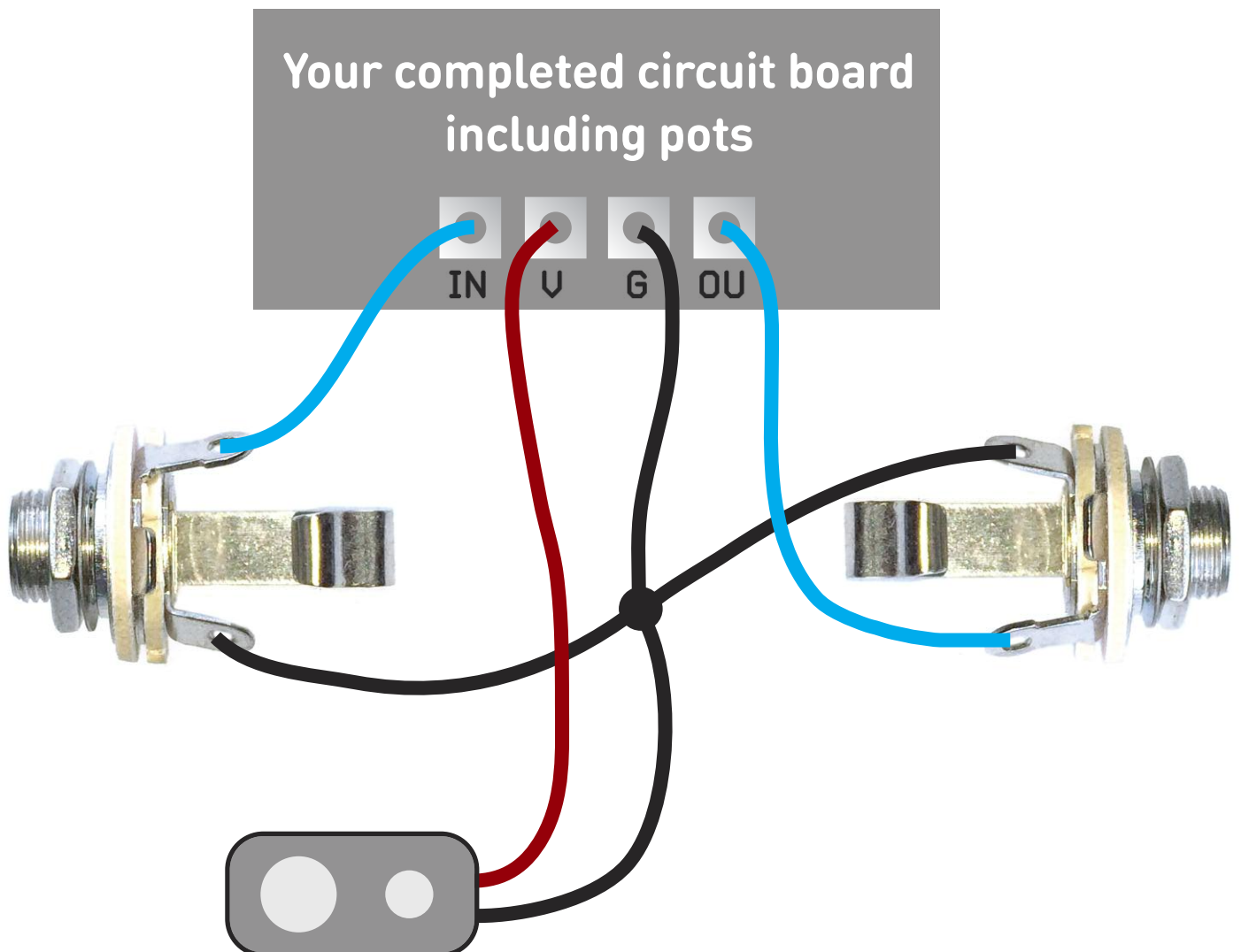
UNDER NO CIRCUMSTANCES will troubleshooting help be offered if you have skipped this stage. No exceptions.

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.

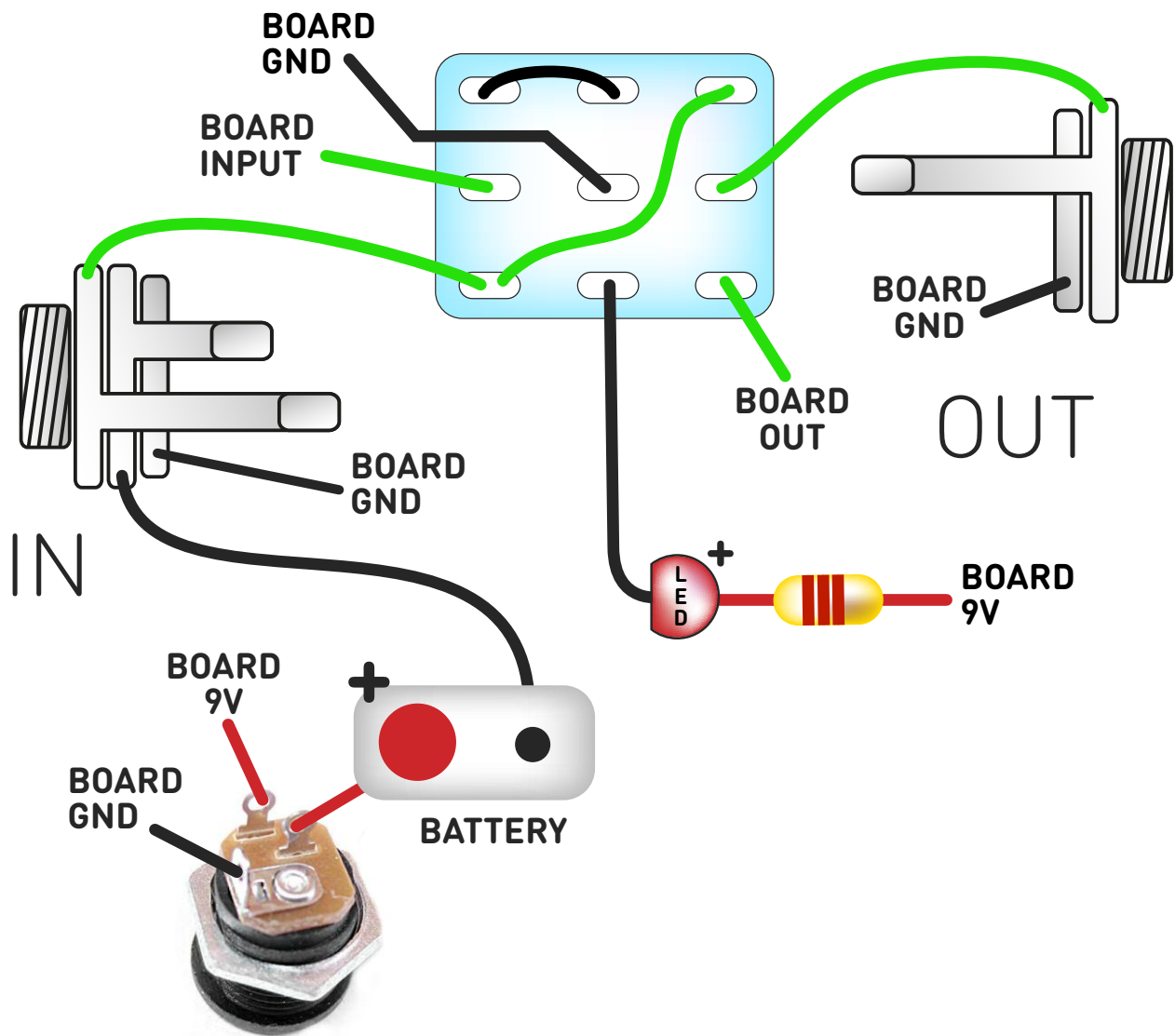
Solder some nice, long lengths of wire to the board connections for 9V, GND, IN and OUT. Connect IN and OUT to the jacks as shown. Connect all the GNDs together (twist them up and add a small amount of solder to tack it). Connect the battery + lead to the 9V wire, same method. Plug in. Go!

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.



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.

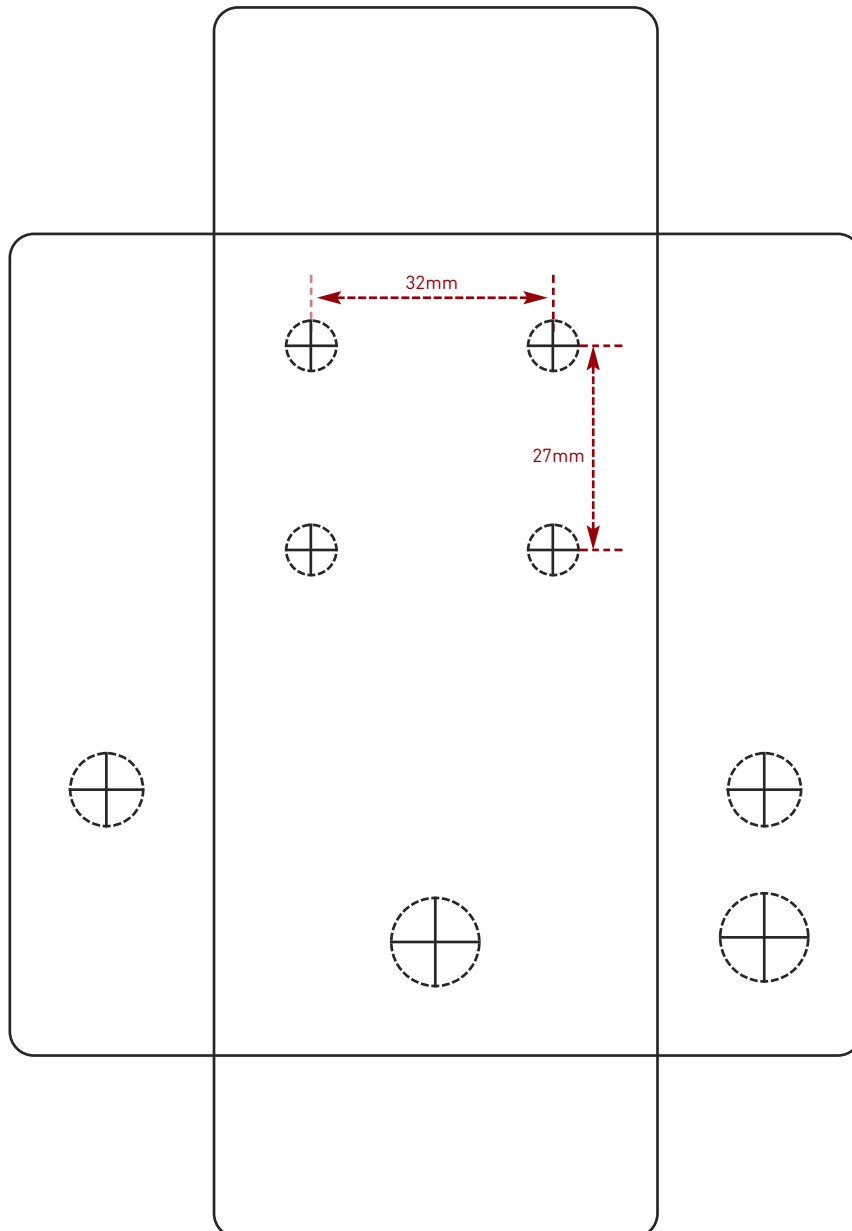
Drilling template

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