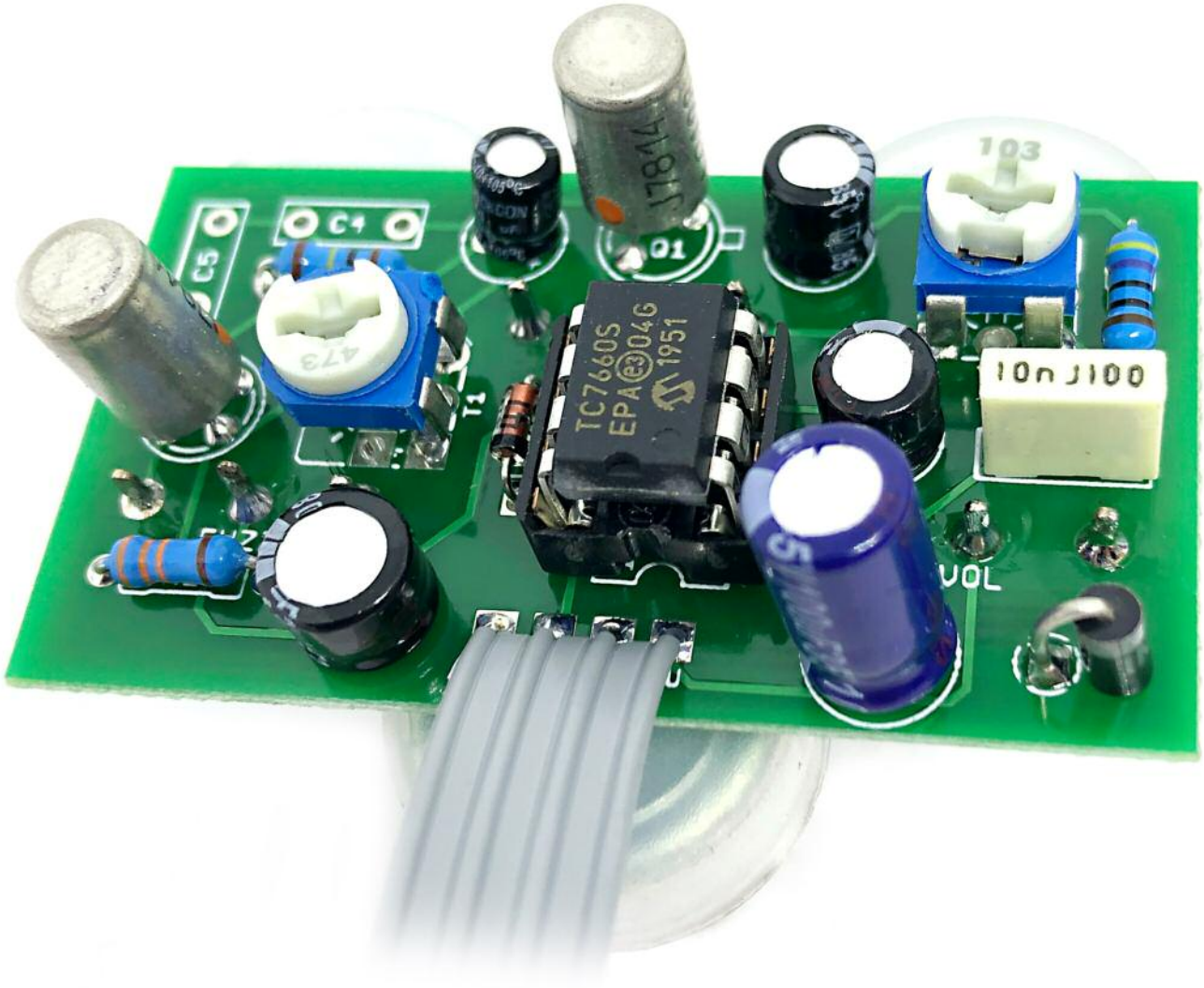


Solar Face

Si or Ge Fuzz Face with a magic bias dial

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:

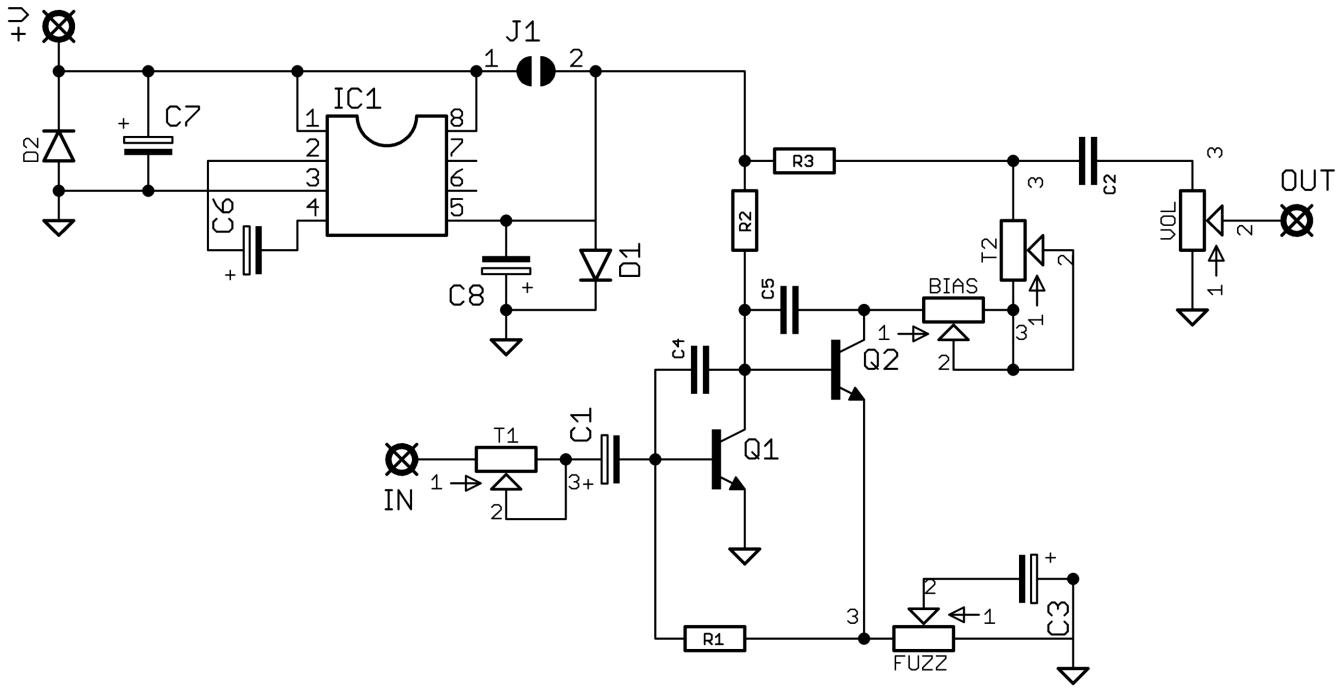
- Resistors should be 0.25W. You can use those with higher ratings but check the physical size of them.
- Electrolytics caps should be at least 25V for 9V circuits, 35V for 18V circuits. Again, check physical size if using higher ratings.

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 + Notes

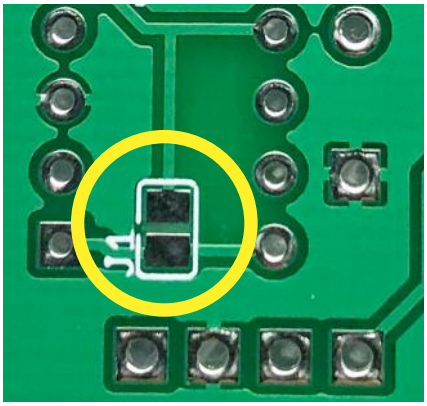


This is the full schematic including the following optional parts:

Voltage Inverter (D1, IC1, C6, C8) to make a positive ground build (using PNP transistors) work with standard negative ground supplies.

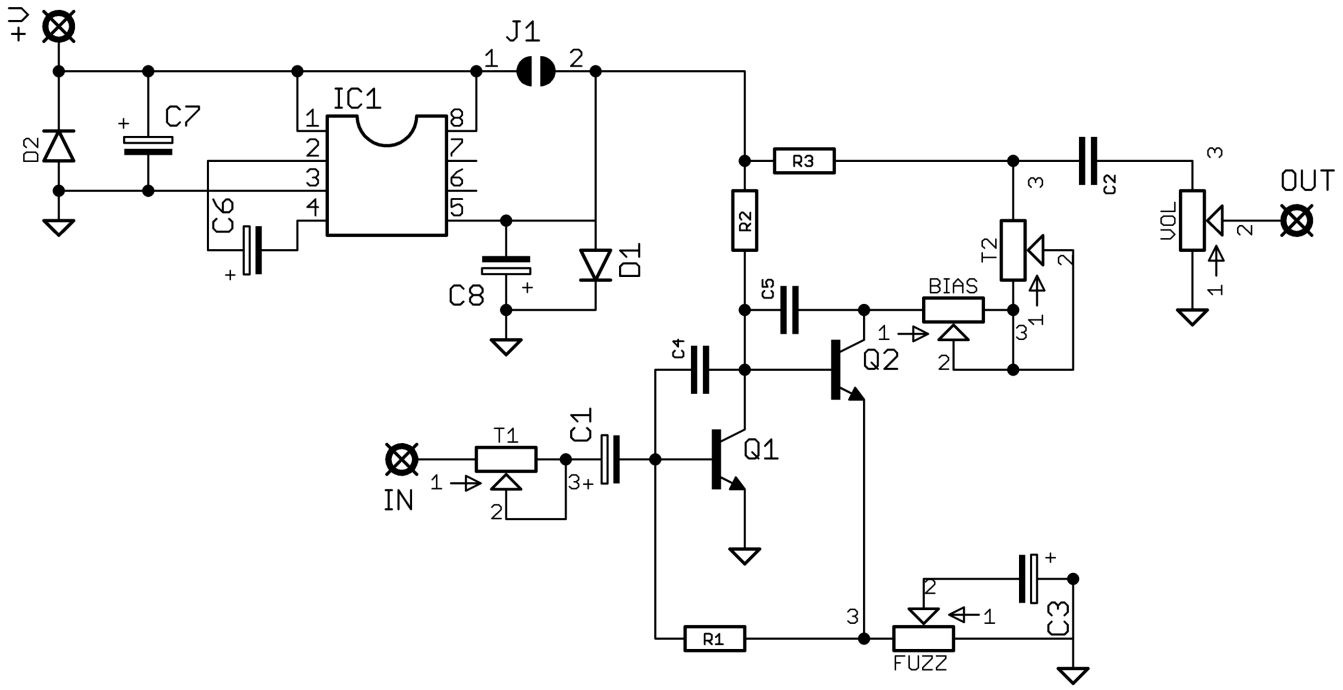
To use the voltage inverter simply add these parts.

If you're building a NPN / negative ground circuit, so don't require the inverter, leave these parts out and jumper these pads on the back of the PCB with a blob of solder.



Smoothing Caps (C4, C5) to prevent high frequency oscillation on builds with higher gain silicon transistors. These aren't required on builds using germanium transistors in the standard Fuzz Face gain ranges.

Schematic + BOM PNP/Ge



R1	100K
R2	33K
R3	470R

C1	1u elec
C2	10n
C3	22u elec
C4	Empty
C5	Empty
C6	10u elec
C7	100u elec*
C8	10u elec

D1	1N4148
D2	1N4001*
IC1	TL7660SEPA
Q1-2**	Ge Fuzz Face set
BIAS	5KB
FUZZ	1KC
VOL	250KA
T1	47-50K
T2	10K

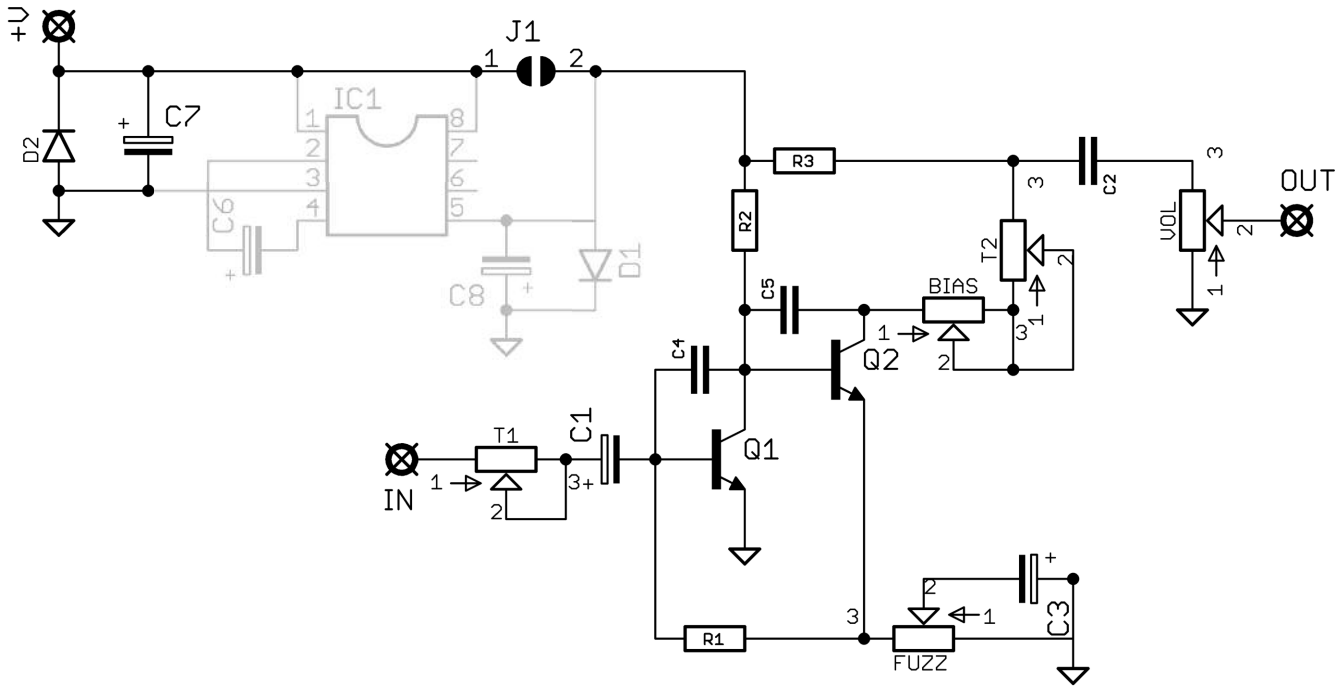
Parts listed in green are the optional voltage inverter. If you're not adding this leave these out and connect the pads on the back of the board as detailed on the previous page with a blob of solder.

You must use the PNP wiring diagram if you aren't using the inverter and reverse C7 and D2.

**Typical transistor gain values for a Fuzz Face are:

Q1 - 70ish
Q2 - 120ish

Schematic + BOM NPN/Si



R1	100K
R2	33K
R3	470R

C1	1u elec***
C2	10n
C3	22u elec***
C4	*
C5	*
C6	Empty
C7	100u elec
C8	Empty

D1	Empty
D2	1N4001
IC1	Empty
Q1-2**	BC108 etc
BIAS	5KB
FUZZ	1KC
VOL	250KA
T1	47-50K
T2	10K

*You should use small value ceramic caps for C4-5. The larger the value the more top-end will be removed from the signal.

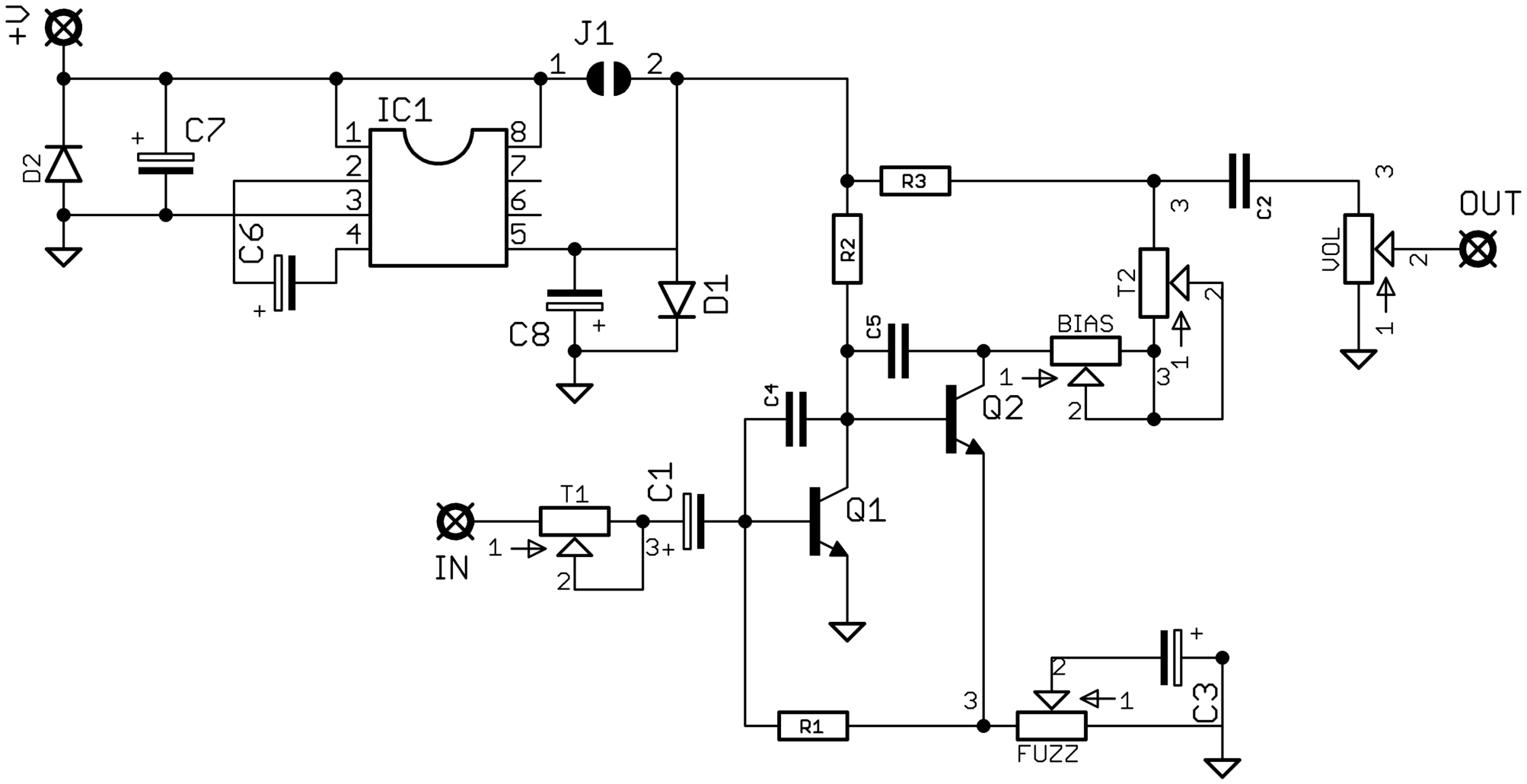
We go for C4 - 470p, C5 100p

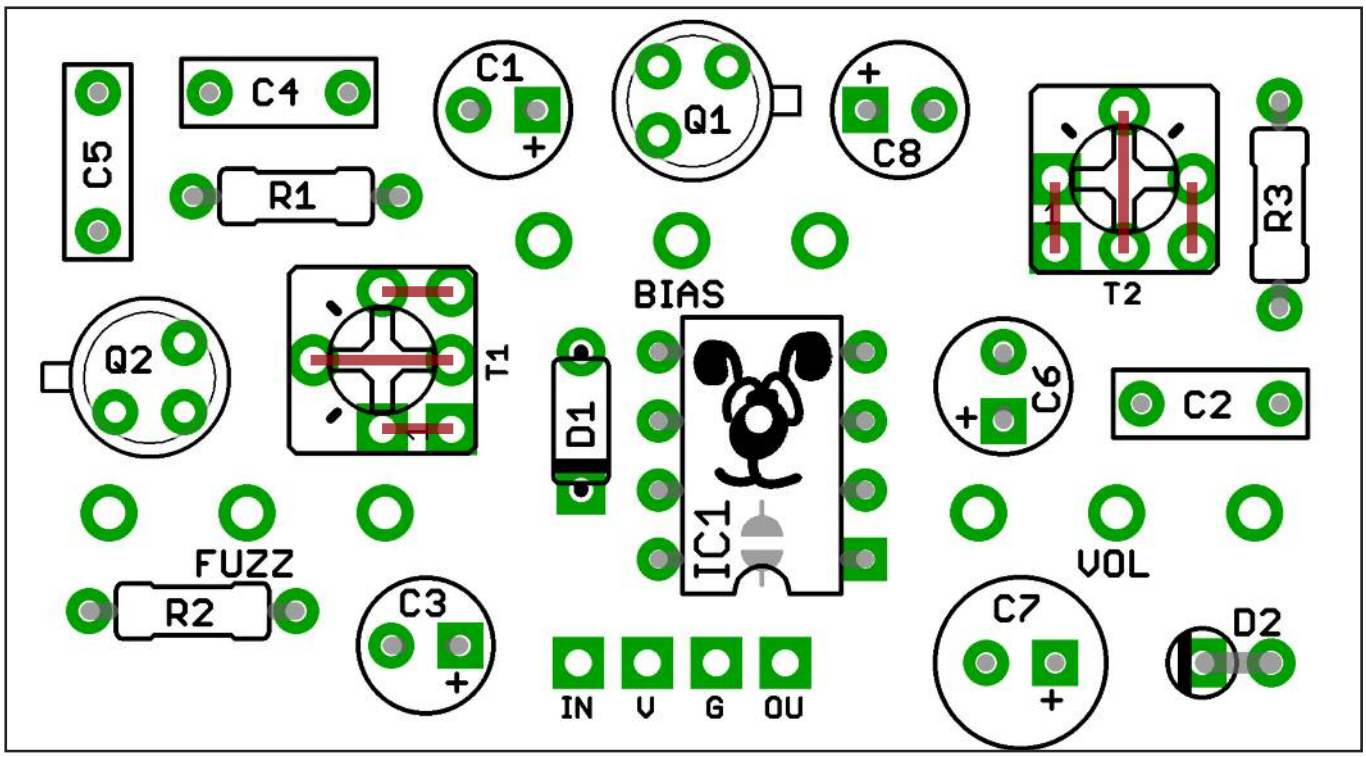
**Feel free to try any low-medium gain BJT transistors.

***Reverse C1 and C3 for this configuration.

Connect the pads on the back of the board as detailed on the page 3 with a blob of solder.

This configuration will also work for NPN Germanium transistors, but you won't need C4-5.


+V
J1
1
2
IC1
C7
D2
C6
C8
D1
R3
R2
C2
T2
VOL
OUT
C5
BIAS
Q2
C4
T1
C1
Q1
IN
R1
FUZZ
C3



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 and transistors. 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 **T1** to adjust the signal going into the circuit for more/less overall gain.

Use **T2** to adjust the minimum resistance for the bias control so your bias pot gives you the range you desire. Somewhere near the centre point is always good, but it's up to you if you want more extreme biasing.

Test the board!

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

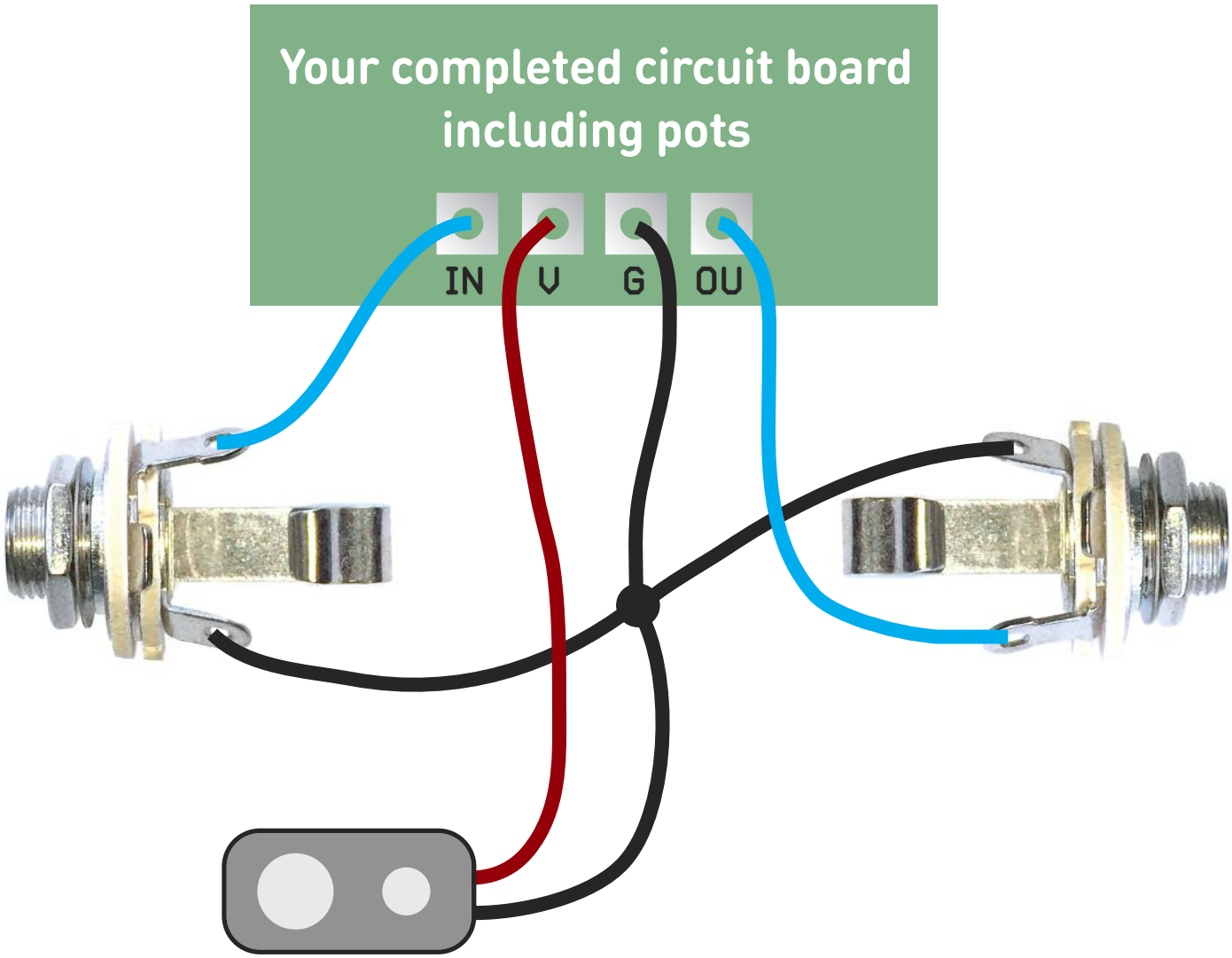
UNDER NO CIRCUMSTANCES will troubleshooting help be offered if you have skipped this stage. No exceptions.

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.

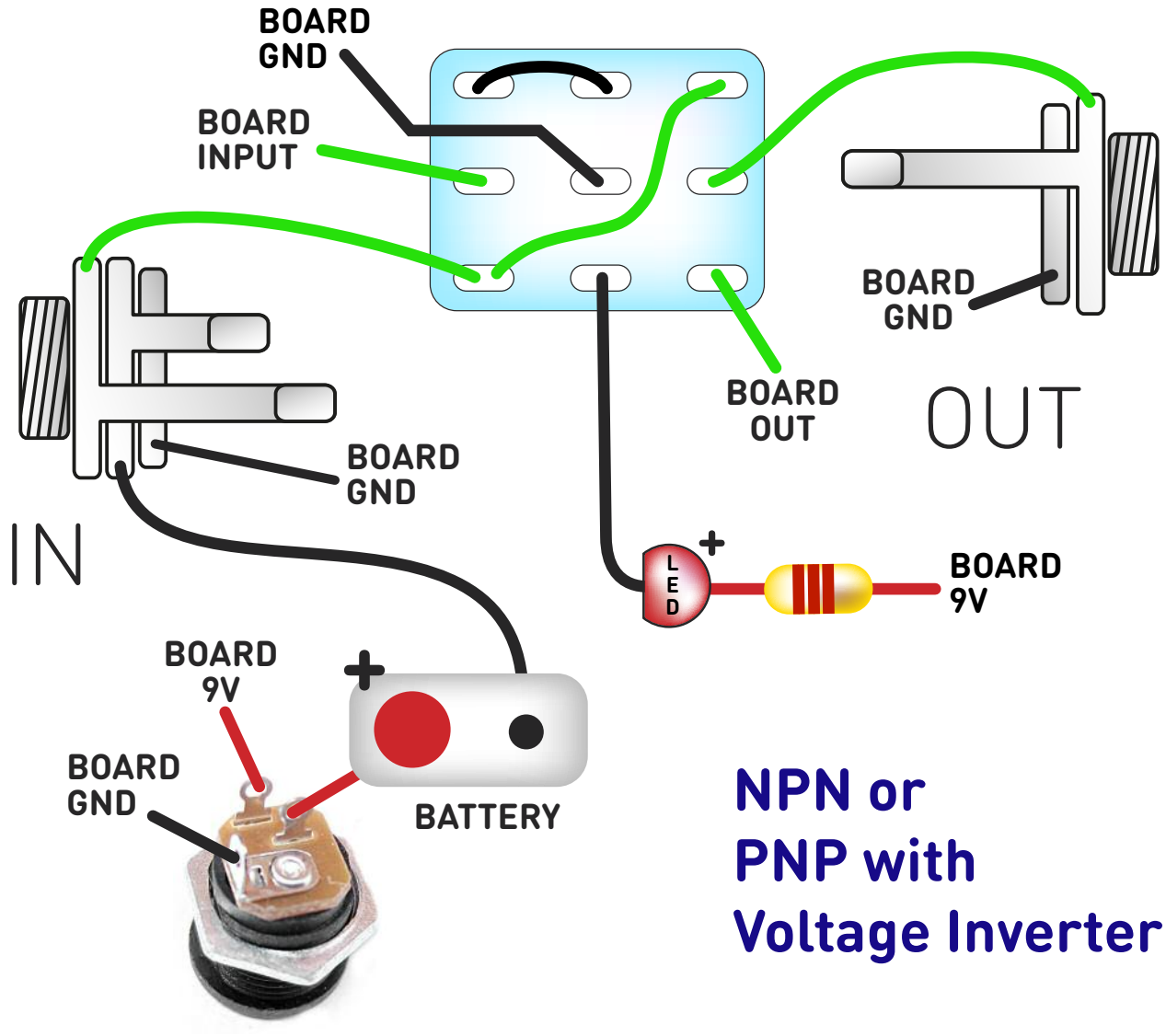
Solder some nice, long lengths of wire to the board connections for 9V, GND, IN and OUT. Connect IN and OUT to the jacks as shown. Connect all the GNDs together (twist them up and add a small amount of solder to tack it). Connect the battery + lead to the 9V wire, same method. Plug in. Go!

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)



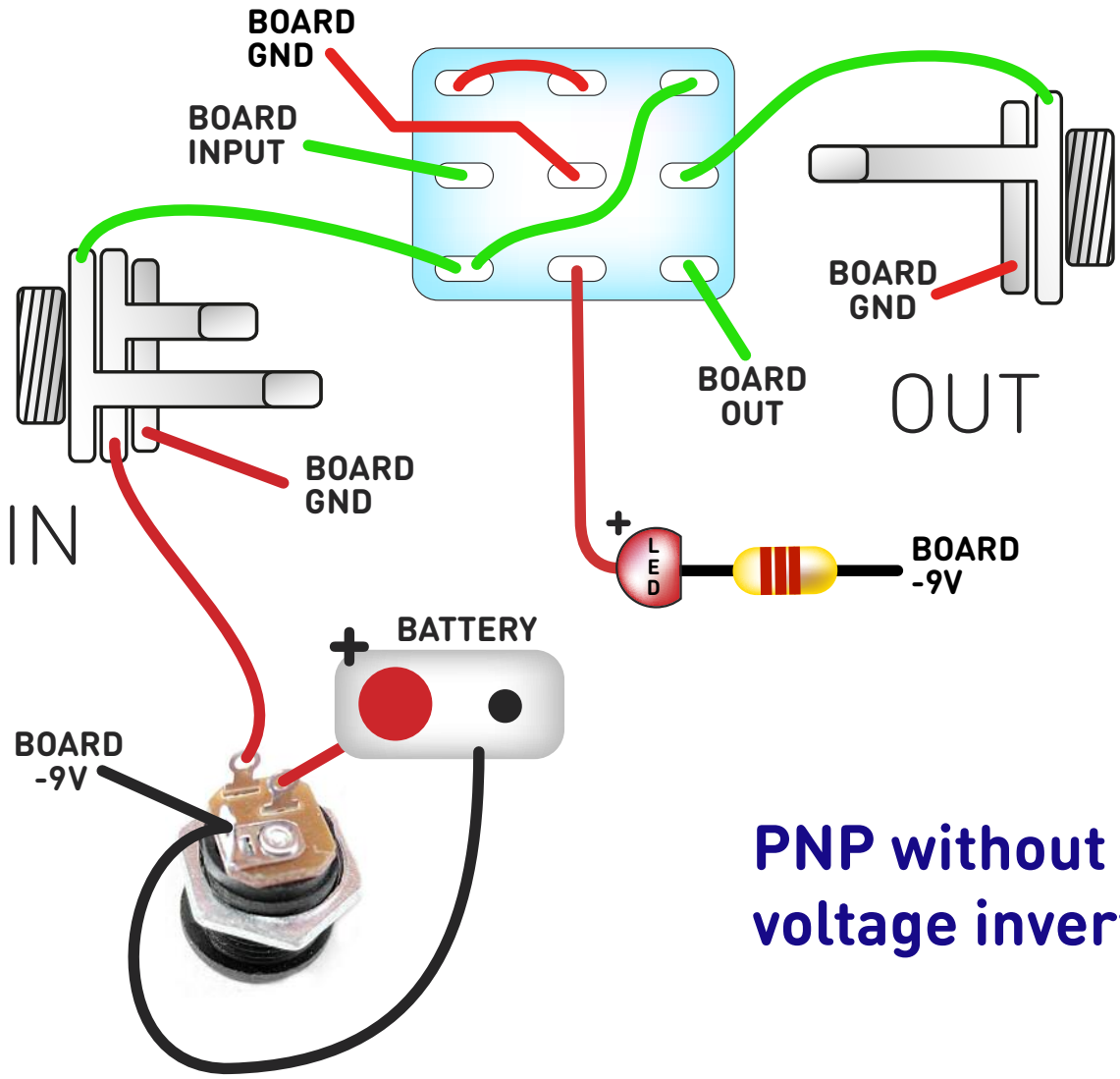
Wiring shown above will disconnect the battery when you remove the jack plug from the input, and also when a DC plug is inserted.

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.

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.

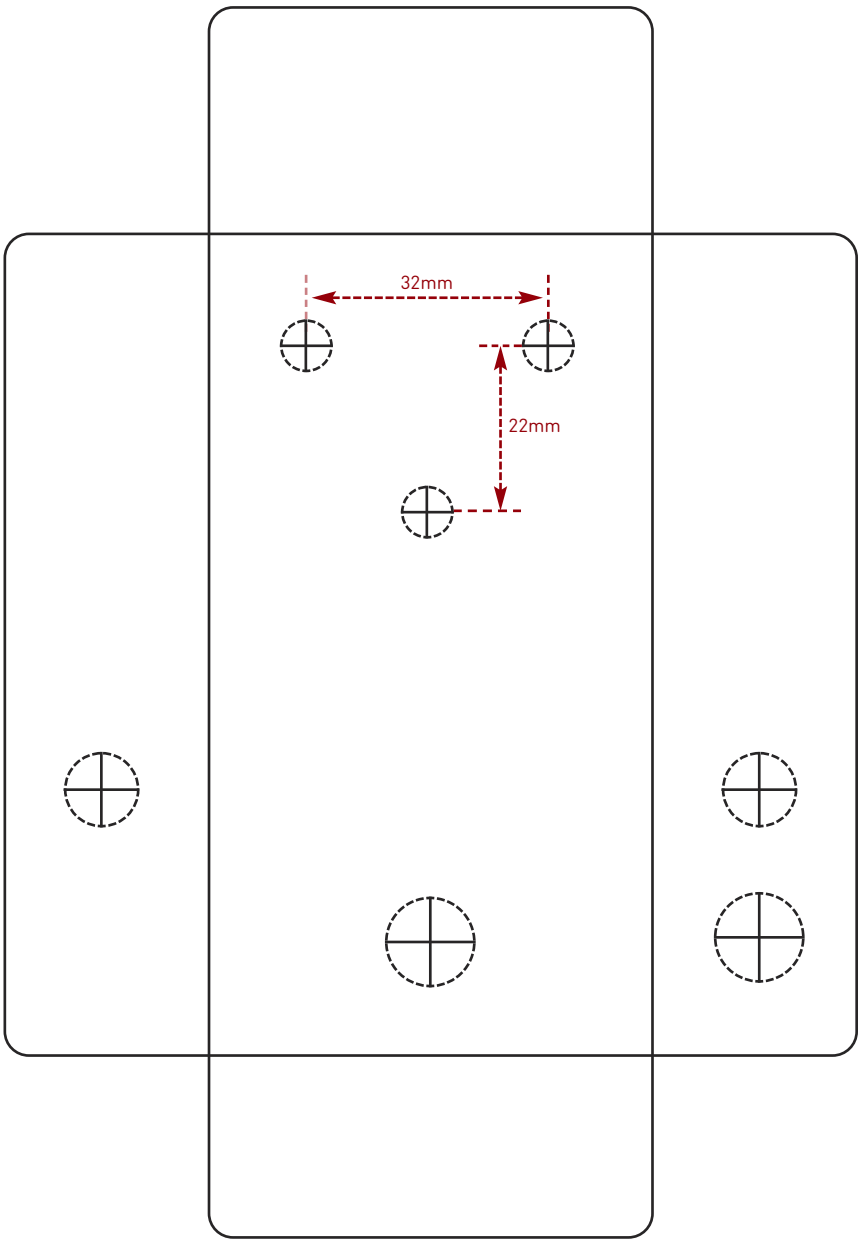
Drilling template without battery

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

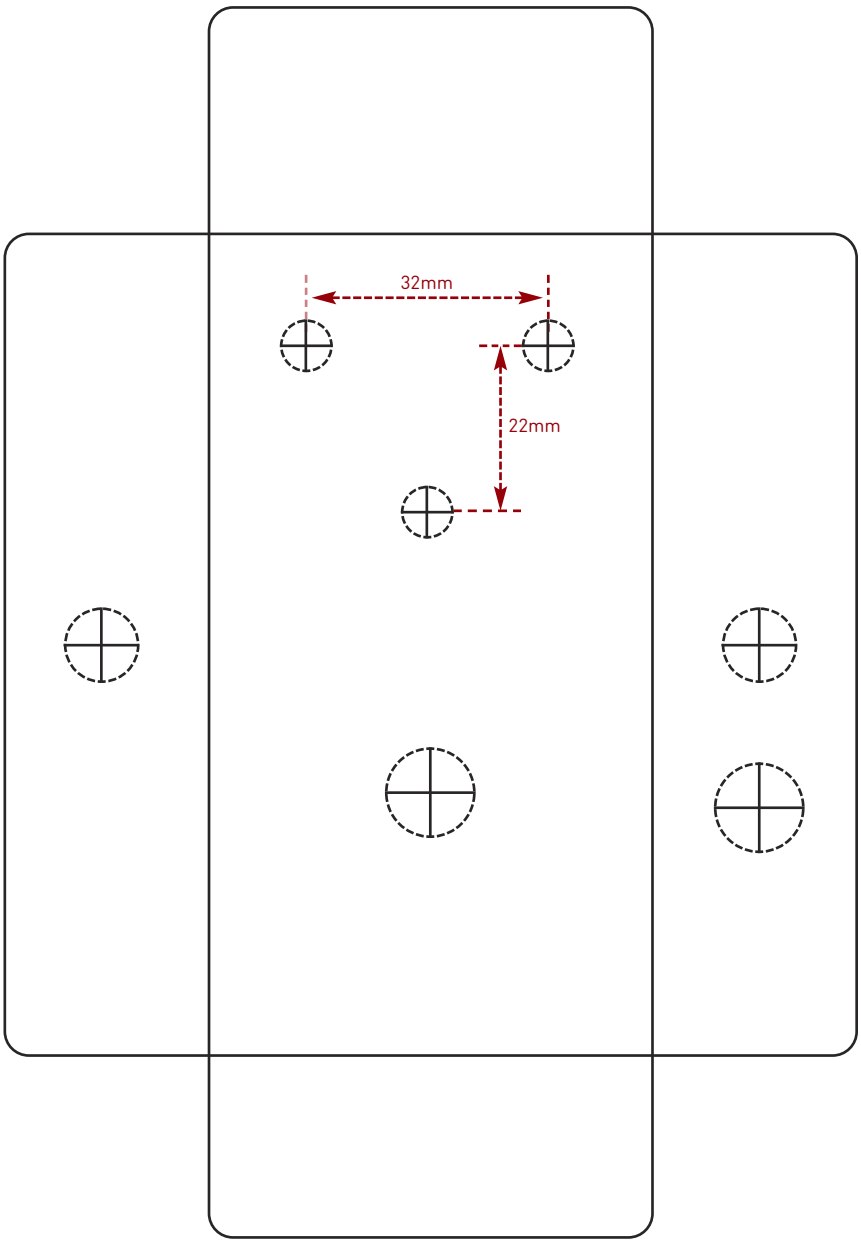
Drilling template with battery

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