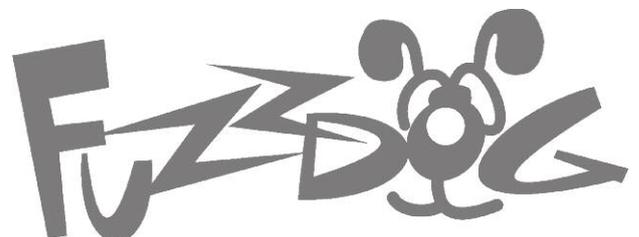




Sonic Tight'un

FET into power amp
distortion fun



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:

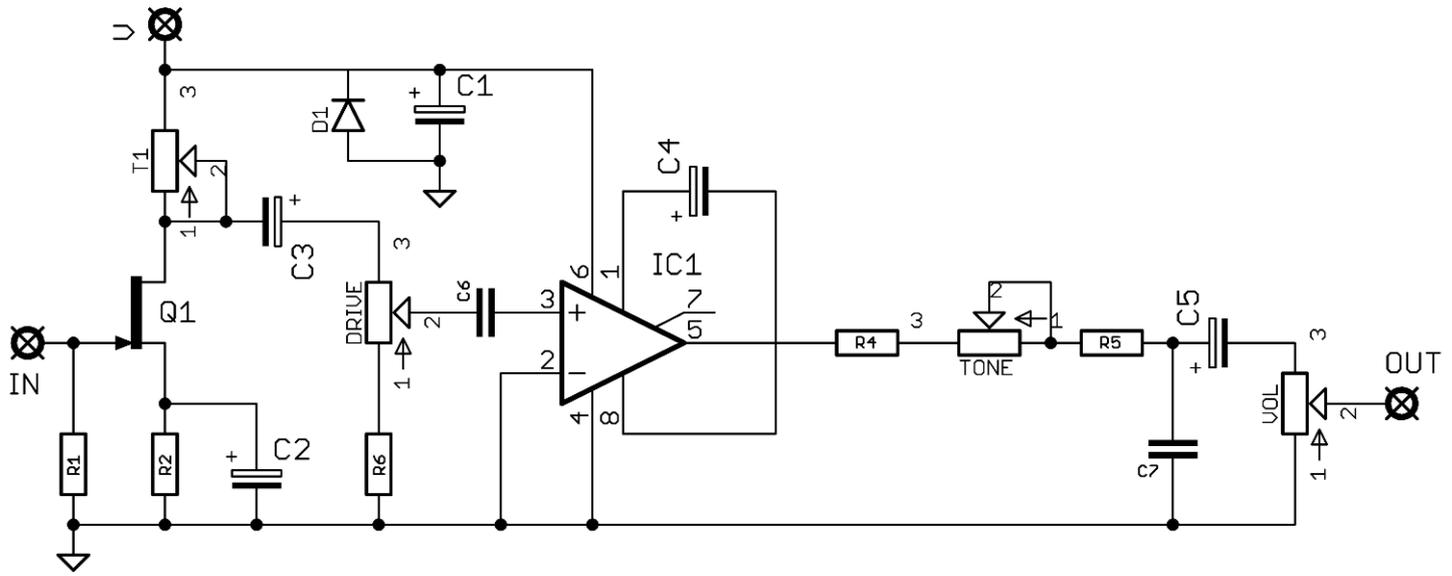
- Resistors should be 0.25W. You can use those with higher ratings but check the physical size of them.
- Electrolytics caps should be at least 25V for 9V circuits, 35V for 18V circuits. Again, check physical size if using higher ratings.

LAYOUT CONVENTIONS

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

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

Schematic + BOM



- R1 4M7
- R2 2K2
- R4 1K
- R5 1K
- R6 1K

- C1 100u elec
- C2 4u7 elec
- C3 4u7 elec
- C4 10u elec
- C5 10u elec
- C6 4n7
- C7 4n7

- IC1 LM386N-3*
- Q1 2N5457**

- D1 1N4001

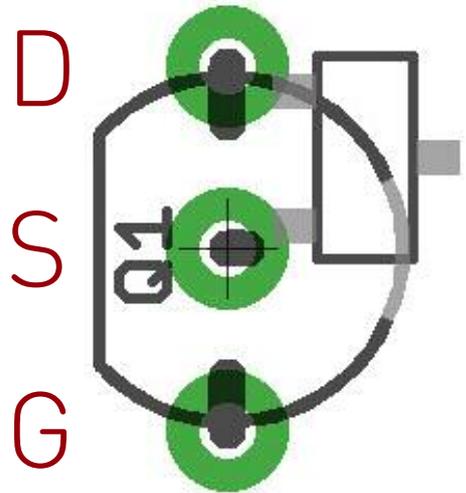
- DRIVE 10KA
- TONE 100KB
- VOL 100KA

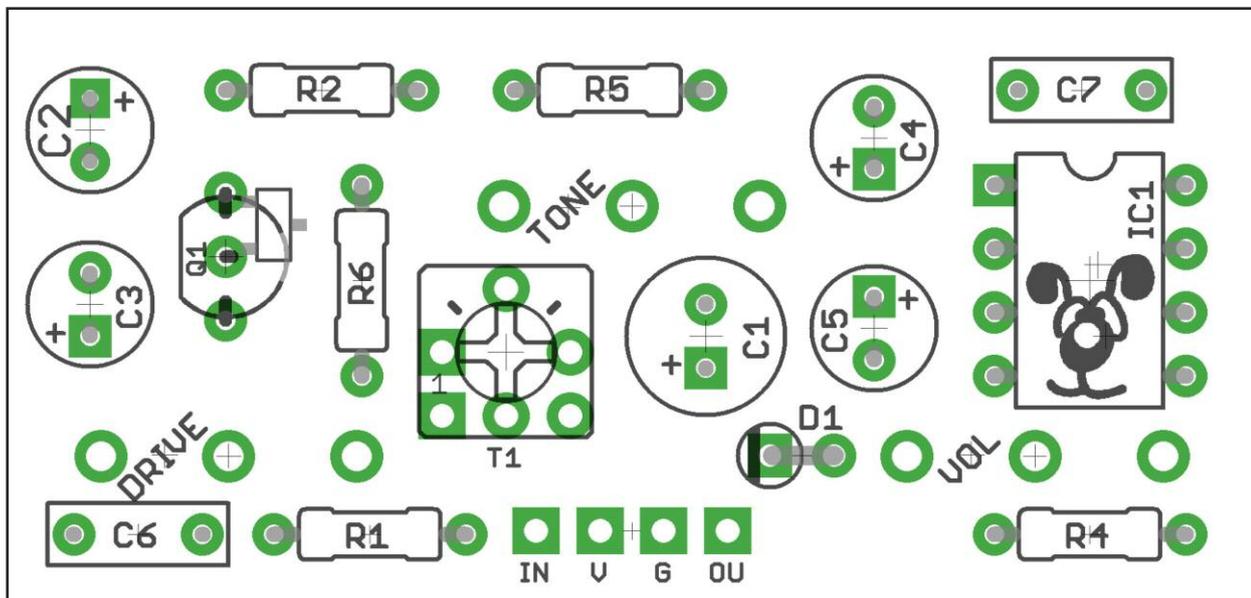
- T1 10K

*LM386N-1 can be used, but the N-3 variant works better. The sound is less compressed and note decay is less of an issue.

**The PCB has pads for both through-hole and SMT FETs. The original circuit uses 2N3819 which has a DGS pinout. We've designed the PCB to take the more readily available 2N5457 or J201 which have a DSG pinout.

If you want to use 2N3819 you'll have to twist the legs to fit the pads.





PCB layout ©2018 Pedal Parts Ltd.

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 handling and soldering the FET. They're very sensitive to heat and static.

It's best to use a socket for the IC. If you're not, be very careful when soldering.

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.

BIASING

See the next page for the test wiring instructions. Once you have everything set up, come back here.

You need to adjust the trimmer T1 to adjust the bias voltage at the drain of Q1.

To do this, set your multimeter to DC Voltage, with a range that includes 9V.

Put your common probe on any ground point on the board, or one of your jack ground lugs. It helps if this probe is a clip-type rather than just a point.

Check your supply voltage by putting the second probe on the V pad of the PCB.

Now place the second probe onto the drain pad of Q1 (see previous page). Adjust your trimmer until you get to around half your supply voltage.

Don't get hung up on exact numbers. Tweak until you like what you hear, but around half supply is the best starting point.

If you don't have a meter you can just do this by ear. It'll only sound good in a certain range of the turn.

Test the board!

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

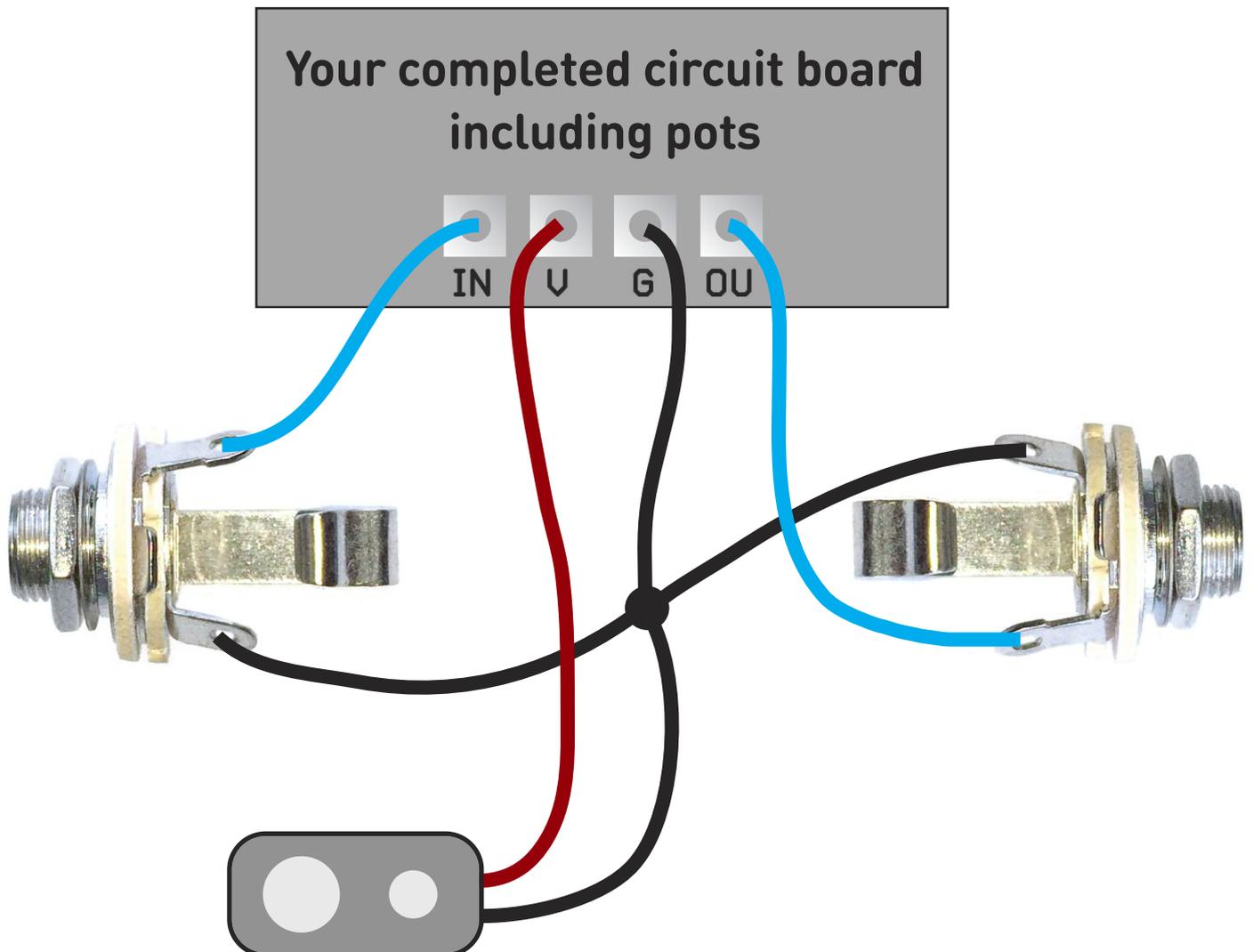
UNDER NO CIRCUMSTANCES will troubleshooting help be offered if you have skipped this stage. No exceptions.

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.

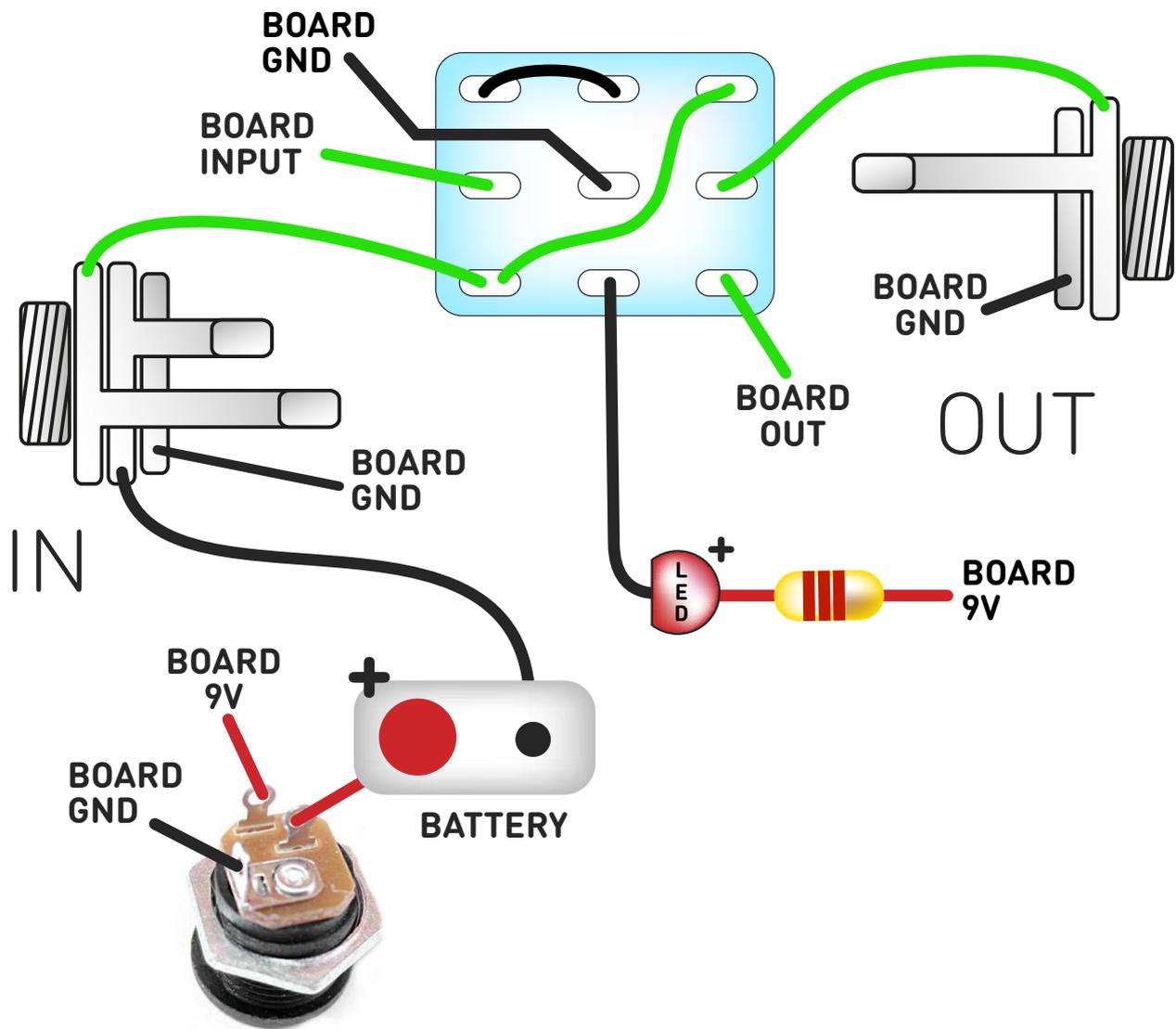
Solder some nice, long lengths of wire to the board connections for 9V, GND, IN and OUT. Connect IN and OUT to the jacks as shown. Connect all the GNDs together (twist them up and add a small amount of solder to tack it). Connect the battery + lead to the 9V wire, same method. Plug in. Go!

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.

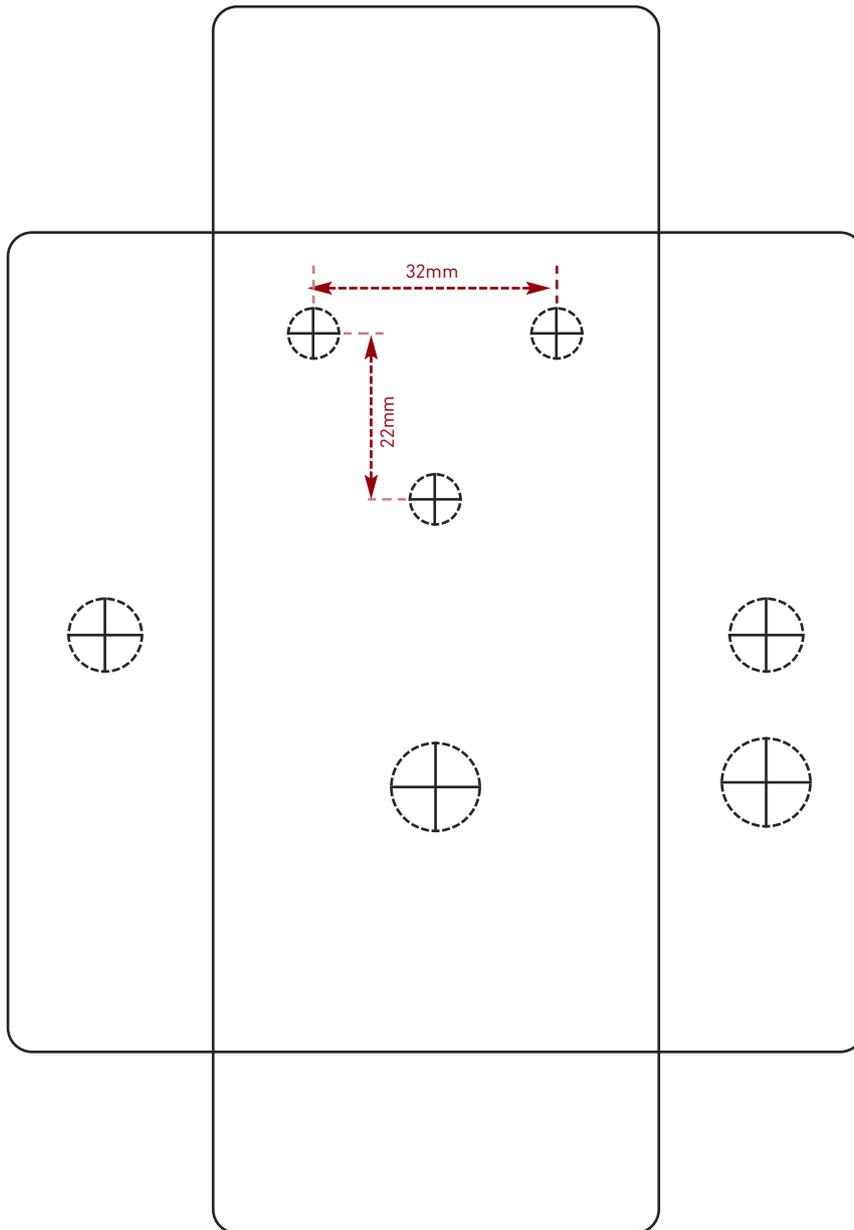
Drilling template

Hammond 1590B
60 x 111 x 31mm

Recommended drill sizes:

Pots	7mm
Jacks	10mm
Footswitch	12mm
DC Socket	12mm

It's a good idea to drill the potholes 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