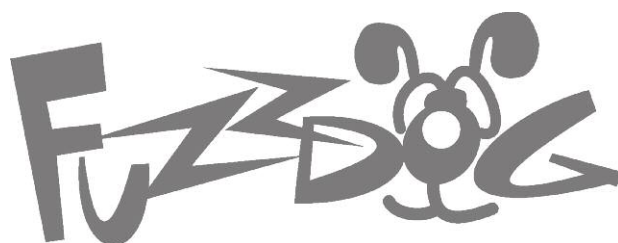


# PHASE 90

One knob of glorious  
phasey phun



# 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:

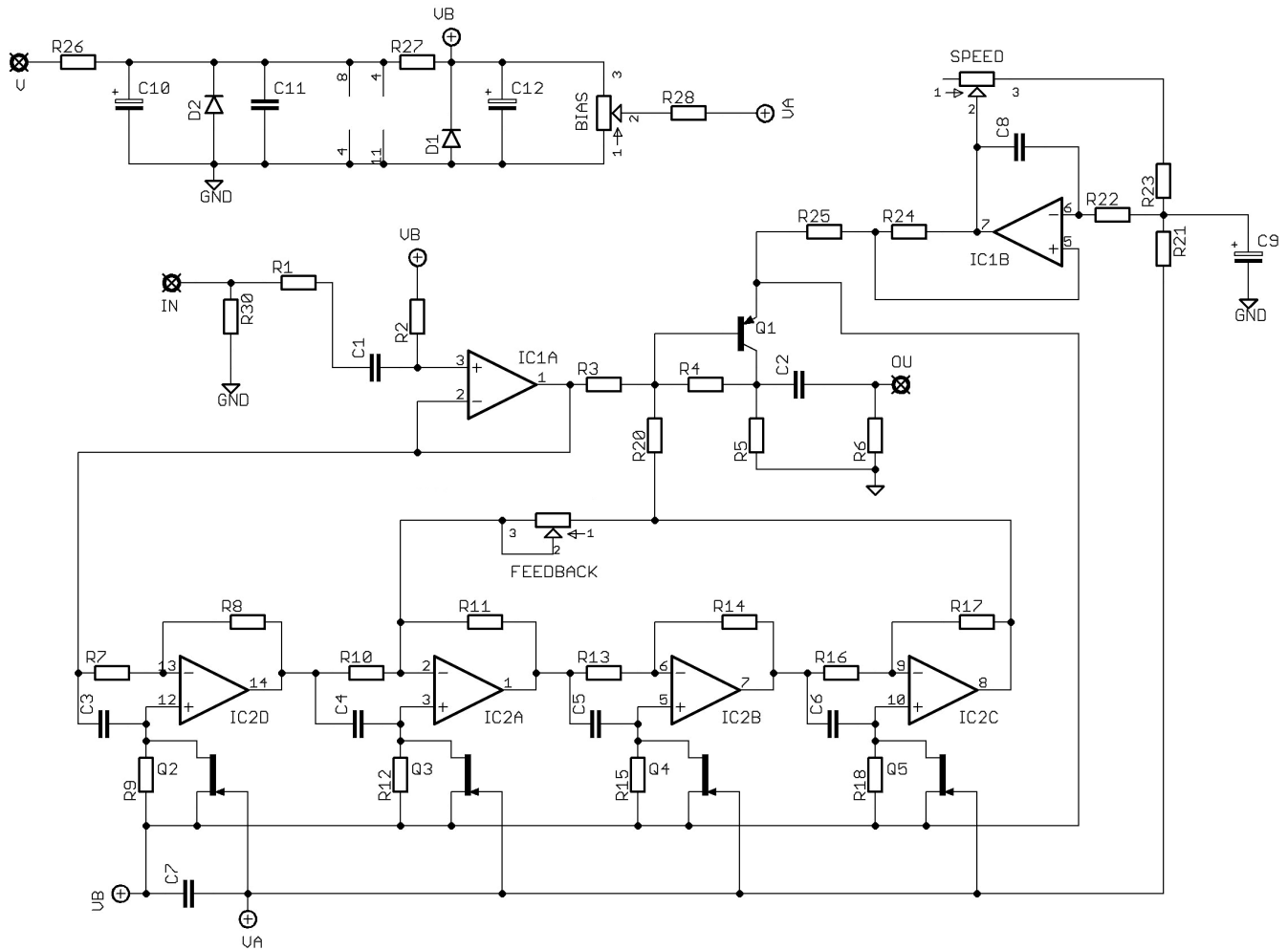
- Resistors should be 0.25W. You can use those with higher ratings but check the physical size of them.
- Electrolytics caps should be at least 25V for 9V circuits, 35V for 18V circuits. Again, check physical size if using higher ratings.

## **LAYOUT CONVENTIONS**

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

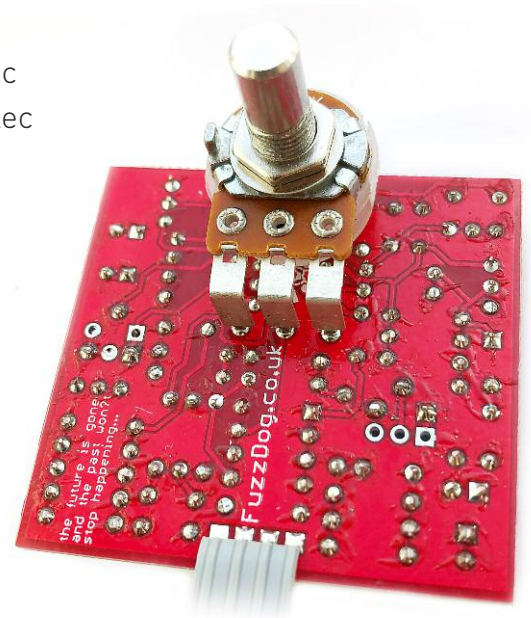
- **Electrolytic capacitors:**  
Long leg (anode) to square pad.
- **Diodes/LEDs:**  
Striped leg (cathode) to square pad. Short leg to square pad for LEDs.
- **ICs:**  
Square pad indicates pin 1.

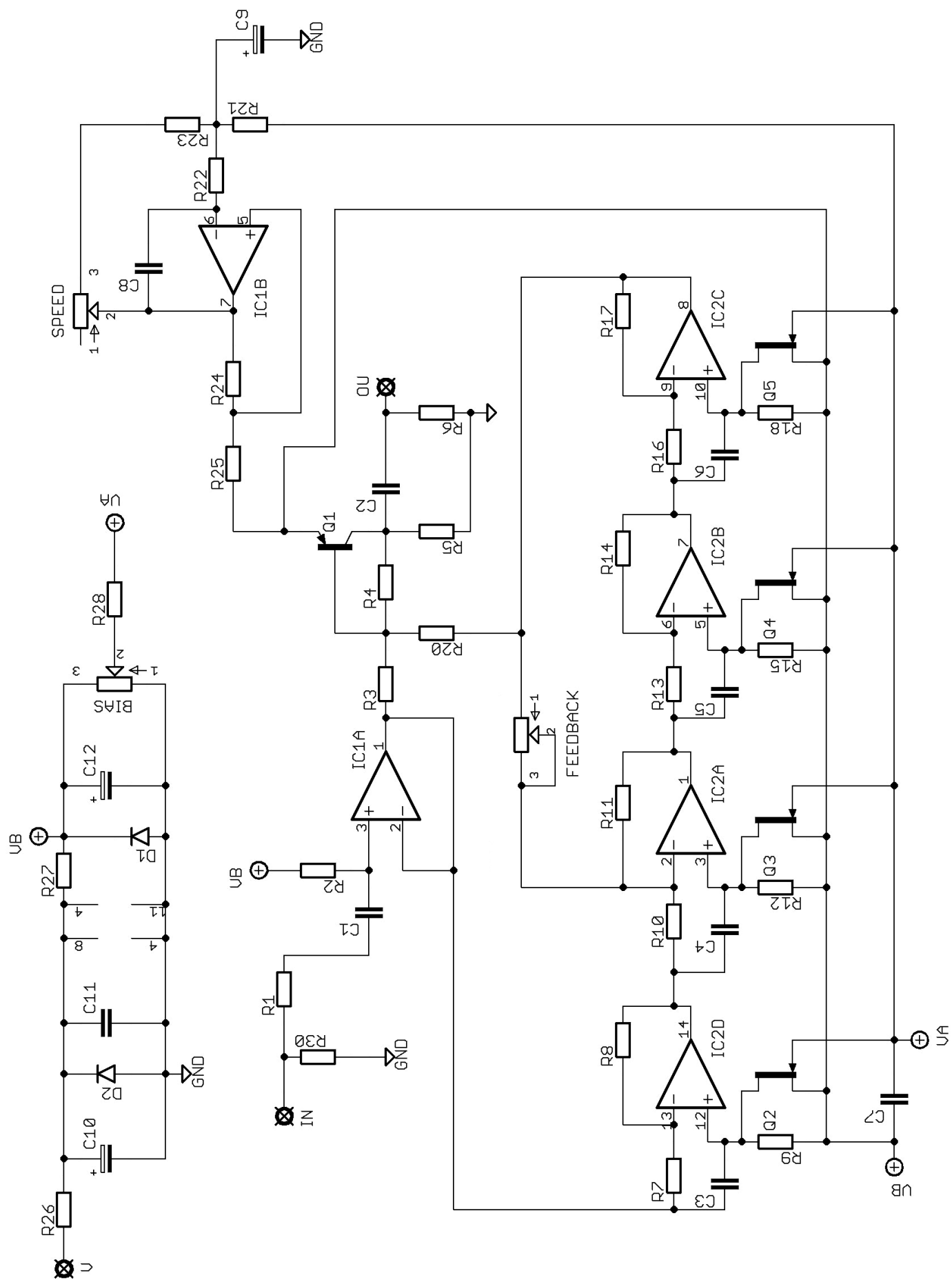
# Schematic + BOM



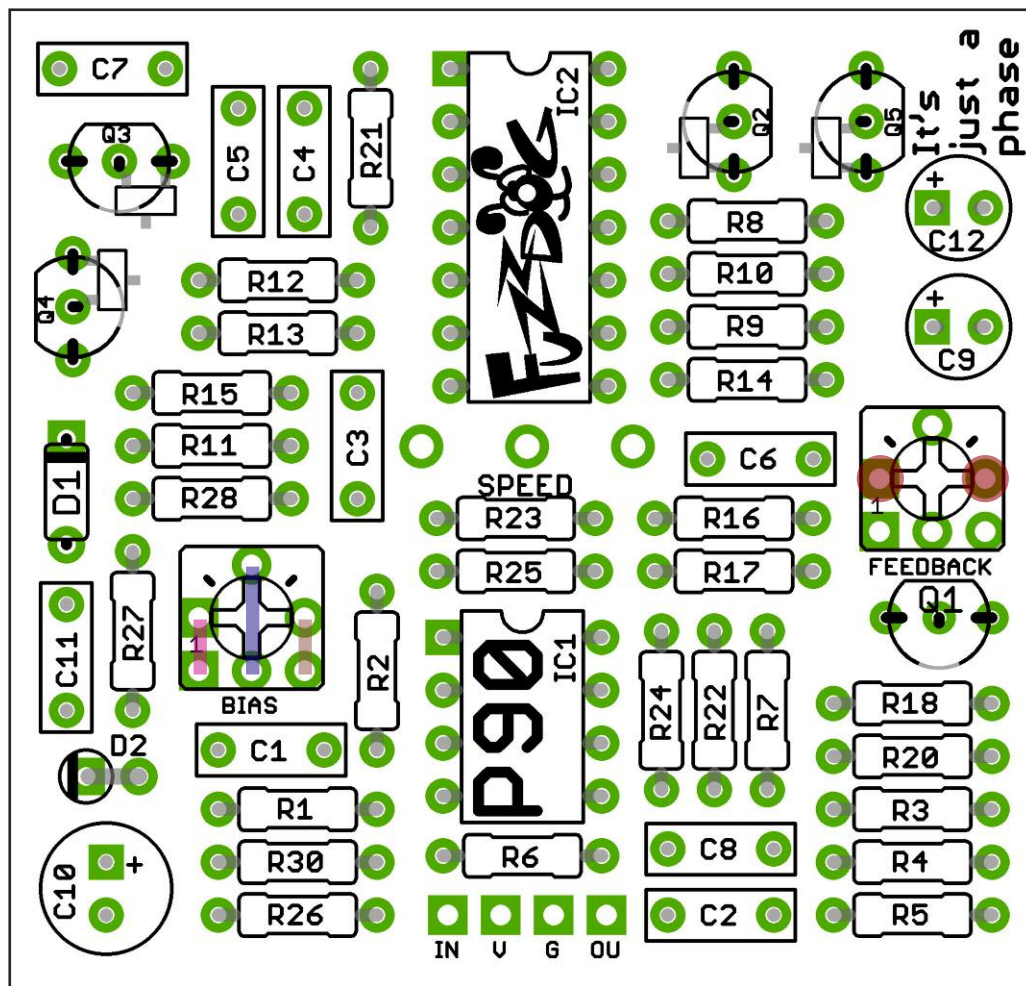
R1	10K	R22	150K	C1	10n	SPEED	500KC
R2	470K	R23	4K7	C2	47n		
R3	150K	R24	470K	C3	47n	FBACK	47K trim**
R4	150K	R25	150K	C4	47n	BIAS	250K trim
R5	56K	R26	47R	C5	47n		
R6	150K	R27	10K	C6	47n		
R7	10K	R28	1M	C7	47n		
R8	10K	R30	1M	C8	10n		
R9	22K			C9	15u elec		
R10	10K	D1	5.1v zener	C10	100u elec		
R11	10K	D2	1N4001	C11	100n		
R12	22K	D3	LED	C12	22u		
R13	10K	Q1	2N5087				
R14	10K	Q2-5	Matched FETs*				
R15	22K						
R16	10K	IC1	TL072				
R17	10K	IC2	TL074				
R18	22K						
R20	150K						
R21	3M9						

\* / \*\* see page 5









PCB layout ©2020 Pedal Parts Ltd.

22K resistor across these pads if going for that option. See below.

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). Same goes for the ICs if you aren't using sockets.

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 pot. Once it's in place you'll have no access to some of the board.

## \*FETs

You need to use a matched set of FETs for this build. The board is designed for 2N5457. If you're using different FETs check your pinout.

## BIASING

Use the bias trimmer to adjust the voltage at the gates of the FETs. You can do this by ear, simply tweaking until you get the best phasing effect. If you want to do it by numbers it should be around 3V.

## \*\*FEEDBACK

We've added space for a trimmer for the feedback setting. The standard parts for this are:

Script logo version - empty - no feedback

Block logo version - 22K resistor

So it's up to you. Leave it out, put the 22K across the trimmer pads shown in red above, or use a trimmer and adjust to taste. If the latter, use a 47-50K trimmer, set it to the middle position and tweak it either way until you're happy.

There are extra pads provided for the trimmers so you can use several different types. Just drop the pins into the holes as they naturally fall. Lines shown above show which pads are internally connected on the PCB.

# Test the board!

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

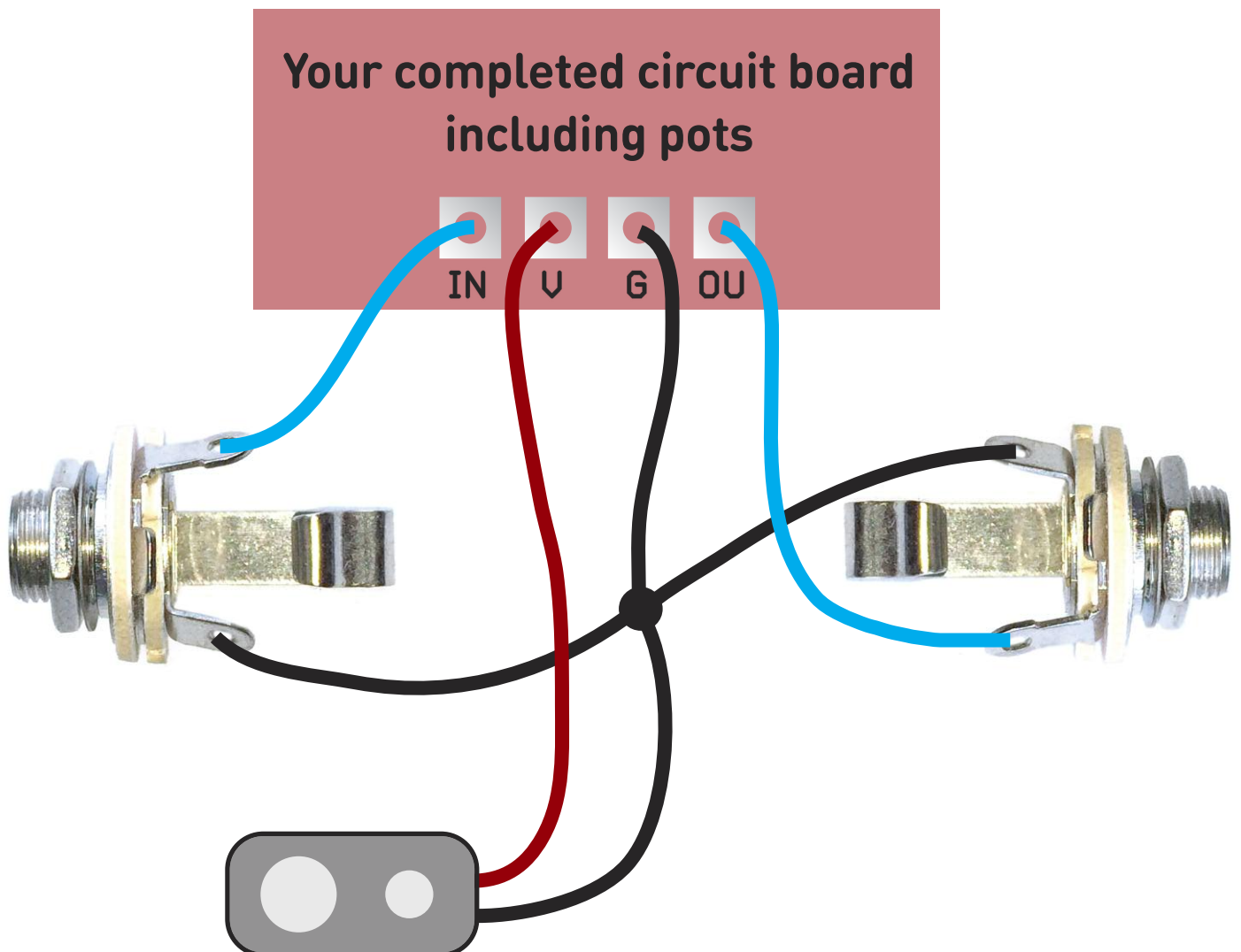
**UNDER NO CIRCUMSTANCES will troubleshooting help be offered if you have skipped this stage. No exceptions.**

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.

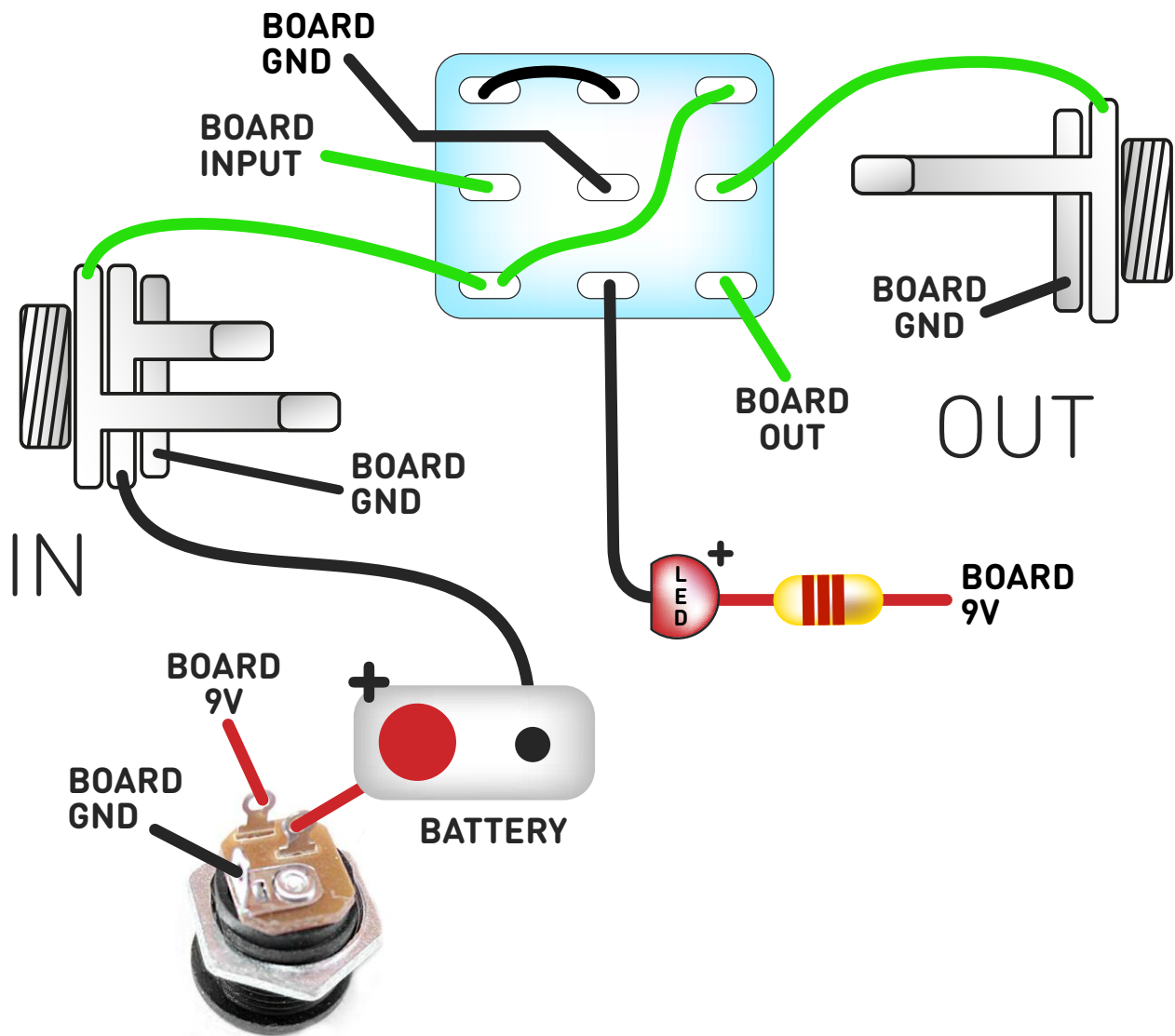
Solder some nice, long lengths of wire to the board connections for 9V, GND, IN and OUT. Connect IN and OUT to the jacks as shown. Connect all the GNDs together (twist them up and add a small amount of solder to tack it). Connect the battery + lead to the 9V wire, same method. Plug in. Go!

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

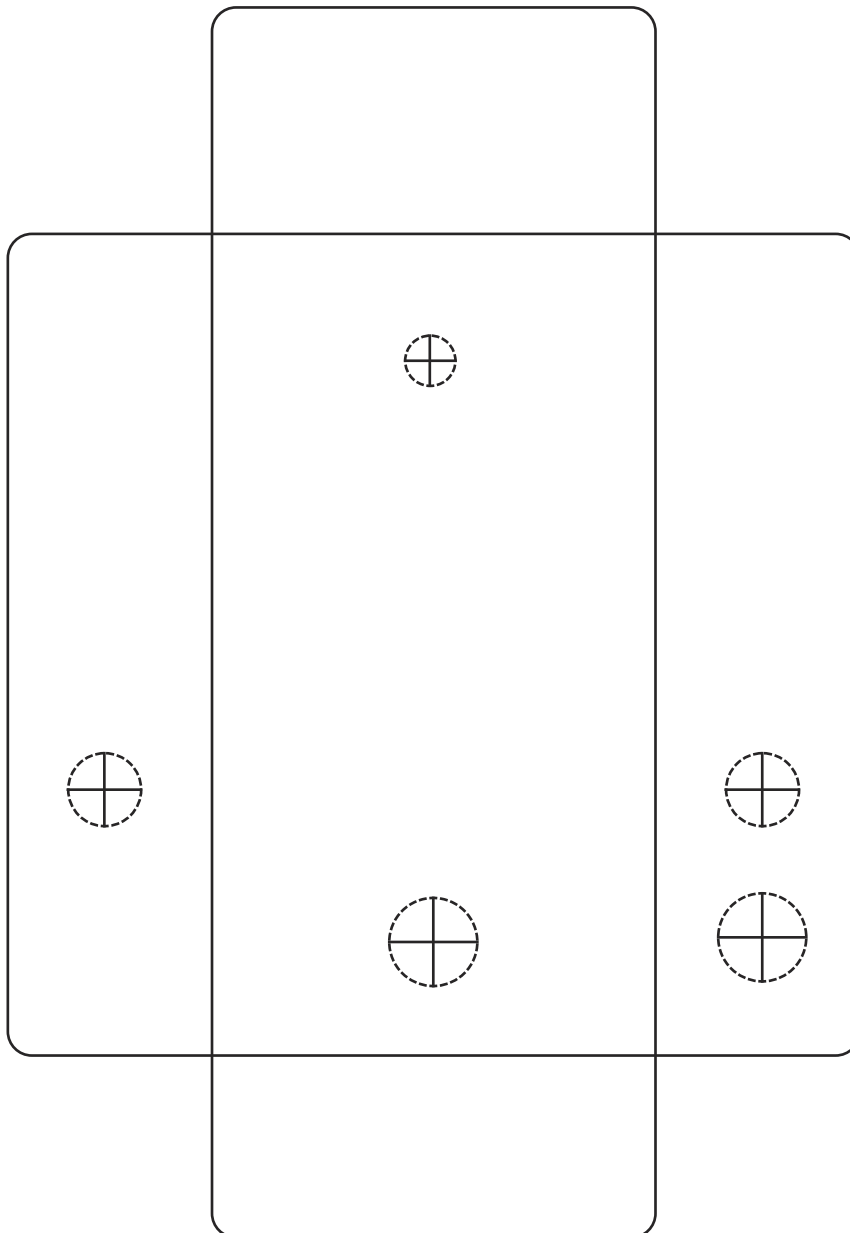
60 x 111 x 31mm

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