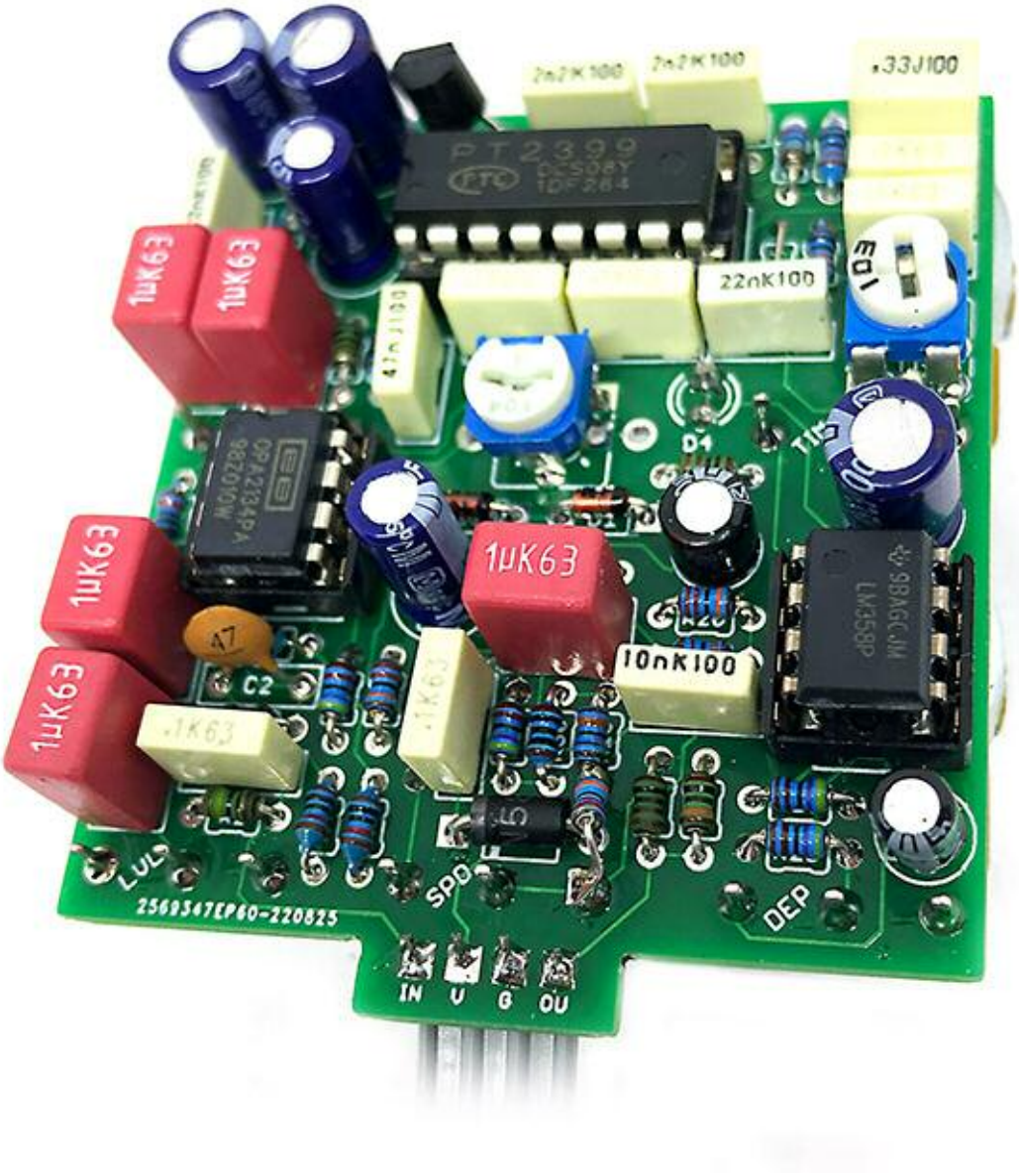



# Alluring Echo

## Splendid modulated tape-style delay

# 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:

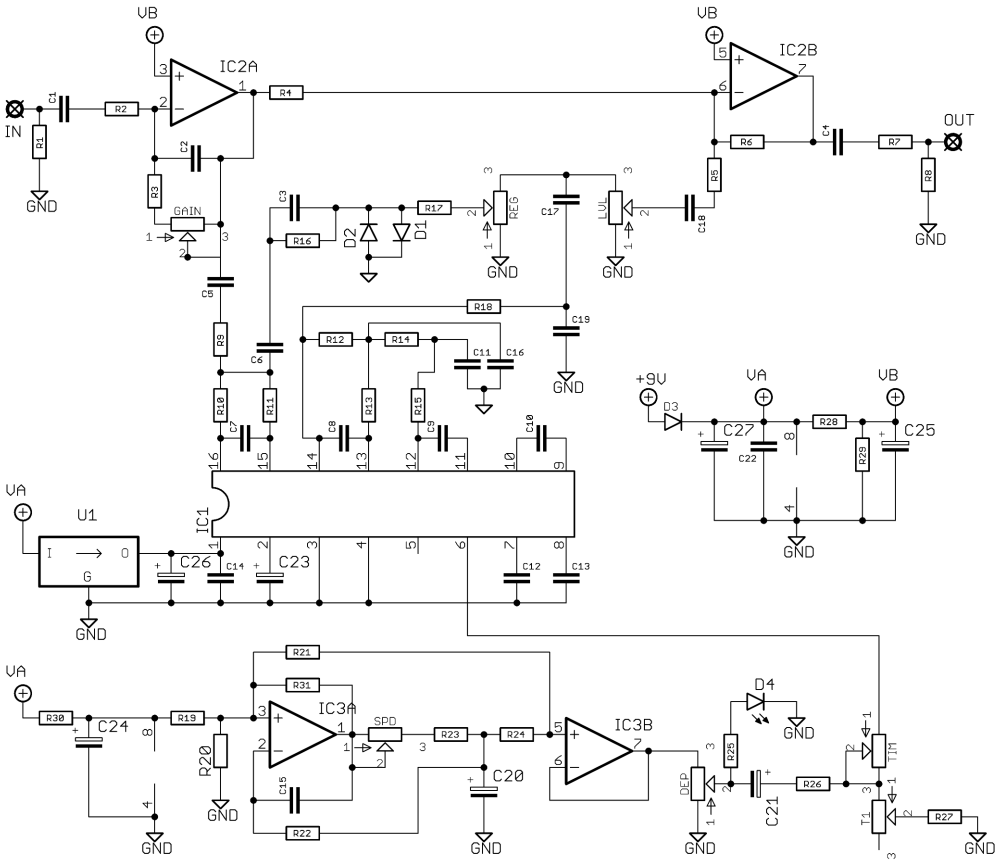
- Resistors should be 0.125W. You can use those with higher ratings but check the physical size of them. 0.4W can also be found in 3mm length.
- Electrolytics caps should be at least 25V for 9V circuits, 35V for 18V circuits. Again, check physical size if using higher ratings.

## 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 |
| R2 | 100K |
| R3 | 47K |
| R4 | 10K |
| R5 | 10K |
| R6 | 12K |
| R7 | 1K |
| R8 | 100K |
| R9 | 10K |
| R10 | 10K |
| R11 | 10K |
| R12 | 15K |
| R13 | 10K |
| R14 | Jumper* |
| R15 | 10K |
| R16 | 10K |
| R17 | 1K |
| R18 | 10K |
| R19 | 120K |
| R20 | 220K |
| R21 | 120K |
| R22 | 220K |
| R23 | 4K7 |
| R24 | 100K |
| R25 | 4K7 |
| R26 | 4K7 |
| R27 | 1K |
| R28 | 47K |
| R29 | 47K |
| R30 | 100R |
| R31 | 220K** |

| | |
|---|---|
| C1 | 100n |
| C2 | 47p |
| C3 | 47n |
| C4 | 1u |
| C5 | 1u |
| C6 | 1u |
| C7 | 2n2 |
| C8 | 2n2 |
| C9 | 100n |
| C10 | 330n |
| C11 | 22n |
| C12 | 100n |
| C13 | 100n |
| C14 | 100n |

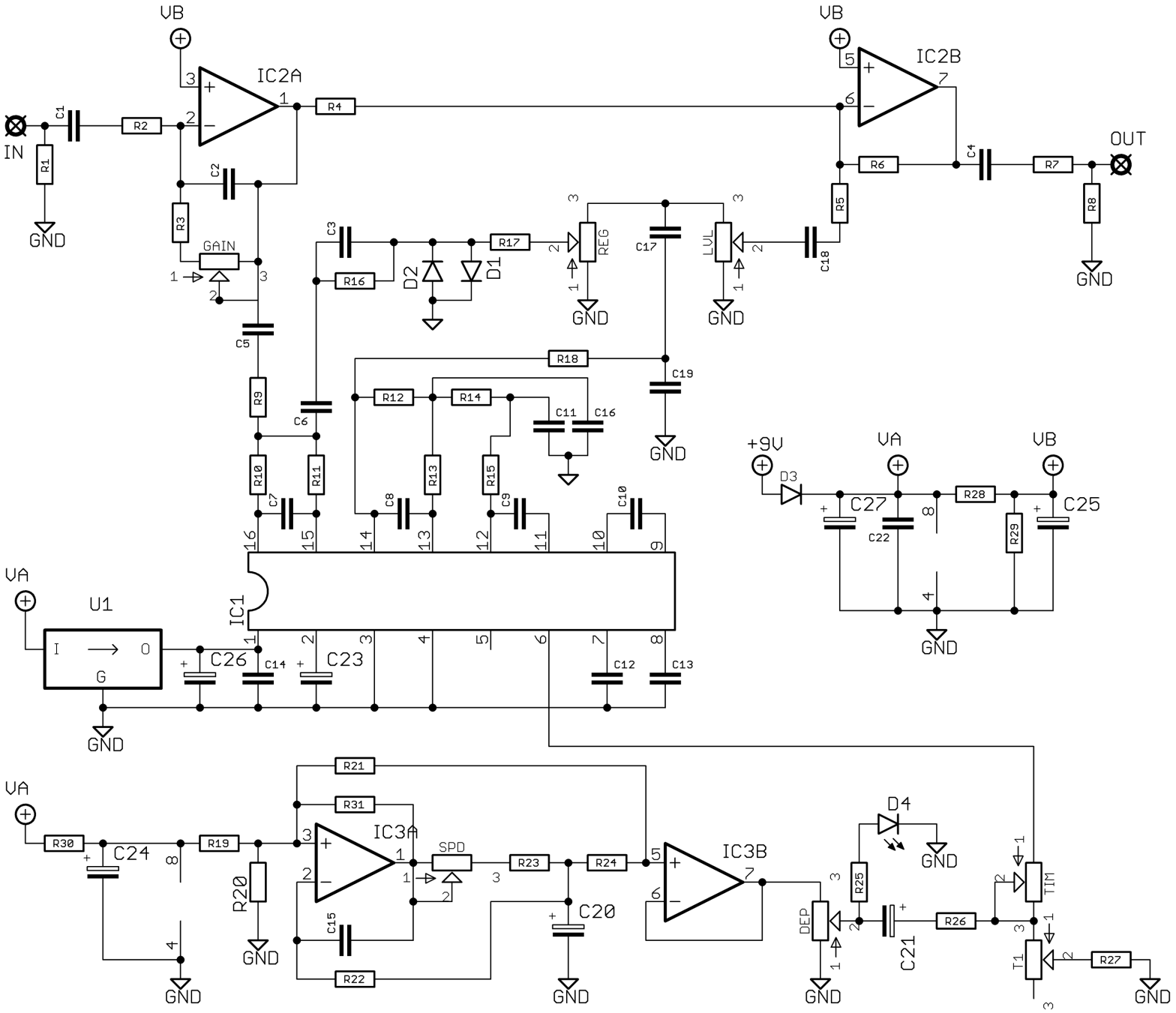
| | |
|---|---|
| C15 | 10n |
| C16 | Empty* |
| C17 | 1u |
| C18 | 1u |
| C19 | 22n |
| C20 | 10u elec |
| C21 | 4u7 elec |
| C22 | 100n |
| C23 | 47u elec |
| C24 | 100u elec |
| C25 | 47u elec |
| C26 | 100u elec |
| C27 | 100u elec |

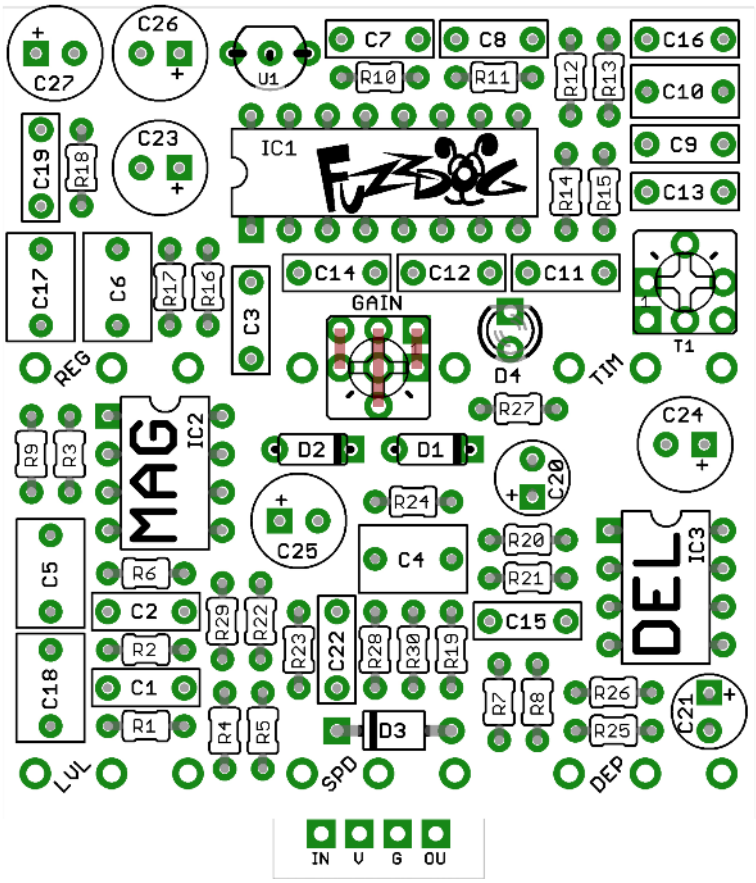
| | |
|---|---|
| IC1 | PT2399 |
| IC2 | OPA2134 |
| IC3 | LM358 |
| U1 | 78L05 |
| D1-2 | 1N4148 |
| D3 | 1N5817 |
| D4 | LED |

| | |
|---|---|
| DEP | 100KB |
| GAIN | 100KB/TRIM |
| LVL | 100KB |
| REG | 100KA |
| SPD | 50KB |
| TIM | 50KB* |
| T1 | 10K TRIM |

*Extended delay time mod - R14 10K, C16 10n, TIM 100KB. Repeats will have more noise.
**Missing from the board and needs to be hacked in. See later in doc.

You can use an external pot for gain if you like, but it's best to set and forget with a trimmer.


VB
IC2A
IC2B
IN
OUT
C1
R1
R2
R3
R4
R5
R6
R7
R8
C2
C3
C4
C5
C6
C7
C8
C9
C10
C11
C12
C13
C14
C15
C16
C17
C18
C19
C20
C21
C22
C23
C24
C25
C26
C27
R9
R10
R11
R12
R13
R14
R15
R16
R17
R18
R19
R20
R21
R22
R23
R24
R25
R26
R27
R28
R29
R30
R31
GAIN
REG
LVL
SPD
TIM
DEP
T1
D1
D2
D3
D4
GND
VA
+9V
U1
IC1
IC3A
IC3B
I
O
G




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, U1 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). Same goes for the ICs if you aren't using sockets.

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.

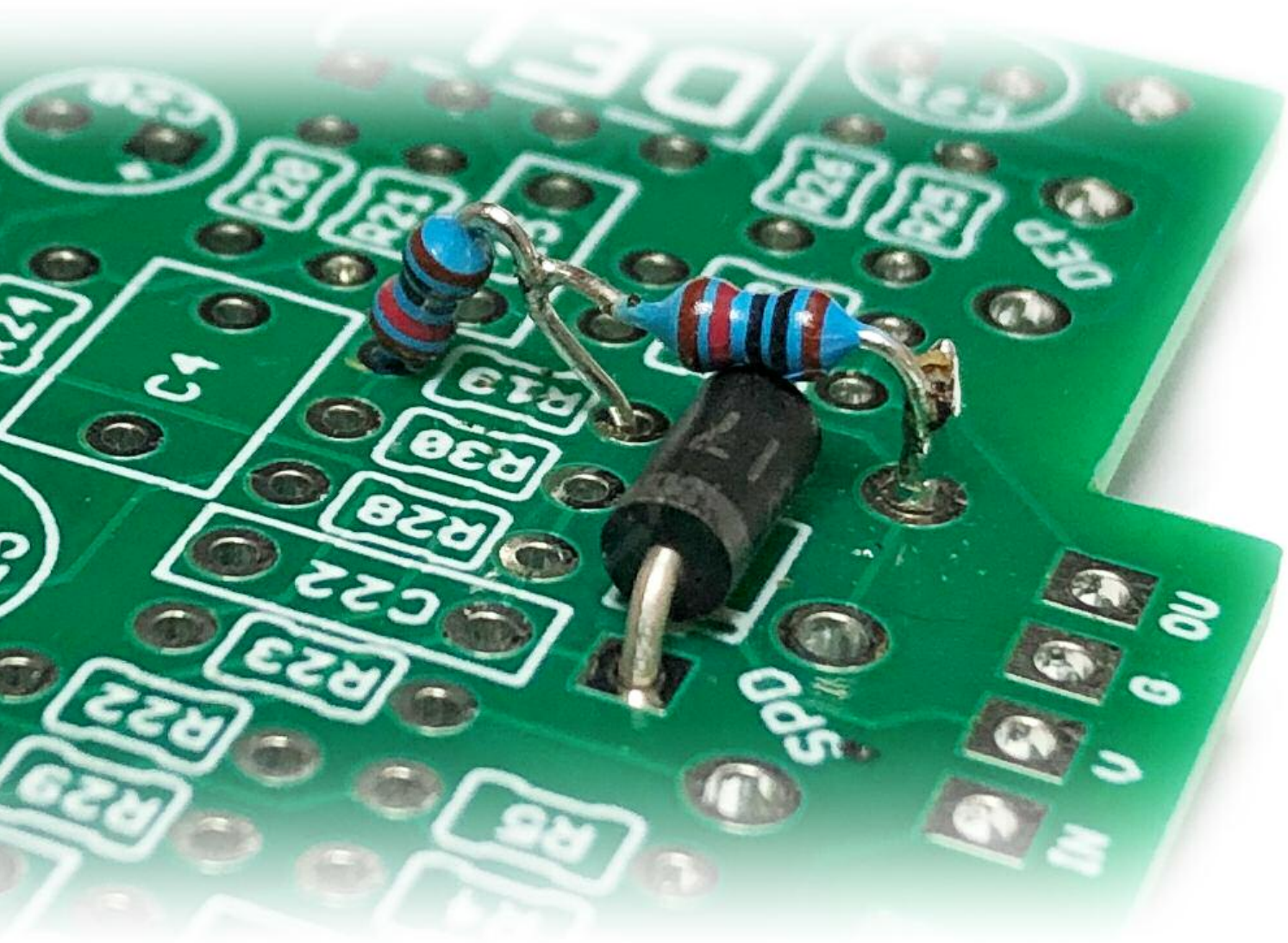
There are extra pads for the Gain trimmer so you can use different types. As long as there's a leg in each column marked above you're OK.

D4 will get brighter when the modulation depth is higher, and will flash to indicate the modulation speed.

# R31 Hack

Oops. It'll be fixed on V1.1 of the PCB, but for now this needs to be hacked in. It's simple.

Leave some of the leg of R19 above the board when mounting. When everything else is assembled simple place R31 from the exposed leg of R19 to pin 1 of the SPD pot, above D3.




# Adjusting the trimmers

GAIN - turn all the controls down to zero. Adjust the trimmer until your engaged signal is the same level as your bypassed signal, or louder if you want some boost.

T1 adjusts the overall modulation depth. Tweak to taste. There's no right or wrong.

# Test the board!

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

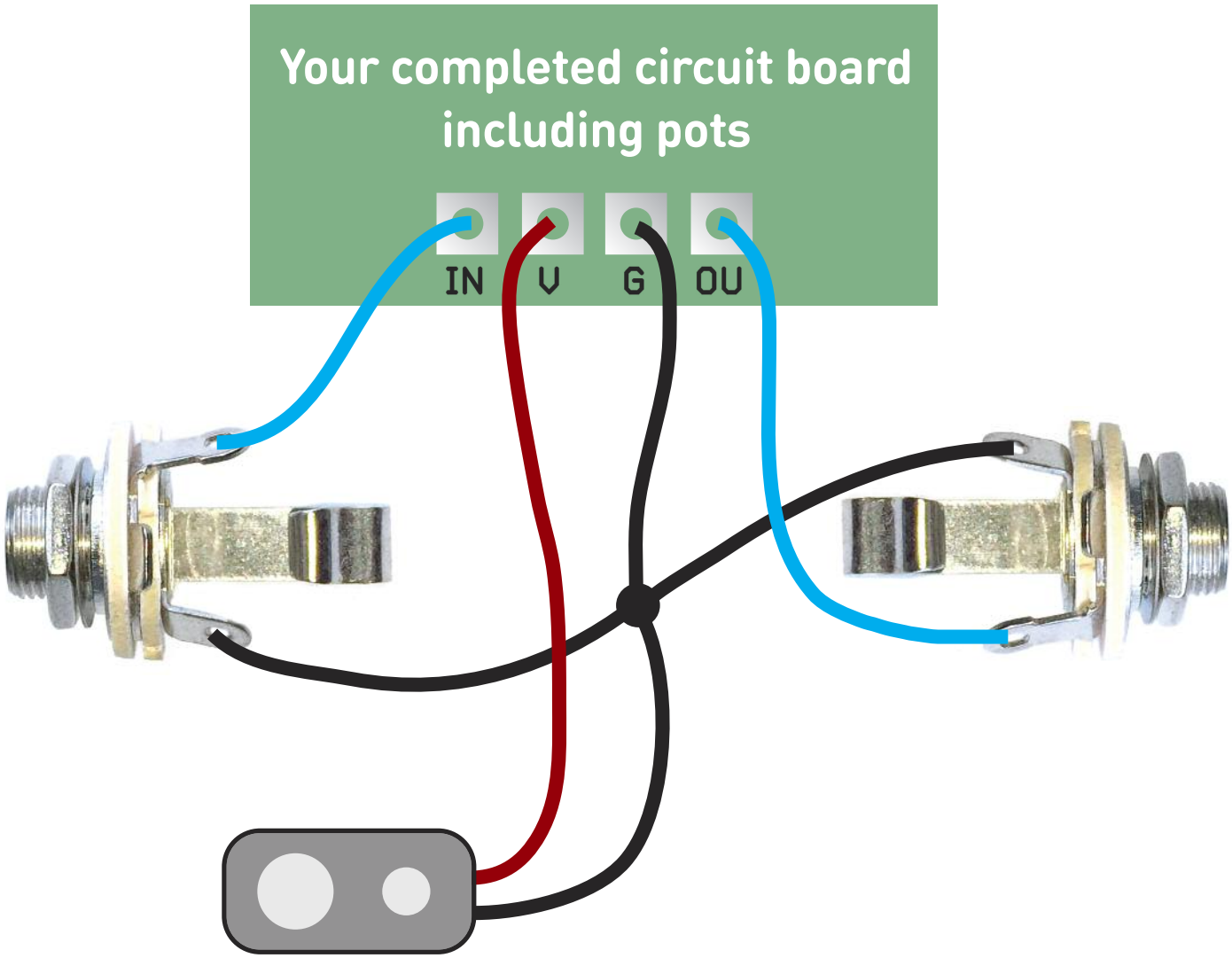
## UNDER NO CIRCUMSTANCES will troubleshooting help be offered if you have skipped this stage. No exceptions.

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.

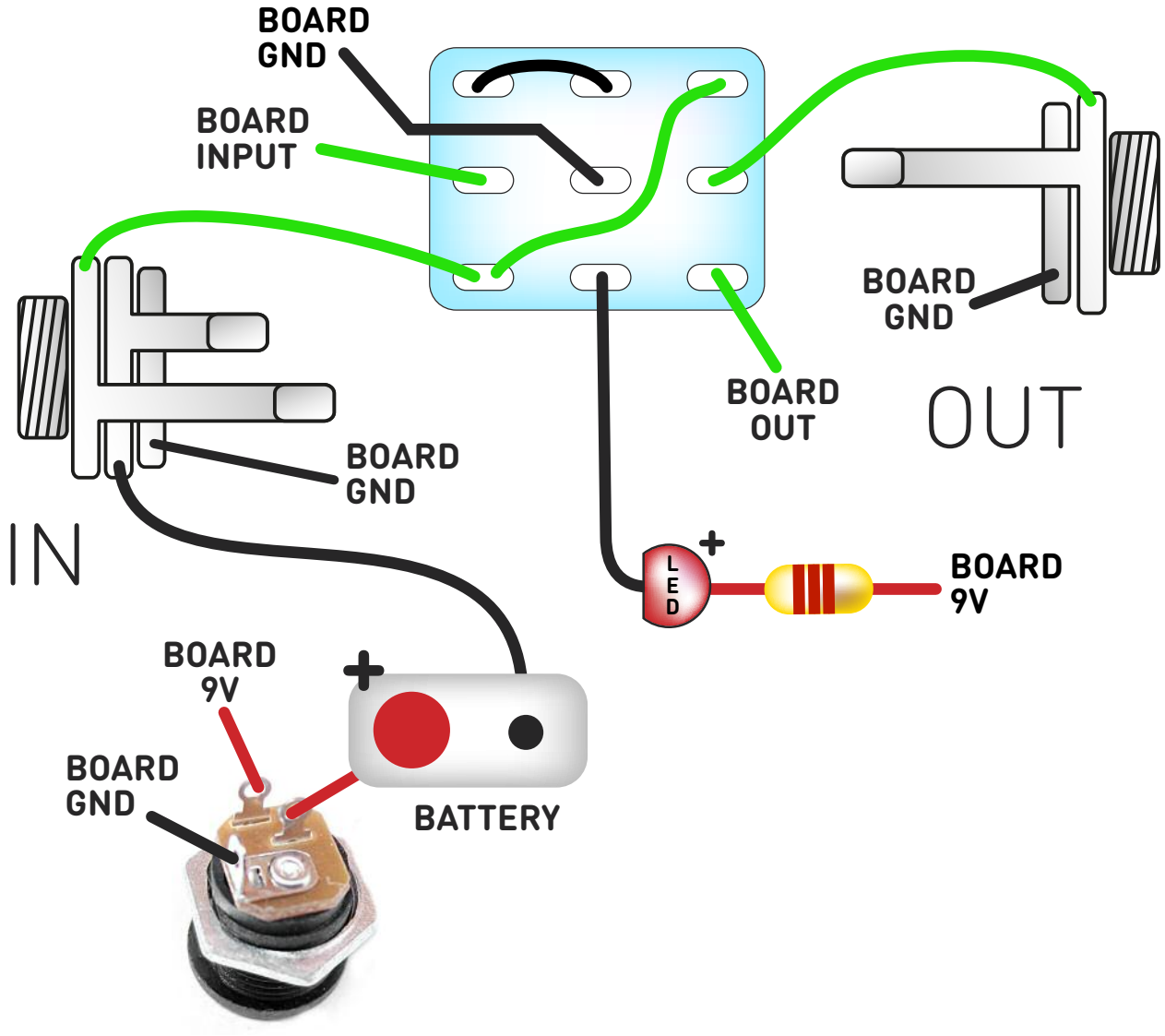
Solder some nice, long lengths of wire to the board connections for 9V, GND, IN and OUT. Connect IN and OUT to the jacks as shown. Connect all the GNDs together (twist them up and add a small amount of solder to tack it). Connect the battery + lead to the 9V wire, same method. Plug in. Go!

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

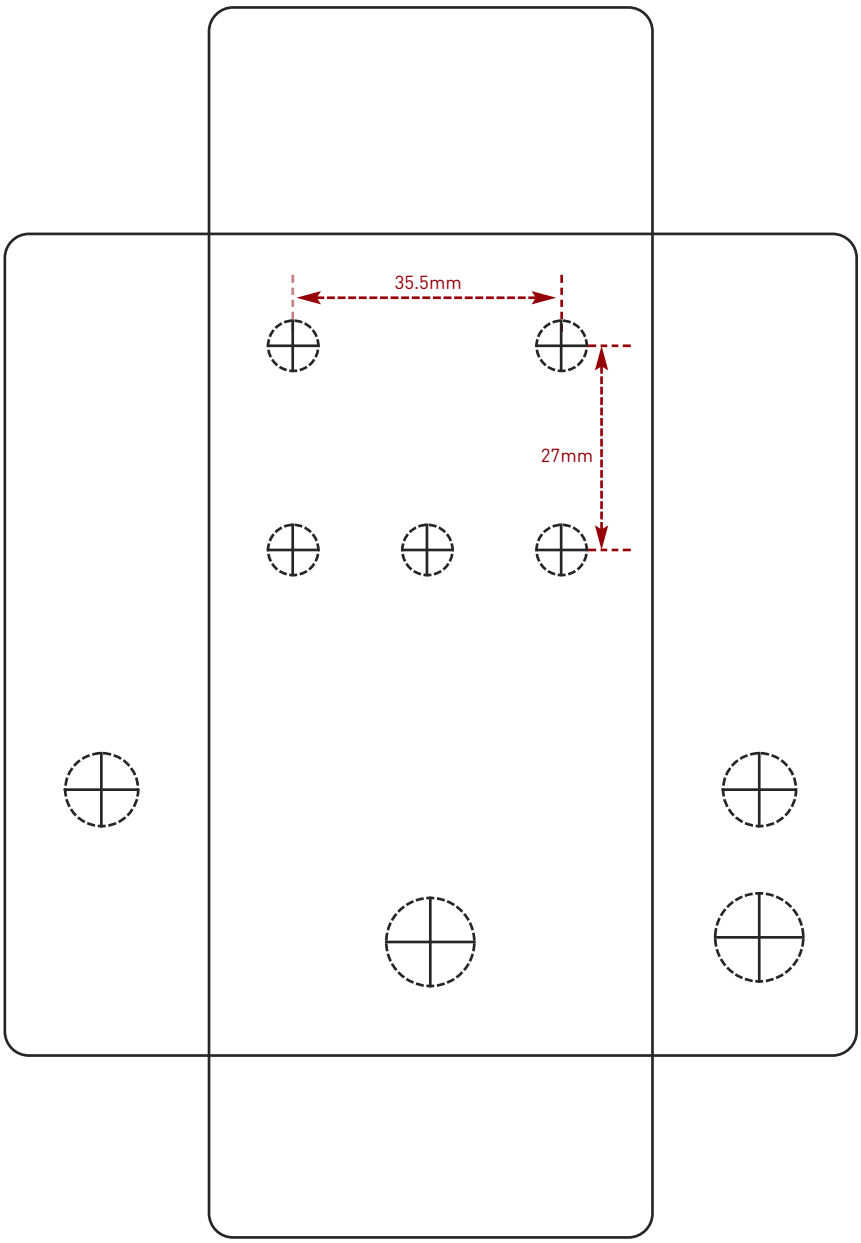
Hammond 1590B

60 x 111 x 31mm

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