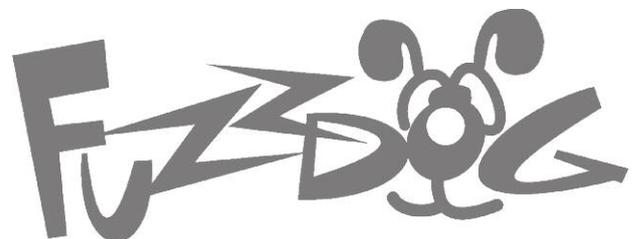


Emperor Dual OD

Blues Breaker-based
boutique drive action



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.

Yet more important notes specific to this circuit

CHANNEL ORDER

Our version of this circuit runs left channel (A) into right channel (B). Why? Because that's how it should be. If you don't like it you're welcome to wire up the daughterboard differently, wiring IA and OA to IB and OB, and vice versa. You should also flip the daughterboard and put the switches on the other side. You'll have to use long wires for the jack connections, but that's the price of putting it together differently to how it was designed.

HIGH GAIN VERSION

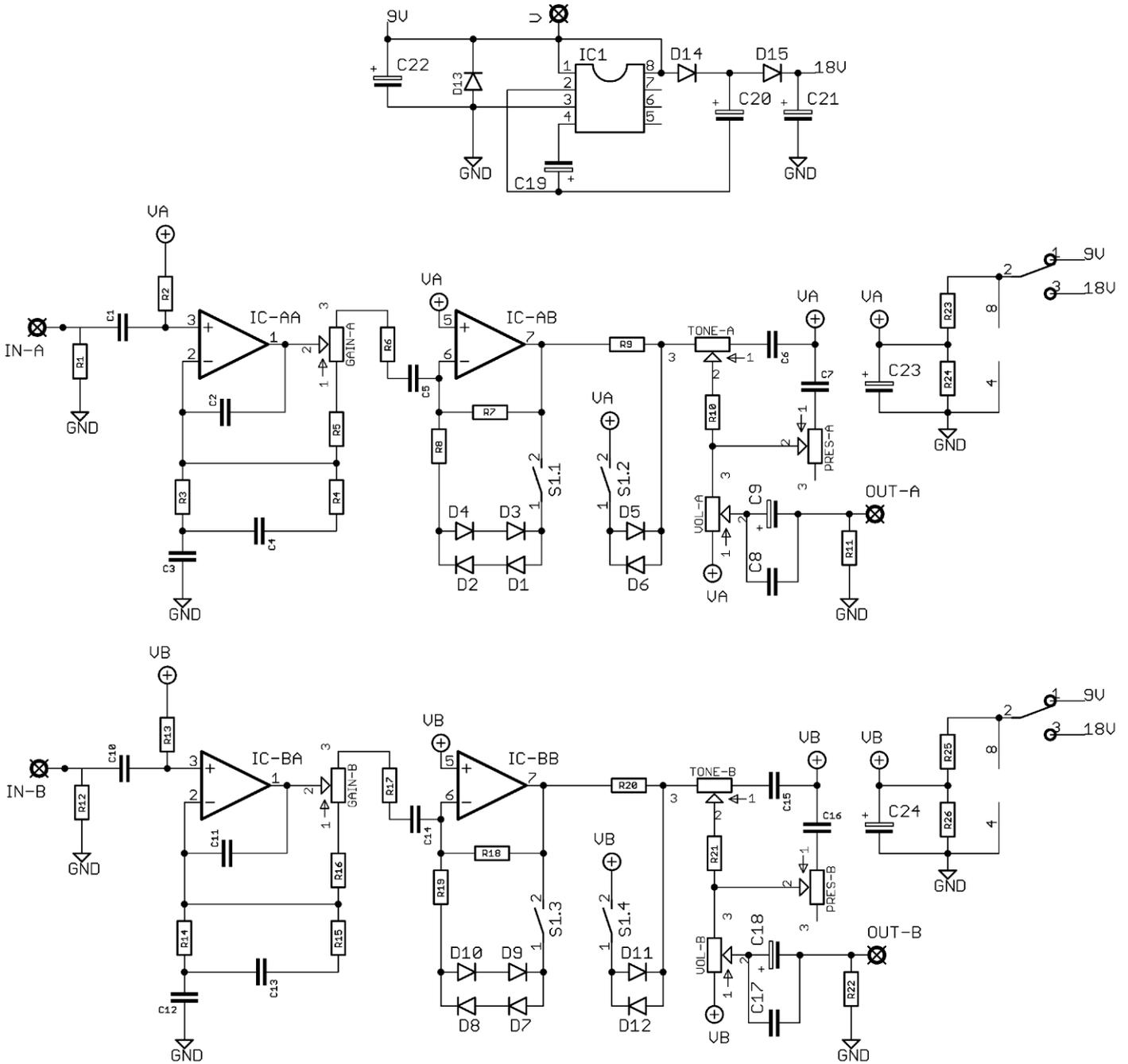
We can only go off the info that is out there in space. As far as we know, no-one has identified the IC used in the new high-gain channel. Other components have been identified and are (using Channel A BOM as ref):

R5 100K
C1 22n*
D5-6 1N1418 or 1N914

We cannot confirm these changes, so it's up to you whether you want to implement them. Without the specific IC used it's unlikely to sound the same as the original.

*The input cap is supposedly 22n on the input of both channels in the high-gain version.

Schematic - Main PCB



The two channels are identical. The small circuit shown around IC1 is the charge pump.

The connections for the 4-way DIL clipping selector switch are shown above (S1.1 - S1.4). Switches 1 and 2 will connect the two sets of diodes for circuit A, 3 and 4 for circuit B. You can engage one, both or neither set of diodes for each circuit. You'll notice big changes in volume with the switches in different positions.

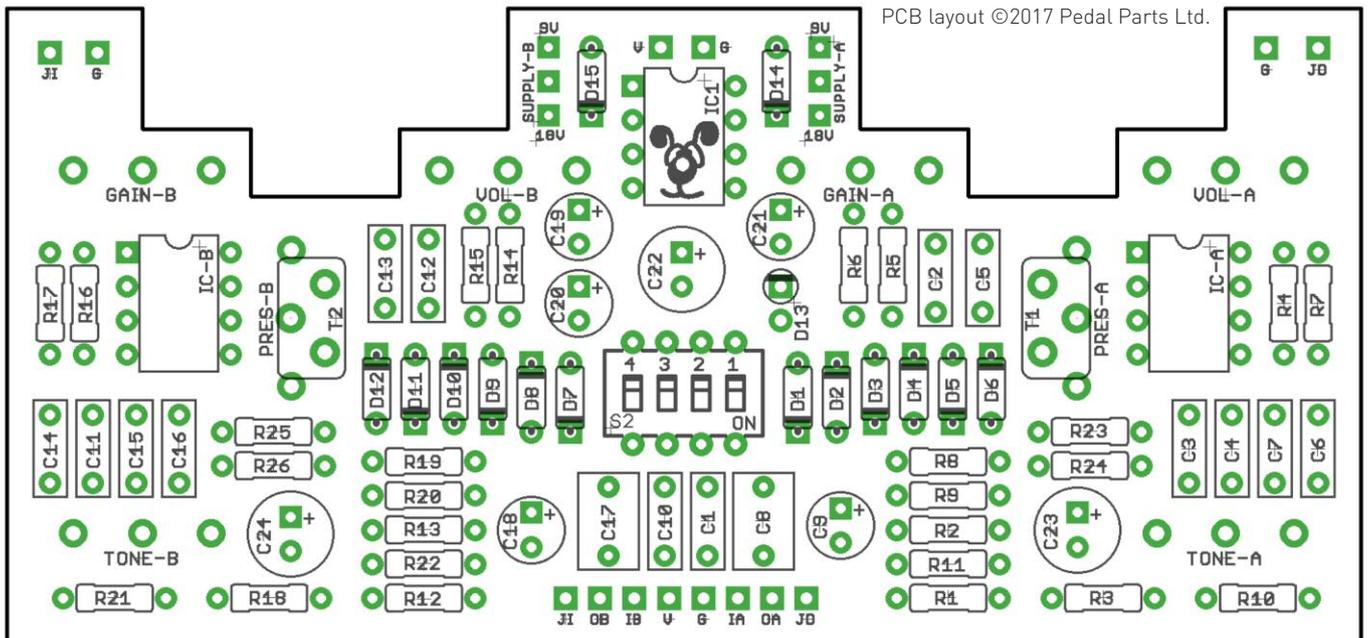
BOM - Main PCB

R1	1M	C1	10n	D1-4	MA856
R2	1M	C2	100p	D5-6	1S1588
R3	27K	C3	10n	D7-10	MA856
R4	33K	C4	10n	D11-12	1S1588
R5	10K	C5	100n	D13	1N4001
R6	10K	C6	10n	D14-15	1N4148*
R7	220K	C7	10n		
R8	6K8	C8	1u	IC1	7660S*
R9	1K	C9	1u	ICA-B	4580
R10	6K8	C10	10n		
R11	1M	C11	100p	VOL	100KA
R12	1M	C12	10n	GAIN	100KB
R13	1M	C13	10n	TONE	25KB
R14	27K	C14	100n	PRES	50KB
R15	33K	C15	10n		OR 47/50K
R16	10K	C16	10n		TRIMMER
R17	10K	C17	1u		
R18	220K	C18	1u	S1	4-WAY DIL
R19	6K8	C19	10u elec*		SWITCH
R20	1K	C20	10u elec*		
R21	6K8	C21	10u elec*		
R22	1M	C22	100u elec		
R23	47K	C23	100u elec		
R24	47K	C24	100u elec		
R25	47K				
R26	47K				

*Only required if you're including the charge pump.

The pads for SUPPLY-A and SUPPLY-B are 2.5mm apart, and will take standard header pins or SPDT slide switches with that pitch. Use these to select 9 or 18V supply for each circuit. You can hardwire either selection with jumper wires.

Diodes D1-D12 are the 'magic ingredients' in the original circuit, but they can be replaced with others to suit your taste. Experiment.



PCB layout ©2017 Pedal Parts Ltd.

Snap the small metal tag off the pots so they can be mounted flush in the box. See later in the document for tips on mounting.

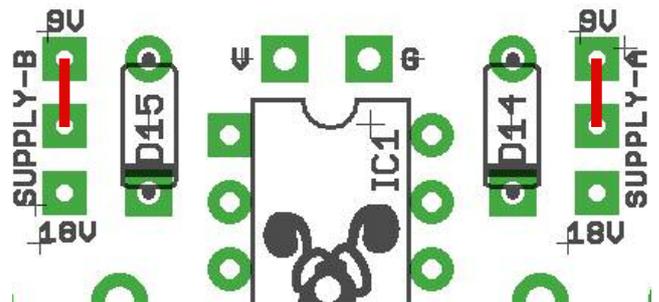
Positive (anode) legs of the electrolytic caps go to the square pads.

Negative (cathode) legs of the diodes go to the square pads.

Be very careful when soldering the diodes and LEDs. 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.

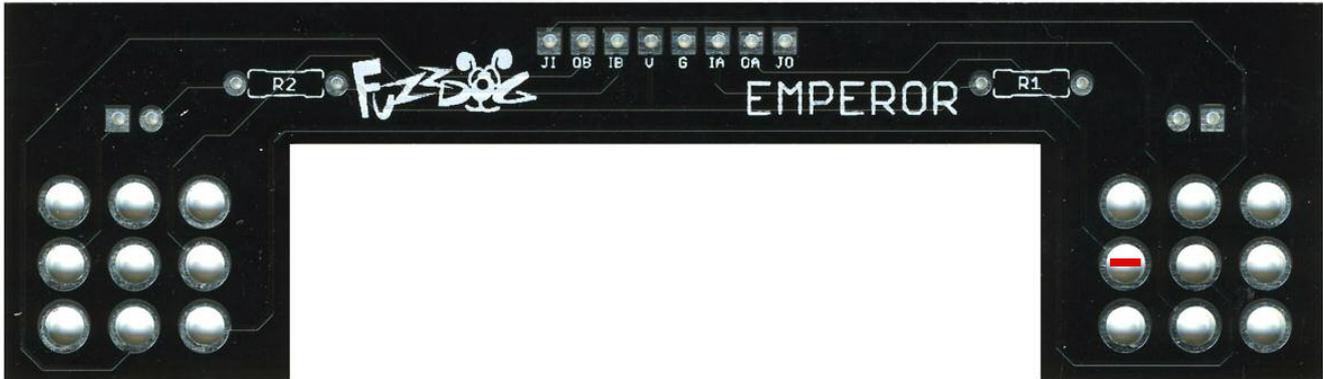
You should solder all other board-mounted components before you solder the pots. Once they're in place you'll have no access to much of the board.

If you aren't adding the charge pump leave out C19-21, D14-15, IC1, and place jumpers as shown below:



Though we supply header pins with the kit, there's no reason you can't attach the two PCBs using ribbon cable or simply wire. Totally up to you.

BOM - Daughterboards



TRUE BYPASS

R1-2 CLR - we use 2k2
SW1-2 3PDT ON-ON

Ensure switch lugs are horizontal
as shown in red above.

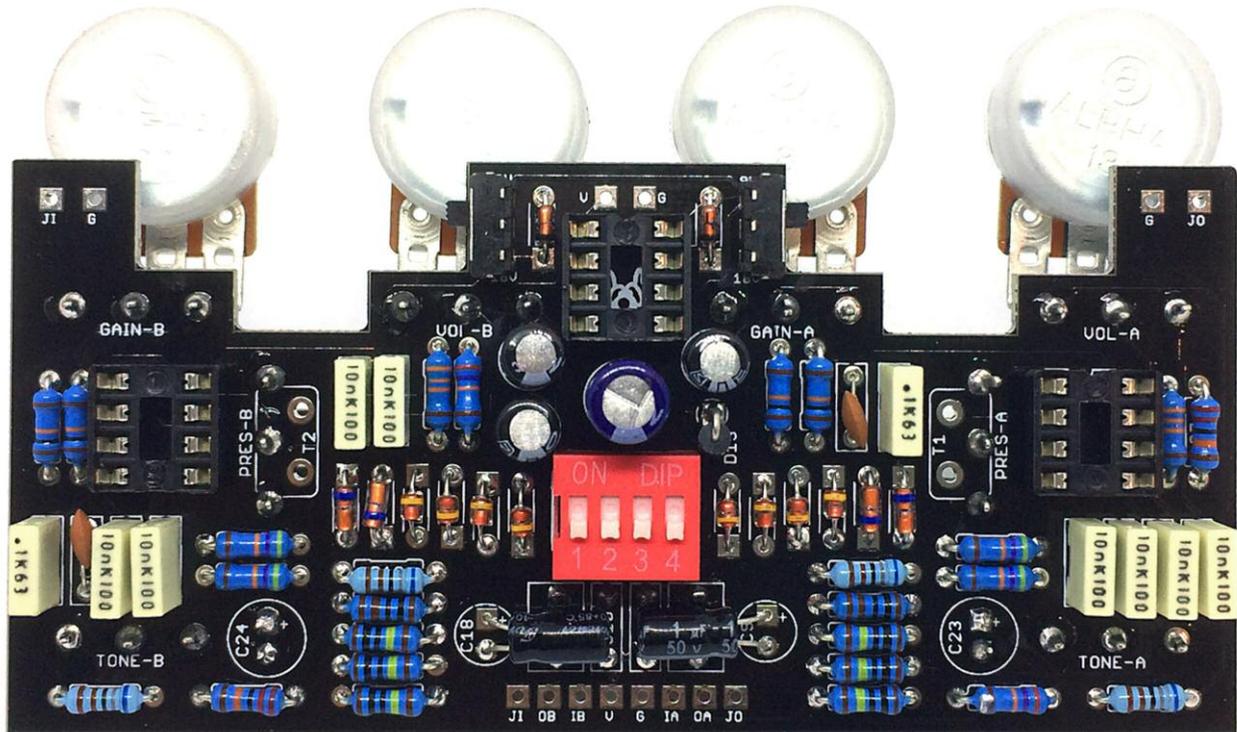


OPTICAL BYPASS

R1-2 2K2
C1-2 220n (these go on the underside of the PCB with the switches)
OK1-2 TLP222G
SW1-2 DPDT ON-ON

Ignore the two small pads marked in red above.
They're simply vias used in routing the signals.

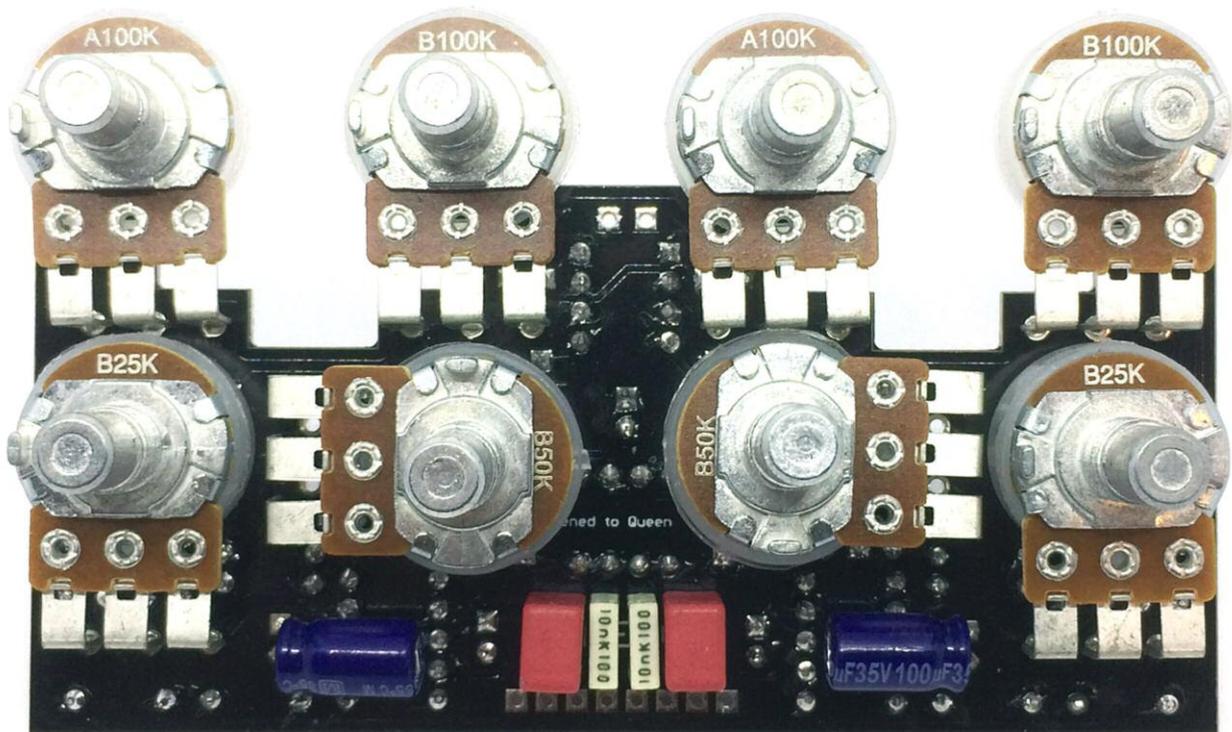
MAIN PCB ASSEMBLY



Nothing too unusual about it. We recommend placing some components on the pot side of the PCB (as shown below) to give better clearance when joining the two PCBs together. It's best to lay C5, C18, C23 and C24 flat against the board.

Use EITHER a pot or trimmer for each PRESENCE control. You can't use both. If using trimmers make sure you put them on the opposite side of the PCB to the pots.

Place pots after all the other PCB-mounted parts. See next page for tips on getting them mounted correctly.



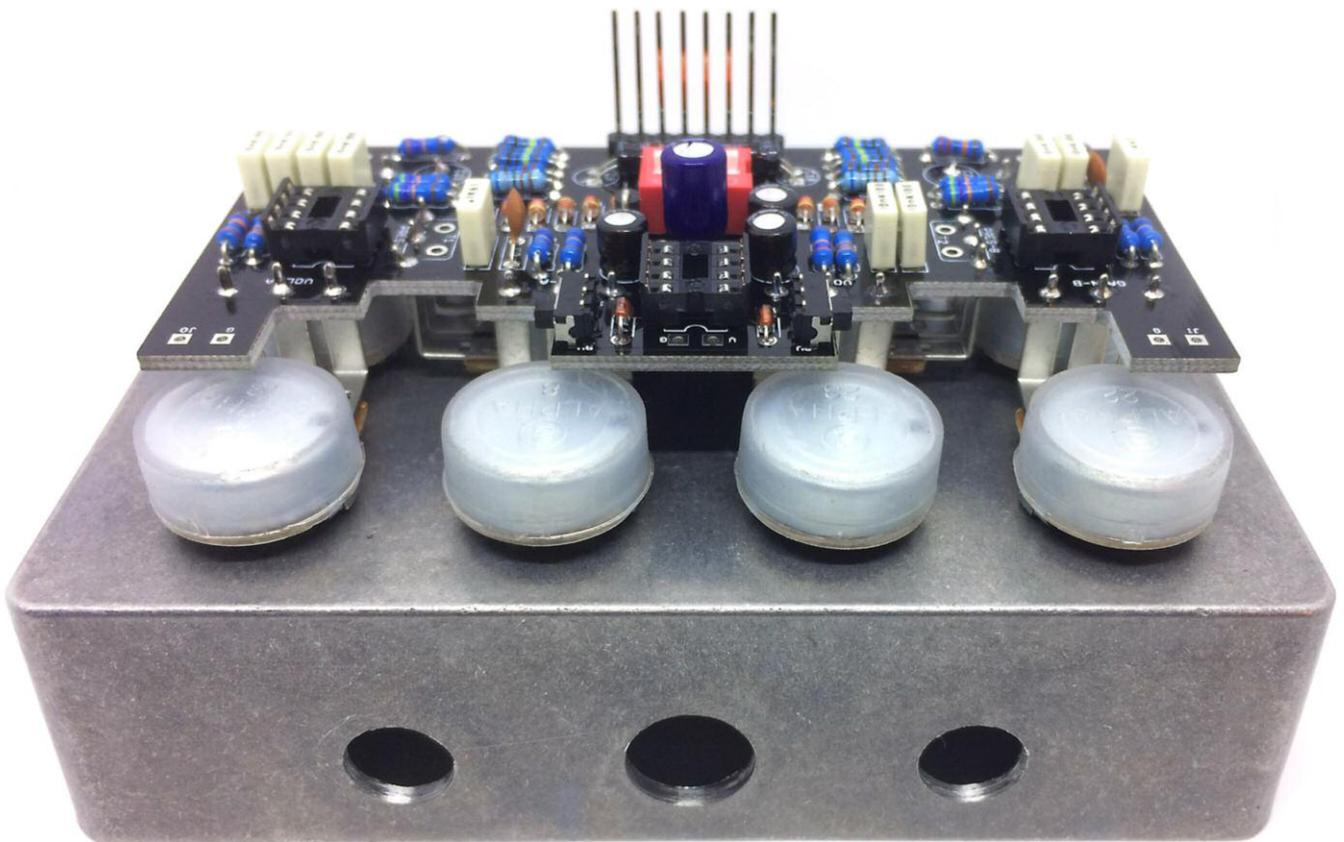
MOUNTING THE POTS

We have a lot of pots on there and they all need to line up nicely, both with the PCB and each other. The best way to do this is a combination of eye and using the enclosure for alignment.

Snap the small locator tags off each pot so they can sit right against the enclosure. Now put each pot in place on the PCB. Now place the enclosure over the top of them as shown below, so the pots sit in the holes. Do a quick visual check above and below the PCB to see if they're all straight.

Now solder in a single pin of each pot. Choose the one that's most accessible. Once they're all tagged in, take the board away from the enclosure and do a visual check of the alignment. If any are sitting at a different height to the others, melt the joint and adjust. Once you're happy everything is as it should be you can solder in the rest of the pins.

If you're using long header pins to join the main PCB to the daughterboard, now is the time to solder that in too. Put the short end of the headers into the main PCB, get it sitting at 90° to the board and solder one pin. Check it is perfectly straight and solder the rest.



If you're leaving out the Presence pots and want to have the Tone pots centered below the Vol and Gain, you're going to have to wire them. Simply bend the legs back on themselves so they are well clear of the PCB and wire them up to the appropriate pads.

ASSEMBLE DAUGHTERBOARD AND JOIN THE PCBS

This procedure is the same for both versions of the daughterboard.

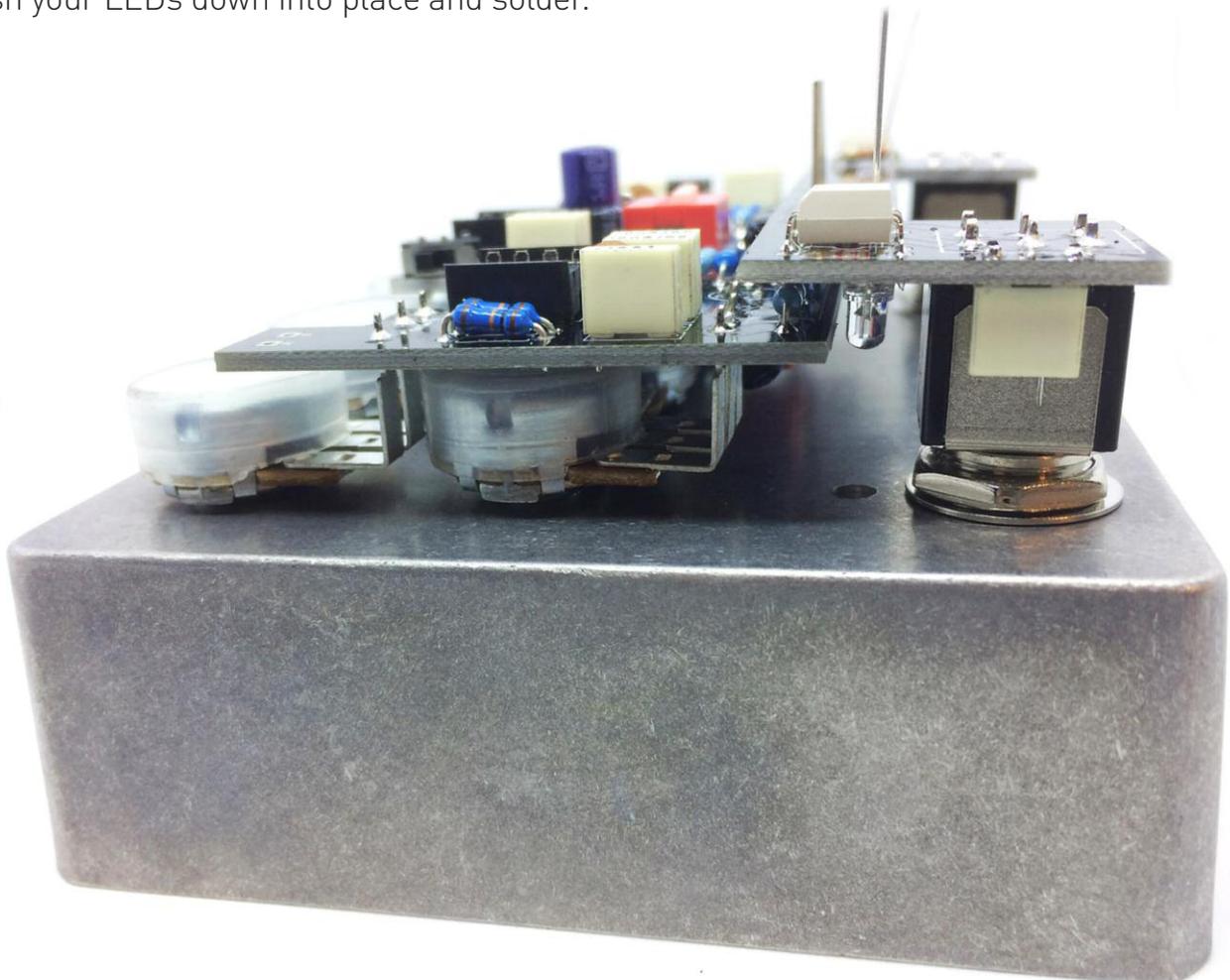
Solder in your daughterboard resistors, caps and ICs as appropriate. Push your LEDs through their pads (long leg to round pad), and bend the legs slightly so they can't slide back out.

Set the height of your footswitches as desired using the lower nut in each case. It's best to have it at least one turn away from the main switch body so you aren't putting any pressure on that when attaching them to the enclosure. Now drop them into place on top of the enclosure, including any internal hardware such as anti-vibration washer or locknut. Loosely fasten them in from the other side using the other nut. Ensure they can still be moved.

Do the same with your main PCB - drop the pots through the holes in the enclosure and fasten two or three of them from the other side.

Now drop your daughterboard over the header pins on the main PCB. They should line up very well. Once located, drop the daughterboard further down and onto your switch pins. The enclosure should have plenty of wiggle room where the pots and footswitches mount, so getting a good fit should be no problem. Once you're happy with placement, solder in your footswitches and the headers.

Once you've done your final assembly and have the circuits in place actually INSIDE the box, push your LEDs down into place and solder.



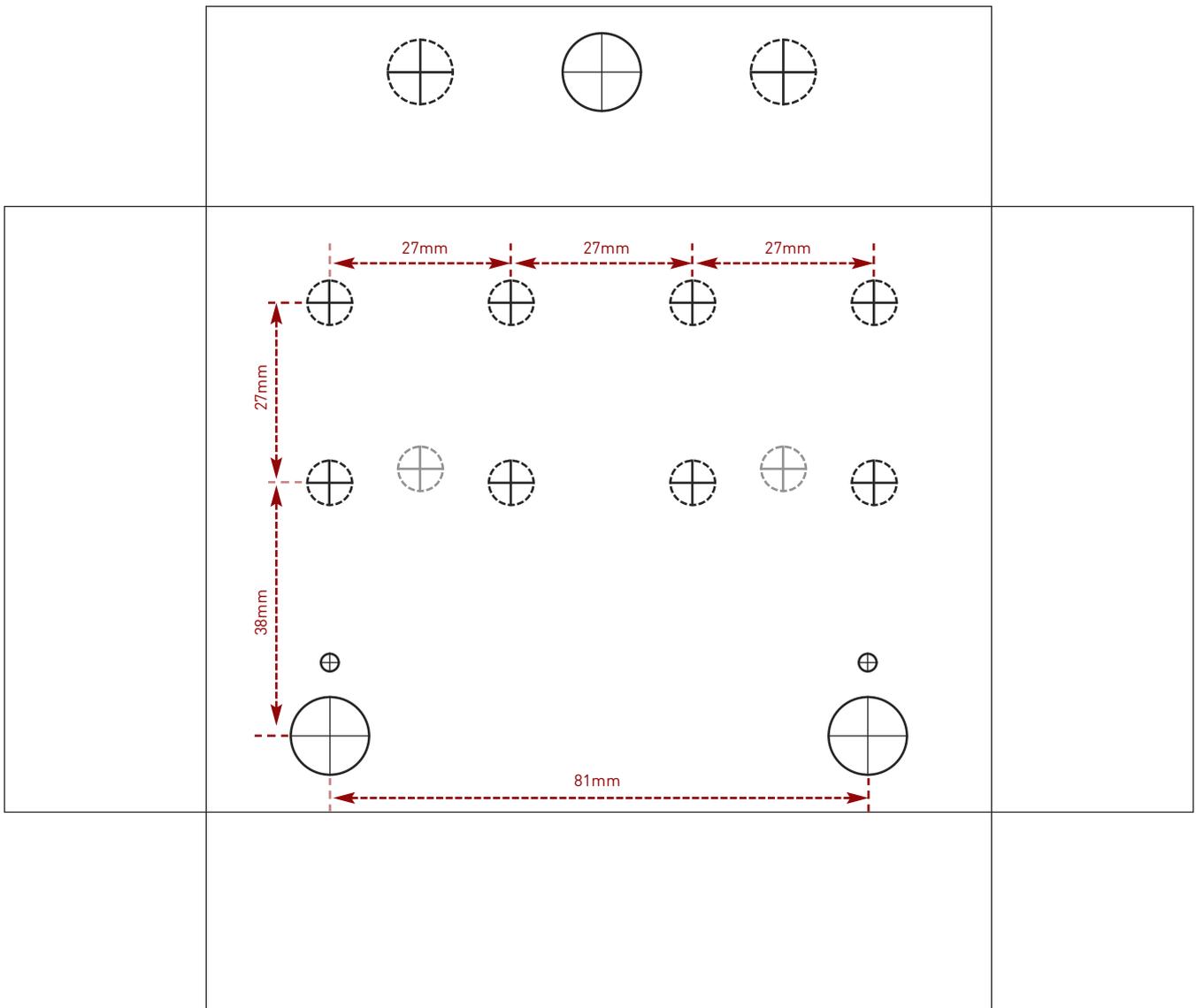
Drilling template

Hammond 1590BB

Recommended drill sizes:

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

It's a good idea to drill the pots and footswitch 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.

If you print this out, please measure the distances to ensure your printer hasn't increased or decreased the size.

FuzzDog.co.uk