

Wrecktifier v2.1

High Gain Distortion fun



A few notes to start with...

This build is not for the beginner. Please do not attempt unless you know what you're doing. The high parts count will make troubleshooting very difficult.

The board has been designed so you can easily add a Boner Boost as the post boost, as the original boost section hasn't won many hearts. Boner is recommended if you want to add some oomph without changing the tone too much. The original is... weird.

Version 2.1 makes adding the Boner simple - no hacks required.

Post boost is not stand-alone - it will only be engaged when the distortion is engaged.

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.

The board is designed so you can easily run the Distortion part of the circuit at 18V by adding the optional charge pump circuitry. The Boost section will still run at 9V. You can even add a small switch or headers + jumper clip to select between 9/18V.

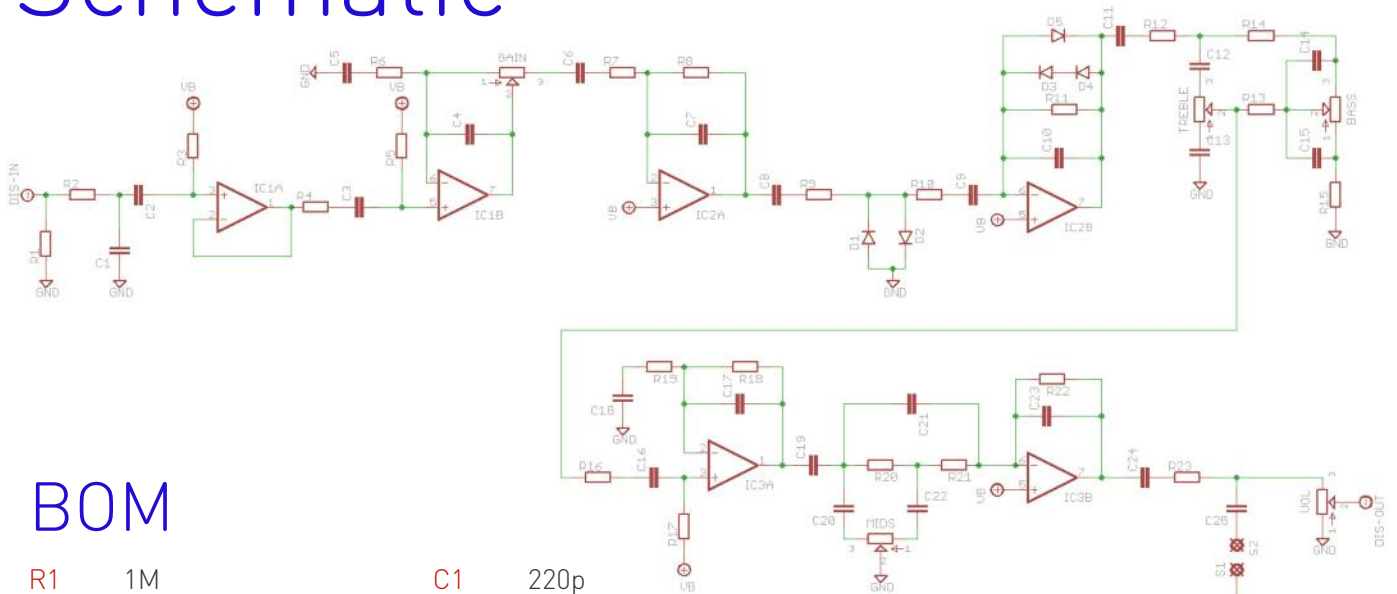
Be very careful when soldering the diodes, LEDs and transistors. They're very sensitive to heat. You should use some kind of heat sink (crocodile clip or reverse action tweezers) on each leg as you solder them. Keep exposure to heat to a minimum (under 2 seconds). Sockets are recommended for the ICs.

The long leg (anode) of the electrolytic capacitors go into the square pads.

The cathode of the LEDs (short leg) and diodes (striped leg) go into the square pads.

Snap the small metal tag off the pots so they can be mounted flush in the box.

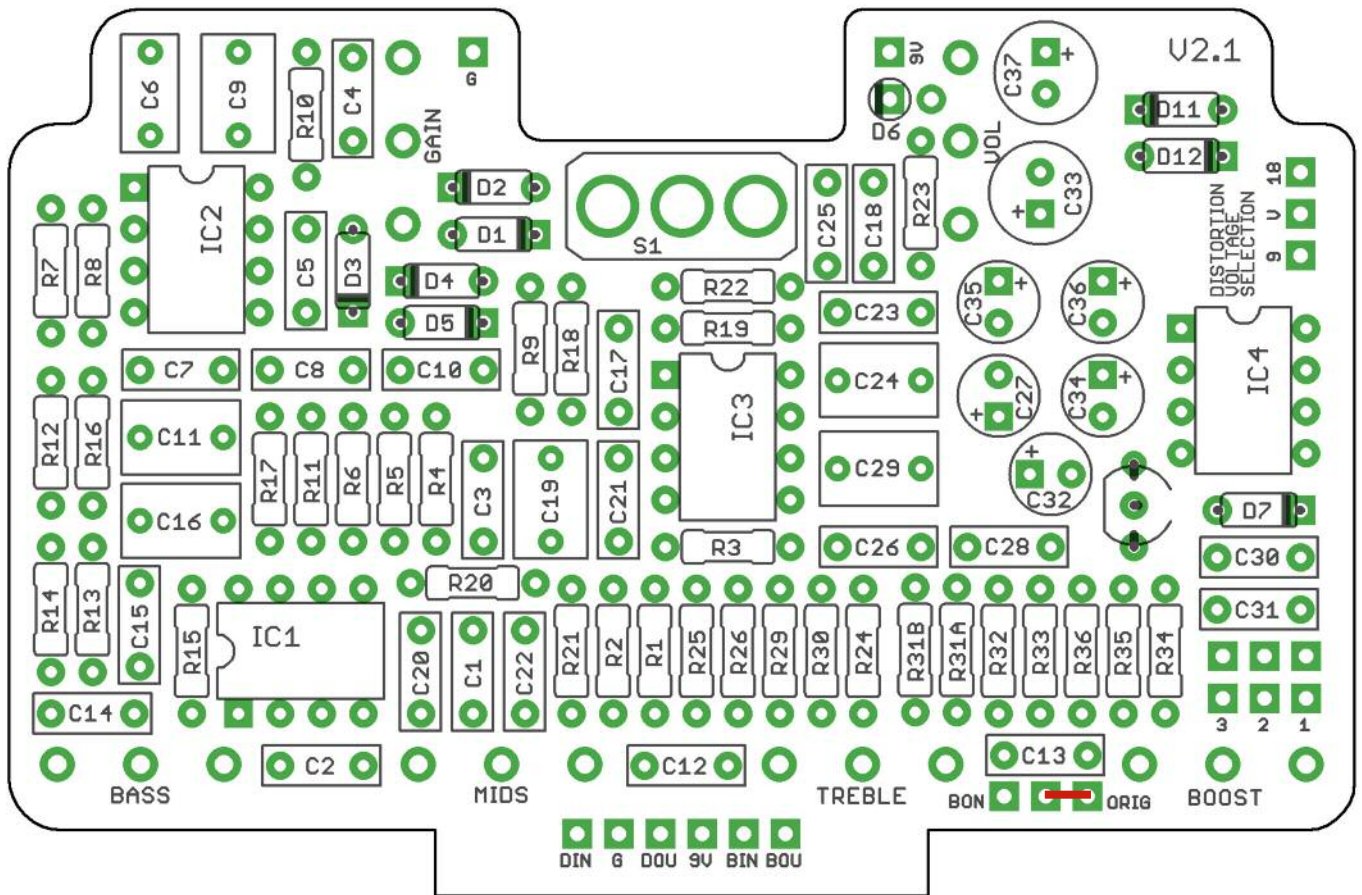
Schematic



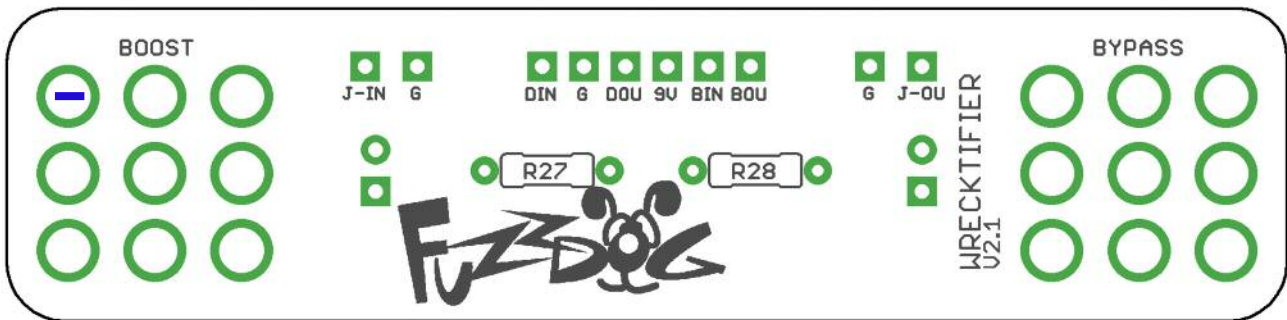
BOM

R1	1M	C1	220p	Q1	2N5088/2N5089
R2	1K	C2	220n	IC1-3	4580
R3	470K	C3	33n	IC4	7660S
R4	10K	C4	560p	D1-5	1N4148 / 1N914
R5	470K	C5	220n	D6	1N4001
R6	1K	C6	1u	D11-12	1N4148 / 1N914
R7	10K	C7	220p	GAIN	100KA
R8	470K	C8	220n	BASS	100KC
R9	1K	C9	1u	MIDS	100KA
R10	47K	C10	220p	TREBLE	50KA
R11	100K	C11	1u	VOL	100KA
R12	1K	C12	4n7	BOOST	100KA
R13	4K7	C13	47n	SW1-2	3PDT FOOTSW.
R14	10K	C14	47n	SW3	SPDT TOGGLE
R15	1K	C15	100n		
R16	470K	C16	1u		
R17	470K	C17	470p		
R18	47K	C18	47n		
R19	47K	C19	1u		
R20	33K	C20	100n		
R21	33K	C21	1n		
R22	680K	C22	47n		
R23	10K	C23	330p		
R24	47R	C24	1u		
R25	10K	C25	4n7		
R26	10K	C26	10n		
R27	2K2 (CLR)	C27	10u elec		
R28	2K2 (CLR)	C28	100p		
R29	1M	C29	1u		
R30	100R	C30	10n		
R31A	1M	C31	10n		
R31B	Empty	C32	2u2 elec		
R32	100K	C33	100u elec		
R33	47K	C34	10u elec		
R34	1K	C35	10u elec		
R35	47K	C36	10u elec		
R36	10K	C37	100u elec		

Part numbers shown in BLUE are for the original boost. Ignore these if building the Boner Boost version. See page 5.
Parts listed in green are for the optional charge pump. See page 6 for power options.



PCB Layout ©2015 Pedal Parts Ltd.



Leave the pots and switches until last. You won't have access to much of the underside of the PCB once they're in place. When soldering these, a good tip is to only attach a single pin of each at first. That way you can easily melt the joint and reposition the pot or switch to get it aligned with the others before attaching the rest. They're difficult to move once fully attached.

If your pots don't have plastic covers you'll have to leave plenty of space between the board and the pot body, otherwise you'll short out the circuit. Place some thick card beneath the pots while you solder them in. Once you have your first pin connected, take it out and ensure you have a good clearance.

The labels on the connector pads on the main pcb and the daughterboard are:
 DI - Distortion IN, DO, Distortion Out, BI - Boost In, BO - Boost Out. Not that you need to know that. just connect them together.

Footswitches mount with the tags horizontal, as the blue line of SW2 shows above.

R28 and R28 can go on either side of the daughterboard.

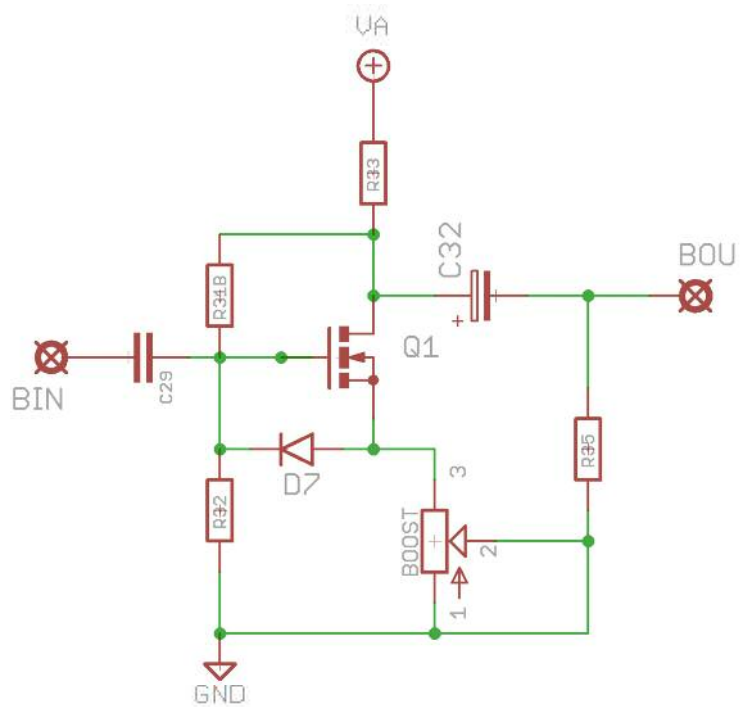
IMPORTANT! If you're building a version with the original boost, place a jumper as shown in red above.

Boner Boost

The Boner Boost mod utilises component spaces that already exist for the original boost along with a few jumpers.

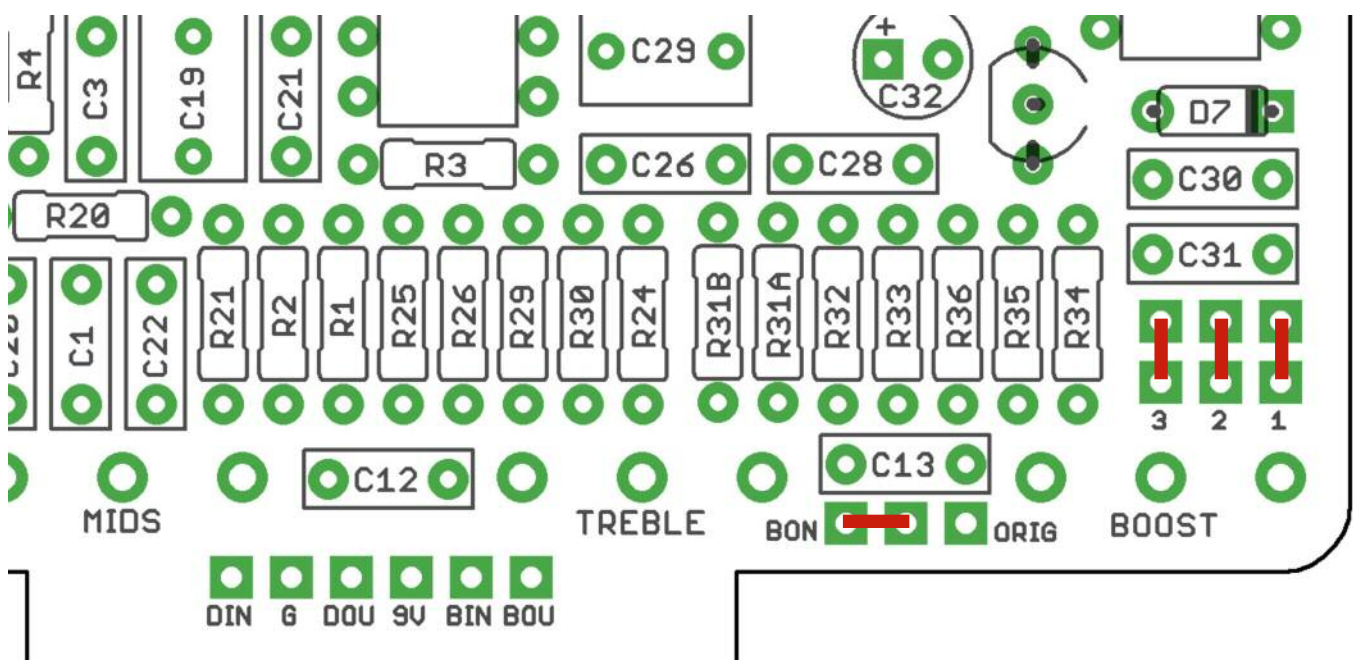
Place parts as listed in the BOM below.

Remember Q1 needs to go in the opposite way around to how it is shown on the PCB screen print.



R29	empty	C28	empty
R30	jumper	C29	100n
R31A	Empty	C30	empty
R31B	10M	C31	empty
R32	10M	C32	10u elec
R33	5K1		
R34	empty	D7	9.1V Zener
R35	100K		
R36	empty	Q1	BS170 (reverse orientation)
		BOOST	5KC

Now you just need to place four jumper wires as shown below.



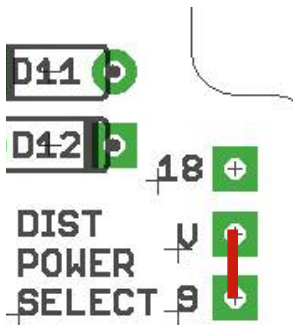
Power

9V? 18V? Up to you. If you choose to add the optional charge pump that will only have a bearing on the distortion circuit. The boost, whether original or Boner, stays at 9V.

NO CHARGE PUMP, EVERYTHING 9V

Leave out all the charge pump parts (listed in green in the BOM).

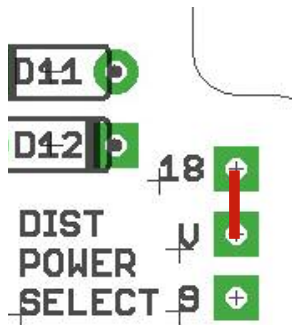
Connect these two pads with a jumper.



CHARGE PUMP, DISTORTION 18V

Place all the charge pump parts (listed in green in the BOM).

Connect these two pads with a jumper.



CHARGE PUMP, DISTORTION 9V/18V

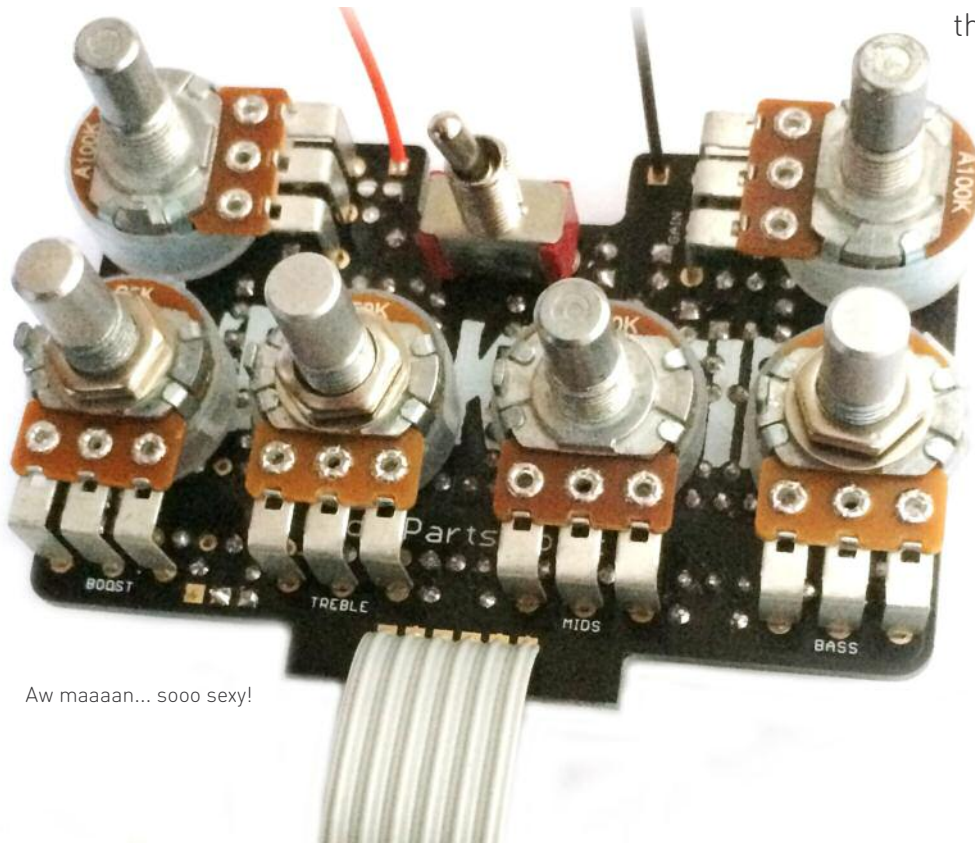
Place all the charge pump parts (listed in green in the BOM).

The 18 / V / 9 pads are spaced at 0.1" (2.54mm), so you have some neat options:

Use a SPDT slide switch with that pitch and drop it straight in there.

Use header pins and a jumper clip. Place the jumper across the top or bottom two sets of pins to choose the power.

Wire a SPDT toggle switch offboard so you can select the power outside of the box. Middle pin of the switch to the V pad.

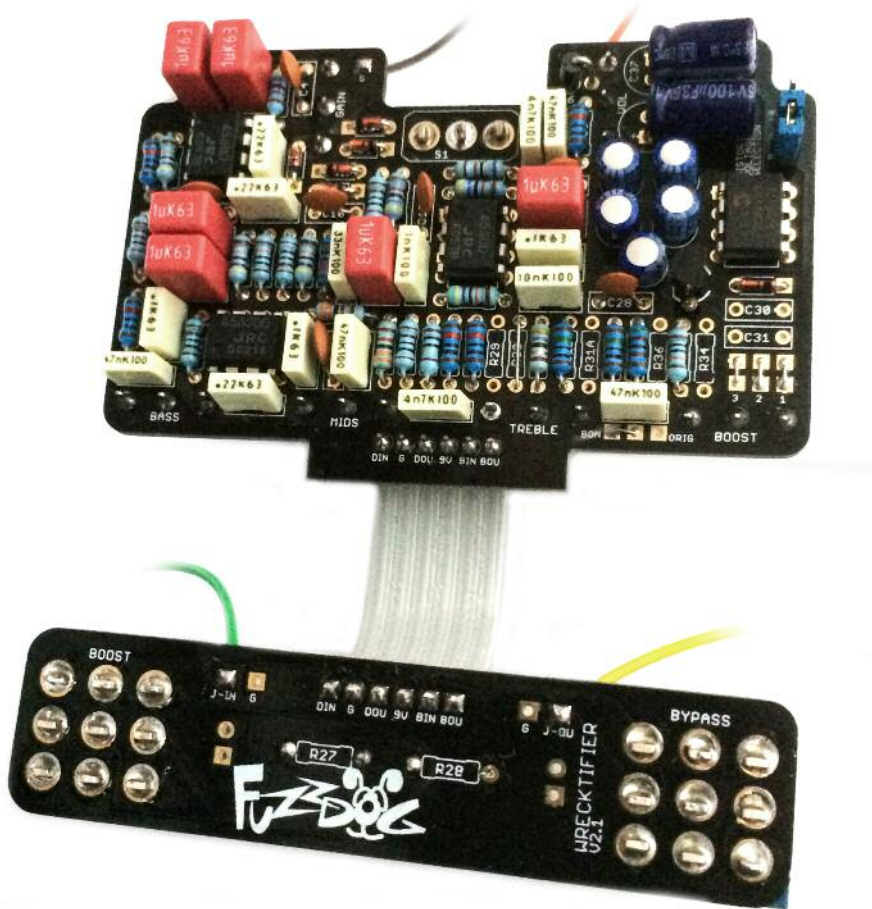


Aw maaaaan... sooo sexy!

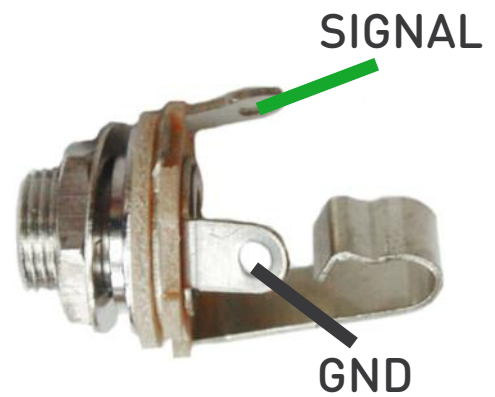
Connect everything up...

Connect everything up but the LEDs. That includes the footswitches. Connect the main PCB to the daughterboard. The pads align perfectly so you have nice straight wire runs.

Ensure your power supply is 9V Tip Negative, or connect up a battery for now. If connecting a battery, solder long lengths of wire to the +V and GND pads on the PCB, then attach the battery to the other end of these. This saves desoldering stuff from the board, which is a pain.

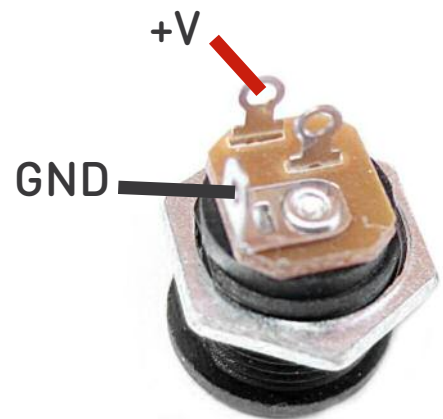


JACK SOCKETS



SIGNAL is JACK IN and JACK OUT on the PCB. Each socket has its own GND connection conveniently placed.

DC SOCKET



Once the sockets are connected... Plug in. Go!

If it works, crack on and get it in the box. If not, troubleshoot. Check you have everything in the right place and reflow any poor joints.

First thing to check - have you put in the appropriate power jumper..?

Ignore the third tag - you only need that if wiring up for battery.

What about my LEDs?

Pre-drilled enclosures are supplied with 3mm holes for the LEDs, rather than larger ones that require a mounting bezel. Why? Because you can hold the LED securely in place with the PCB. When you come to box up your lovely new circuit, get the pots in place first. Then the jacks. Now, slide your LEDs all the way into the PCB (short leg to square pad) and bend the legs ever so slightly so they don't fall out. Alternatively put a little bluetac on there. Now locate your footswitches into place and tighten.

When everything is secure, let your LEDs slide down into the holes - use some needle-nosed pliers or skinny fingers to position them fully in the holes. I won't kid you, its a little fiddly but worth it for the neat finish you'll get.
No bezel = happy pedal.

Once in place, solder.

Those little lights aren't going anywhere!

Drilling template

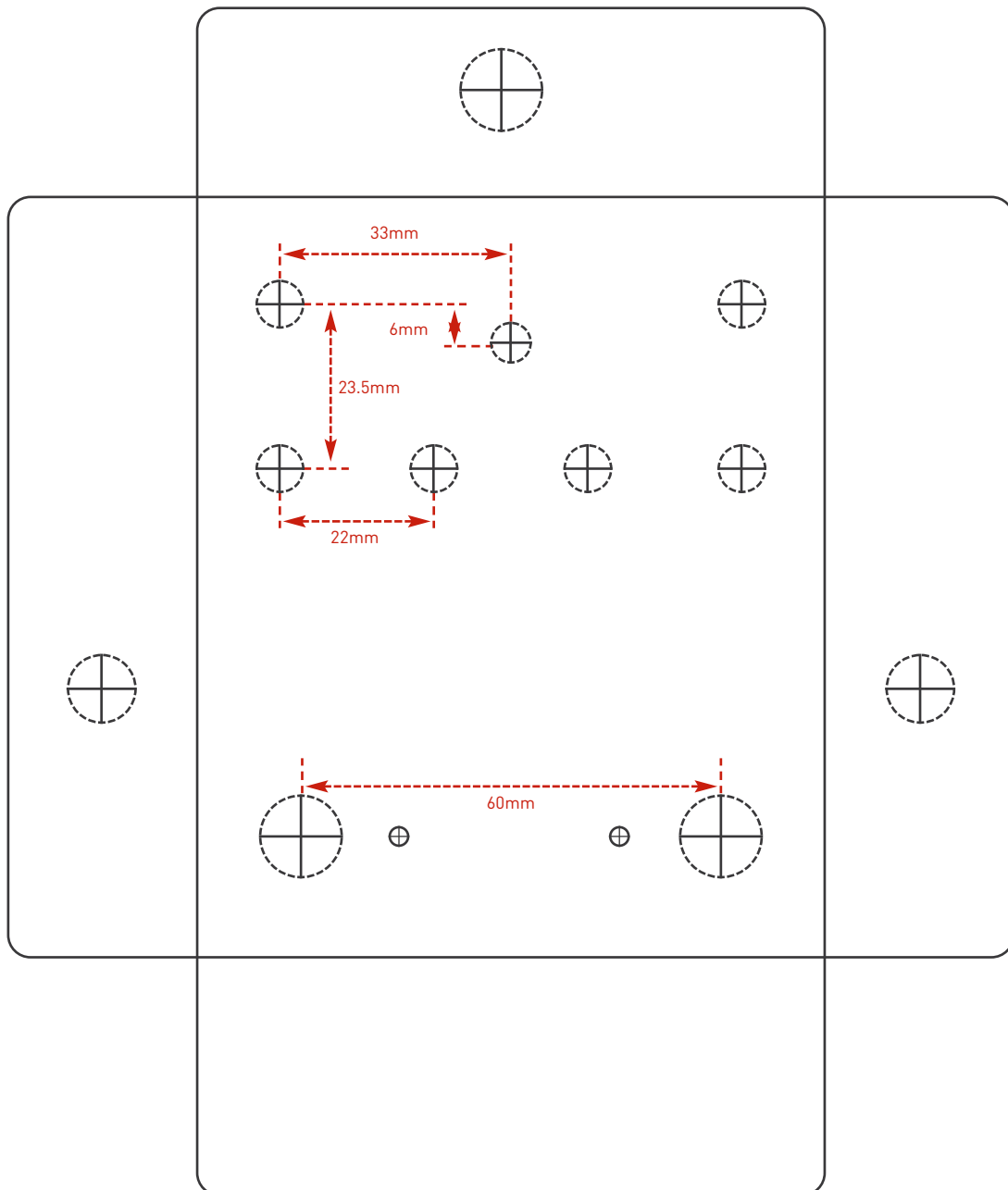
Hammond 1590BB (top face measurements)

90 x 116 x 27mm (without base)

It's a good idea to drill the holes for the board-mounted parts 1mm larger to give yourself some wiggle room unless you're a drill ninja.

Recommended drill sizes:

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

PedalParts.co.uk