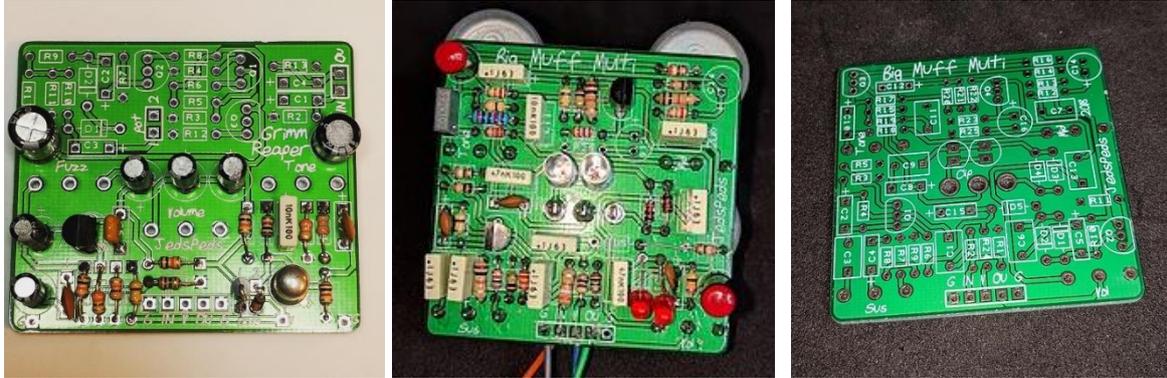


[www.Jedspeds.co.uk](http://www.Jedspeds.co.uk)  
Whoof Rapper PCB Kit



### Parts List – Reaper Side

R1	470K	C1	100PF	<b>Pots</b>	
R2	10K	C2	100UF	Fuzz	250KB
R3	330R	C3	1UF	Vol	100KA
R4	47K	C4	10UF	Tone	100KB
R5	1K	C5	47PF		
R6	10K	C6	1UF		
R7	1K	C7	1UF	D1	4001
R8	33K	C8	22UF	D2	4148/914
R9	33K	C9	470PF		
R10	---	C10	10NF	Q1	MPSA18
R11	10K	C11	1UF	Q2	AC176
R12	1M				

### Parts List – Green Side

R1	22K	R11	22K		
R2	1M	R12	10K		
R3	150K	R13	47K	Q1	2n5088
R4	6K2			Q2	2N3906
R5	560K	C1	47N	Q3	2N5088
R6	18K	C2	47N		
R7	10K	C3	47N	D1	1n34A
R8	10K	C4	100N	D2	1N34A
R9	68K			D3	1N4001
R10	68K				

## Parts List – Whoof Muff

R1	1m	R12	12k	R23	2k7	C1	100n	C12	100n
R2	39k	R13	10k	R24	100r	C2	470p	C13	100n
R3	100k	R14	100k			C3	100n	C14	100u
R4	470k	R15	470k			C4	100n	C15	100n
R5	390r	R16	390r			C5	470p		
R6	12k	R17	12k			C6	47n	D1-4	914
R7	1k	R18	22k			C7	100n		
R8	10k	R19	20k			C8	470p	VOL	500ka
R9	100k	R20	470k			C9	47n	Sus	50KA
R10	470k	R21	100k			C10	3n9	TONE	100kb
R11	390r	R22	10k			C11	10n	Mid	25kb
								Q1 +4	3904
								Q2 + 3	GE NPN

Please read the guide in full before starting your build. If it is blatantly obvious you haven't read it and contact us for help then don't be surprised if we tell you to read the guide again... harsh I know.

### **Kit Specific Build Notes – Green Reaper.**

- The Reaper is a modified Tone Bender Fuzz pedal. It uses a hybrid pairing of transistors to produce a variety of fuzzy tones.
- The Reaper only uses half the PCB, the other half is a Green Ringer circuit to add an octave up section for our Green Reaper kit. The image above shows which half to build the Reaper side on. The top half is the Green Ringer side.
- The diodes are designed to stand up. The striped side of the diodes goes to the square pad on the PCB.
- There are two extra ground pads in the bottom corners of the PCB to use as you wish.
- The pots on this kit will fit like this if using the right-angled variety.

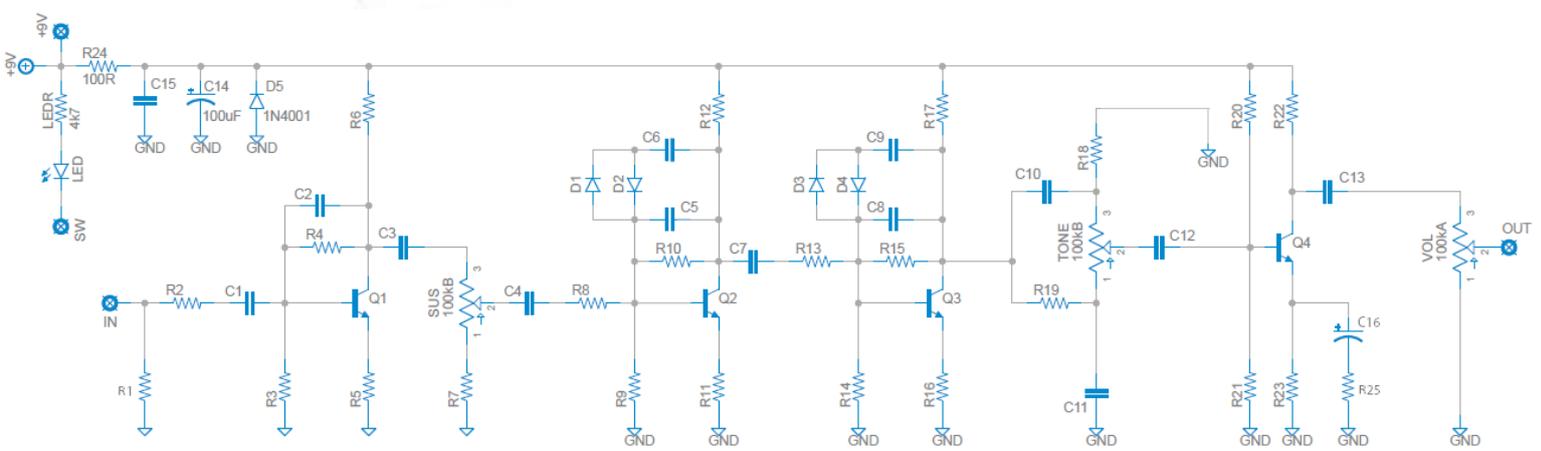
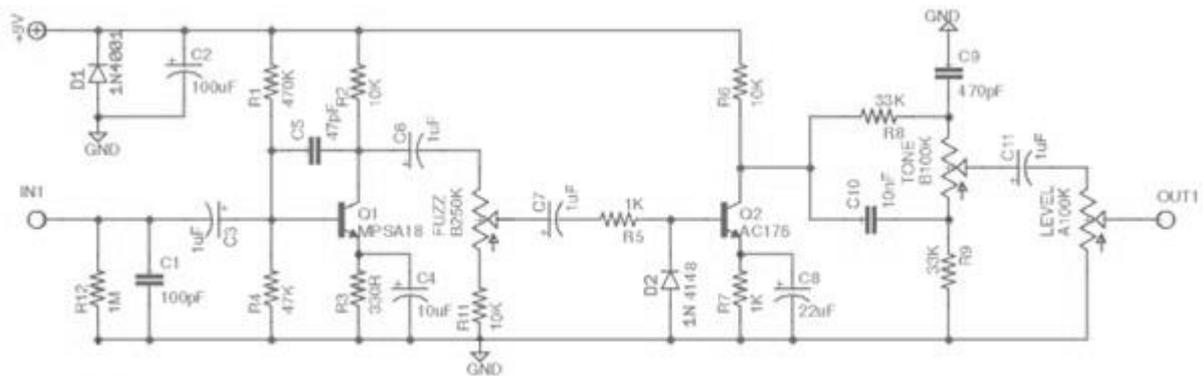


- There is also an extra V pad, which you could put your CLR into if you wish. Its below Q2.
- The two pads in the Green Ringer marked POT need to be jumpered with a resistor leg off cut. You can, if you wish add the 8<sup>th</sup> pot here, wire pad 1 and 2 of a 20kb pot to each lug it it slightly alters the shrillness of the octave. Not essential.

### **Kit Specific Build Notes**

- D5 is featured on the PCB. It is an optional extra bit of power filtering. There is already a fair bit of filtering in the circuit in my opinion...it's up to you, experience noise – drop in a 4001
- The toggle is not included on the Whoof Rapper kit. Therefore you need to hard solder from the centre pad to the right pad as you look at the PCB from the front.

## Schematic for Reference



## General Build Instruction

The first thing you must do is identify your parts from the pack. As a rule, I build from small to big – therefore step one will be to identify the resistors and any diodes inside your kit.

Each kit has a parts list at the top of the guide. So as an example, if R1 is listed as a 1M resistor then you dig out a 1M from the pack and place it on the PCB in the R1 spot. It's your call if you put one in and solder or put a few in or even put them all in and solder. I'll let you decide.

The front of the PCB is the side with the white writing on (the silkscreen)

You then need to follow the same process for all the other parts included. Working in height order mount the rest of the parts to the PCB ensuring that you solder them in place well as you go.

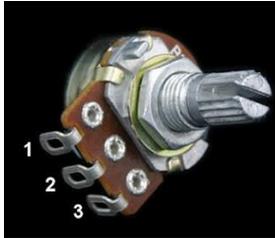
Some parts are quite heat sensitive so you must be careful when soldering them. Diodes, chips and transistors mainly. They can easily burn out by overheating.

I would then add the offboard wires, starting with nice long ones leaving them to be trimmed to length later. Finally, I add the pots either mounting them to the board or wiring them into place.

## Useful links

[Resistor calculator](#)

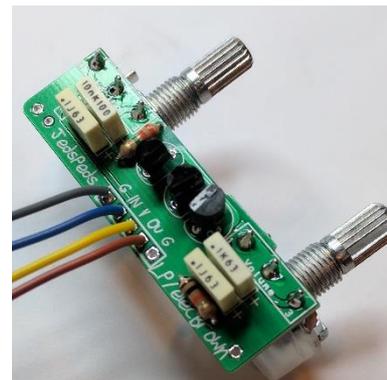
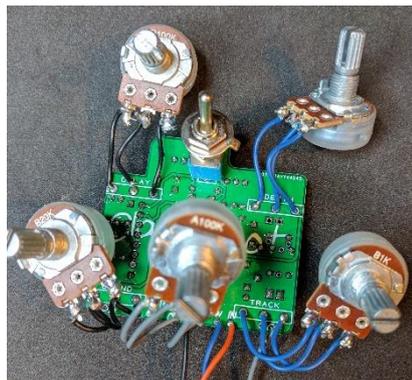
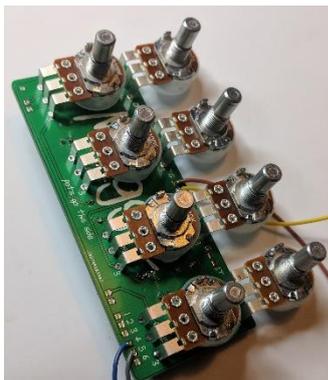
## Pots



Pin Numbers for the pots. Snap the little metal tag off before trying to put it through the holes in the box.

Some of our PCBs are now designed to use right angled mount pcb potentiometers. They mount from the rear as indicated by the pcb. We will provide right angled pots as stocked, this means that you may receive a “normal” pot on occasion and you will have to wire it. It’s not that inconvenient now is it.

The other options for pots are that they are wired into place or the PCB hangs from them.



## Hooking up the Jacks

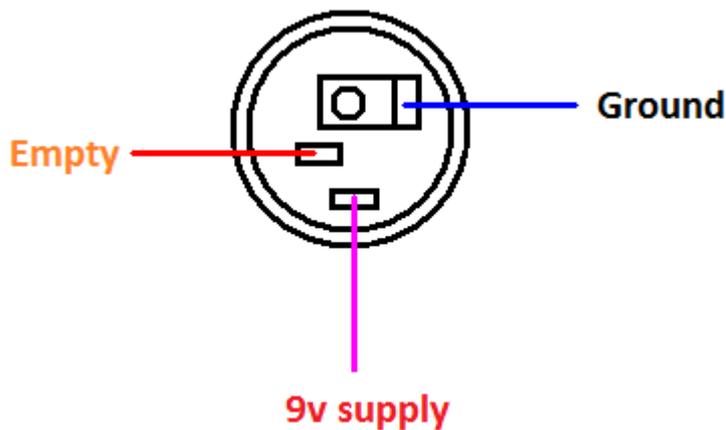
Our kits come with mono jacks. They have two connections, a positive or the tip connection. And a negative, or sleeve connection.

The tip connections will connect to the 3PDT as shown later in the guide unless there are instructions otherwise.

The sleeve connections are ground points, all grounds throughout a build must connect. The Jack socket will then connect to the enclosure and ground the case.



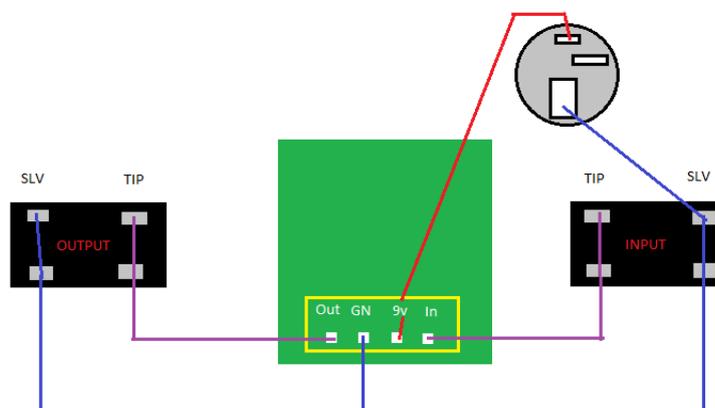
## Power Socket



## TEST THE BOARD

When you get to the stage with the board and pots wired you need to test the board before you add the foot switch. Firstly, don't put it in the box – I see people building inside a tiny enclosure and I wonder why. Some PCB's are tight enough as it is without reducing your workspace to 6cm wide!

Connect Input pad to TIP of Input jack. Same with output. Then connect the power to the 9v pad. Connect all grounds together (board, jack sleeves and 9v) If your board is working at this point then you can start work on the switch! If not...get double checking! If you email asking me why your pedal isn't working the first thing I will ask is... "did it work without the switch?", so make sure you follow this step.



If your PCB is working you are ready to connect the board to the footswitch! Follow the next stages meticulously and all being well by the end of it you will have a lovely finished pedal!

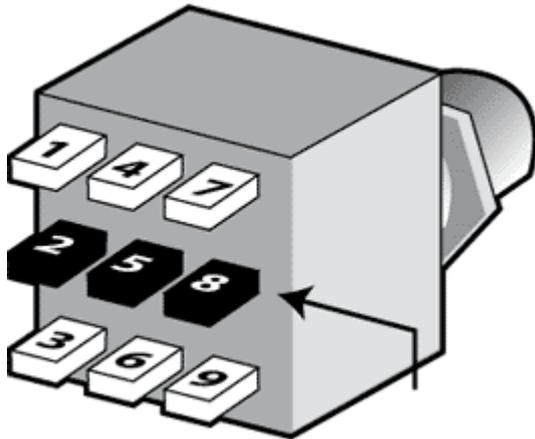
The next step involves undoing the tip wires from the PCB as they will now move to the 3PDT.

If its not working at this stage then you need to find out why. Do not continue beyond this point until you get it working. Adding a footswitch will not magically make a circuit work.

Troubleshooting is a necessary evil at times.

## The Footswitch.

Have a look at the footswitch. It has 9 pins. The orientation of the switch is crucial. If you do it wrong your switch will not work, you might ruin it and you will certainly have to un do it.

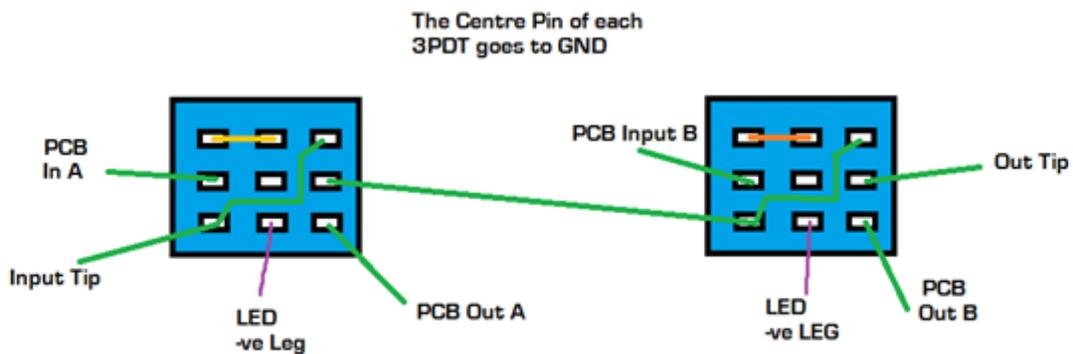


The pins on the switch must run left to right like this, so the top row of pins would be 1, 4, and 7.

If you put it in with 1, 2, and 3 across the top row you are wrong.

There are loads of ways to wire up the footswitch. I use this one, I always have. Its never let me down and is easy to hook up.

**Please make sure you get the switch the correct way!**



Repeat the pattern for the third switch, with out tip heading to In tip as shown between these two. Hopefully that makes sense...you could go forever and have a million switches 😊

The CLR 2k2 resistor can be any value up to 4k7. You will have a spare in the kit to use.

Good luck and happy soldering!

## PCB Design Notes



1. All PCBs are designed by ourselves, and tested before sale.
2. Box caps are shown with a + polarity mark on the silk screen, this is irrelevant as the box caps are non-polarised so can go either way.
3. Spare ground points may be provided on the PCB layout, use them if you wish but there is no requirement to.