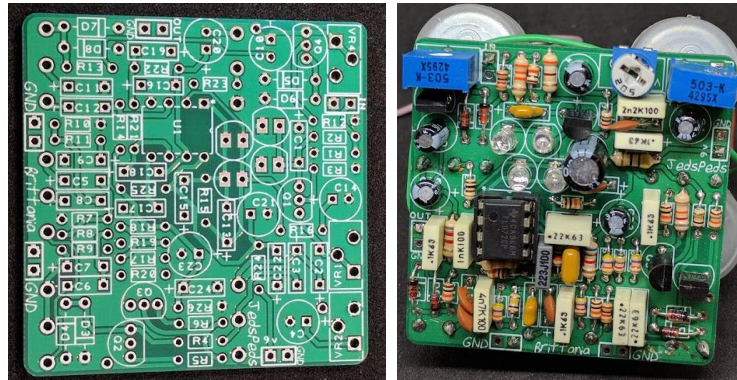


www.Jedspeds.co.uk
R.O.G Britannia PCB Kit



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.

Parts List

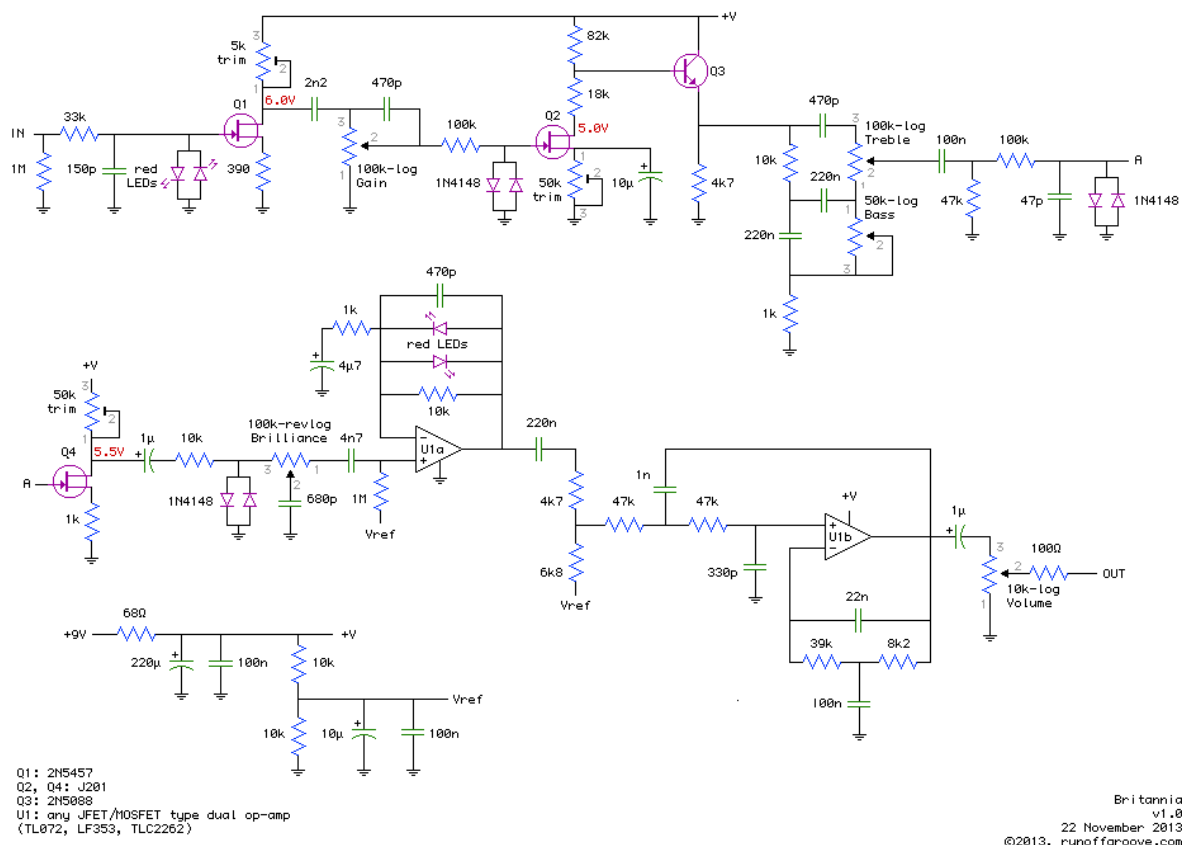
R1	1M	R14	1M	C1	150PF	C14	4U7	VR1	5K	Q1	5457
R2	33K	R15	10K	C2	2N2	C15	220N	VR2	50K	Q2	J201
R3	390R	R16	1K	C3	470P	C16	1N	VR4	50K	Q3	5088
R4	100K	R17	4K7	C4	10UF	C17	330PF	D1+2	RED LED	Q4	J201
R5	18K	R18	6K8	C5	470PF	C18	22N	D3+4	1N4148		
R6	82K	R19	47K	C6	220NF	C19	100N	D5+6	1N4148		
R7	4K7	R20	47K	C7	220NF	C20	1UF (E)	D7+8	1N4148		
R8	10K	R21	39K	C8	100NF	C21	220U	D9+10	RED LED		
R9	1k	R22	8K2	C9	47PF	C22	100N	GAIN	100KA		
R10	4k7	R23	100R	C10	1UF (E)	C23	10UF	TREB	100KA		
R11	100k	R24	68R	C11	680PF	C24	100N	BASS	50KA		
R12	1k	R25	10K	C12	4N7			BRILL	100KC		
R13	10k	R26	10K	C13	470PF	U1	TL072	VOL	10KA		

Kit Specific Build Notes

- This is a hard kit. It's very involved. Its tight. Its fiddly, just look how many parts there are.
- I found that the Brilliance pot is quite a subtle setting.
- **This is a circuit designed by runoffgroove.com.** I highly recommend reading their detailed breakdown on their website <http://www.runoffgroove.com/britannia.html>. This includes instructions on how to set the pedal up the first time you use it.
- You have to bias the transistors on this one. Near to each transistor is a pad marked T.P with a voltage indicated on it, you are looking to set the trimmer until that pad reads that voltage with a multi meter. Biasing is fairly straight forward, but without doing it your pedal will probably sound a bit poo. You will need to use a multi meter. The black lead goes to a ground pad, and the red lead to the test pad, use the trimmer to find the required voltage.

- VR1 pad needs to be set to 6.0v.
- VR2 pad needs to be set to 5.0v.
- VR4 pad needs to be set to 5.5v.
- As the project uses thru hole Jfets, appropriate subs may be sent depending on what we have in stock. Global depletion is a phrase that springs to mind.

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.

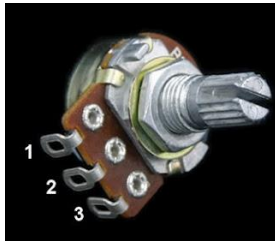
Some parts are quite heat sensitive so you must be careful when soldering them. Diodes, chips and transistors mainly. They can easily burn out by overheating.

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](#)

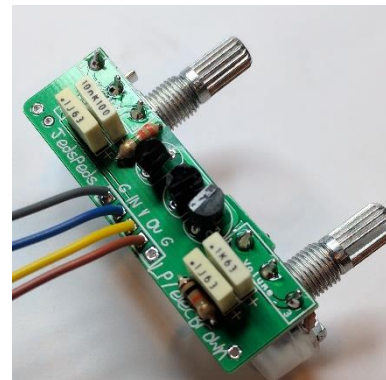
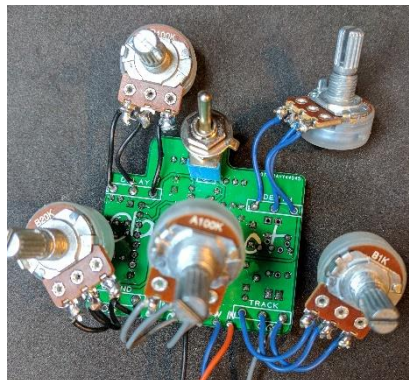
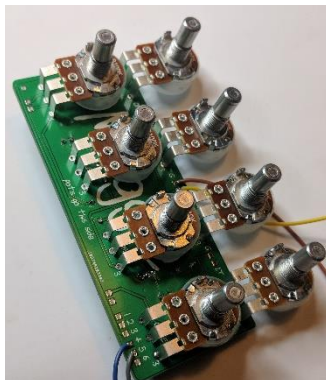
Pots



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.

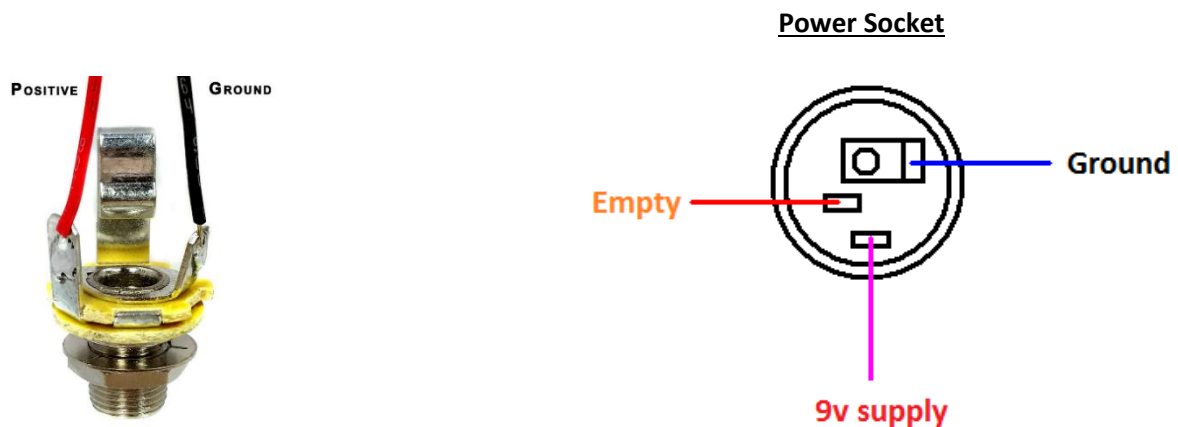


Hooking up the Jacks

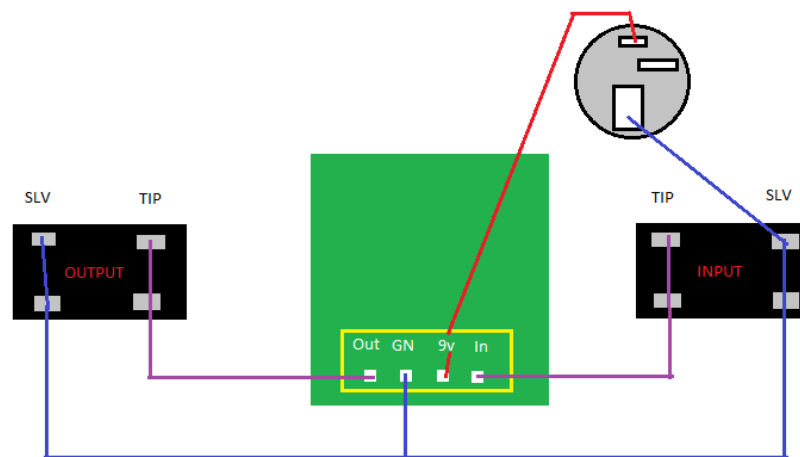
Our kits come with mono jacks. They have two connections, a positive or the tip connection. And a negative, or sleeve connection.

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.



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!

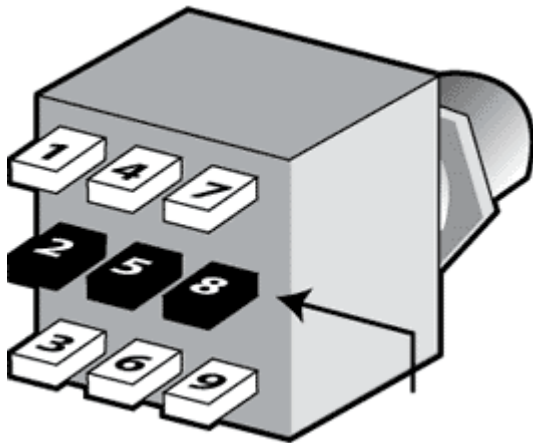
The next step involves undoing the tip wires from the PCB as they will now move to the 3PDT.

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.

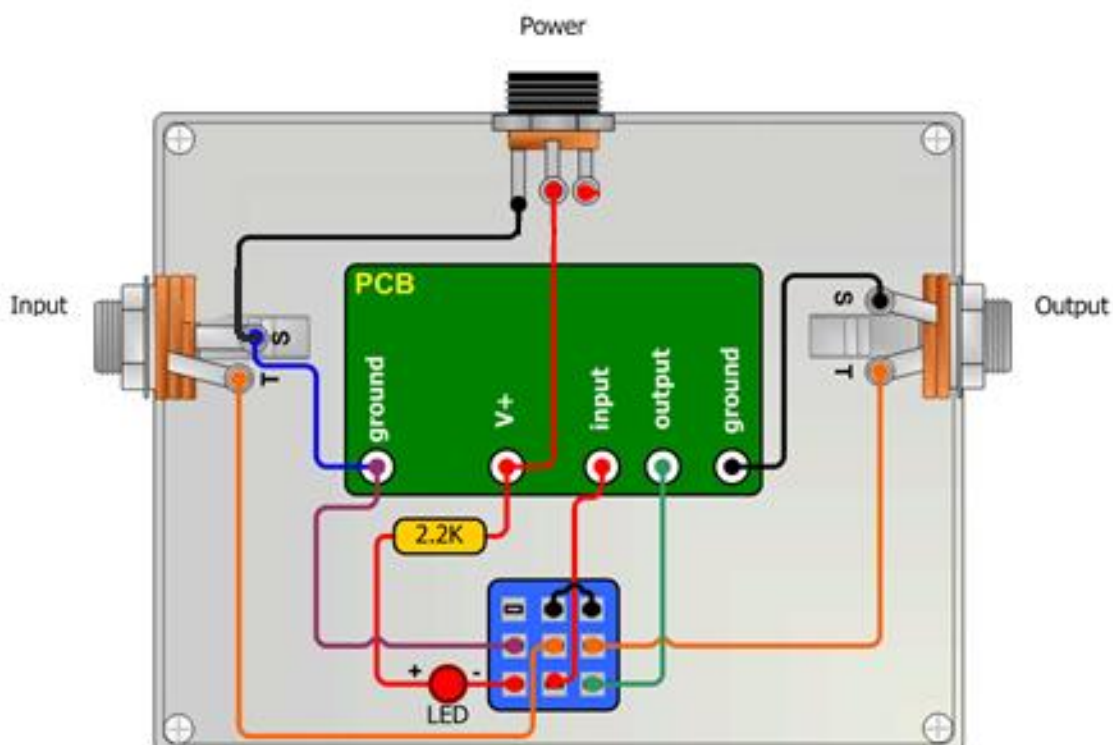
If you put it in with 1, 2, and 3 across the top row you are wrong.

To help you a little further with ensuring the orientation of your switch is correct. To compare both 3PDT images you would have this...

- Pin 1 – Empty
- Pin 2 – Ground
- Pin 3 – LED
- Pin 4 – Pin 7
- Pin 5 – Input Jack tip
- Pin 6 – PCB Input
- Pin 7 – Pin 4
- Pin 8 – Output Jack Tip
- Pin 9 – PCB Output

Please make sure you get the switch the correct way!

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.



The 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.