Overunder PCB Kit

Please read the guide in full before starting your build. It contains a lot of really important information which will give you the best chance of a successful build.

For build support, please join our [Facebook Forum.]

Please check the kit against the parts list upon receipt, I hand pick kits to order and I am only human.

 truebypass@jedspeds.co.uk

Contents

Parts List (main pcb)							
R1	1K	**C1**	10u	**Q1**	2n3904		
R2	10k	**C2**	100n	**Q2 + 3**	AC128		
R3	47k	**C3**	10u	**D1**	4001		
R4	470r	**C4**	10u				
R5	5k1	**C5**	47u	**Pre**	250Kb	**Vol**	5kb
R6	220k	**C6**	18n	**Blend**	100kb	**Stab**	10kb
R8	1M	**C7**	100n	**Drive**	10kb	**Gate**	10kb
		C8	10u	**Tone**	10kb	**Comp**	10kb

Offboard Components	
3PDT footswitch	
Footswitch PCB	Included with the kit unless out of stock
LED	Coloured
4K7 Resistor	This is for limiting the current to the LED (also known as a CLR)
Mono Jacks x2	
Power Socket 2.1mm	
Wire	
Socket + Sil row	

Suggested Tools

Essential	**Desirable**
Soldering station	
Solder - 60/40 or good quality lead free	
Wire snipping pliers	
Wire stripping pliers	
Solder sucker / de solder braid	
PCB Clamp	
Multi Meter	
	Centre Punch
	Drill bits
	Tweezers

The essential tools are all the bare minimum of what I use to build a pedal. The list is not exhaustive, but we do get a lot of enquiries about what tools people need.

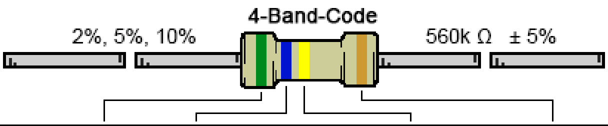
I sell a large range of pedal building related tools here.

A Detailed Overview of the Components in this Kit

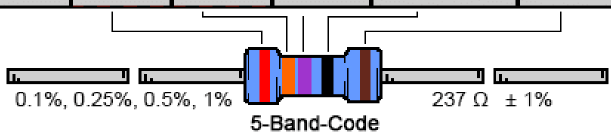
Resistors

If we use the analogy of water rather than electricity, then resistors would be called restrictors. They restrict the flow. Resistors are non-polarised (they can be inserted either way round). The body of a resistor can be one of many colours. Most commonly seen in our kits are blue or buff.

What really matters is the colour of the bands around the body. There may be 4 or 5 of these. on a 4 band resistor the first two colours give you the value for the resistor and the 3rd gives you the number of zeros to add (aka the multiplier). With a 5 band resistor the first three give you the value and the 4th gives you the zeros. See the table below.



COLOR	1ST BAND	2ND BAND	3RD BAND	MULTIPLIER	TOLERANCE
Black	0	0	0	1Ω	
Brown	1	1	1	10Ω	± 1% (F)
Red	2	2	2	100Ω	± 2% (G)
Orange	3	3	3	1KΩ	
Yellow	4	4	4	10KΩ	
Green	5	5	5	100KΩ	± 0.5% (D)
Blue	6	6	6	1MΩ	± 0.25% (C)
Violet	7	7	7	10MΩ	± 0.10% (B)
Grey	8	8	8	100MΩ	± 0.05%
White	9	9	9	1GΩ	
Gold				0.1Ω	± 5% (J)
Silver				0.01Ω	± 10% (K)



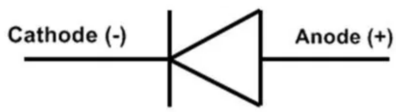
So the 4 band resistor above that has rings of green, blue, yellow and gold is a 560000Ω (or 560kΩ). You will see this 'value plus zeros' reappear when we get to capacitors. Note that a 4.7kΩ resistor would normally be written by me as 4k7.

Note that if the multiplier is gold or silver you divide the leading figures by 10 or 100.

You can either decode them yourself following the above table, alternatively this online resistor calculator or you can use a multi meter. If you want any further explanation about how to decode resistor values, this video *may* help.

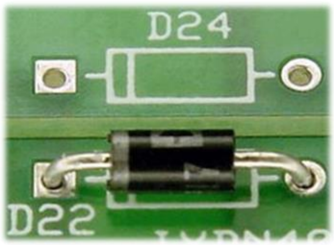
Diodes

Again, going back to the water analogy, diodes can be thought of as one-way valves. They allow electricity to flow in only one direction. The electrical symbol for a diode is shown below. They must be inserted in the correct direction for the circuit to work.



The anode is the positive end and the cathode is the negative. electricity will only flow from the anode to the cathode. Diodes can be various colours but usually have one or sometimes two bands at one end.

In this kit you should have 3 diodes. The symbol on the PCB will be something like this, simply line up the band on the diode with the line on the board.



The diodes in this kit should look like this;

1N4001

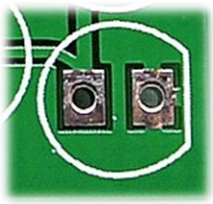
Be careful when you bend the legs.

Diodes, like all silicon components, are heat sensitive. When soldering, limit the contact time as much as possible. Aim for a maximum of 4 seconds. Less is good! If you have more than one diode to solder it is good practice to do one leg on each then go back for the second. This way the diode has time to cool between each leg being soldered.

LED’s

These perform like other diodes except that the glow when a current passes through them.

Their position on the PCB will be shown like this



The LED will also have a flat side.

Generally, the rule would be to match the two and most of the time that is what you will do. However, to save you de soldering your LED’s I recommend testing them with a multi meter (set to diode test mode) to be 100% sure you are connecting the positive side where it needs to be. On the example above the positive feed goes into the round side.

I have come across LED’s orientated opposite to what I expected previously, after installation… so this quick test is sensible and is one less thing to debug later on. Check twice…solder once.

In this kit you have two LED’s. One that emits white light, this is used within the main circuit. The second LED is coloured and is the LED used to indicate when the pedal is turned on.

Transistors

The body of the transistor is usually a small black ‘can’ with a flat on one side.

The shape on the PCB corresponds to the ‘can’

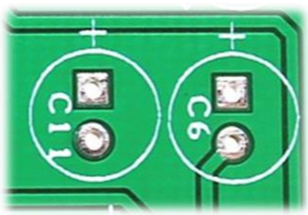
The same warning about heat when soldering diodes also applies to transistors. Take your time not to overheat.

Q2 and 3 are AC128 germaniums. They have a metal can, the emitter is indicated with a small metal tag. This is also drawn onto the PCB silkscreen.

If you wish, you can substitute these for silicon transistors.

Capacitors

Capacitors come in many shapes and sizes but fall into two basic camps. Polarised or non polarised. Capacitors can store energy. For a really good explanation of how they work try this video starting at 4:25.



We will start with the polarised as they are the simplest. The most common that you will see are 'electrolytic' capacitors. These look like a small beer can with two protruding wires, usually from the same end, sometimes one from each end. On the outside of the can will be the value and the max voltage. Polarised capacitors can only be inserted into the board in one direction. The PCB will have a + next to the positive terminal.

On the capacitor you will see that one side of the can is marked with a stripe usually containing negative symbols. The wire nearest to this is the negative wire. You will also notice that the positive wire is longer.

Now onto non polarised capacitors. The good news is that just like resistors you can insert them into the PCB either way round.

Now the bad news. they come in a variety of shapes and sizes and although there are supposed to be standards for markings they do vary. So, what do they look like? This kit predominantly contains box caps, like this one.

Now, the marking. Capacitance is measured in Farads (f). 1f is a huge capacitor. Going down in order there are Millifarads, mf (rarely used) 1 mf is 1/1000th of 1f. Microfarads uf 1/1000th of 1mf. Nanofarads nf 1/1000th of 1uf and Picofarads 1/1000th of 1nf.

Another way to put it is that there are 1000pf in 1nf and 1000nf in 1uf.

Polyester box capacitors with values of 1nf but less than 100nf will have the value written on the top, either in a very obvious way, or a code.

The capacitor shown above has a value of 100nf. A 1uf capacitor would begin with 1***, value of 1000, so with the additional decimal point, .1k is 100nf. The K is the tolerance (10%) and 63 is the max voltage. Manufacturers do have other ways of indicating values though unfortunately.

Where things can get difficult is with poly boxes of 100nf and above. sometimes you will see a 100nf marked as 104xxxx ie the picofarad notation has been used for the value. Equally you may see .1xxxx ie 0.1uf (the x's denote the tolerance and voltage). It is still the same 100nf capacitor. The reason for this is that only 3 digits can be used for the value so 100n cannot be used and 100 would denote 100pf. Still with me? this continues for high values until we get to 1uf which will be marked as 1uxxxx.

Eventually these capacitor and resistor codes will start to sink in! Trust me.

So now it's time to get down to building your pedal!

We are hoping to go from this to this

The first thing you must do is identify your parts from the pack. I suggest you separate them into groups of like components, working on one smaller group at a time.

Start with the main PCB and set aside anything to do with the offboard section. You won't need to worry about that bit for a while yet.

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. Working in height order mount the rest of the parts to the PCB ensuring that you solder them in place well as you go.

I then add the pots either mounting them to the board or wiring them into place.

There's quite a bit going on with this kit, and there's no obvious test stage so you may want to take lots of photographs as you go in case you need to de bug.

Pots

As a rule, I design my PCBS to accept two styles of potentiometer.

Overunder can take 8 board mounted pots. I know, 8 pots in a 125B 😉

The pots will probably look like this, a right-angled potentiometer. The shaft could be splined or round.

Often if a kit is designed for these then you will receive them.

Occasionally you may receive a mixture of this style and the next. If so, you can use wires to attach the pot to the PCB. There are various reasons this may happen such as supply issues or it may also be a part of the design.

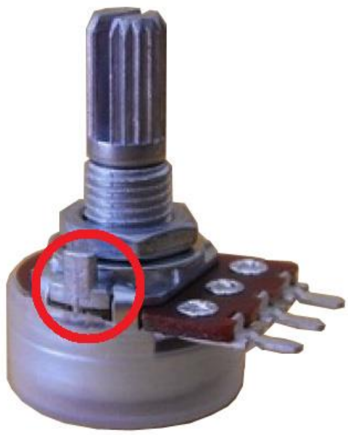
If you received any pots that look like this, then they will need to be wired to the PCB. You can achieve this by using normal wire, or, you can use header pins which is what I did when I prototyped my Jefferson.

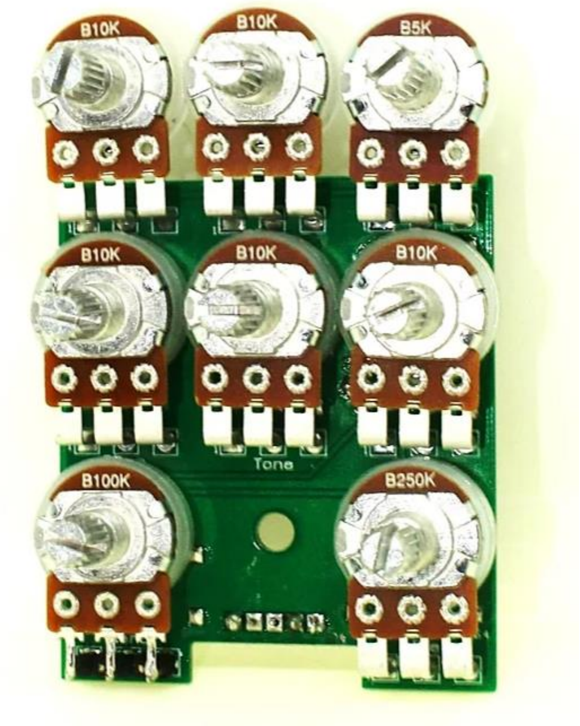
The pots have their values written on them

The pins are numbered, but that is not written on the part. Pin 1 is indicated on the PCB with either a square pad or a number 1, or both. The pin numbers are the same for both types of pot.

Just behind pin 1 and next to the shaft is a small metal tag.

Take a pair of pliers and snap this off. It comes away easily.



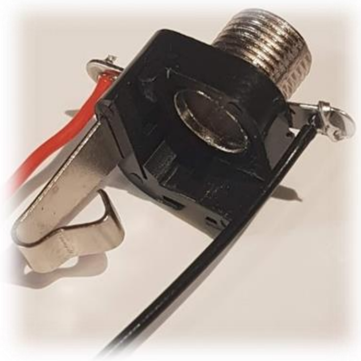


This is my prototype Overunder. I designed and built it during the Covid pandemic and I was low on pots, so I had to improvise with the 100KB

Note how I used header pins to create the right-angled joints for it.

Hooking up the Jacks

This kit comes with mono jacks, and because they are 'top mounted' they are also 'mini'. They have two connections, a positive AKA the tip connection. And a negative AKA the sleeve connection.



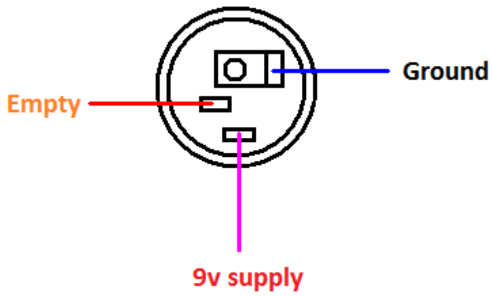
Using this picture as a reference the lug to the left with the red wire coming from it is the positive/tip wire.

The black wire is therefore the negative/sleeve wire. This is essentially a ground point.

Connect wires to both jacks like this and set them to one side. The wires need to be long enough to go from one end of your pedal to the other as they eventually connect to the footswitch.

Once you put them in the enclosure, they will ground it.

Power Socket



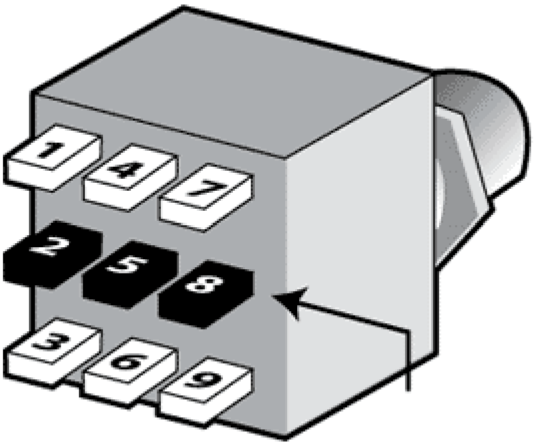
The black wire is the ground point.

The red wire is the 9V supply

At this stage, connect two wires to it and set it aside. There is no need to put anything in the enclosure just yet.

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.

The pins on the component will not be numbered. If you mount it with the lugs facing left to right as shown here. Pins 1,4,7 and 3, 6, 9 will do the same thing.

There are various ways to connect a switch to a circuit. Most people think you get a 'true bypass' switch. It's just a method of wiring. A switch is merely a switch.

I have designed a PCB specifically for the footswitch and whilst over time the layout of the switch may change slightly, the mechanics of it do not.

The two fit together like this. Take some time to study the footswitch and PCB.

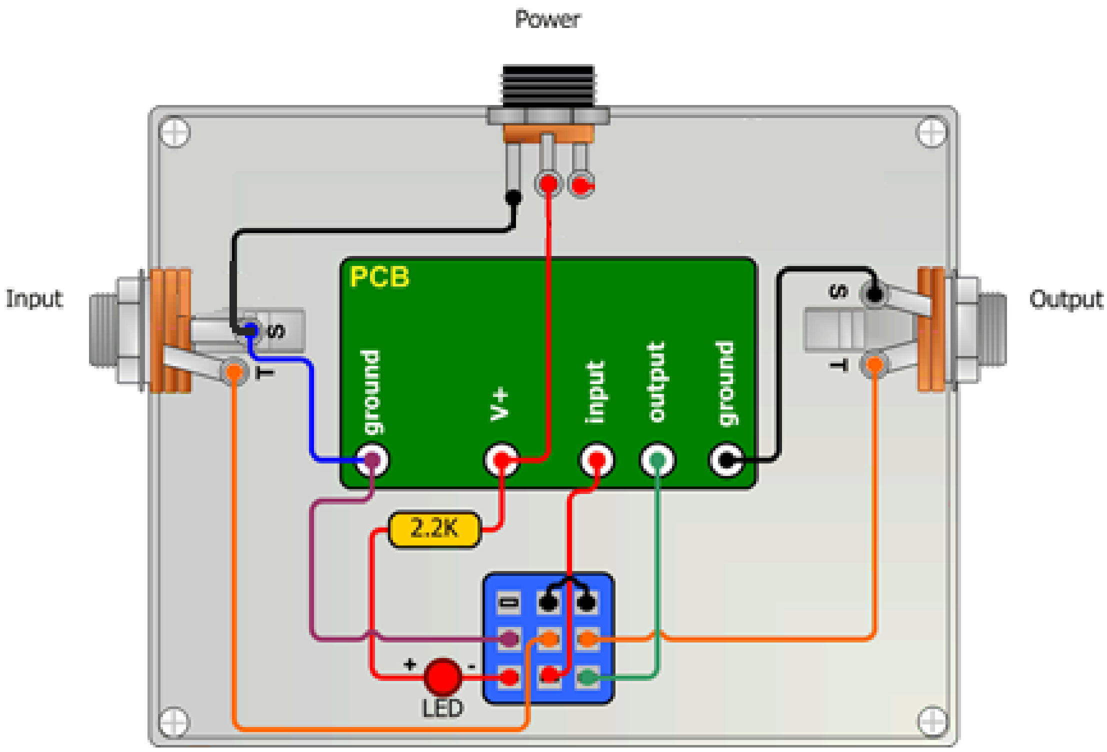
NOTE: the lugs run from left to right as I said above.

The PCB has two sides, it must go on like this. The other side looks quite different and does not say 'jeds peds' on it.

This PCB will sit above the main PCB allowing you to use the socket and SIL connector and I will explain that shortly.

Once you are satisfied it is on the right way then it must be soldered into place, but don't do it yet as I want to explain a second method of switch wiring that you may decide suits you better. (if you have already decided that you want to use the 3PDT PCB, then follow this link)

The other way you could hook up the footswitch would involve manual wiring. This is a method that can be used on any pedal. The following image gives a good visual representation of how to hook up the entire pedal, albeit in a different enclosure.



To compare this to the earlier 3PDT image, 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 lugs the correct way!

There are loads of other ways to wire up a foot switch, but I use this one, I always have, and it's never let me down.

On this picture the CLR is shown as 2k2. We supply 4k7 but you can use any value up to around 10k.

all your ground points must connect somehow, there are usually 5.

- The PCB has a ground plane and a couple of pads marked G.
- The power supply has a ground lug, you should have a black wire on it from earlier.
- Both jacks have a sleeve which needs to connect to ground.
- The footswitch has a ground lug.

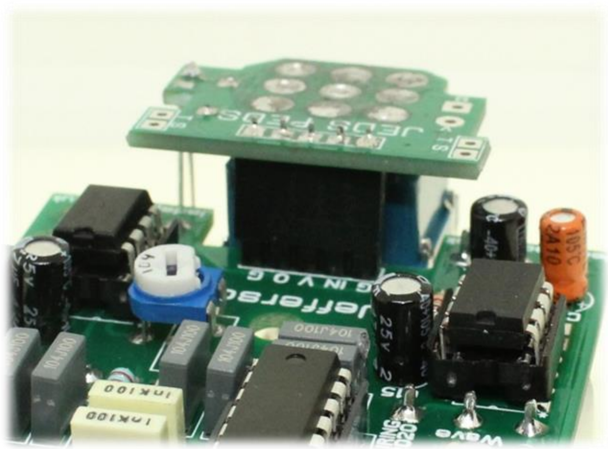
The same goes for all V pads. V is the direct power feed. On Jefferson, the main board has 2 'V' spots, one at the very top pretty much underneath where the power socket will sit and another at the very bottom next to the 3PDT. If you are manually wiring you can connect the supply at the top of the PCB, then use the lower V pad as the feed for your CLR.

If you want to use this method, and wire it all manually, then you can start to box up your circuit as its easier and neater to finish this off inside the enclosure so you can snip your wires to length. Take your time and think each stage through methodically.

HOWEVER,

This kit has been designed to have a simple connection between the 3PDT footswitch PCB and the main board. It uses a 5-pin socket and connector. This minimises offboard wiring to a few wires. (The design is identical to that used in Jefferson.)

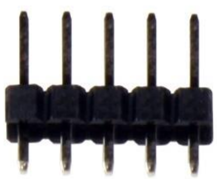
It looks long winded, but it really isn't.



This was part of the design for this kit and to ensure a neat build I recommend it. It's your call though. It does take a little experimenting to get the heights right so do not solder anything until you are 100% sure everything is correct.

The 5 pin connector needs to be soldered into place on the main PCB.

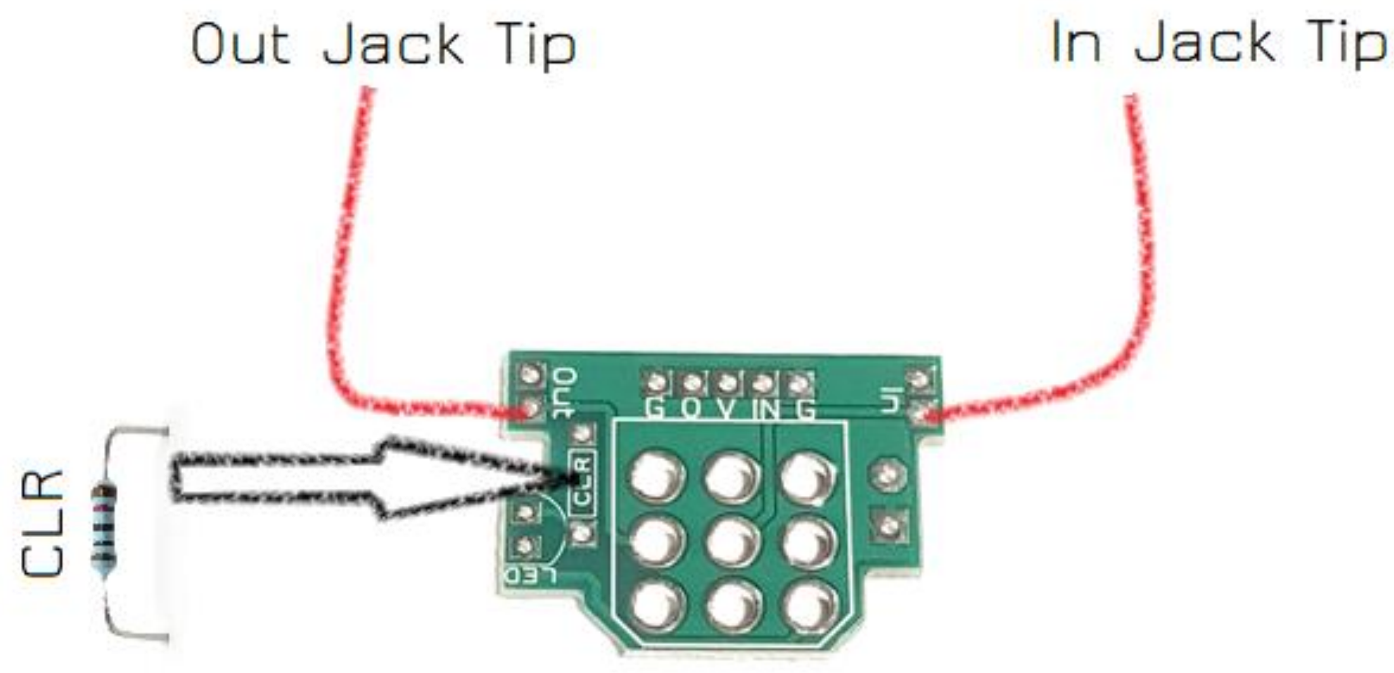
It has a long side, and a short side. The long side should stick and the short side soldered in.



So far so good? Hope so 😊

The 5 PIN socket Is soldered to the foot switch PCB.

The CLR (usually 4k7) needs to be soldered in place on the 3PDT board. Its clearly marked where.

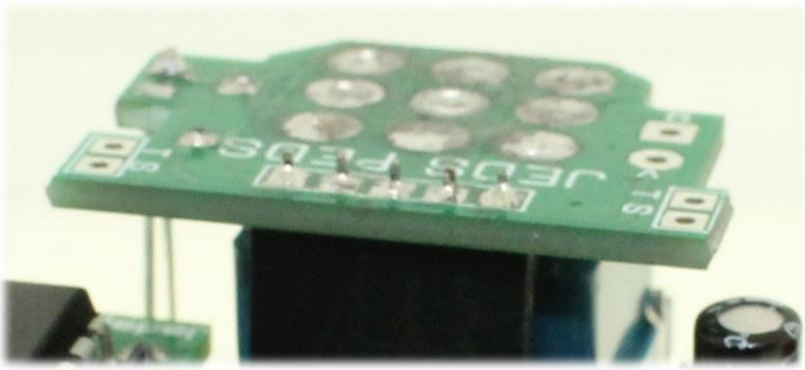


Its now time to set the height of the footswitch. Remove the nut and two washers from the 3PDT, leaving one nut at around halfway up the thread.

Put the main PCB into the enclosure and fix it in place with a couple of the pot nuts and washers.

Now put the footswitch in its hole and add its external nut. Just finger tighten it for now.

Then take the footswitch PCB and put it over the lugs of the footswitch. You are hoping to get the right balance of it closing onto the connector, whilst leaving enough lug to solder the board too. If you get it right first time, then you are a better guess than I was! As you can see from the next two images the foot switch will eventually sit quite high from the board, with the lugs almost flush to the board.

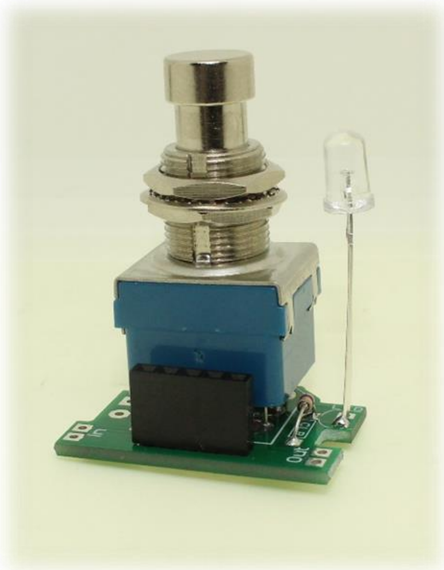


Once you are certain you have it at the most appropriate height, you need to put your LED into its spot on the foot switch board and set that.

Remember earlier when I said to test the LED with a multi meter, and you identified its positive leg, well that leg goes here.

Again, push the foot switch pcb into place, ensuring that the connection is made with the socket, and the lugs and this time (using your third hand) poke the led through its hole in the case. (It should be long enough but if it happens not to be it will have to be wired)

When you are happy everything is good to go, solder it together. Remember I said check twice solder once earlier. I checked mine at least 5 times… Just be sure before you solder.

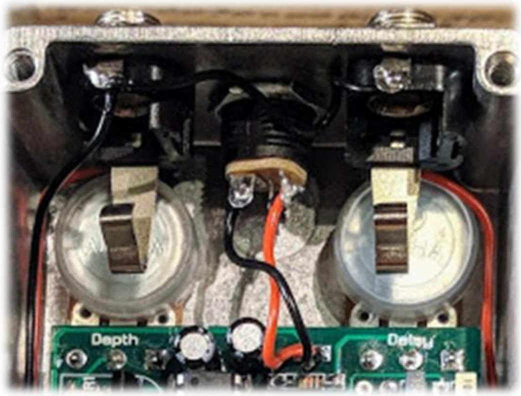


You have just constructed this inside the box.

Cute.

Mount the power socket into place and solder both wires in position at the power connection just near to them.

They are marked on the board.

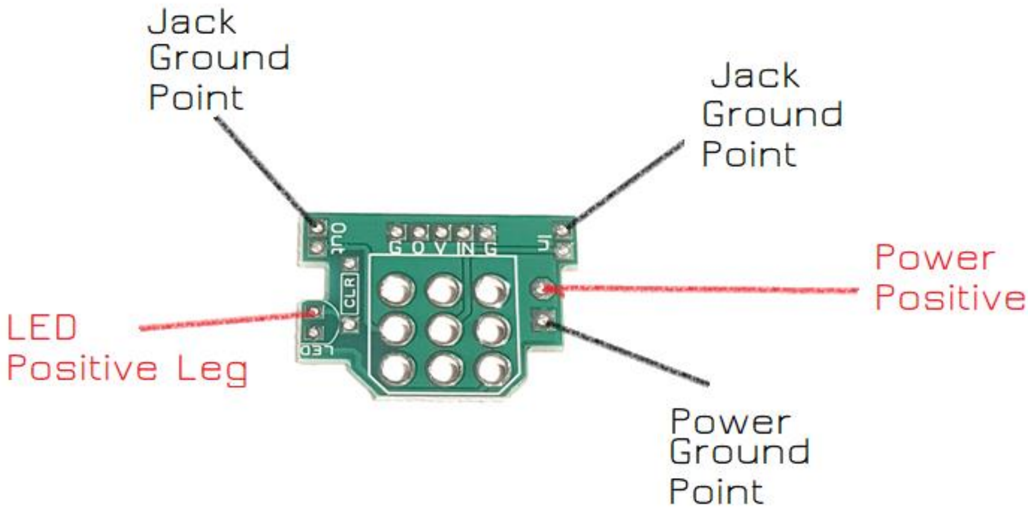


Now add the jacks into to the case. Face the Tip connection to the rear of the pots, like so. The ground lugs may need bending over slightly (*ooer missus*)

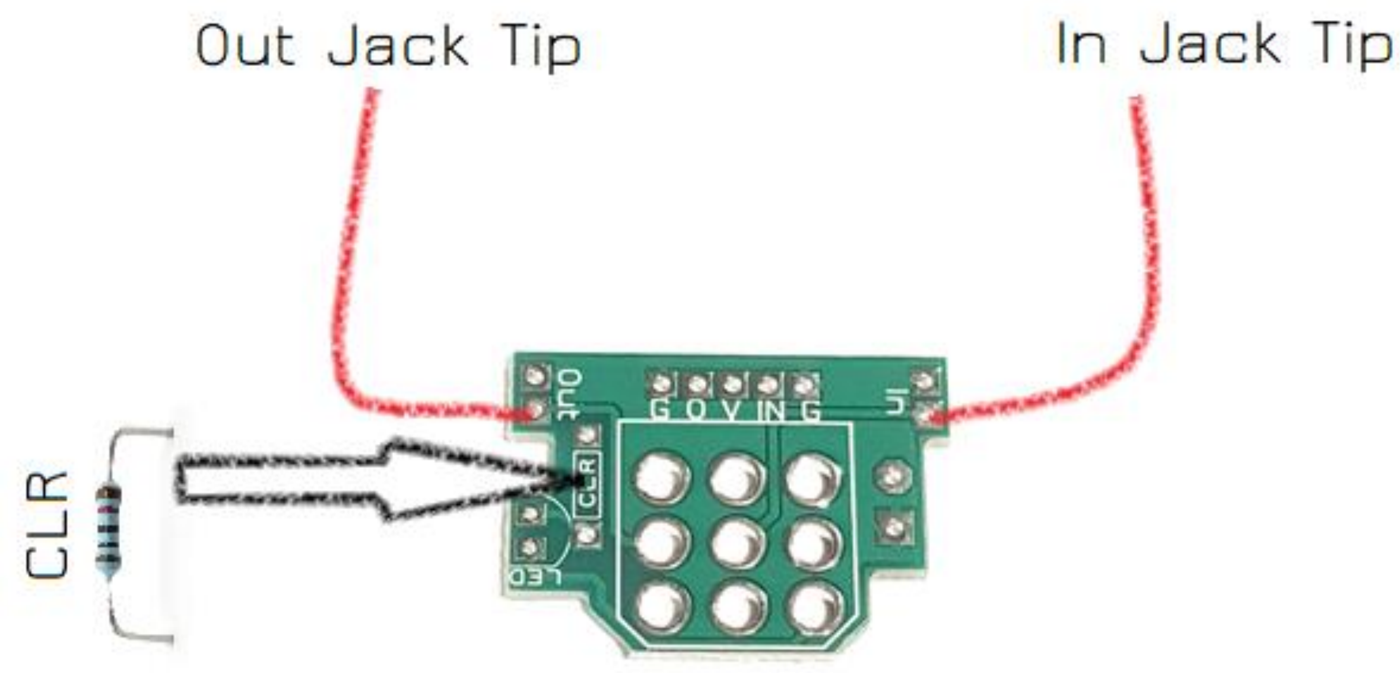
It does not matter if the ground lugs touch the case, as they are already in contact with the case, but they may prevent the lid closing if they get in the way

You can't let the tip lugs touch the case though.

As you can see, I ran a wire from one jack ground lug to the other and then ran one black wire down to the footswitch to complete my 'ground net'.



Now connect the red tip wires to the in and out pads marked T on the footswitch pcb.



And, finally connect the black wire (ground) to the pad marked S. Push the 3PDT back into place, ensuring the connection is made and he LED sits properly in its hole.

Tighten the external switch nut.

And have a cuppa.

Dare I say it, but it is time to test it all. Hopefully, it's all alive and kicking. But if it isn't, and you can't figure out why, then the place to head is our [Facebook forum]() and start a thread.

Good luck and happy soldering

Love Jed x


Jeds
Peds
Overunder
Vol
Stab
Drive
Gate
Tone
Comp
Pre
Blend
G IN V OU G
JEDS PEDS