

General Build Guide

Your first stop
for build info



CONTENTS

03 What's this all about?

04 THE BASICS

05 Introduction

06 Tools

07 Preparation

08 Components

09 Resistors

10 Resistor Colour Codes

11 Capacitors

12 Capacitor Codes

13 Diodes

14 Transistors

15 Integrated Circuits (ICs)

16 Potentiometers

17 Trimmers (Presets)

18 Toggle Switches

19 Rotary Switches

20 LEDs

21 Jack Sockets

22 DC Sockets

23 Footswitches

25 GENERAL BUILD GUIDELINES

26 Introduction

27 Parts and layout conventions

28 Holding parts in place

29 Placing pots and switches

30 Ribbon connectors

31 TESTING

32 Why test?

33 Simple test rig

34 Test wiring

35 FINAL ASSEMBLY

36 Assembly order

39 LED

40 BYPASS WIRING

41 Introduction

42 3PDT - Original

44 3PDT - Compact

46 3PDT - Top LED

48 DPDT - OptoPuss

51 Other variations

52 Top-mounted DC socket

53 Top-mounted DC socket and jacks

54 Positive ground wiring

**These contents are clickable
- no need to scroll around.**

**Click the link in the top left
of any page to return here.**

WHAT'S THIS ALL ABOUT?

We'll give you some basic information about what all those bits and pieces are in front of you, the best way to populate the PCBs, and how to do the final assembly.

Hopefully you'll have a lovely working pedal at the end of it.

There are four distinct sections:

- **The Basics**

This was originally a separate document altogether, but no harm having it here. You can easily jump past this if you know what's what. This will give an outline of the different components, including connectors and switches.

- **General build guidelines**

Directions on the best way to approach getting all those parts soldered onto the PCB.

- **Testing**

In almost every case it's best to test the main circuit board before connecting up the offboard elements.

- **Bypass and final assembly**

Instructions for the different bypass methods we offer and how to connect everything together once the circuit is complete.

Some of the above - particularly the last two sections - may not be relevant to your kit. In these cases the individual documents for those kits will be more comprehensive than those that follow these guidelines. You won't be left hanging.

Some of the images used in this document are from the excellent Pedal Builders Vector Pack from On the Road Effects.



THE BASICS

What's that stripe? Why does that hurt?

What's in this section?

The basic information you'll need to recognise a component, read its value and find where to put it on a PCB.

What is it not?

A technical document or an indepth explanation of the ins, outs, whys and wherefores of all things electronic. There are many online resources which will give you as much information as you need about components, schematics, electricity, soldering techniques etc. They'll explain it far better than we could, so go out and learn if you want to do more than put your nice new kit together.

There's a lot of text...

Sorry, but that's entirely necessary. You'll only need to read it through once, maybe twice.

**Hopefully by the end of it
you'll have gone from a
bag of bits to a finished pedal.**



TOOLS

There's a bare minimum you should have in your arsenal if you're going to get through this. Sure, you can probably get away with trying to clip component legs with kitchen scissors, but you're going to get sick of it.

Soldering Iron / Solder

You want something with a fine tip, not a massive plumber's spade tip. Soldering stations are easier to work with, but a simple iron with an in-line power cord will be fine, though you will need a stand with cleaning sponge. Then ditch the sponge.

Get some brass wool tip cleaner and stick it where the sponge used to be. Fine solder is best for small work like this - 0.7-1mm.

Wire Strippers / Cutters

Separate or combined. Stripping wires with a Stanley knife is tedious and gives you grey hair much sooner than is necessary.

Pliers

Fine needle-nose will serve you well.

Desolder Pump and Solder Wick

You WILL put something in the wrong place and have to remove it. These will help. Fluxed wick is well worth the extra cost.

Helping Hands

Moveable arms with crocodile clips to hold your work. Handy. There are loads to choose from, so go with what your budget allows.

A NOTE ON STATIC

Some components are very sensitive to static, so you should take precautions to ensure you aren't going to damage them. We wouldn't expect someone starting out to invest in specialist items to do this, so please do a search on '**ESD protection**'. It can be as simple as touching something metal before handling items.



Heat Sinks

Anything metal that will grip onto something on its own. Crocodile clips will do it. Self-closing tweezers are great.

Multi Meter

The best £20 you'll ever spend. Again, budget-dependent, but even the cheaper end of things will do most of what you need.

Screwdrivers / Spanners / Hex Keys

You know, just general tools. A set of precision screwdrivers will serve you well. There are a few spanner sizes that you'll repeatedly use when assembling - 14mm, 12mm, 10mm for sure. You could take the really sensible route and get a set of [Pepers' Pedals Rocket Sockets](#) - a specially designed tightening tool set that won't scratch your enclosure.



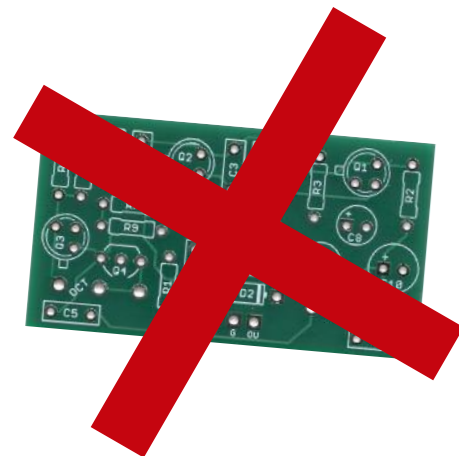
PREPARE THYSELF...

Soldering your first PCB

Don't.

If you haven't soldered before, please do a bit of practice first.

Get hold of some strip board (also known as vero board), watch some How-To videos and practice on that.



Lesson 1

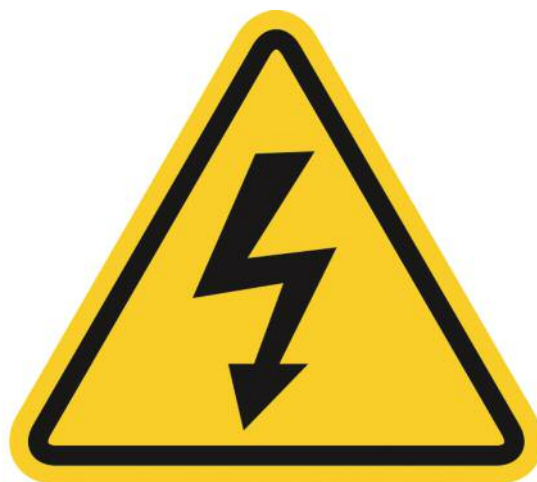
Soldering irons are hot. Solder gets hot. Everything a soldering iron touches gets hot. Your hands don't like it, and neither do the components you'll be soldering.

Sure, some will complain less than others, but the basic rule is - only apply heat for as long as necessary.

Learn which components are more resilient than others, and take the right precautions where needed.

Lesson 2

Electricity kills. If you're reading this you probably don't know nearly enough to go anywhere near mains power. Stick to batteries and 9V power supplies, and then only use the latter when you've tested your circuit with the former.



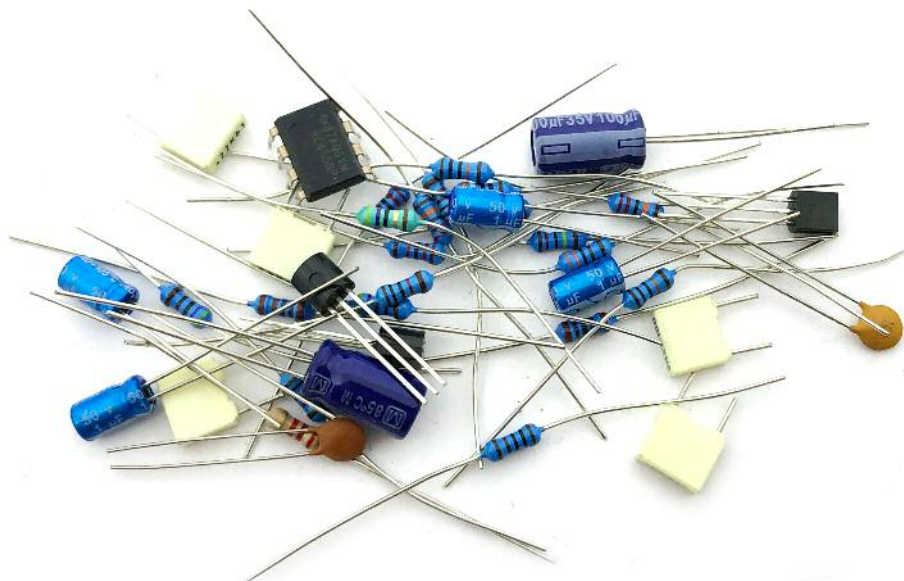
OK, I'm ready

Cool. Now you need to learn how to recognise what that bewildering pile of components is and where they go. Read on...

COMPONENTS

You won't get an explanation of what components actually do in this document. Again, there are many resources that will do that far better than we could.

This document deals entirely with through-hole components. That is, components with bits sticking out that will go through holes on a PCB. Surface mount is a whole different ball game.



Hopefully the following pages will have you recognising components with ease.

Seriously - invest in **Electronics for Dummies** and read it cover to cover. Twice. Then go back and read the bits you weren't sure of for a third time.

You don't need to know how electrons move to build a fuzz box, but it sure is fascinating.

Recognising the parts on your PCB



RESISTOR



DIODE



DIODE



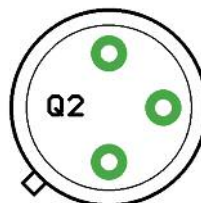
BOX OR CERAMIC CAPACITOR



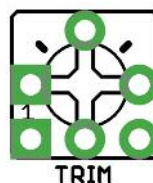
ELECTROLYTIC CAPACITOR



TRANSISTOR



TRANSISTOR



TRIMMER



POT

RESISTORS

Heat tolerance: **HIGH**

There are lots of different looking ones around, but for the most part you'll deal with two kinds that look very similar.

Most resistors supplied with our kits are rated at **0.25W** (quarter watt) which is more than adequate for the power they need to handle.

They'll be made of either **Carbon Film** (usually beige/yellowish case) or **Metal Film** (blue case). They do the same job, they're just made of different stuff and have different tolerances. In most kits these parts will be around 6mm in length. Some kits have smaller 3mm resistors which are fiddly to handle but do the same job in a tighter space..

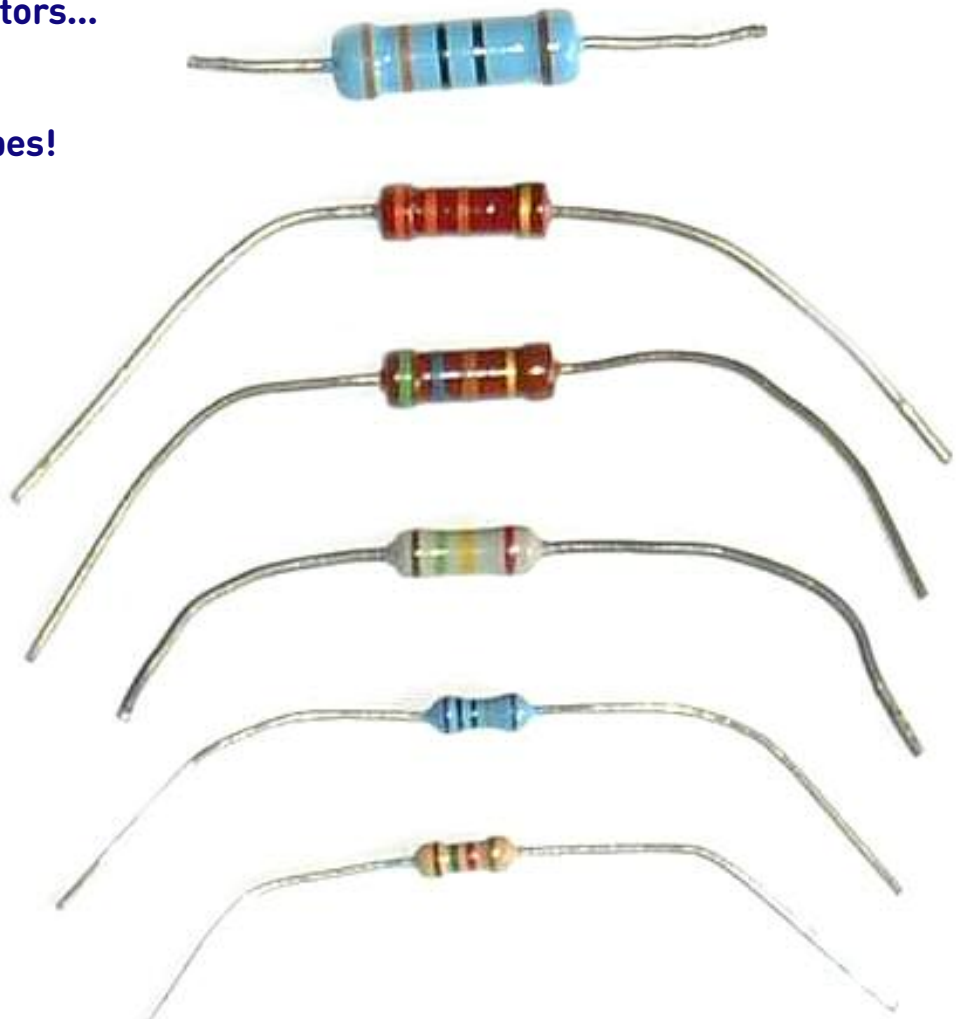
Carbon film resistors normally have quite a high tolerance - around 5%, meaning a 100K resistor may measure anywhere between 95-105K. This is fine in a lot of cases, but sometimes something more precise is required.

Metal Film resistors are normally 1% tolerance, even 0.1% if you want to spend a lot of money, but that kind of precision simply isn't required here.

So, you recognise resistors...

Now, which is which?

They're just full of stripes!



RESISTOR COLOUR CODES

You can learn how to read these, or just Google '[Resistor Code Calculator](#)', select your colours and it'll show you the value. Or you can set your newly purchased multimeter to Resistance and just measure the value by placing your probes on each leg. Or learn how to read them - here you go:

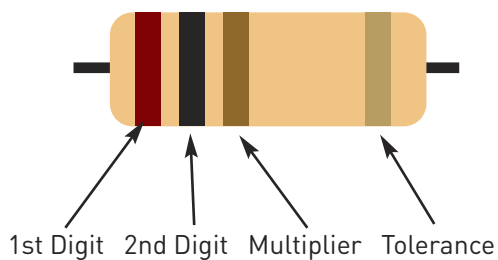
Resistors are measured in **ohms**, and the codes are shown as multiples of 1 ohm.

Each stripe represents a number:

0	BLACK	5	GREEN
1	BROWN	6	BLUE
2	RED	7	VIOLET
3	ORANGE	8	GREY
4	YELLOW	9	WHITE

The **tolerance** band shows you the % margin of possible variation in the shown value. There are a few of these, but you'll mostly come across Silver (10%), Gold (5%), Brown (1%)

4-BAND



The first two bands of this resistor give you two digits. In this case 2 and 0.

The third band gives you the multiplier, which is the number of zeros to add after the first two digits, in this case 1.

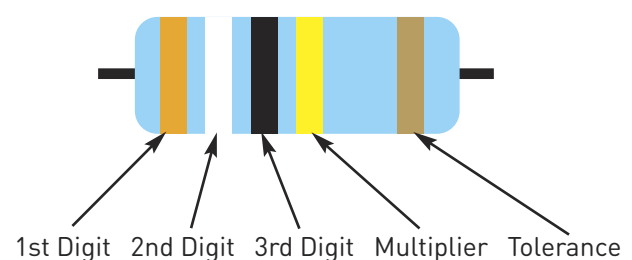
The resistor above measures 200 ohms.



This one is:

5, 6 and 2, which is 56 followed by 2 zeros, so 5600 ohms, or 5.6K (or 5K6).

5-BAND



It would be a bit much having to write out 75,000 ohm and such, so we don't:

1000 ohm = 1K (1 Kilohm)

1000K = 1M (1 Megaohm)

Resistor values under 1K will often be written as the value followed by R. Resistance, get it?

So you may see a 100 ohm resistor written as 100R.

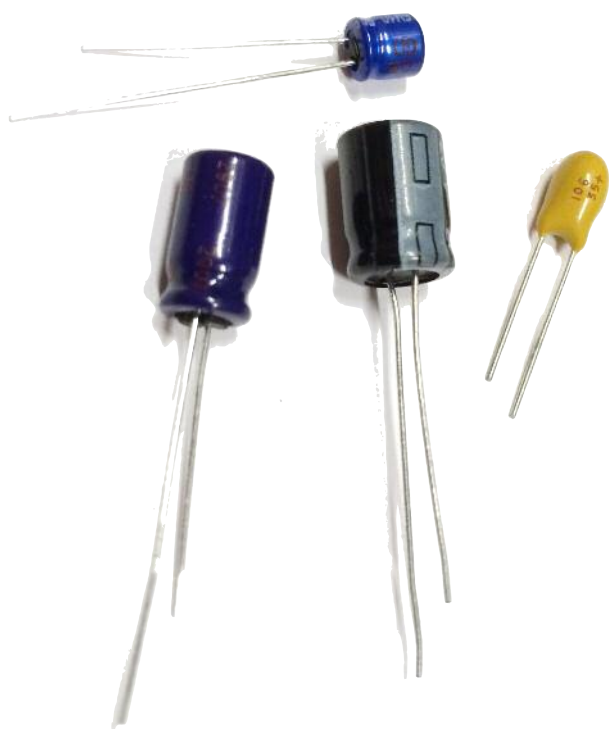
CAPACITORS

Heat tolerance: **MEDIUM**

There are many different shapes and sizes of capacitors, or 'caps' out there, made from lots of different materials, but they'll generally fit into two distinct categories.

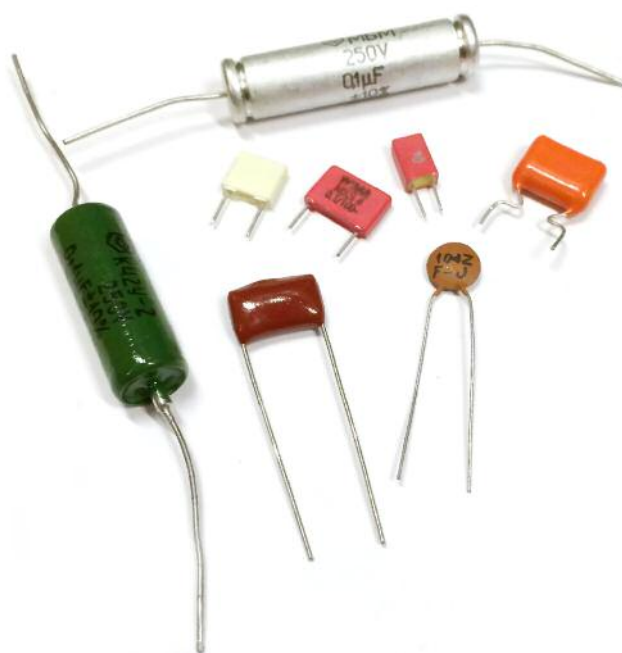
POLARISED means that the capacitor has a positive (+) side and a negative (-) side. In essence that means that in a circuit a polarised cap has to go in a certain way.

The positive leg will normally be longer than the negative. There'll usually also be some kind of marking on the case to indicate which is which. In the image below the white stripe on the middle cap indicates the negative lead. The small yellow cap is made of Tantalum, and the positive leg is indicated by a +.



NON-POLARISED means there isn't a positive or negative leg. Non-polarised caps can go into the circuit either way round.

The caps below are all different shapes, sizes and materials, but they're all non-polar and, believe it or not, all have exactly the same value. If placed in a circuit they'll all do exactly the same job, though different materials can have an effect on the sound of a circuit. Most of the non-polar caps that come with our kits are either polybox type (like the small cream one, though they can be many other colours), or ceramic like the small brown disc.



100n? 10u? 100p? What the....?

Capacitance is measured in Farads.

A Farad is huge. You will not be coming across any caps with whole Farad measurements until you start building fuzz boxes based on tesla coils and military laser systems. We'll be dealing with tiny fractions of Farads - microfarads (uf or, if you want to be pedantic, μf), nanofarads (nf) and picofarads (pf). Everything is based on multiples of 1000 so they aren't too hard to remember.

RECOGNISING CAPACITOR VALUES

Most caps you come across will have their value shown in uf, nf or pf.

1uf	=	1,000nf	=	1,000,000pf
0.1uf	=	100nf	=	100,000pf
0.01uf	=	10nf	=	10,000pf

See where that's going? Obviously there are circumstances where one unit of measurement is going to be more appropriate than another, and manufacturers tend to use their common sense. You wouldn't label a 1uf cap as 1,000,000pf now would you?

That being said, sometimes they don't make as much sense as you'd like.

POLY BOX CAPS

These will be marked in one of two ways, either in nf or uf. It will be easy to tell which. Different manufacturers follow different rules, but generally the following will apply.

Caps between 1n and 100n will normally be shown as the actual value followed by an n, so for instance, 10n.

Over 100n and it can get trickier. They may be shown with their actual nf value, but often they are marked in **uf**. In this case it will be shown as a decimal fraction, such as **.22**, and won't be followed by a letter.

In this case, the cap value is **0.22uf**, which, in nf, is **220n**. As there are 1000 nf in 1 uf, simply put three zeros after the uf value shown. 0.22 followed by three zeros is 220.

There are other markings on there which indicate tolerance and voltage rating. Don't worry about them. **.1J63** indicates a **100n** cap with **5%** tolerance (J) and rated to handle **63 volts**. Everything supplied in the kits is well within necessary tolerances and voltage ratings. Only concern yourself with the values.

Hopefully the whole x1000 thing is making a little sense by now..?

CERAMIC CAPS

These tend to be used for lower values (though not always), and are measured in pf. The markings take the format of three digits, and similar to resistors they are:

Digit Digit Multiplier

So a cap marked as **101** would be 10 with one trailing zero, which would be **100pf**.

223 would be 22 with three zeros - so 22,000pf, which is **22nf**.

We won't normally supply ceramic caps for values above 1nf.

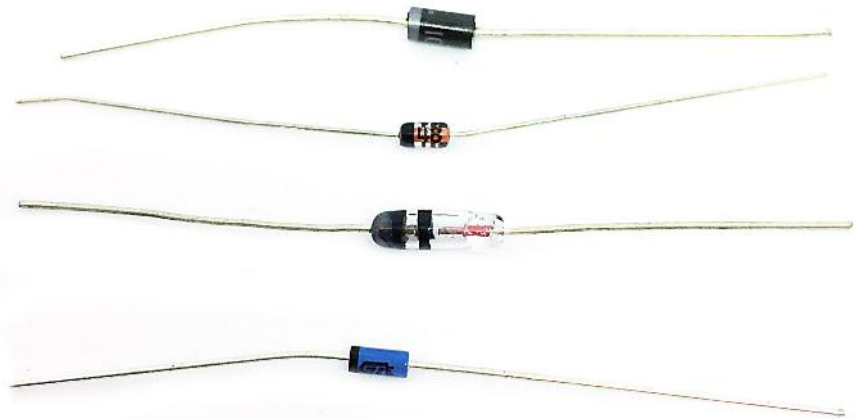
DIODES

Heat tolerance: **VERY LOW**

Once again, these vary greatly in size and appearance. This electronics thing isn't as straight forward as you were expecting, eh?

Though they look fairly different, diodes will normally follow one format - a barrel with a leg sticking out of each end, one leg being positive (anode), one negative (cathode).

In most cases you'll find the cathode end is indicated by one or more stripes, and in most cases this leg will go to a square pad on the PCB.



In MOST cases, but....

Say hello to our Russian friends. These follow their own rules, and will usually have stripes indicating the anode. Helpful, no?



As with all components you should treat diodes with care, especially when bending the legs. The points where they meet the glass casing are easily cracked, especially on the large DO-7 style cases of the 1N34A or Russian germaniums. The best way to approach them is to grab the leg right up against the body with your smallest needle-nosed pliers and bend the leg with your finger, letting the pliers take the strain away from the casing.

Diodes have many purposes in the circuits we're dealing with. This document is about identifying them, not psychoanalysing them.

One thing diodes do have in common is their dislike of excessive heat.

CAREFUL!

TRANSISTORS

Heat tolerance: **VERY LOW**

Do we really need to keep repeating the fact that most components come in many different forms? Thought not.

Transistors are perhaps the most varied though, and probably the parts that'll cause you the most heartache. They're very easy to fry with excessive heat, especially FETs and MOSFETs such as 2N5457 or BS170. Take your time with them.

They are also sensitive to static (especially FETs and MOSFETs), so make sure you're suitably discharged before handling them, and leave at least ten seconds between soldering each leg.

You'll mostly deal with two different classes of transistors in stomps.

BJT - Bi Junction Transistor

FET - Field Effect Transistor

You'll be dealing with three legs pretty much all the time. There are transistors with more, but we're not getting too exotic right now.



BJTs consist of:

Collector Base Emitter

On older style metal hat casings the emitter is normally identified by a small metal tag, and the collector leg will often be shorter than the others. On the most common, more modern TO-92 black plastic casings we aren't so lucky.

They can be arranged pretty much any which way, and it's a case of checking with an online datasheet or using a component tester to identify which is which.

Fortunately PCBs are usually designed with specific transistors in mind. Certainly any FuzzDog PCB you come across will show the transistor outline on the PCB exactly as it should be placed, or there'll be an explanation of any deviation in the build document.

FETs consist of:

Drain Gate Source

Again, looking at the case is no help whatsoever. Check the build document or datasheet for more info on each type as the pin-out - the order the legs are in - will differ between models.

Once again, Russian stuff plays by its own rules. Tester or datasheet required, though we'll always mark the legs on any we supply in kits.

There are others transistor types, such as the **Uni Junction Transistor** (UJT) found in the Repeat Percussion, but we could fill a whole document with them.

Sometimes you'll find an IC in a transistor-style case, i.e. voltage regulator 78L05.

INTEGRATED CIRCUITS

Heat tolerance: **VERY LOW**

...or ICs, or chips. Little (usually) black packages of goodness, containing many tiny components. One part to do the job of many. We recommend you always use a socket for these parts.

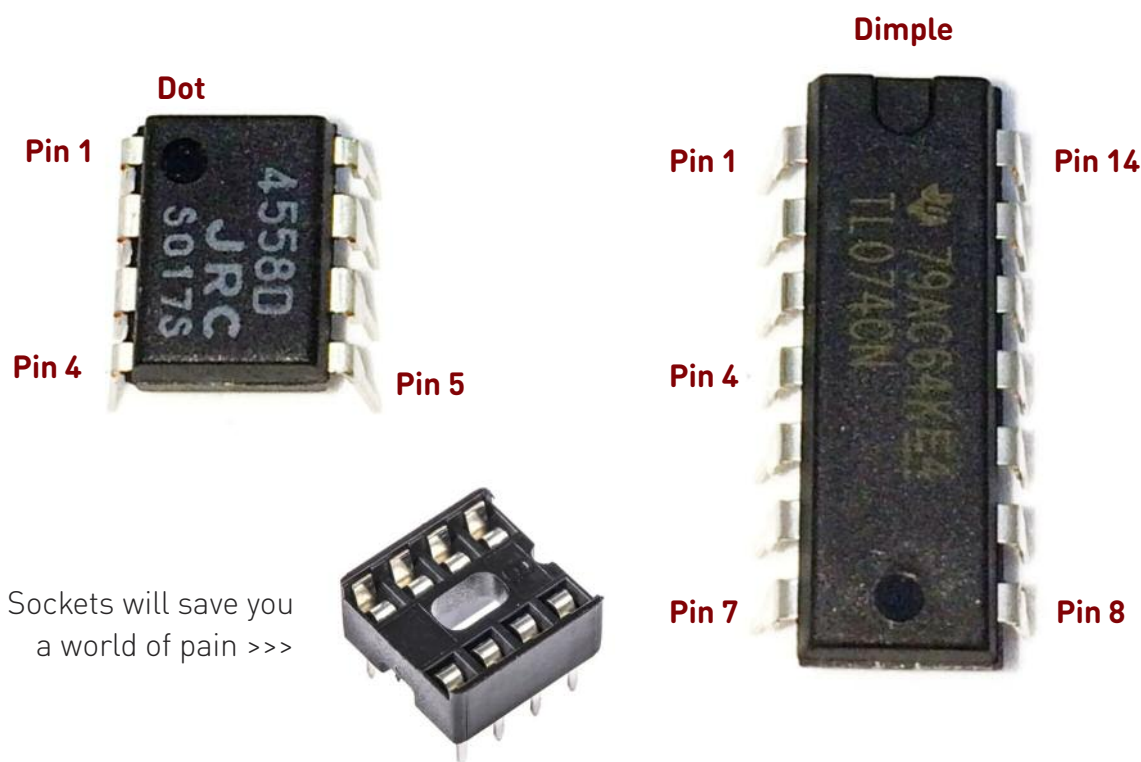
Once again, this is purely about identification and orientation/placement.

You'll normally come across a black case with two rows of legs, like a lovely robotic insect from the future. These are known as **Dual InLine (DIL)** packages. Occasionally you may have to deal with a single row of legs, which are **Single InLine packages (SIL)**, but these are less common.

You need to know which way round they go, and which leg is which. If you put them in the wrong way and power up, chances are it's goodnight IC. Hope you have a spare.

The silkscreen legend on our PCBs will have a notch at one end of the IC. This indicates the top. The pin to the left of that notch is always pin 1. Sometimes an IC won't have a notch, but will have a small dot printed on the top of the case instead. This indicates pin 1.

From pin 1 you count down the package, then back up the other side to find your leg numbers.



HEAT AND STATIC ALERT!

ICs don't like either. Handle with care, ground yourself before picking them up, and use sockets instead of soldering them direct to the PCB where possible.

....and **CONFUSION ALERT** - the 78L05 voltage regular mentioned on the last page is actually an IC in a transistor package. Don't worry about it.

POTENTIOMETERS

Heat tolerance: **MEDIUM**

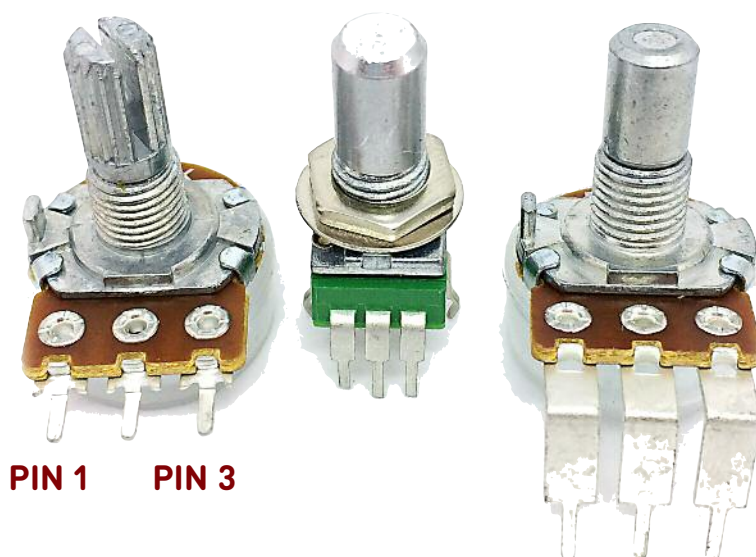
The fun part. These, along with switches, are likely your only point of interaction with a finished circuit, so get them right.

They'll either mount straight into the PCB or will be wired. The important thing to get right is the leg numbering.

They look like tough cookies, but guess what? Beneath that tough exterior they have delicate souls.

The variable resistance they produce is normally via a thin wafer of carbon that is easily damaged by too much heat on the pins. Careful now.

As well as a value, pots are marked with a code to indicate their taper. What's taper? It's the way in which the pot changes the resistance when turned.



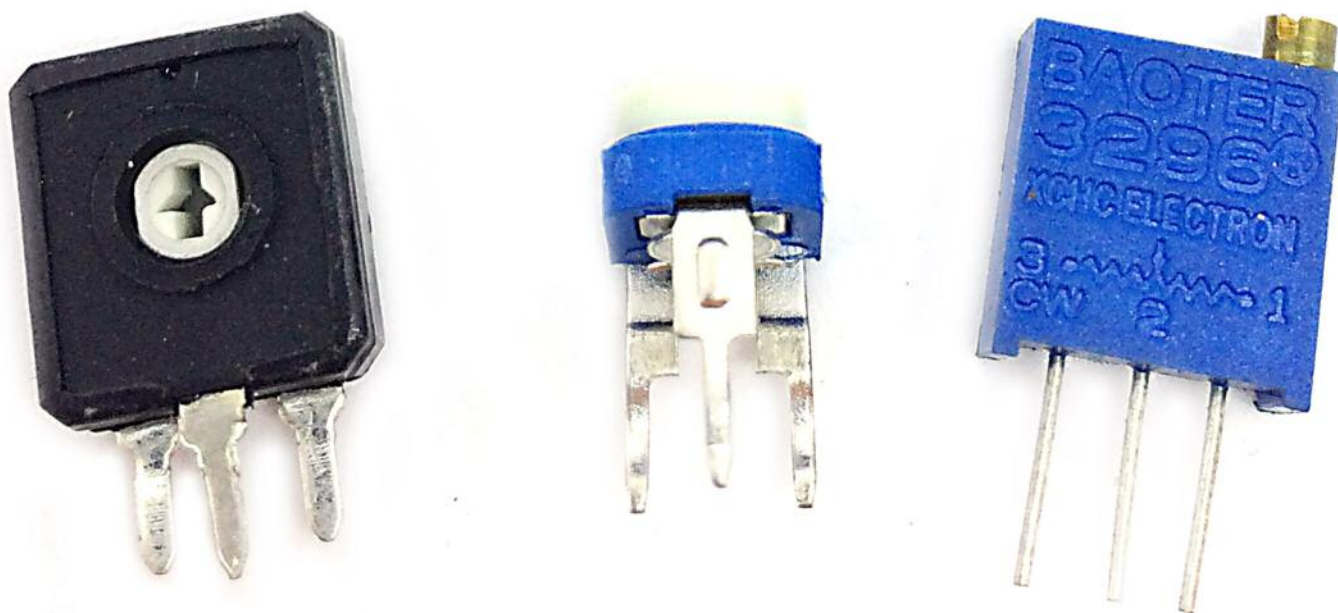
- A** = **Logarithmic** (also sometimes known as Audio)
The resistance changes slowly at the start of the turn from fully counter-clockwise, but gets increasingly quicker towards the end of the turn.
- B** = **Linear**
Change in resistance is the same throughout the turn.
- C** = **Anti-Logarithmic / Reverse Logarithmic**
Changes quickly at first but slows down towards the end of the turn.
- W** = A combination taper sometimes called '**Tone**' taper.
Not so common, but seen in Tube Screamer type circuits.
Kind of does its own thing.

Just to complicate things a little you may also come across DUAL-GANG pots. These have two rows of pins - so six in total, and are essentially two pots in one. They enable control of two different elements of a circuit with one knob. The two sets of resistances are almost always of the same value, i.e. a 100KB dual-gang. If you ever come across a circuit that requires a dual-gang with two different values, move on. That's a world of pain, and thankfully very rare.

TRIMMERS (or presets)

Heat tolerance: **MEDIUM**

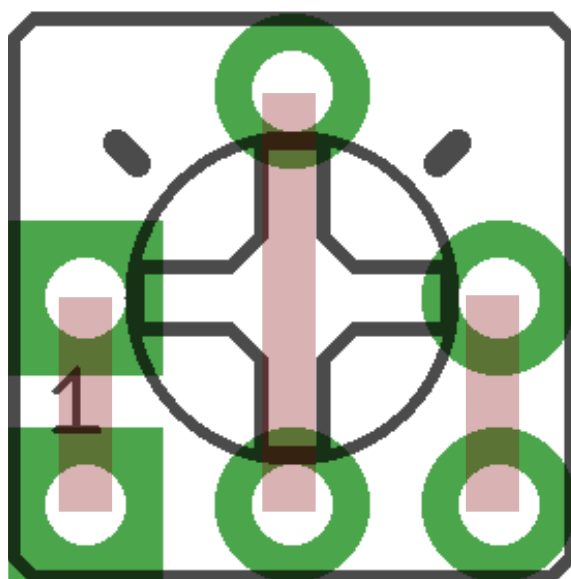
The potentiometer's shy cousin. These are normally used for set-and-forget adjustments such as transistor biasing. If you want to get really tweaky you can normally replace a trimmer with an external pot, but more often than not there's a good reason that control has been left inside.



These come in lots of shapes and sizes, as well as having different operational characteristics. Some, such as lefty and centre up there, cover their entire resistance range in one single turn. Others, such as righty, will require many turns - usually 13 or 25 - to get from one end to the other. Easy does it. In both cases it can be quite easy to force the turn too far and snap the mechanism, especially with the multiturns. Adjust these very slowly and delicately - you'll feel a slight resistance to the turn when they reach the end of the range. Don't push it.

On most boards we've produced since around 2018 the PCB footprint for trimmers has multiple pads to allow for different formats.

The pads are connected within the copper traces of the PCB as shown. As long as you have one leg of your trimmer in each column of pads you're good to go.



TOGGLE SWITCHES

Heat tolerance: **HIGH**

These come in lots of different shapes, sizes and configurations.

You shouldn't have to worry about that, as we'll provide whatever is required, or list them with enough info to help you source the correct unit.

You'll find them described by the number of poles and the type of throw they have. Say what?

OK, a switch with one column of lugs will be described as **single-pole**.

Two columns will be **double-pole**. Three columns... you get the idea.

This is followed by the throw - single or double. That's usually all we'll be dealing with, so no need to get overly complicated here.

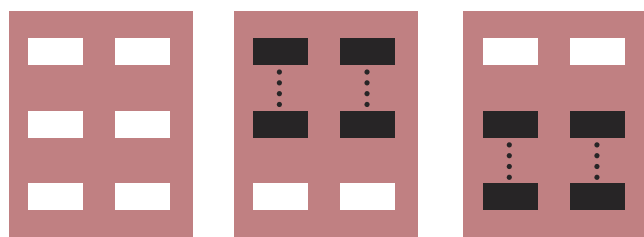
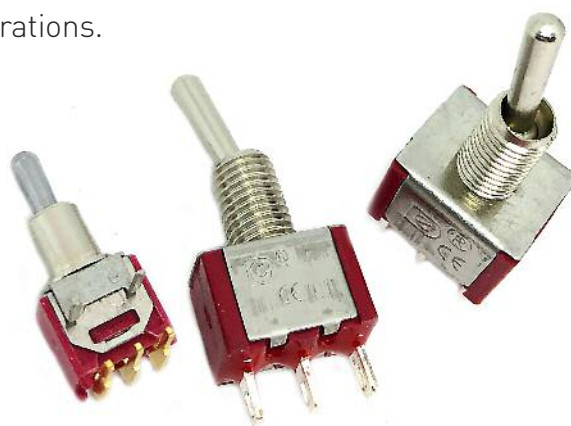
One more thing - the way they switch between those sets of poles. Maybe the switch has two positions, switching from one sets of poles (ON) to the other (ON), which would be described as **ON-ON**. If stops in the middle, where the centre pole connects to neither of the outer poles, so totally off. This would be **ON-OFF-ON**. Simple, right?

You'd think so. We also deal with another type - **ON-ON-ON** - whereby in the centre position two opposite sets of poles are connected. OK, brain freeze. It'll be easier with illustrations - these show the lugs on the bottom of the switch.

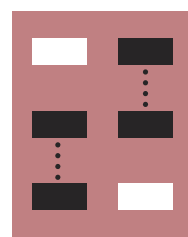
Below we have a **DPDT ON-ON** switch (double pole, double throw, two position).

The first simply shows the 6 poles - two sets of three. Centre pic shows which lugs are connected when the switch is in the down position - the connection will always be the opposite side to that which the switch is pointing in. The third pic shows the connections with the switch up. With an ON-ON switch the centre two lugs will always be connected to one or the other sets of outer poles. For an ON-OFF-ON, with the switch in the middle position the centre lugs aren't connected to either of the outer sets.

We'll touch on ON-ON-ON (far right pic). In the centre position the lugs are connected as shown. There are two types though, one connects as here, the other the opposite way. Phew!



Don't sweat the ON-ON-ON - where this is used the complicated bits are dealt with in the PCB design.



ROTARY SWITCHES

Heat tolerance: **MEDIUM**

Sometimes you need more than the two or three options offered by a simple toggle, no matter how many poles you cram on there. In this case we can turn to the rotary.

Once again, these come in lots of different configurations depending on the number of poles and throws required.

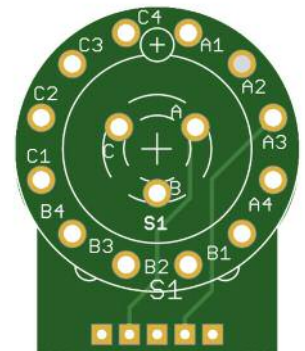
For instance, if you needed one part of your circuit to have twelve different switchable options, you could use a 1P12T (one pole, twelve throw) switch. Perhaps you want to change two things in the circuit, and have six different options of each - 2P6T. And so on, for 3P4T (three poles, four throws) and 4P3T (four poles, three throws).

Here's a 3P4T, which is what we mostly use. Notice the locator lug - where there is a space on one of our PCBs for one of these switches there's a circle with a cross inside to indicate the correct position for this.

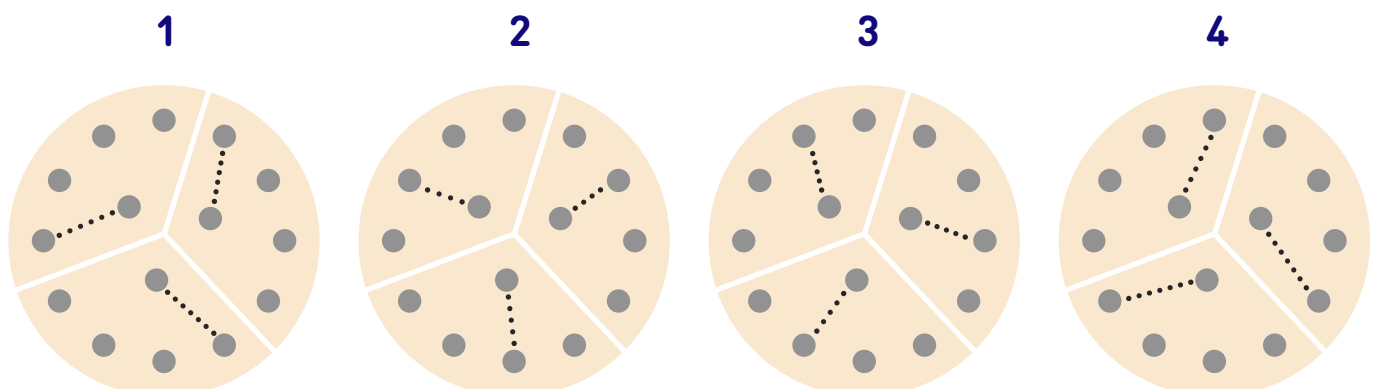
The washer under the nut has a lug on it. This can be located in one of the holes around the body of the switch to limit the number of positions it can be turned.

We may wish to switch three different elements of the circuit, but only want three different variations. We'd still need three poles, but you'd only use three of the four throws.

These often have long shafts, so will require some hacksaw action to shorten them.



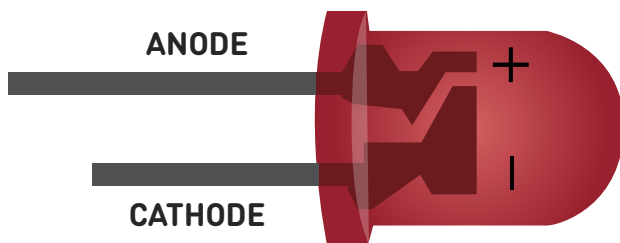
Here's a view from below of a 3PDT rotary. It shows the central three poles connecting to each of the corresponding outer poles as the switch is rotated through its four positions:



LEDs

Heat tolerance: **VERY LOW**

Shh. Be very quiet. These little fellas are incredibly delicate. And difficult to clearly photograph, so...



LEDs are diodes that light up when current flows through them. It's that simple.

The **ANODE** (+) leg will always be longer than the **CATHODE** (-) leg, so it's easy enough to identify which is which. Once those legs are snipped, you can recognise the cathode by a slightly flattened edge on the casing.

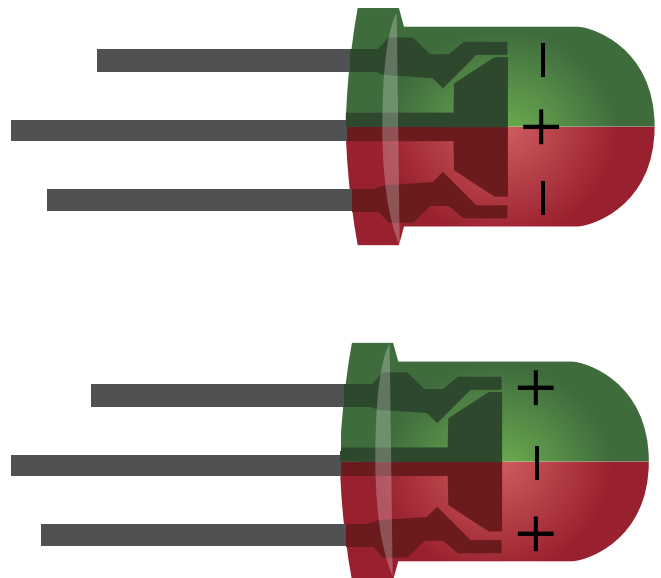
As well as being a handy indicator as to whether your pedal is on or off, they can be used to clip your signal, just like a regular diode. So, with that in mind, don't worry about seeing them on circuit boards but not poking out of the enclosure. They're busy inside.

You need to take extra care when soldering these fellas. A heat-sink of sorts is a must. There's no need to poke the legs all the way through the board before soldering - feel free to leave the body a good distance away from the surface of the PCB.

As with most other components, these come in many shapes and sizes. We normally supply 3mm for your on-off indicator, but others within kits may be 5mm.

The image to the left is a single colour LED. There are others, but you'll rarely come across them in our kits. Just for clarity, here's examples of others that we supply with a small amount of kits.

Bi-colour LED will display two different colours depending on which side of the device is powered. These come in two different types - **common anode** and **common cathode**.



Common anode - the positive supply is always applied to the single anode. The negative supply will be applied to one cathode or the other, lighting up whichever colour is powered. You can also supply both cathodes at once and have a combination of both colours, but why?

Common cathode is the same but the other way around, if that makes sense?

We highly recommend you avoid colour-cycling LEDs which will constantly change colour when powered. These can cause noise in your circuit.

JACKS

Heat tolerance: **HIGH**

The gateway in and out of your pedal. We deal almost entirely with 6.35mm (1/4"), and use three particular types across our range.

OPEN FRAME

The type used in most of our kits. We tend to stick with Neutrik as a manufacturer as the quality is second to none. Beware cheap imitations.

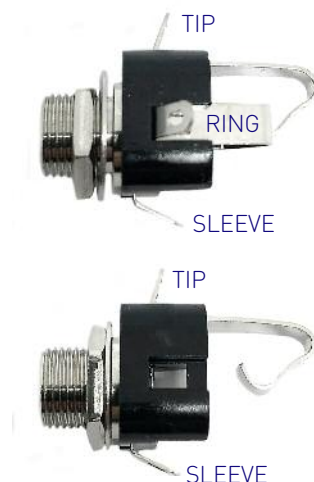
These come in two configurations: two-pole (mono) and three-pole (Tip Ring Sleeve - sometimes referred to as stereo).

You'll normally deal with mono unless you want battery provision in your pedal, in which case the TRS jack is used as a nifty way to disconnect the battery when the input jack is removed.



LUMBERG

Handy in a tight spot, and also great quality. Same deal as above regarding configs.



THROUGH-HOLE

There are lots of types of these, but we tend to stick to those shown. Again, available mono and TRS. Where these are used on our PCBs we normally include extra pads so TRS can be used even when mono will suffice. These jacks have two sets of pins for each connection, one set being switched. This switch is opened when the jack is inserted. Don't worry about it. Our PCBs have you covered.



DC SOCKETS

Heat tolerance: **HIGH**

You have the power!

Once again, three types across our kits.

All are 2.1mm barrel, normally wired with the tip negative.

BOSS-STYLE SWITCHED

Used on almost everything we supply.

These can be wired as a simple +/- or in conjunction with a TRS jack you can use them to include a battery in your pedal. There's a switch mechanism which disconnects the battery supply when a DC plug is inserted.



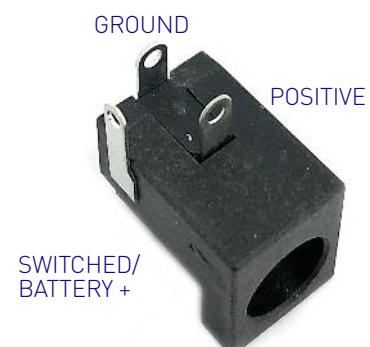
MINI

Used for tighter builds such as FuzzPups (hey - that's a whole different document), or for pedals with top-mounted DC and jacks. These have only two pins. The longer pin is normally for your positive connection unless otherwise stated in an individual build doc.



THROUGH-HOLE

A PCB-mounted jack used on a few of our kits, such as tester units or buffers. It's difficult to get a square hole drilled into an enclosure, so these will normally sit inside the box with a round hole offering access.



FOOTSWITCHES

Heat tolerance: **HIGH**

Very important, as in most cases this is your main point of contact with your pedal.

We currently deal with two different types of these, one of which has two variants.

By far the most common type used is **3PDT** (or three-pole, double throw). The actual mechanics of these should be fairly clear from the description of the toggle switches.

We've been at this a long time, and have tried these switches from many different manufacturers. We now have a standard 3PDT switch which we're entirely confident in. This is the blue one.

We also offer a premium 3PDT from **Gørva**. This has a softer click than the standard switch and is, in our opinion, worth the extra investment for a pedal you're going to be stomping on a lot.

Our **OptoPuss** bypass system uses a **DPDT** switch, along with some clever circuitry to provide the functionality of the 3PDT with one less pole. This has a much shorter throw - the distance the switch travels before engaging - than the two 3PDT. It still has a definite 'click', somewhere in between the other two.

Honestly, if you're reading this section of the document you're probably new to this. We'd recommend avoiding the OptoPuss bypass as the optocoupler used in the circuit is incredibly sensitive to heat, and very easily damaged.



A faint, light gray background image of a printed circuit board (PCB) layout. It shows various electronic components including resistors (labeled R15, R12, R19, R8, R9, R10), capacitors (labeled C8, C10, C6), and integrated circuits (labeled IC1B). There are also labels for ground (GND) and a component labeled HI. The layout includes traces connecting these components and some numerical labels like 1, 2, 3, 5, 6, 7, 9, 10, 11, 12, 13, 14, 15, 16, 17, 18, 19, 20, 21, 22, 23, 24, 25, 26, 27, 28, 29, 30, 31, 32, 33, 34, 35, 36, 37, 38, 39, 40, 41, 42, 43, 44, 45, 46, 47, 48, 49, 50, 51, 52, 53, 54, 55, 56, 57, 58, 59, 60, 61, 62, 63, 64, 65, 66, 67, 68, 69, 70, 71, 72, 73, 74, 75, 76, 77, 78, 79, 80, 81, 82, 83, 84, 85, 86, 87, 88, 89, 90, 91, 92, 93, 94, 95, 96, 97, 98, 99, 100.

Hopefully this spot of light reading will have set you on your way to populating your PCB.

It's by no means a comprehensive field guide for engineers, but as a part-spotter's guide it should see you right.

Here endeth **The Basics**



GENERAL BUILD GUIDELINES

What to do... what not to do.

This isn't paint by numbers

Sorry, no hand-holding here, just general tips on putting the bits in the right place and getting your pedal together.

- 1 Read your kit-specific build document from start to finish.
- 2 Separate out the parts, read the values and check against the bill of materials (BOM).
- 3 Contact us immediately if something is missing, incorrect, or you simply can't identify it.
- 4 Decide which order to assemble the parts - usually height is a good indicator. It's good practice to start with the smallest and least susceptible to heat, i.e. resistors. Here's our recommended order:

- Resistors
- Diodes (flat)
- IC sockets
- Film box caps
- Electrolytic caps
- Diodes (vertical)
- Transistors
- LEDs

Of course this list will totally depend on the individual kit, and you'll find your own way after a few builds.

- 5 TEST! See later in this doc.
- 6 Fix.

Repeat steps 5-6 until everything sounds good. It's entirely pointless boxing up a partially working circuit then trying to troubleshoot.

- 7 Embelish your enclosure. Of course you can do this first, but we sometimes hear from people who've spent forever making the box look sweet but just can't get the circuit working. What a waste of time and effort.
- 8 Box it up with the off-board bits in place.
- 9 Enjoy.

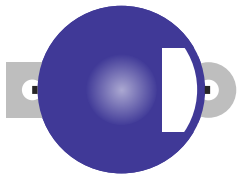
Putting parts on to (in to?) the PCB

Our PCBs have clear silkscreen part placement guides printed on them, including part numbers. Follow this and you can't really go wrong.

As mentioned on the previous page, READ THE KIT SPECIFIC BUILD DOCUMENT before you start. Maybe we've messed up and mislabelled something.

General Layout Conventions

Unless otherwise stated in the build doc, these are universal across our PCBs:



Electrolytic capacitors:

Long leg (anode) to square pad. Stripe indicates cathode.



Diodes/LEDs:

Striped leg (cathode) to square pad. Short leg to square pad for LEDs. The exception to this is with Russian germanium diodes - stripe = anode.

ICs:

Square pad indicates pin 1.

Making components fit

Resistors - you need to bend the legs, obviously.

Capacitors - most will drop straight into the holes in the space provided, though with ceramic caps the legs may need to splay slightly.

Transistors - most will need the two outer pins to splay slightly, some will be pre-formed to fit.

Diodes - bend those legs. Do this VERY carefully with large-bodied (DO-7) germanium diodes. The body will very easily break if you don't offer some strain relief. Grip the leg with needle-nosed pliers tight against the body and bend the leg with your finger, letting the pliers take the strain. Sometimes diodes need to be placed vertically - just bend one leg 180° to be parallel with the other.

ICs - the legs need to be eased inwards slightly to fit into their sockets.

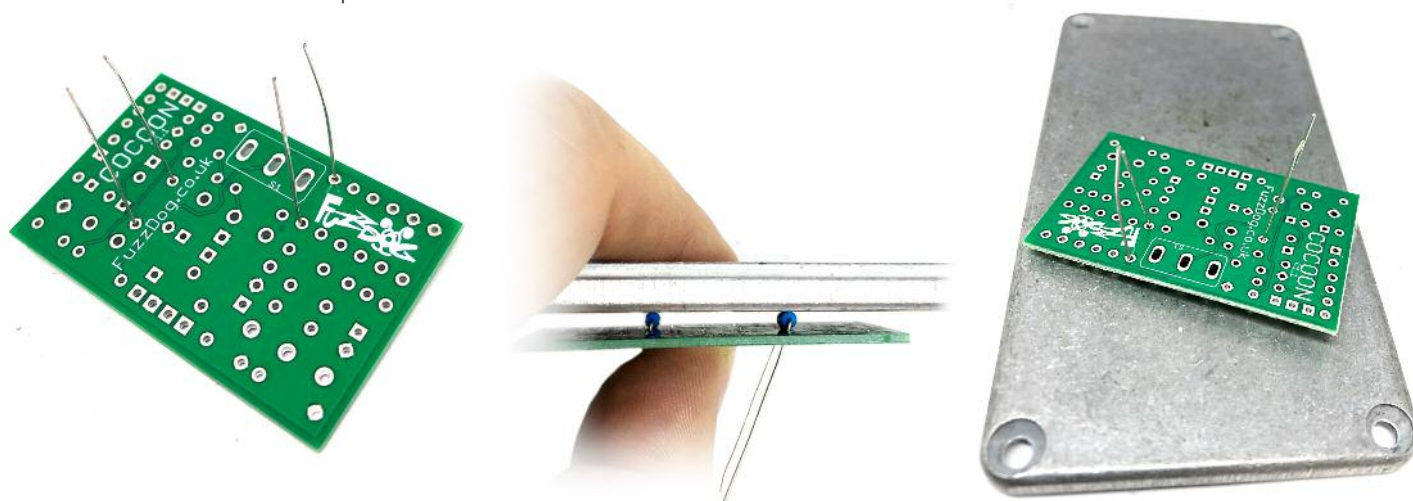
Holding parts in place

We use two main methods for this. The tip and balance, and the leg bend.

We use a combination of both. Where there are a bunch of parts of the same height - i.e. resistors, box caps etc - we use the first. Once you get onto electrolytic caps and transistors the latter is definitely more practical.

Tip and balance

With your PCB in your helping hands, silkscreen side up, place your group of parts (i.e. resistors). Leave the legs straight as shown left. Now take your enclosure lid, or any other hand flat, stiff object, and hold it onto the parts to keep them held in place. Now flip the whole thing over so the PCB is sitting on the lid. Boom! You can now solder everything in place, snip the legs and continue with the next batch of parts.



Leg bend

Place your components one at a time and bend the legs out slightly against the pad so they can't fall back through. Done. Why don't we just use this method for everything? We find the bent legs can get in the way more when making your way through with the soldering iron. You could of course just do a few parts at a time. Wish we'd thought of that...



Pots and toggle switches

Almost always best to do these last. In a lot of cases, once you solder the pots in place you'll have covered up a lot of the underside of the PCB, restricting access to the pads. You can bend the pots out of the way if you really need to, but you can only subject them to so much of that before the legs give up and snap.

Getting pots straight

Get the pot in place, solder one pin. Do a visual check to see that all three pins are all the way into the PCB and the pot is straight. If not, melt that one joint and push the pot firmly into the correct position. Once all three are snugly home you can solder in the remaining two.

Now, sometimes the pot pins will be hard to access once all the other parts are soldered in - see below.

You really don't want to be poking a hot soldering iron into that tight space where the bass and treble pot pins are, risking burning nearby components. Fortunately we're dealing with double sided PCBs, so you can solder the pots from either side of the board.

In this case it'd be best to place the two rear pots first, soldering on the same side as the pots with unhindered access. Then you can approach the front pair from either side.



Toggles

You shouldn't ever have to worry about accessing the pins for these - there'll always be plenty of space. Use the same method to get them straight as per the pots.

Single pin first, melt, adjust with some wiggling until you get it nice and straight, then commit the other pins.

Ribbon connectors

These aren't necessary - four lengths of wire will do the same job - but they do make a neat finish.

Some people don't like them as the joints where they meet the board can become brittle and snap with repeated bending, but the same could be said of any wire.

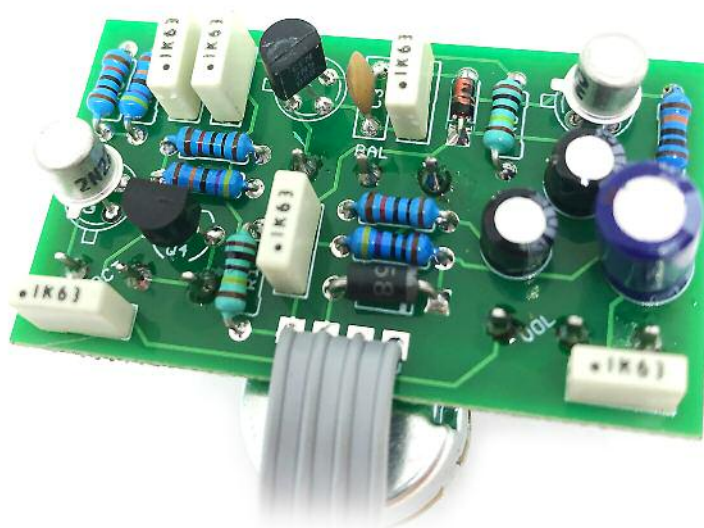
On the majority of our kits these are 4-way, i.e. there are four individual wires in the ribbon. Sometimes there are six or more, but individual kit documents will deal with that.

There's really only two things to watch out for with these.

- **Don't try to make sharp bends.** They will break internally. Keep your bends nicely curved.
- **Determine which side of the PCB they're going to go on before you break out the iron.**
In some cases the ribbon will have to attach to the main component side of the board, in others the pot side. You really don't want to try to remove one of these once they're attached. They are difficult to strip if you need to shorten them, so it's best to plan how they're going to sit in the enclosure to work out if you need to cut them or not.

Here the ribbon is on the same side as the pots. It runs vertically straight down to the face of the enclosure, has a gentle bend, runs below the jacks, another gentle bend and up to the underside of the daughterboard PCB. Nice. If it was still a little long it could have bent back and gone under the main PCB a little.

In the case below there's a pesky pot right where the ribbon would normally attach. Here we keep it on the component side of the PCB and solder it in before that pot. From here it can arch over the top of the jacks and into the top side of the daughterboard. No rules, just common sense and care.





TESTING

You really should y'know

Why test?

Yes, you're keen to get this thing boxed and cranked, but testing is important. Why's that?

- **It MAY save time in the long run**

There's a good chance your finished pedal will work fine if you skip the test stage, in which case - kudos. But don't push your luck. One of your builds at some point is going to fizzle and pop or do nothing at all when you stomp it. If only you'd done this test stage it'd save having to pull it all apart.

- **It will definitely narrow down any issues**

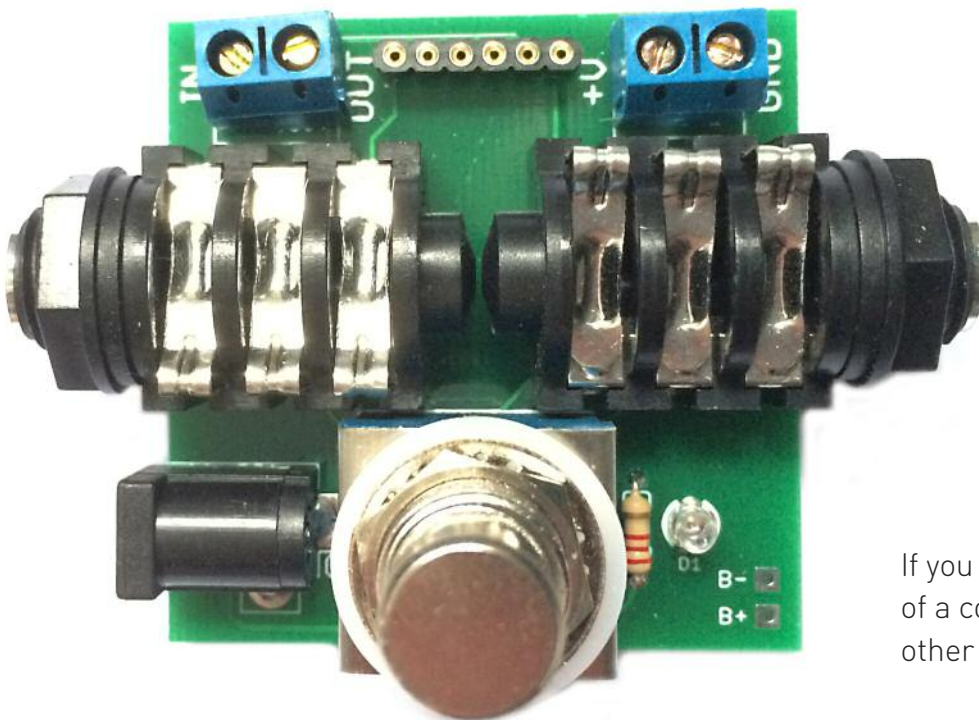
If the worst does happen, and your circuit isn't sounding like it oughta at this stage, at least you've got it pinned down to something on the main PCB. If you'd already attached your daughterboard, jacks and footswitch then they also come into play as possible problems.

- **It really doesn't take much effort**

OK, if you haven't done the smart thing and purchased one of our tester units there's a little bit of time and faff involved, but nothing too outrageous. If you want to save a few quid you can knock up a test rig with just two jacks, a battery snap and a 4-way SIL socket. Stay tuned.

By far the simplest way is to use one of our testers

There are several models, but in most cases you'll be fine with the Simple Circuit Tester (pictured). This has a strip of SIL sockets to insert your ribbon connector if using one, or screw terminals if you prefer to use individual wires on your builds.

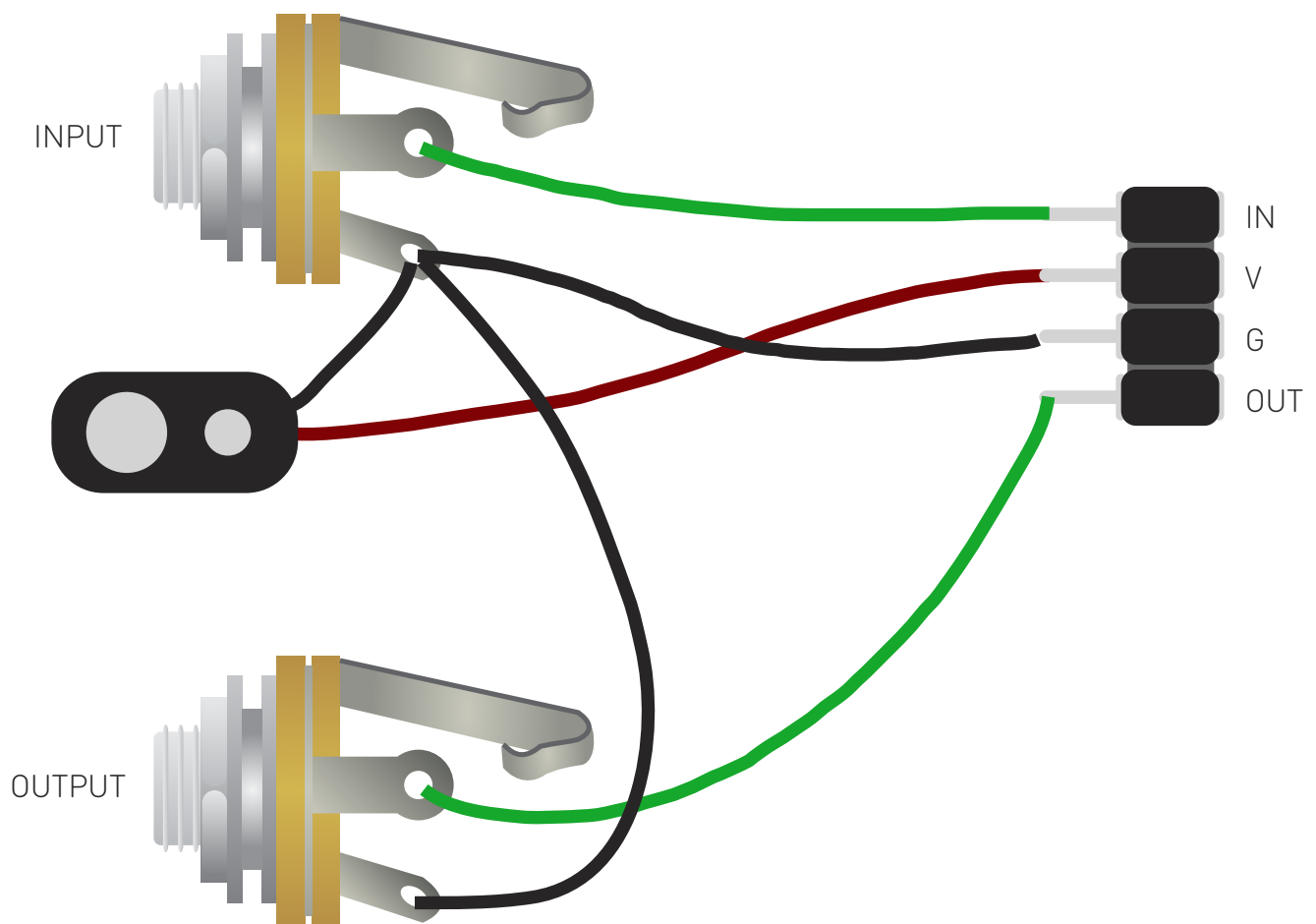


If you don't want to invest the price of a couple of beers, there are other methods detailed overleaf.

Multi-use bodge rig

You can make a reuseable test rig with two jacks, a battery snap and a 4-way SIL socket.

You can then test board with ribbon connectors as much as you like.



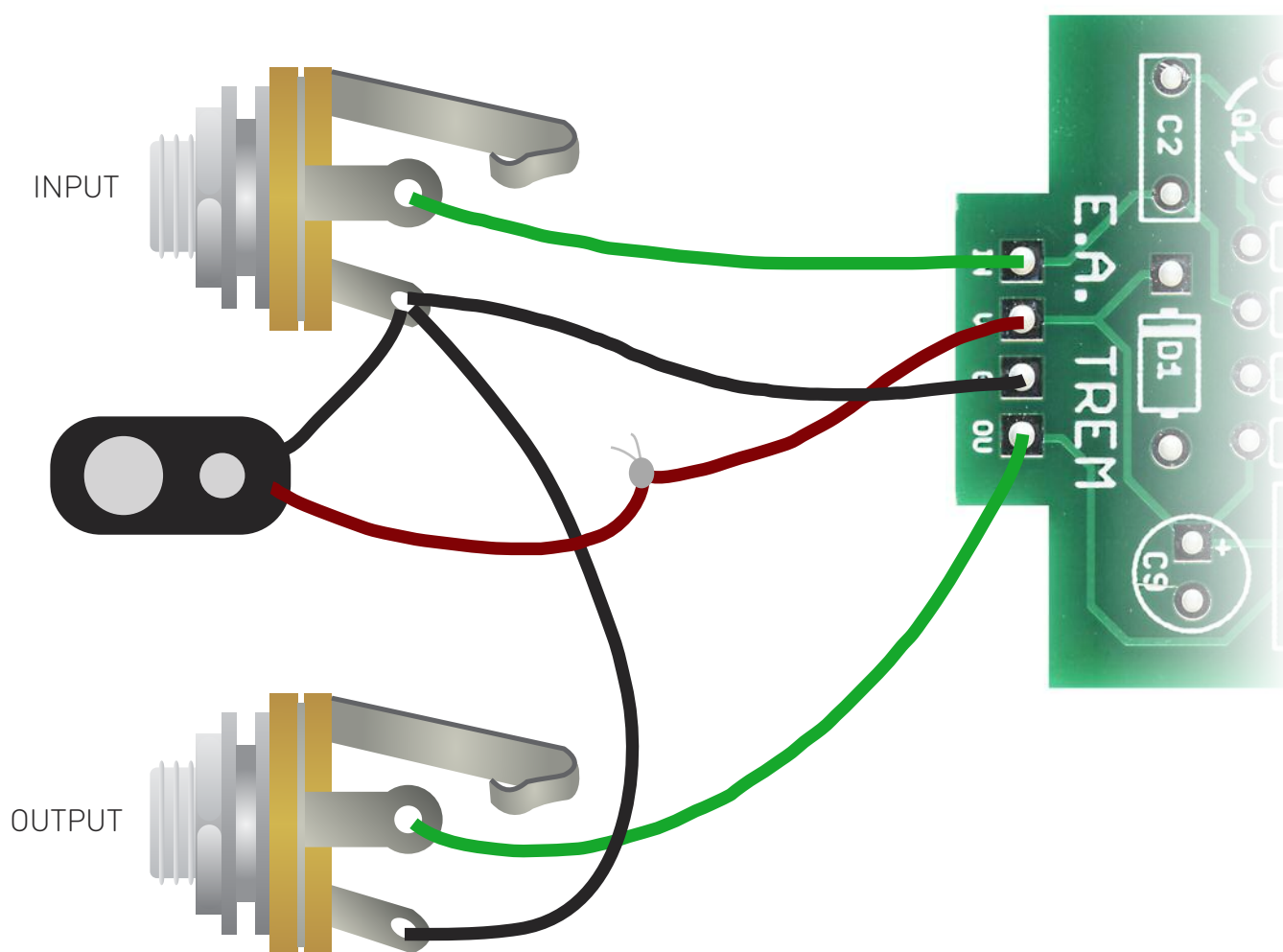
It looks a little messy due to all the ground connections having to go together, but it works.

Not using ribbon? No prob

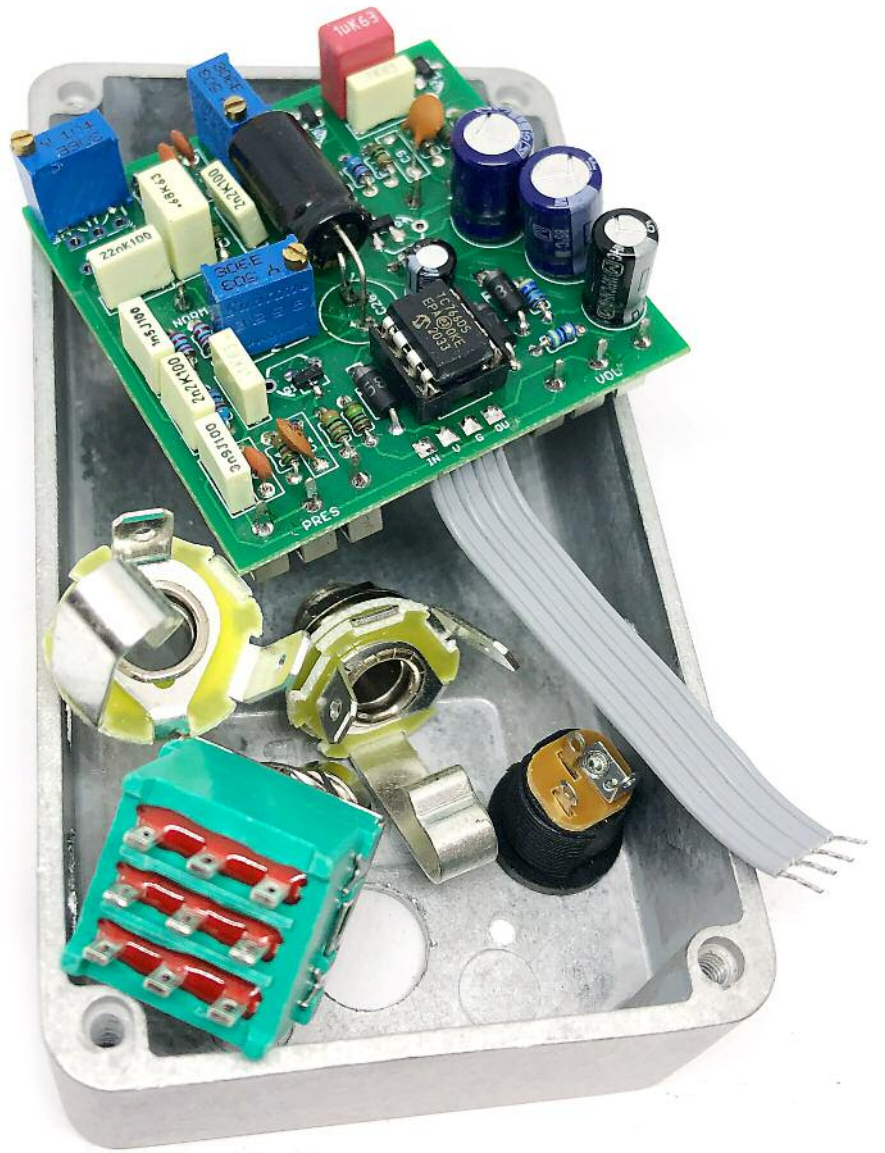
You're just going to have to do a little desoldering on the jacks and battery snap after testing.

Build your circuit and add your four lengths of wire to the I V G O pads. Make them nice and lengthy as you're probably going to cut them after testing.

You then wire up a temporary test configuration a lot like the previous page, but without the socket. Instead of wiring the battery + to the main PCB, join the battery wire to the length of wire you've connected to the main board.



Once tested, desolder the wires from the jacks, trim them to a better length for final assembly and move onto the next stage. Job done.



FINAL ASSEMBLY

Nearly there...

Still a little way to go

You're going to have to scroll on further to find specific instructions for your chosen bypass type, but the basic method is the same regardless.

In most cases you're going to have two jacks, a DC socket and a footswitch of some sort. These all connect to a daughterboard which in turn drops onto your footswitch and connects to your main circuit, completing the picture.

Our recommended assembly order in the enclosure:

1 DC socket

It's a bit fiddly trying to get the nut on once the footswitch is in place, so do this first.

2 Footswitch

It's important to assemble this correctly. There are two nuts. One goes right down the threaded part until it almost touches the body of the switch. Don't set it right against the body. This nut is there to put all the strain on the threads rather than where the body meets them. If you don't include this nut the switch will just pull apart when fastened up.

Next there's either a serrated anti-vibration washer (3PDT switches) or a split washer (Alpha DPDT switches) that goes on the inside of the enclosure against this first nut. Slip the footswitch through the hole in the enclosure. Add the smooth washer if you have one, or indeed want to include one, then the final nut to fasten it. Ensure it's straight and tighten.

3 Jacks

Washer and nut on the outside of the enclosure. Fasten very loosely at first. Take note of the position of the lugs in case they'll interfere with the circuit board. Don't fully tighten until the circuit board is in place and you've manoeuvred the ribbon cable below them if necessary.

4 Main circuit board

Once this is in you can fasten those pots right up.

5 Footswitch daughterboard

...which you've already prepped after looking at that specific section.

Then it's just a case of soldering everything to the daughterboard. We recommend doing the DC socket wires first, as they'll be hard to get to once the daughterboard is placed onto the footswitch. We then locate the ribbon connector into the daughterboard pads and place the board onto the footswitch before soldering them in, just to ensure everything sits nicely. Then just make all your other connections. Everything should be readily accessible.

Ensure your footswitch is straight, otherwise your ribbon connector will be twisted when soldered to the daughterboard.

And it just looks nicer than wonkly.

We prefer to have the DC socket oriented as shown. It makes for easy access for the wiring.



Daughterboard prep

It's easiest to add your wires before soldering the PCB to the footswitch. Here's a typical example for a standard box layout as above.

As a rough guide we'd say:

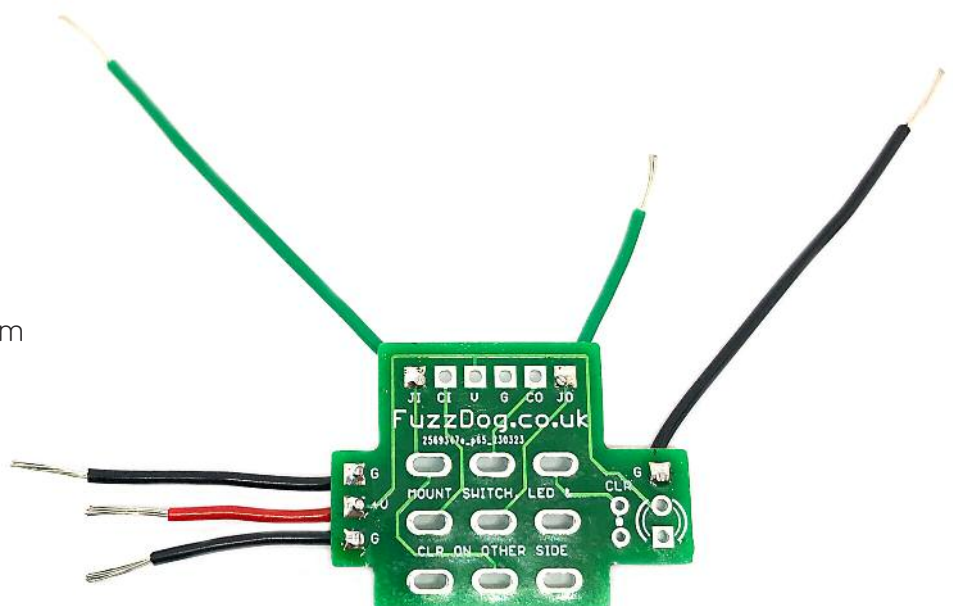
DC + and G - 20mm

Input jack ground - 25mm

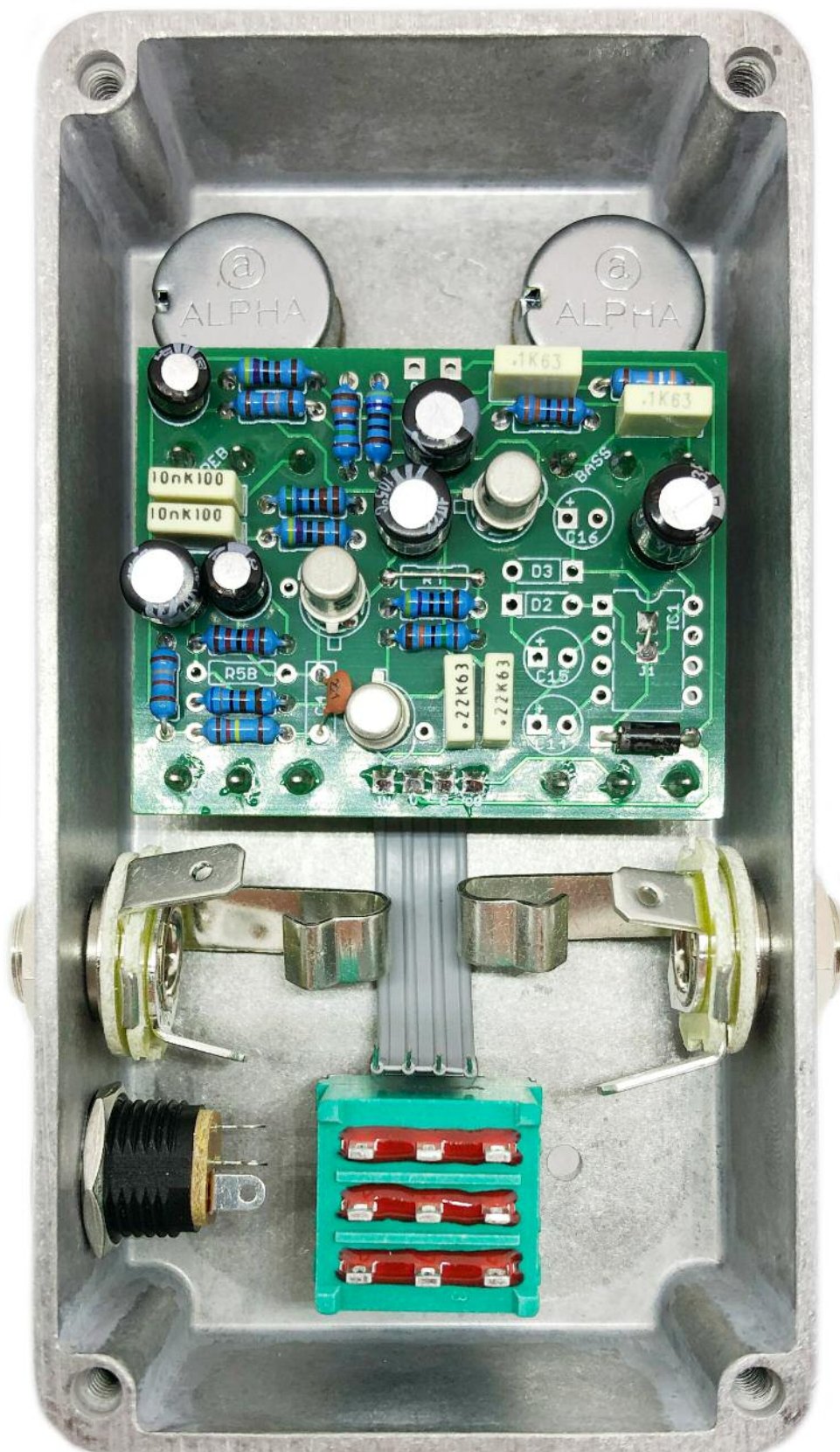
Input jack signal (JI) - 40mm

Output jack signal (JO) - 20mm

Output jack ground - 40mm



Note the position of the ribbon cable. It's been curved below the jacks and is pointing straight up around the same level as the footswitch poles, ready to drop the daughterboard onto.



Dealing with the LED

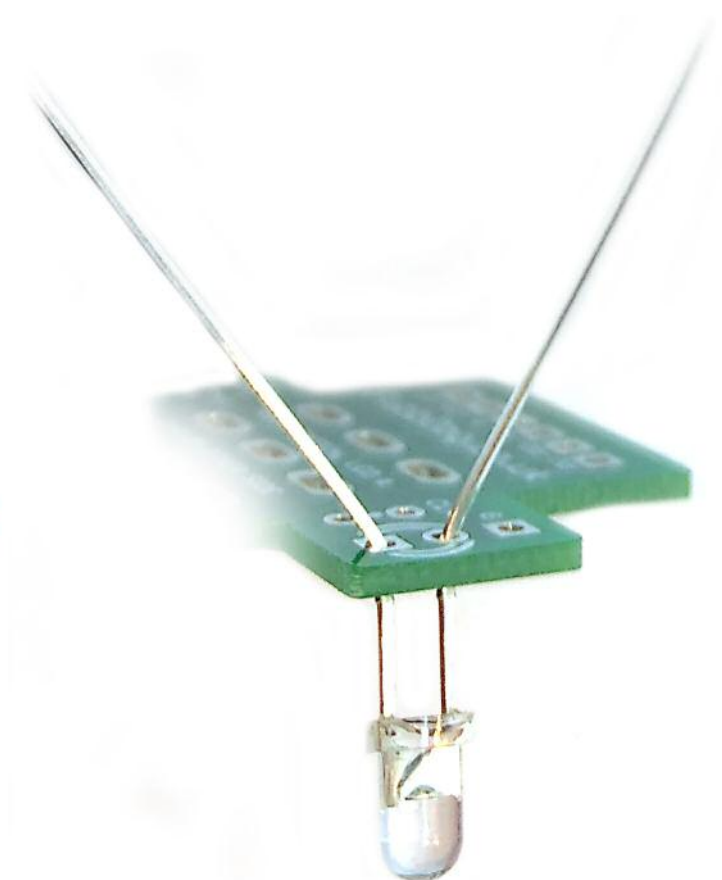
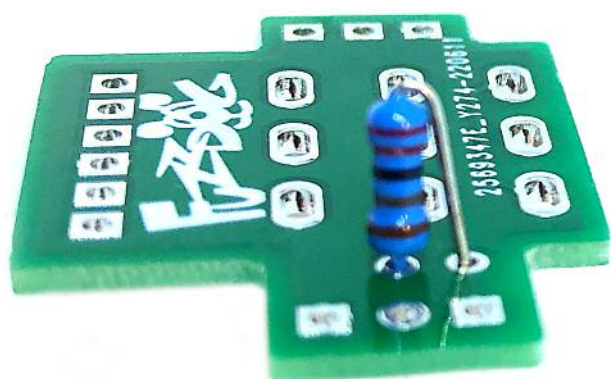
We don't use bezels, so this is what we do.

Before you solder the daughterboard to the footswitch you need to get the LED in there.

Before that, solder your Current Limiting Resistor, which stops the power destroying your LED, into the two pads marked CLR. It should go on the underside of the daughterboard, positioned vertically as shown below.

You're not soldering the LED yet. Just push the legs into the pads, slide it up a ways and then bend the legs to stop it falling out.

Do the rest of your assembly, then this final step. Push the LED down through the daughterboard and locate it into the hole in the enclosure. It'll be nice and snug, and may need a little push with your pliers. Ensure the legs aren't twisted at all or touching each other. Once you're happy, solder it in. It's now just as secure as if it had one of those ugly bezels. It's good enough for EHX, Boss and many others, so it should be good enough for you. If not, source a bezel. Myeh.





BYPASS

Several variations, same results

More than one way...

We offer several bypass switching options. The basic idea is the same in each case, but different circuit boards are utilised in each. You don't have to use a daughterboard, but we highly recommend them. If you want to do without you'll have to find a way to secure your LED in the enclosure. There are plenty of resources for wiring diagrams - search '**3PDT true bypass wiring**'.

In all cases we're dealing with the same four main connections between your main circuit board and your footswitch bypass daughterboard:

In V G Out

- I or CI** The signal being sent to the main circuit
- V** Your positive power supply, normally 9V DC
- G** The ground connection
- O or CO** The output of the main circuit

The other connections on the daughterboard are also the same in each case:

- J1** The connection to your input jack socket
- V** Positive connection to the DC power socket
- G** Negative connection to the DC power socket
- JO** The output jack socket.

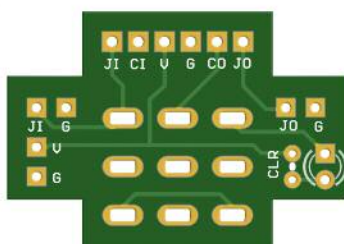
However, not every connection will necessarily be made on the daughterboard depending on the configuration of the main circuit board and your chosen pedal layout. This will be outlined later.

Identify the daughterboard you have below and click on it to go to that section. The first three utilise our standard 3PDT or Gørva footswitches, the OptoPuss uses Alpha DPDT switches. The original 3PDT PCB is being phased out as supplies run out, being replaced with the compact version. The connections are the same for each board but positioned in different places, so we'll show each in the following pages.

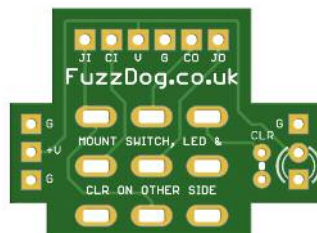
IMPORTANT - in every variant the Current Limiting Resistor (CLR) should be placed on the daughterboard. Our older main circuit boards have a space for it, but you should leave this empty. If your main PCB has a pad marked LED to the right of the IVGO connections, this is the case.



3PDT - ORIGINAL



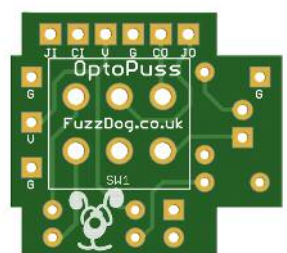
3PDT - COMPACT



3PDT - TOP LED



OPTOPUSS

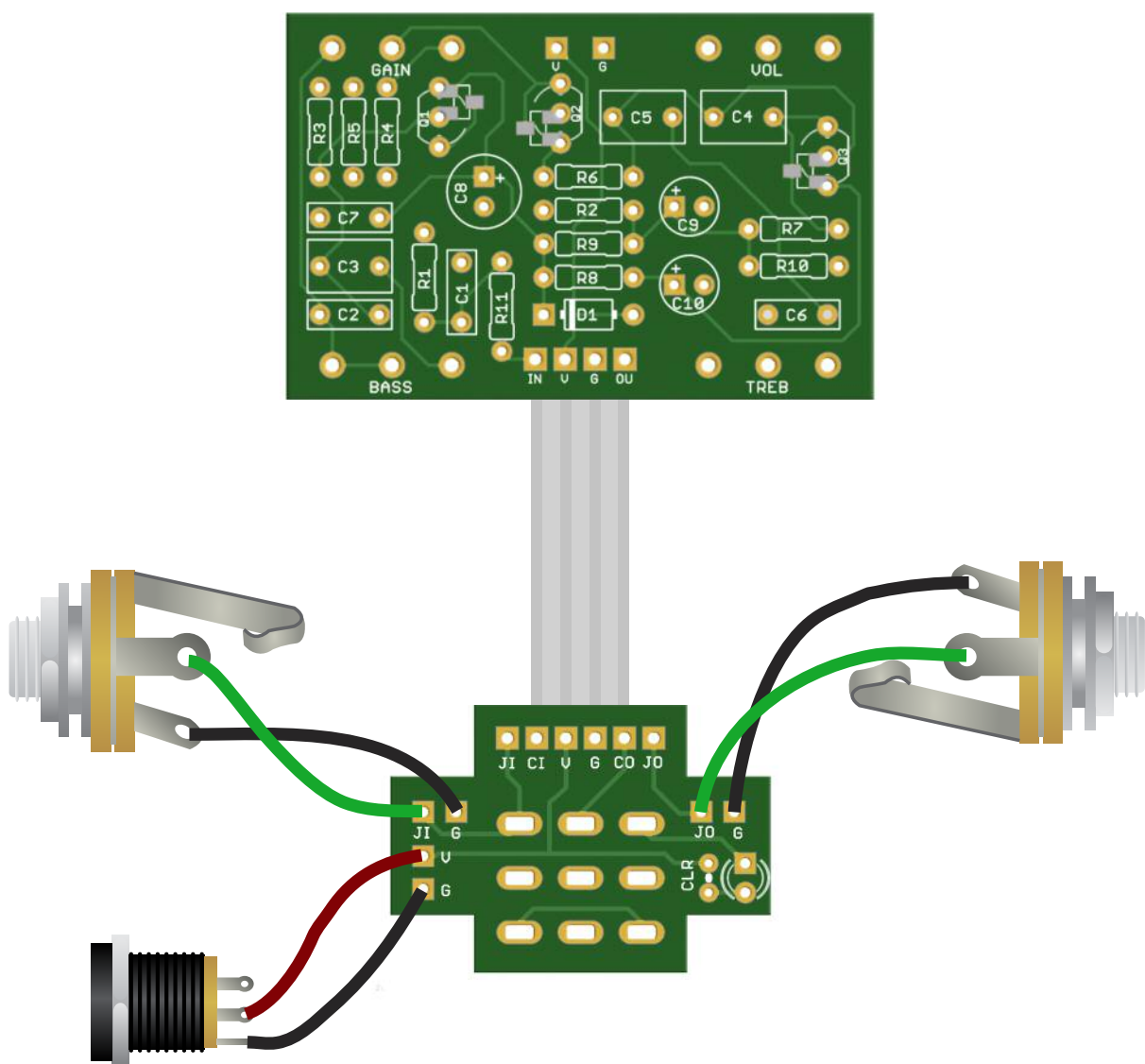


3PDT - Original

No battery provision

This configuration is for DC power supply only.

If using the mini-DC socket the red wire goes to the longer pin.



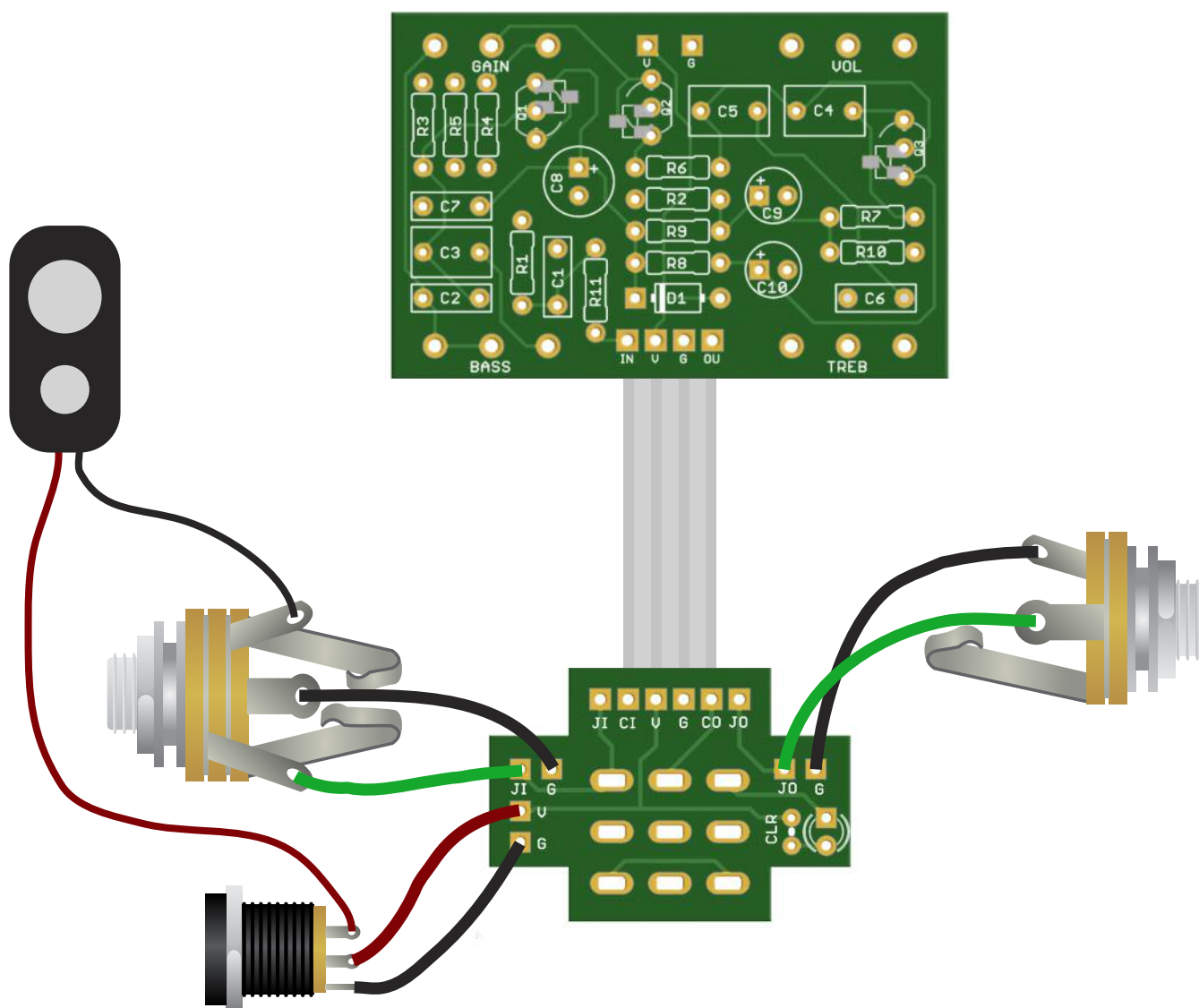
3PDT - Original

Battery provision

This configuration allows for battery or DC power.

The battery ground is disconnected when your input jack is removed, so your battery won't drain.

The battery is disconnected when you insert a DC power plug. This requires a TRS jack socket.

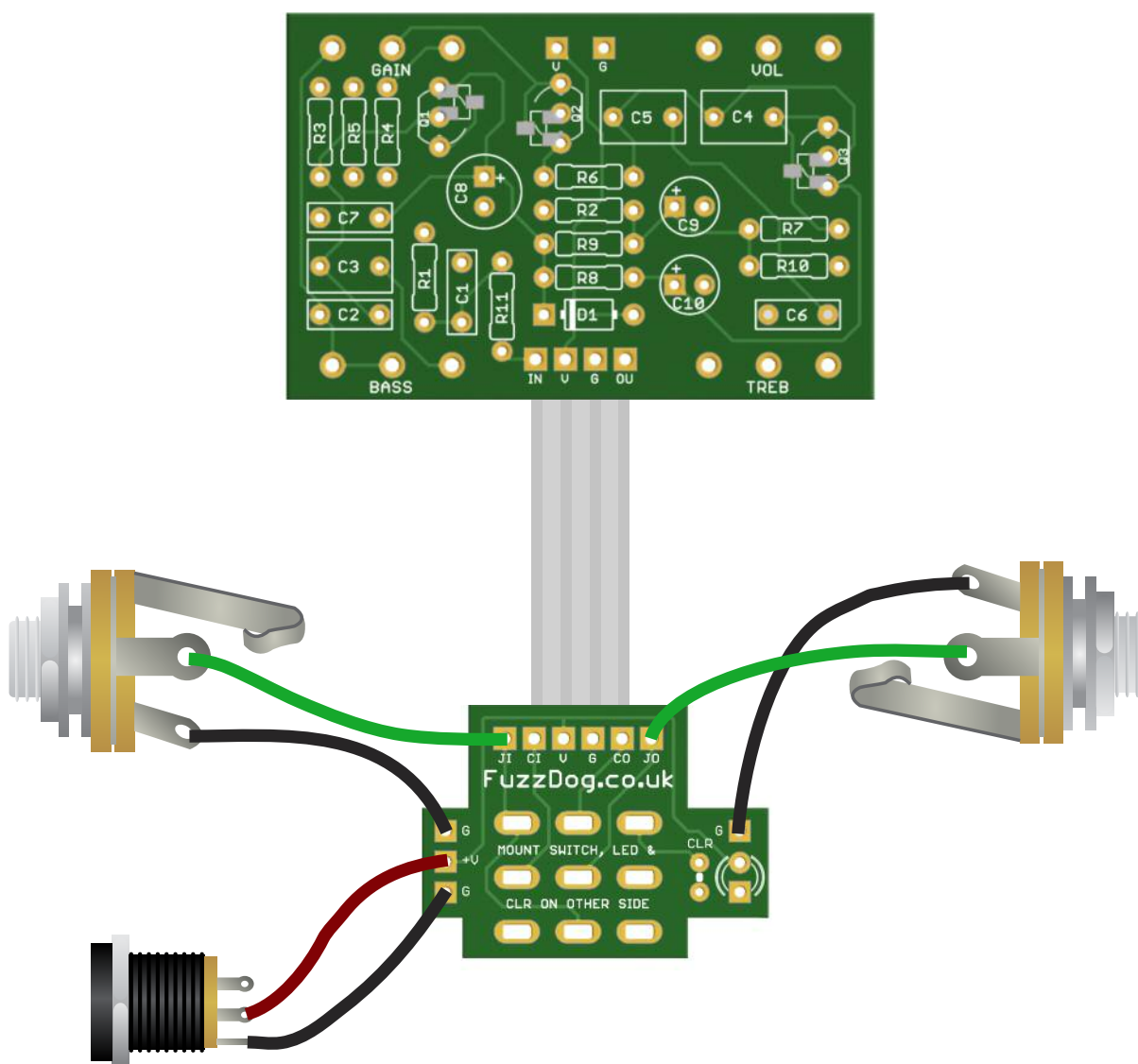


3PDT - COMPACT

No battery provision

This configuration is for DC power supply only.

If using the mini-DC socket the red wire goes to the longer pin.



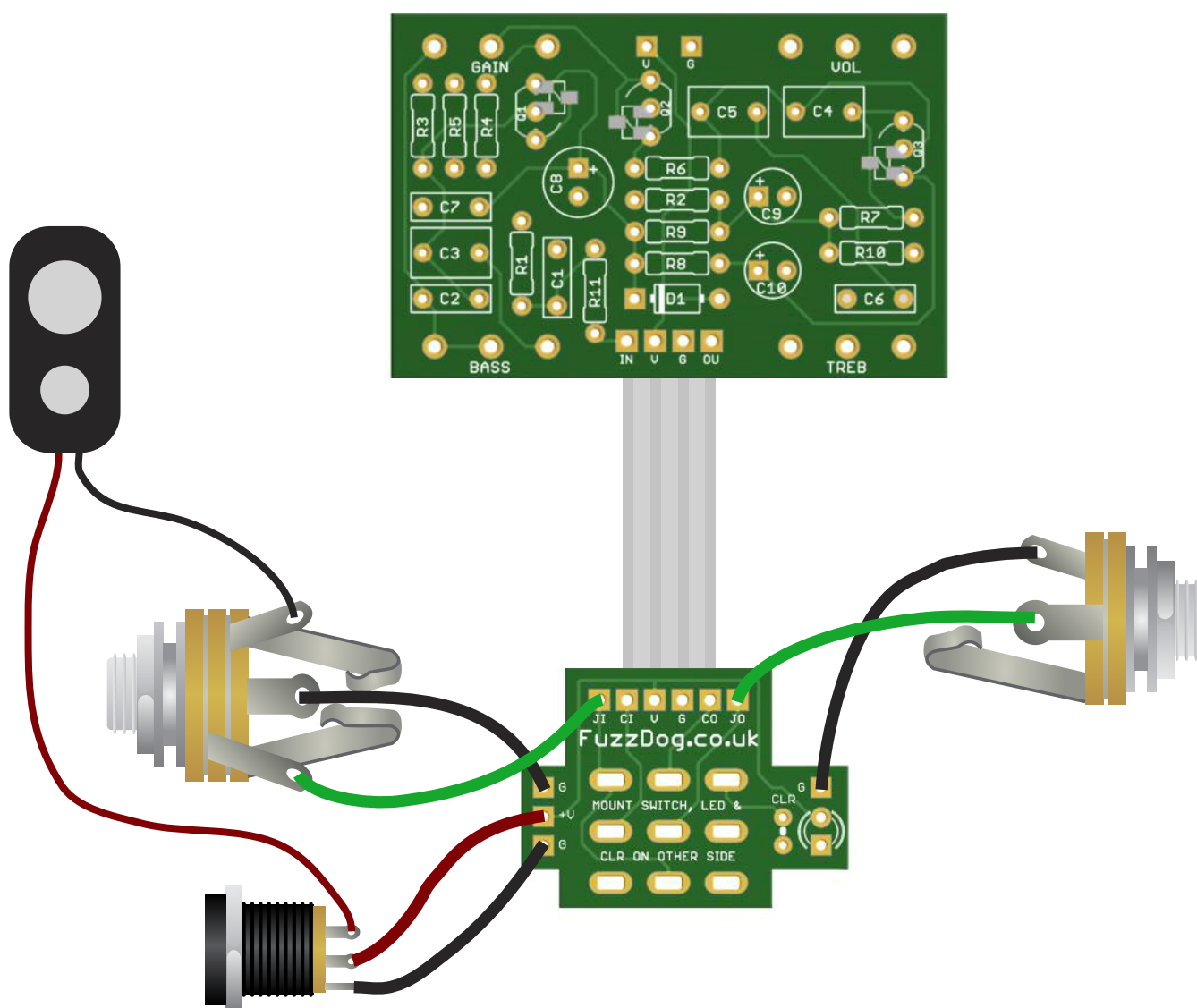
3PDT - COMPACT

Battery provision

This configuration allows for battery or DC power.

The battery ground is disconnected when your input jack is removed, so your battery won't drain.

The battery is disconnected when you insert a DC power plug. This requires a TRS jack socket.



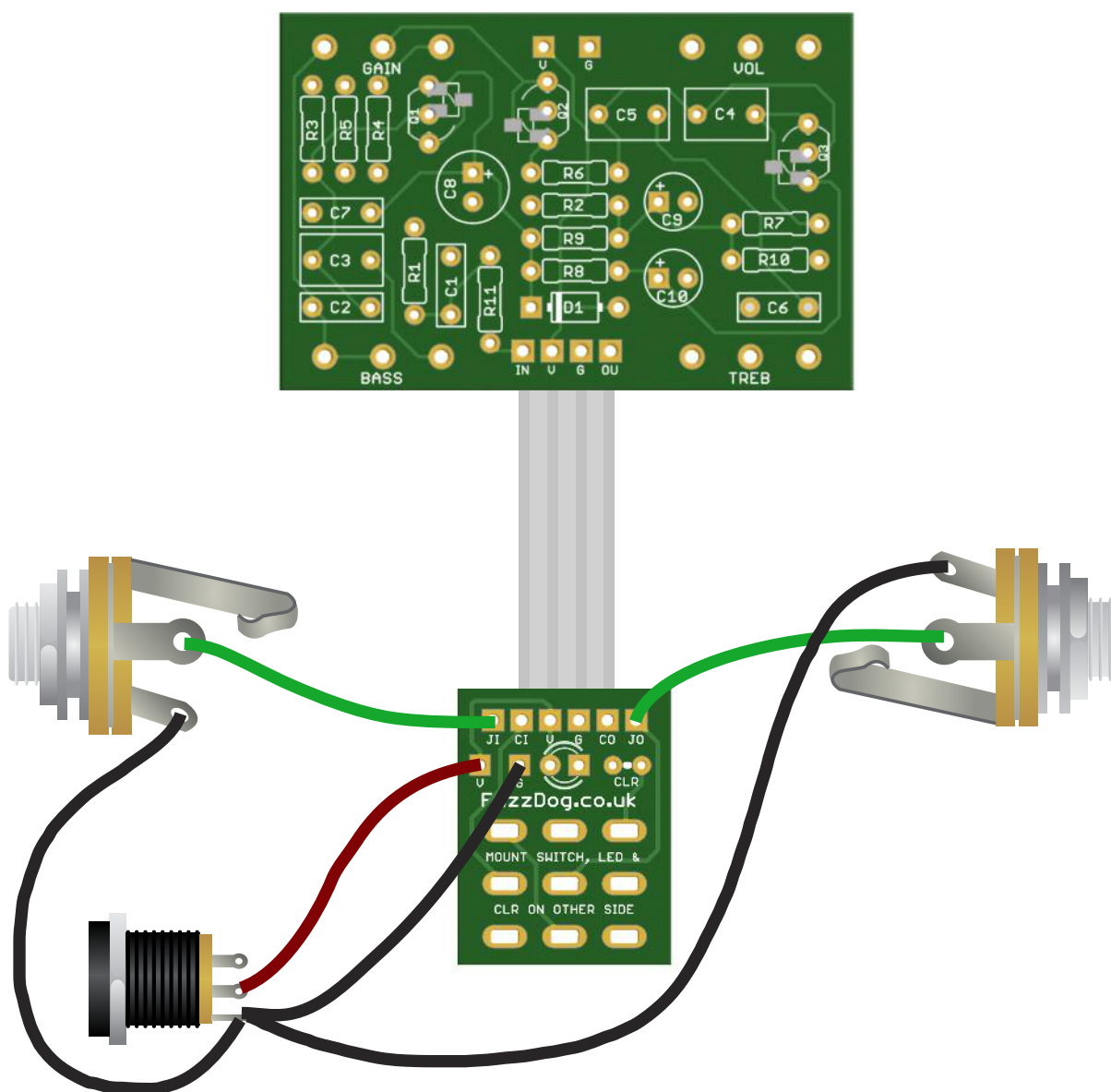
3PDT - TOP LED

No battery provision

This configuration is for DC power supply only.

If using the mini-DC socket the red wire goes to the longer pin.

This board was designed to go with main circuit boards that provide the jack connections on-board. There are no pads for the jack grounds, so it's a bit messy to use on 'normal' builds. See later.



3PDT - TOP LED

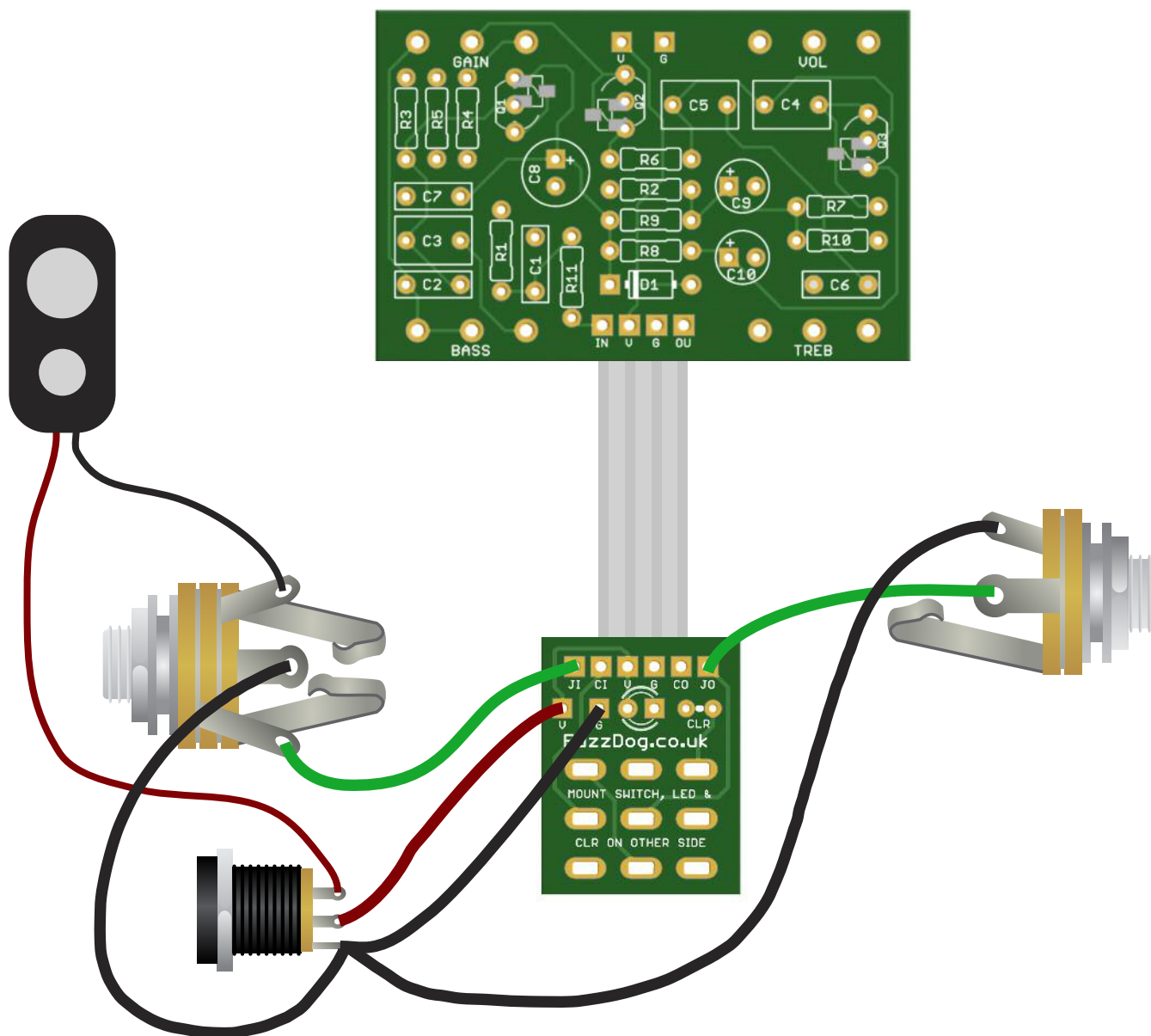
Battery provision

This configuration allows for battery or DC power.

The battery ground is disconnected when your input jack is removed, so your battery won't drain.

The battery is disconnected when you insert a DC power plug. This requires a TRS jack socket.

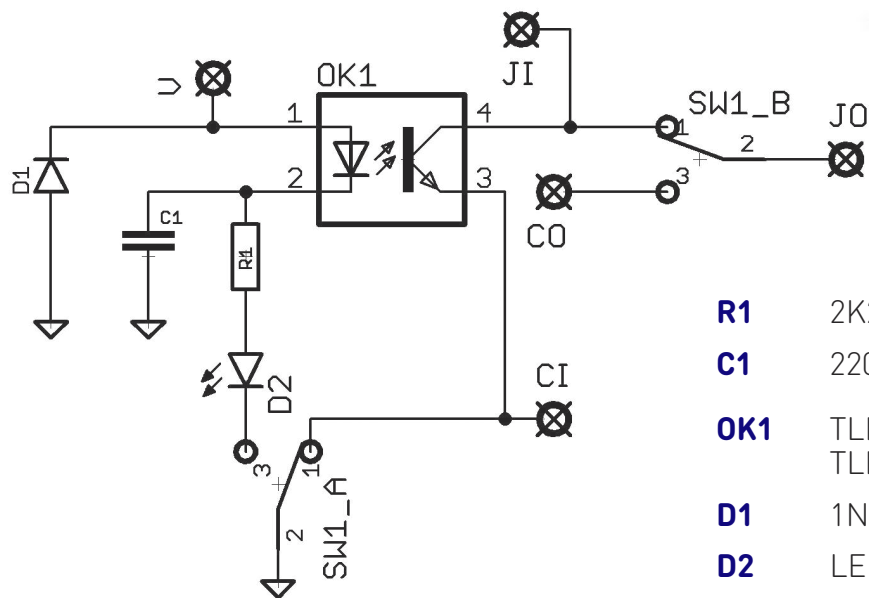
As you can see, the ground connections will get a bit messy. We don't recommend this other than to use with main circuit boards designed specifically to go with it.



DPDT - OPTOPUSS

The OptoPuss utilises some circuitry to switch your signal.

This all goes on the same side of the PCB as the footswitch.



R1	2K2-4K7
C1	220n
OK1	TLP222G, TLP222A, TLP240A, TLP240D*
D1	1N4001
D2	LED**
SW1	Latching DPDT



How does it work?

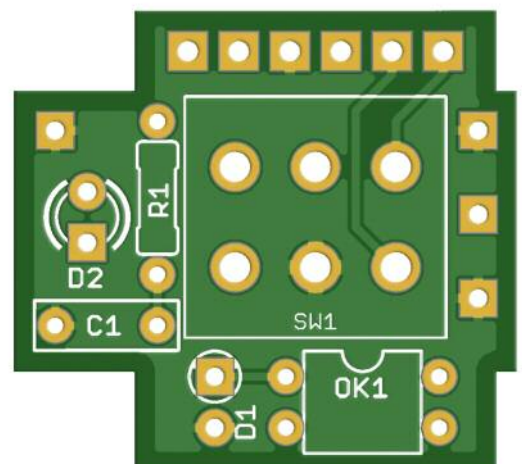
When bypassed there's no power going to the internal LED of the Optocoupler (pins 1+2). This keeps the connections between pins 3+4 (the FET) open. This means the input of the connected circuit (CI - Circuit In) is connected to ground and your signal goes from JI (Jack In) straight to JO (Jack Out).

IMPORTANT - be super careful soldering the optocoupler - they're very easy to overheat.

When the switch is latched you're connecting pins 1+2 from V to GND via C1, R1 and the LED. This current flow lights the optocoupler's internal LED, causing a drop in resistance between pins 3+4 to almost nothing. Now JI is connected to CI, and CO is connected to JO - your signal goes from Jack In > Circuit In > Circuit Out > Jack Out and all is well.

*Note orientation of OK1 on the image. Dot indicates pin 1. Originally designed for TLP222G which is now out of production. Many others will work, but the 240 worked best in our search for alternatives.

The board has been designed for Alpha brand footswitches. Others may fit.

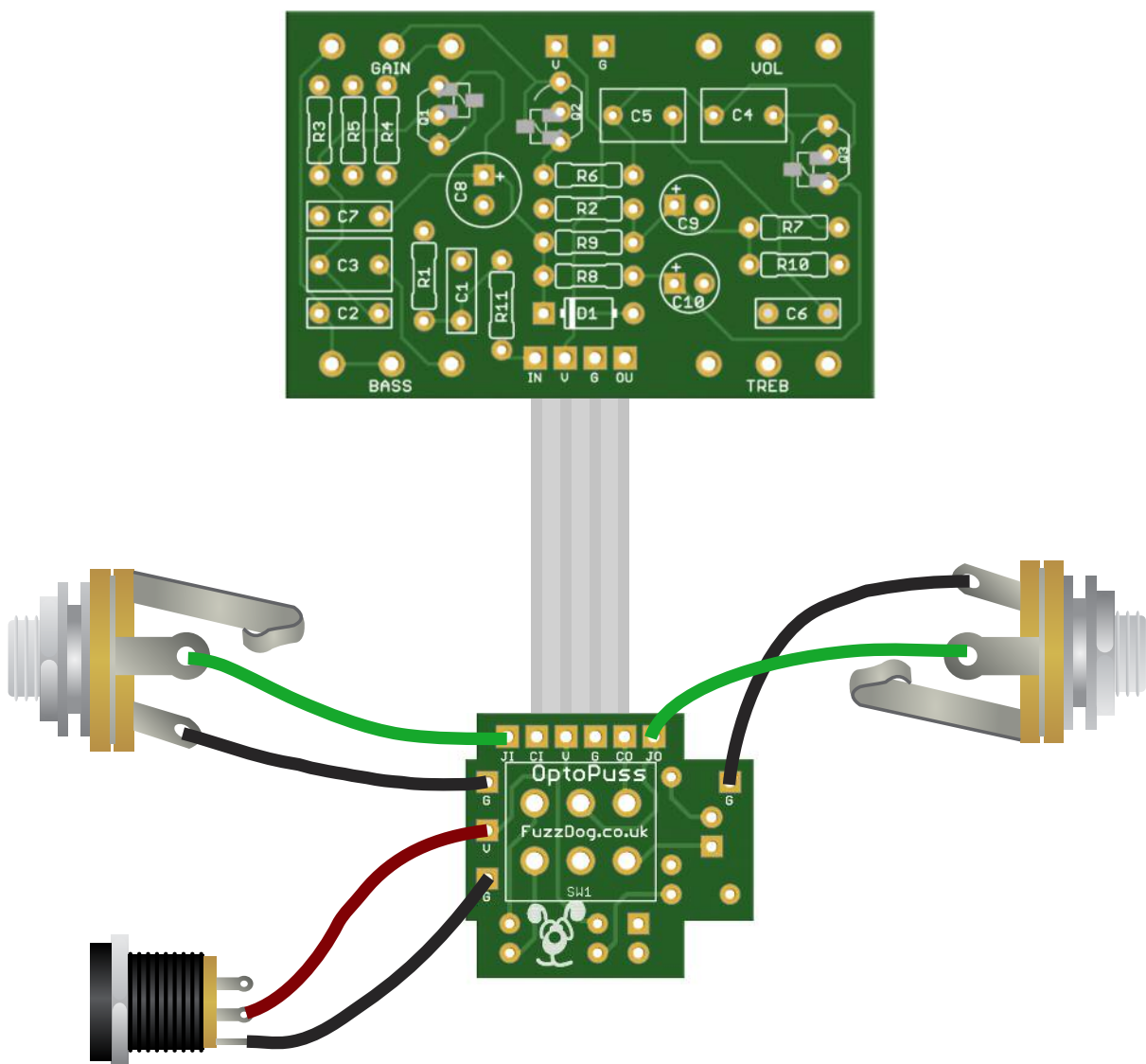


DPDT - OPTOPUSS

No battery provision

This configuration is for DC power supply only.

If using the mini-DC socket the red wire goes to the longer pin.

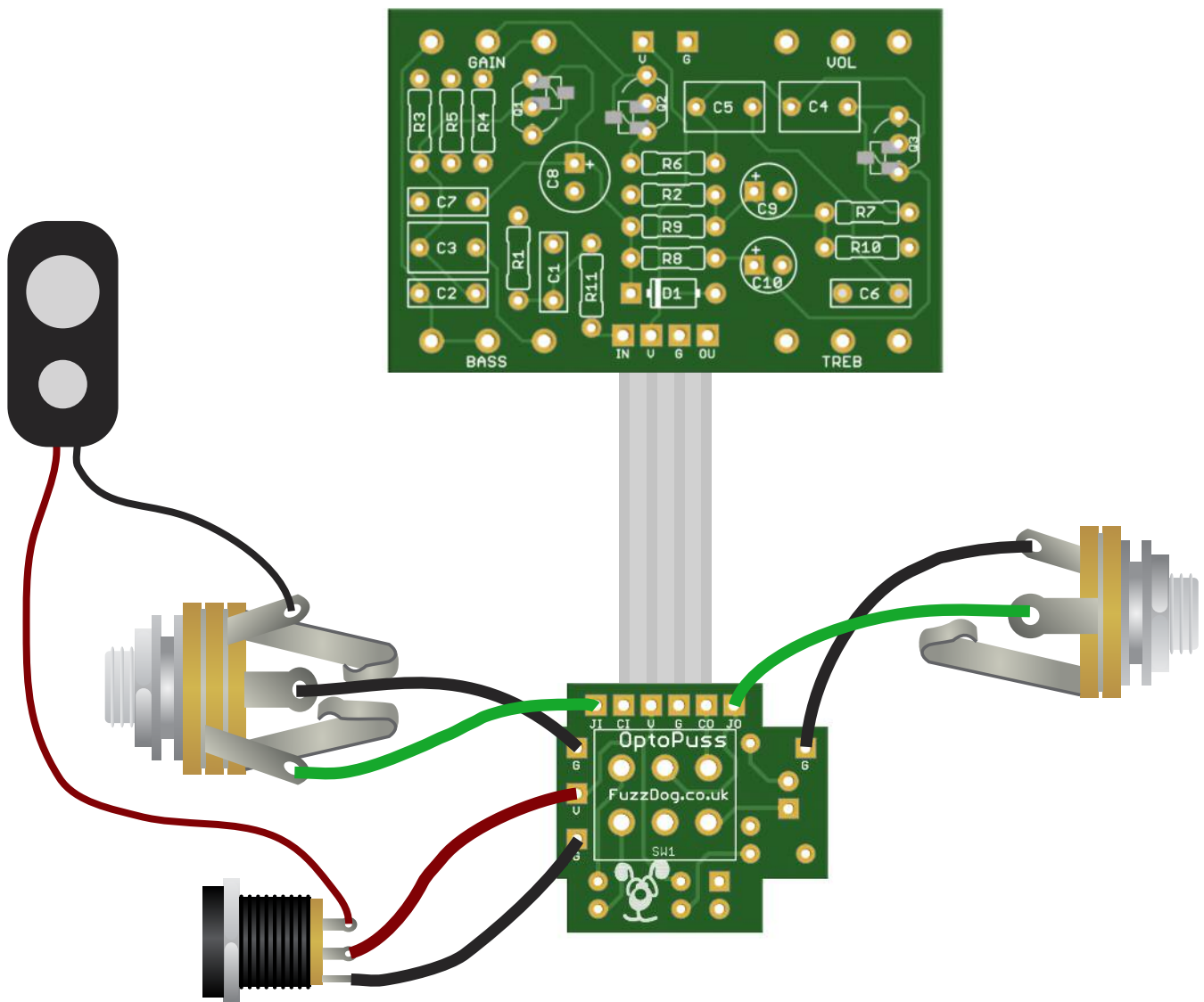


DPDT - OPTOPUSS

Battery provision

This configuration allows for battery or DC power.

The battery ground is disconnected when your input jack is removed, so your battery won't drain. The battery is disconnected when you insert a DC power plug. This requires a TRS jack socket.



Variations on a theme

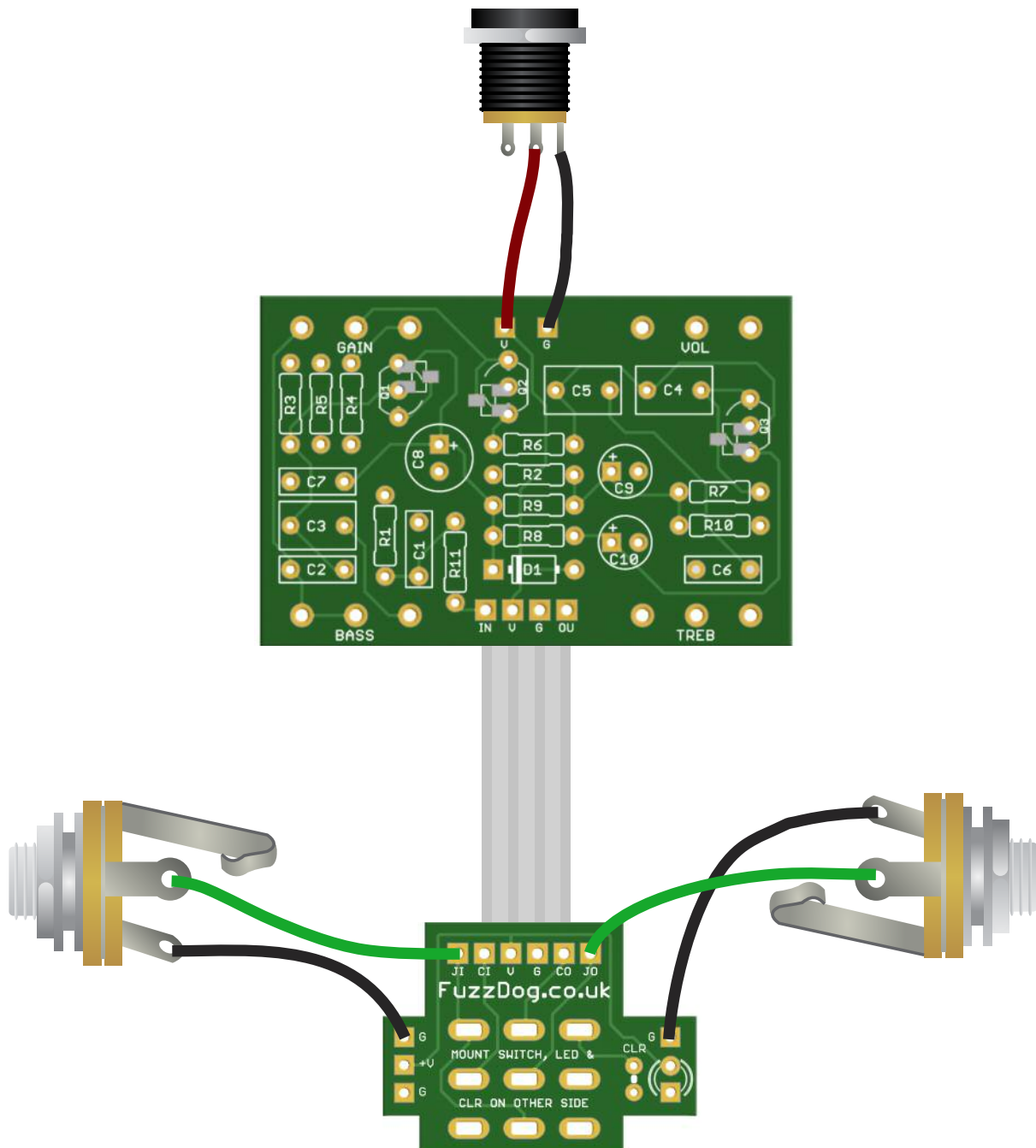
There are times other wiring variations will be available on your build, when a main circuit board has provision for a top-side DC socket or even top-side jacks.

We'll only illustrate one of each case here, as the basic principle is the same for every one of our bypass boards. You simply make the appropriate connections on the main PCB instead of the daughterboard, and in the case of top-jacks you should use all six of the connecting pads between the main PCB and the daughterboard.



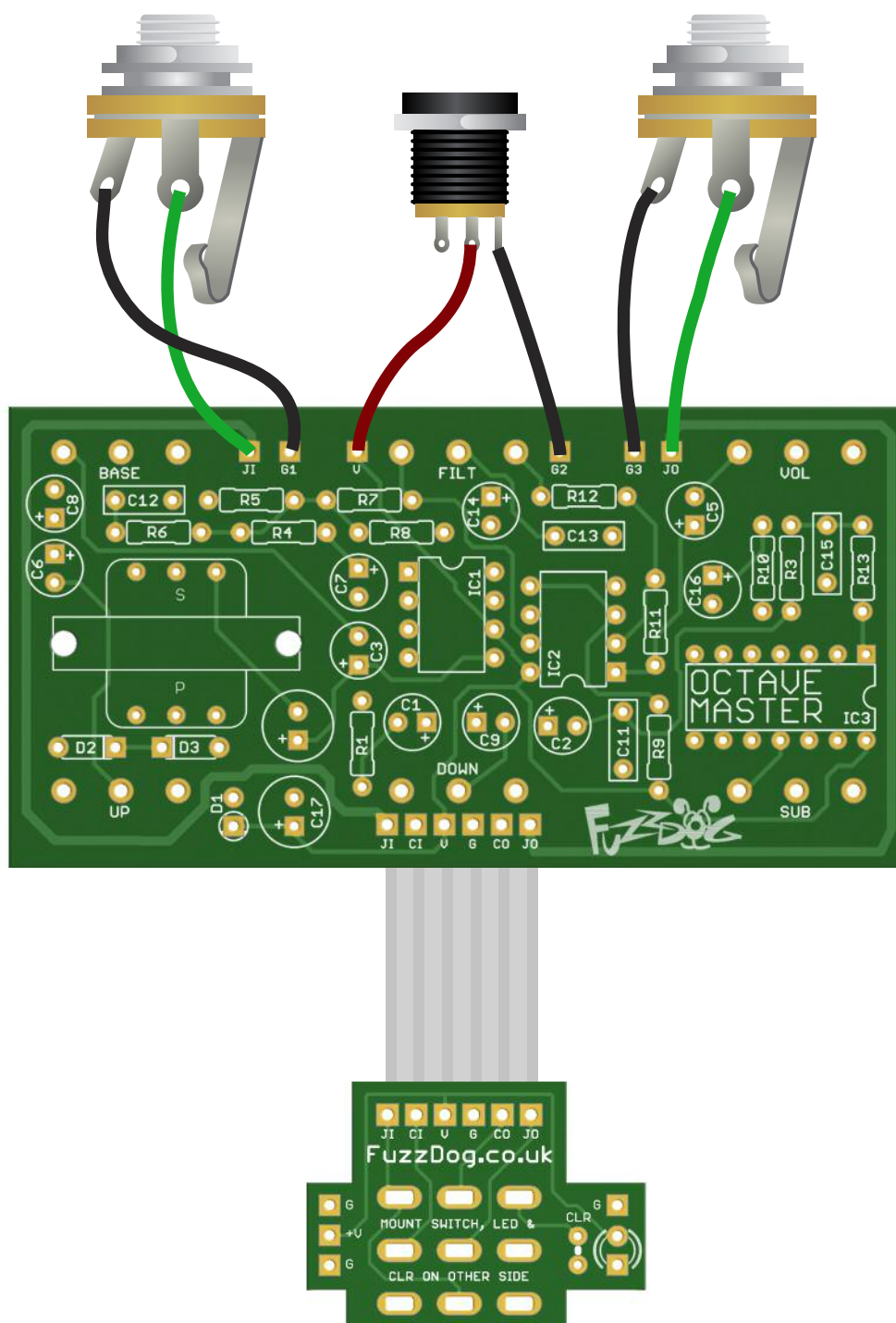
TOP-MOUNTED DC SOCKET

Sometimes our main PCBs will have pads at the top of the board for the V and G connections, as seen in the illustration here. In this case you can simply eliminate the DC V and G connections from the daughterboard and make them on the main PCB instead.



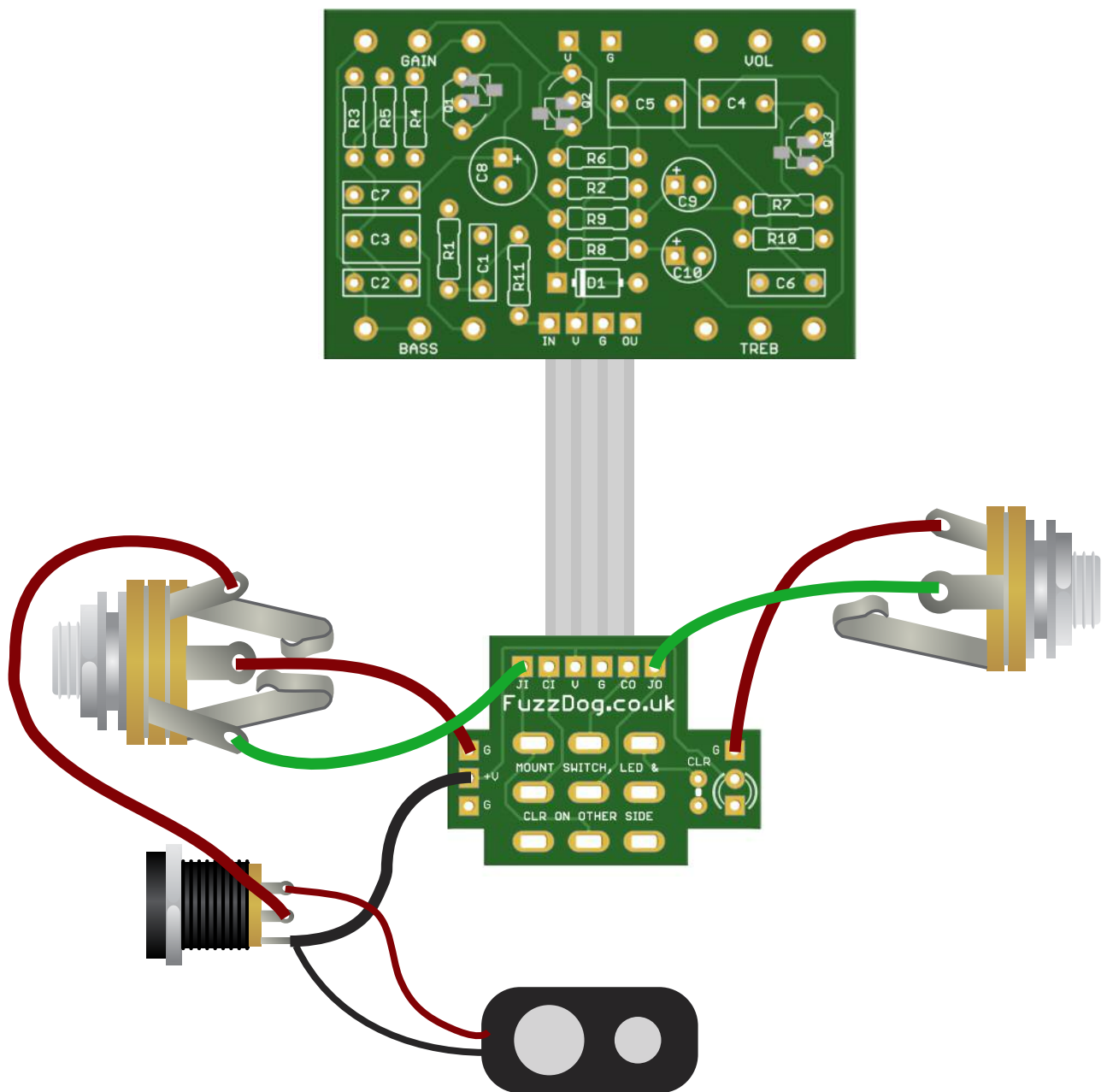
TOP DC AND JACKS

In some cases our main PCBs will have pads at the top of the board for the power and jacks connections, as seen below. In this case the only connections you need to make to the daughterboard are the six pads corresponding to the main PCB. This is entirely optional of course. You could ignore the pads on the main PCB and wire it as normal with just the four connections, using the daughterboard for your jacks and DC.



POSITIVE-GROUND

Some circuits are positive-ground. Often these will have an onboard voltage inverter which enables them to be powered by a standard negative-ground supply. If not you can wire them with the power connections reversed. A downside of this is you can't daisychain this with standard negative ground pedals. It will need its own supply or battery power.



We hope this document has been of help.

We've tried to make it as comprehensive as possible.

If you have any comments or suggestions for improvement, please feel free to get in touch.

Cheers from Team FuzzDog

