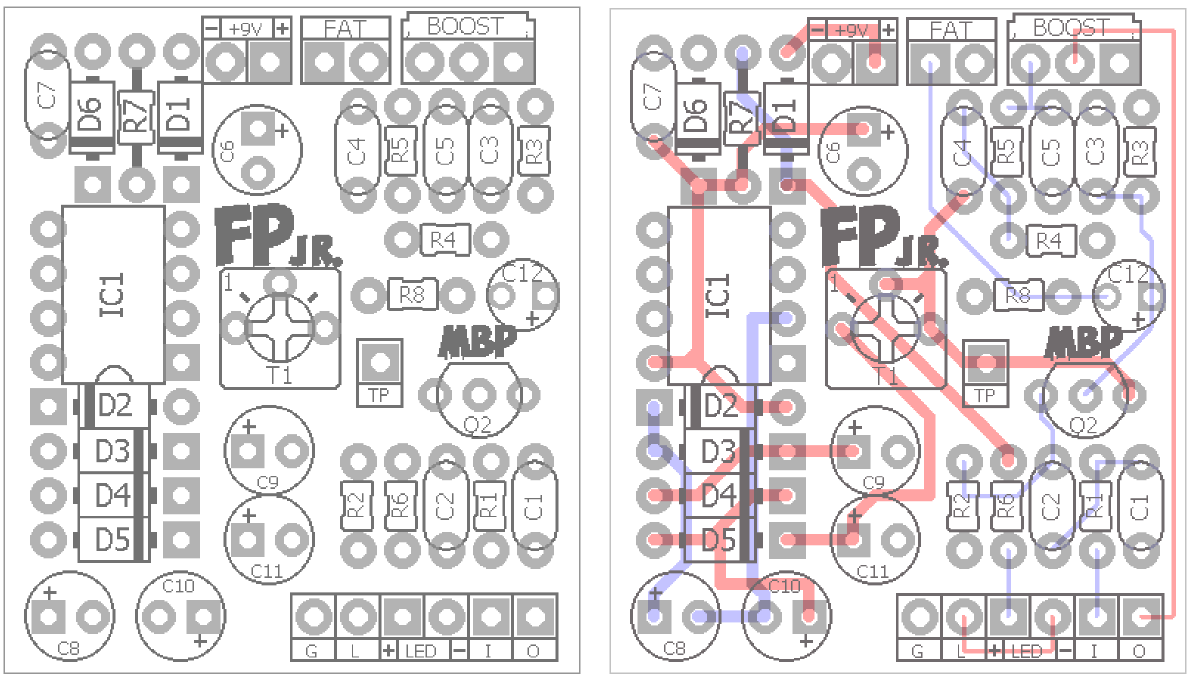
UNDERSTANDING PCB LAYOUTS

When building any PCB project, it's a good idea to familiarize yourself with the component and trace layout on the actual board you are building. This will help you understand how the components are interconnected and get you used to seeing certain layout patterns. This type of information can become very useful when things go wrong. Once you can properly read a PCB layout, tasks like debugging stubborn builds will become much easier. Let's take a closer look.

All madbeanpedals PCBs follow the same standard, more or less. Each fabricated PCB has two copper layers, a top and bottom. Each of these layers contains small copper strips (called traces) that connect the components that make up the circuit. Top and bottom traces are connected to one another by the solder pads the components go into, or by vias which are small pads that do not hold components.

Here is a simple example of a PCB layout, and very typical of what you see on "page 1" of every madbeanpedals build doc.

Fig.1: The silkscreen and trace layout for the FatPants Jr.



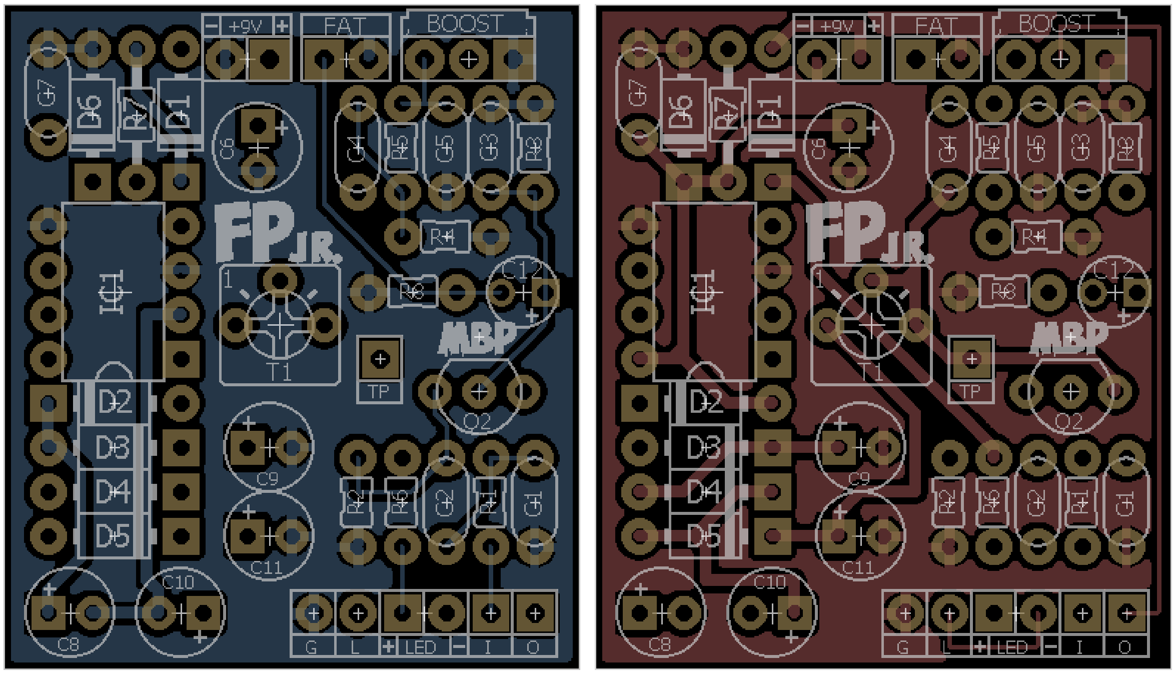
The left image shows the component layout on this PCB. This is the image that is silk-screened to the circuit board by the manufacturer. The image on the right includes the top and bottom traces that connect the components together. The red traces are ALWAYS top layer traces and blue are ALWAYS bottom layer traces (at least on MBP PCBs).

Two observations:

1) Some traces are different sizes. This is not an accident. All MBP PCBs have two defined trace widths; one for power and one for audio. Power traces are thicker (24mil) whereas audio traces are thin (10mil). In general, we like for our power traces to be thick to help dissipate heat, and we like audio traces to be thin to keep any parasitic capacitance (and thus impedance) as low as possible. It's not important that you understand what that means exactly. Only that when you see a thick trace it is likely some element of the power supply and a thin trace is likely an audio portion of the circuit.
2) Some component pads appear to have no connections at all. Take for instance the round pad on C9. There are no traces connected to it. This is because C9 is connected to ground, and on all MBP PCBs, grounding is done through copper pours instead of traces. A copper pour is a large rectangle of copper that surrounds all the components on the PCB. Essentially, it "fills in" the negative space on the circuit board. These pours are used for connecting all grounded components. The reason we do this is it can help reduce noise in some situations. It also makes PCB layout easier because we do not need to draw as many traces on the circuit board. Some people like to make the copper pour on the top connected to the power supply and the bottom connected to ground. This is also a good practice, just not the one I use.

Here are the examples to the bottom and top grounding planes on the FatPants JR PCB.

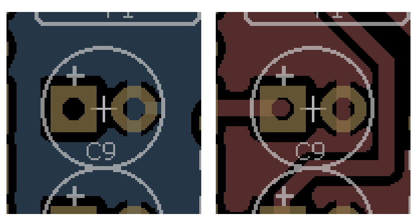
Fig.2: Bottom and Top grounding planes



Once again, the blue image is for bottom traces and the red one is for top traces.

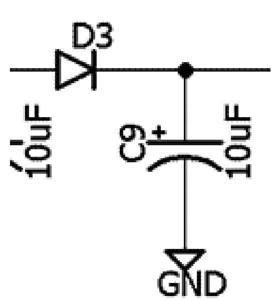
Now let’s look at C9 again.

Fig.3: Bottom and Top grounding planes



Now if you look at the round pad on C9 you can see that there are small “bridges” that connect the pads to their respective grounding planes. These bridges are called “thermals”. We use them because it makes soldering easier. Without the thermals, you actually have to apply more heat to the pads to get a good solder connection, and that is something we try to avoid since it can damage the PCB. You might also notice on the right image that the square pad on C9 (the + lead) has a trace going off to the left. This is the trace that connects C9 to D3, as shown in this snippet from the schematic.

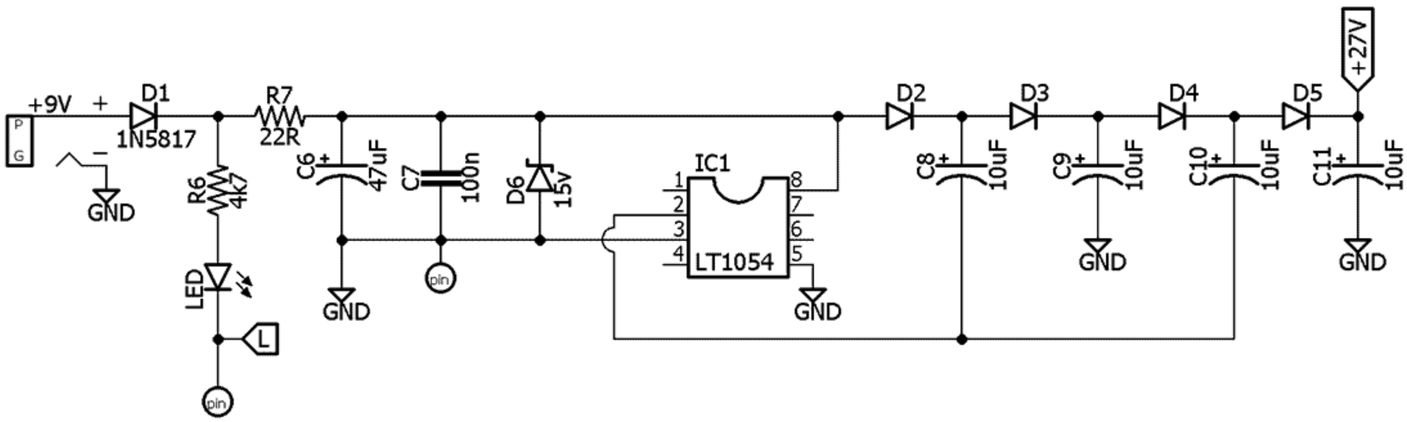
Fig.4: D3 and C9 on the FatPants Jr. schematic



BTW: because of the magic of EagleCAD software (the software I use to make PCB layouts) it is impossible to connect traces on the PCB layout incorrectly. As long as everything is drawn right in the schematic, you literally cannot connect traces the wrong way on the PCB. Pretty cool, huh?

Let's take a larger look at the schematic. Here is the power supply section of the FatPants Jr. in full.

Fig.5: Power supply of the FatPants Jr.

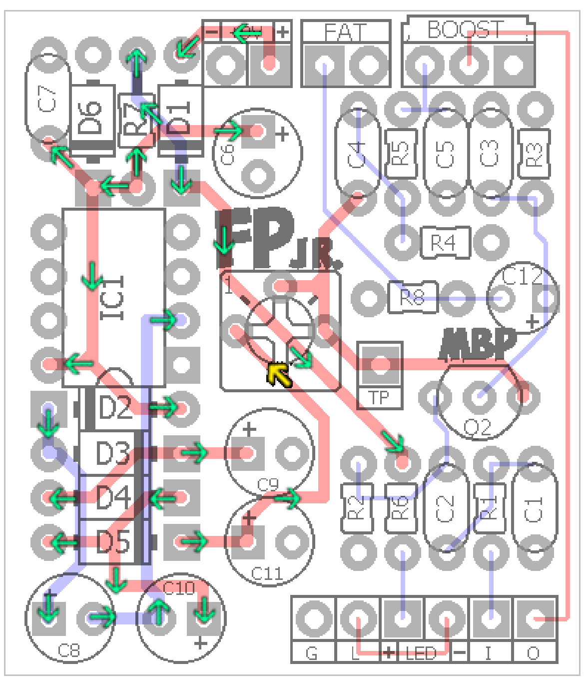


I'm not going to explain what all of this means, as that is an entire tutorial on its own. But, here is the basic walkthrough:

- The +9v jack connects to the reverse polarity protection diode, D1.
- The supply splits off; one to the CLR (R6) for the indicator LED and one to R7. R7 is current limiting and helps reduce noise as well as prevent D6 from burning out if a very large DC supply is attached.
- C6 and C7 are for power decoupling to smooth the incoming DC. They also form a small LP filter with R7.
- D6 prevents excess voltage from damaging our charge pump.
- The charge pump and associated circuitry pumps the 9v supply up to about 27v to the JFET used in the audio part of the circuit.

Let's compare this section of the schematic to the accompanying PCB layout.

Fig.6: Power supply trace routing

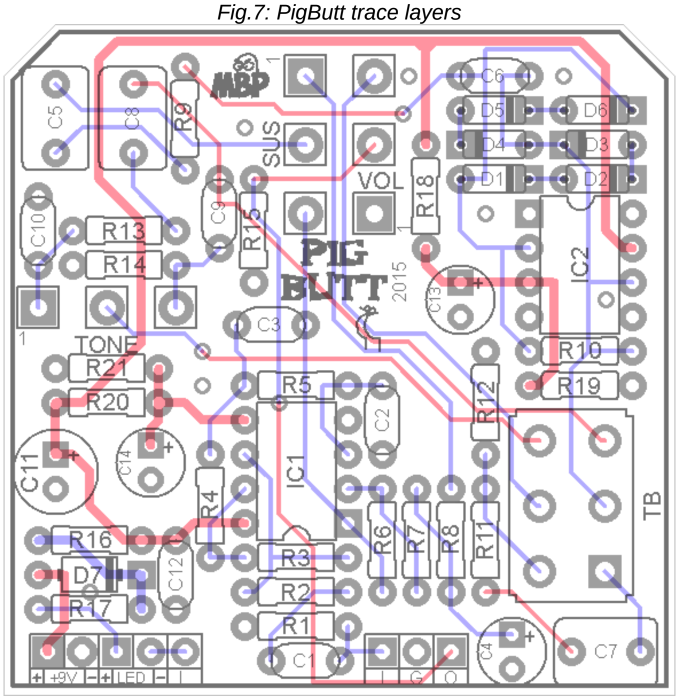


Starting at the 9v jack, the green arrows traces the path that the power supply takes all the way to the T1 trimmer (shown by the yellow arrow). The arrows trace the route the positive DC voltage takes to supply the audio portion of the circuit with power. Quite a lot! You will also notice that some traces are on the top (red) and some on the bottom (blue). This is the beauty of double-sided layouts. It allows us to make PCB layout much more compact.

Remember that the grounding is not shown…all the grounds are linked through the copper pour which is not illustrated in this diagram.

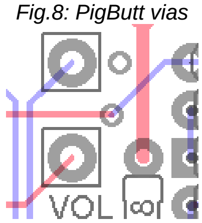
Let's take a look at another PCB example.

Fig.7: PigButt trace layers



Here are the trace layers for the PigButt. You'll notice some things are different; the 9v jack in on the bottom left instead of the top, and there are a lot more traces (because there are more components). But, here's something else that's different. Notice the extra little round pads that show up? Take a look near the Volume pot; there are two little pads there.

Fig.8: PigButt vias



Those pads are called vias. They are used to connect the top and bottom traces just like we do with the pads the components are soldered to. In the case of vias, though, we use them when we need to connect top and bottom in a place where there are no components.

The via closest to the top edge connects the top ground plane to the bottom ground plane. Why do this? Because sometimes our ground pours do not end up covering all the "negative space" in the PCB layout. Thus, you might have a small area where there is a top ground pour but not a bottom one. By using a via, we can fill in the "missing" pour.

The second via below that one is used to connect a top and bottom trace. In this area we need to get the top trace over to the C6 cap. But, there is another top trace connecting R18 to the power supply. So, we use a via to shift the trace from top to bottom to cross over the R18 trace.

The two things to take away from this are, 1) when you are following traces on a PCB be sure to pay attention to any vias because they connect the top to the bottom and 2) *never solder anything on or in a via*. They are there only to make the PCB functional and do not require you to solder anything to them.