PALE HORSE BUILD GUIDE

Nov 2017 update – This build guide shows this process using the old version of the Switching Board. It is still useful as a guide although there are a few differences with the new v2 Switching Board. Please refer to that documentation for a complete picutre.

This guide shows most of the process of building one of the VFE projects on madbeanpedals. The illustrated steps focus on the actual circuit building and not assembly. Assembly for these projects is much like any other pedal build. If you follow these steps and take care with drilling your enclosure it should go smoothly. However, there are some important tips in this tutorial so please read carefully. It might same you some time and frustration down the road!

Building the VFE projects is a bit different form the average DIY pedal. These boards are designed for production and utilize board-mounted hardware. This can greatly speed up turnaround in assembly, however it can present some challenges for DIY. Since we do not usually have access to CNC drilled enclosures that will guarantee a consistent drill spec in our homes, there is a bit more care required by the DIY'er.

Another advantage of production ready boards is that they can quickly be plugged into a guitar and amp for rapid testing. No messing around with clunky prototyping boards with wires going all over the place…just plug in and rock out.

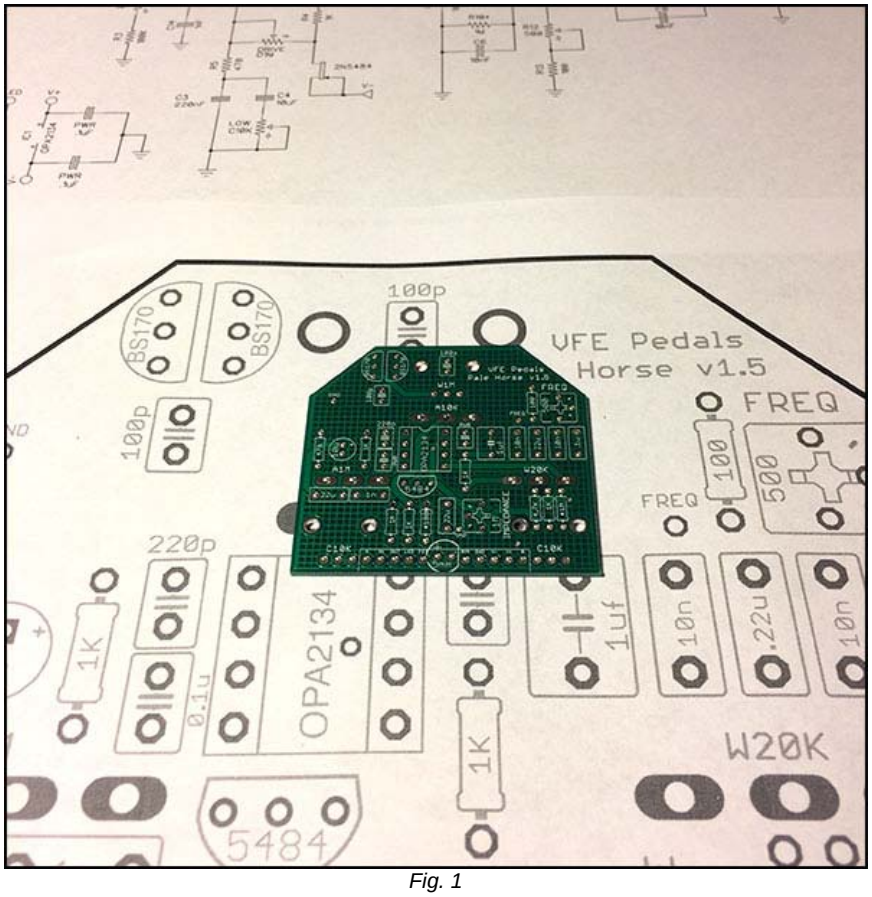

Fig. 1

First we'll begin with the audio circuit portion of the Pale Horse. Remember that each VFE project has two PCBs; audio and switching. The two boards are wired together to form the complete circuit for each project.

I generally work off just the schematic when building pedals, but since I did not design this PCB I printed up a copy of the PCB silkscreen as a helper.

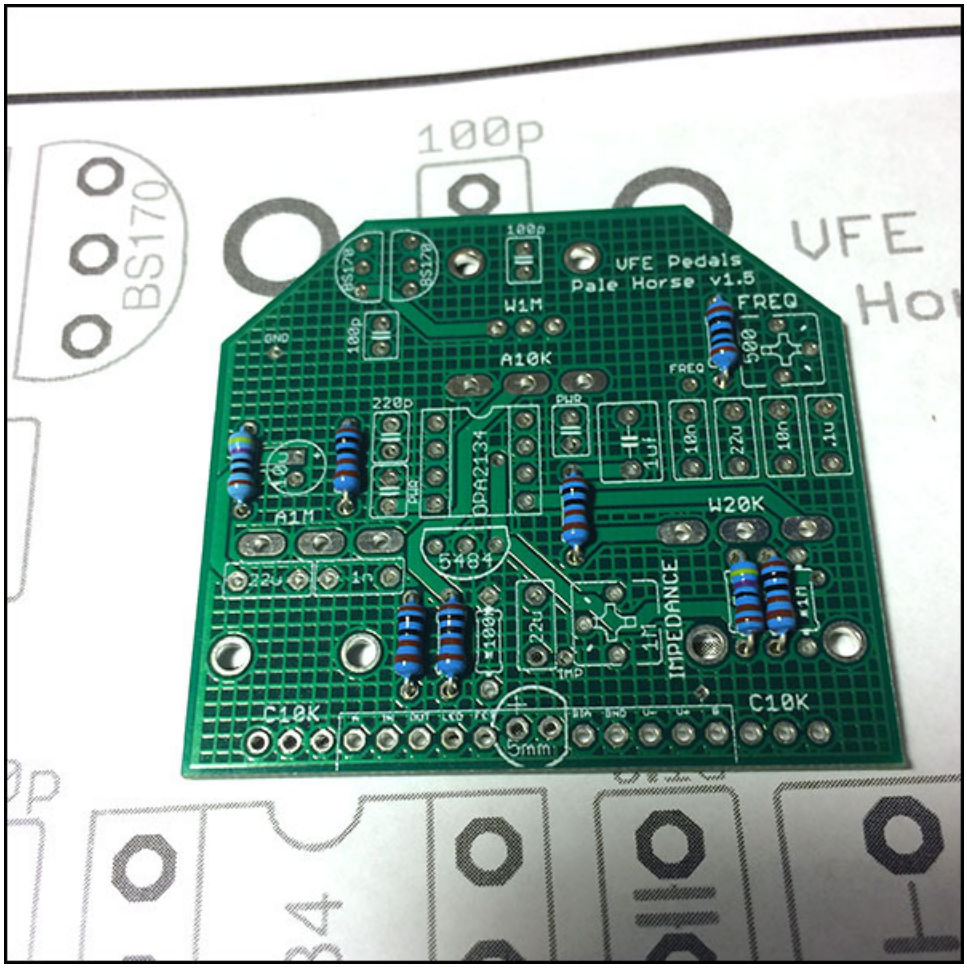

Fig. 2

There are different ways of stuffing and soldering circuit boards. Personally, I usually put all the components on and solder everything at once. Some people like to go by groups (resistors, then caps, diodes, etc) and still more that (ugh) solder one component at a time. Ain't nobody got time for that! Whatever style you do is fine…as long as you practice good soldering technique and keep focused on the task.

I'm going to stuff this entire board first but show you the steps. Here I've started with the resistors. I've left two resistors blank – these are potential mods as described in the PaleHorse build doc and at this point I wasn't sure what I would do with them.

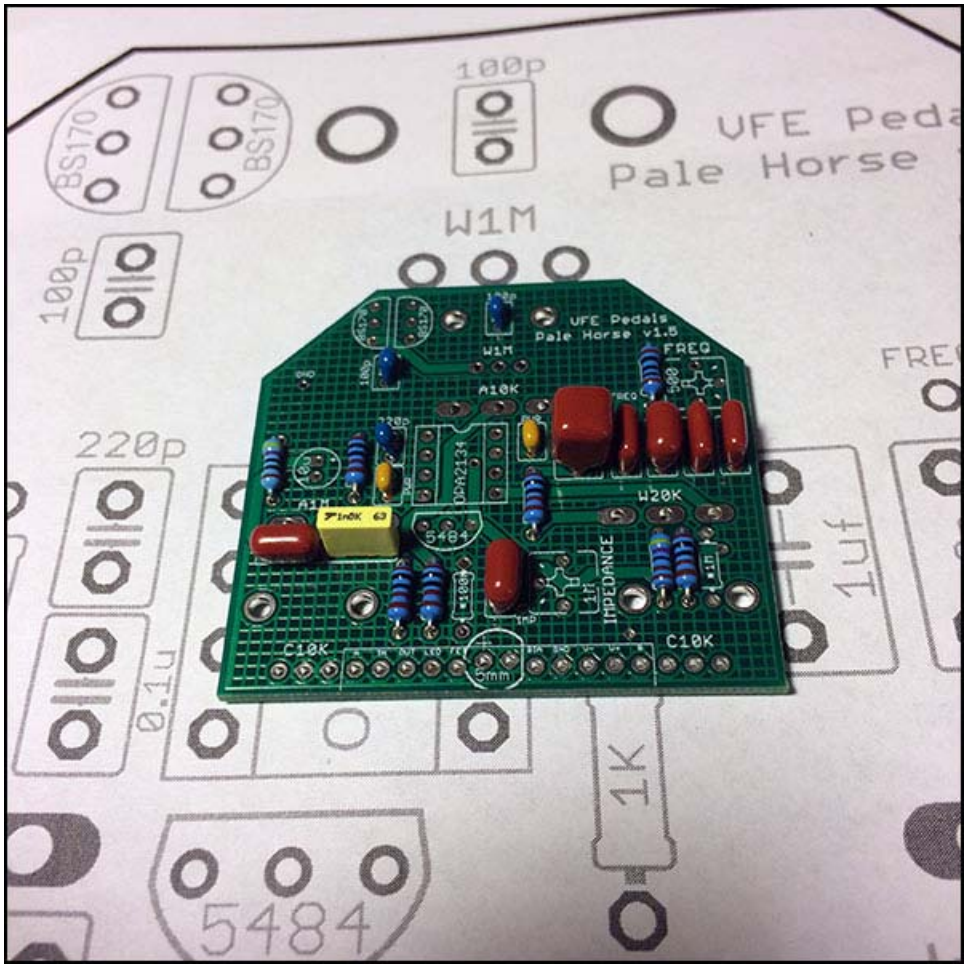

Fig. 3

Next up are the film caps and MLCC. I'm using mostly Panasonic ECQ-V. These are no longer made but I bought a decent stock of them when they were obsoleted. Since I don't have them in values under 10n, I've used an old AVX 1n box cap where it was called for.

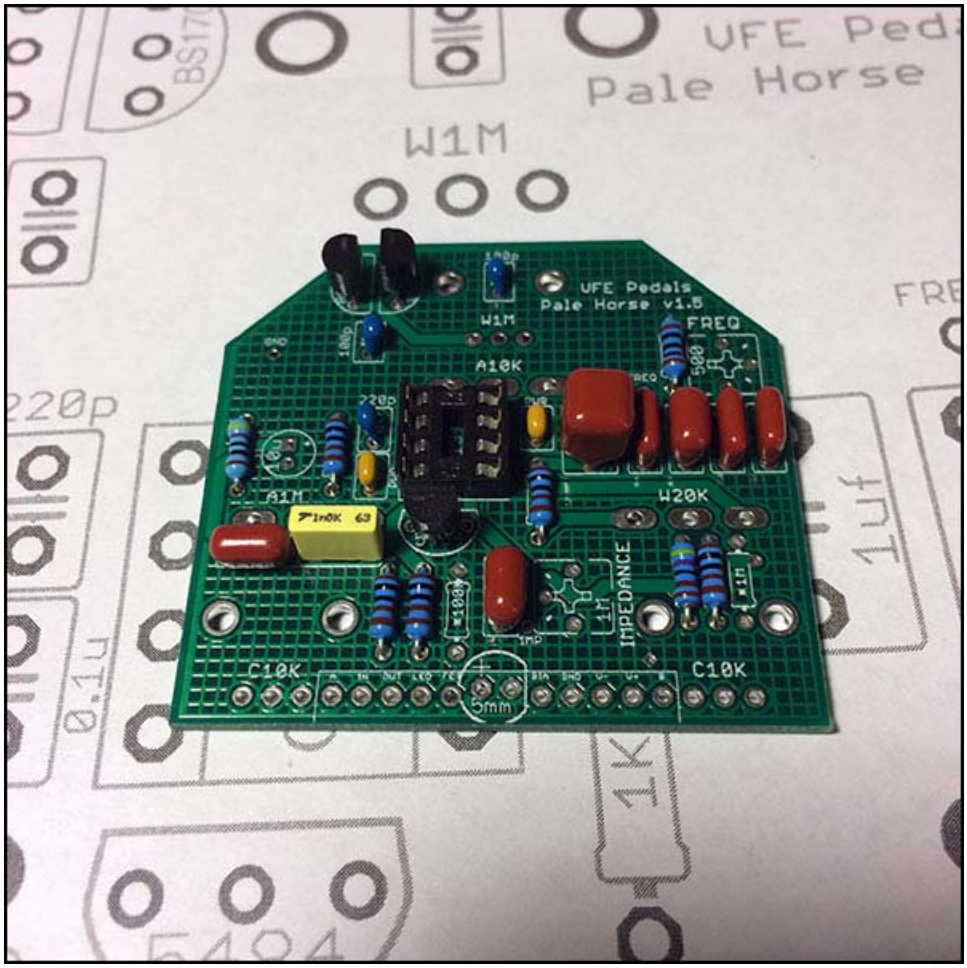

Fig. 4

The transistors and IC socket are next. I almost always use a socket for IC's. These are ghetto sockets from Tayda but I never have a problem with them…plus they are cheap!

Later on I changed my mind and socketed one transistor (the 2n5484) because I did not have the correct part. I'm glad I did because the J112 I had on hand (even though very close in spec to the J113 sub the build doc mentions) turned out not to work correctly in this circuit (it caused a sputtered output from the gain stage). That's okay – it worked enough to verify the build was correct and later on I got the right part.

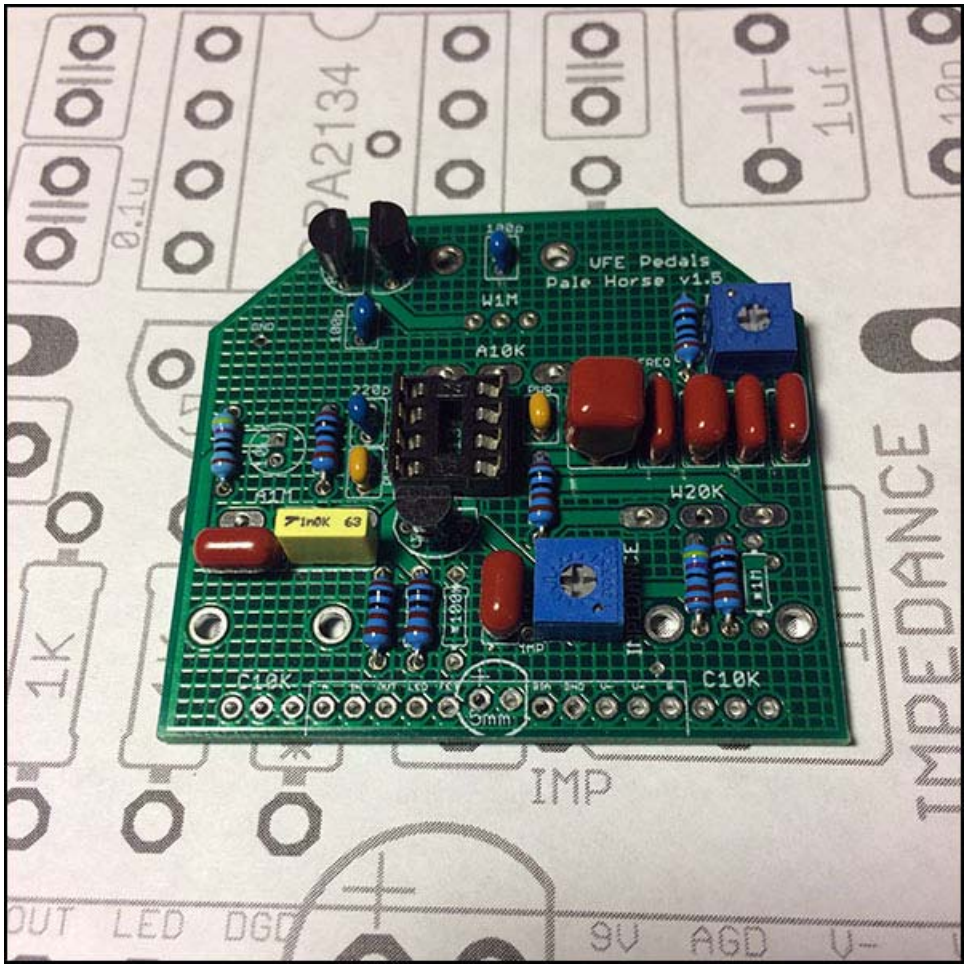

Fig. 5

There are two trimmers in the Pale Horse for tweaking the input impedance and frequency center of the tone pot. These are Bourns 3362p which is what I used in all my builds. BTW – Bourns makes a version of these with a little shaft on it so you can use your grubby guitar fingers to adjust them instead of a screwdriver. They have the suffix “TLF” instead of just “LF” on Mouser:
http://www.mouser.com/Passive-Components/Potentiometers-Trimmers-Rheostats/Trimmer-Resistors-Through-Hole/3362-Series/_/N-9q0yt?P=1yzv7fy&Keyword=TLF&FS=True

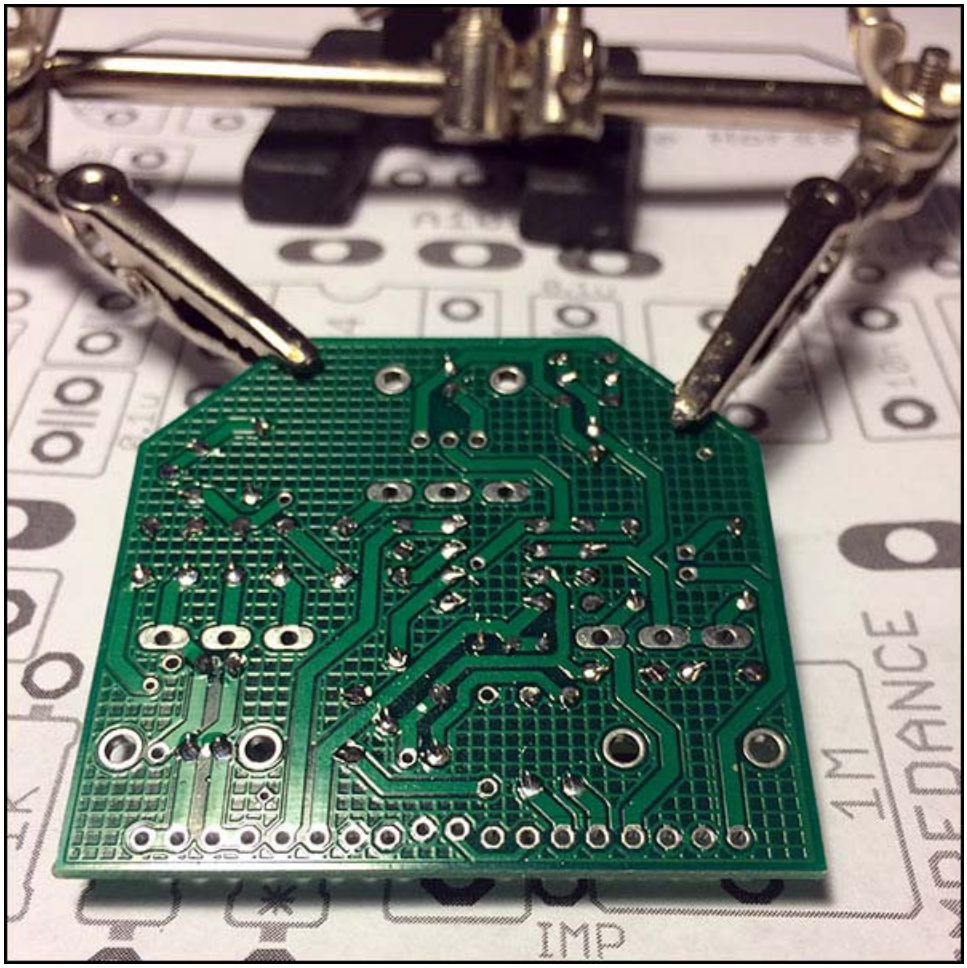

Fig. 6

Time to solder! I use some Helping Hands to hold my PCBs. They are cheap and do the job. I've bought a couple of small vises for PCB soldering but I always come back to the Hands.

You may have noticed I did not stuff the one electrolytic cap called for on this build. That's because it is a 10uF Bi-Polar and I did not have any of those when I did this build. That's okay – I'll show my workaround next.

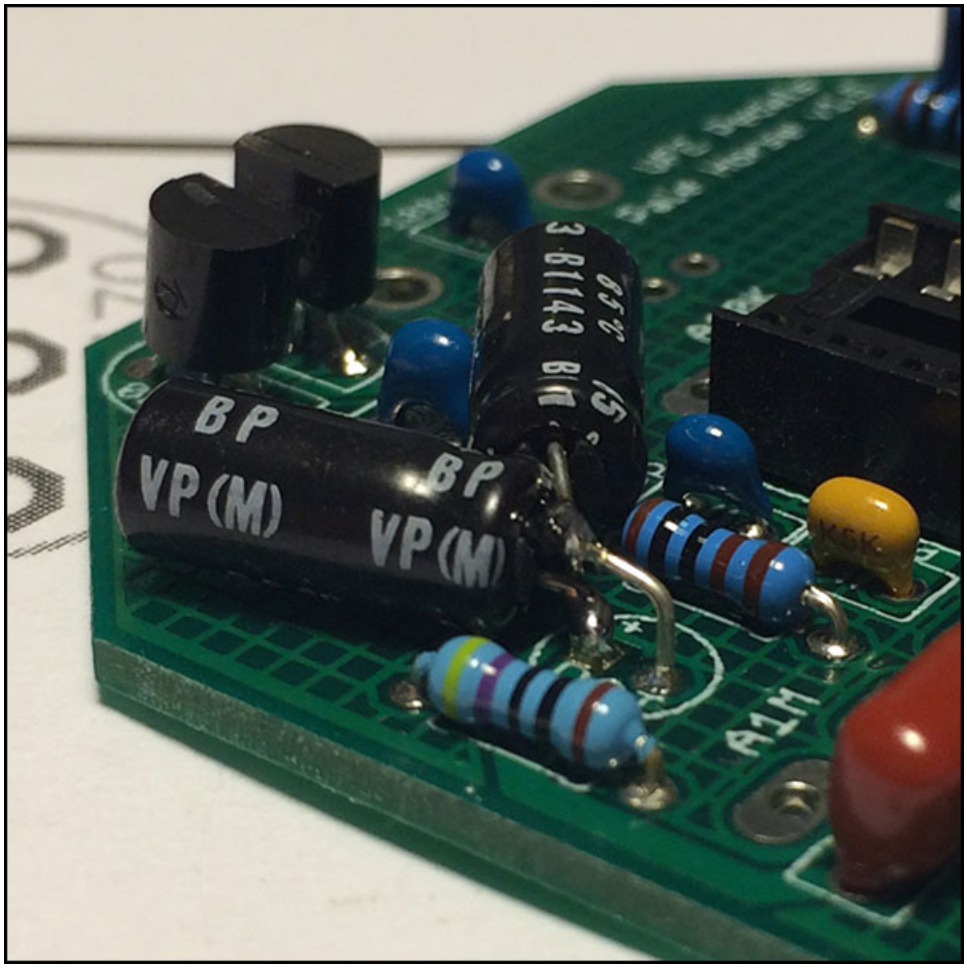

Fig. 7

Since I did not have the 10uF BP cap, I had to decide on a sub. At first I thought I might be able to get away with 4u7 BP (which I happen to have) but after plugging in some numbers to my LP/HP calculator (I have it saved in an Excel file) I realized that wouldn't do. With only a 4u7 cap when the Fat control turned all the way up the corner frequency of the added bass would be around 68Hz. With a 10uF it would be 33Hz – definitely within the audio range.

So, I had to come up with another idea. Since I had the 4u7 BP I decided to solder two of them in parallel. This yields a 9.4uF Bi-Polar cap (within tolerance) which was close enough. I bent the leads of one into place and soldered in the first cap. Then I carefully soldered the second cap to the first. Looks totally DIY!

BTW: I could have just as easily used polarized 4u7 caps here and soldered them + to - on each leg. That would make a BiPolar cap of about 9u4.

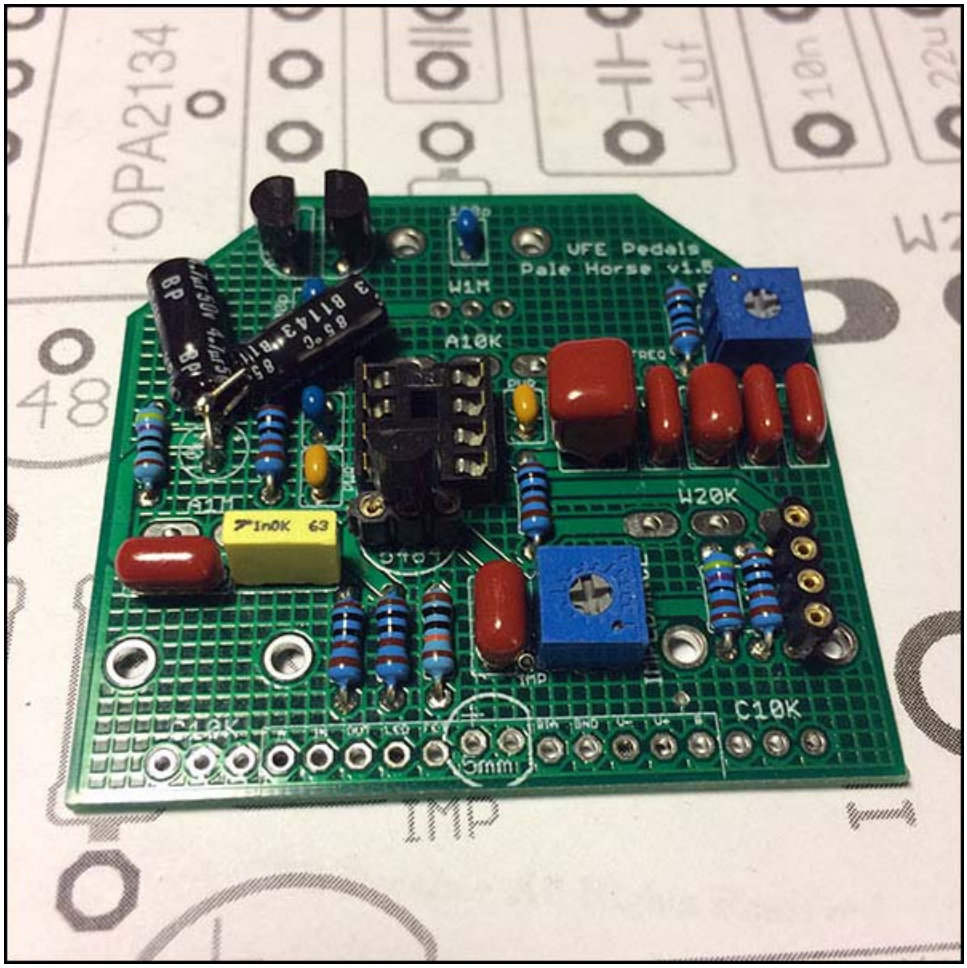

Fig. 8

I decided I would just stick with a 100k in that one mod spot. I figured (correctly) that I would be happy with this value as the minimum with the Impedance trimmer. I socketed the 1M resistor spot because I wasn't sure if it would be needed (turns out it wasn't). I also desoldered my transistor sub I mentioned earlier and put a socket in there instead. Good call!

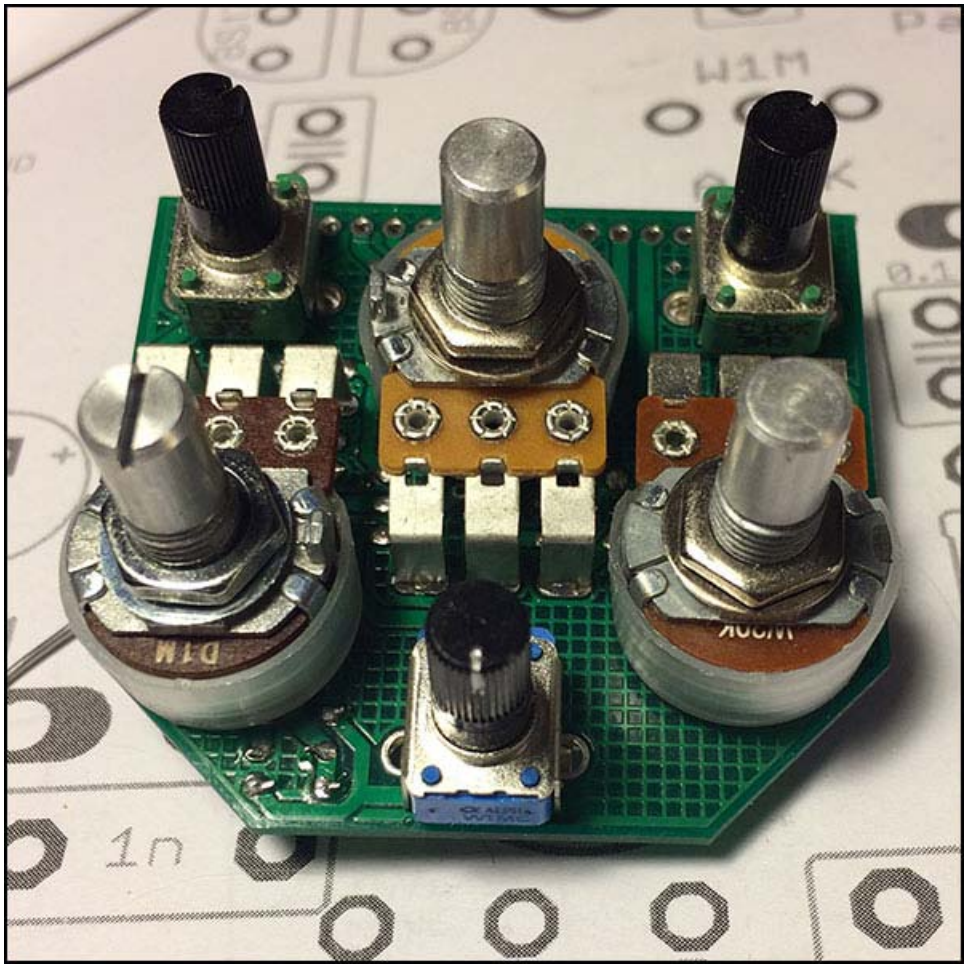

Fig. 9

With all the components on might as well do the pots, too! Well, not really. The smart thing to do here would have been

- Drill my enclosure
- Mount the 16mm pots on them and then solder them to the PCB
- Load the 9mm pots, check their clearance with their respective drill holes and soldered them in place on the PCB

However, at this point I had not even made the drill template for these projects and I was anxious to get going. Turns out everything aligned okay, but I would recommend doing those steps.

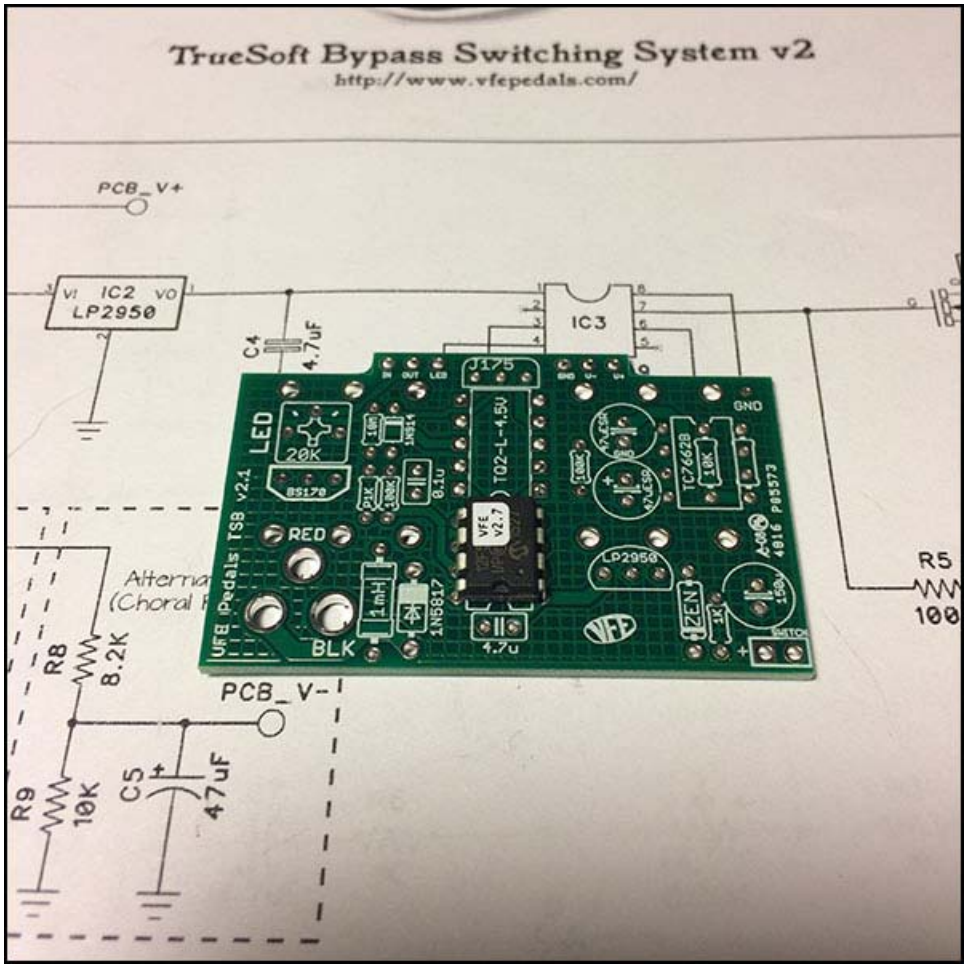

Fig. 10

OMG – I'M SO BOARED….ISNT THIS DONE YET?
Nah, still a bit to go. Time to build the switching board. This is required for all the VFE projects, (although the specs are different on some of them). The PaleHorse uses a split rail (+ and -9v) for operation so we'll build up the whole thing for this project.

I've already loaded in the micro-controller (this is how you get the PCB, too) without a socket. A socket is not used for these due to low clearance when the Switching Board is mounted in the enclosure. More on that later.

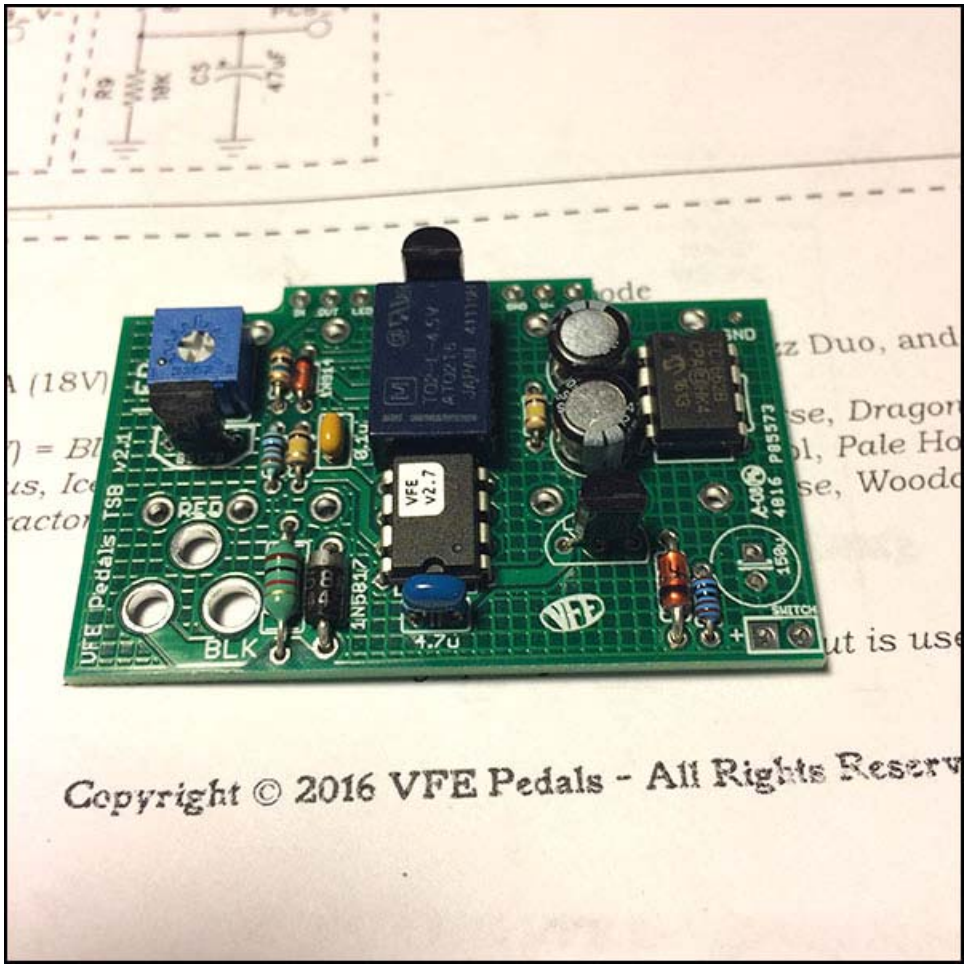

Fig. 11

I decided to just load everything up in one go here. It's a small PCB and there's no point in putting you through the drudgery of step by step stuffery now. I would have preferred to use metal film for the 100k and 10M but I just didn't have them. Not a big deal, but I do stick to metal film for just about everything, these days.

You can see I've also loaded the 7662 inverter for the split-rail power, also without a socket. The top of that relay is pretty much right at the "I'm barely clearing the enclosure lid" point so that gives you an idea of why sockets might cause problems. However, low-profile sockets do exist and could be used here. I just don't have any. The transistors will get folded down later for clearance (as Peter recommends).

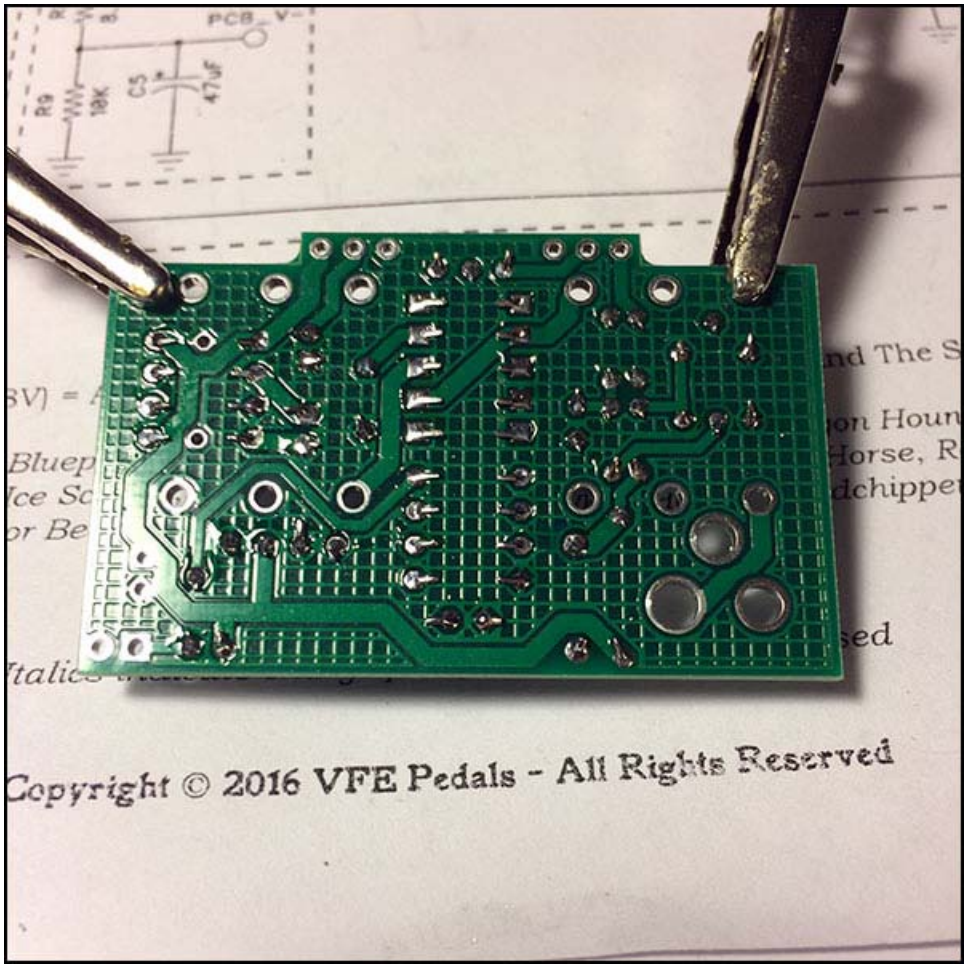

Fig. 12

Since I really haven't talked enough about IC sockets yet, let's talk some more about IC sockets. Or, the absence thereof. When you solder IC's directly to a PCB, it's best to do one leg at a time (or two if you do them on opposite ends). This prevents the chip from getting too hot. I tend to solder one leg then solder a few other components while the IC cools off. Then I solder another leg and repeat. Some people like to use heat-sinks and that's fine. I am a very, very lazy builder so I always go for best result with least effort (something you get in the habit of after you build a few thousand PCBs).

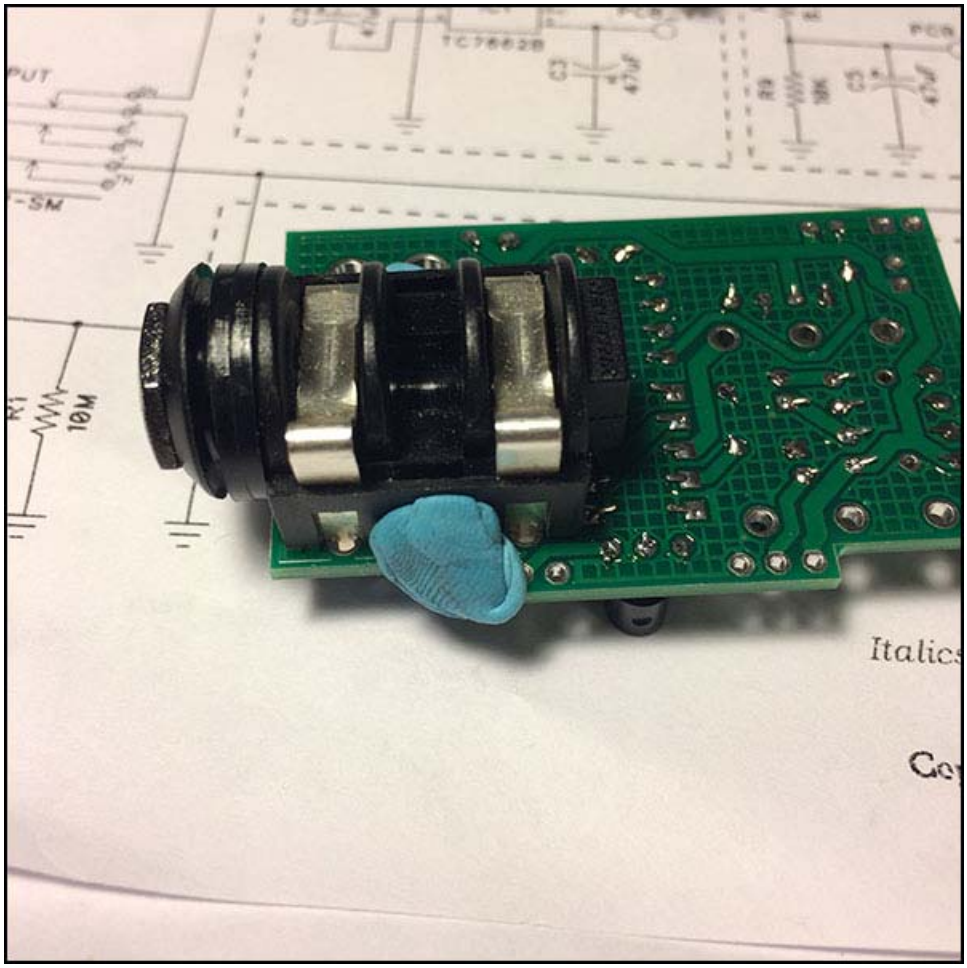

Fig. 13

Now it's time to tackle the bottom-mount components. The Switching board has two different types of components that go on the bottom of the PCB – the jacks and the 150uF decoupler. Again, the smart thing to do here would be to load the input and output jacks on the enclosure then fit the PCB and solder it in place. But, I wasn't at that point yet so I broke off a chunk of my oft-used Blue Tac to hold the jacks in place while I soldered.

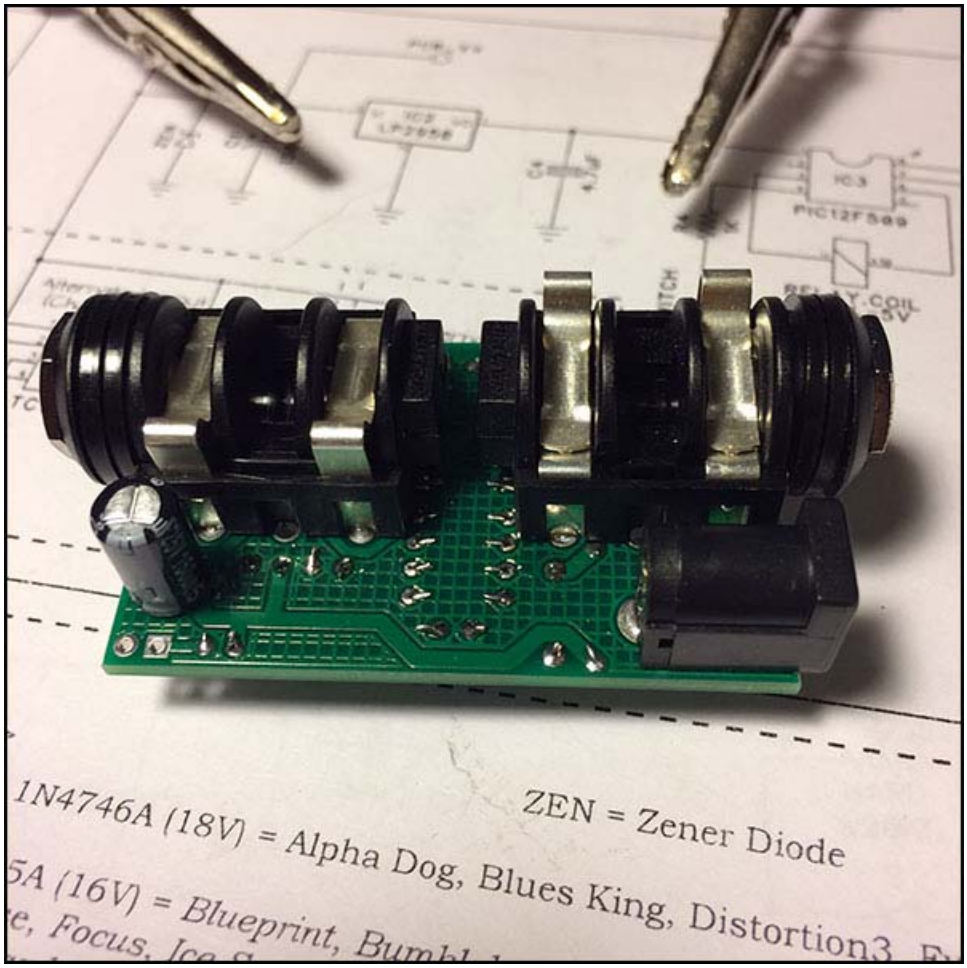

Fig. 14

Well, it turned out that this was a huge mistake. The reason is the DC Jack. The DC Jack has minimal clearance with the enclosure side-wall and if it sits just a little bit proud it's very tough to get the whole assembly mounted without forcing it. I did not know that at this point and I ended up having to de-solder this DC Jack, screw in the assembly to the enclosure and re-solder the DC Jack to get it right. Unfortunately, I do not have pictures of this process. However, if you've already read the Switching Board documentation you know that Peter made a video for us on how to properly build this stuff including how he does his DC Jacks. Please watch that instead of doing what I did here!
https://www.youtube.com/watch?v=vAvK-yB_29M

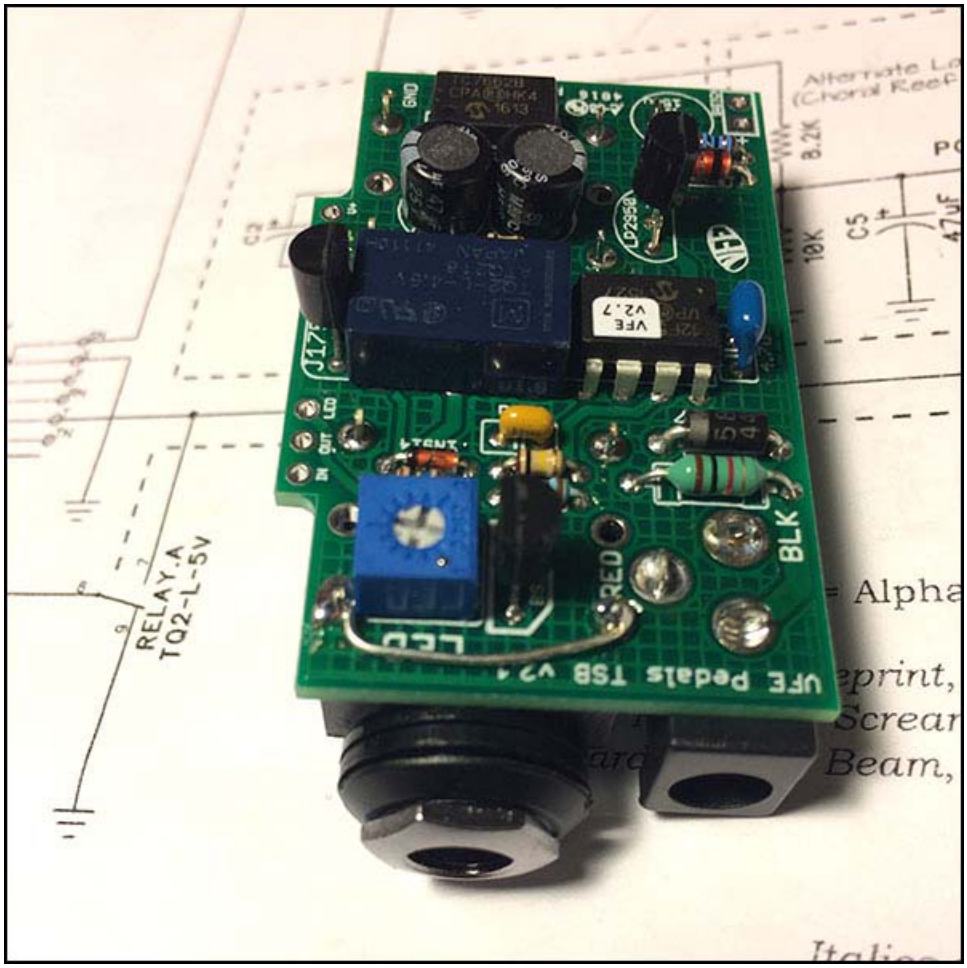

Fig. 15

Since I am not using the “sleeve make” jack spec’d for these builds a jumper is necessary to complete the input grounding. I go into a lot of detail in the Switching Board documentation on the ¼” jacks used in the VFE Projects so please read that if you haven’t already.

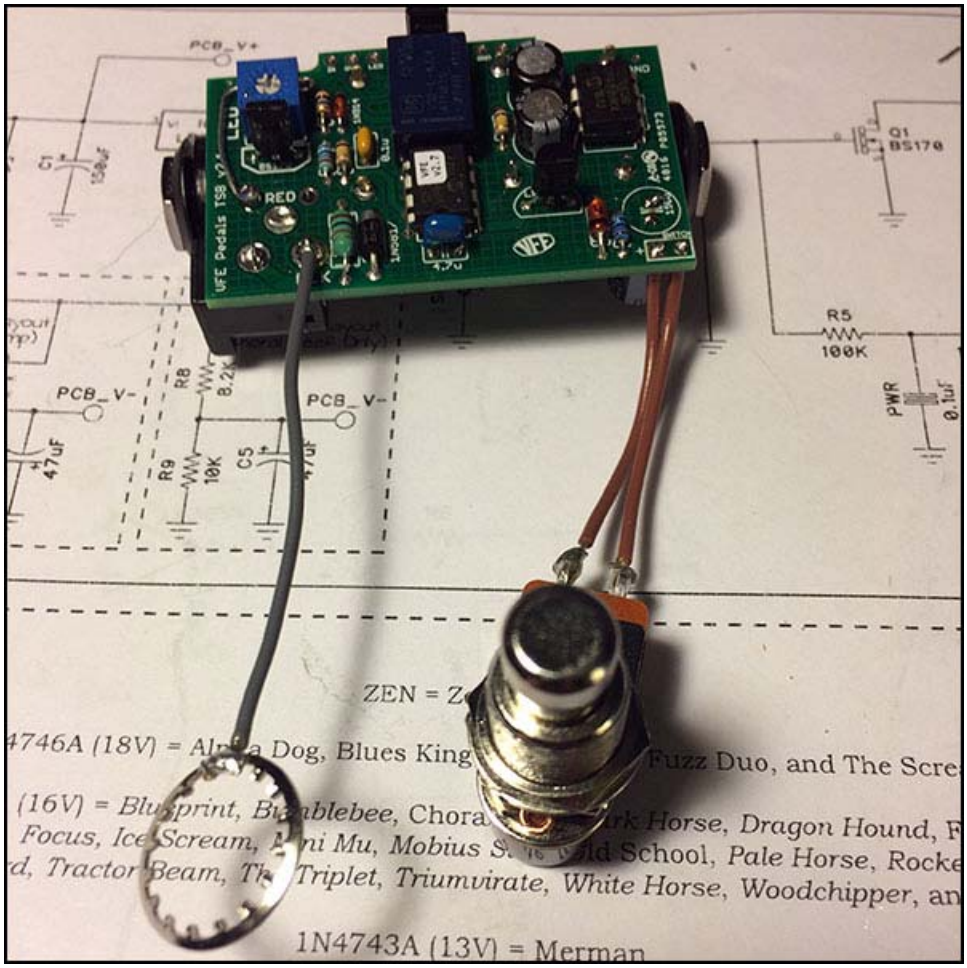

Fig. 16

Another departure from the VFE process: I'm using a bit of wire soldered to the lock washer to do the enclosure grounding instead of the method Peter describes in the video linked above. Either way will work – this is just my preference. Again, more details in the Switching Board documentation.

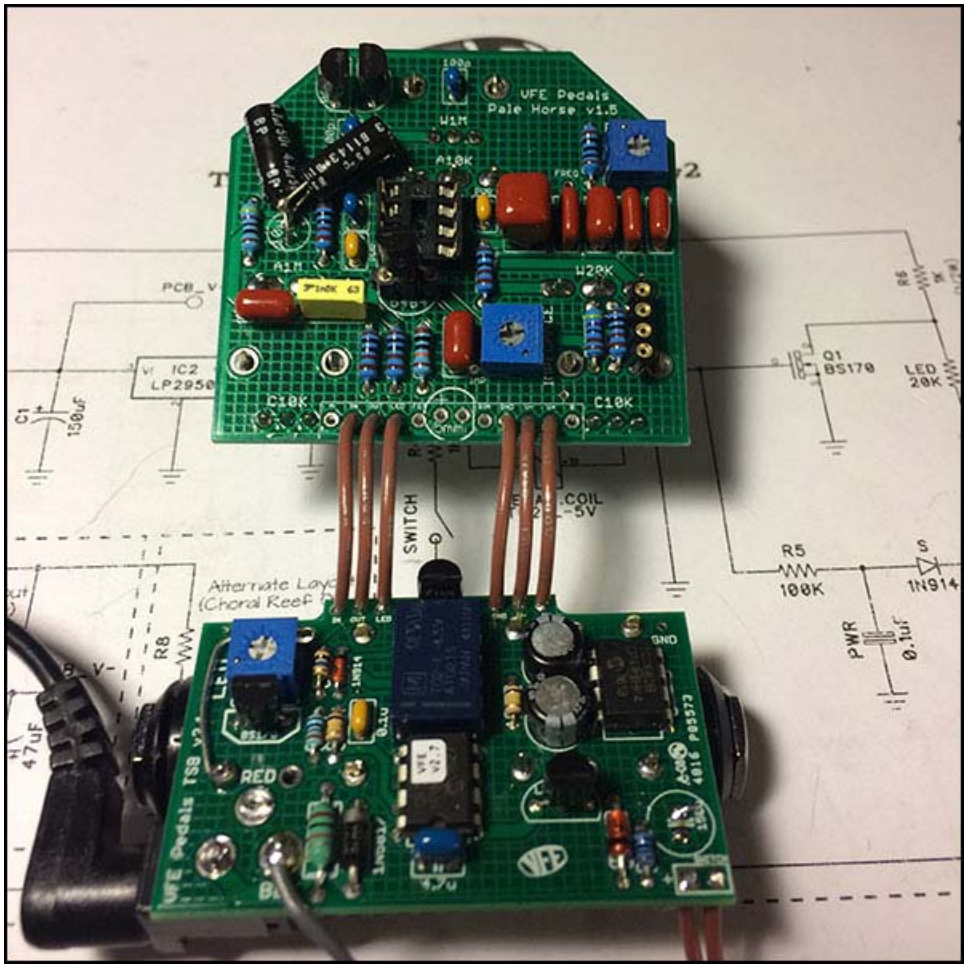

Fig. 17

Sweet! Getting really close now. I've done the wiring between the two boards to complete the circuit and plugged in the power. At this stage I have not loaded in the OPA2604 IC. First I am going to use my DMM to test some voltages (not shown). What I am looking for is that I have -9v on pin4 of the IC and +9 on pin8. It's always a good idea to confirm proper power before putting in an IC. Incorrect voltages can cause an IC to heat up and potentially damage it.

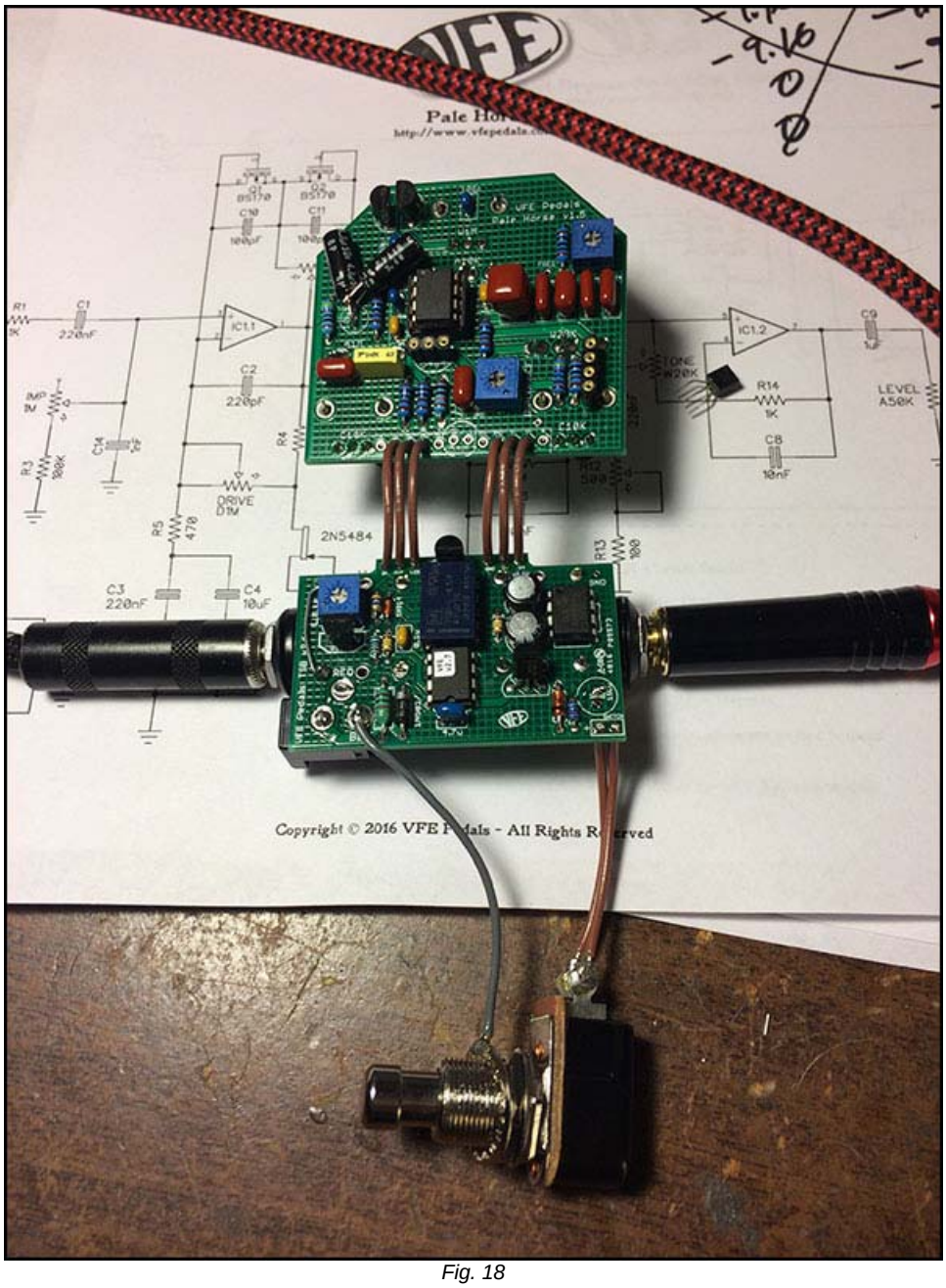

Fig. 18

My voltages seemed correct so it was time to test this thing out. Success (mostly)! The build was working, audio was being passed in bypass and effect mode and things were generally what they should be. The one exception was my transistor sub. At this point I realized it was causing sputtering and weird artifacts (esp. with the Comp pot in the center), so I removed it from the socket. I sat and stared at this build for 5 full days while I waited for the proper transistor to arrive. Really!

Fig. 19

Once the proper transistor arrived I was able to confirm that the circuit worked 100% so I moved on to the enclosure drilling and artwork. Unfortunately, I didn't take any pictures during this process but you most likely already know what's involved in this (assuming you have built a few pedals before). If you have followed all the tips up to this point it should be easy enough to load your build in an enclosure….that's actually the only step to assembly. There are no other wires to attach or much else to do.

Oh wait- there is one thing! The LED indicator. You may have noticed I never soldered one in. I waited until assembly for that part. I simply loaded the LED loose into the PCB, assembled the pedal with all the pots and jacks screwed down, then moved the LED into place and soldered it directly to the board. That was all there was left to do ☺

Fig. 20

Obligatory guts. This went together surprisingly well for the first crack at building a VFE production pedal. There were a few headaches along the way which I have already described, but if you keep those in mind you should have an even easier time than I did. Just remember; there is a good way to do things and an evil way to do things. And, evil always wins because good is dumb.