

The Smart Switch

Designed for the Line 6 DL4

Installation Guide

One must be careful when working with an expensive pedal such as the Line 6 DL4. It is important to read through the entirety of this manual before attempting the modification! If the worst happens and you break your pedal, you have my deepest sympathies as I have been there many, many times. However, we have to assume zero responsibility for the mishap due to the nature of the modification. If you do run into a problem, please feel free to email me at joe@schoolcraftspecialties.com. I will do the best I can at responding quickly, however, be aware that I have a job and I am a full time student outside of this small enterprise so it may be a few days before I can get around to answering your questions. Thanks for your patience!

Okay here we go...

What You Need:

Tools:

- Adjustable Wrench
- Needle Nose Pliers
- Soldering Iron
- Multimeter
- No. 2 Phillips Head Screwdriver.
- Flathead Screwdriver (for prying)
- Drill, 5mm (13/64") and 1/2" Drill bit

Supplies:

- 60/40 Lead Solder
- 22 Gauge Wire
- Parts Cup
- Electrical Tape
- Super Glue (Not super necessary)

Parts List:

- 3.3k Resistor – 2
- 10k Resistor – 1
- 22k Resistor – 1
- *68k Resistor – 2
- .1uF Capacitor – 1
- 47uF Capacitor – 1
- RGB LED – 1
- 78L05 Voltage Reg. – 1
- ATtiny85 – 1
- 8 DIP IC Socket – 1
- Switching MOSFET – 2
- Mo. N.O. Foot Switch – 1
- LED PCB – 1
- Smart Switch PCB – 1

**For Volume Drop Modification*

Step One: *Assemble the Smart Switch PCB.*

It's best to start the mod by assembling the Smart Switch PCB before you take anything apart. Go ahead and get your soldering iron fired up and locate all of the abovementioned parts. **Note:** Leave some room to bend C1 and the 78L05 voltage regulator over so they don't hit the top of the case (pictures don't show this step).

1. Install the four resistors on the backside of the PCB where their names are visible. (Figure 1.1)
The values are as follows:
 - R1 = 10k
 - R2 = 22k
 - R3 = R4 = 3.3k
2. Flip the PCB over and install C2, the .1uF capacitor. Note, this cap will be in a little different position than seen in Figure 1.2 due to a board update.
3. Next install 8 DIP IC Socket where it says ATTiny85. Make sure to get the orientation correct!
4. Install the LM7805 Voltage Regulator (Q1).
5. Install MOSFET1 & MOSFET2 (both are SiHU5N50D switching MOSFETs; note the orientation).
6. Install C1, the 47uF capacitor (the positive leg (long leg) goes in the square pad).
7. Install the ATTiny85 chip; note the orientation! (See Figure 1.2 for the finished PCB.)

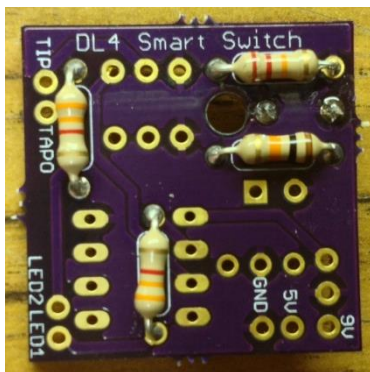


Figure 1.1

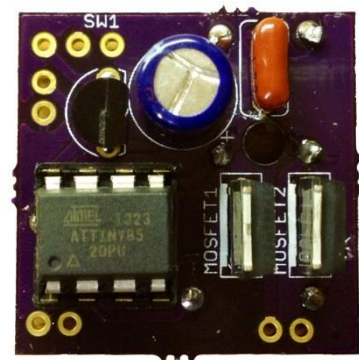


Figure 1.2

Step Two: *Take the pedal apart.*

This step seems pretty obvious, but there are a few tricks that you will need to make sure you get the motherboard out of the pedal without tearing anything up. Follow the steps below.

1. Remove the knobs from the shafts of the potentiometers using your fingers/Flat Head Screwdriver.
2. Remove the 5 plastic nuts off of the input/output jacks on the backside of the pedal.
3. Remove the 5 Phillips head screws from the back plate and remove the back plate.
4. Remove the battery clips from the pedal (be careful not to break the wires off of the PCB; this is very easy to do!).
5. Remove the 4 Phillips head screws and the 1 stud screw from the PCB/Case assembly.
6. Pull the PCB from the case. (Note that it is not necessary to remove the four foot switches unless you have had them replaced with aftermarket switches.)

Step Three: *Drilling the case*

This step is rather important to get right the first time. You can put the new footswitch and indicator LED anywhere you want on the case so long as there is room for it on the insides with the PCB in the case and everything. These steps outline how I normally do it.

1. Place two dots on the case, one about 1/2" to the left of the Line 6 logo and the other 3/8" to the right of the logo at the top of the case, centered vertically on the logo. (Figure 3.1)
2. Drill a 1/2" hole for the footswitch on the left dot. (Figure 3.1)
3. Drill a 5mm (13/64") hole on the dot opposite of the 1/2" hole for the indicator LED. (Figure 3.2)



Figure 3.1

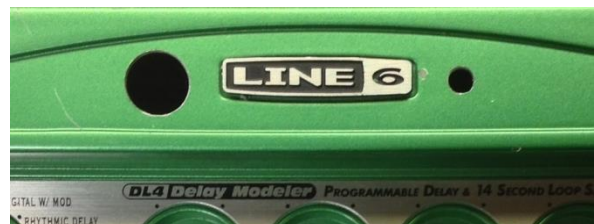


Figure 3.2

Step Four*: *Volume Drop Mod*

This step is only necessary if you are performing the volume drop mod on your unit. If you are not doing this modification skip this step and proceed to Step Four.

1. Locate R36 and R37 on the DL4's PCB. It should be sort of in between the two output jacks by the third potentiometer from the right. (Figure 4.1)
2. Bend the legs of the 68k resistors so that the bodies of the resistors are vertically oriented. Cut a portion of the resistors' legs off so that you have just enough to work with. (Figure 4.2)
3. Install one 68k resistor in parallel with R37 and the other in parallel with R36. I usually glob some solder onto the pads of both of the resistors before I try and solder the 68k resistors on. (Figure 4.3)
4. Wrap electrical tape/heat shrink wrap around the resistors to insulate them. Also make sure that they aren't standing taller than the body of the potentiometer that they are located next to. I usually carefully bend the resistors over a bit. (Figure 4.4)

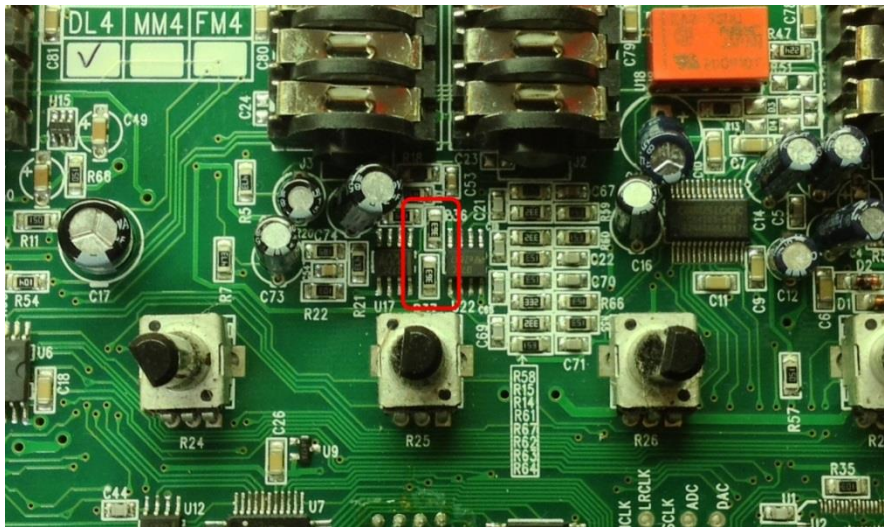


Figure 4.1



Figure 4.2



Figure 4.3



Figure 4.4

Step Five: *Populate the LED PCB*

1. With the letters on the PCB facing up, install the RGB LED with its flat side closest to the hole in the PCB in the holes labeled without the "W". (Figure 5.1)

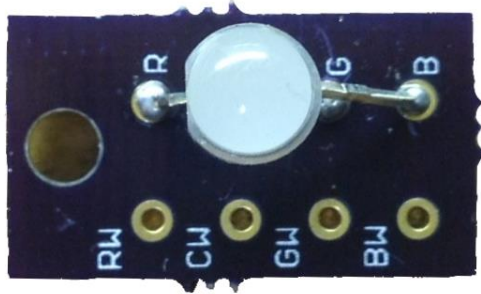


Figure 5.1

Step Six: *Wire up the Smart Switch PCB (SS PCB) to the DL4 Motherboard*

You are almost there! It's down to connecting a few wires. Be careful not to press too hard when soldering to the back side of the DL4 PCB because it is very easy to bend the LEDs on the other side. Also, notice that I also come from underneath the SS PCB with the wires and solder on the top side of the board. This helps keep the insulation of the wires in-between the two boards. However, some people have had issues with the components hitting the bottom plate of the case, so you may want to come from the top side of the PCB with the wires instead.

1. Locate the SS PCB target location on the back side of your DL4. I usually place it on the bottom right-hand side in-between switch 1 and switch 2 (the labels for the switches are on the top side of the DL4 PCB right above the plastic pieces on the switches). Note that the hole in the SS PCB will line up with the screw hole in the DL4 PCB. Make sure that all wires can reach from their spot on the DL4 PCB to the screw hole. (Figure 6.1)

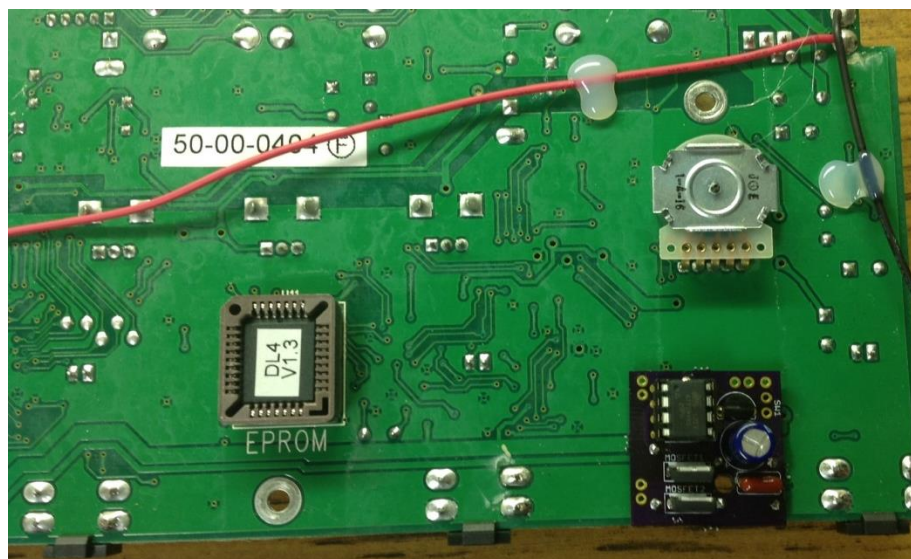


Figure 6.1

2. Flip the DL4 PCB over and find the Expression Jack (it will be the jack farthest to the left by a large capacitor and the power jack). Bend the three tabs up so that they are no longer in contact with their respective terminals. (Figure 6.2)

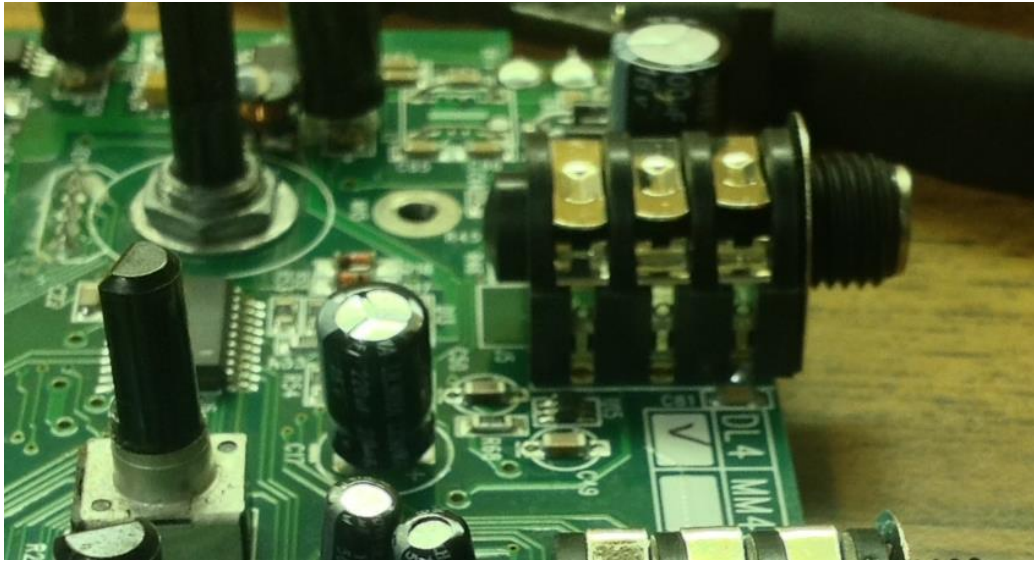


Figure 6.2

3. Flip the DL4 PCB back over and locate the solder pads for the Expression Jack. Jumper the Right Sleeve and Ring pads together with a jumper wire. Attach a wire from the Right Tip pad to the "TIP" pad on the SS PCB (the labels for the pads are on the back side of the SS PCB). (Figure 6.3)

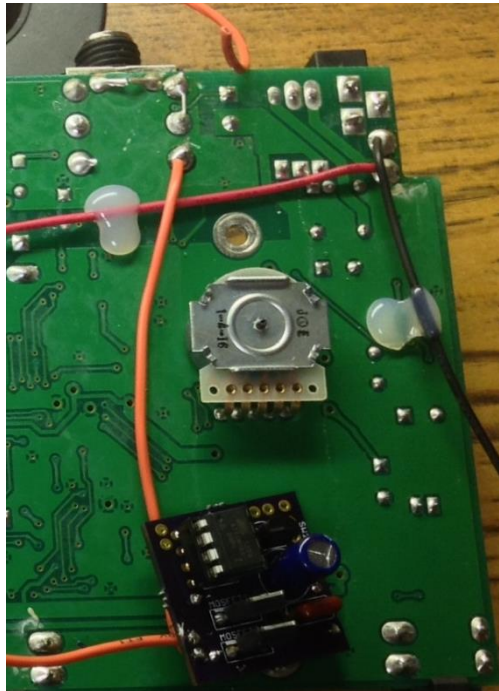


Figure 6.3

4. Connect a wire from the top right corner pad of switch 4 to the pad on the SS PCB labeled "TAPO" (it's right next to the "TIP" pad). (Figure 6.4)

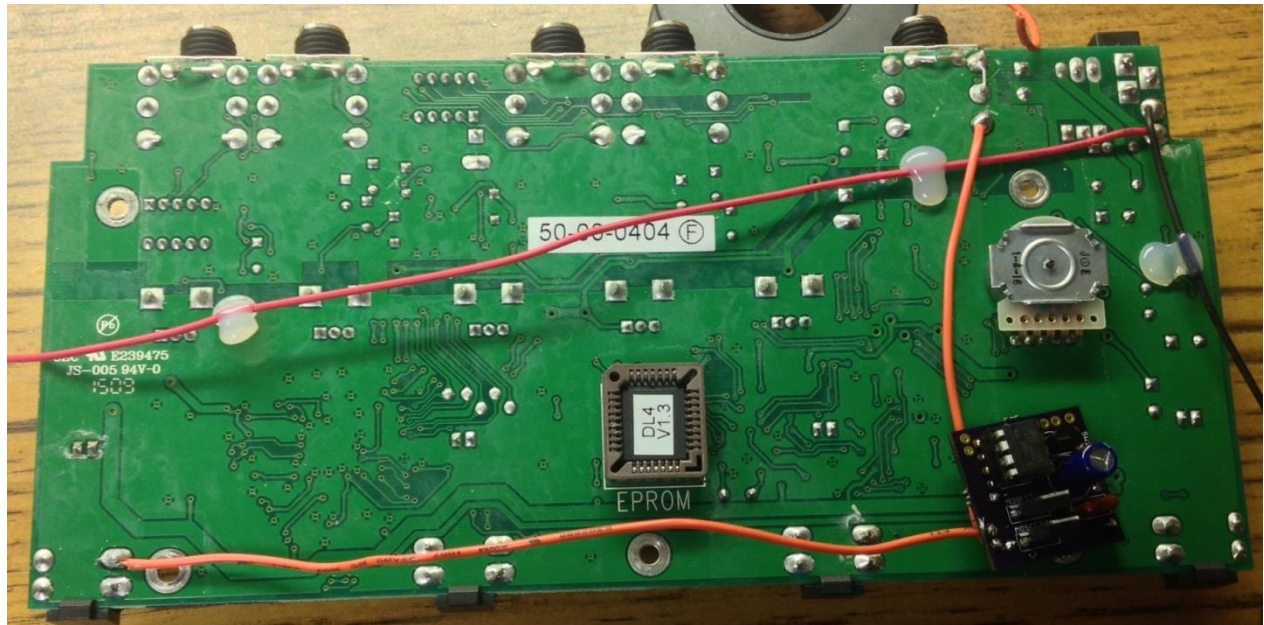


Figure 6.4

5. Connect a wire from the positive (red) battery terminal on the backside of the DL4 PCB to the pad on the SS PCB labeled "9V". (Figure 6.5)

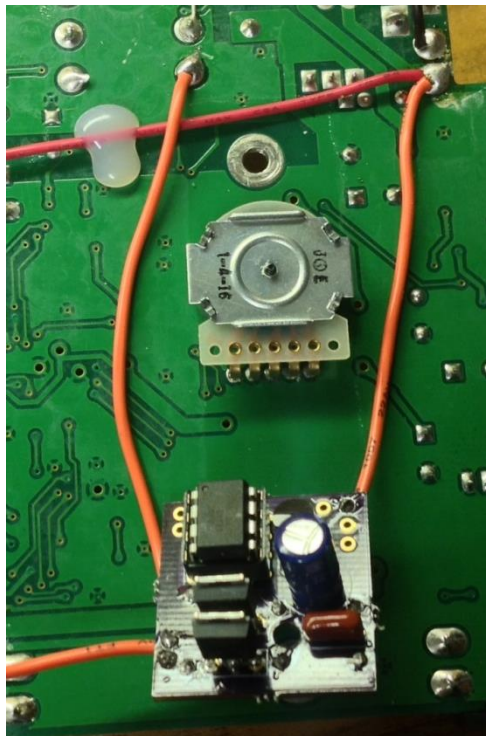


Figure 6.5

6. Connect a wire from the bottom left solder pad of switch 1 on the DL4 PCB to the "GND" solder pad on the SS PCB. (Figure 6.6)

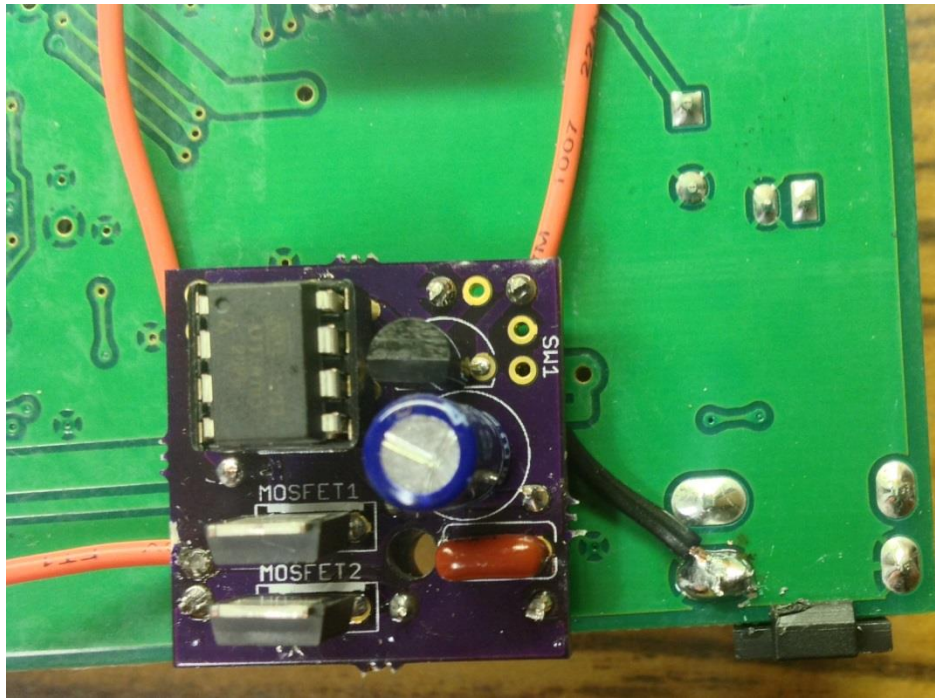


Figure 6.6

7. Next, connect wires to the "LED1", "LED2", and "5V" pads on the SS PCB. Make sure that they stretch about 1.5" past the edge of the DL4 PCB in-between the input and output jacks. (Figure 6.7)

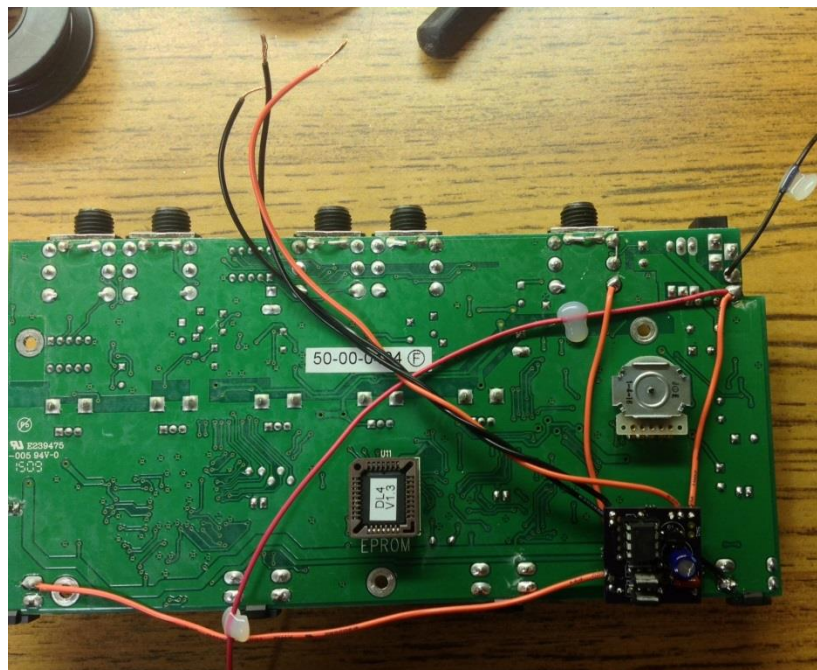


Figure 6.7

8. Attach the wire coming from the “5V” pad on the SS PCB to the pad labeled “CW” on the LED PCB. Now, choose what two colors you want to indicate your different presets (RW = Red, GW = Green, BW = Blue) and attach the wire coming from “LED1” on the SS PCB to one and the wire coming from “LED2” on the SS PCB to the other. As you can see in the picture, I chose Blue (BW) and Red (RW). (Figure 6.8)

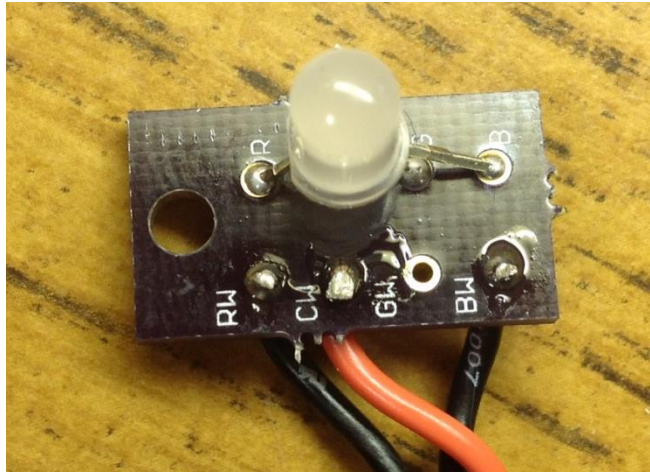


Figure 6.8

9. Last step! Attach two wires to the pads labeled “SW1” on the SS PCB and solder them to the Momentary Normally Open footswitch. Make sure they stretch about 1.5” past the edge of the DL4 PCB in-between the expression jack and the output jacks (it doesn’t matter which wire goes to which pad/tag on the footswitch). (Figure 6.9)

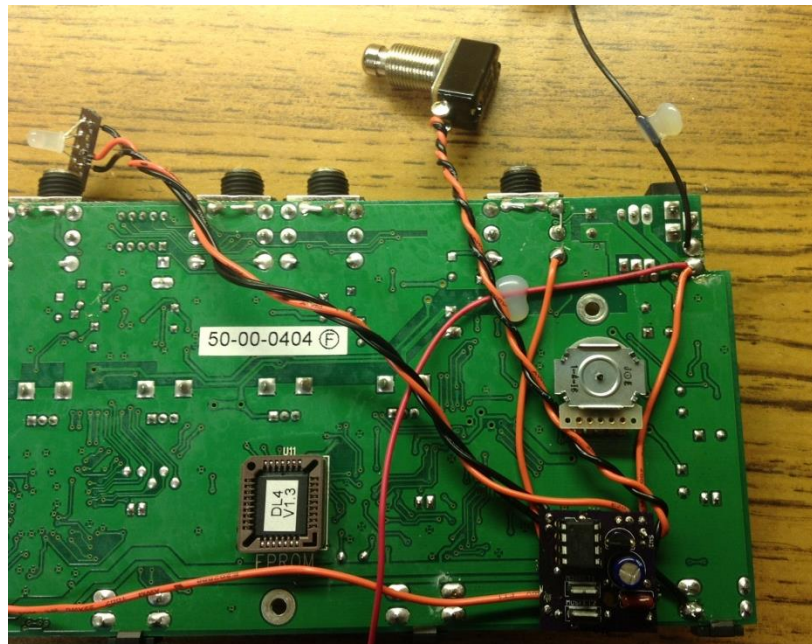
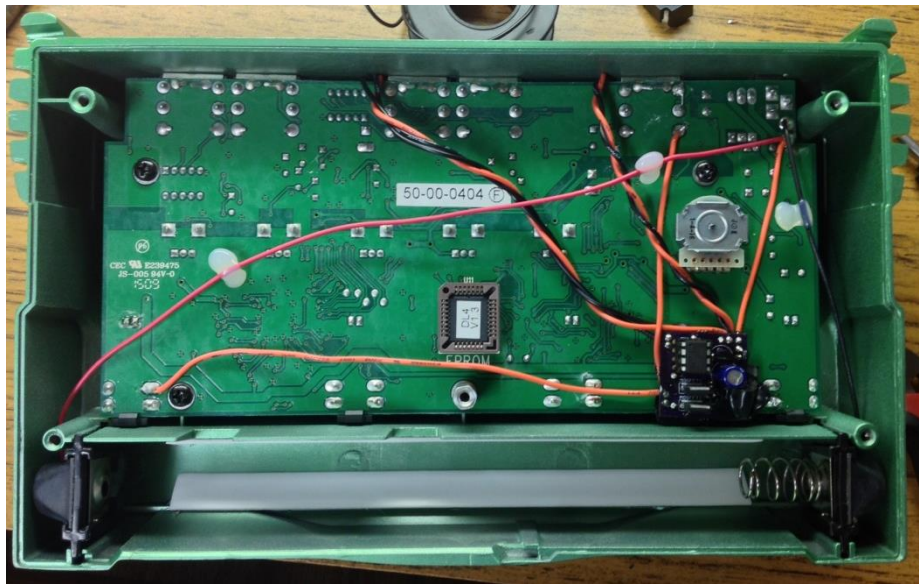


Figure 6.9

Step Seven: *Final Instructions*

Congratulations on making it this far! The hard part is over; all you have to do is place the DL4 Motherboard back into its case and tighten everything down. Before you do though, I'd recommend placing electrical tape on the backside of the SS PCB to prevent any shorts from happening. Use the screw/screw hole nearest the SS PCB in conjunction with the hole in the SS PCB to hold the SS PCB in place. The hole in the SS PCB is a tight fit, and you may have to run the screw through it a few times in order for it to work properly. Don't tighten the screw down very tightly because it can cause a short if you do. If you put the pedal together and the mod isn't working, try loosening that screw a smidge until you see the indicator LED come back on. Beyond that, installation is pretty simple. I recommend either gluing or taping the LED PCB into place in the case. Also, I usually start the re-installation process with the LED PCB and the new footswitch going in first. ***Make sure that the tags on the footswitch aren't touching the case!*** I hold the switch still with a screwdriver while I tighten down the nut before I fully install the DL4 PCB. Take a look at the finished product picture below to get an idea of how everything should look. I hope you enjoy this mod as much as I have!



Finished Installation