

Build Your Own Clone British Blues Overdrive Kit Instructions



Warranty:

BYOC, LLC guarantees that your kit will be complete and that all parts and components will arrive as described, functioning and free of defect. Soldering, clipping, cutting, stripping, or using any of the components in any way voids this guarantee. BYOC, LLC guarantees that the instructions for your kit will be free of any major errors that would cause you to permanently damage any components in your kit, but does not guarantee that the instructions will be free of typos or minor errors. BYOC, LLC does not warranty the completed pedal as a whole functioning unit nor do we warranty any of the individual parts once they have been used. If you have a component that is used, but feel it was defective prior to you using it, we reserve the right to determine whether or not the component was faulty upon arrival. Please direct all warranty issues to: sales@buildyourownclone.com This would include any missing parts issues.

Return:

BYOC, LLC accepts returns and exchanges on all products for any reason, as long as they are unused. We do not accept partial kit returns. Returns and exchanges are for the full purchase price less the cost of shipping and/or any promotional pricing. Return shipping is the customer's responsibility. This responsibility not only includes the cost of shipping, but accountability of delivery as well. Please contact sales@buildyourownclone.com to receive a return authorization before mailing.

Tech Support:

BYOC, LLC makes no promises or guarantees that you will successfully complete your kit in a satisfactory manner. Nor does BYOC, LLC promise or guarantee that you will

receive any technical support. Purchasing a product from BYOC, LLC does not entitle you to any amount of technical support. BYOC, LLC does not promise or guarantee that any technical support you may receive will be able to resolve any or all issues you may be experiencing.

That being said, we will do our best to help you as much as we can. Our philosophy at BYOC is that we will help you only as much as you are willing to help yourself. We have a wonderful and friendly DIY discussion forum with an entire section devoted to the technical support and modifications of BYOC kits.

www.buildyourownclone.com/board

When posting a tech support thread on the BYOC forum, please post it in the correct lounge, and please title your thread appropriately. If everyone titles their threads "HELP!", then it makes it impossible for the people who are helping you to keep track of your progress. A very brief discription of your specific problem will do. It will also make it easier to see if someone else is having or has had the same problem as you. The question you are about to ask may already be answered. Here are a list of things that you should include in the body of your tech support thread:

1. A detailed explanation of what the problem is. (not just, "It doesn't work, help")
2. Pic of the top side of your PCB.
3. Pic of the underside of your PCB.
4. Pic that clearly shows your footswitch/jack wiring and the wires going to the PCB
5. A pic that clearly shows your wiring going from the PCB to the pots and any other switches(only if your kit has non-PC mounted pots and switches)
6. Is bypass working?
7. Does the LED come on?
8. If you answer yes to 6 and 7, what does the pedal do when it is "on"?
9. Battery or adapter.(if battery, is it good? If adapter, what type?)

Also, please only post pics that are in focus. You're only wasting both parties' time if you post out of focus, low res pics from your cell phone.

Revision Notes:

Rev 1.0 Omit the R10 100k resistor. [Click here for Rev1.0 instructions.](#)

Rev 1.1 Footswitch PCB assembly discontinued.

Copyrights:

All material in this document is copyrighted 2011 by BYOC, Inc.



**BRITISH BLUES OVERDRIVE
KIT
INSTRUCTION INDEX**

Parts Checklist.....page 4 - 5

Populating the Circuit Board.....page 6 - 12

Assembly.....page 13 - 15

Wiring.....page 16 - 19

Installing the ICs.....page 20

Modifications.....page 21 - 22

Operation Overview.....page 23

Schematic.....page 24 - 25

Parts Checklist for BYOC British Blues Overdrive

Resistors:

- 1 - 1k (brown/black/black/brown/brown)
- 1 - 3k3 (orange/orange/black/brown/brown)
- 2 - 4k7 (yellow/purple/black/brown/brown)
- 2 - 6k8 (blue/gray/black/brown/brown)
- 3 - 10k (brown/black/black/red/brown)
- 1 - 100k (brown/black/black/orange/brown)
- 1 - 220k (red/red/black/orange/brown)
- 2 - 1M (brown/black/black/yellow/brown)

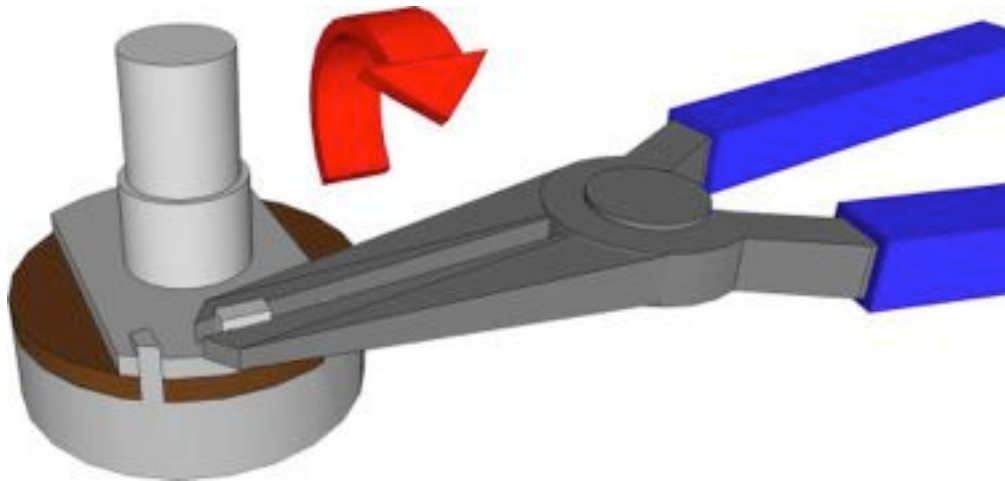
Capacitors:

- 1 - 47pf ceramic disc (small orange cap)
- 5 - .01 μ f/10n film (103)
- 3 - 0.047 μ f/47n film (473) these are extras for modifications
- 1 - 0.1 μ f/100n film (104)
- 1 - 0.22 μ f/220n film (154)
- 2 - 100 μ f aluminum electrolytic

Diodes:

- 4 - 1N914 or 1N4148 (small orange glass with black stripe)
- 1 - 1N4001 (larger black plastic with silver stripe)

Potentiometers: Be sure to snap off the small tab on the side of each panel mounted pot.



- 1 - B25k linear (tone knob)
- 1 - B100k linear (drive knob)
- 1 - A100k audio (level knob)

IC's:

- 1 - TL072
- 1 - DIP 8 socket

Hardware:

1 - drilled enclosure w/ 4 screws

1 - byoc British Blues OD PCB

1 - 3PDT footswitch

3 - knobs

1 - AC adaptor jack

1 - 1/4" mono jack

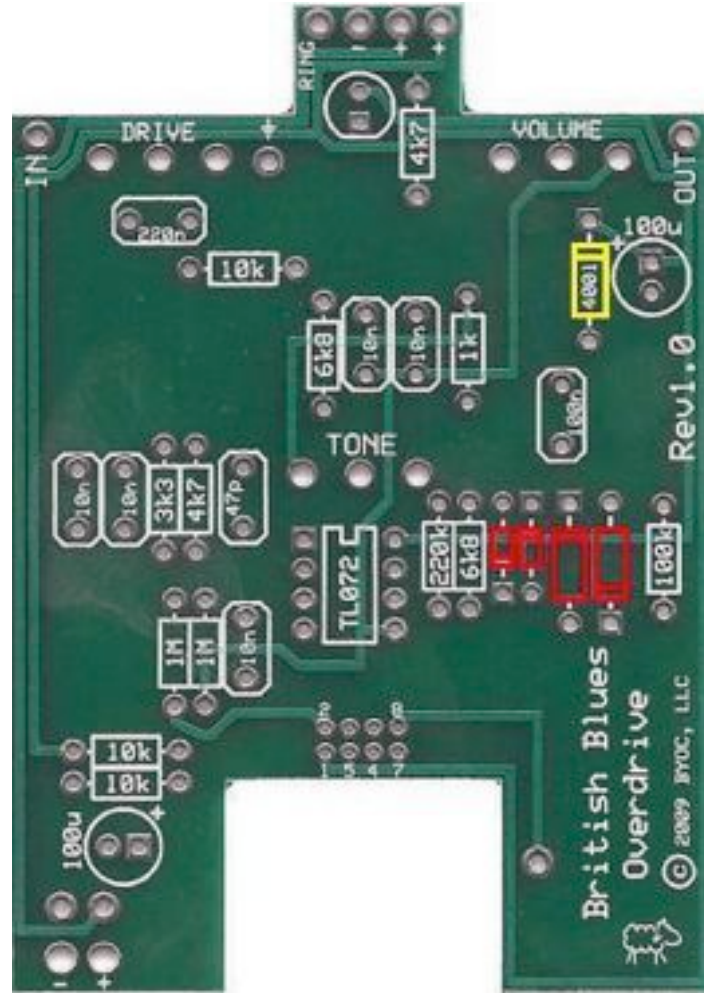
1 - 1/4" stereo jack

1 - red LED

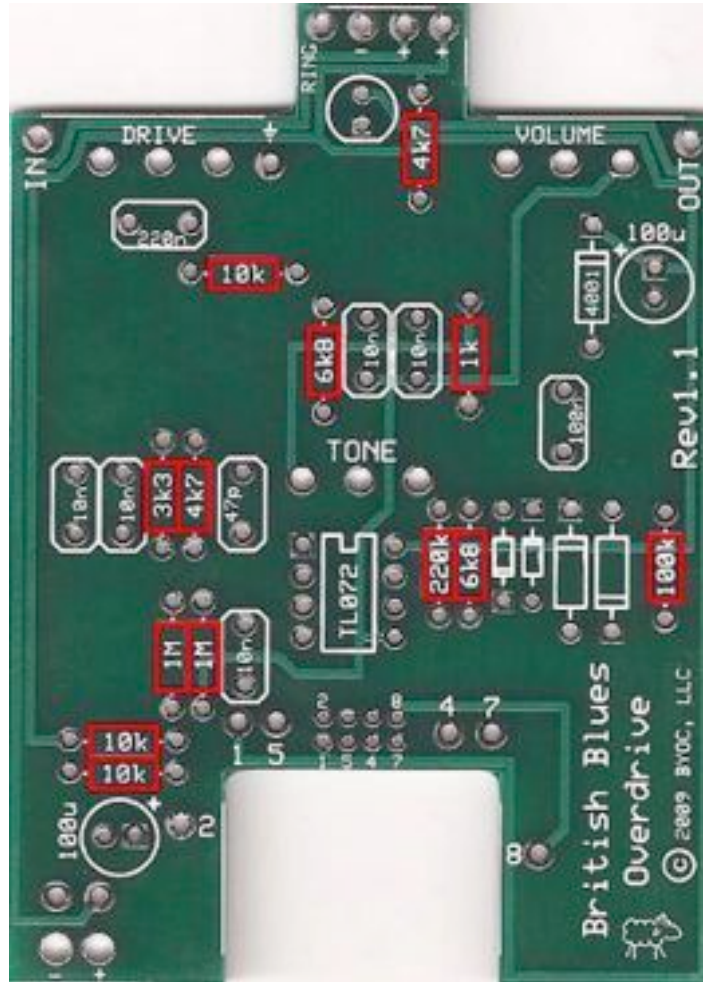
1 - battery snap

hook-up wire

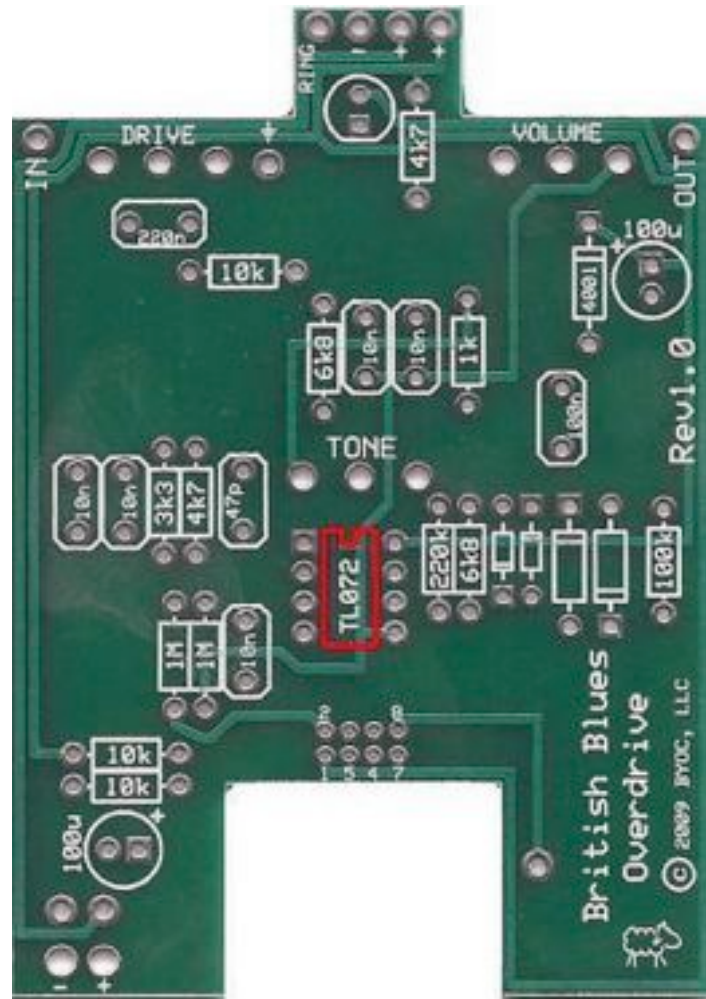
Populating the Circuit Board



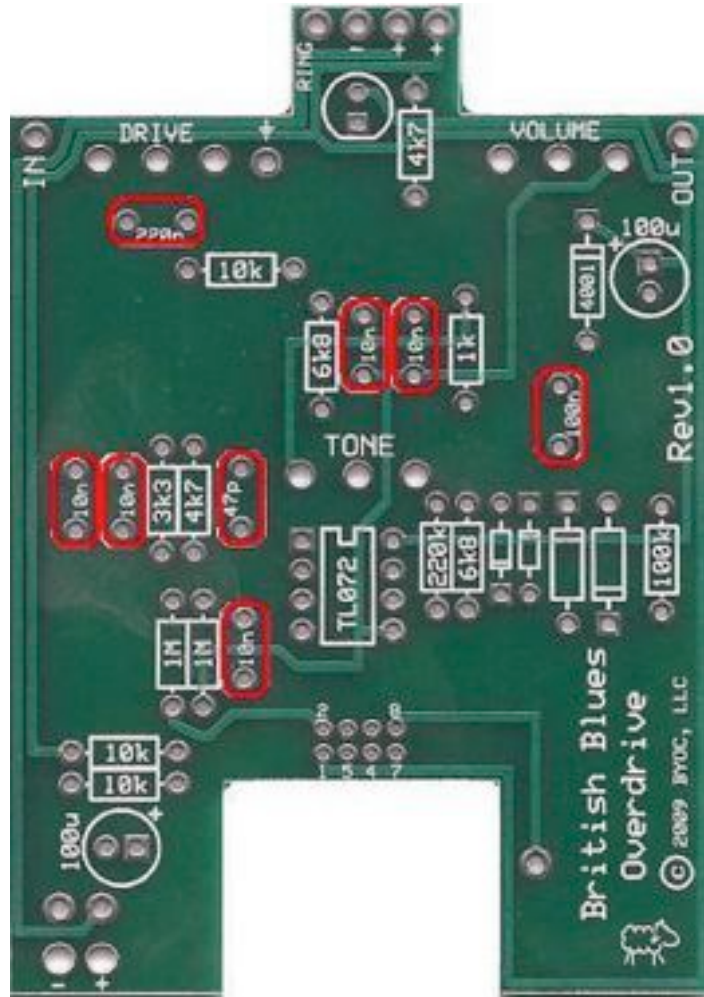
STEP 1: Add the diodes. Be sure to match the end of the diode with the stripe to the layout on the PCB. The stripped end should go in the square solder pad. Note that the 1N4001 diode goes in the space highlighted in yellow. The four small orange 1N4148 diodes go in the space highlighted in red. Two of the spaces will be larger, but are still intended for the 1N4148 diodes. The two larger spaces are there for experimentation with other clipping diodes of various size.



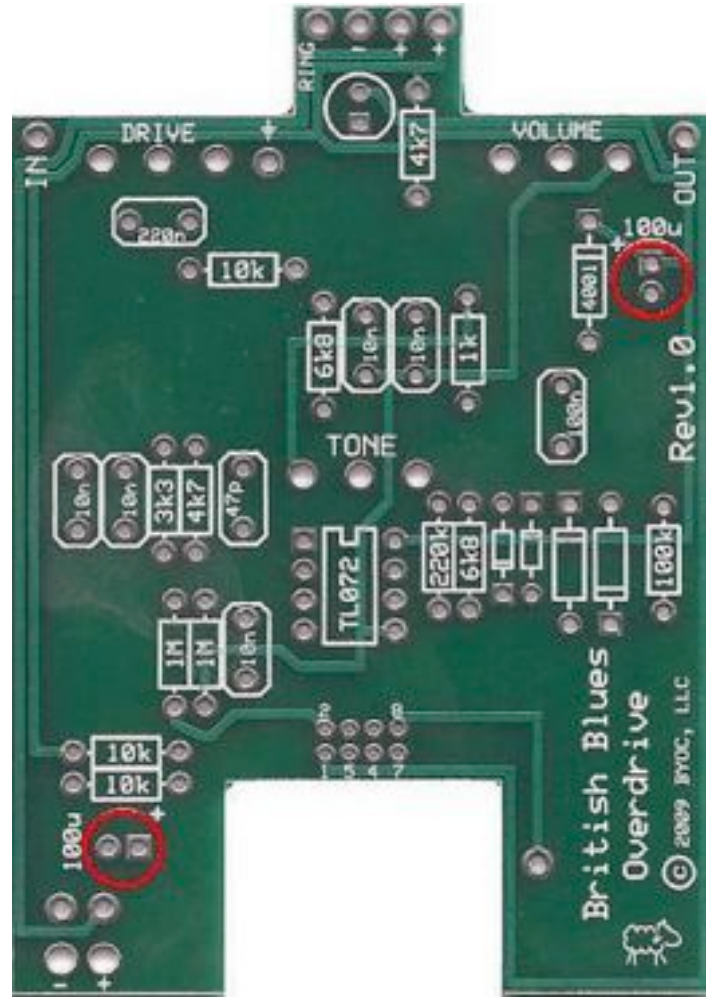
STEP 2: Add the resistors . Resistors are not polarized, so it does not matter which end goes in which solder pad.



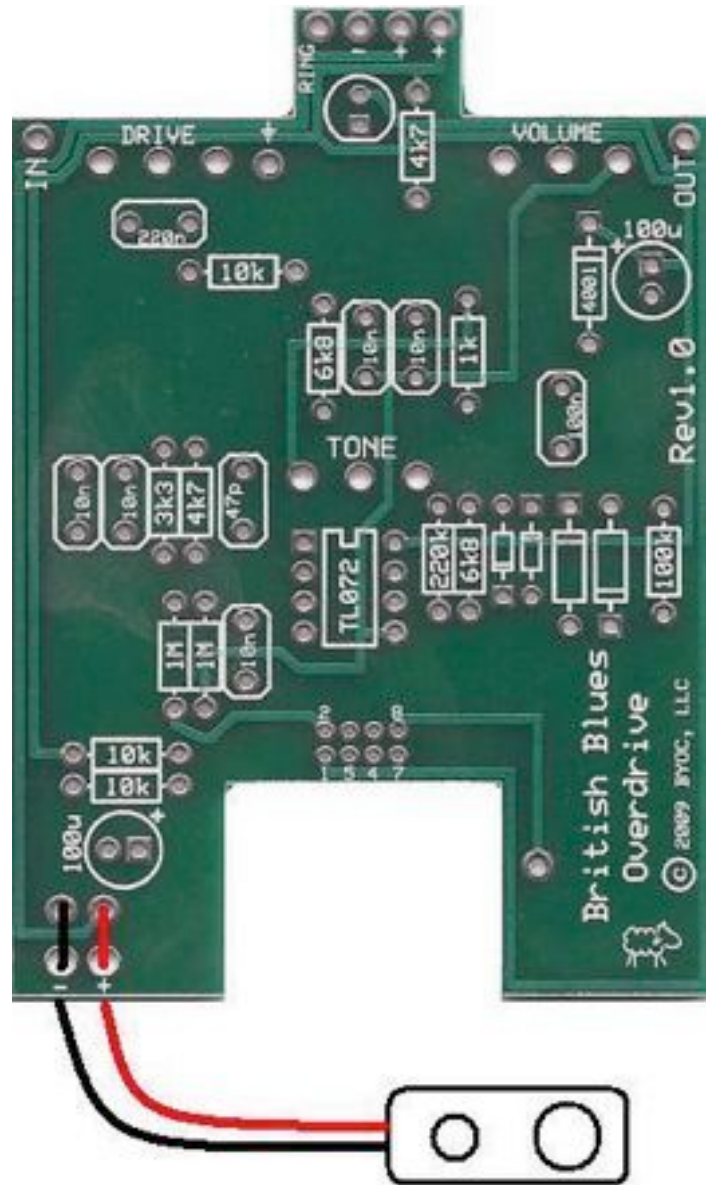
Step3: Add the 8 pin socket. Line up the socket so that the “u-shaped” notch matches up with the end that has a notch on the layout. Only add the socket. DO NOT solder the actual chip to the PCB. DO NOT solder the chip to the socket. Only the socket gets soldered. You will never solder the chip(s). Installing the chips or IC's into the socket will be explained on page 26.



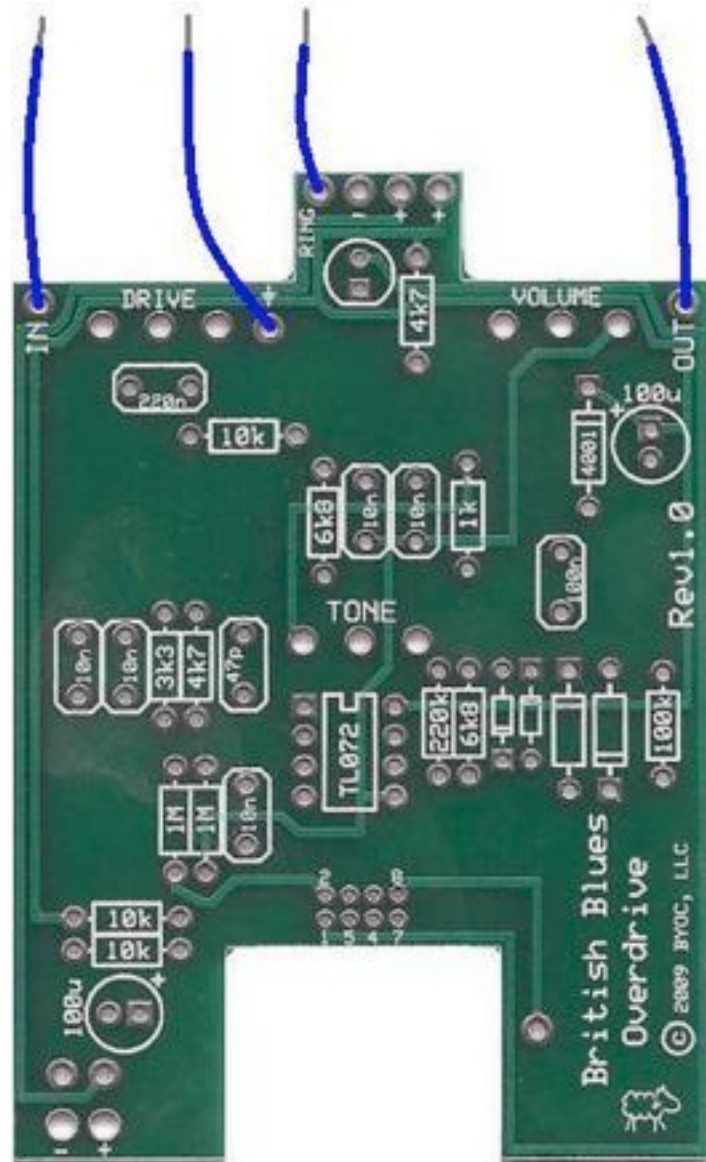
STEP 4: Add the film capacitors. These are not polarized so they can be inserted into the PCB in either direction.



STEP 5: Add the aluminum electrolytic capacitors. These are polarized. The positive end will have a longer lead and should go in the square solder pad. The negative end will have a shorter lead with a black strip running down the body of the capacitor.



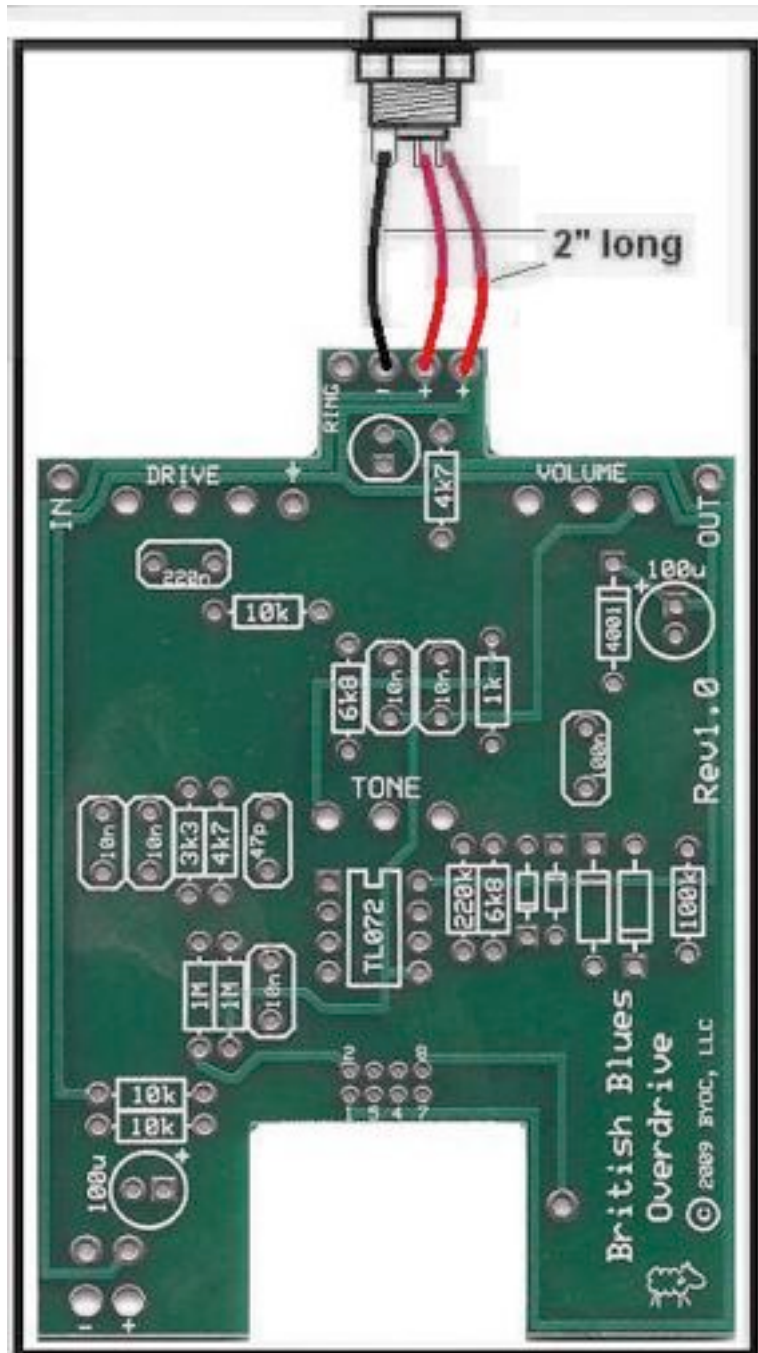
Step 6: Add the battery snap. Thread the solder ends of the battery snap into the strain relief holes from the bottom solderside of the PCB and out through the top. Insert the solder ends of the battery snap wires into the topside of their respective solder pads. Solder on the bottom side of the PCB. Remember the red wire goes in the “+” hole and the black wire goes in the “-” hole.



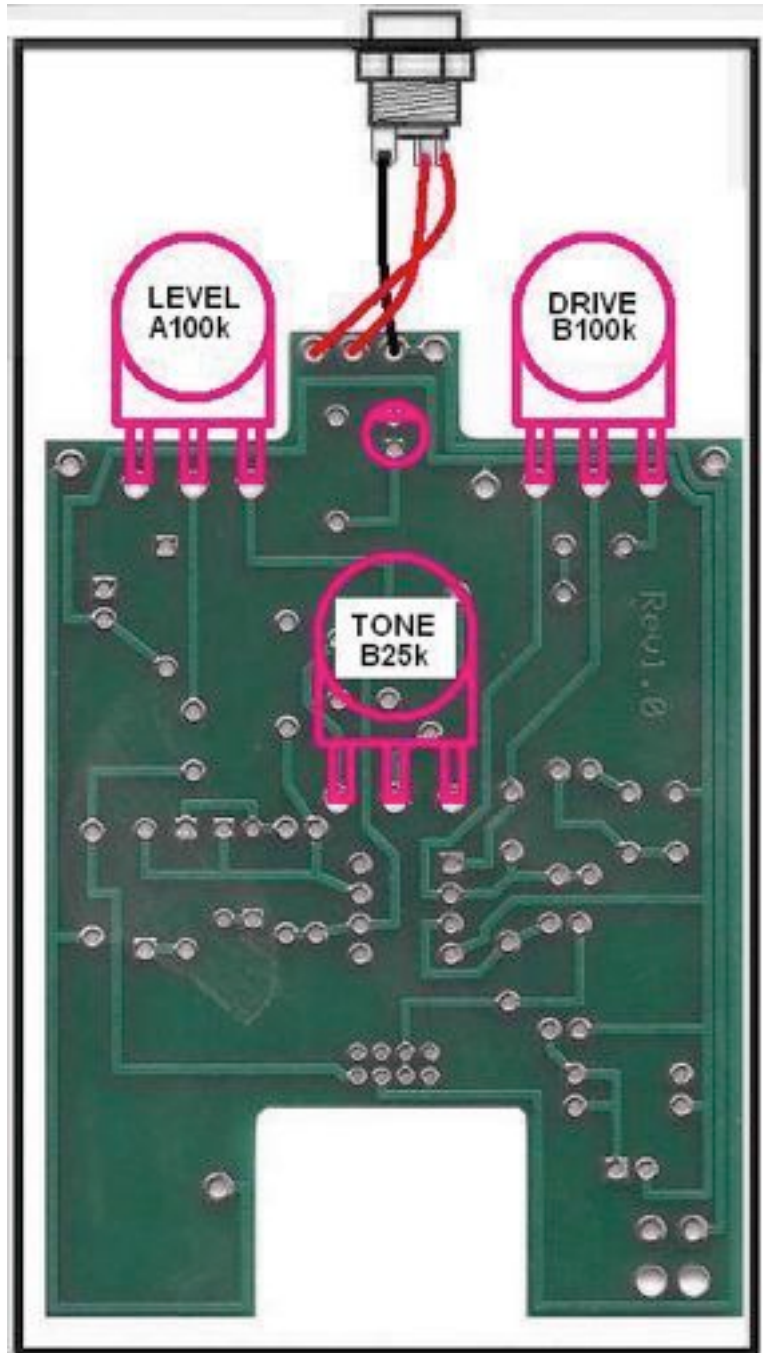
Step 7: Add wires to the IN, OUT, Ground, and RING eyelets. Start by cutting three 2.5" pieces of wire and one 1.5" piece of wire. Strip 1/4" off each end and tin the ends. Tinning means to apply some solder to the stripped ends of the wires. This keeps the strands from fraying and primes the wire for soldering. Solder a 2.5" piece of wire to each of the IN, OUT, and Ground eyelets on the PCB. Solder the 1.5" piece of wire to the RING eyelet on the PCB. Load the wires in from the top and solder on the bottom of the PCB.

Assembly

Step 1: Mount the DC adaptor jack to the enclosure.

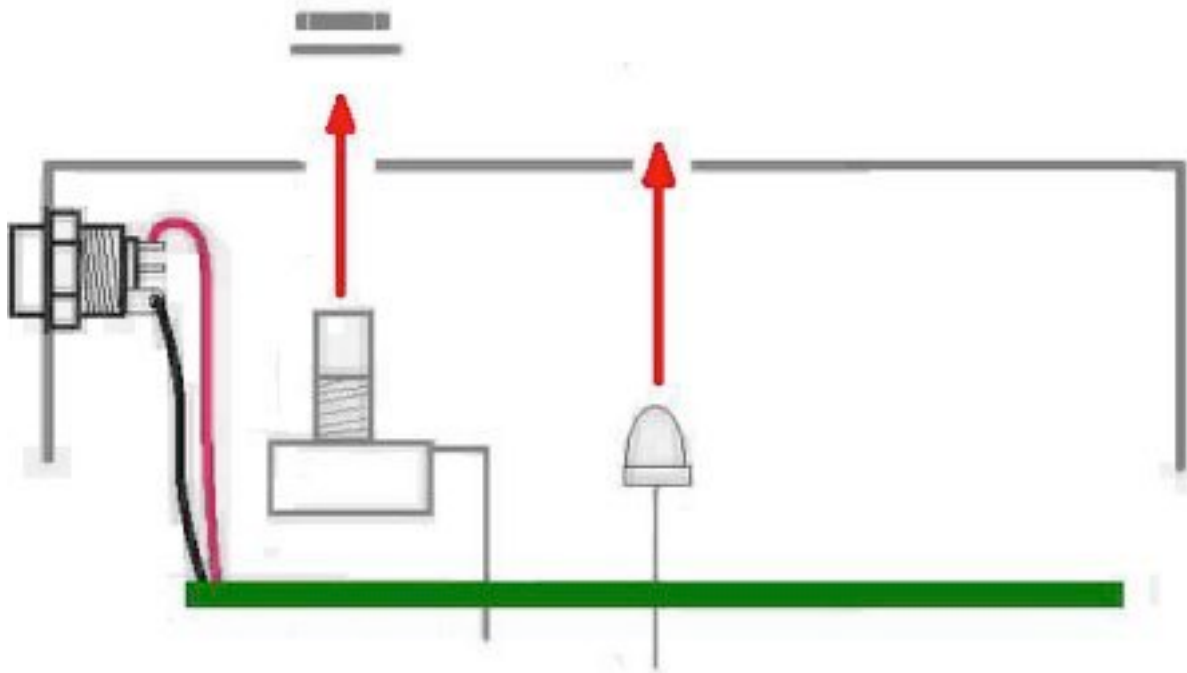


Step 2: Connect the TIP (negative) terminal of the DC adaptor jack to the “-” eyelet on the PCB with 2 inches of hook up wire. Connect the SLEEVE of the DC adaptor jack to the “+” eyelet on the far right side of the PCB with 2 inches of hook up wire. Connect the battery disconnect terminal of the DC adaptor jack to the “+” eyelet more towards the center of the PCB with 2” of hookup wire.



Step 3: Flip the PCB over so that the bottom or solder side is up. Insert the B25k(tone), A100k(volume), B100K(drive) potentiometers, and the LED into the bottom side of the PCB. DO NOT SOLDER YET!!! The LED will have one lead that is longer than the

other. The longer lead goes in the hole of the square solder pad.

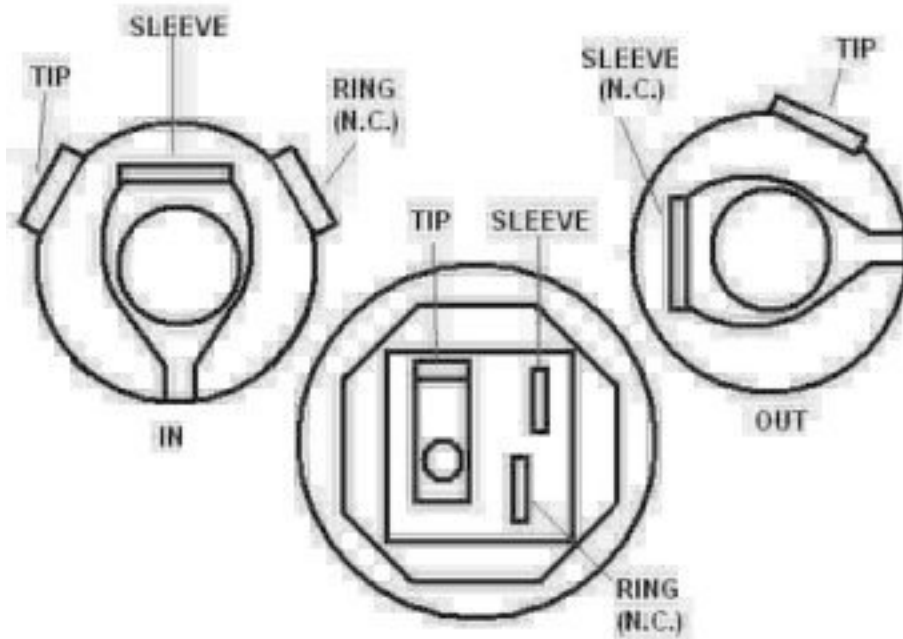


Step 4: Hold the PCB in one hand so that the component side of the PCB is in the palm of your hand and the bottom side with the pots, toggle switch and LED is facing up. Now use your other hand to guide the predrilled enclosure onto the PCB assembly so that the pots and LED all go into their respective holes. Once the PCB assembly is in place, secure it by screwing on the washers and nuts for the pots and toggle switch. Only tighten them with your fingers. You do not want them very tight yet. Be sure to keep your hand on the PCB so that it does not fall off the PC mounting posts of the pots and toggle switch.

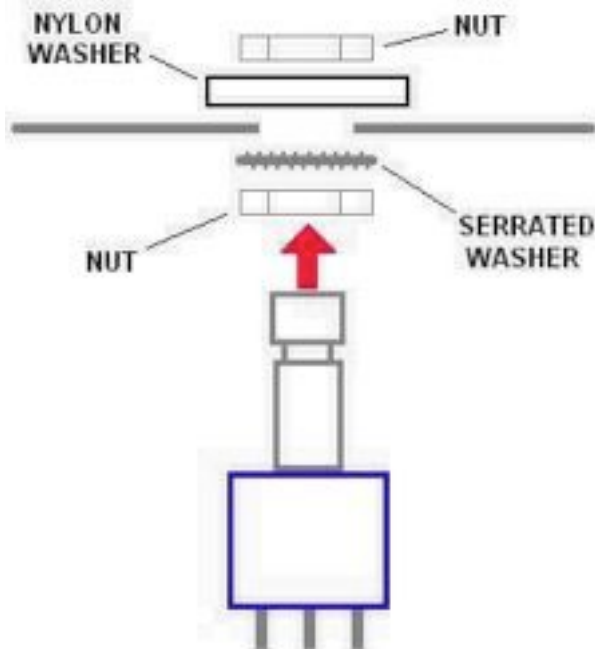
Step 5: Turn the entire pedal over so that the component side of the PCB is facing up. Lift the PCB up off the pots and toggle switch about 2mm just to make sure that the back of the PCB does not short out against that pots. Make sure the PCB is level and symmetrically seated inside the enclosure.

Step 6: Solder the pots and LEDs. You will solder these parts on the component side of the PCB. After you have soldered them in place, be sure to tighten up their nuts.

Wiring



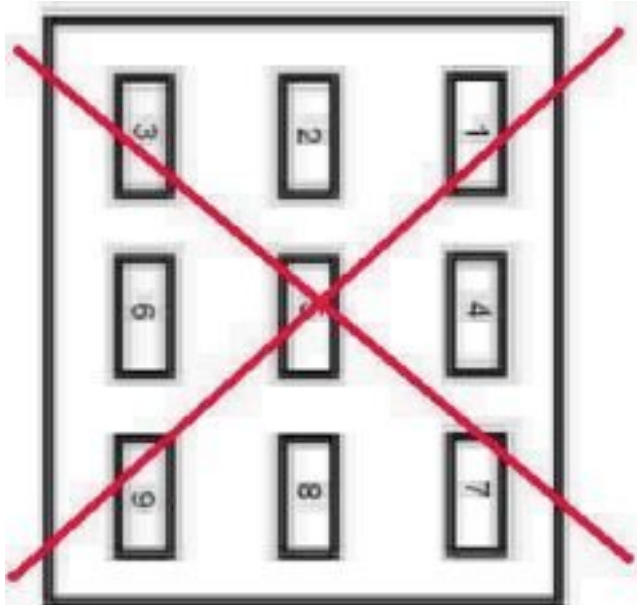
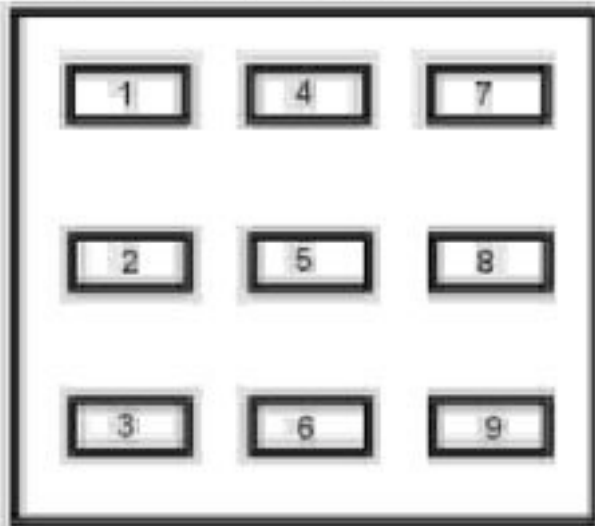
Step 1: Install the 1/4" jacks to the enclosure. Be sure to turn the OUT jack a 1/4 turn counter clockwise so that solder terminal for the tip does not short out against the enclosure.



Step 2: Install the footswitch. Orient the footswitch so that the flat sides of the solder lugs are like the diagram below. NOTE: There are no actual number markings on the footswitch. There are two correct ways you can orient the footswitch. They are both 180 degrees of each other. Either way is fine. It does not matter as long as the flat sides of

the solder lugs are running horizontal, not vertical.

Footswitch Solder Lug Designations

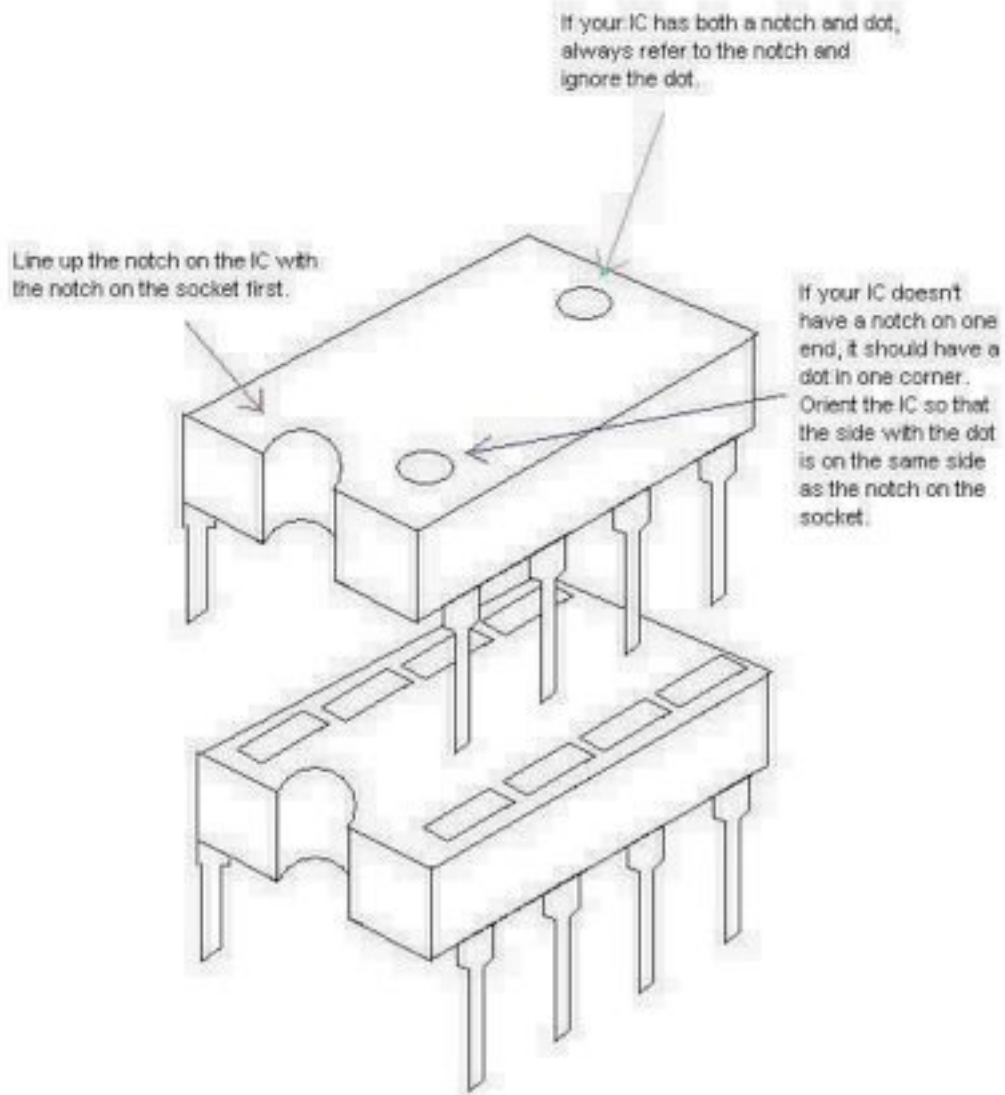


- Cut 3 x 2" pieces of wire. Strip 1/4" off each end. These will be used to connect the tip and sleeve of the IN jack and the tip of the OUT jack to the PCB.
- Cut 1 x 1.5" piece of wire. Strip 1/4" off each end. This will be used to connect the ring of the IN jack to the ring eyelet on the PCB.

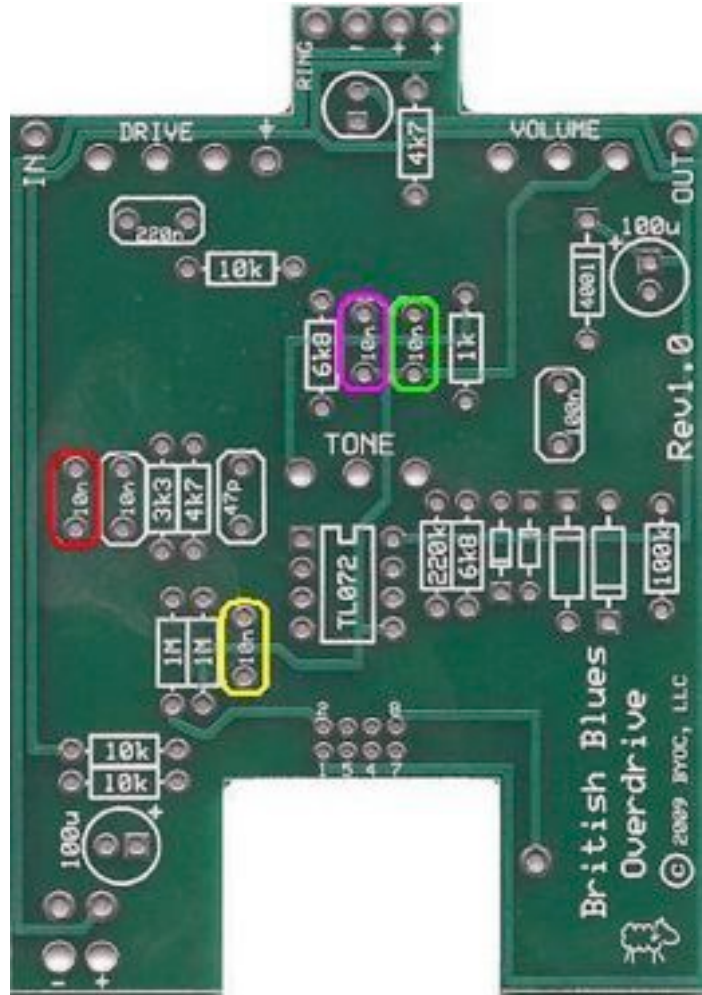
Step 5: Solder one end of the pre-cut and pre-stripped wires to the footswitch.

Step 6: Insert the other remaining ends of the pre-cut and pre-stripped wires into the topside of the PCB and solder. You can solder these on the topside as well. It is easier this way, but you may burn a small amount of the PVC coating on the wires. This is purely aesthetic and won't damage the wires in anyway. But you can avoid this by removing the PCB assembly and footswitch from the enclosure entirely (the PCB assembly will still be attached to the enclosure via the DC jack wiring) so that you have access to solder the underside of the PCB.

Installing the IC



Modifications



The original Marshall Blues Breaker circuit is a great overdrive for guitars with humbuckers or rigs that have a lot of low end. But may sound too thin for single coils or bright rigs. The kit comes with a few extra caps to open up the lows and and make the pedal less bright.

1. Change C1 to a larger value. The 10n input cap highlighted in yellow will let more bass in if you increase the value. The kit comes with an extra 47n cap for this but you can go even bigger if you want.
2. Change C4 to a larger value. The 10n cap highlighted in red behaves very similar to the 47n(.047uf) cap in a tubescreamer that you change to get rid of the midrange hump. Increasing this cap will darken the pedal up a little. The kit comes with a 47n cap but you

can go bigger if you want.

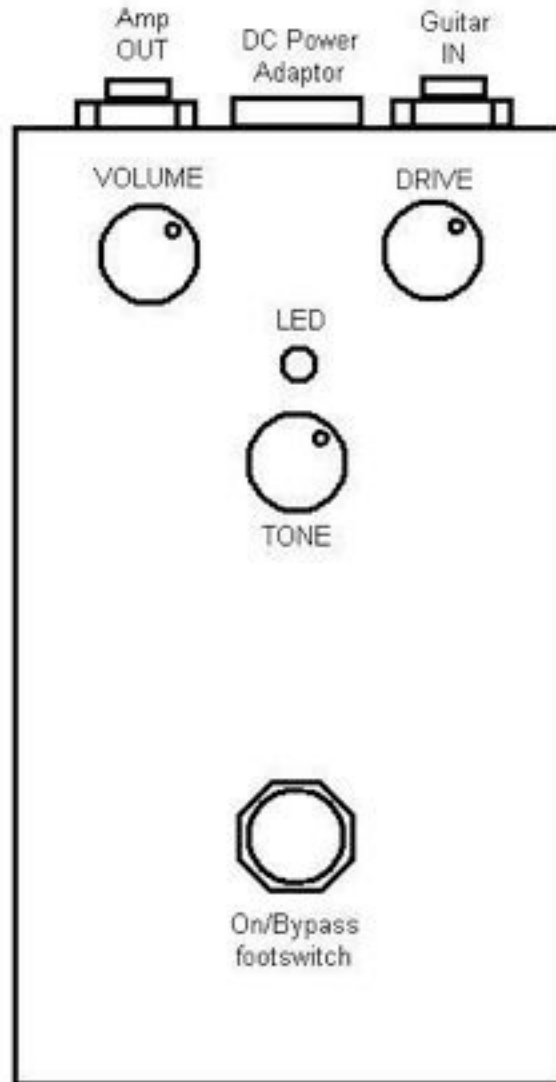
3. Change C6 to a larger value. The 10n cap highlighted in green is part of the tone knob. Changing it to a larger value will make the tone knob more useful in the lower frequencies. The kit comes with a 47n cap, but you can try a larger value if you want more.

4. Change C7 to a larger value. The 10n cap highlighted in pink is part of a low pass filter (or high cut). Increasing the value of this cap won't give you more bottom end, but it will cut out some of the highs. So if after the first three mods, you still find that the pedal is too bright, you might want to play around with this cap value.

5. More gain - You can increase the value of the R7 220k resistor or decrease the value of the R3 4k7 and R4 3k3 resistors. You can also increase the gain by using a larger value potentiometer for the drive knob. If you want less gain, do the opposite to any of these component value suggestions.

6. More low end - In this circuit, increasing the value of any of the caps will give you more low end or reduce highs in some way. We've already discussed some of the more critical capacitors that will give you the most change, but you could even change the 100n and 220n to something like a 1u film cap (5mm spacing). And you could cut some of the highs by using a larger value cap in place of the 47pf.

Operating Overview



Pretty simple. Volume, drive, tone. Guitar plugs in on the left. Amp plugs in on the right.

DC power supply - Use a 2.5mm negative tip adaptor (this is your standard guitar fx style adaptor). This unit can run on 9 - 18VDC. If using battery power, only use a single 9V battery.

Current Draw - 3.5mA

Input Impedance - 1M ohms

Output Impedance - 100k ohms

Please visit
<http://buildyourownclone.com/board>
for any technical support

copyright 2009
B.Y.O.C., LLC