Build Your Own Clone
Classic Overdrive
Kit Instructions

Warranty:

BYOC, Inc. 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, Inc. guarantees that the instructions for your kit will be free of any majors 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, Inc. 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, Inc. 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 and/or PayPal/processing fees. Return shipping is the customer's responsibility. This responsibility not only includes the cost of shipping, but accountability of deliver as well. Please contact sales@buildyourownclone.com to receive a return authorization before mailing.

Tech Support:

BYOC, Inc. makes no promises or guarantees that you will successfully complete your kit in a satisfactory manner. Nor does BYOC, Inc. promise or guarantee that you will receive any technical support. Purchasing a product from BYOC, Inc. does not entitle you to any amount of technical support. BYOC, Inc. 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.byocelectronics.com/board

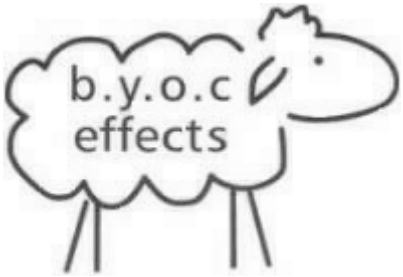
When posting a tech support thread on the BYOC forum, please post it in the correct lounge, and please title your thread appropriately. If everyone titles their threads "HELP!" then it makes it impossible for the people who are helping you to keep track of your progress. A very brief description of your 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 is 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: There are no known errors.
Rev 1.1: Tantalum Caps replaced with 5mm film cap spacing.
Rev 1.2: Added markers for common modifications.



CLASSIC OVERDRIVE KIT
INSTRUCTION INDEX

Parts Checklist for BYOC Classic Overdrive Kit

Resistors:	**Metal Film (5-band)**	/	**Carbon (4-band)**
1 - 100ohm	(brown/black/black/black/brown)		(brown/black/brown/gold)
1 - 220ohm	(red/red/black/black/brown)		(red/red/brown/gold)
4 - 1k	(brown/black/black/brown/brown)		(brown/black/red/gold)
1 – 2k7	(red/purple/black/brown/brown		(red/purple/red/gold)
2 - 4k7	(yellow/purple/black/brown/brown)		(yellow/purple/red/gold)
7 - 10k	(brown/black/black/red/brown)		(brown/black/orange/gold)
1 – 22k	(red/red/black/red/brown)		(red/red/orange/gold)
1 - 51k	(green/brown/black/red/brown)		(green/brown/orange/gold)
3 - 510k	(green/brown/black/orange/brown)		(green/brown/orange/gold)
1 – 1M	(brown/black/black/yellow/brown)		(brown/black/green/gold)

Capacitors:
1 – 47 ~ 51pf ceramic disc (small round orange)
1 - 22n or .022µ film (may say ‘223’ on the body)
1 - 47n or .047µ film (may say ‘473’ on the body)
1 - 100n or 0.1µ film (may say ‘104’ on the body)
2 – 150n or 0.15u film (may say ‘154’ on the body)
3 - 220n or 0.22u film (may say ‘224’ on the body)
2 - 1µ film (may say ‘105’ on the body)
1 - 10µf aluminum electrolytic
1 - 47µf aluminum electrolytic
1 - 100µf aluminum electrolytic

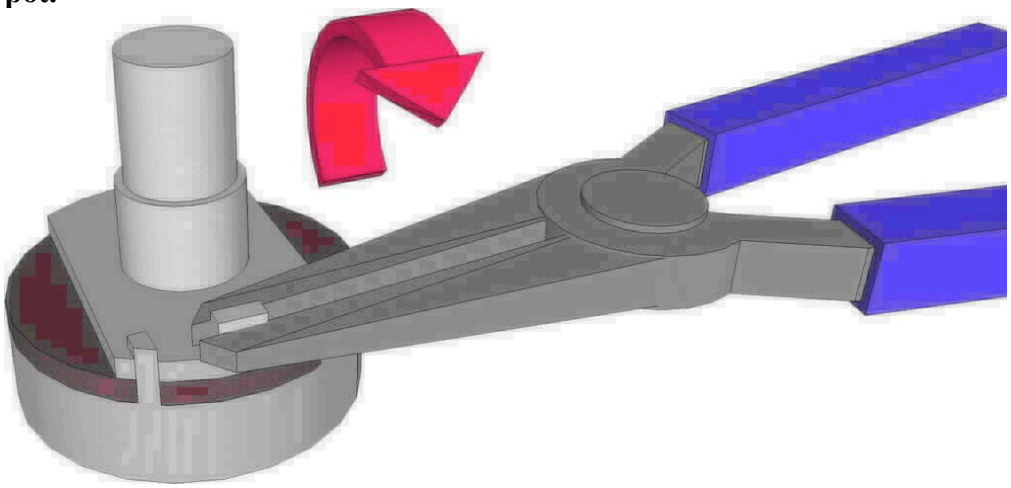
Diodes:

3 - 1N4148 (small orange glass with black stripe)

Transistors:
2 - NPN Transistors (MPSA18, 2N5088, 2N3904, 2N2222 or similar EBC BJT)

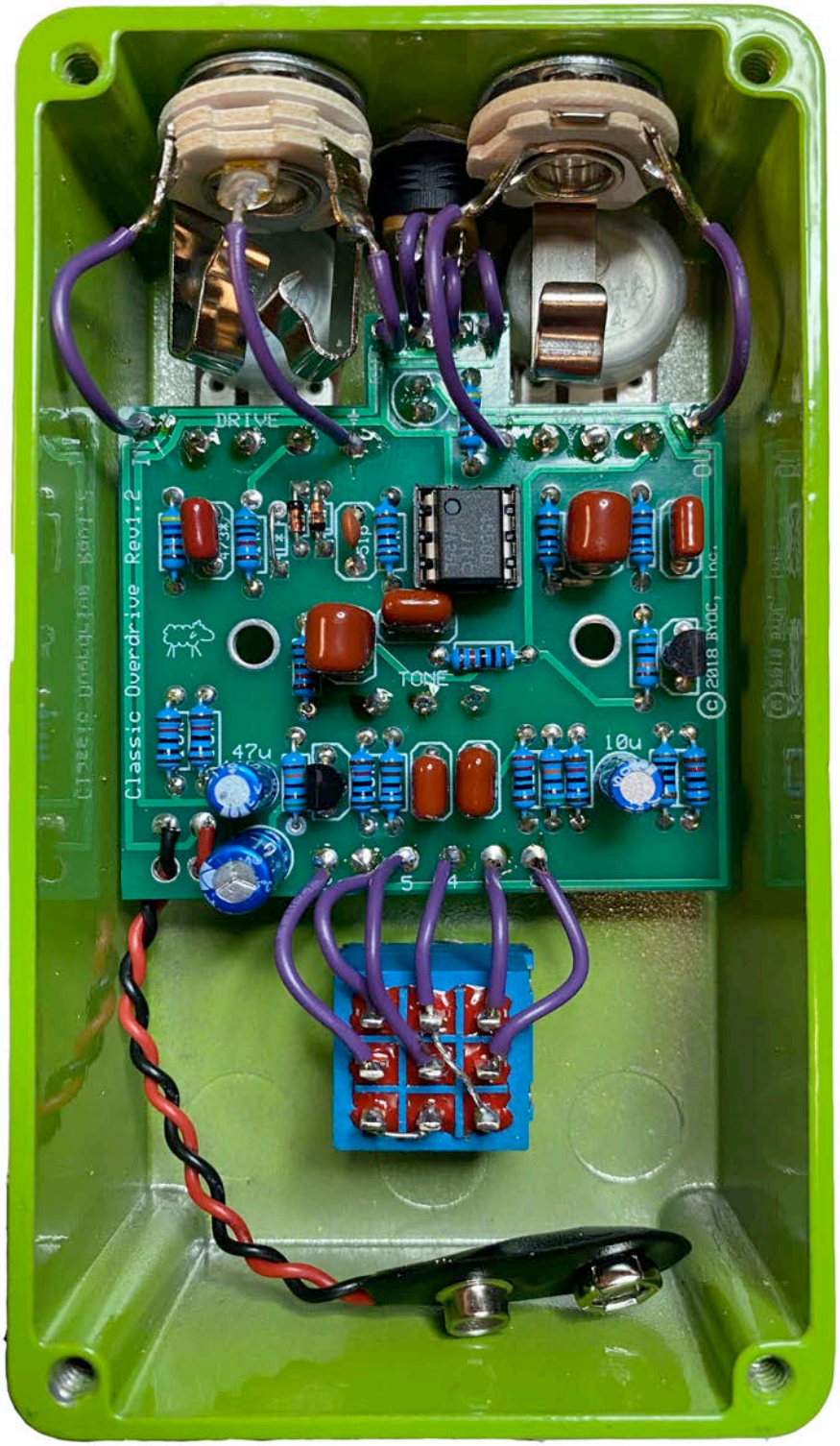
IC's:
1 - 4558 op amp
1 - 8 pin socket

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



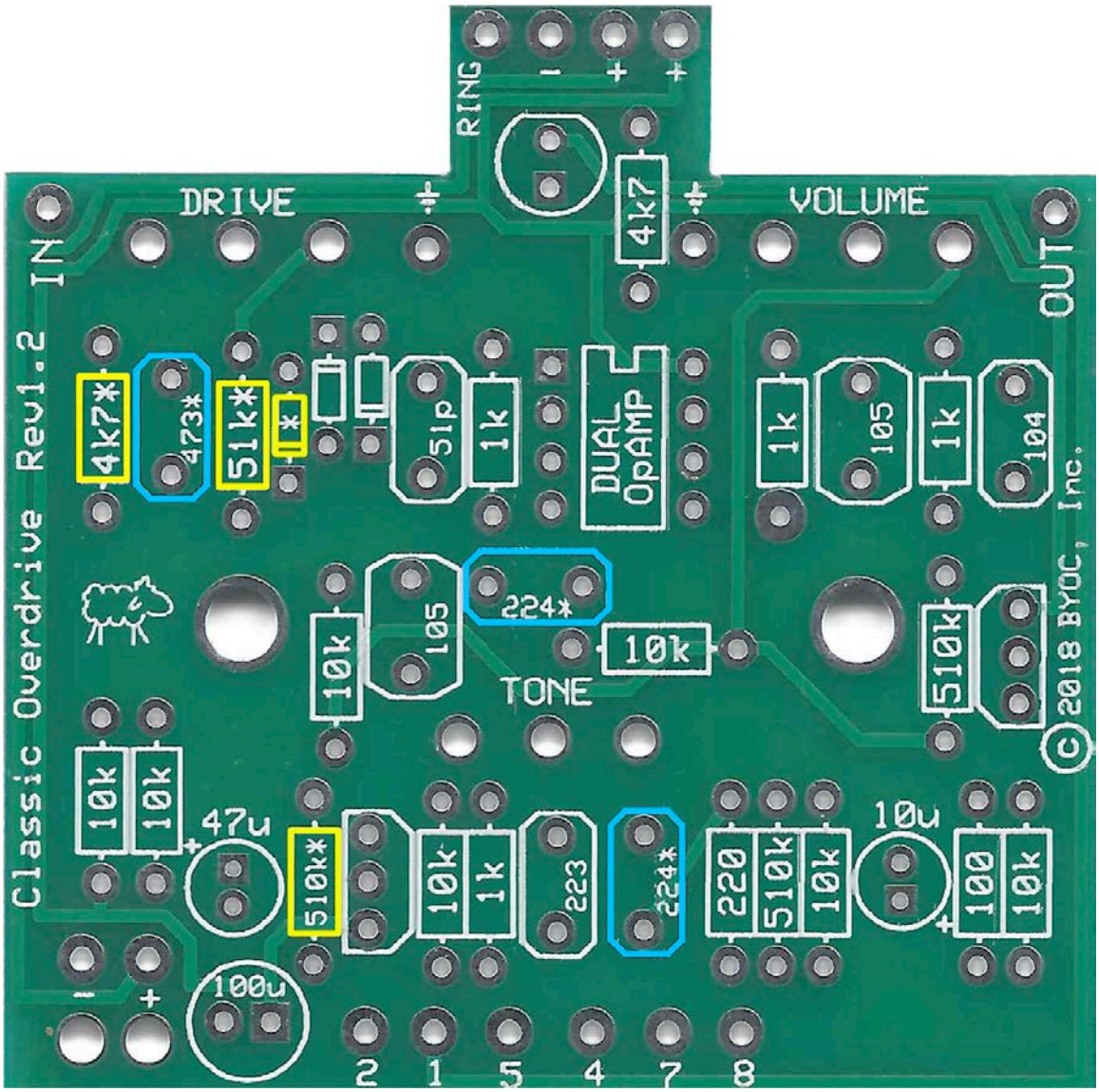
1 - B100k (LEVEL)
1 - W20k or B25k (TONE)
1 - A500k (DRIVE)

Hardware:
1 - drilled enclosure w/ 4 screws
1 - BYOC Classic Overdrive PCB
1 - 3PDT footswitch
4 - knobs (optional)
1 - AC adaptor jack (optional)
1 - ¼"mono jack
1 - ¼"stereo jack
1 - red LED (optional)
1 - battery snap
4 - bumpers
hook-up wire



Example of finished build. Yours may look slightly different depending on the REV number, but they are still the same circuit and will ultimately look like pictured above.

MODIFICATIONS



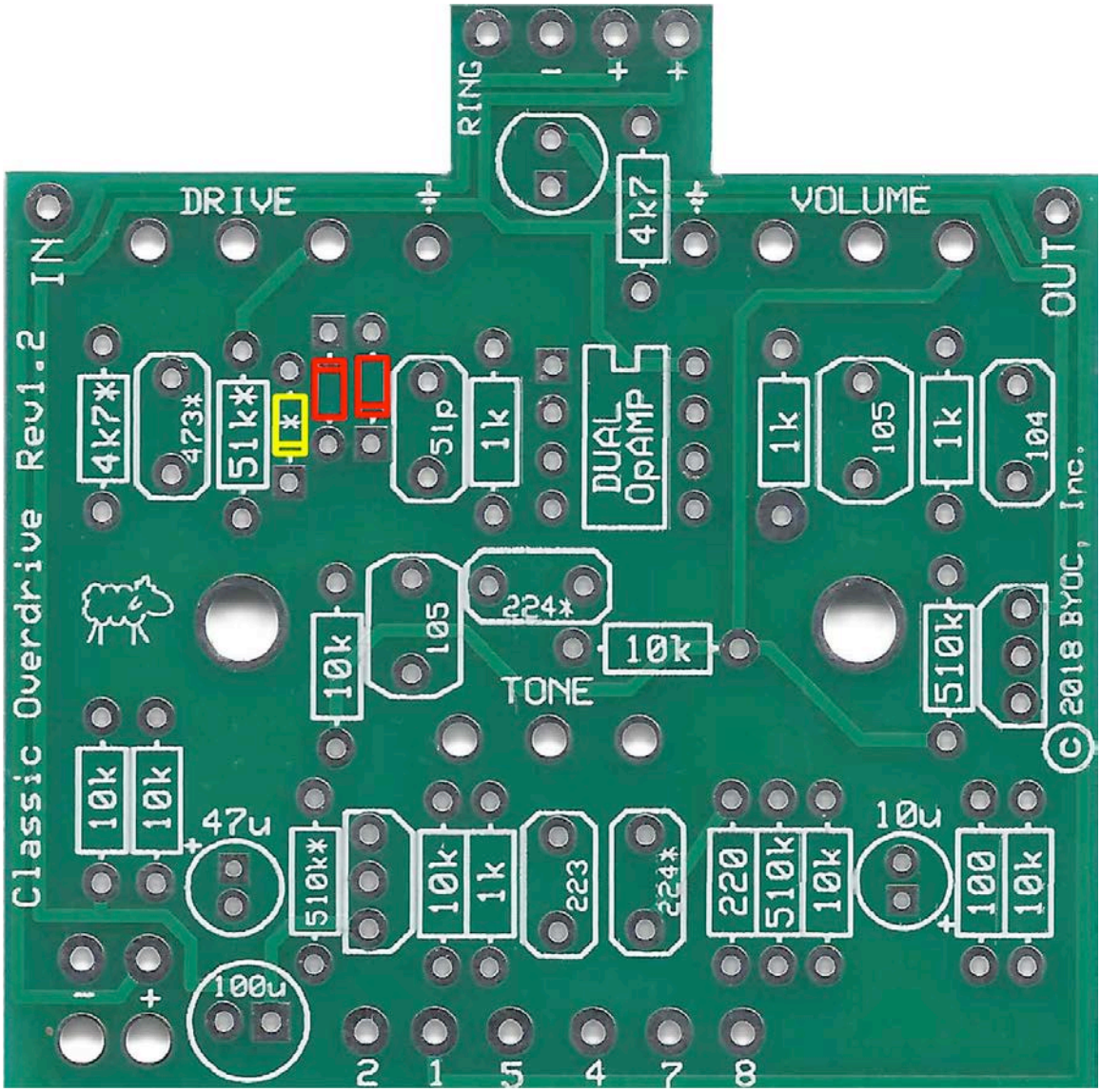
Before you begin populating the PCB, let's take a moment to talk about the various modifications available for this circuit. Your kit comes with parts to perform what are probably the most popular modifications for the Tubescreamer. The spaces on the PCB with an * or asterisk symbol are spaces where you will have the option to use the "stock" component value or a "mod" component value. Be aware that while all of these mods are optional, the spaces on the PCB still need to have a component in them. If you chose not to do the modification, you will still need to install the "stock" component.

1. Asymmetrical Clipping – by adding the extra 3rd 1N4148 diode in the diode space

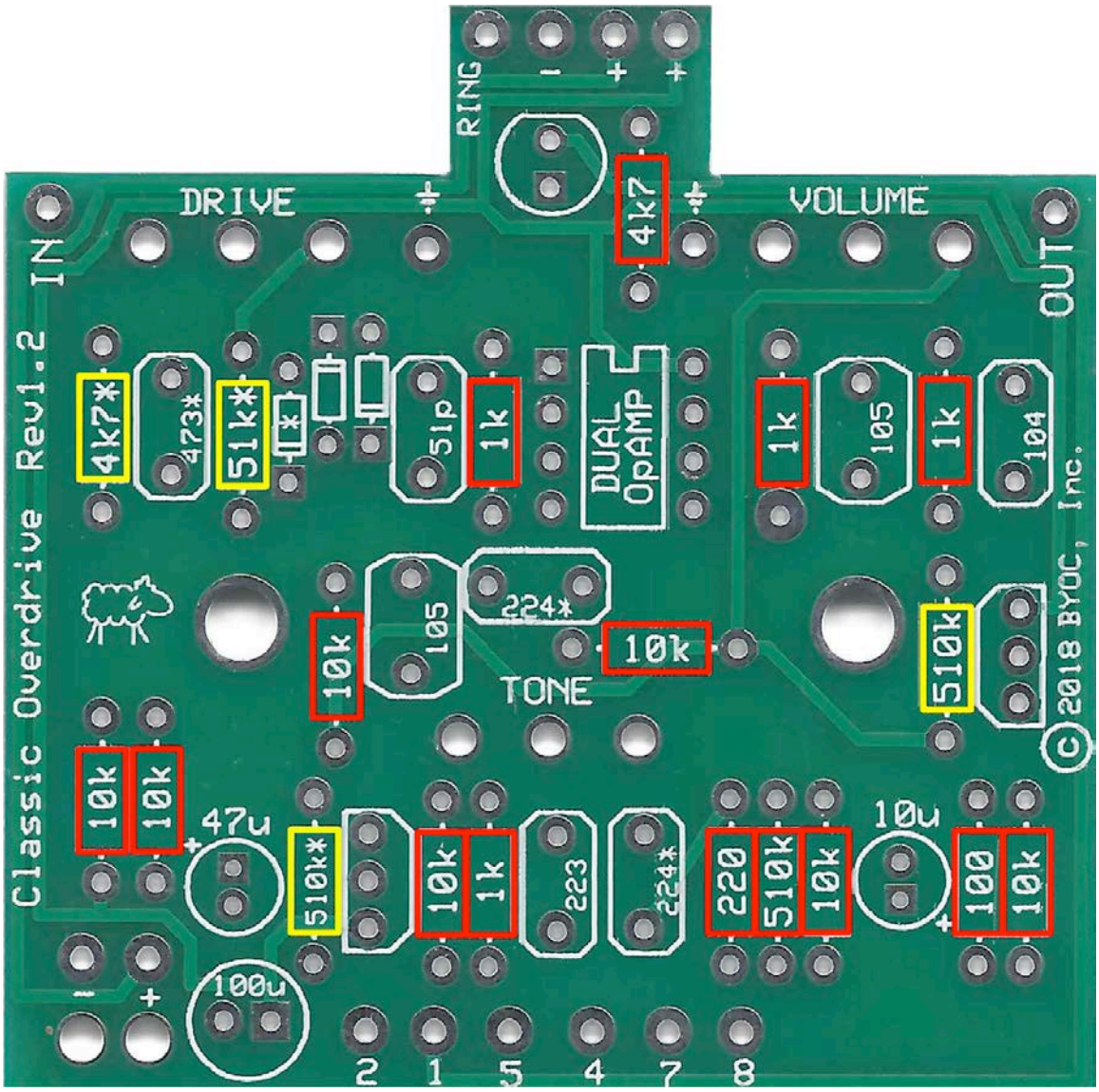
with an * symbol will create asymmetrical clipping. This very subtle change will increase 2rd order harmonics, giving the distortion a richer, more pleasing sound. If you choose not to perform this mod, you will need to add a jumper wire. You CANNOT leave this diode space open.

2. Increased Input Impedance – Changing the R3/510k resistor to a 1M resistor will increase the input impedance. This is another very subtle modification. A higher input impedance increases the performance of your pickups and adds a very slight sparkle to your tone.
3. More Clean – The R6/51k resistor sets the minimum gain limit. By replacing this resistor with a 22k, you reduce the minimum gain limit allowing for less overdrive. This is a useful mod for people who like to use their overdrive as more of a clean boost, than an actual overdrive.
4. More Gain – The R7/4k7 resistor sets the maximum gain limit. By replacing this resistor with a 2k7, you can increase the maximum amount of gain. Who doesn't want more gain? However, it is HIGHLY RECOMMENDED that you also perform the C4 "more bass" mod as well if you perform this mod.
5. More Bass – The C4/473 capacitor sets the high pass roll off frequency of the IC1a gain stage. By replacing this capacitor with a .22uF (224) film capacitor, you reduce the amount of low frequencies that are being cut. The roll off frequency is determined by both the R7 and C4 capacitor. The stock roll off frequency with 4k7 and 473 is 720.8Hz. With a value of 2k7 for R7 and 473 for C4, that's a roll off frequency of 1.25KHz….that's a lot of low end loss. This is why we recommend doing both the "more gain" and "more bass" mods together. With the 224 C4 capacitor and 2k7 R7 resistor, the roll off frequency is now 268Hz.
6. More Highs - The C5/224 capacitor sets the roll off frequency for a low pass filter (or high cut filter). It's set at 723Hz. You can begin to understand why the stock Tubescreamer is so midrange heavy…it's got one filter cutting low frequencies below 720Hz and another filter cutting highs above 720Hz. By changing the C5 capacitor to a .15uF(154), this will change the roll off frequency to 1.06KHz. C5 is the 224 film capacitor space located closest to the Dual OpAmp space.
7. Treble Control Frequency Shift – The "tone" control on the Tubescreamer is actually a 1-band active EQ. "Treble" would be a more accurate name for this knob. It can boost or cut frequencies above 3.2KHz. The frequency is determined by the R10/220ohm resistor and the C6/224 capacitor. By changing C6 to .15uF(154) this will shift the treble controls roll off frequency to 4.8KHz. By shifting the tone control's operating range to a higher frequency, this helps preserve more of your upper mids when using the tone control to cut highs. C6 is the 224 capacitor located next to the 220 resistor.

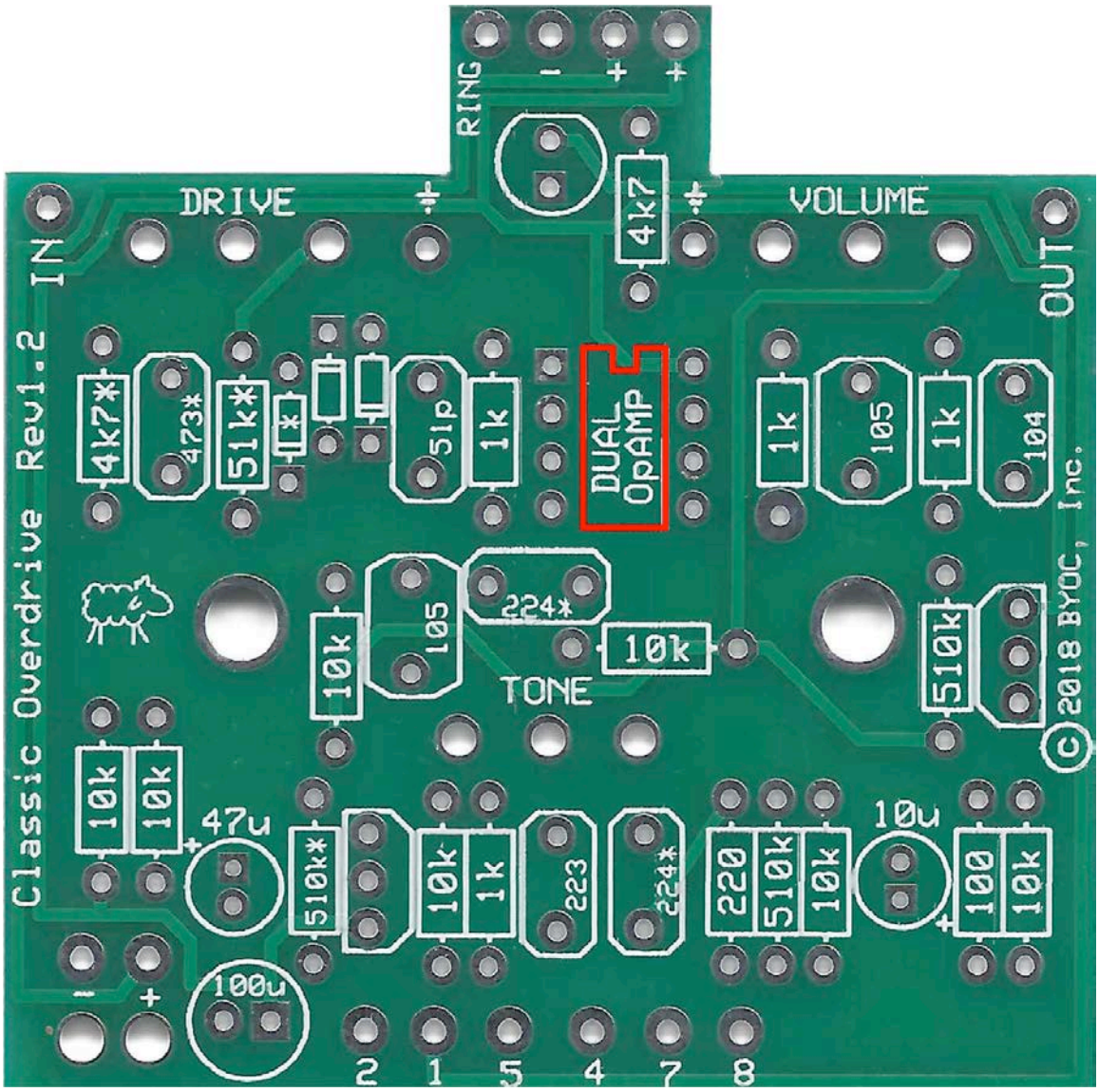
Populating the Circuit Board



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. **The diode highlighted in yellow is for the asymmetric clipping modification. Make a jumper with left over clipping lead if you're building to stock. Refer to page 7 for more info.**

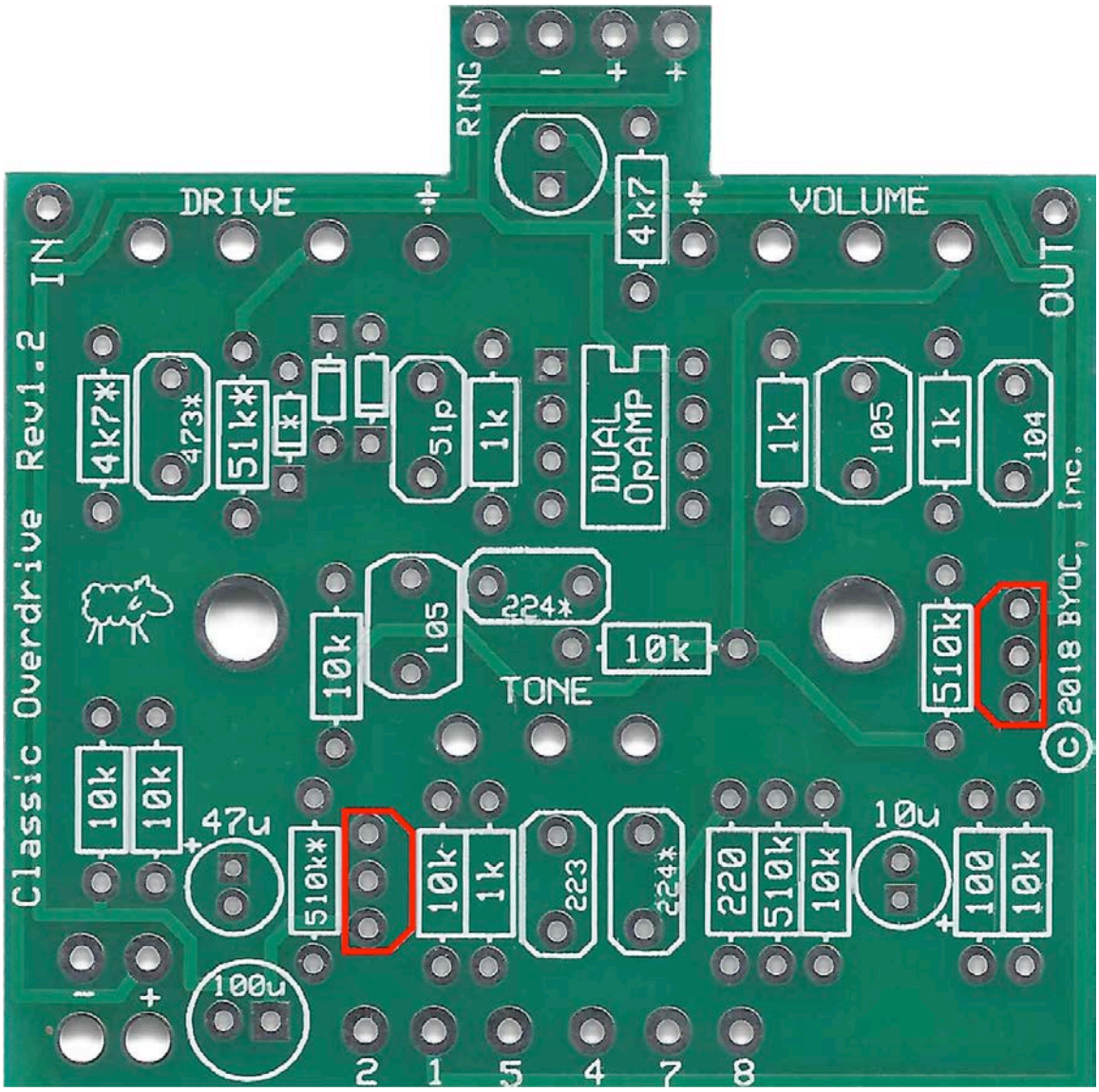


STEP 2: Add the resistors. Resistors are not polarized, so it does not matter which end goes in which solder pad. Take your time and be sure not to confuse similarly banded resistors such as the 510k with the 51k or the 100ohm with the 1k. **The resistor spots highlighted in yellow are for modifications. Refer to page 7 for more info.**

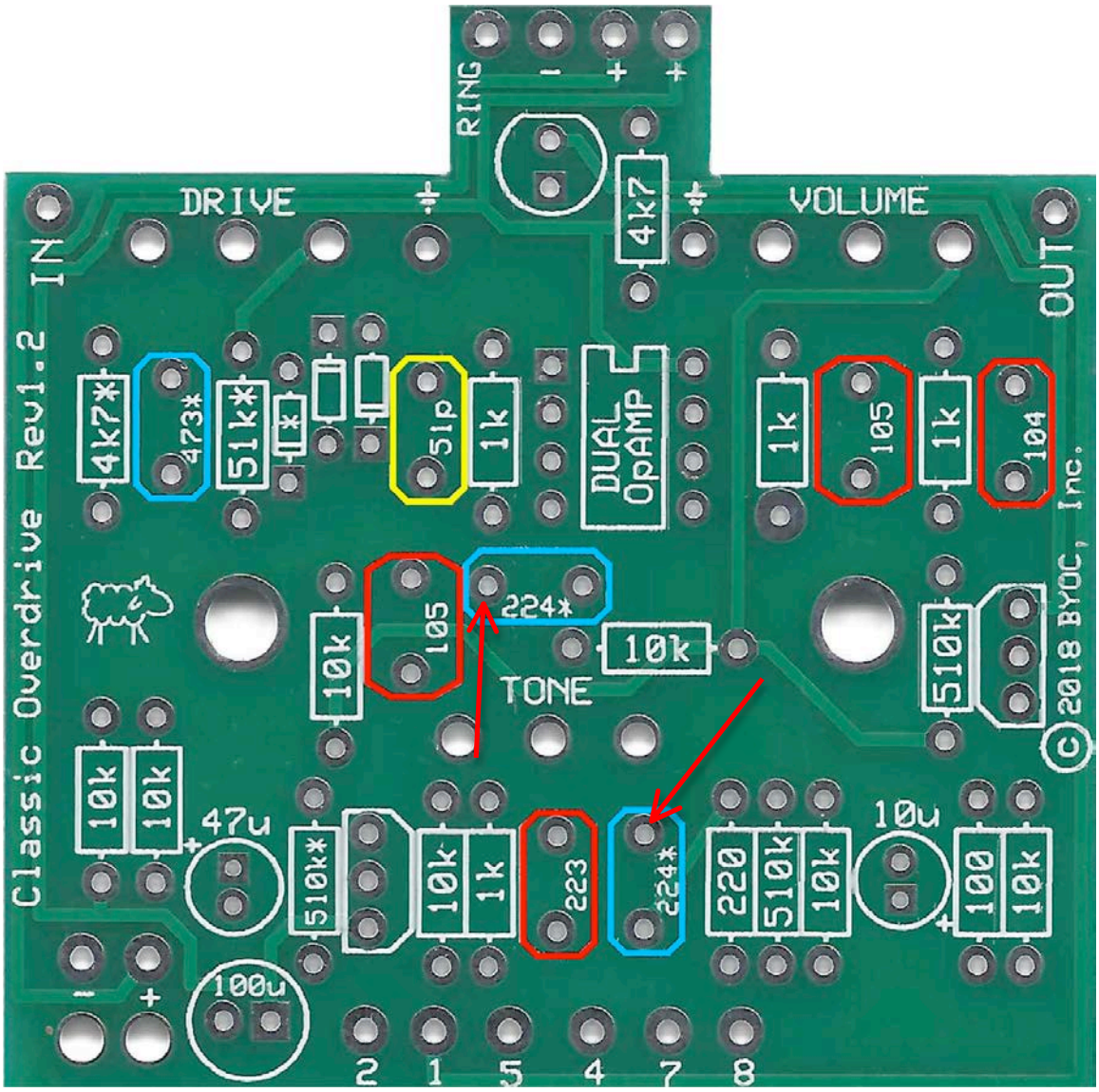


STEP 3: Add the 8 pin IC socket. **ONLY SOLDER THE SOCKET! NOT THE ACTUAL IC!** This is a socket. The sockets get soldered to the PCB. The ICs get inserted into the sockets. The actual IC chip itself, never gets soldered. You will insert the IC into the socket after the entire pedal has been built.

See page 25 for instructions on how to install the IC.



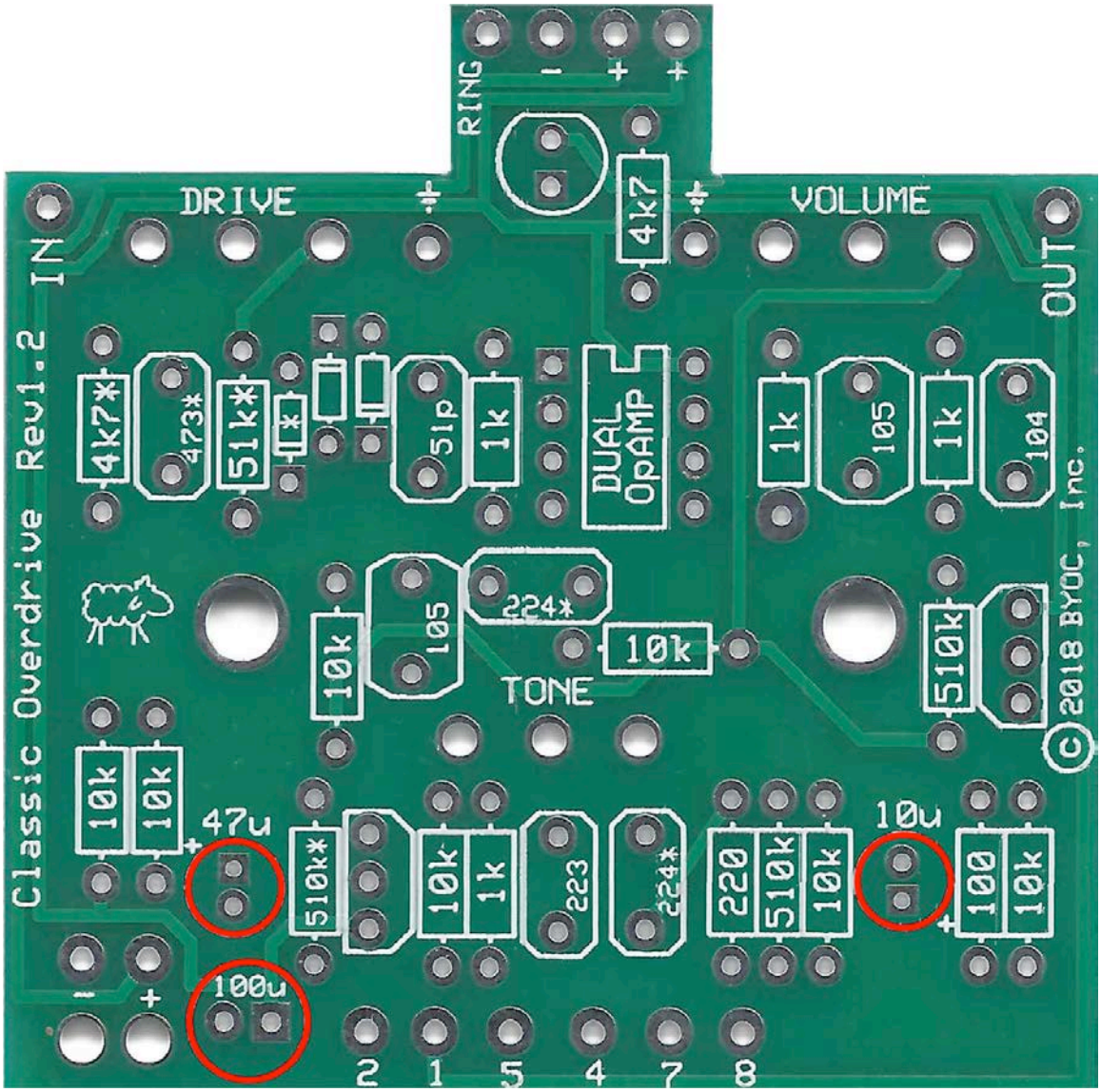
Step 4: Add the Transistors. Be sure to orient the transistors so that the flat side of the transistor body matches up with the flat side on the PCB layout.



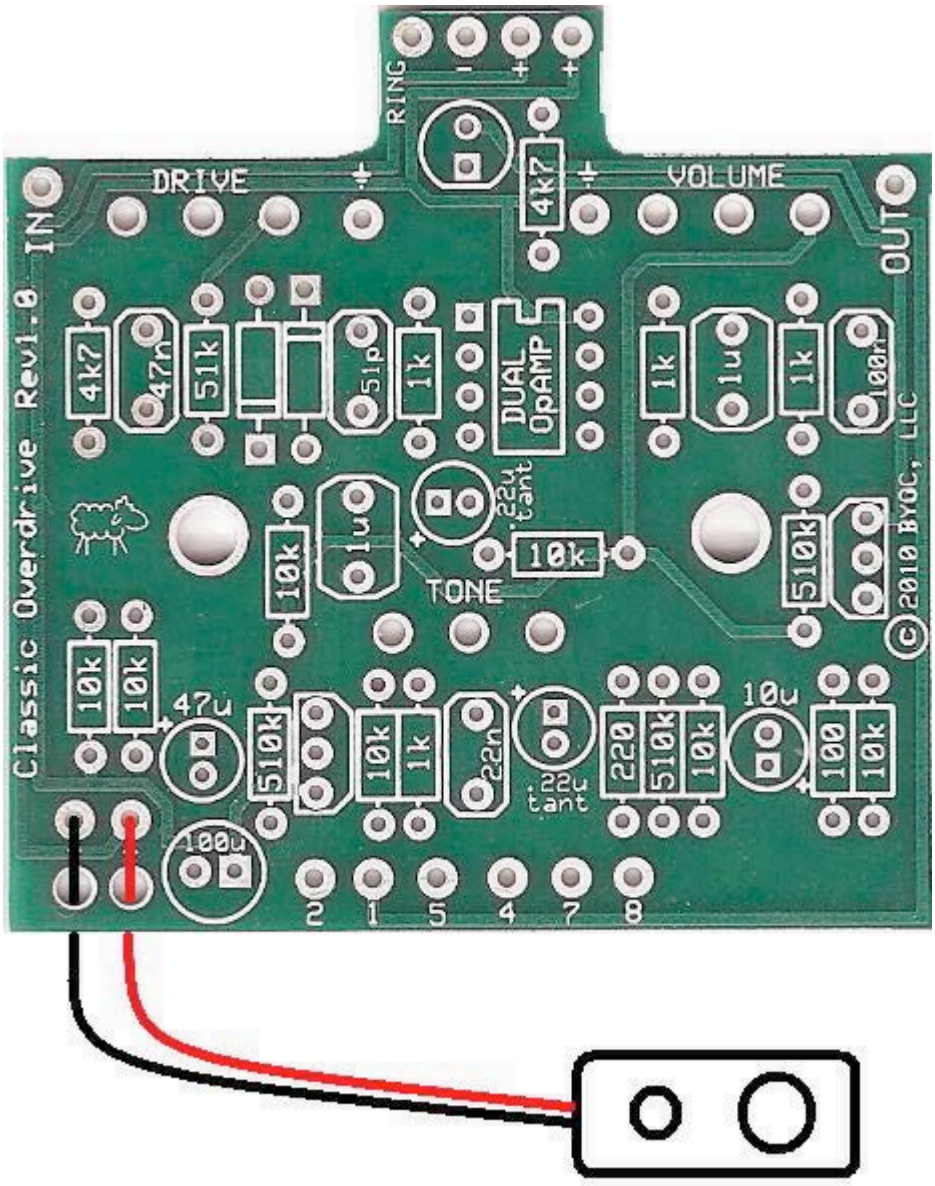
Step 5: Add the film and ceramic disc capacitors. These are not polarized and can be inserted into the PCB either way.

The capacitor highlighted in yellow is the ceramic disc cap. Your kit may come with a 47pF, 50pF, or 51pF depending on availability.

The caps highlighted in blue are where you would use a different value to perform modifications. Refer to page 7 for more info.

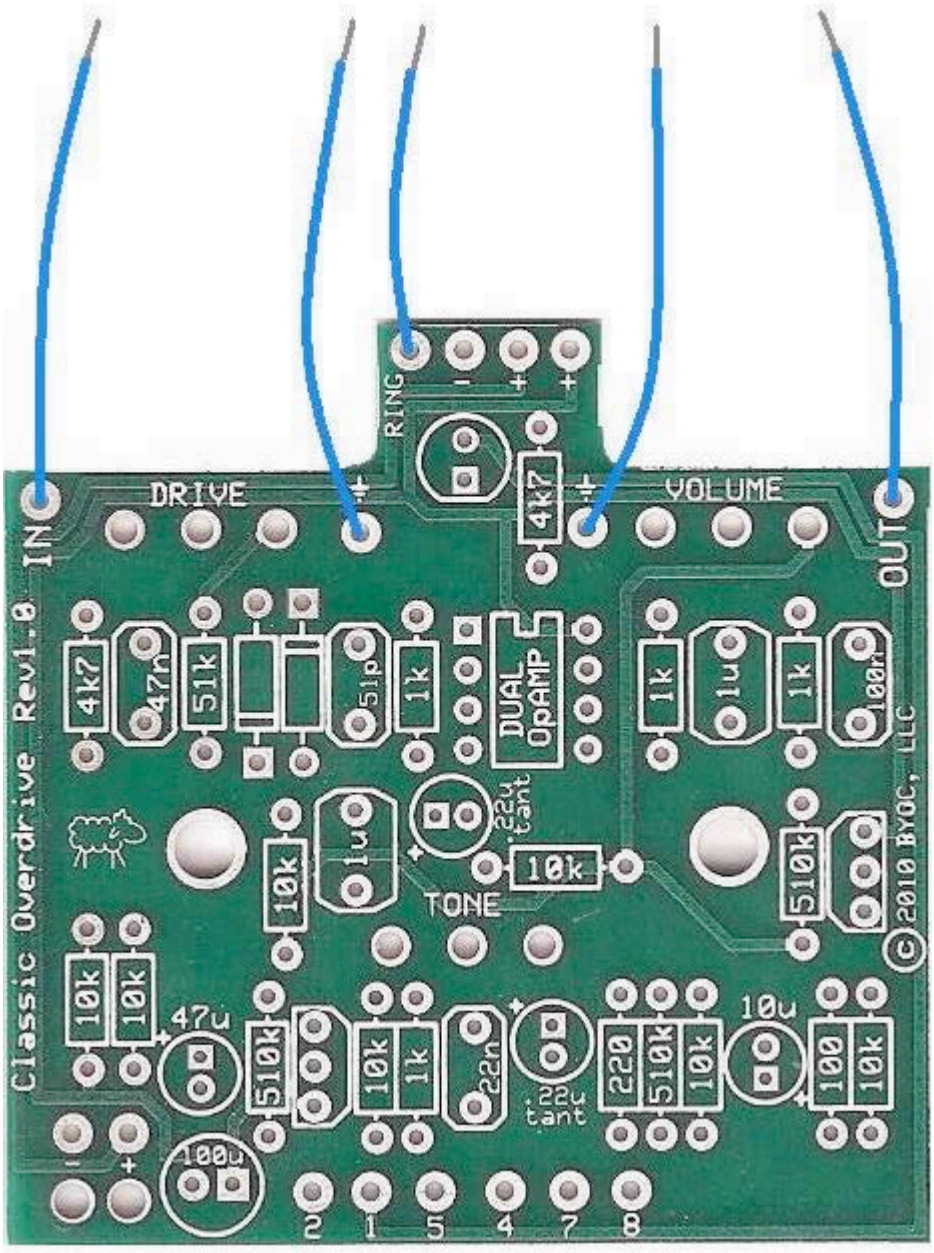


STEP 6: Add the aluminum electrolytic capacitors. These are polarized. The positive end will have a longer lead and should go in the square solder pad. The negative end will have a shorter lead with a black or white stripe running down the body of the capacitor.



Step 7: Add the battery snap. Thread the solder ends of the battery snap into the strain relief holes from the bottom solder-side 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.

The PCB says REV 1.0, but the wiring of the battery snap is exactly the same in all REV versions.

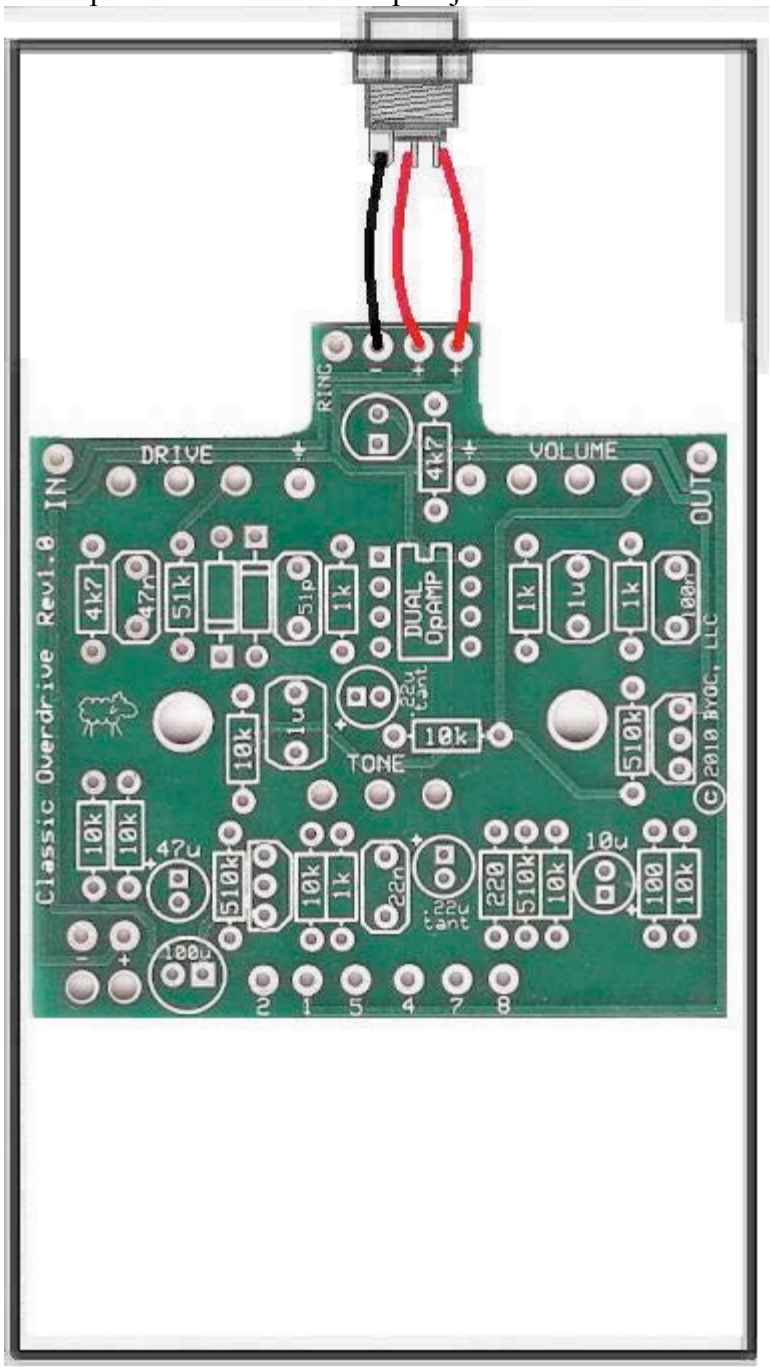


Step 8: Add wires to the IN, OUT, RING, and two Ground eyelets. Start by cutting four 2.5" pieces of wire and one 1.5" piece of wire. Strip 1/4" off each end and tin the ends. Tinning means to apply some solder to the stripped ends of the wires. This keeps the strands from fraying and primes the wire 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.

The PCB says REV 1.0, but the jack wiring is exactly the same through all the REV versions.

Main PCB Assembly

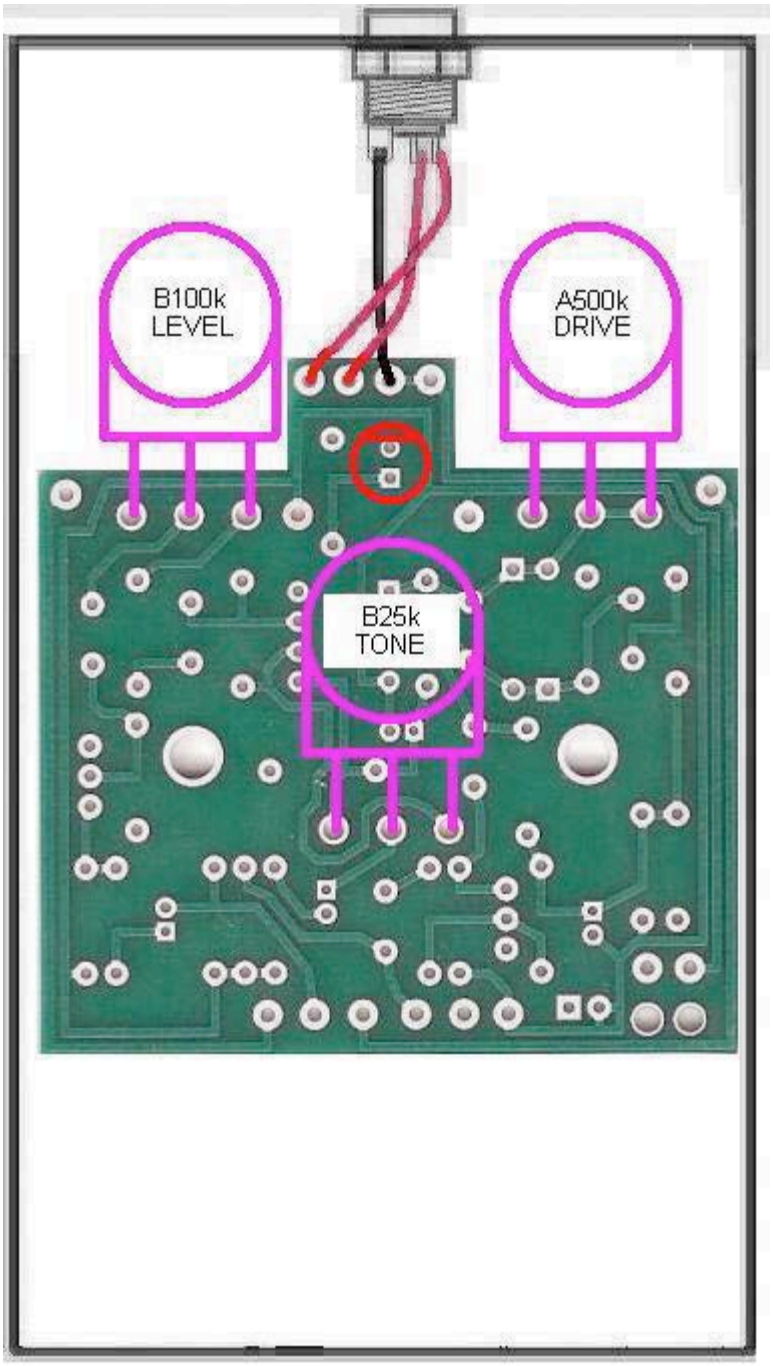
Step 1: Mount the DC adaptor jack to the enclosure.



Step 2: Connect the TIP (negative) terminal of the DC adaptor jack to the "-" eyelet on the PCB with 2

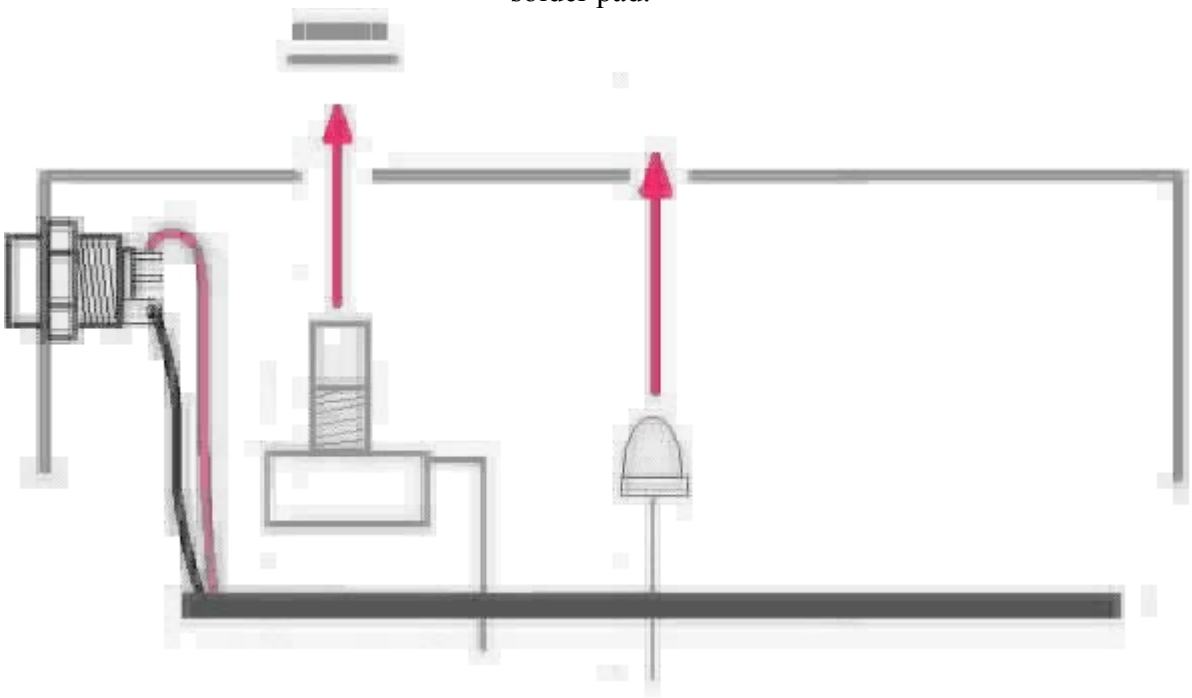
inches of hook up wire. Connect the SLEEVE of the DC adaptor jack to the "+" eyelet on the far right side of the PCB with 2 inches of hook up wire. Connect the battery disconnect terminal of the DC adaptor jack to the "+" eyelet more towards the center of the PCB with 2" of hookup wire. Load the wires in from the bottom of the PCB and solder on the topside.

The PCB says REV1.0, but the power wiring is the exact same through all the REV versions.



Step 3: Flip the PCB over so that the bottom or solder side is up. Insert the

B100k(LEVEL),A500k(DRIVE), B25K(TONE) potentiometers, and the LED into the bottom side of the PCB. DO NOT SOLDER ANYTHING YET!!! The LED will have one lead that is longer than the other. The longer lead goes in the hole with the square solder pad.

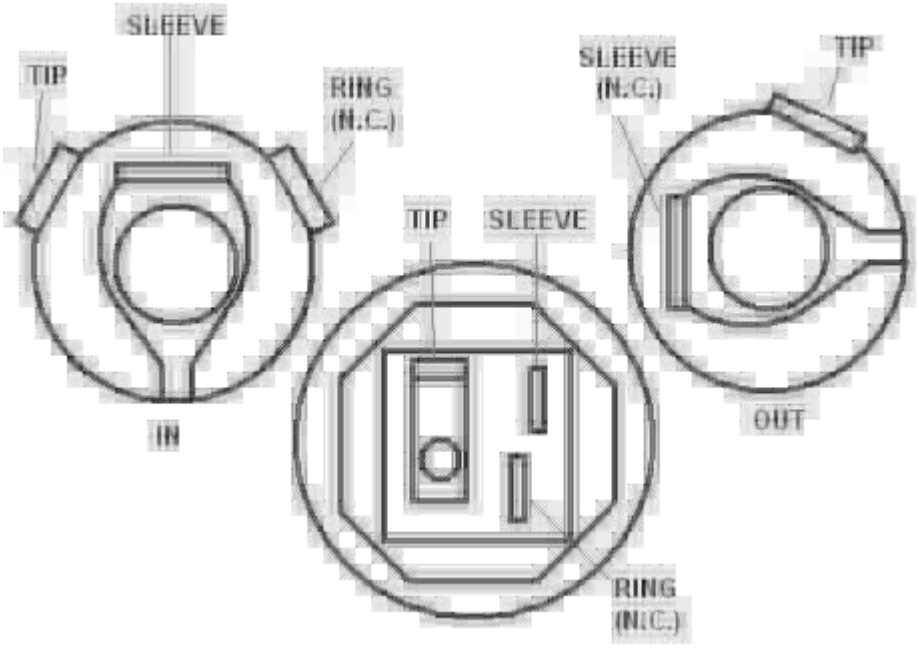


Step 4: Hold the PCB in one hand so that the component side of the PCB is in the palm of your hand and the bottom side with the pots, toggle switch and LED is facing up. Now use your other hand to guide the predrilled enclosure onto the PCB assembly so that the pots and LED all go into their respective 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.

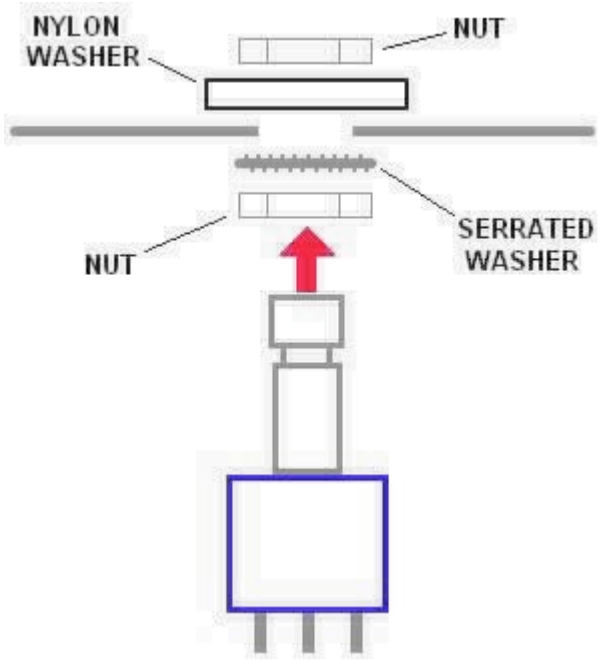
Step 5: Turn the entire pedal over so that the component side of the PCB if facing up. Lift the PCB up off the pots and toggle switch about 2mm just to make sure that the back of the PCB does not short out against that pots. Make sure the PCB is level and symmetrically seated inside the enclosure.

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

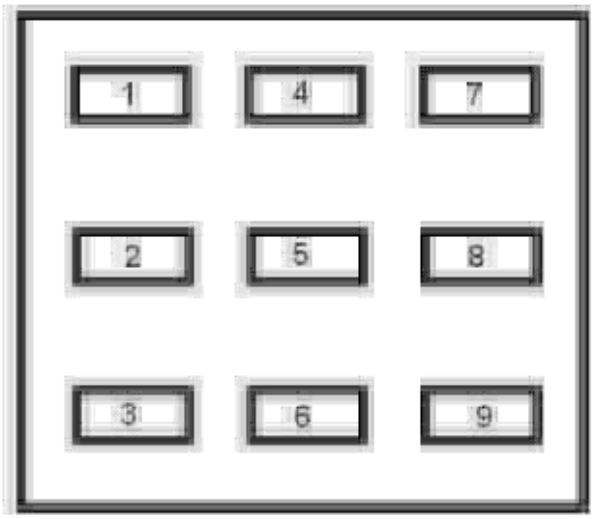


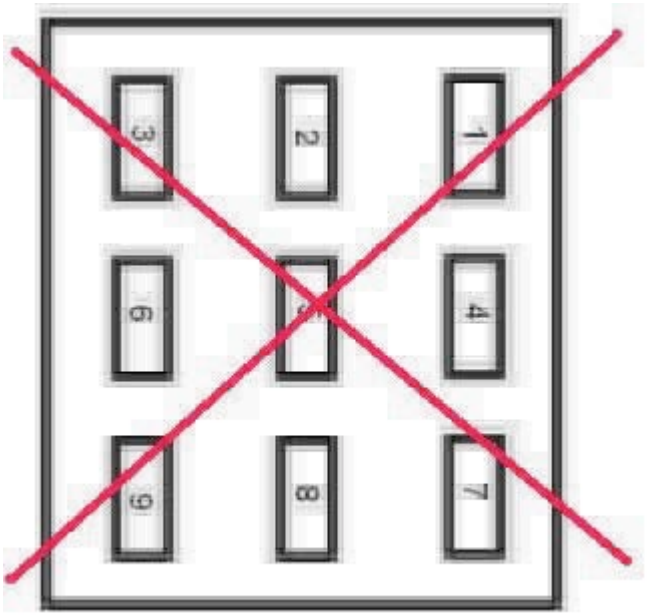
Step 1: Install the 1/4" jacks to the enclosure. Be sure to turn the OUT jack a 1/4 turn counter clockwise so that solder terminal for the tip does not short out against the enclosure.

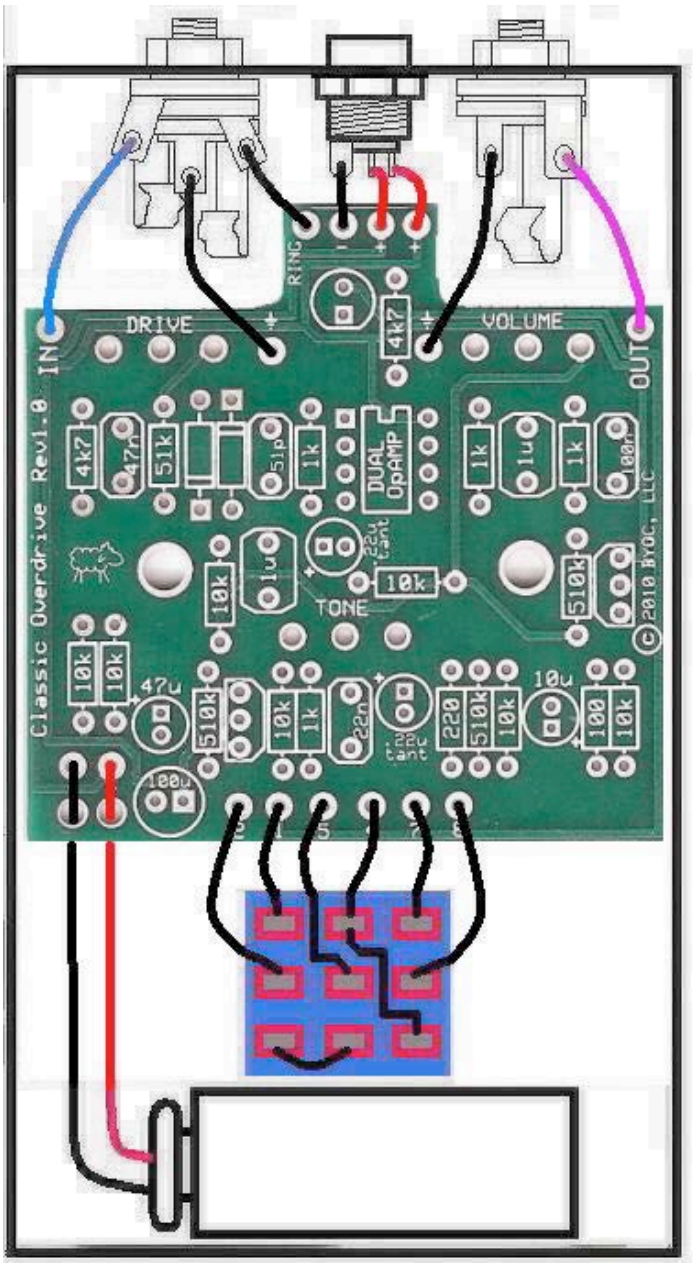


Step 2: Install the footswitch. Orient the footswitch so that the flat sides of the solder lugs are like the diagram below. NOTE: There are no actual number markings on the footswitch. There are two correct ways you can orient the footswitch. They are both 180 degrees of each other. Either way is fine.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

Footswitch Solder Lug Designations







The PCB says REV1.0, but the final wiring is the exact same for all the REV versions.

Step 3: Connect the pre stripped and tinned wires to the 1/4" jacks.

Step 4:

- Cut 4 x 3/4" pieces of wire. Strip 1/8" off each end. These will be used to connect lugs/eyelets 1, 2, 7, & 8
- Cut 1 x 1" piece of wire. Strip 1/8" off each end. This will be used to connect lug/eyelet 5
- Cut 1 x 1.5" peice of wire. Strip 1/8" of one end. Strip 1/2" off the other end. This will be used to connect lug/eyelet 4. The longer stripped end will be used to jumper

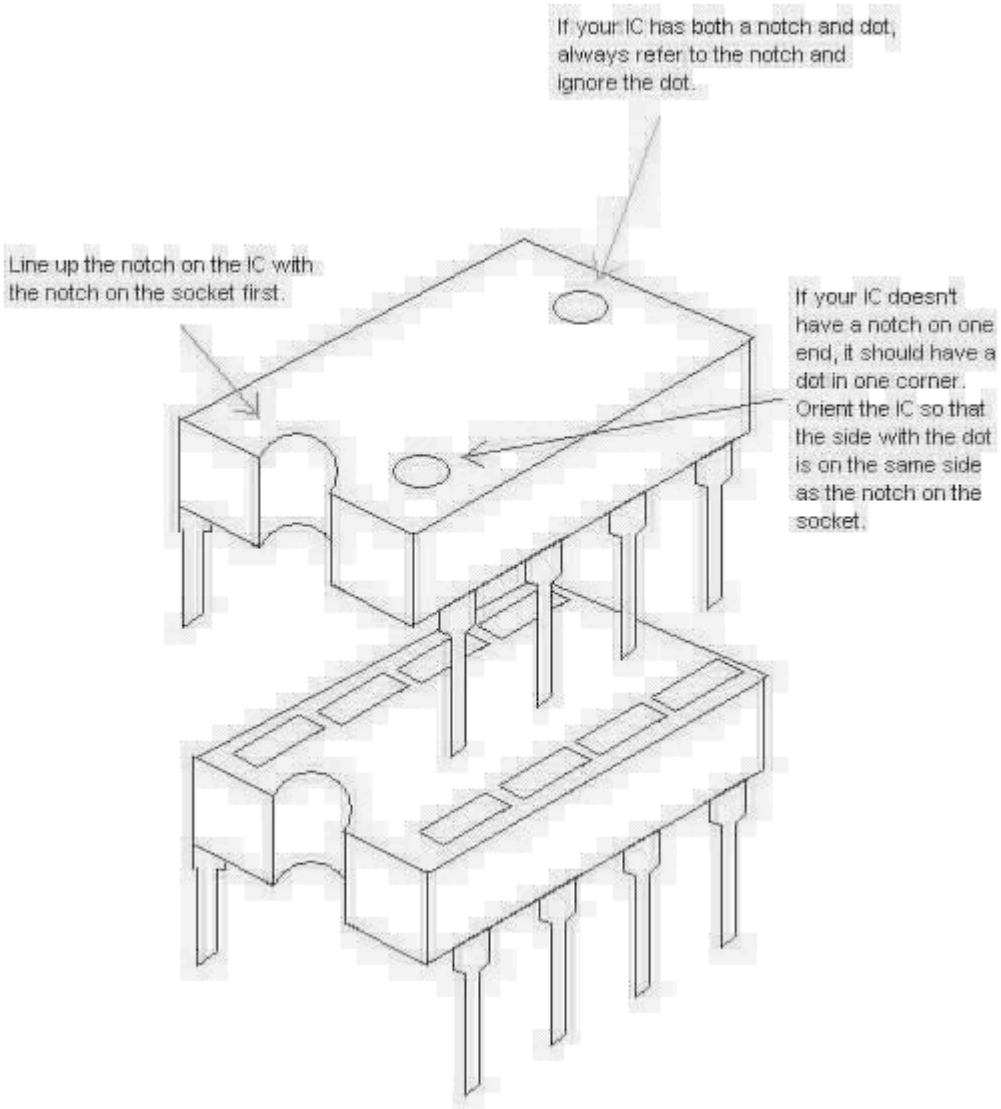
lug 4 to 9.

- 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” peice 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.

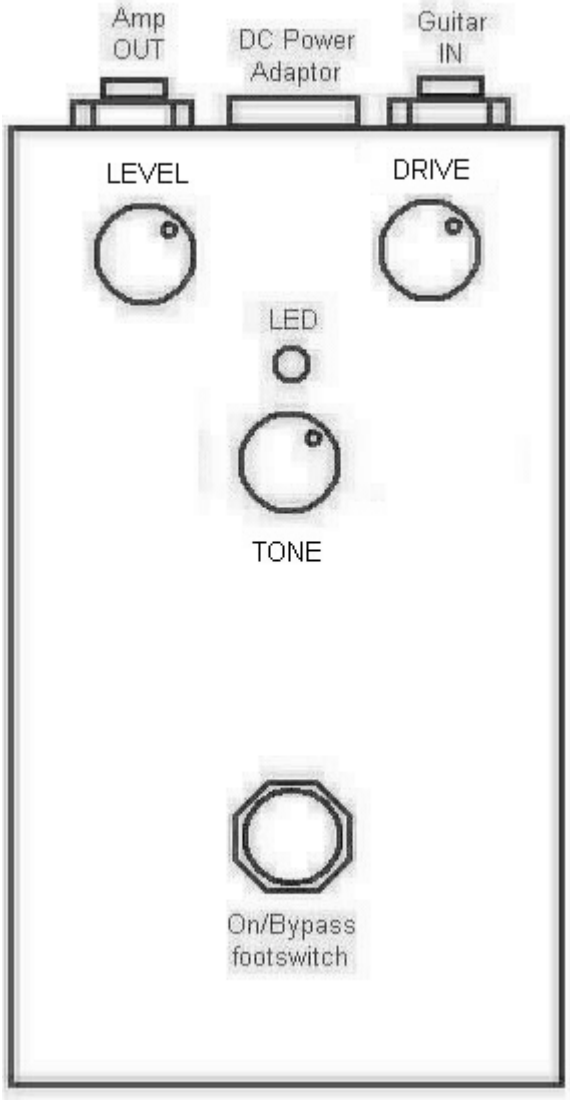
Step 6: Insert the other remaining ends of the pre-cut and pre-stripped wires into the topside of the PCB and solder. You can can solder these on the topside as well. It is easier this way, but you may burn a small amount of the PVC coating on the wires. This is purely asthetic and won't damage the wires 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 ICs and Finishing Up



Don't forget to put the cover on the enclosure and apply the bumpers to the cover if you like to use them.

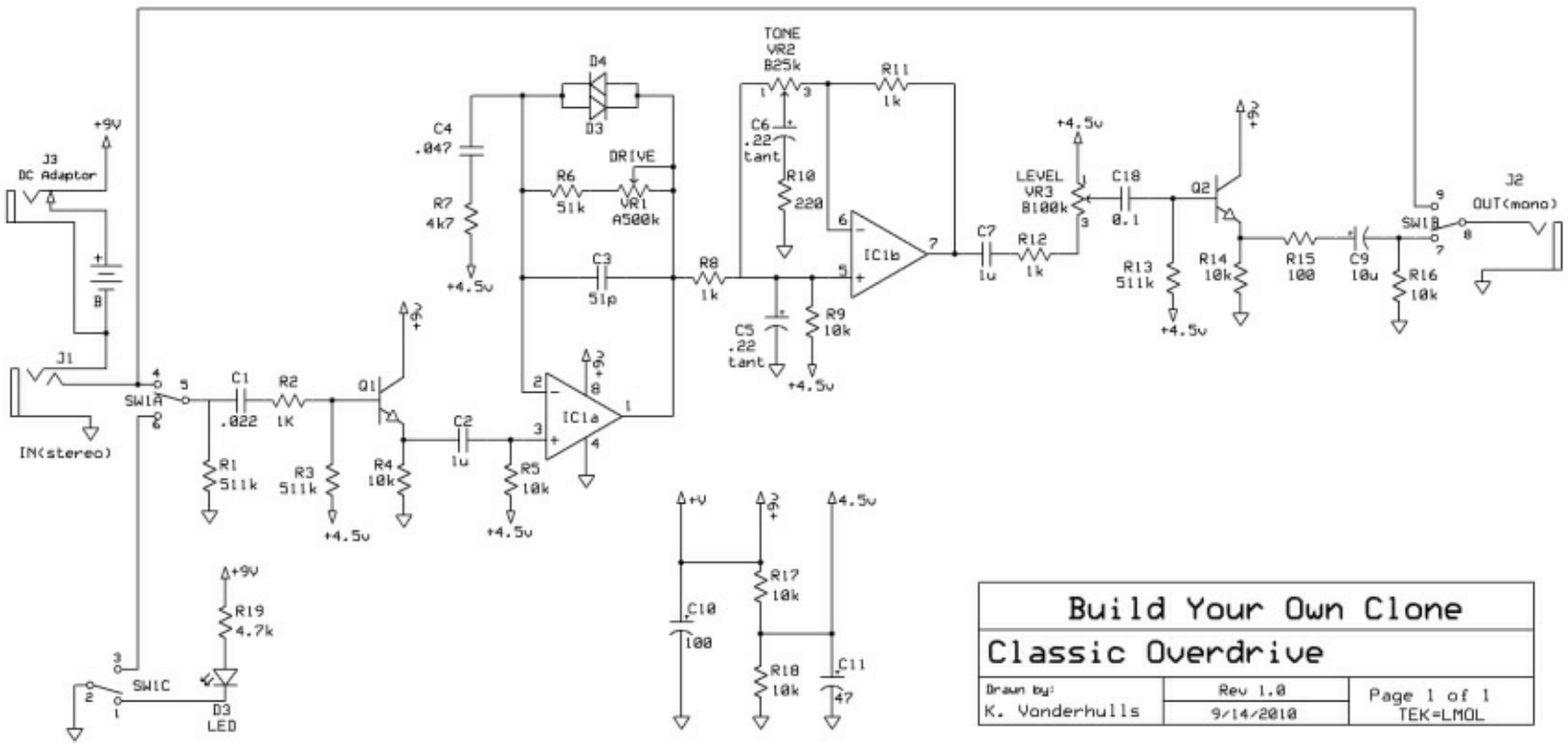
Operating Overview



LEVEL: Controls the overall output volume.
DRIVE: Controls the amount of overdrive.
TONE: Controls the tone of the effect.

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

Current Draw - ~2.5mA
Input Impedance - 511k ohms
Output Impedance - 10k ohms



Build Your Own Clone		
Classic Overdrive		
Drawn by: K. Vanderhulls	Rev 1.0 9/14/2010	Page 1 of 1 TEK=LMOL

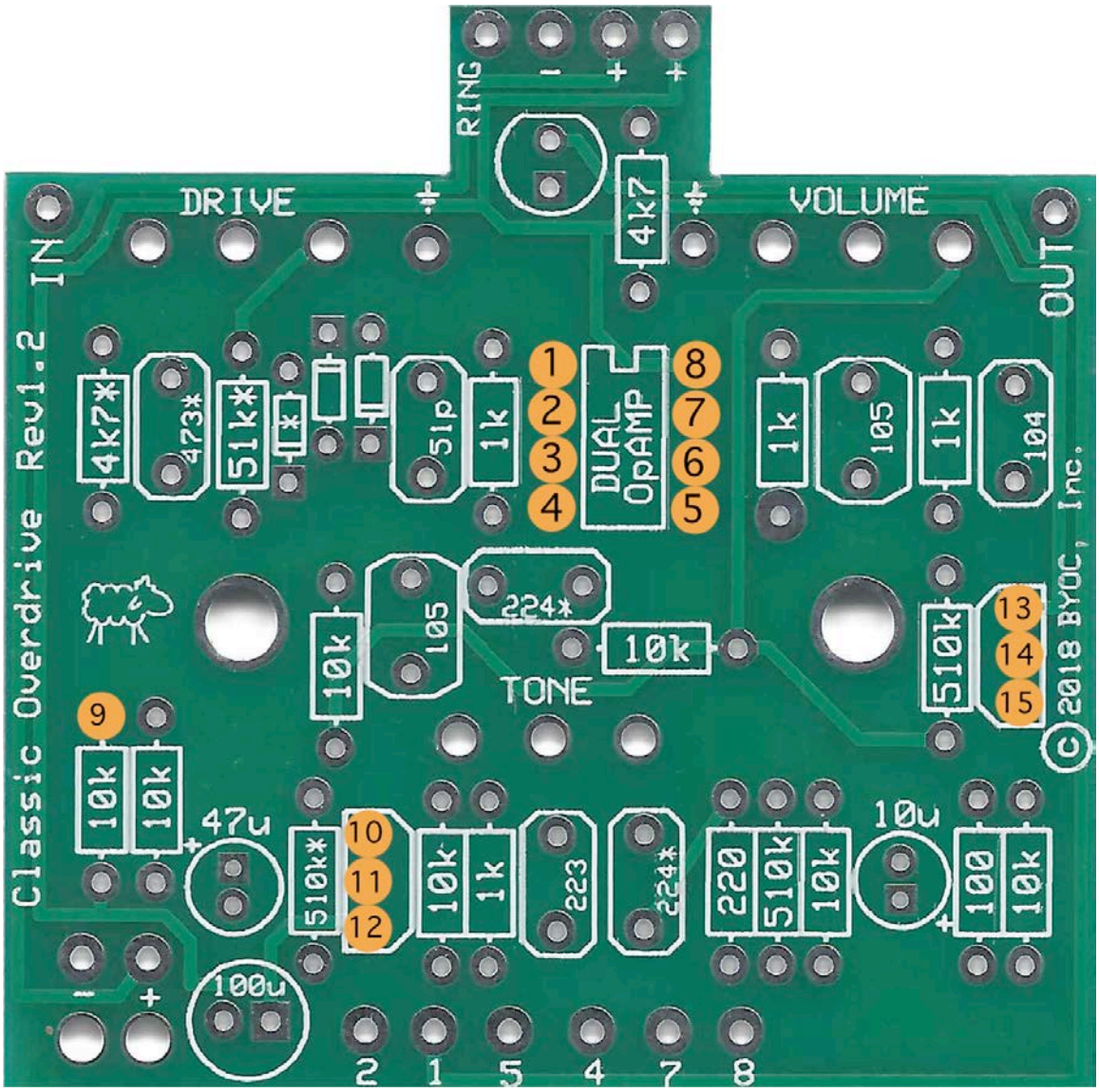
Trouble Shooting

The overwhelming majority of our tech support issues are due to poor soldering….lazy, sloppy, impatient soldering. But even careful experienced pedal builders still make bad solder joints on occasion. If your pedal doesn't fire up right out of the gate, you should reflow all your solder joints. You do this by reheating the solder joint with your iron and applying a tiny bit of fresh solder. **This is the first thing you should do**, even if you are certain your soldering is perfect. Here are some other common issues.

1. Problem: The pedal works in every way, but the LED doesn't come on. Reason: You probably oriented your LED backwards
2. Problem: The pedal works, but it is extremely quiet and/or has low distortion. Reason: You probably have one or more of your resistors mixed up. The R15 100ohm resistor is commonly mistaken for a different value.
3. Problem: Bypass works, but the effect doesn't work and the LED doesn't come on. Reason: You probably didn't wire your DC adapter jack and or foot switch correctly. Or you are using a dead battery.
4. Problem: Bypass works, the LED comes on, but the effect doesn't work. Reason: You might have your input and output cables mixed up, or you have a problem on the PCB and you should take voltage readings.
5. Problem: All I get is a loud hum. Reason: You probably don't have a proper ground connection to the sleeves of your input and/or output jack(s).
6. Problem: The op amp is getting very hot. Reason: You either put the op amp in backwards or you used a positive tip power supply. The op amp is ruined and you will need to replace it.

Taking Voltage Readings. Before you even bother taking voltage readings, YOU MUST have bypass working correctly and the LED status light working correctly. The LED and bypass are indicators that you have power and ground to your circuit and that your jacks and footswitch are wired correctly. If these aren't working correctly, nothing else will work correctly.

1. Make sure your pedal has power. This not only means connecting a 9V battery or 9VDC 2.1mm negative tip power supply. This also means plugging in a cable into the input jack. The input jack acts as the power on/off switch.
2. Set your DMM (digital multimeter) to read DC Voltage for at least 9 volts.
3. Connect the black probe plug to the COM terminal of the DMM and the red probe plug to the to the Volts/ohms terminal of the DMM.
4. Now you are ready to take voltage readings. Touch the black probe to ground. The sleeve or either jack is a good ground point. Touch the red probe to the various test points show in the following diagram.



The following voltages are based on 9.0VDC supply voltage. Your voltages may vary by a few mV depending on your exact supply voltage.

TP1: 4.5VDC
TP2: 4.5VDC
TP3: 4.5VDC
TP4: 0 VDC
TP5: 4.5VDC
TP6: 4.5VDC
TP7: 4.5VDC
TP8: 9VDC

TP9: 4.5VDC
TP10: 3.6VDC
TP11: 2.8VDC
TP12: 9VDC
TP13: 9VDC
TP14: 2.8VDC
TP15: 3.6VDC

If test points 8, 12, and 13 measure 0, or close to zero, this could mean that you are not getting power to your PCB. 8, 12, and 13 all connect to the +9V power supply, so you should be reading the exact same voltage as your power supply. You could however, also have a short along your power rail to ground. Set your DMM to test for continuity (the little speaker or horn icon) and touch the black probe to ground and the red probe to TP8. If the DMM makes a buzzing noise, you have a short.

If test points 1 – 3, 5 – 7, and/or 9 measure significantly lower then 4.5VDC, remove the Op Amp from its socket and retest the voltages at 3, 5, and 9.

> If 3, 5, and 9 all test at 4.5VDC with the Op Amp out of its socket, then the Op Amp has likely been damaged.
>
> If 3, 5, and 9 all test significantly off from 4.5VDC with the Op Amp out of its socket, it's likely that you have an incorrect resistor value at R17 or R18. These would be the two 10k resistors in the lower left hand corner of the PCB just bellow the sheep logo.

If test points 10 and 11 are significantly off, you likely have the Q1 transistor in backwards or you have and incorrect resistor value at R4. Q1 is the transistor closest to the "2" eyelet. R4 is the resistor closest the "1" eyelet. It is also possible that Q1 is damages, but only if TP9 is correctly reading 4.5VDC

If test points 14 and 15 are significantly off, you likely have the Q2 transistor in backwards or you have an incorrect resistor value at R14. Q2 are R14/10k are the transistor and resistor closest to the copyright symbol. It is also possible that Q2 is damaged, but only if TP9 is correctly reading 4.5VDC

If you need further technical support, please visit
http://buildyourownclone.com/board