

Build Your Own Clone Overdrive 2 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.

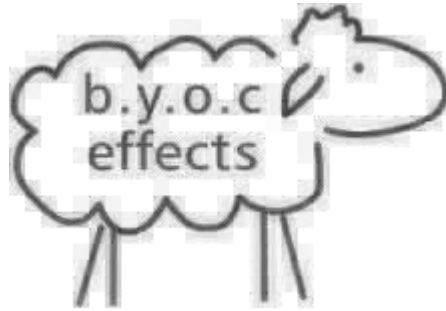
Revision Notes:

Rev 1.0 There is a trace touching one of the eyelets for the “mids” toggle switch.

Rev 1.1 (released 2/5/09) Fixed the trace error.

Copyrights:

All material in this document is copyrighted 2009 by BYOC, LLC



OVERDRIVE 2 KIT INSTRUCTION INDEX

Parts Checklist.....	page 4 - 6
Standard Spec Build Notes.....	page 7 - 8
MOSFET Spec Build Notes.....	page 9 - 11
Populating the Circuit Board.....	page 12 - 22
Assembly.....	page 23 - 25
Wiring the Footswitch & Jacks.....	page 26 - 28
Installing the ICs.....	page 29
Setting up the Internal Trimpots.....	page 30 -31
More Modifications.....	page 32 - 35
Schematic.....	page 36 -37

Parts Checklist for BYOC Overdrive 2

Resistors:

- 1 - 100ohm (brown/black/black/black/brown)
- 1 - 220ohm (red/red/black/black/brown)
- 1 - 470ohm (yellow/purple/black/black/brown)
- 3 - 1k (brown/black/black/brown/brown)
- 2 - 4.7k (yellow/purple/black/brown/brown)
- 8 - 10k (brown/black/black/red/brown)
- 1 - 100k (brown/black/black/orange/brown)
- 2 - 511k (green/brown/brown/orange/brown)
- 2 - 1M (brown/black/black/yellow/brown) or (brown/black/green/gold)

Capacitors:

- 1 - 51pf ceramic disc (small orange cap)
- 1 - .022 μ f/22n film (223)
- 2 - 0.047 μ f/47n film (473)
- 2 - 0.1 μ f/100n film (104)
- 1 - 0.15 μ f/150n film (154)
- 2 - 1 μ f/1 μ 0 film (105 or 1J63)
- 2 - 10 μ f aluminum electrolytic
- 1 - 47 μ f aluminum electrolytic
- 1 - 100 μ f aluminum electrolytic
- 2 - 0.15 μ f tantalum (154)
- 2 - 0.22 μ f tantalum (225)

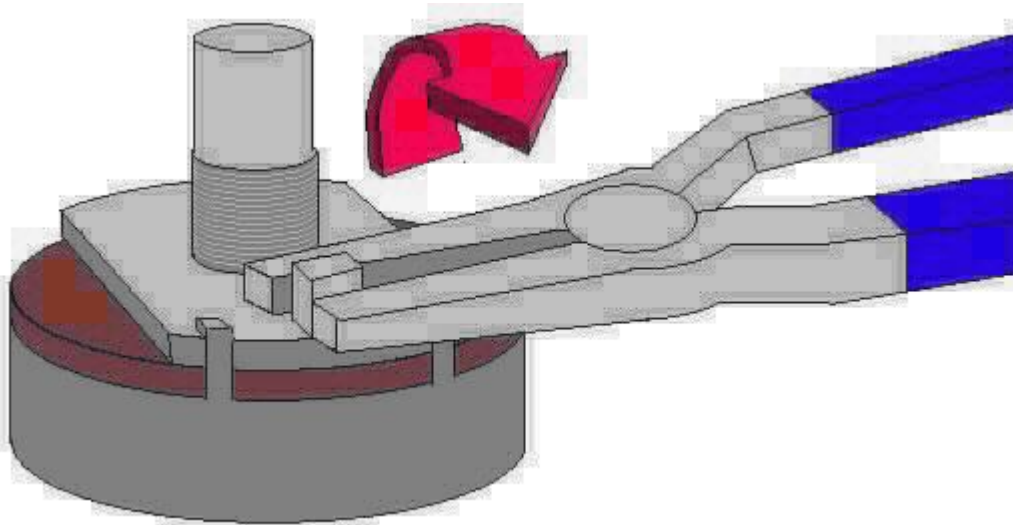
Diodes:

- 2 - 1N914 or 1N4148 (small orange glass)
- 2 - 3mm red LED

Transistors:

- 3 - MPSA18 (or 2N3904)

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 (OD level knob)
- 1 - A500k audio (drive knob)
- 1 - A100k audio (boost level knob)
- 1 - 10k “max” trimpot (will be labelled “103”)
- 1 - 50k “louder” trimpot (will be labelled “503”)
- 1 - 100k “min” trimpot (will be labelled “104”)

IC's:

- 1 - JRC4558D
- 1 - OP2134
- 1 - DIP 8 socket

Hardware:

- 1 - drilled enclosure w/ 4 screws
- 1 - byoc OD2 circuit board
- 2 - SPDT PC mounted on-off-on toggle switch
- 2 - 3PDT footswitch
- 4 - knobs
- 1 - AC adaptor jack
- 1 - ¼”mono jack
- 1 - ¼”stereo jack
- 2 - red LED
- 1 - battery snap
- hook-up wire

MOSFET CONVERSION KIT

Resistors:

1 - 4.7k (yellow/purple/black/brown/brown)

5 - 1M (brown/black/black/yellow/brown)

Diodes:

6 - 1N914 or 1N4148 (small orange glass)

Transistors:

5 - BS170 MOSFET **(these go in backwards from the transistor labels on the PCB!)**

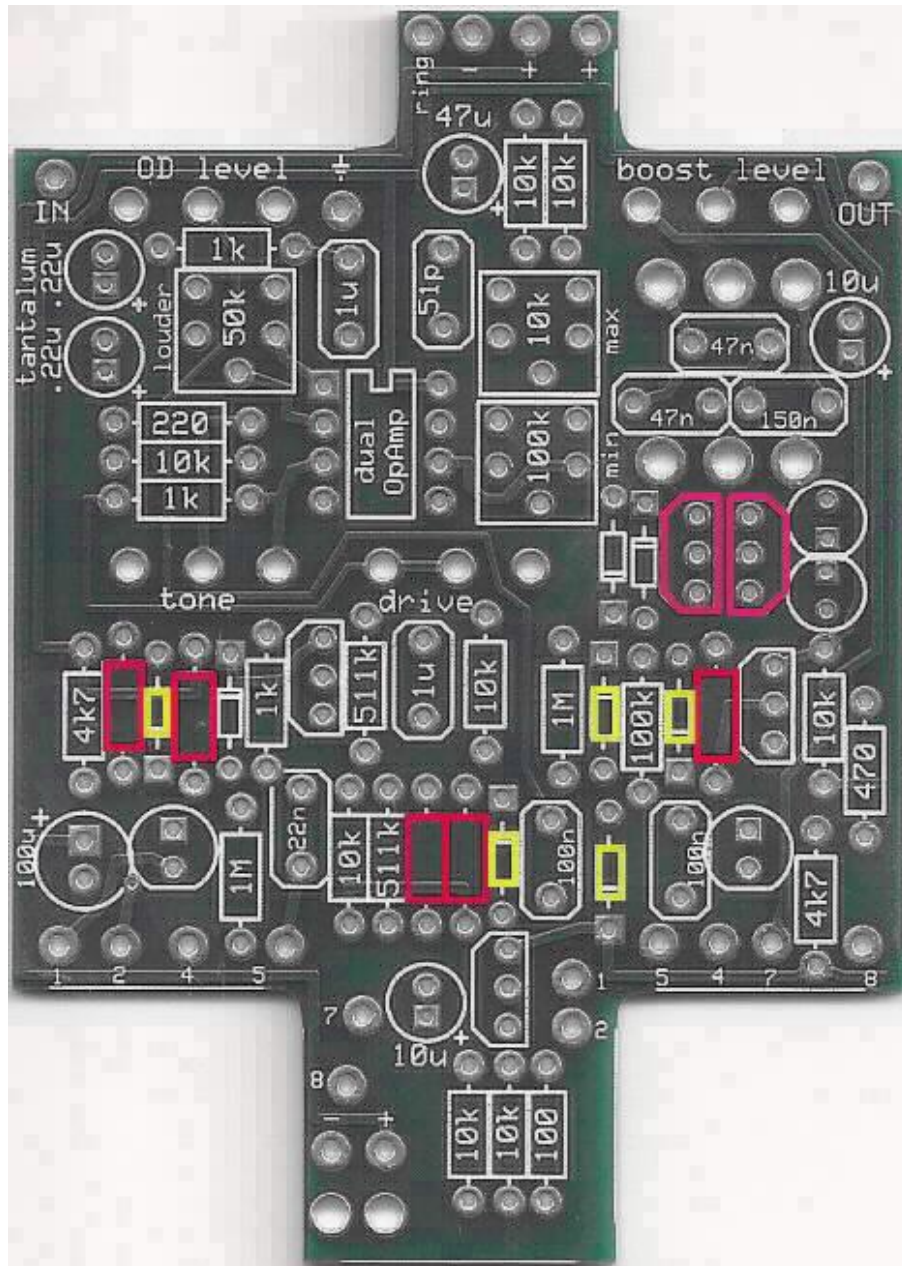
IC's:

1 - CA3260 Dual MOSFET OpAmp

Potentiometer:

1 - B10k linear (tone knob)

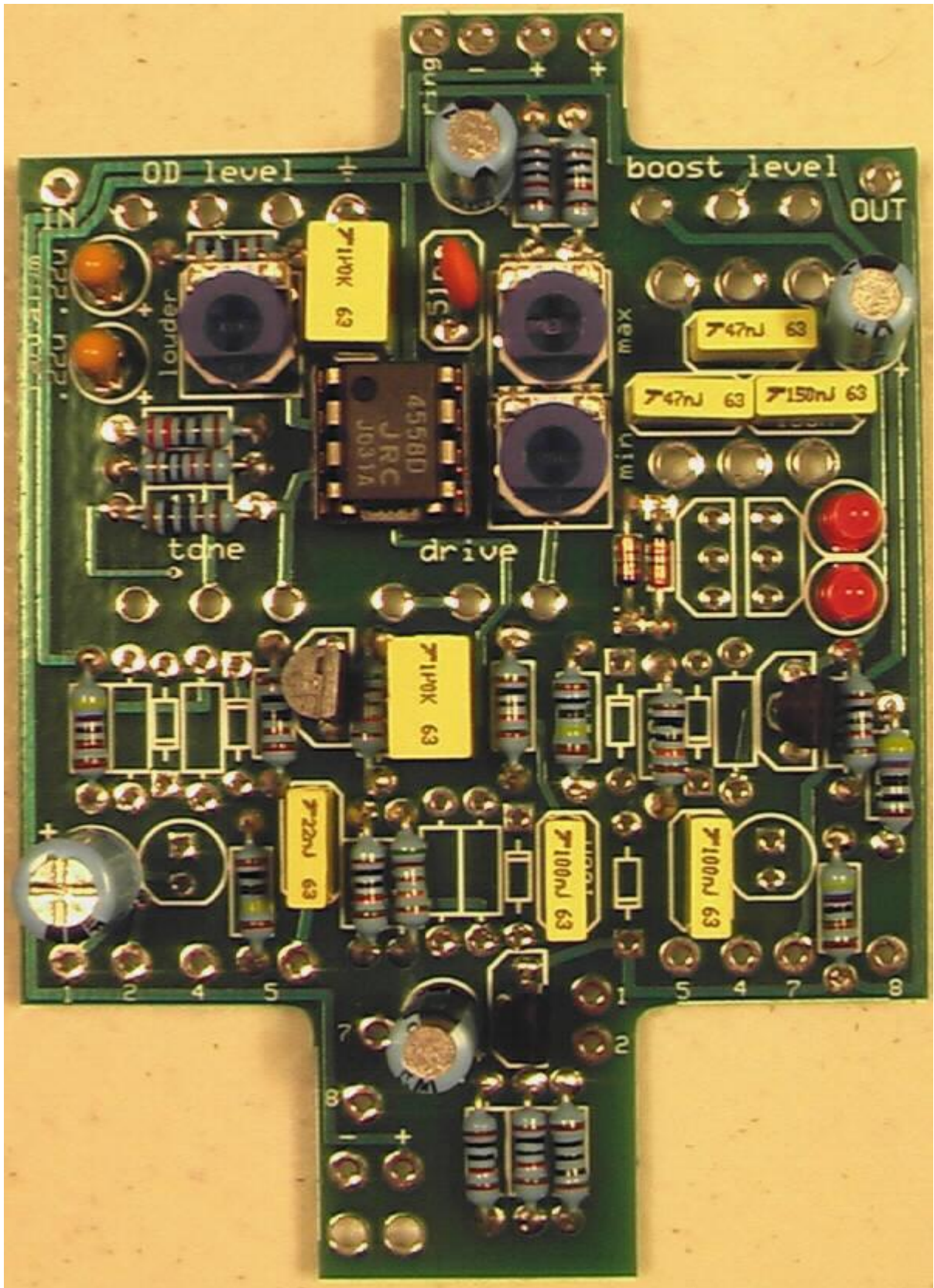
Standard Spec Build Notes



If you are building your Overdrive 2 kit to standard tubescreamer specs, before you begin, you should be aware that there are several spaces on the PCB that should be left empty.

There are 5 resistor spaces, 6 diode spaces, and 2 transistors spaces, highlighted in the diagram above that should be left empty. Do not add any components to these spaces. Omit these components. There are no component values that need to be changed from what is labelled on the PCB.

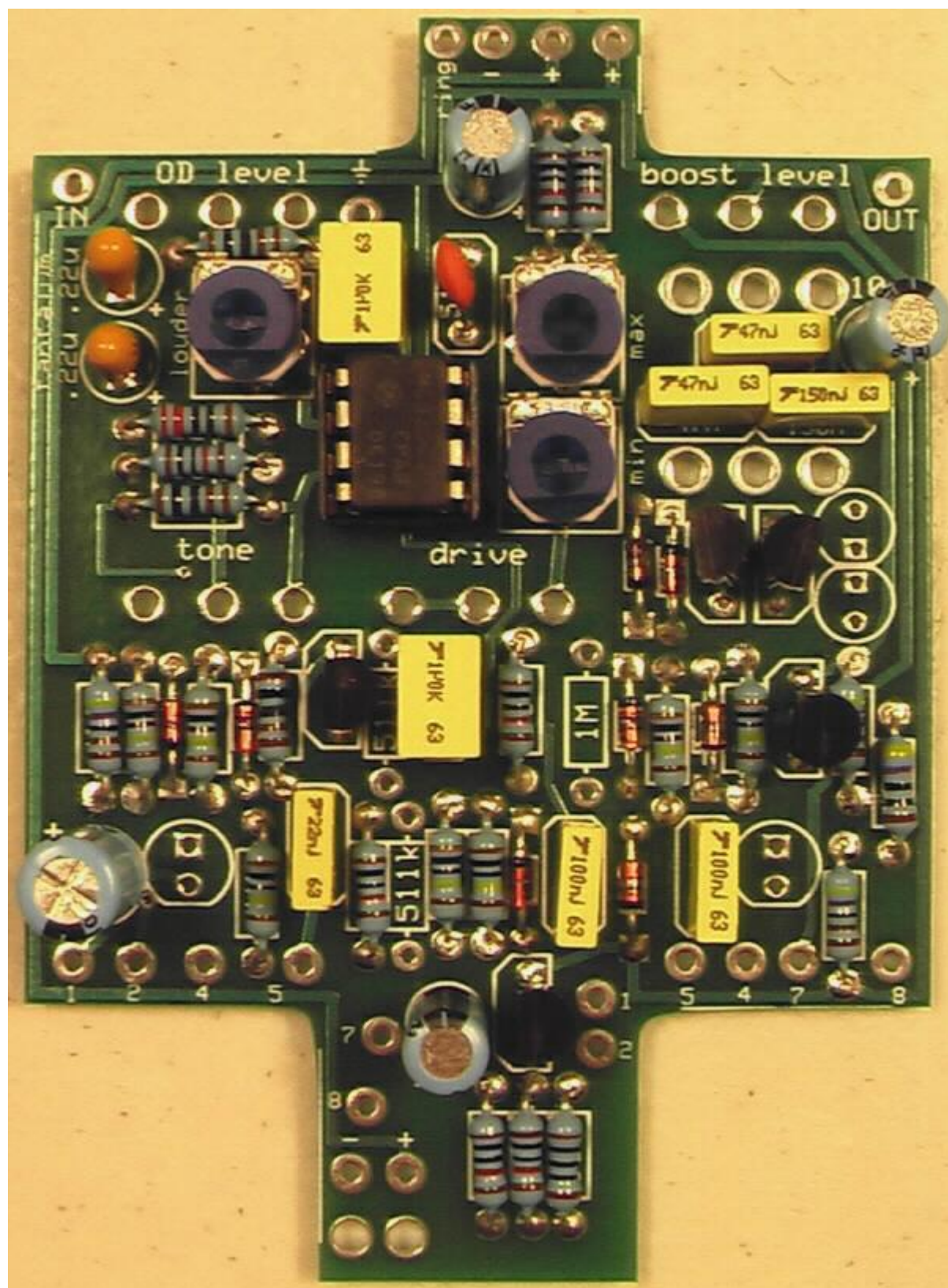
Just be aware of this for now. You will be reminded throughout the “populating the PCB” portion of the instructions as we come to each relevant component.



1. Leave the one 1M resistor space, the two 511k resistor spaces, and the two red LED clipping diode spaces highlighted in RED empty. Do not add any components to these spaces.
2. Use 1M resistors in all the resistor spaces that are highlighted in orange.
3. Change the 100k resistor highlighted in yellow to a 1M resistor.

4. Change the 10k resistor highlighted in yellow to a 4.7k resistor.
 5. Change the 3 MPSA18 transistors highlighted in pink to BS170 MOSFETs. **The BS170 MOSFET transistors need to go in backwards.**
- 80""Tgræg'vj g'47nltqpg't qvgpvkqo gvg't'y kj 'vj g'32nlt qvgpvkqo gvg't0**

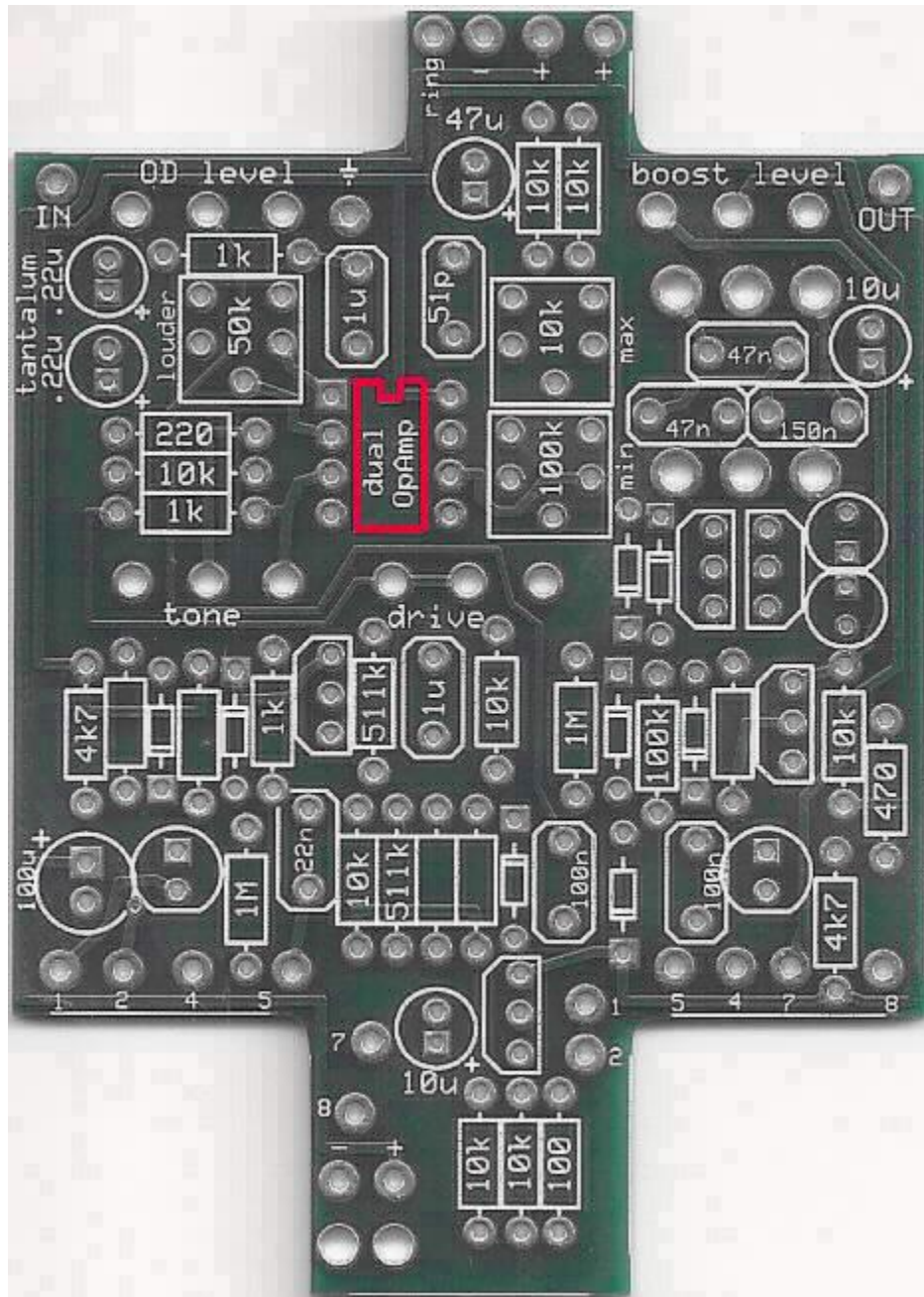
Just be aware of this for now. You will be reminded throughout the “populating the PCB” portion of the instructions as we come to each relevant component. All of the other typical “tubescreamer” mods will still apply to the circuit even after MOSFET conversion.



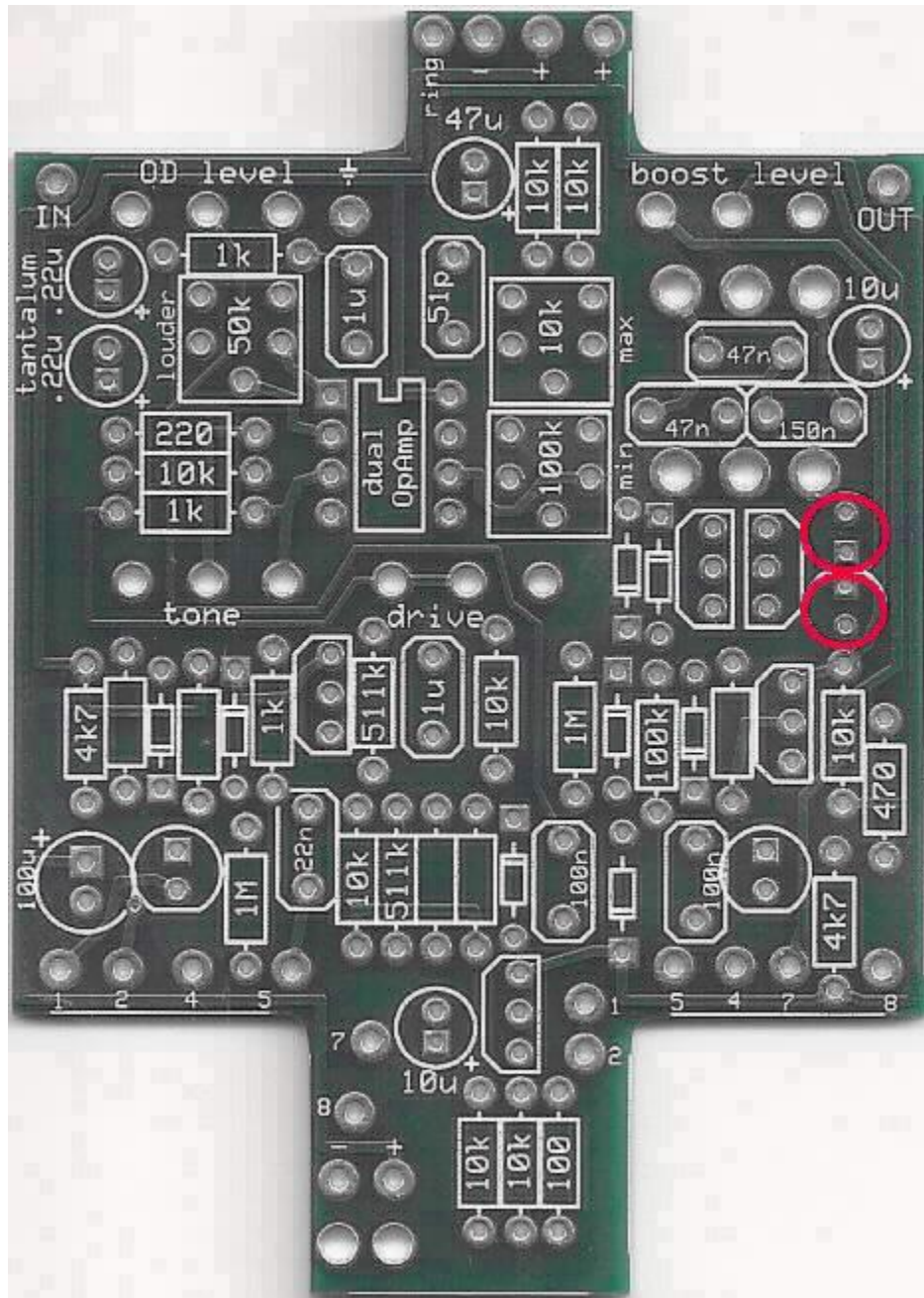
STEP 1: Add the diodes highlighted in red. Be sure to match the end of the diode with the stripe to the layout on the PCB. The stripped end should go in the square solder pad. Only add the diodes highlighted in yellow if you are building to MOSFET specs. If you are building to standard specs, leave them empty.

- Add all the resistors highlighted in RED according to the values labelled on the PCB.
- Omitt the two 511k and one 1M highlighted in pink and leave these three spaces empty.
- Add 1M resistors to the 5 blank spaces highlighted in green.

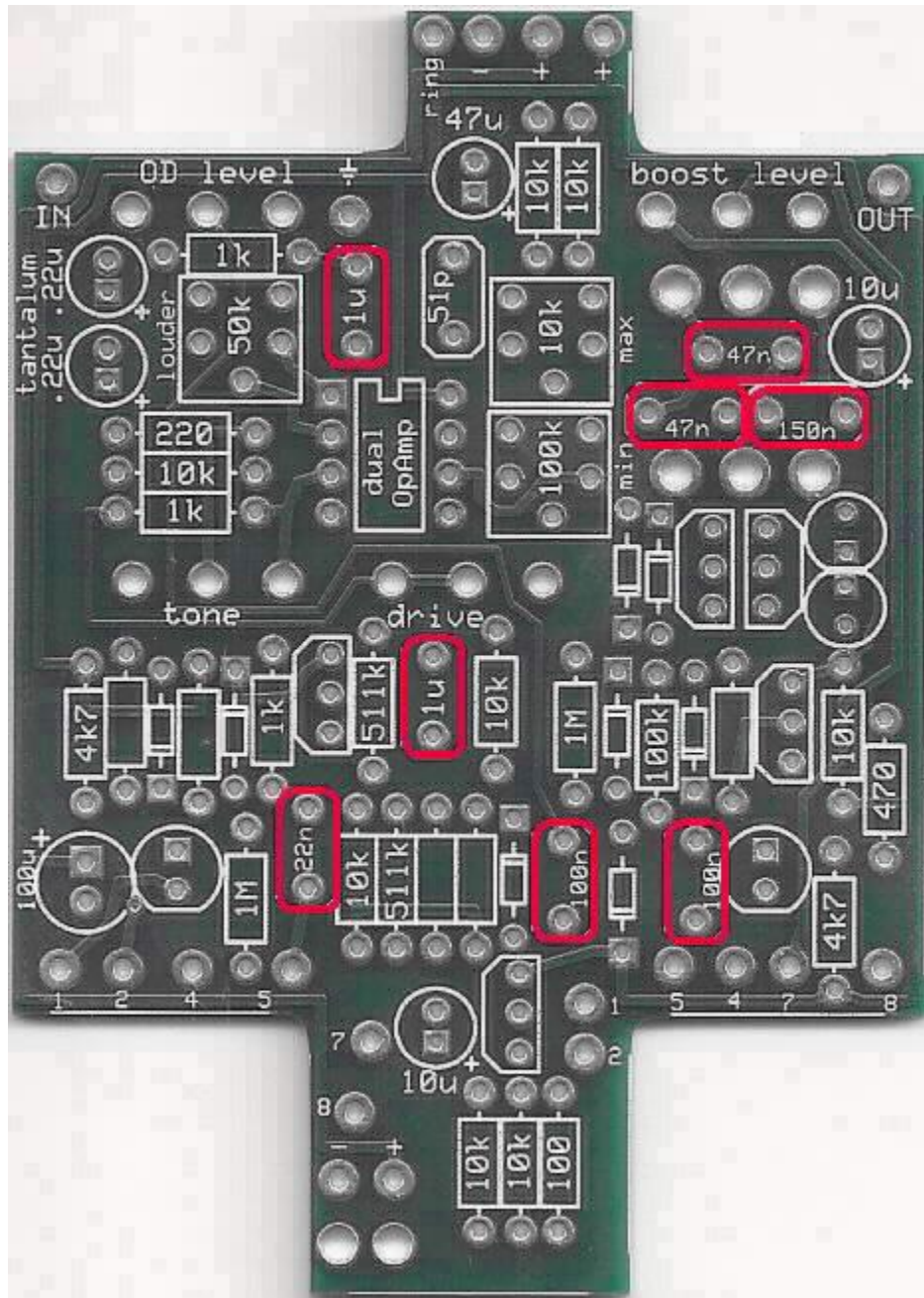
- d. Change the 100k to 1M and change the 10k to 4.7k in the spaces highlighted in yellow.



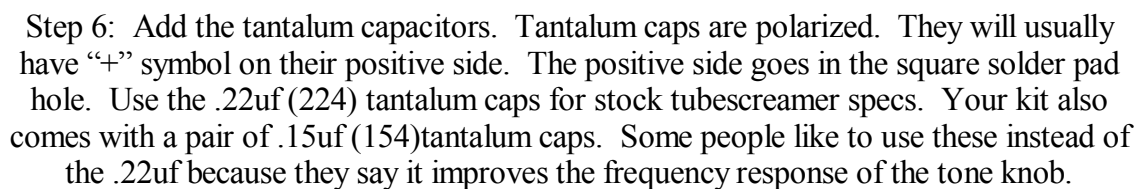
Step3: Add the 8 pin socket. Line up the socket so that the “u-shaped” notch matches up with the end that has a notch on the layout. Only add the socket. DO NOT solder the actual chip to the PCB. DO NOT solder the chip to the socket. Only the socket gets soldered. You will never solder the chip(s). Installing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

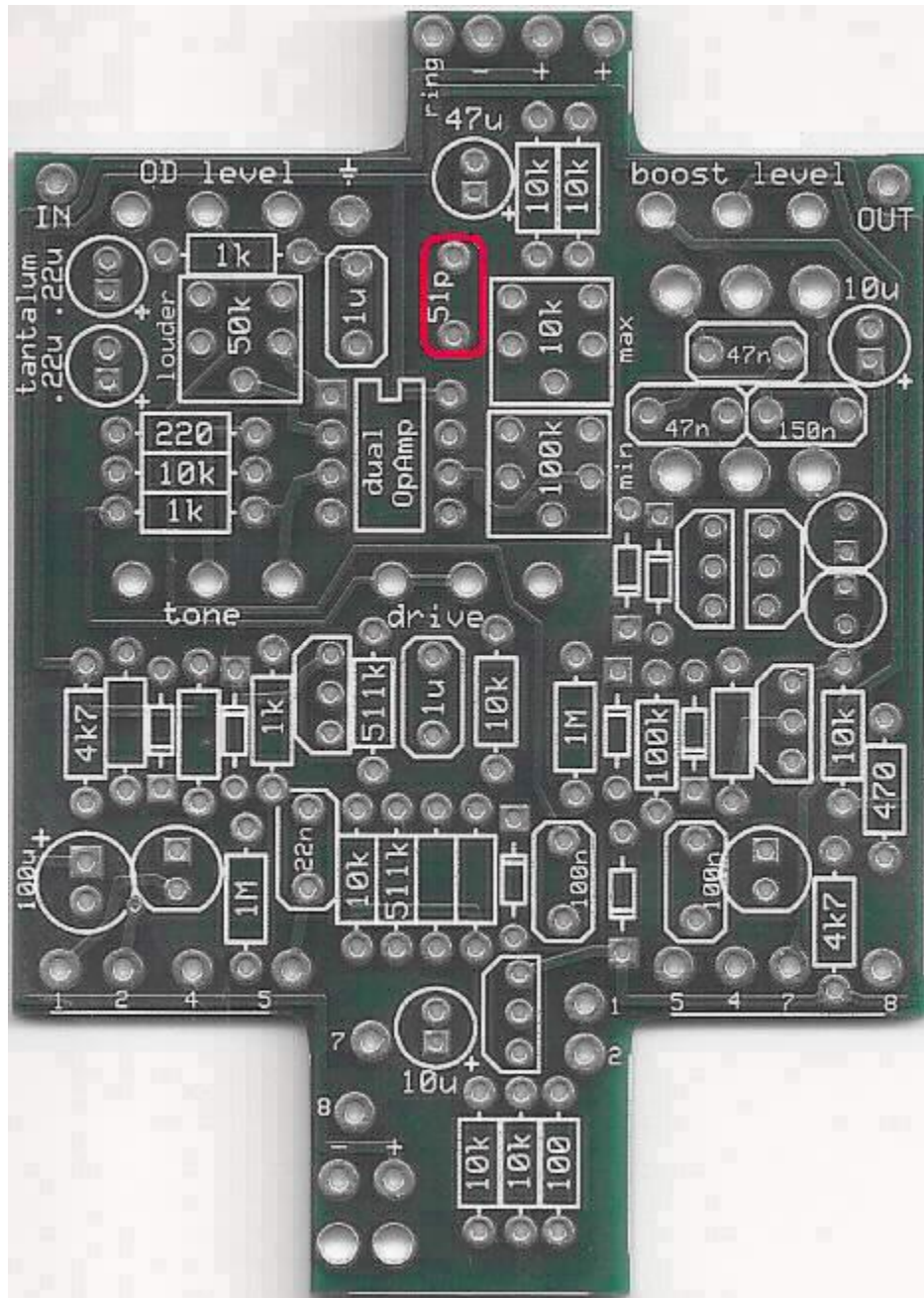


Step 4: Add the 3mm red clipping LEDs. The LEDs will have one lead that is longer than the other. The longer lead goes in the square solder pad hole. The shorter lead goes in the round solder pad hole. DO NOT add these if you plan on using the MOSFET clipping. You cannot use both at the same time. You can only use one or the other.

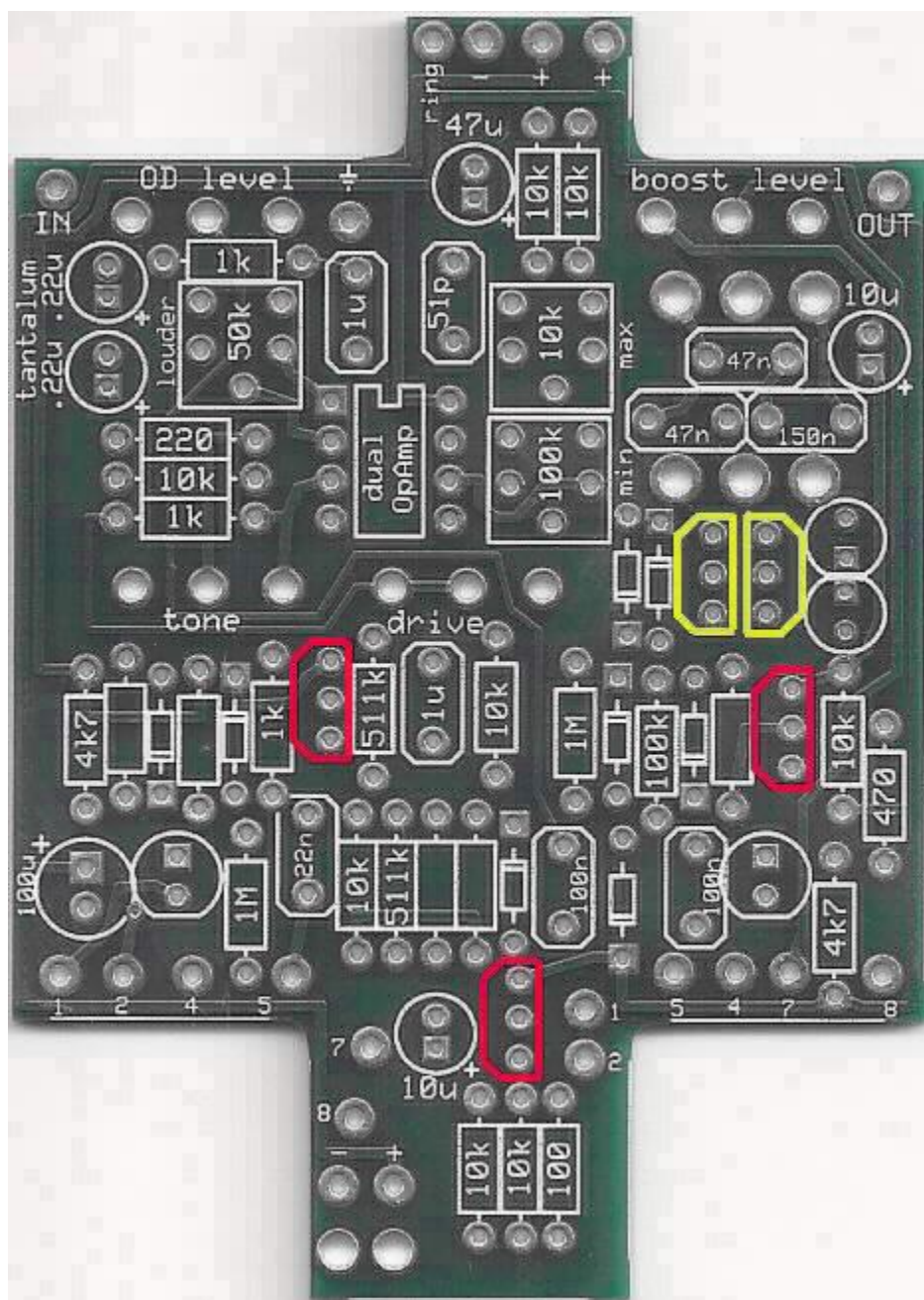


STEP 5: Add the film capacitors. These are not polarized so they can be inserted into the PCB in either direction.





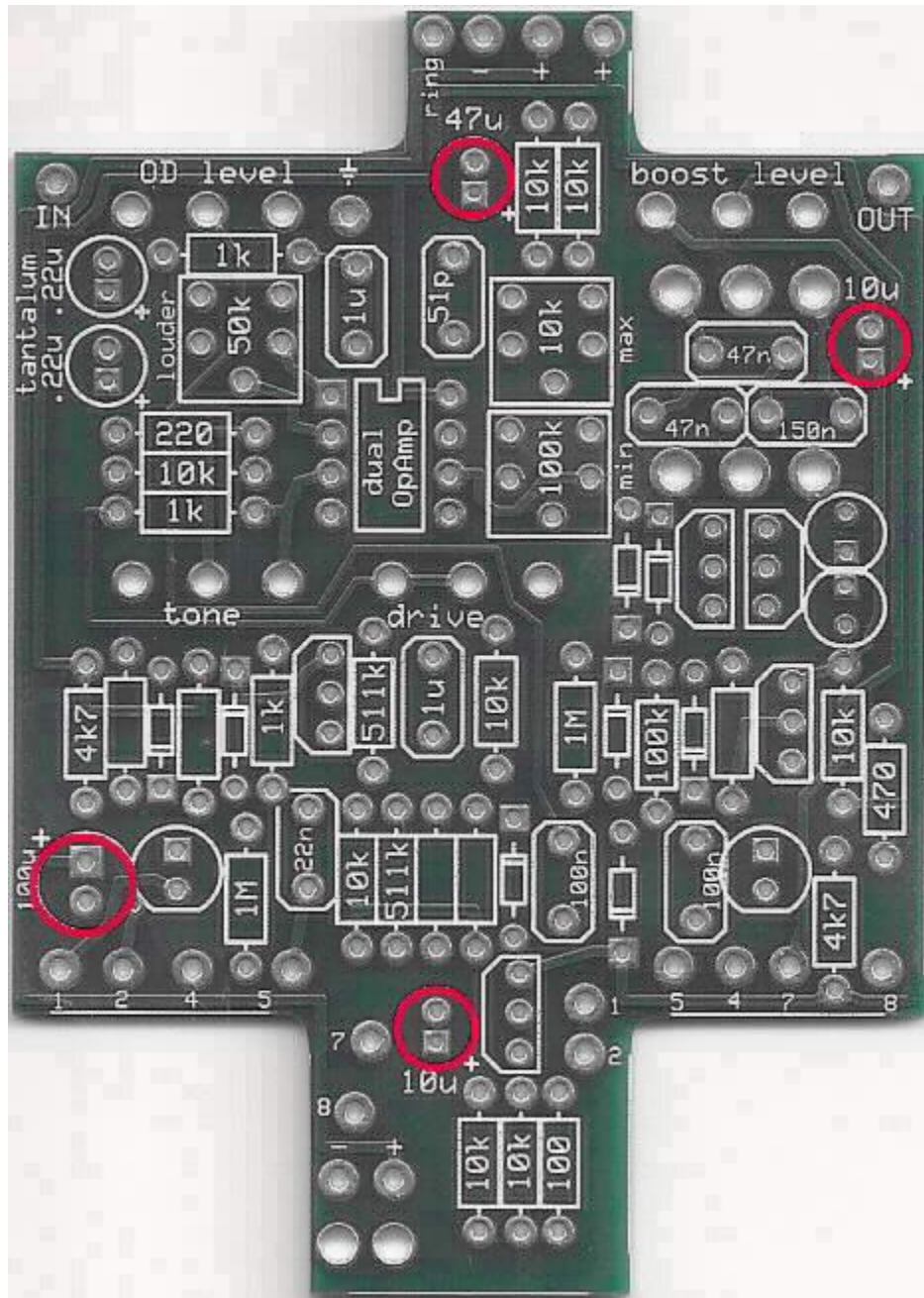
Step 7: Add the ceramic disc cap. These are not polarized and can go into the PCB in either direction.



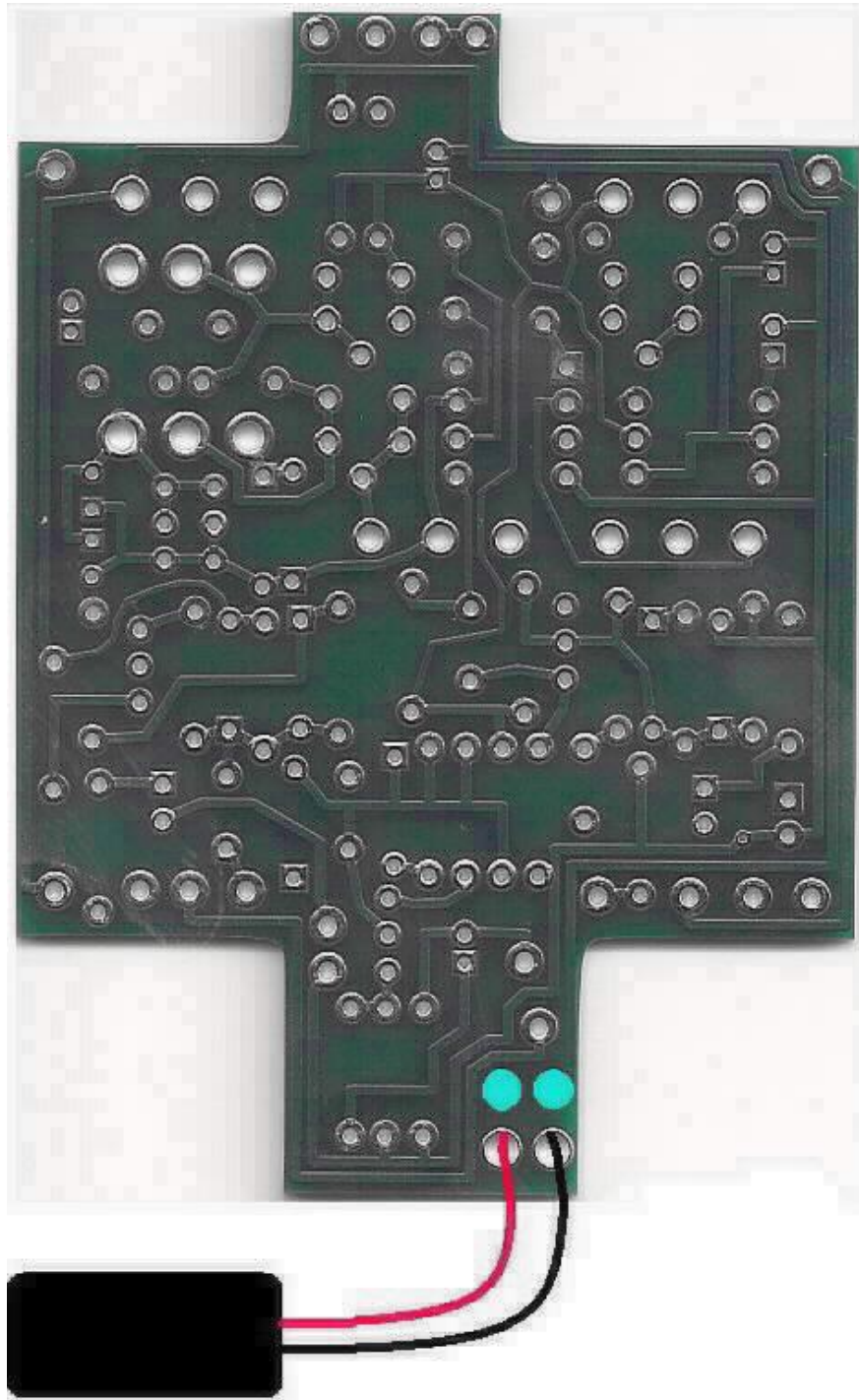
STEP 9:

Standard specs: Add the MPSA18 transistors to the spaces highlighted in RED. Be sure to match up the flat side of the transistor with the flat side on the layout. The spaces highlighted in yellow are for MOSFET clipping transistors. If you are using the clipping LEDs, do not use the MOSFET clipping transistors. You cannot use both at the same time.

MOSFET specs: Add the BS170 transistors to all the spaces highlighted in RED and YELLOW. **Be sure to put the BS170's in backwards from the PCB layout.** Do not add the clipping LEDs if you will be using clipping MOSFETS highlighted in yellow.



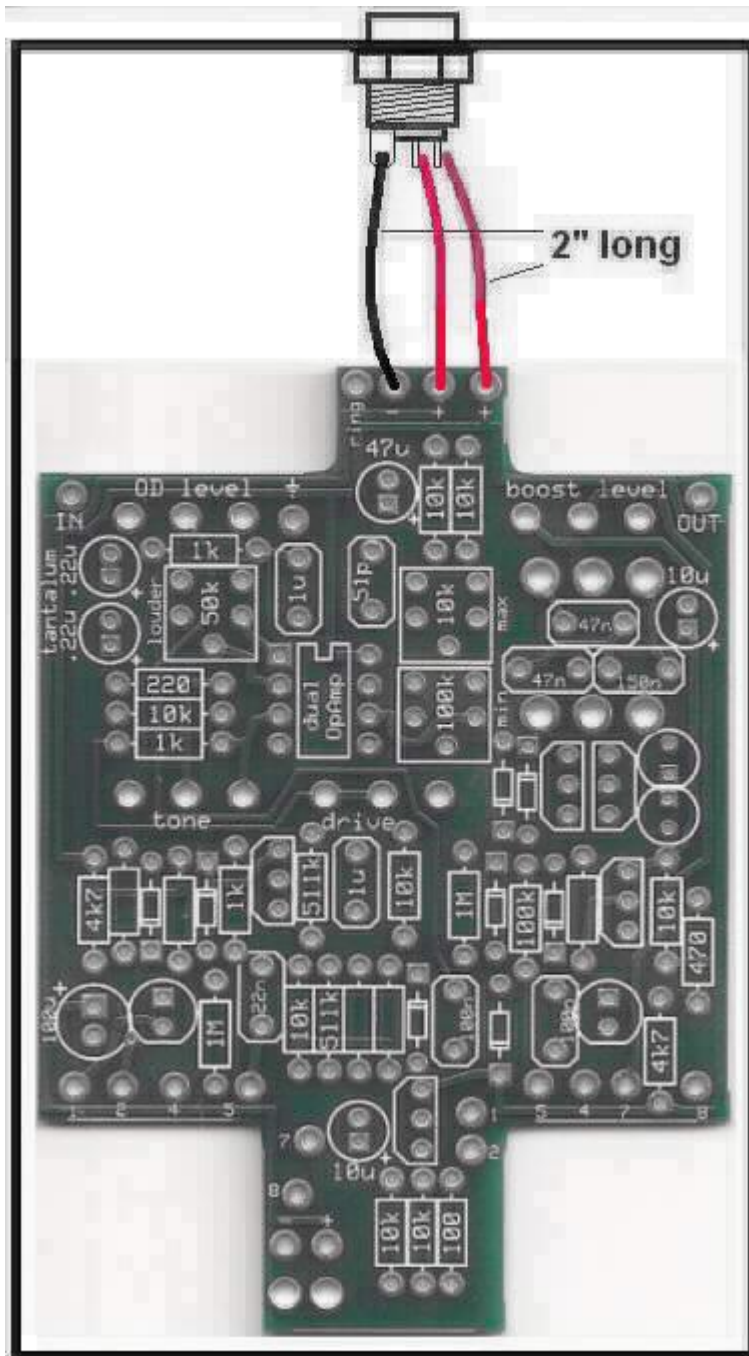
STEP 10: Add the aluminum electrolytic capacitors. These are polarized. The positive end will have a longer lead and should go in the square solder pad. The negative end will have a shorter lead with a black strip running down the body of the capacitor.



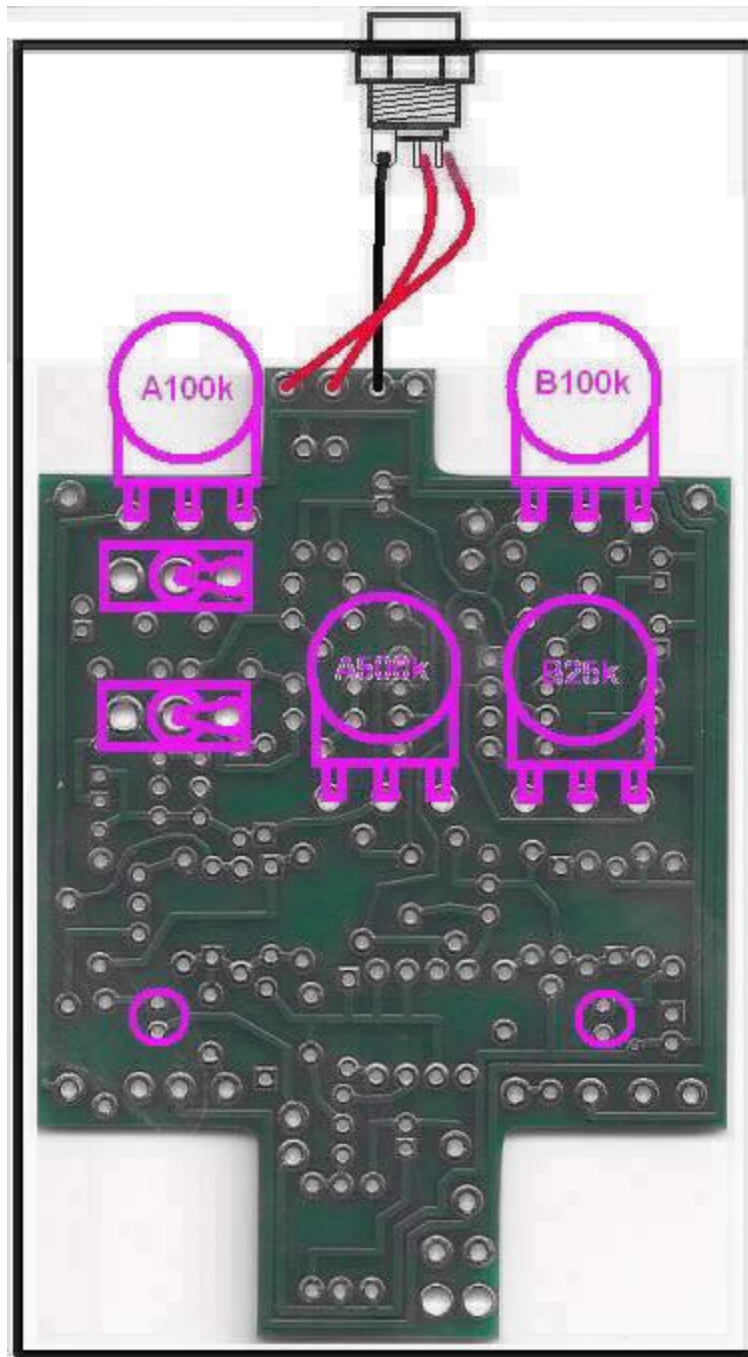
Step 11: Add the battery snap. Thread the solder ends of the snap through the under side of the strain relief holes. Insert the stripped ends of the battery snap wires into the topside of their respective solder pads. Solder on the bottom side where highlighted in blue. Remember the red wire goes in the “+” hole and the black wire goes in the “-” hole.

Assembly

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

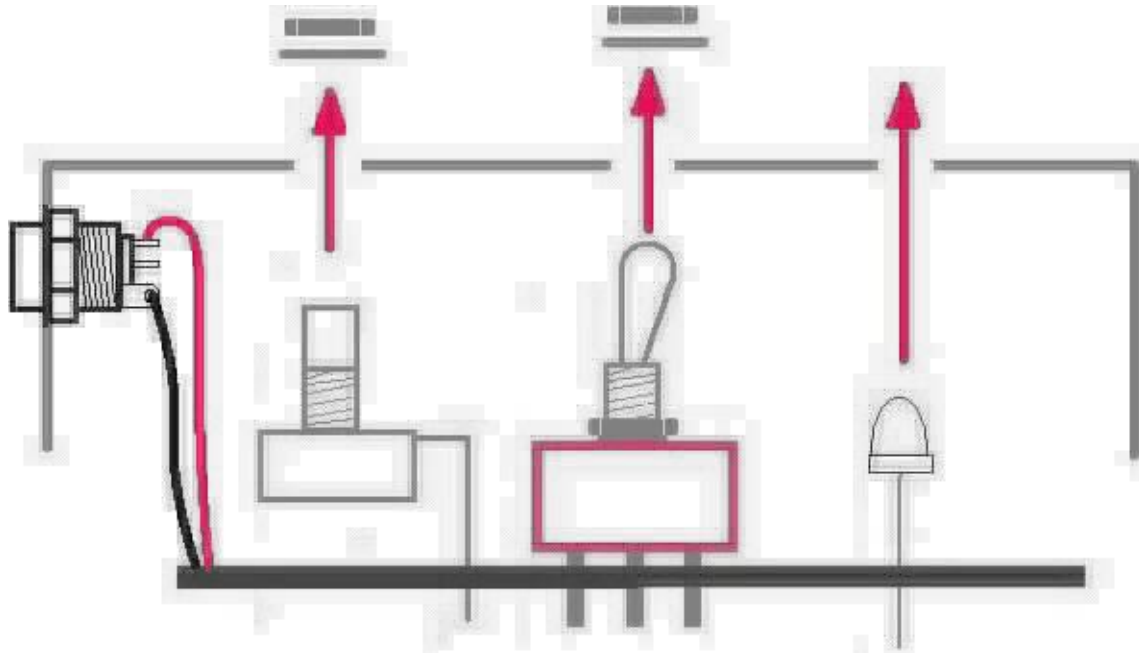


Step 2: Connect the TIP (negative) terminal of the DC adaptor jack to the “-” eyelet on the PCB with 2 inches of hook up wire. Connect the SLEEVE of the DC adaptor jack to the “+” eyelet on the far right side of the PCB with 2 inches of hook up wire. Connect the battery disconnect terminal of the DC 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, A100k, B100K, and A500k potentiometers, the LED(long leads go in the square solder pad holes), and the two toggle switch into the bottom side of the PCB. DO NOT SOLDER!!!There are only two ways in which the toggle switch can fit into the PCB. Either way is fine. Each LED will have one lead that is longer than the other. The longer lead goes in the hole of the square solder pad.

Remember to replace the 25k tone pot with the 10k if building to MOSFET Specs.

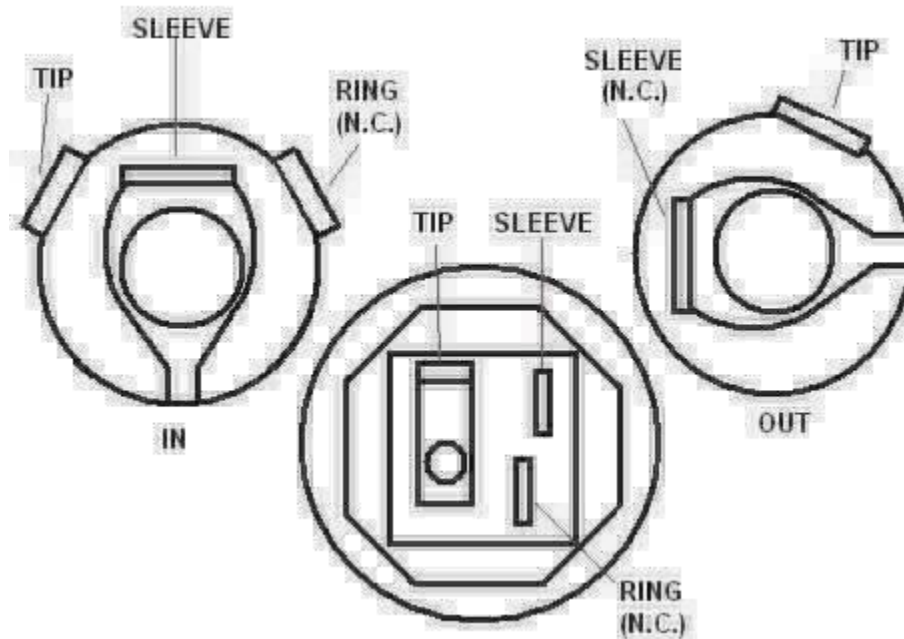


Step 4: Hold the PCB in one hand so that the component side of the PCB is in the palm of your hand and the bottom side with the pots, toggle switch and LED is facing up. Now use your other hand to guide the predrilled enclosure onto the PCB assembly so that the pots, toggle switch and LED all go into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

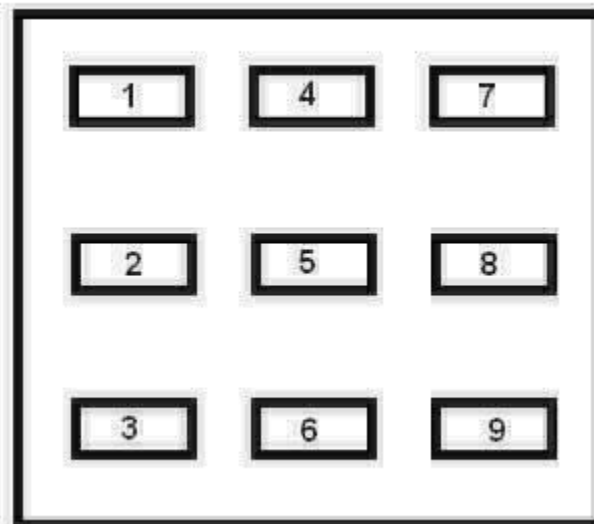
Step 6: Solder the pots, toggle switches, and LEDs. You will solder these parts on the component side of the PCB. After you have soldered them in place, be sure to tighten up their nuts.

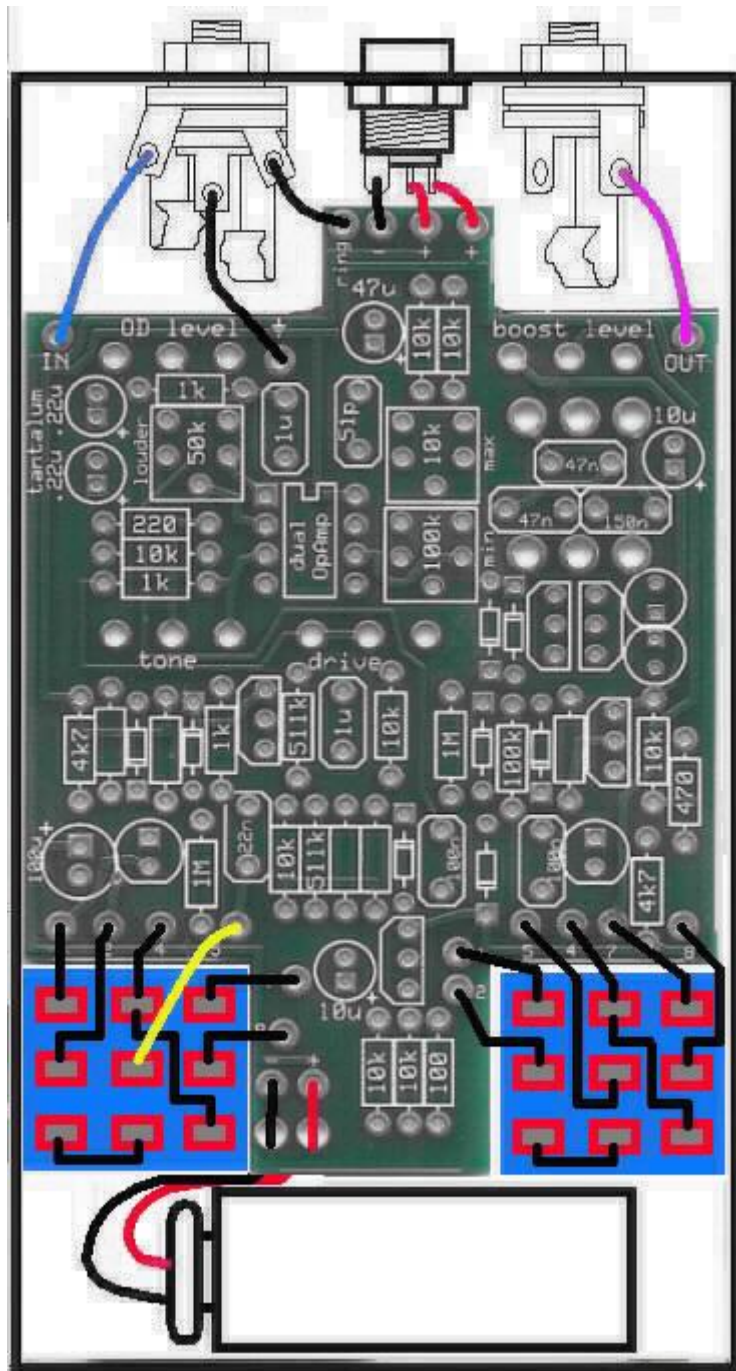
Wiring the footswitch, and jacks



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.

Footswitch Solder Lug Designations





Step 2:

- Cut 7 x 3/4" pieces of wire. Strip 1/8" off each end. These will be used to connect lugs/eyelets 1, 7, & 8 of the footswitch on the left and lugs/eyelets 1, 2, 7, & 8 of the footswitch on the right.
- Cut 4 x 1" pieces of wire. Strip 1/8" off each end. These will be used to connect lugs/eyelets 2 & 5 of the footswitch on the left and lugs/eyelets 5 & 8 of the footswitch on the right.
- Cut 2 x 1.5" peices of wire. Strip 1/8" of one end. Strip 1/2" off the other end.

These will be used to connect lugs/eyelets 4 with the longer 1/2" stripped ends being used to jumper lugs 4 to 9 of both footswitches.

- 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.

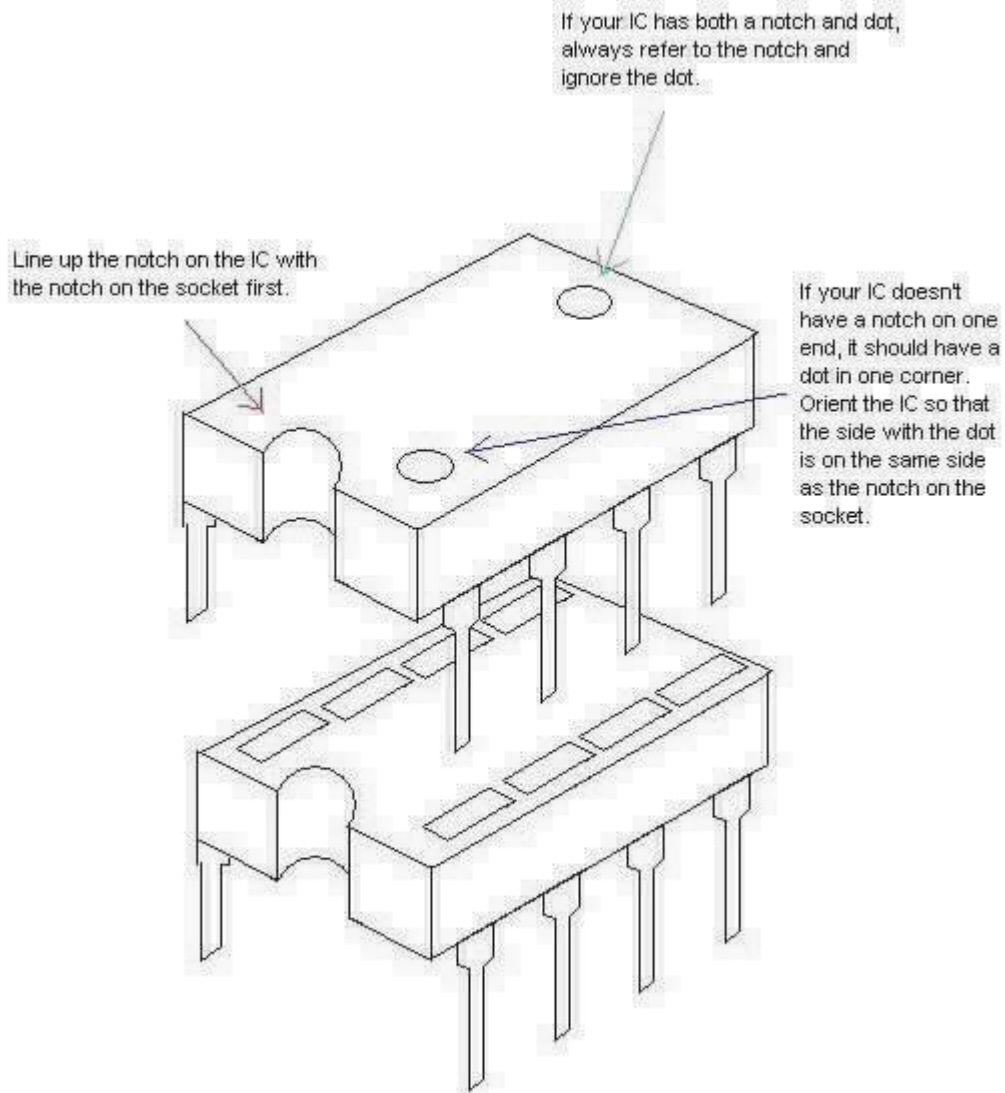
Step 3: Solder the wires for the footswitch.

Step 4: Remove the PCB assembly from the enclosure. Solder the open ends of the wires that you just soldered to the footswitch to their respective eyelets on the PCB. Load the wires in from the top and solder on the bottom side.

Step 5: Reinstall the PCB assembly to the enclosure. Install the footswitch to the enclosure. It will have a white nylon washer and a silver metal washer. You can decide which washer you want to be visible on the outside of the enclosure.

Step 6. Solder the jacks to the PCB.

Installing the IC



The image shows a green PCB for a guitar pedal. It features several sections labeled: "IN" at the top left, "OUT" at the top right, "boost level" near the output, "tone" and "drive" for control knobs, and "dual OpAmp" for the central IC. Other labels include "ring", "min", "max", and "louder". Pin headers are numbered 1-8 along the bottom and sides. Components include resistors (e.g., 10k, 100k, 1M, 47k), capacitors (e.g., 10uF, 1uF, 100nF, 47nF), and potentiometers. Two specific areas are highlighted with red rectangles: one containing a 50k resistor and a 1uF capacitor, and another containing two 10k resistors and a 100k resistor.

MIN trimmer: This trimmer will affect the amount of minimum distortion. Originally it was a 51k resistors, so setting this trimpot to exactly halfway will give you stock TS808 specs. Turning it up will give you more drive, turning it down will give you less drive.

MAX trimmer: This trimmer will affect the amount of maximum distortion. Originally it was a 4.7k resistor, so setting this trimpot to exactly halfway will give you stock TS808 specs. Turning it up will give you more drive, turning it down will give you less drive.

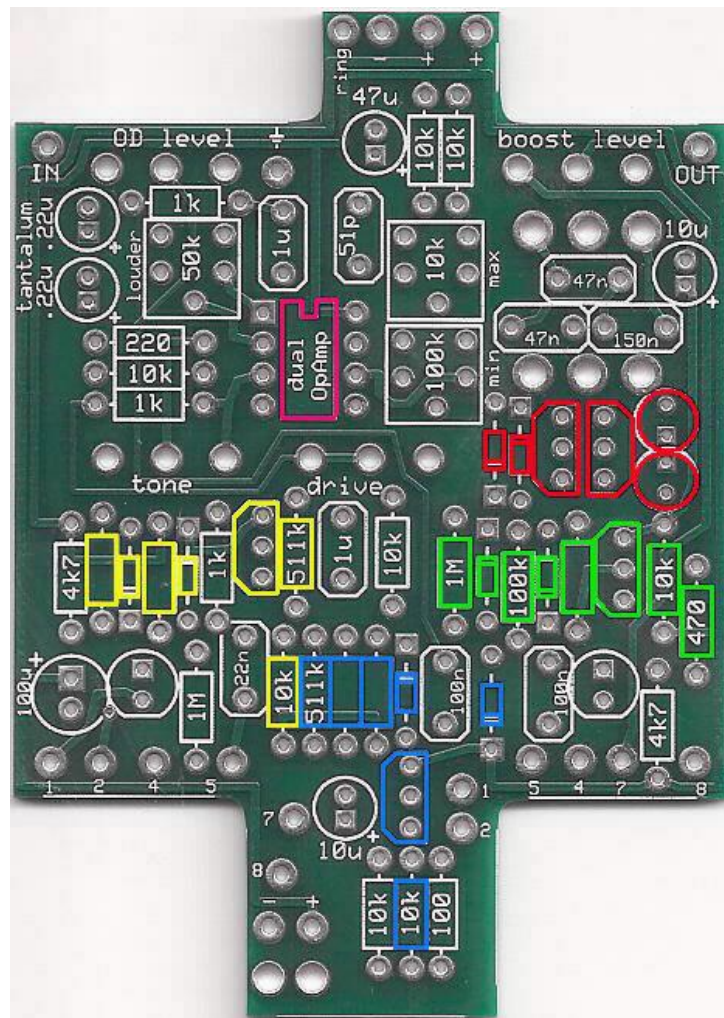
Tuning it up too much may cause squealing or self-oscillation.

LOUDER trimmer: This trimmer affects the gain of the second half of the op amp. Originally it was a 1k resistor, so setting this trimpot to just shy of full turn counter clockwise will give you stock TS808 specs. Turning it up will increase the amount of drive and overall output. Turning it up too much may cause squealing or self-oscillation on certain settings. Particularly if you turn the tone knob up all the way full turn clockwise. This mod does not work very well if you built your pedal to MOSFET specs. It will add a lot of noise, so if you built your kit to MOSFET specs, you should turn it down just about as low as it can go and still give you full range on the tone knob (turning it full turn counter clockwise will make the tone knob less useful).

More Modifications

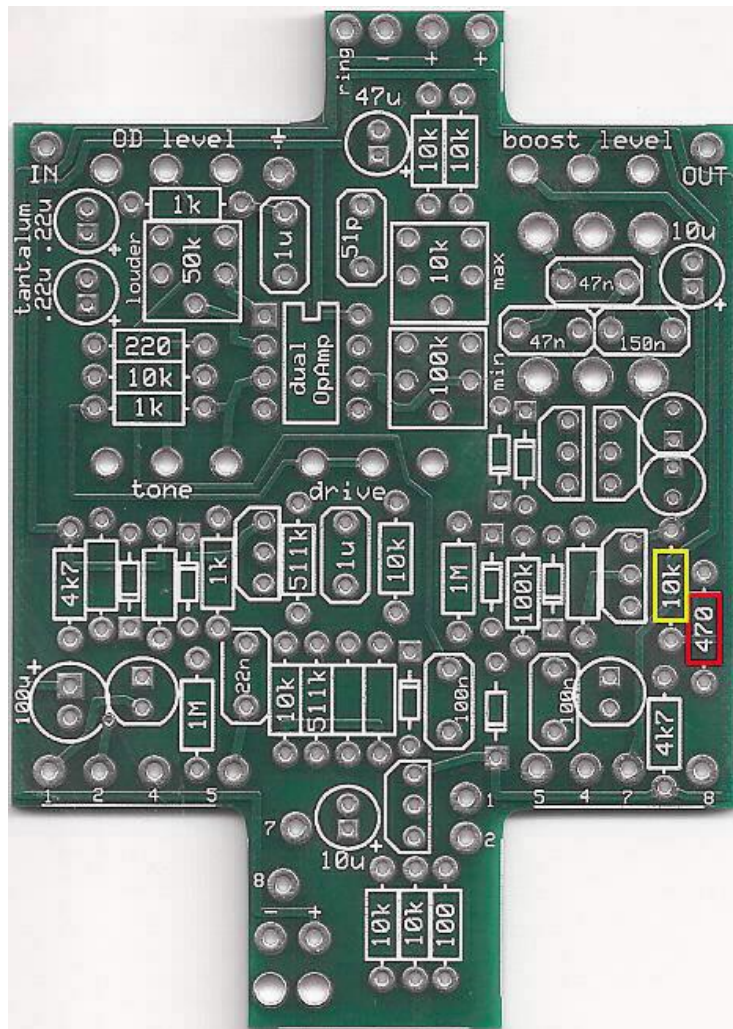
MOSFET or Standard Specs? Or a little of each?

By most peoples' accounts, MOSFETs produce a smoother, more tube-like solid state distortion. This is a good thing, right? But MOSFETs are noisier than the usual components used in the TS circuit. So there is a give and take. The neat thing about this build is that it is not “all or nothing” or “one or the other”. You can build some of the circuit to MOSFET specs and some of the circuit to standard specs. The parts of the circuit that can be built to either MOSFET or standard specs can be broken down into 5 different sections: the input buffer, the op amp, the clipping section, the output buffer, and the boost circuit. The various components specific to each section are mapped out in the diagram below.



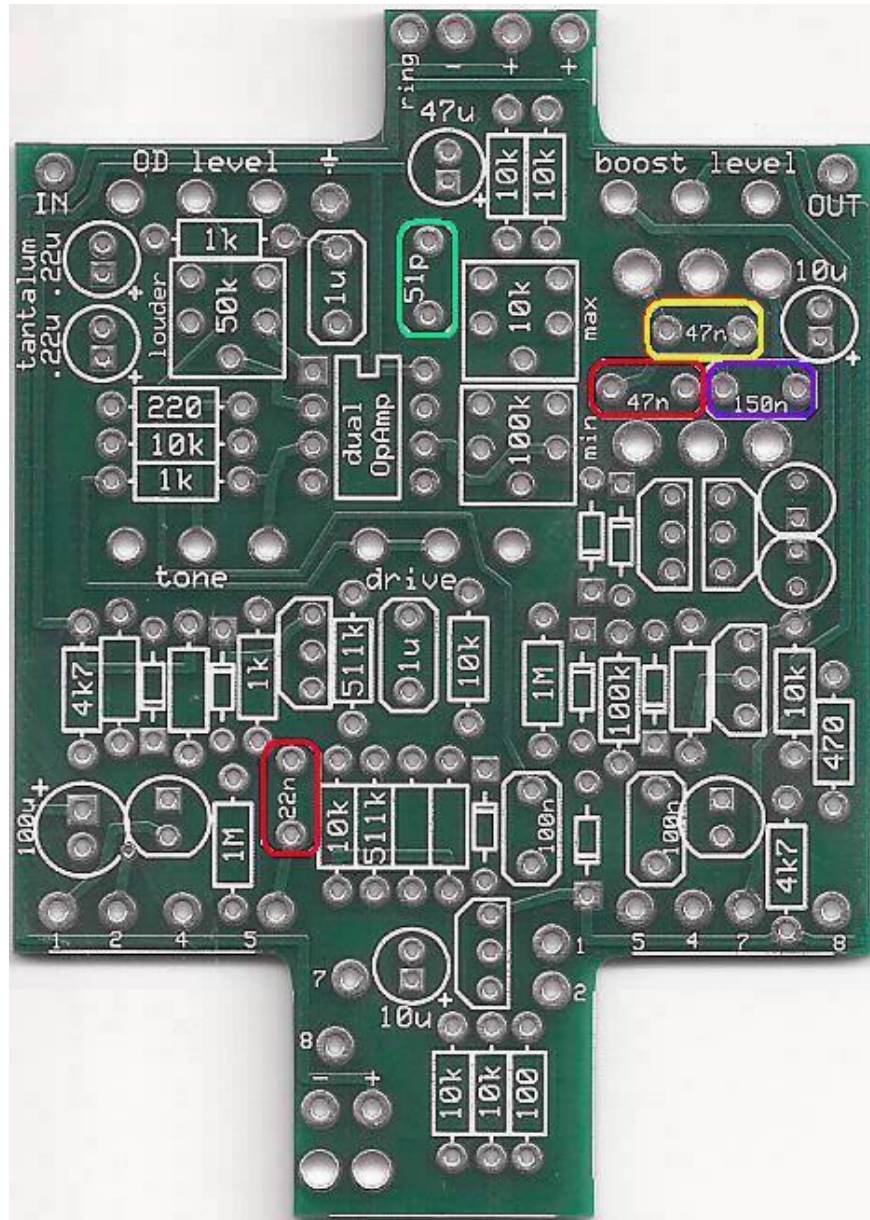
- Input Buffer highlighted in yellow. The buffers are where most of the mosfet noise come from
- Op Amp highlighted in pink. This is socketed, so it should be pretty easy to

- Output buffer highlighted in blue. The buffers are where most of the mosfet noise comes from.
- Clipping section highlighted in red. Note that while you can use either the MOSFET transistors as clipping devices or the LEDs as clipping devices regardless of what specs you build the rest of the pedal to, you cannot have components on the PCB in both of these spaces at the same time. It's one or the other in this case. If you wanted both, you'd need to get rid of the stock silicon diode clippers and put the LEDs in their place and then leave the spaces for the LEDs on the PCB empty.
- Boost Circuit highlighted in green. The standard silicon boost will sound fuller and will be louder but cannot run at more than 9V. The MOSFET boost will not be as loud as the silicon boost, but it will have some “sparkle” to it that many guitarists like and it can run at 18V.



Making the Boost louder - 1. You can squeeze more volume out of your boost by adjusting the 470ohm resistor highlighted in red. This could be considered the “gain” resistor of the boost circuit. By lowering the value of this resistor, the boost circuit will

2. You can also make the standard silicon boost louder and dirtier by increasing the value of the 10k resistor highlighted in yellow. Try a 15k resistor. This mod will not work with MOSFET build specs.

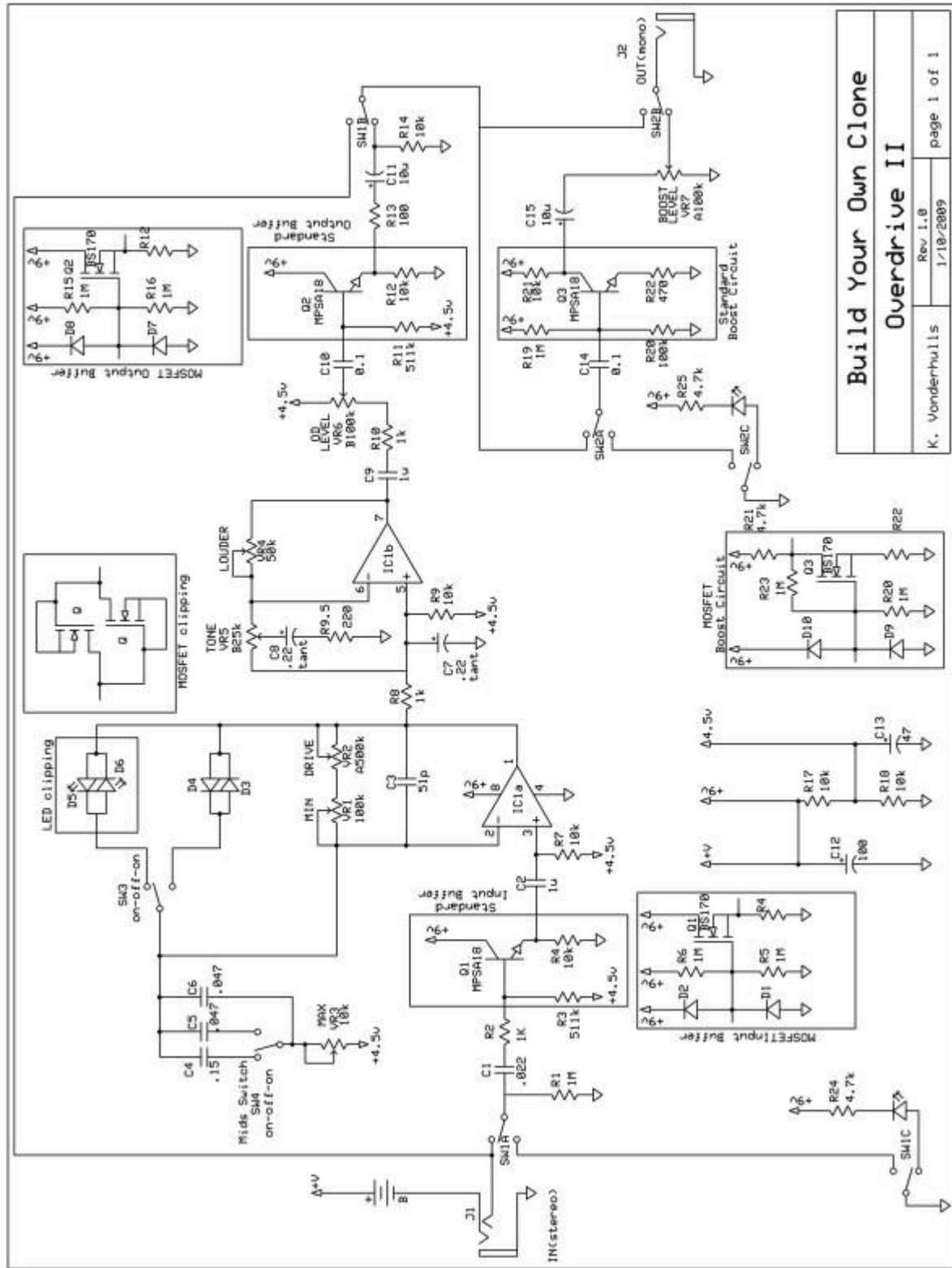


2. Make the “mids” caps larger for fuller bottom end. The 47n highlighted in red would be your center position (position2 on the toggle switch) cap. Try a 100n or .1uf cap here. The 47n cap highlighted in yellow gets put in parallel with the other 47n cap when the toggle switch is to the left(position 1). Try a 100n or .1uf cap here. And the 150n cap

highlighted in purple gets put into parallel with the cap highlighted in red with the toggle switch is to the right (position 3). Try a 220n or .22uf cap here.

3. Make the 51pf feedback cap larger. Try a 220pf or 330pf.

18 volt operation? - This can be a little tricky. Because the standard spec overdrive circuit can run just fine at 18v. But the standard silicon boost circuit cannot. 18v will not hurt it, but it will be distorted. The MOSFET boost can run at 18v. But the MOSFET overdrive circuit cannot. Anymore than 9v will fry the MOSFET op amp. So if you want to run the pedal at 18v, you'd need to build the overdrive portion of the circuit to standard specs and build the boost to MOSFET specs.



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

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