

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.byoelectronics.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. (more than, "It doesn't work, help")
2. Pic of the topside 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 answered yes to 6 and 7, what does the pedal do when it is in the "on" position?
9. Battery or adapter (if battery, is it good? If adapter, what type?)

Also, please only post photos that are in focus.

Copyrights:

All material in this document is copyrighted 2016 by BYOC, Inc.

Li'l Black Key Kit Instruction Index

Parts Checklist.....	page 4
Populating the Circuit Board.....	page 7
Enclosure Assembly.....	page 12
Wiring.....	page 17
Operation Overview.....	page 21
Schematic.....	page 22
PCB Back Trace Photo.....	page 23

Parts Checklist for the Li'l Black Key Kit

Resistors:

- 1 - 2k2/222 (Red/Red/Black/Brown/Brown) or (Red/Red/Red/Gold)
- 1 - 4k7/472 (Yellow/Purple/Black/Brown/Brown) or (Yellow/Purple/Red/Gold)
- 1 - 10k/103 (Brown/Black/Black/Red/Brown) or (Brown/Black/Orange/Gold)
- 1 - 470k/473 (Yellow/Purple/Black/Orange/Brown) or (Yellow/Purple/Yellow/Gold)
- 3 - 1M/105 (Brown/Black/Black/Yellow/Brown) or (Brown/Black/Green/Gold)
- 2 - 2M2/225 (Red/Red/Black/Yellow/Brown) or (Red/Red/Green/Gold)

Visit www.byoelectronics.com/resistorcodes.pdf for more information on how to differentiate resistors.

Capacitors:

- 1 - 220pf ceramic disc (may have "221" printed on the body)
- 1 - 100n/ .1uF film cap (may have "104" printed on the body)
- 1 - 220n/ .22uF film cap (may have "224" printed on the body)
- 3 - 2.2uF Aluminum Electrolytic

Visit www.byoelectronics.com/capcodes.pdf for more info on how to differentiate capacitors.

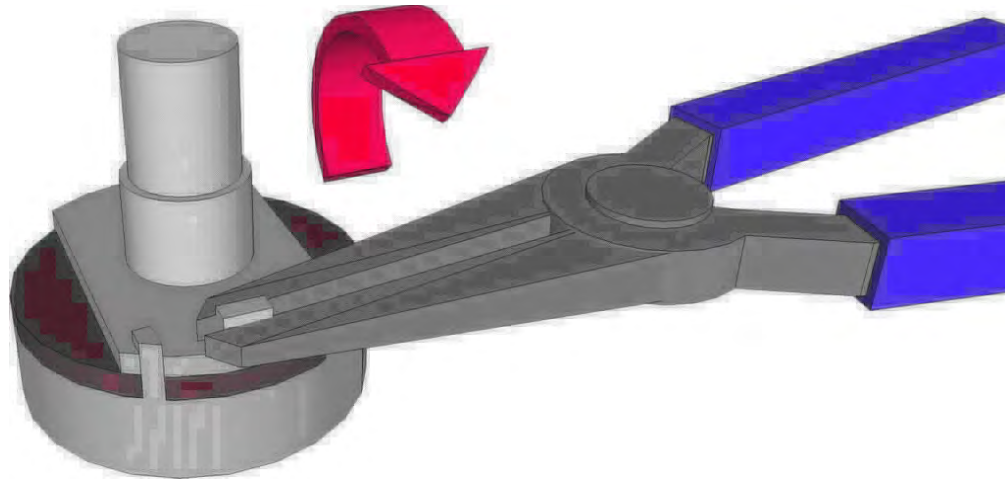
Diodes:

- 4 - 1N4148 or 1N914

ICs:

- 1 - 4558

Potentiometers: SNAP THE SMALL TABS ON THE TOP OF THE POTS OFF WITH A PAIR OF NEEDLE NOSE PLIERS



2 - A100k (VOL, DRIVE)

Hardware:

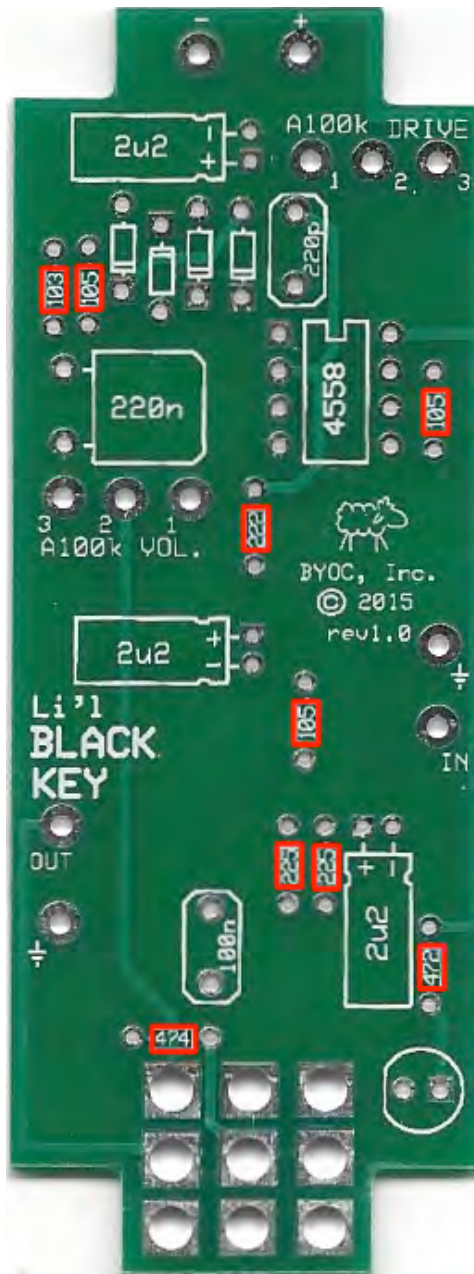
- 1 - predrilled enclosure w/ 4 screws
- 1 - Li'l Black Key circuit board
- 1 - 3pdt footswitch
- 1 - External Nut AC Jack
- 2 - Enclosed Audio Jacks
- 4 - rubber bumpers
- 2 - lock washers (for in and out jacks)
- hook-up wire



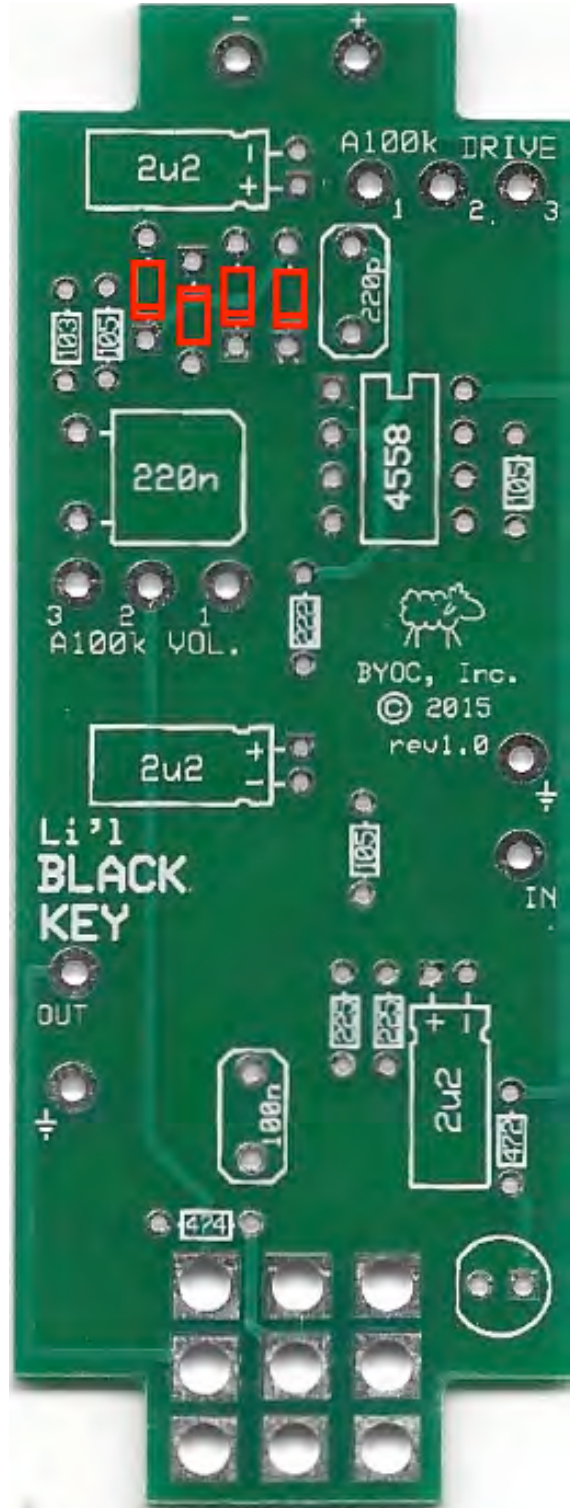
Your build should look similar to this when you're finished.

Populating the Circuit Board

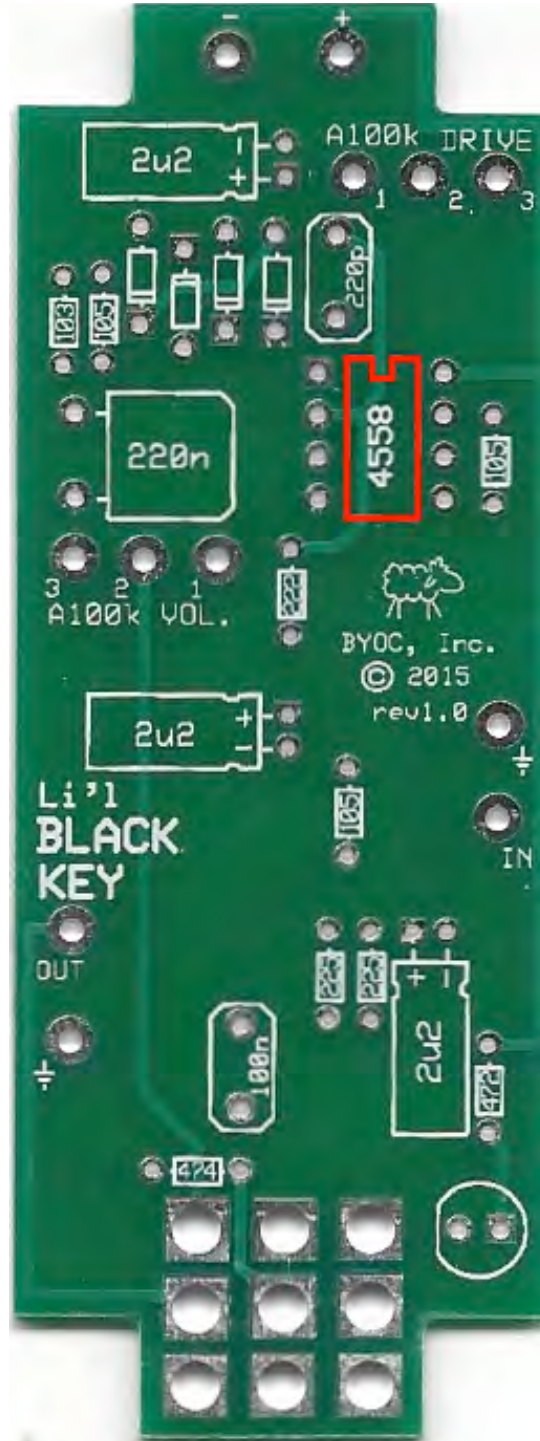
Step 1: Add all the resistors. Resistors are not polarized and can be inserted in either direction.



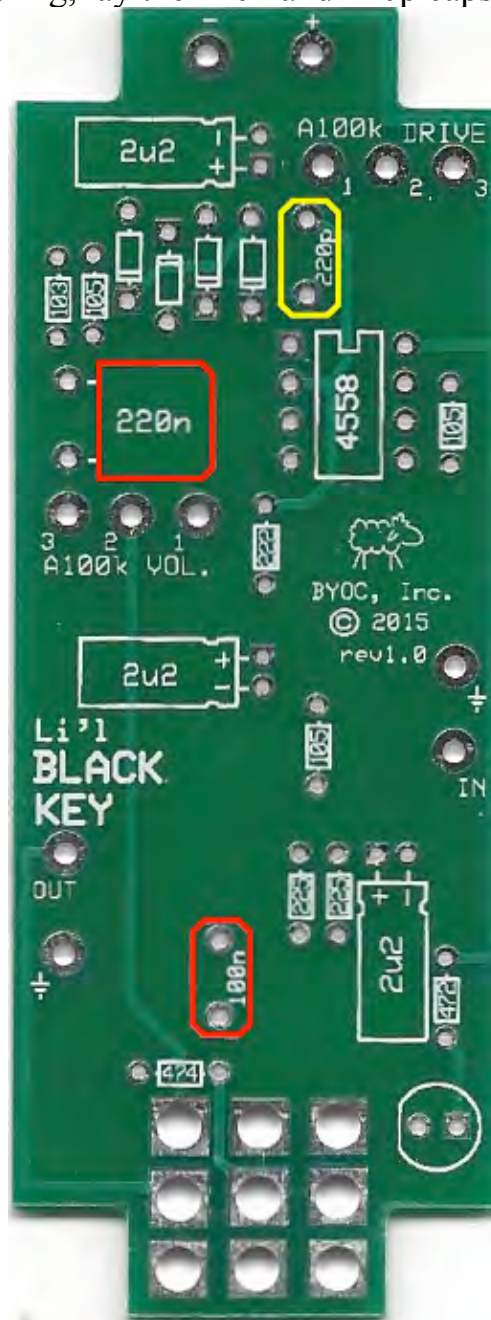
Step 2: Add the diodes. Be sure to match the end of the diodes with the stripe to the layout on the PCB. The striped end should go in the square solder pad.



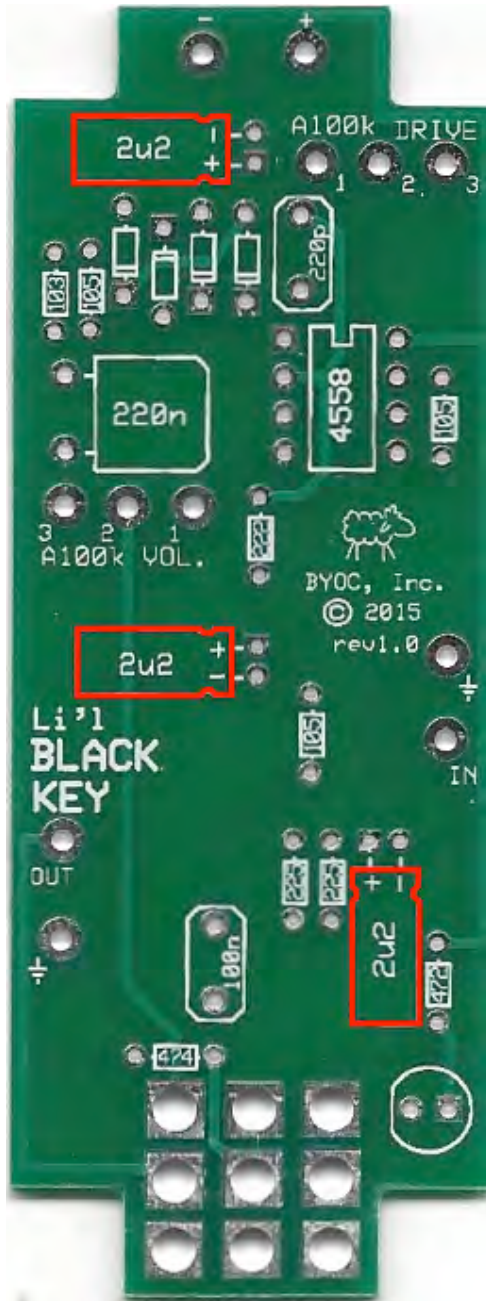
Step 3: Add the IC. Be sure to orient it correctly. There may be a notch on the IC between pins 1 and 8. Line that up with the screenprint on the PCB. If your IC does not have a notch, look for a dot to indicate pin 1. Pin one on the PCB will be a square hole.



Step 4: Add the film and ceramic disc capacitors. These are non-polarized and can go in either direction. The ceramic disc capacitor is highlighted in yellow. Before soldering, lay the 220n and 220p caps flat against the PCB.

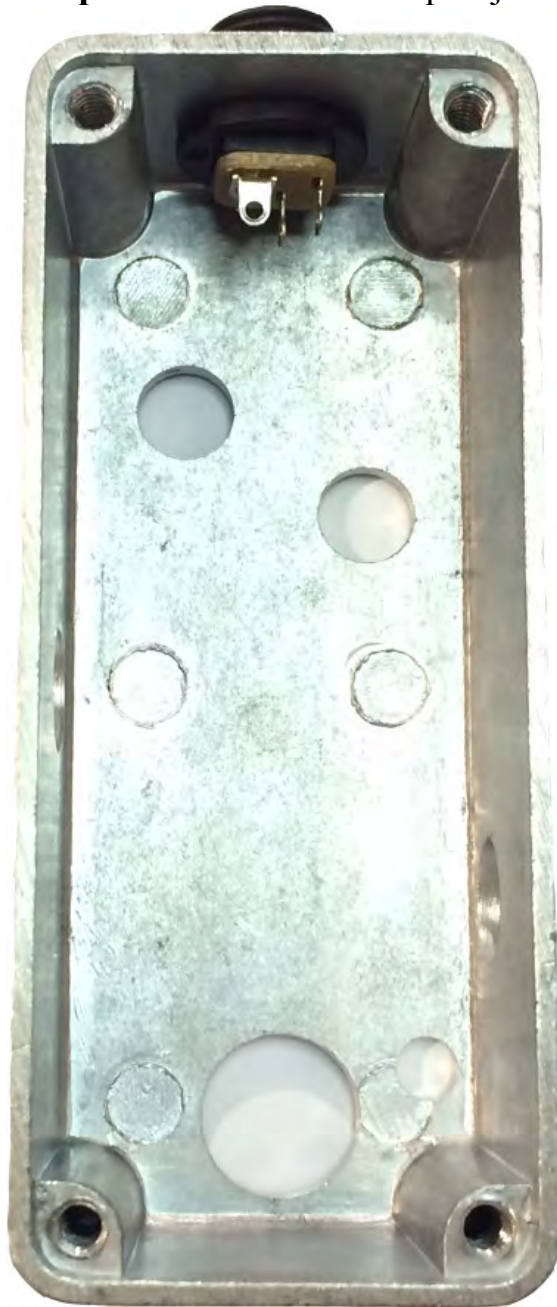


Step 5: Add the aluminum electrolytic capacitors. These ARE polarized, meaning there is a positive and negative end. The positive side will have a longer lead and goes in the square solder pad. The negative side will have a shorter lead and a stripe running along the body of the cap, and goes in the round solder pad. Before soldering, bend the capacitor so it is laying down like below

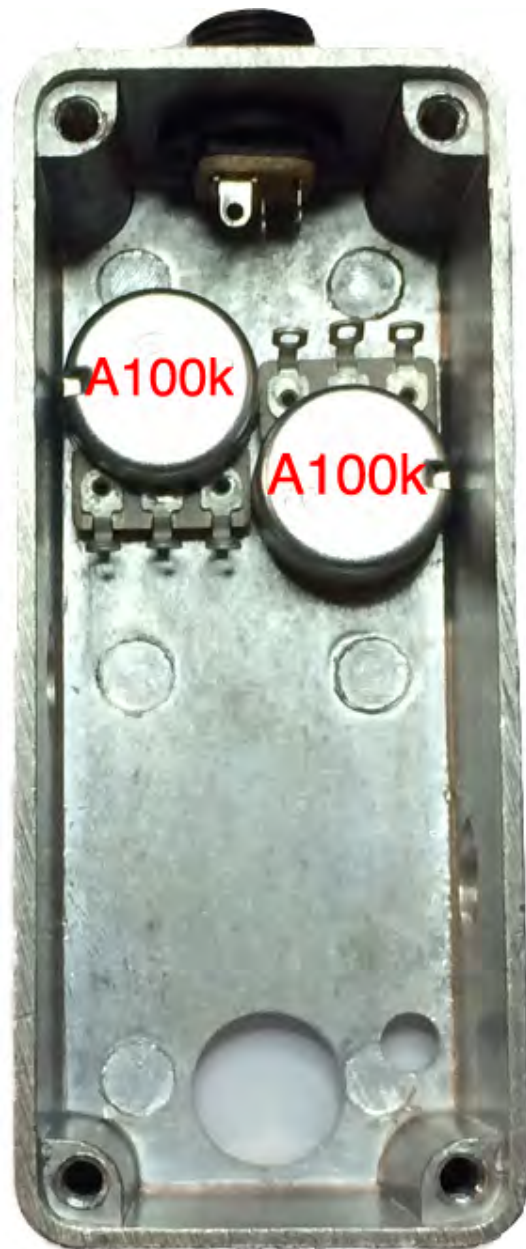


Enclosure Hardware Assembly

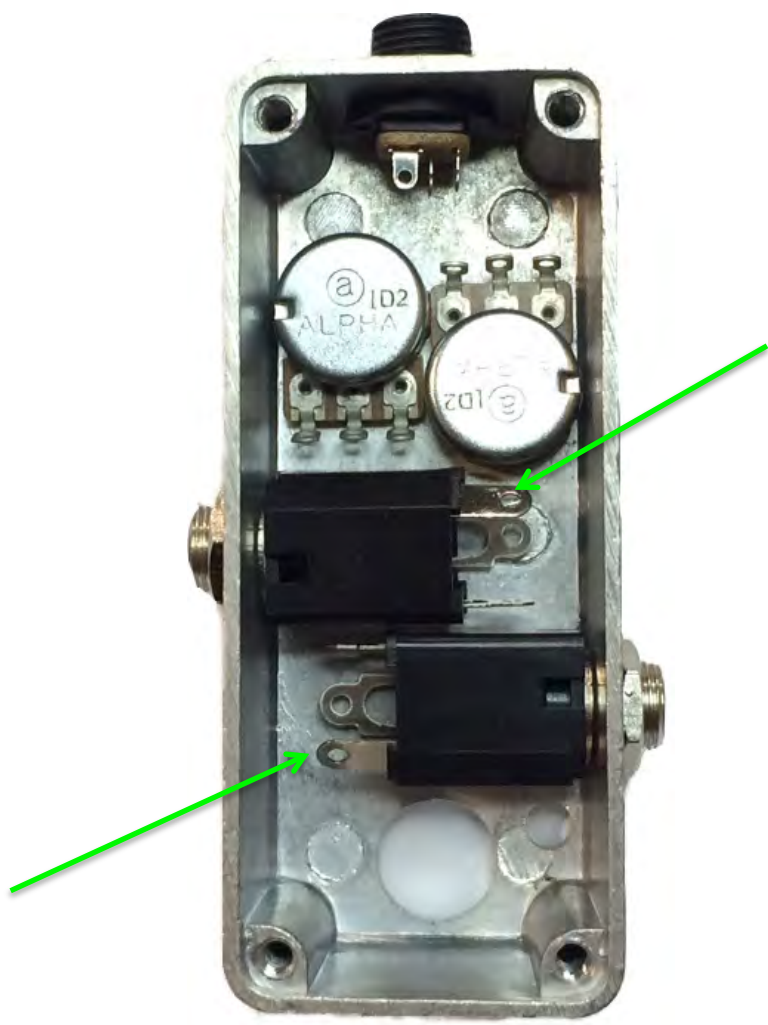
Step 1: Mount the DC adapter jack.



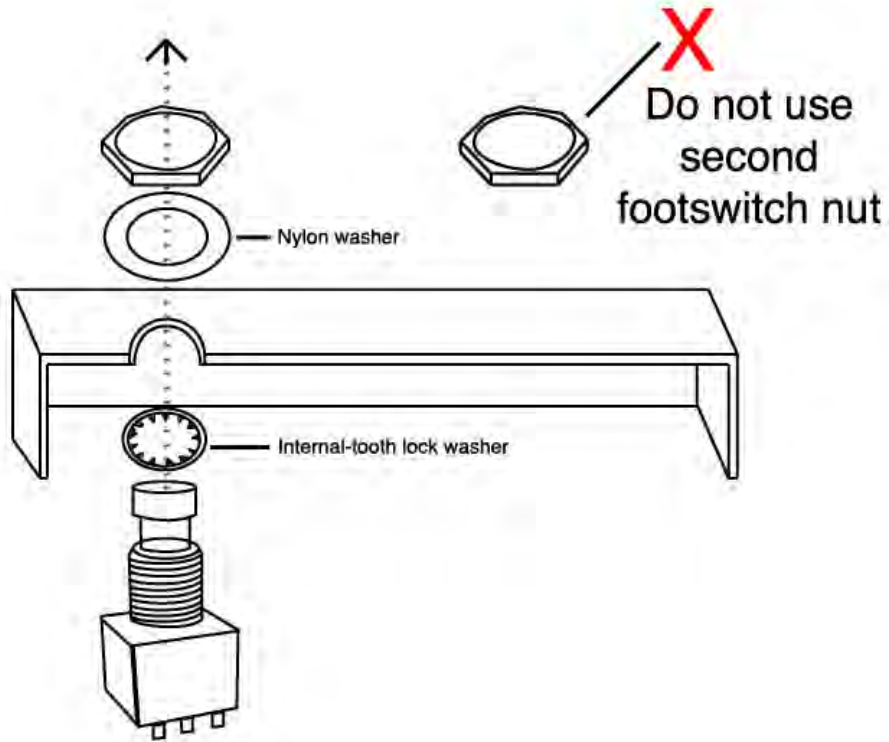
Step 2: Mount the potentiometer. You will mount it with the solder lugs facing towards the DC Jack.



Step 3: Mount the Audio Jacks. You want to orient them so that the SLEEVE terminals (beveled corner of the jack) are facing away from each other. If looking at the inside of the enclosure, the INPUT jack will have its sleeve terminal facing towards the AC jack. The OUTPUT jack will have its sleeve terminal facing towards the footswitch hole. The green arrows are pointing at the sleeves.



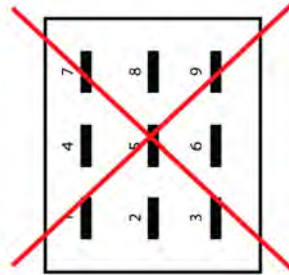
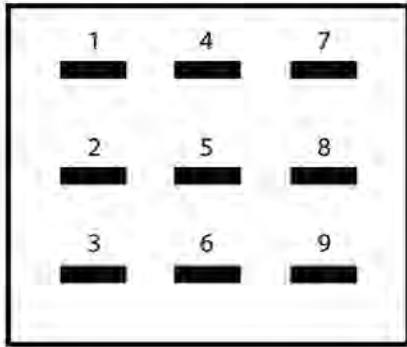
Step 4: Remove both nuts from the footswitch and mount the footswitch.



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.

FOOT SWITCH SOLDER LUG DESIGNATIONS

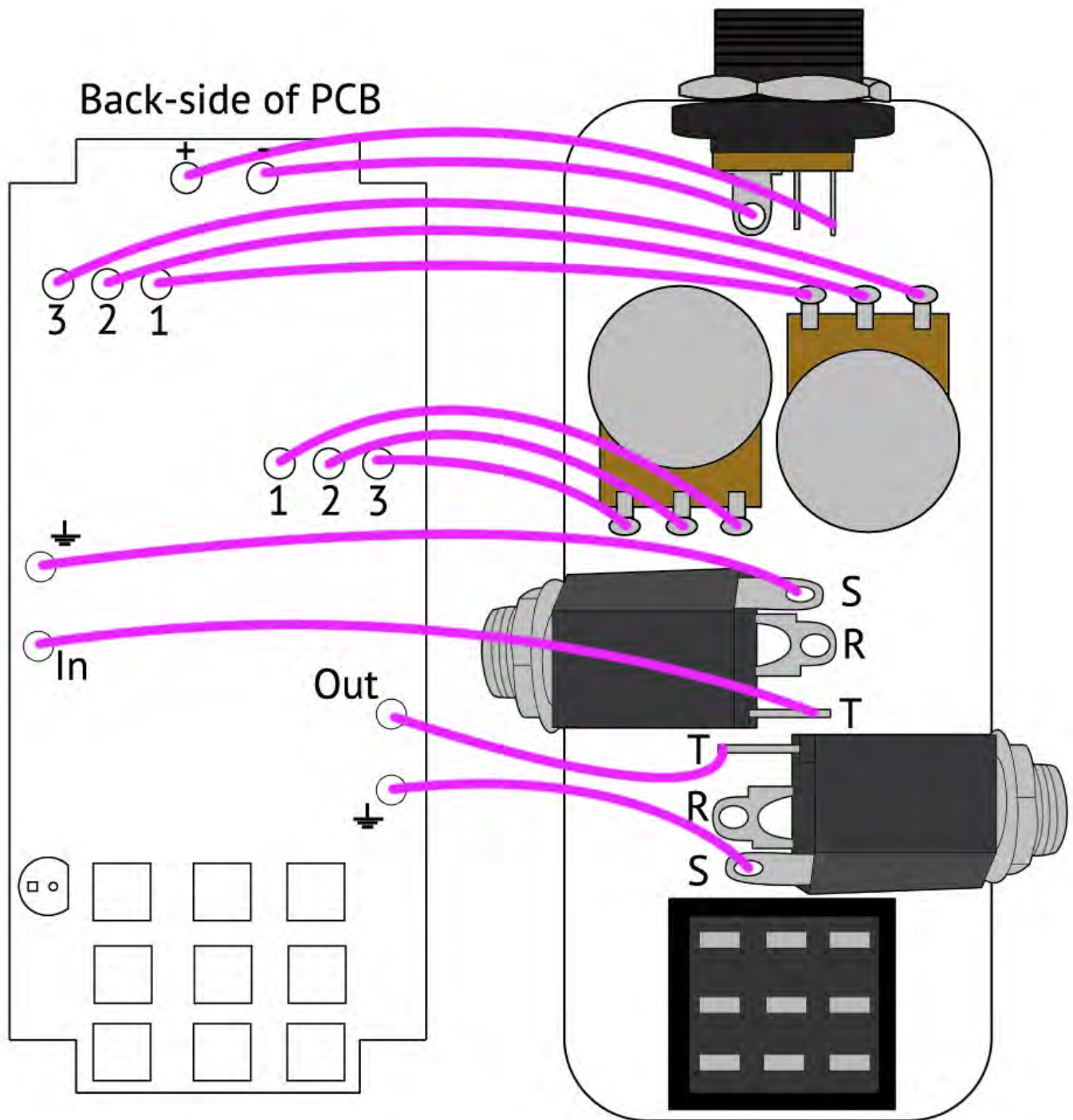


Wiring



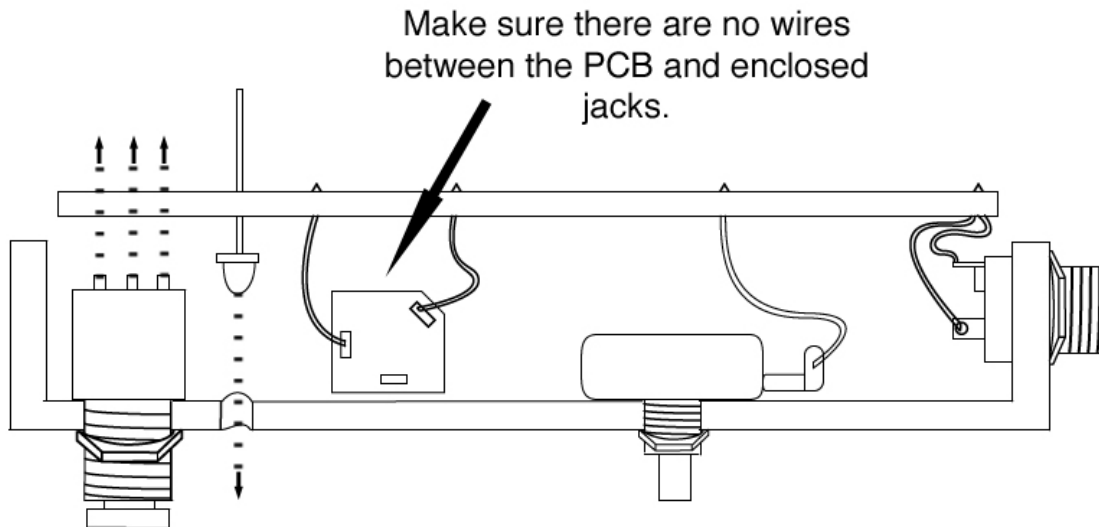
FLIP PCB OVER!!!

STEP 1: Wire the PCB as shown in the diagram below. Make all connections to the back side of the PCB and solder on the top (screen printed) side of the PCB. Make the wires as short as possible but allow enough length so that if you need to do any trouble shooting later, you will be able to do so without having to remove all the enclosure mounted components.



Step 2: Once the AC jack, potentiometer, and IN/OUT jacks are wired, insert the LED in its hole. **Insert the long lead into the square hole!!!** You might want to slightly bend the leads away from each other to keep them in the holes for now. DO NOT SOLDER YET

Step 3: Flip the PCB right-side-up, tuck the wires into the enclosure, and place the PCB onto the footswitch. **DO NOT SOLDER YET!!!!**



It is extremely important that when you place the PCB on the footswitch, you make sure to tuck all the wires out of the way so that that PCB can rest flush against the enclosed jacks and footswitch.

Step 4: Solder ONLY 1 of the footswitch lugs.

Your PCB should be recessed inside the enclosure about 1cm. It should be level.

Some of your components may be sticking up out of the enclosure just a bit, particularly the film capacitors. This is to be expected. This is OK. The lid has a deep recess.

If you need to adjust the position of the PCB so that it fits correctly, reheat the single solder joint you just made on the footswitch. Adjust the position of the PCB while the keeping the solder joint hot. Remove the heat and hold the PCB in position while allowing the solder joint to cool completely. Now solder the rest of the footswitch lugs.

Step 5: Once you have your PCB positioned and soldered, guide the LED into its enclosure hole using the leads that are sticking up through the PCB. Once the LED is in position, solder and trim the excess leads.

Operating Overview



VOL: Controls overall output volume.

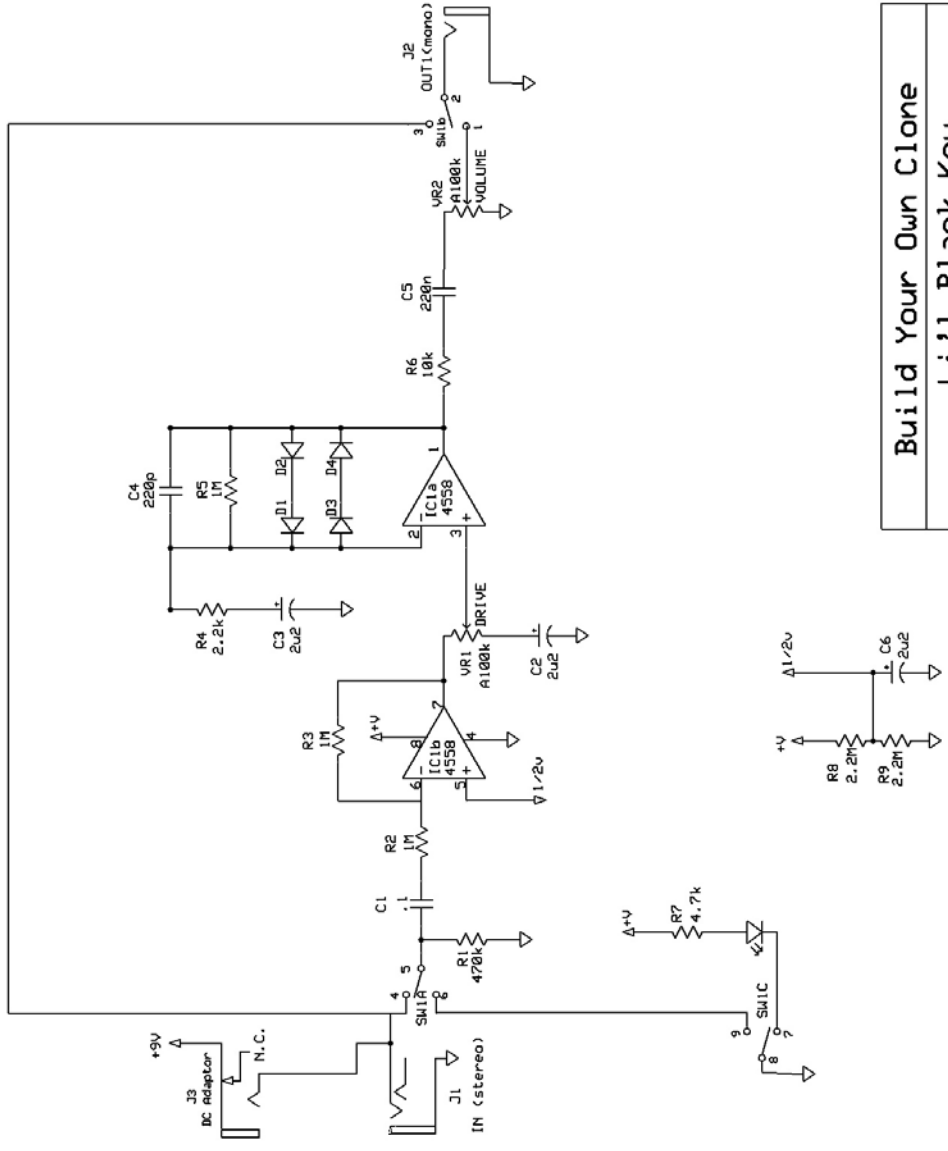
DRIVE: Controls the amount of distortion.

Power supply: 2.1mm negative tip.

Current Draw: 5.5mA

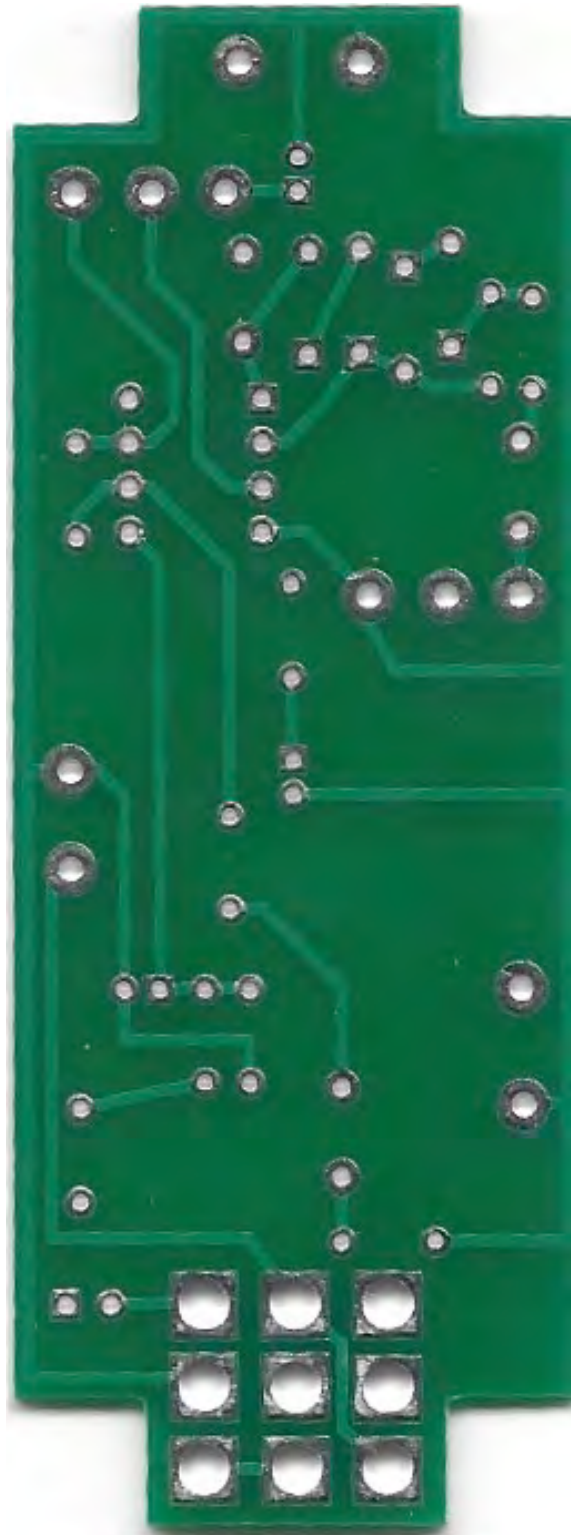
Input Impedance: 470k ohms

Output Impedance: 100k ohms



Build Your Own Clone	
Li'l Black Key	
Designed by:	Rev 1.0
N.H. Kenning	12/30/2015
	HTCH ₁ UB.808? RSH

PCB Back Trace Photo



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

Copyright 2016
BYOC, Inc.