Build Your Own Clone
Mimosa
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. 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 manor. 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:

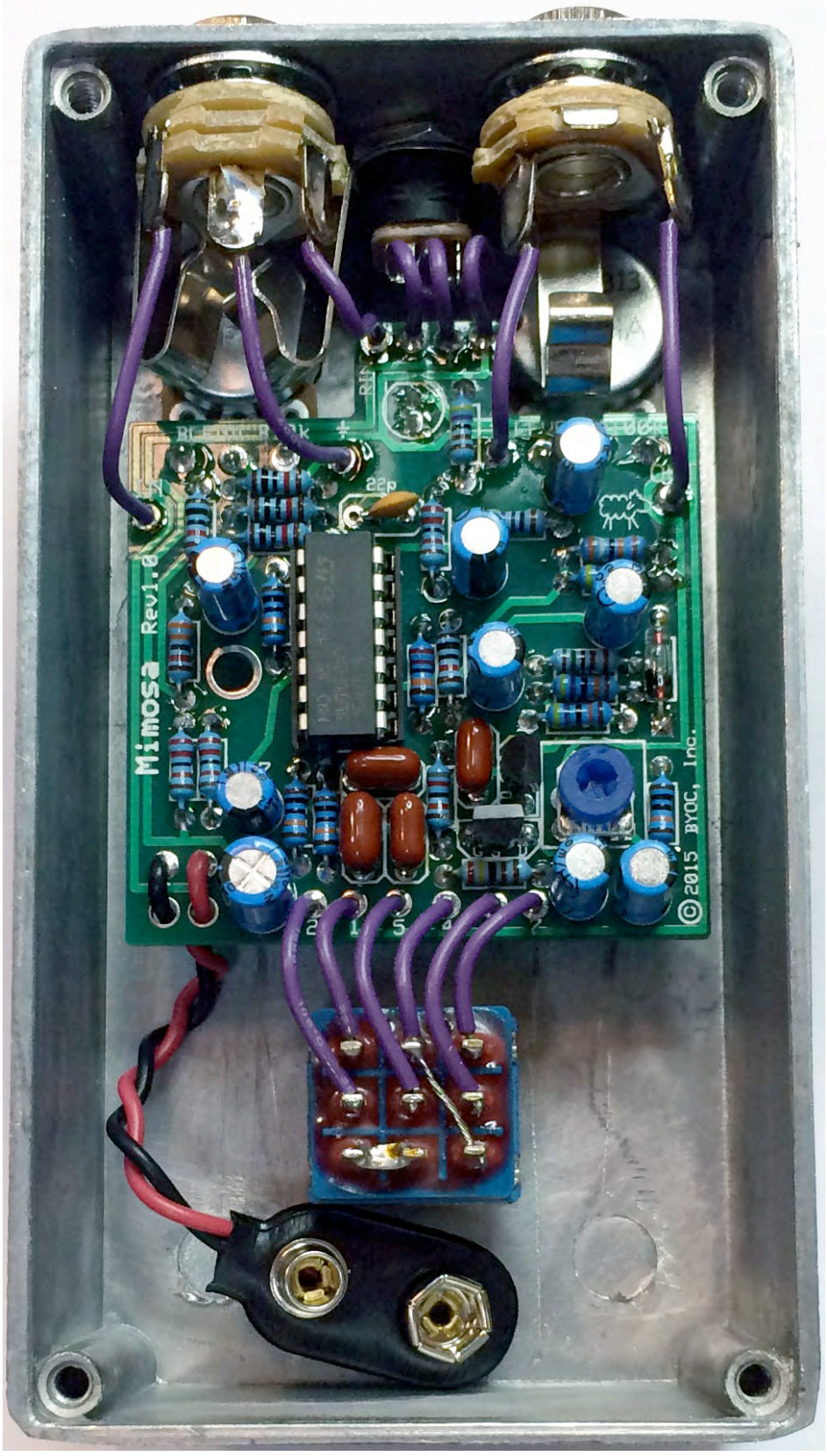
1. A detailed explanation of what the problem is. (more than, "It doesn't work, help")
2. Pic of the topside of your PCB.
3. Pic of the underside of your PCB.
4. Pic that clearly shows your footswitch/jack wiring and the wires going to the PCB
5. A pic that clearly shows your wiring going from the PCB 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:

Mimosa
Kit Instruction Index

This is what your kit should look like when it's complete. Your kit may come with different looking components, don't be alarmed by this.



Parts Checklist for the BYOC Mimosa Kit

Resistors:	**Metal Film (5-band)**	/	**(Carbon (4-band)**
1 - 470R	(Yellow/Purple/Black/Black/Brown)		(Yellow/Purple/Brown/Gold)
1 - 1k5	(Brown/Green/Black/Brown/Brown)		(Brown/Green/Red/Gold)
1 - 2k4	(Red/Yellow/Black/Brown/Brown)		(Red/Yellow/Red/Gold)
1 - 4k7	(yellow/purple/black/brown/brown)		(Yellow/Purple/Red/Gold)
5 - 10k	(brown/black/black/red/brown)		(Brown/Black/Orange/Gold)
3 - 22k	(Red/Red/black/red/brown)		(Red/Red/Orange/Gold)
1 - 33k	(Orange/Orange/Black/Red/Brown)		(Orange/Orange/Orange/Gold)
1 - 82k	(Gray/Red/black/red/brown)		(Gray/Red/Orange/Gold)
4 - 100k	(brown/Black/black/orange/brown)		(Brown/Black/Yellow/Gold)
1 - 220k	(Red/Red/black/orange/brown)		(Red/Red/Yellow/Gold)
1 - 390k	(Orange/White/Black/Orange/Brown)		(Orange/White/Yellow/Gold)
3 - 470k	(Yellow/Purple/Black/Orange/Brown)		(Yellow/Purple/Yellow/Gold)

Capacitors:
1 - 22pf (22) or 20pF (20) ceramic disc (small round orange)
1 - 2n2 or .0022µ (222)
2 - 47n or .047µ film (473)
1 - 100n or .1µ film (104)
3 - 330n or .33u film (334) (*used for bass modification)
1 - 2u2 aluminum electrolytic
6 - 4u7 aluminum electrolytic
1 - 47µf aluminum electrolytic
1 - 100µf aluminum electrolytic

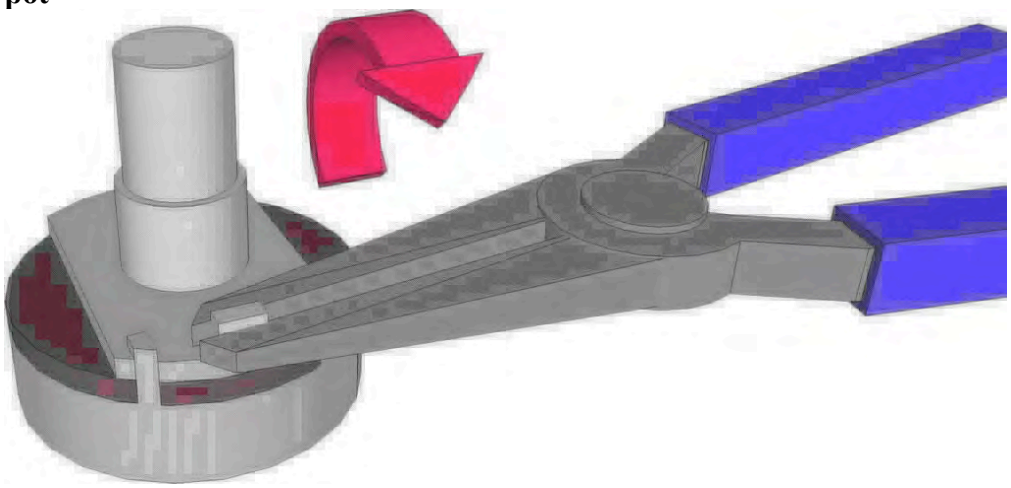
Diodes:

1 - Germanium Diode (1N34A, 1N60, 1n100, or similar)

Transistors:
2 - 2N5457

IC's:
1 - MC3403
1 - 14 pin socket

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

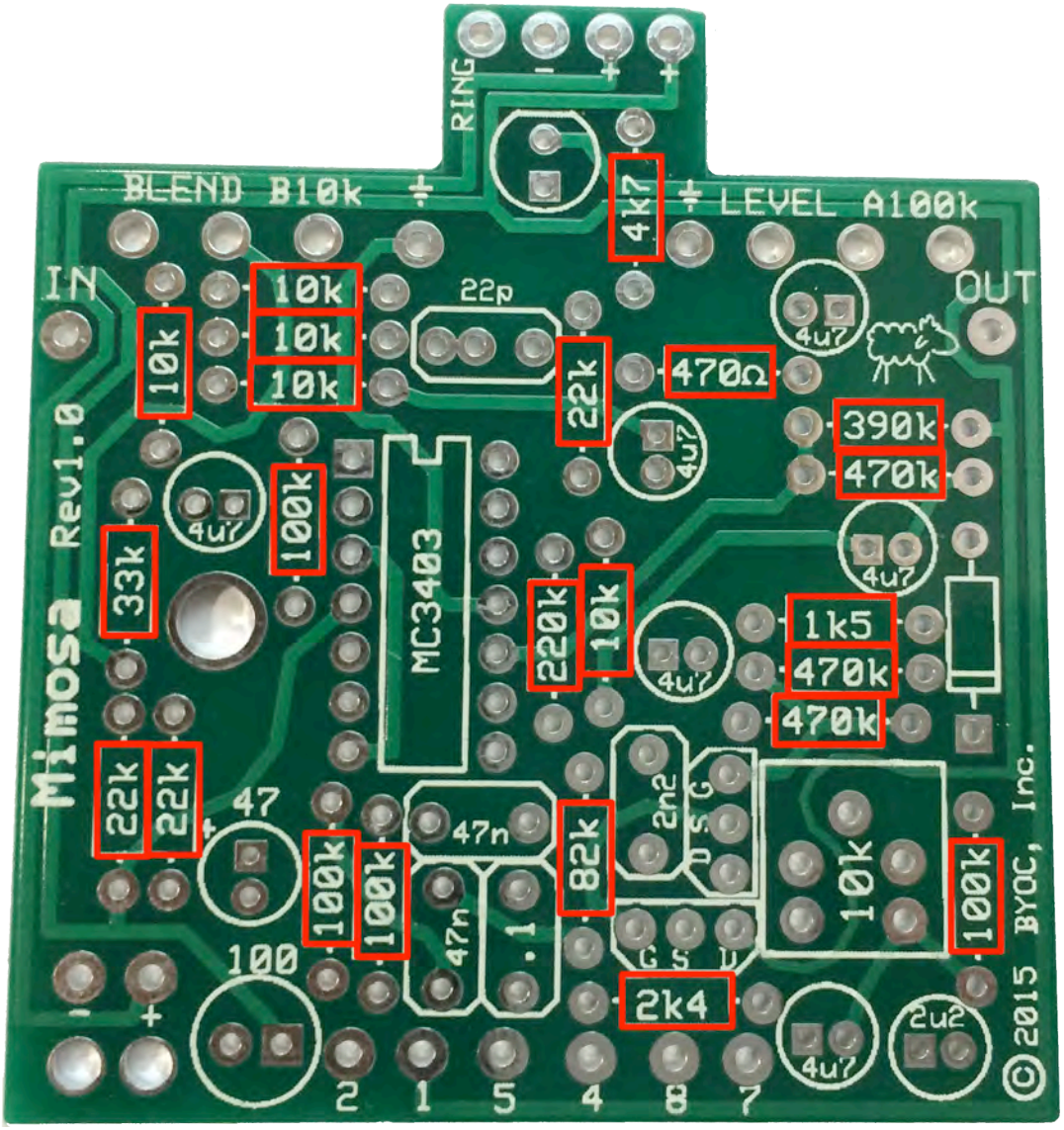


1 - A100k (LEVEL)
1 - B10k (BLEND)
1 - 10k Trimpot

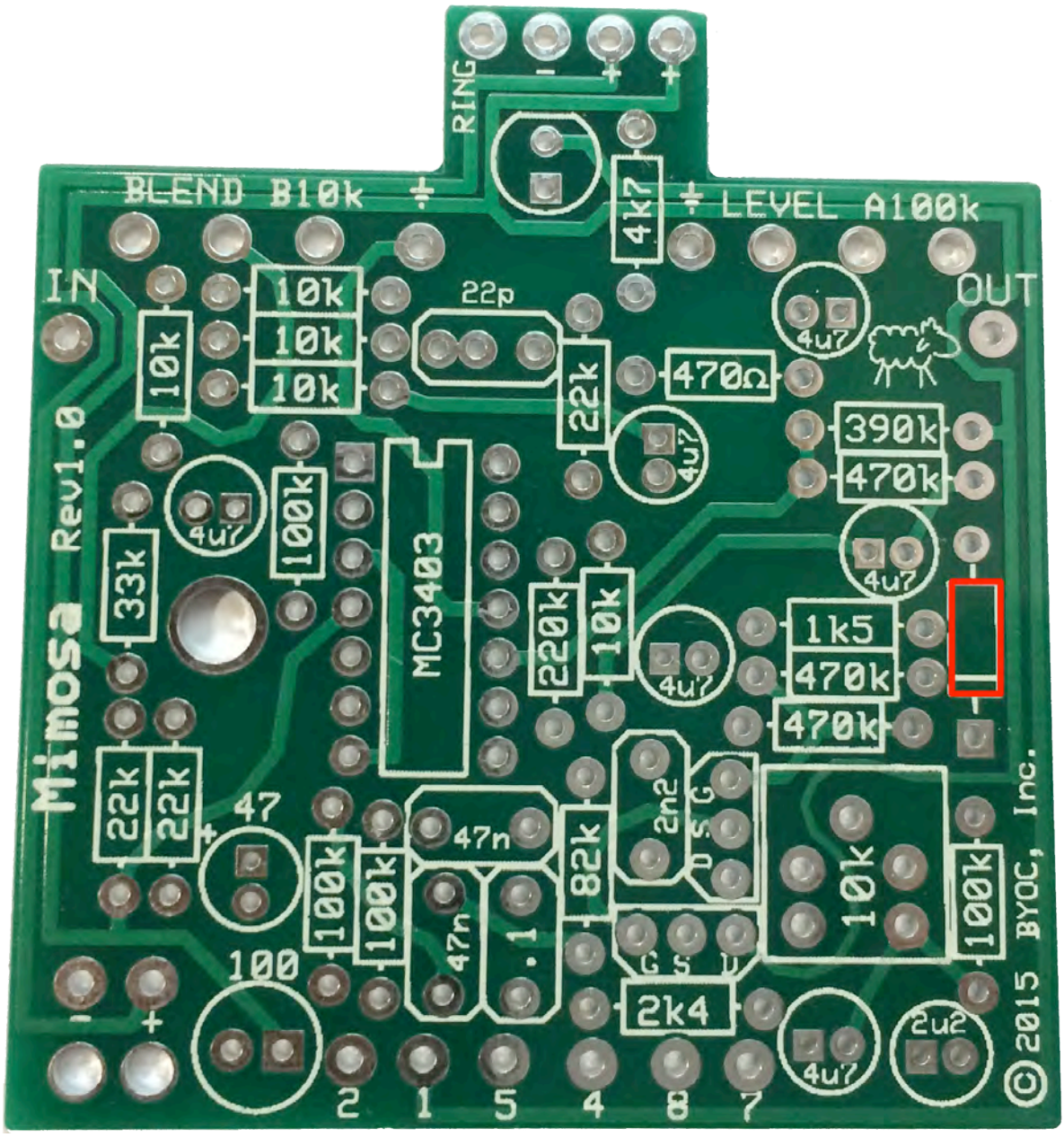
Hardware:
1 - drilled enclosure w/ 4 screws
1 - BYOC Mimosa PCB
1 - 3PDT footswitch
1 - AC adaptor jack
1 - ¼”mono jack
1 - ¼”stereo jack
1 - battery snap
4 - bumpers
hook-up wire

Populating the Circuit Board

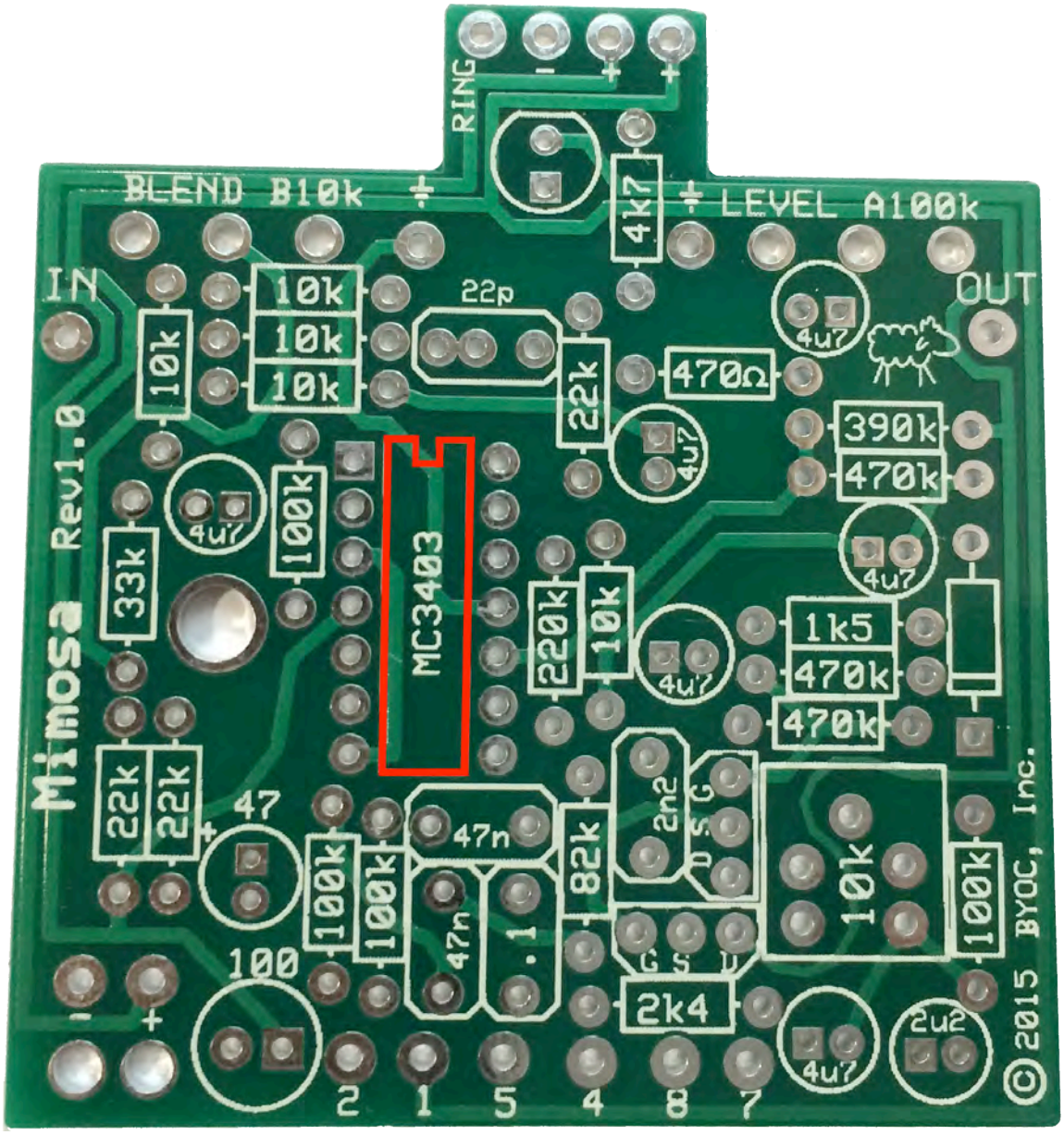
Step 1: Add all of the resistors. These are non-polarized so they can go in either direction.



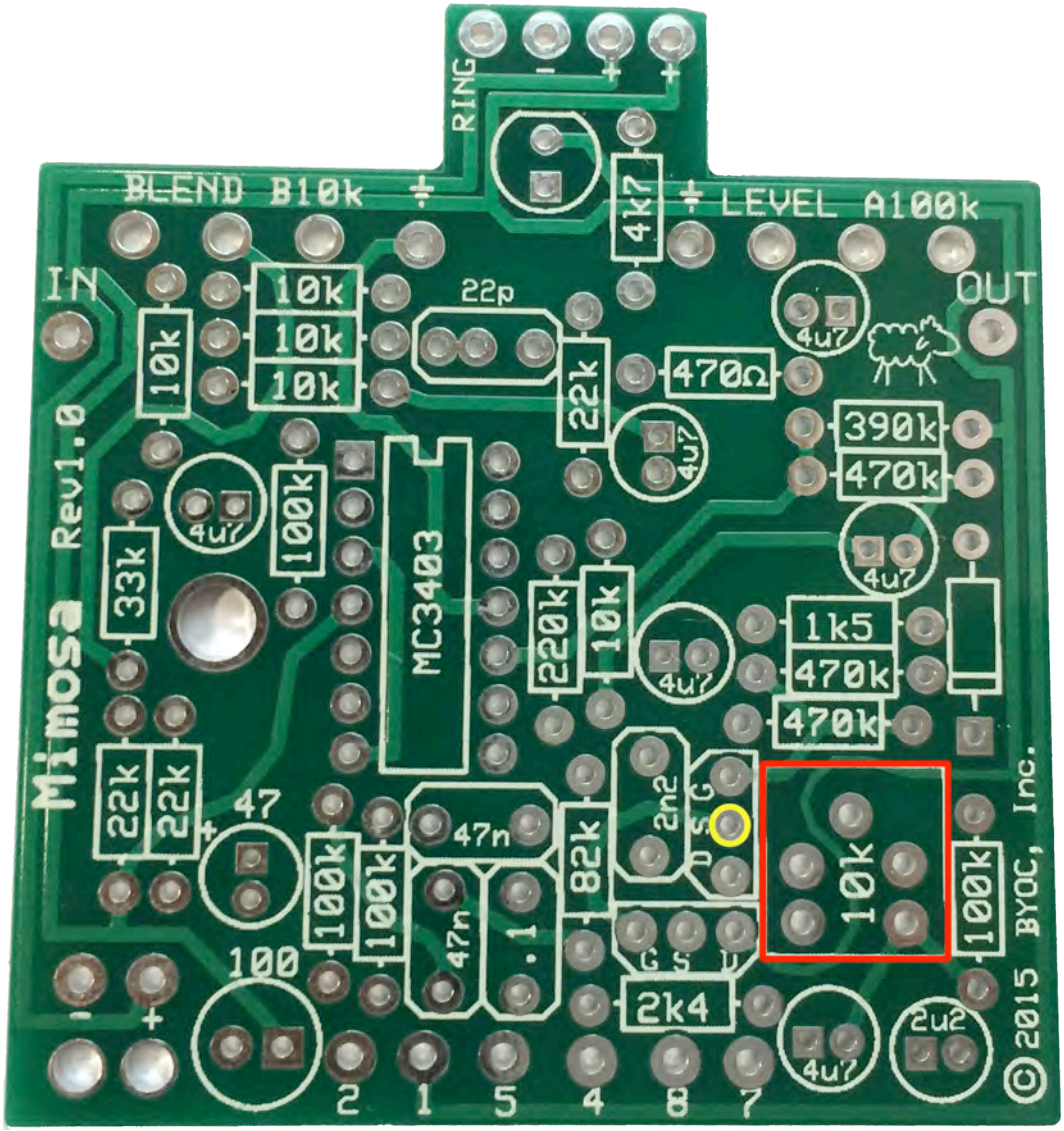
Step 2: Add the diode. Be sure to match the end of the diode with the stripe to the layout on the PCB. The striped end should go in the square solder pad. **If your diode has one white and one gray stripe, place the gray stripe in the square hole. If your diode has only a white dot, place the white dot side into the round hole.**



Step 3: Add the 14 pin 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 orient and install the ICs.



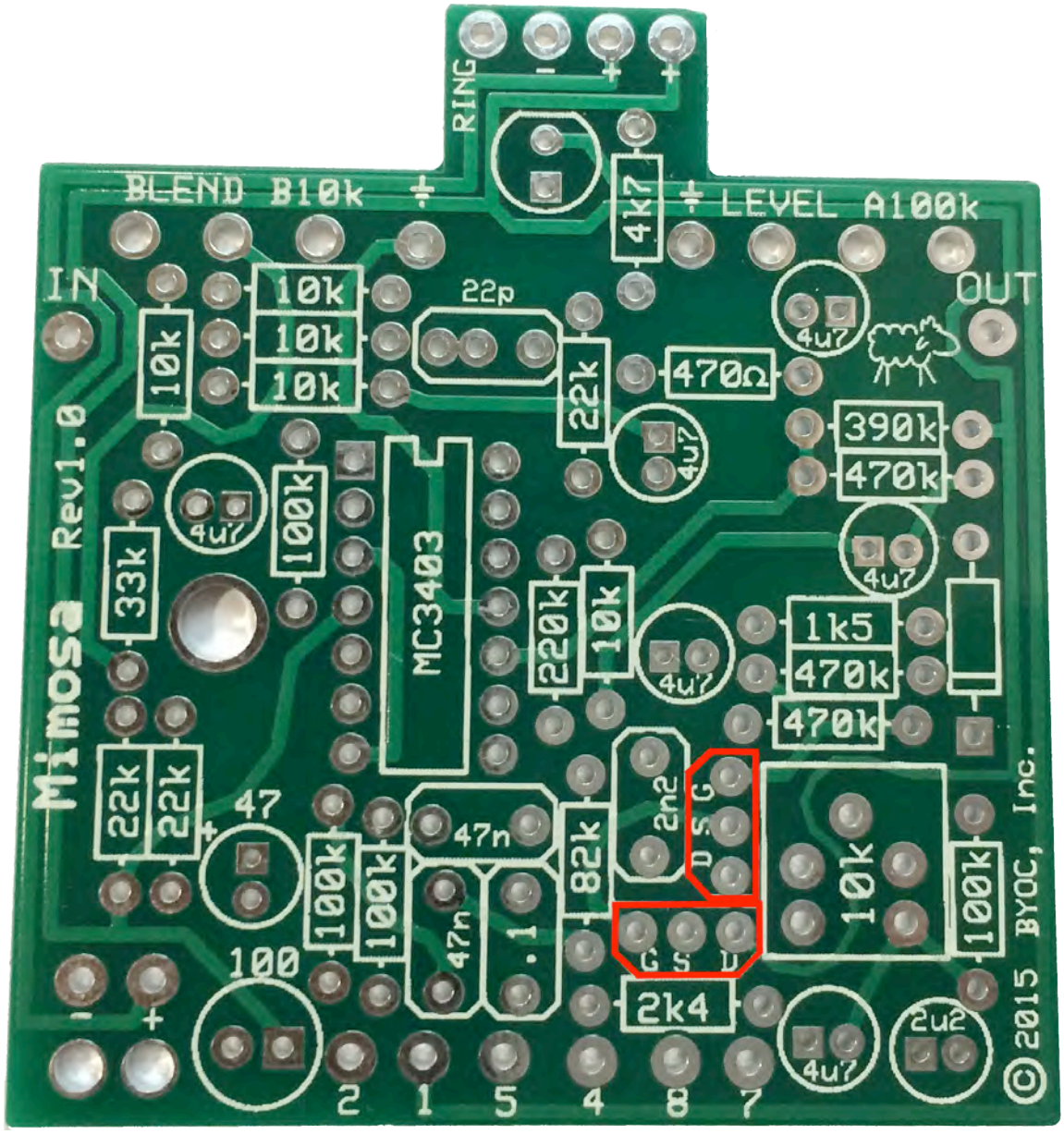
Step 4: Add the bias trimpot. There are 5 holes to accommodate different size trimpots, you will only use 3 holes.



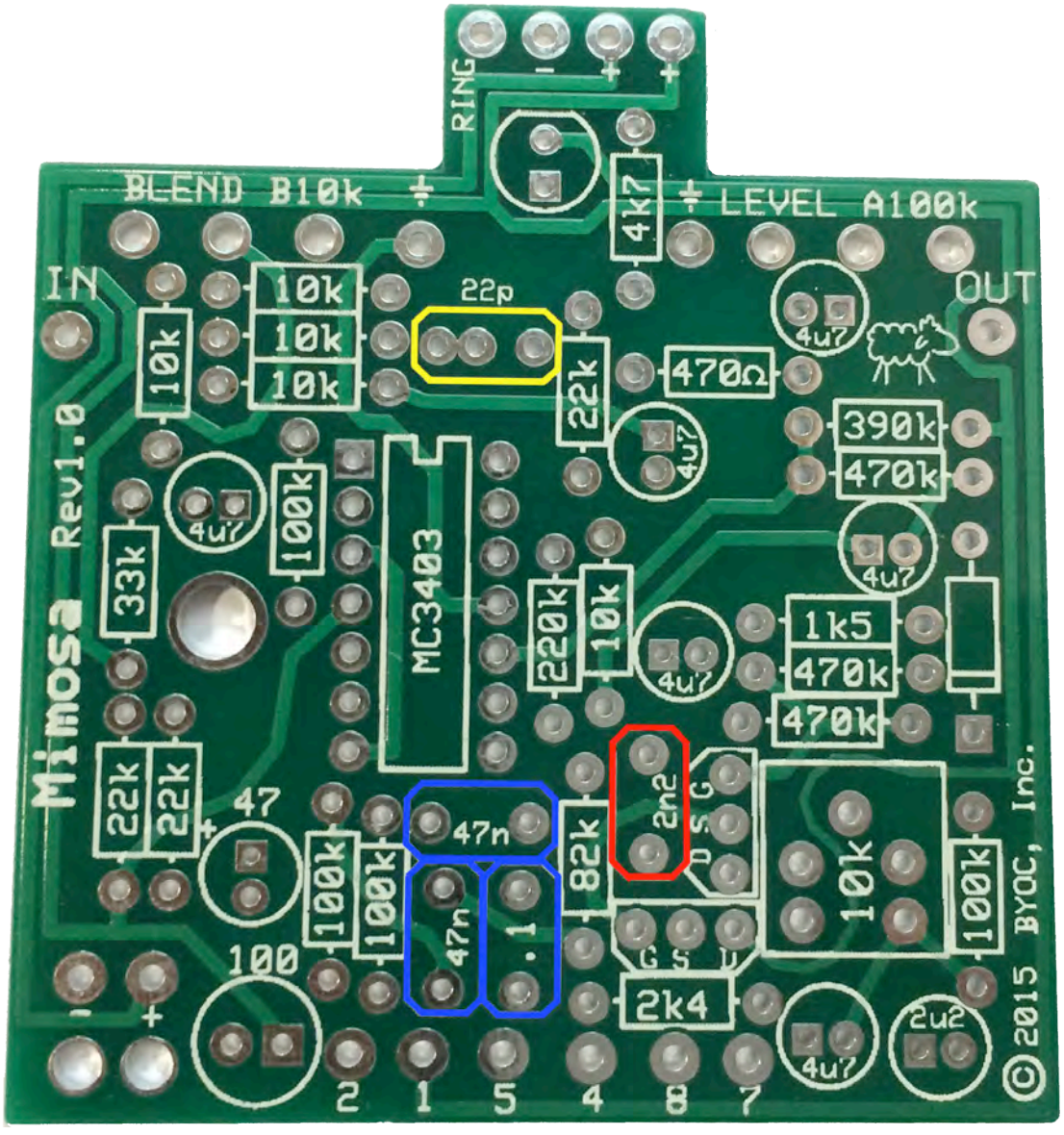
When you have finished building your mimosa kit, you will need to adjust the bias trimpot before the pedal will actually compress. You want to adjust the trimpot so that you get approximately 1.5-1.7V on the source of Q2. This would be the "S" eyelet highlighted in yellow.

Or you can just use your ears if you know what compression sounds like. Be sure to turn the BLEND knob full turn clockwise so that you are hearing 100% wet, compressed signal and no dry signal. You wil have a very hard time trying to adjust the compressor bias by ear when you are listening to the dry, uncompressed signal.

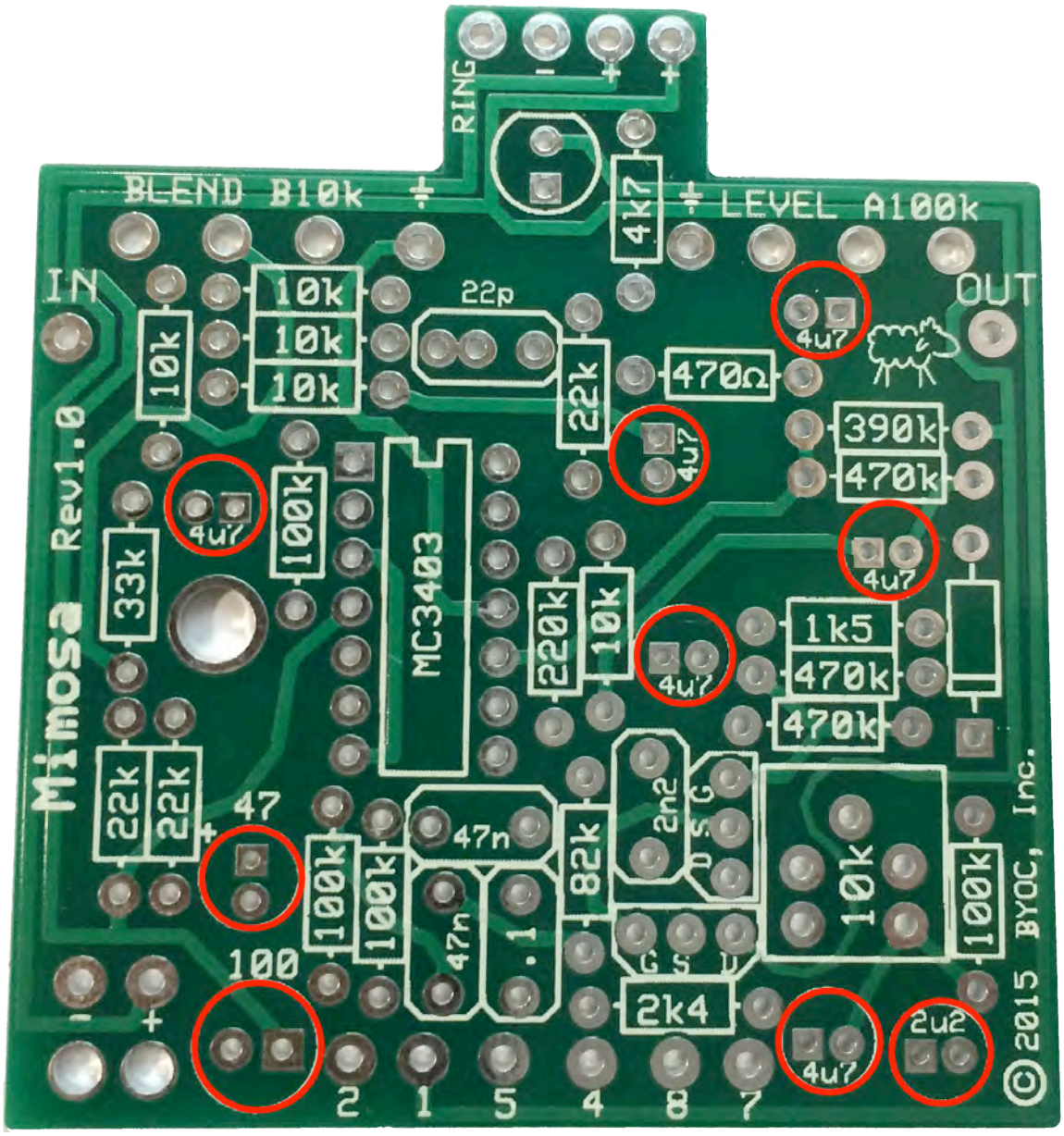
Step 5: Add the transistors. Be sure to match the flat side of the transistor with the flat side of the transistor on PCB layout.



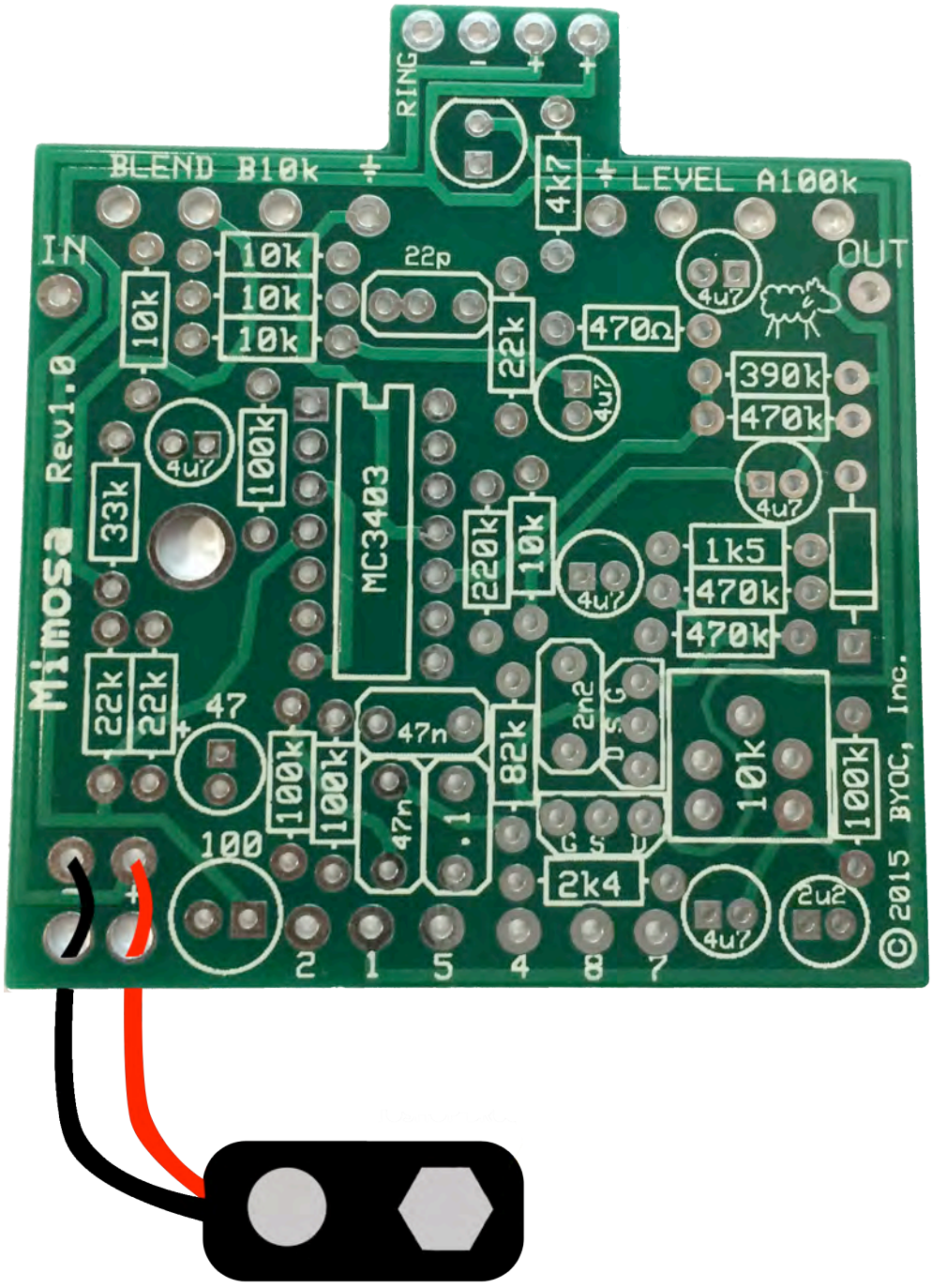
Step 6: Add the film and the ceramic disc capacitors. These are non-polarized so they can go in either direction. For the normal build, use the values listed on the PCB. To modify your Mimosa kit for use with a bass guitar, change the capacitors highlighted in blue to the .33u provided in your kit. **NOTE: Your kit may come with either a 22pf or 20pf ceramic disc capacitor based on availability.**



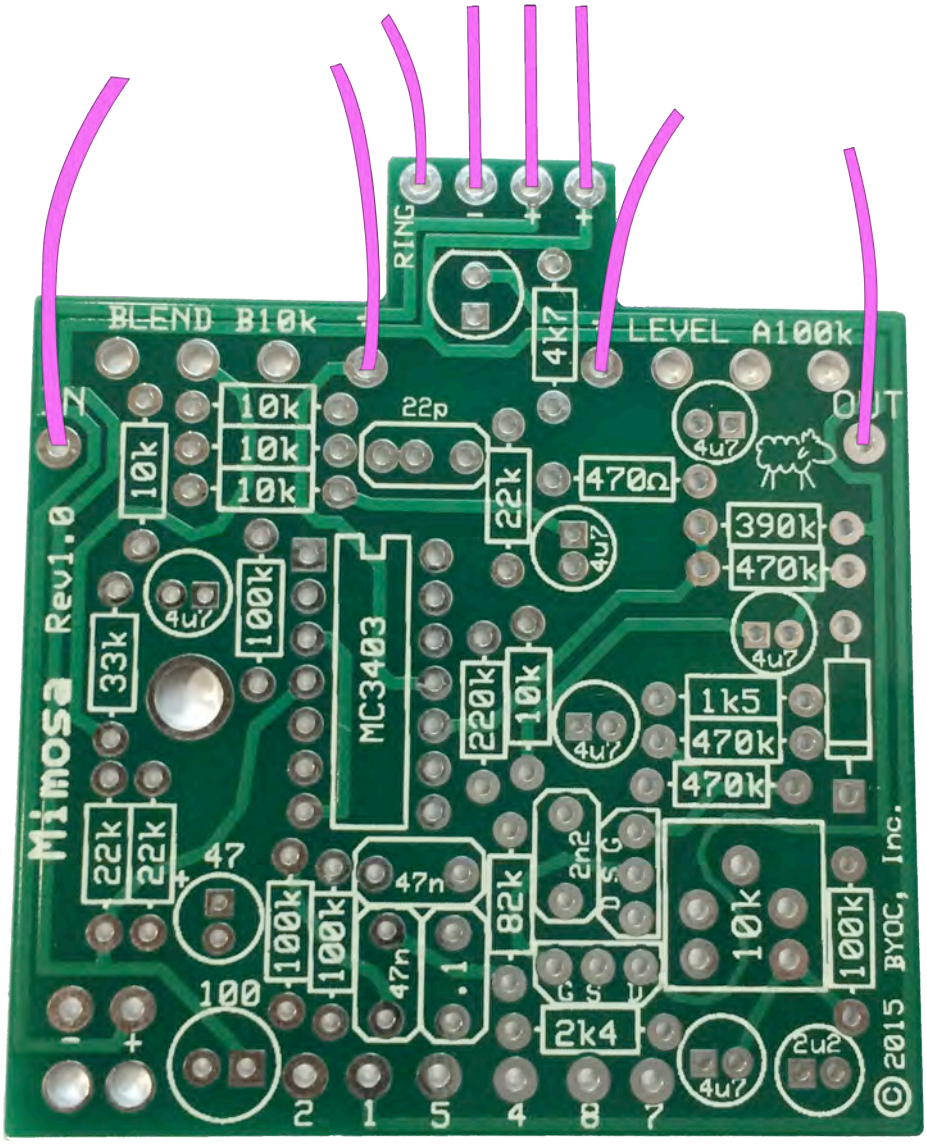
Step 7: Add the aluminum electrolytic capacitors. These ARE polarized, meaning there is a positive and negative end. The positive side will have a longer lead and goes in the square solder pad. The negative side will have a shorter lead and a stripe running along the body of the cap, and goes in the round solder pad.



Step 8: Add the battery snap. Thread the leads into the strain relief holes first through the bottom side of the PCB. Then insert the leads into their respective solder pad hole on the top side of the PCB. The red lead goes in the + hole and the black lead goes in the – hole.

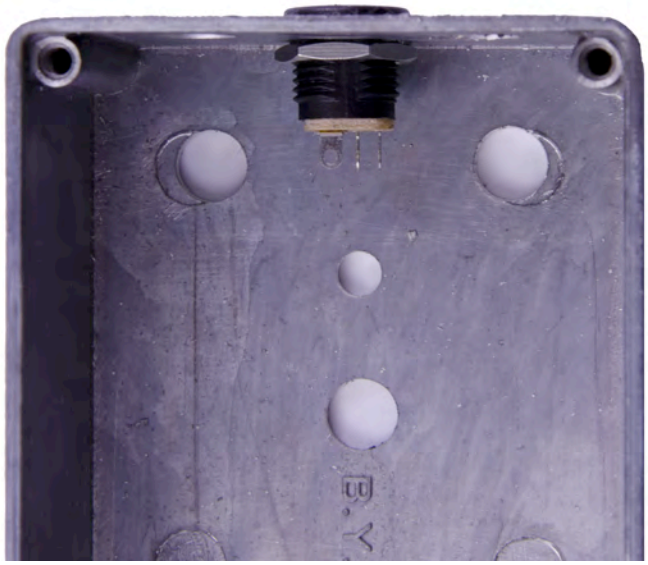


Step 9: Add wires to the IN, OUT, RING, -, +, (+), and the two Ground ⬇ eyelets. Start by cutting three 2.5” pieces of wire and 4 pieces of 1.5” wire. Strip 1/4” off each end and tin the ends. Tinning means to apply some solder to the stripped ends of the wires. This keeps the strands from fraying and primes the wire for soldering. Solder one end a 2.5” piece of wire to each of the IN, OUT, and Ground eyelets on the PCB. Load the wires in from the top and solder on the bottom of the PCB. Do the same with the 1.5” pieces of wire for the RING, -, +, and (+) eyelets



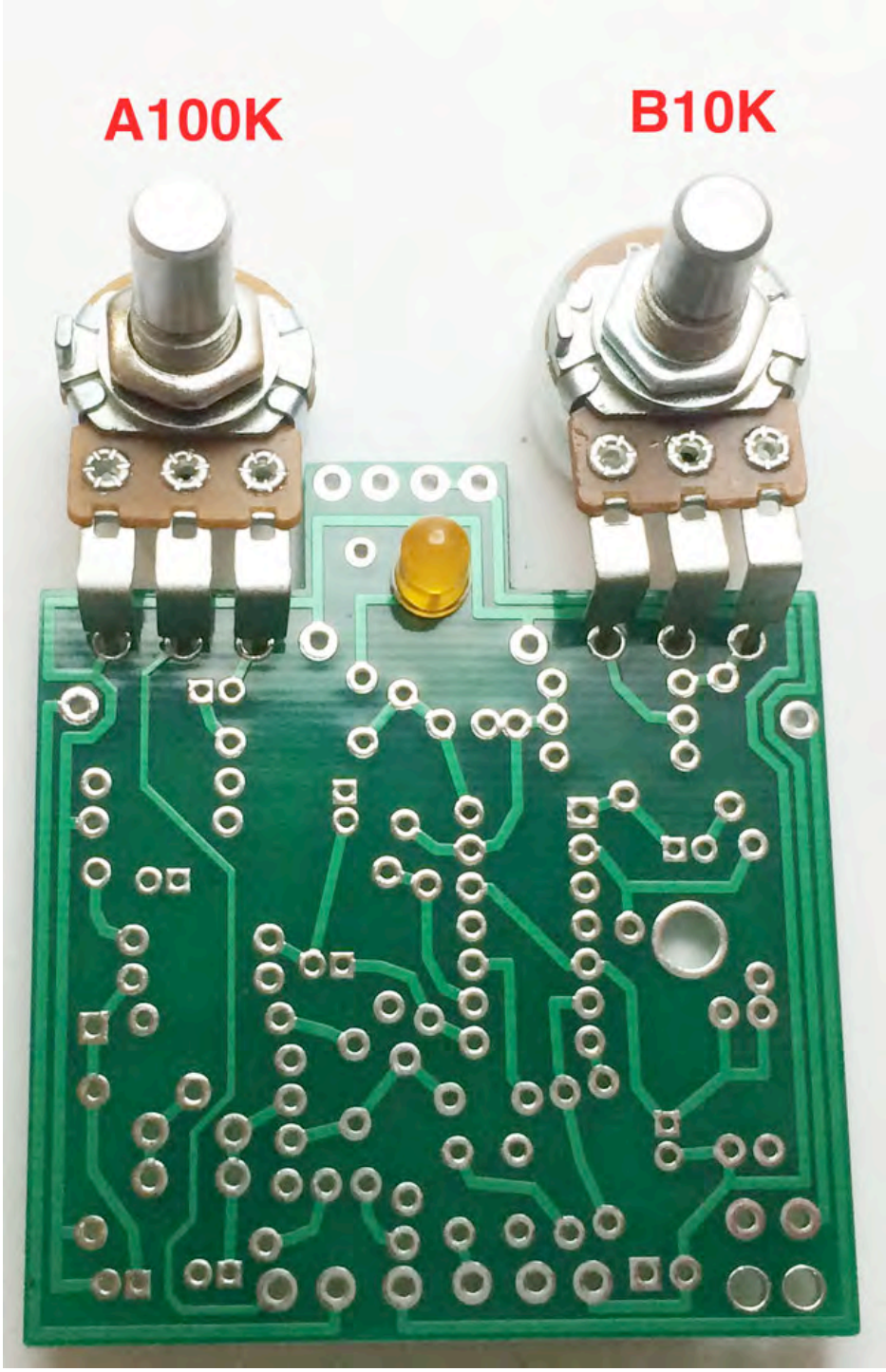
Main PCB Assembly

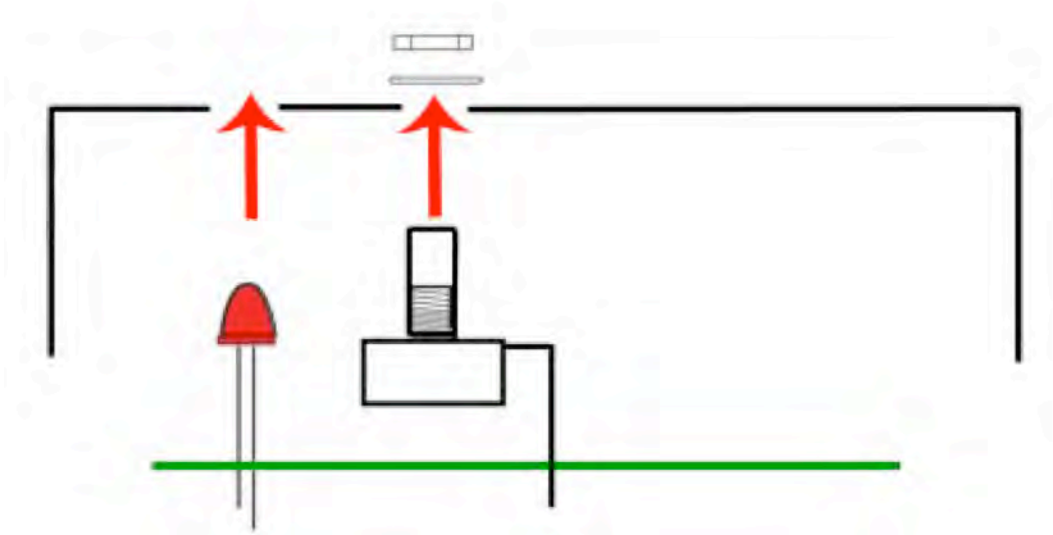
Step 1: Mount the AC adapter jack to the enclosure. Your kit may come with either an external thread or internal thread. Don't get confused by this. They still function exactly the same. You just thread the external nut on the outside and the internal nut on the inside. The picture below is of an internal nut jack.



Step 2: Flip the PCB over so that the bottom or solder side is up. Insert the five 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. THIS WILL GO INTO THE SQUARE SOLDER HOLE.



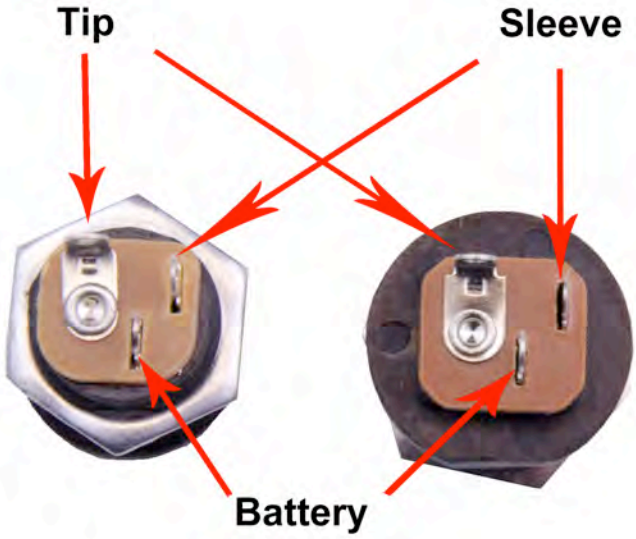


Step 3: Hold the PCB in one hand so that the component side of the PCB is in the palm of your hand and the bottom side with the pots, toggle switch and LED is facing up. Now use your other hand to guide the predrilled enclosure onto the PCB assembly so that the pots, toggle switch and LED all go into their respective holes. Once the PCB assembly is in place, secure it by screwing on the washers and nuts for the pots. Only tighten them with your fingers. You do not want them very tight yet. Make sure you've removed the nuts and washers from the pots and that you've also snapped the tabs off the pots as well before installing.

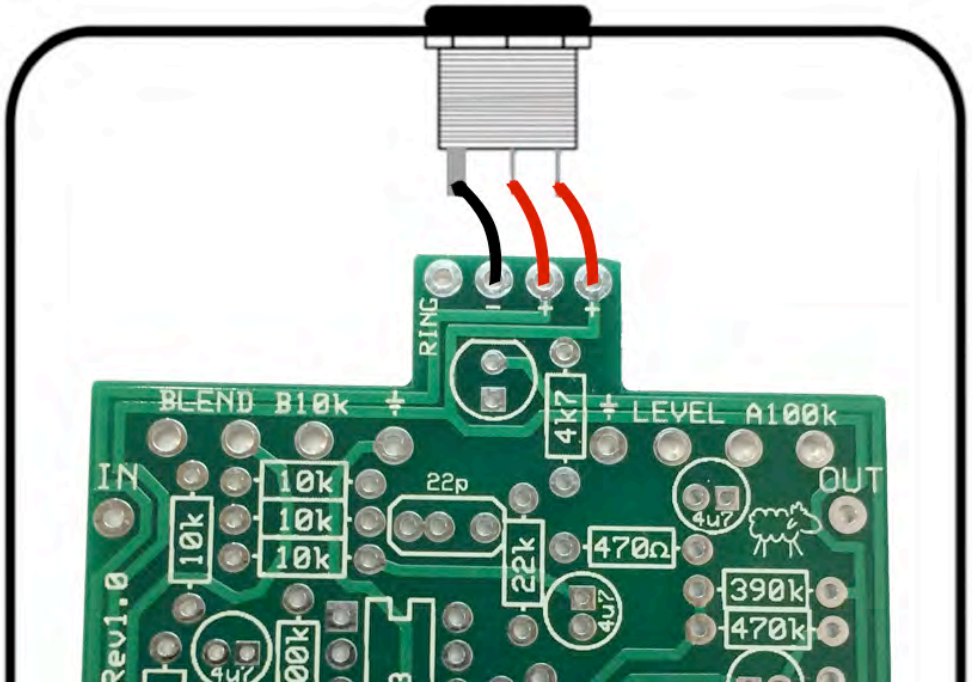
Step 4: Turn the entire pedal over so that the component side of the PCB is facing up. Lift the PCB up off the pots about 2mm just to make sure that the back of the PCB does not short out against the pots. Make sure the PCB is level and symmetrically seated inside the enclosure.

Step 5: Solder the pots, toggle switch and LED. You will be soldering on the component side (top) of the PCB. After you have soldered them in place, be sure to tighten up their nuts. TIP: only solder one lug of each component at first. This will secure everything in place and still allow you to wiggle things around if you need to adjust the fit of anything. Once you have everything perfect, go ahead and solder everything else.

WIRING



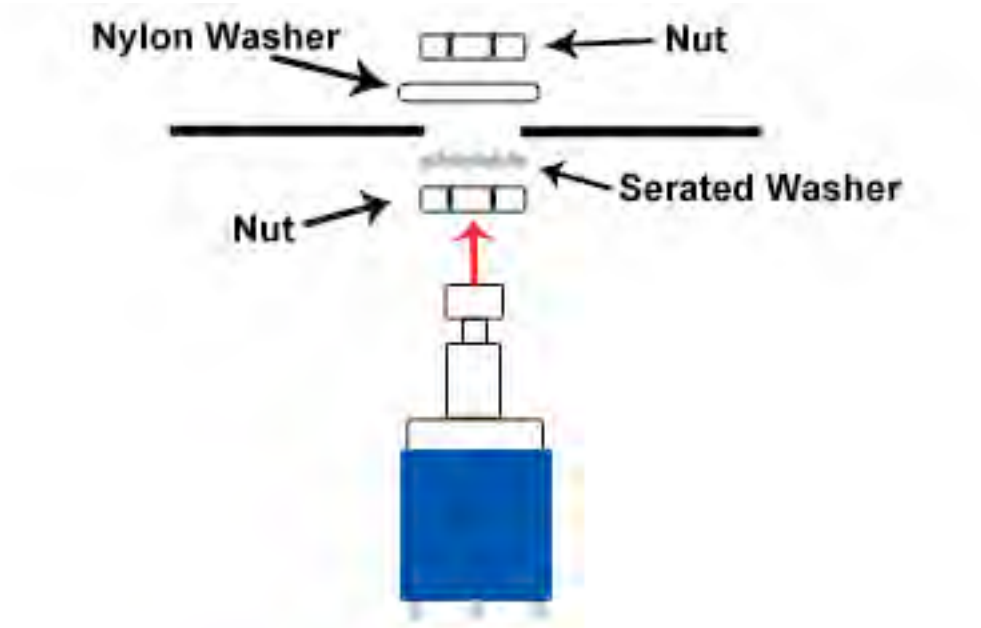
Step 6: Connect the TIP (negative) terminal of the DC adapter jack to the eyelet on the PCB labeled “-“. Connect the SLEEVE of the DC adapter jack to the eyelet on the PCB labeled “+” farthest to the right. Connect the battery disconnect terminal of the DC adapter jack to the second eyelet on the PCB labeled “+” located in the middle of the other two eyelets .



Stereo (input) Jack

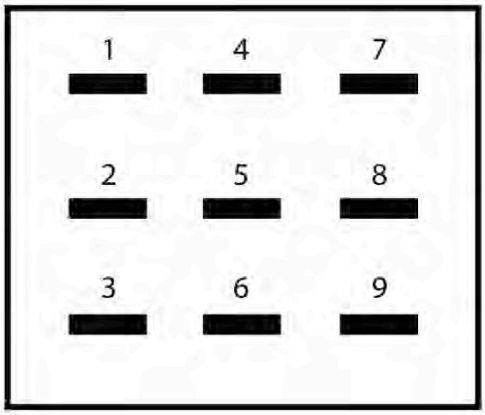
Mono (output) Jack

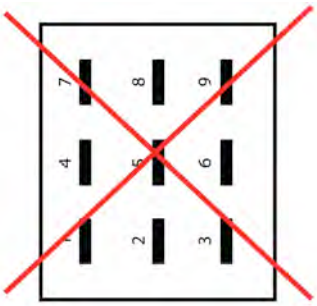
Step 1: Install the 1/4” jacks to the enclosure.



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.
NOTE: It may be easier to wire up part of the foot switch before installing it into the pedal. There will be more room to work & it will be much easier to thread the lug 4 to lug 9 jumper.

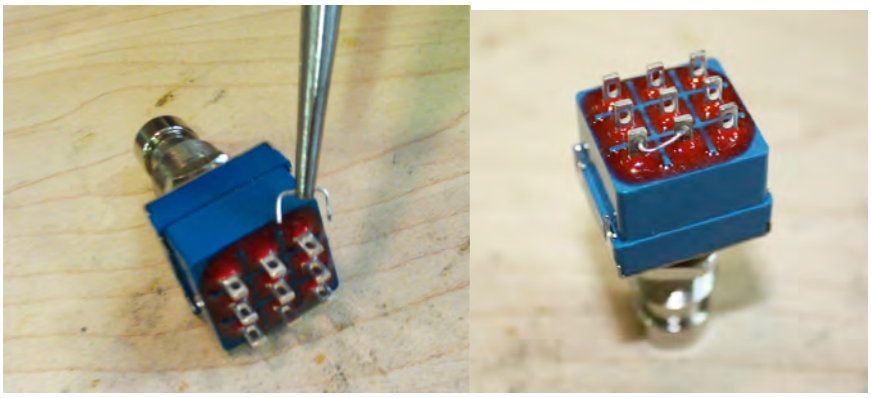
FOOT SWITCH SOLDER LUG DESIGNATIONS

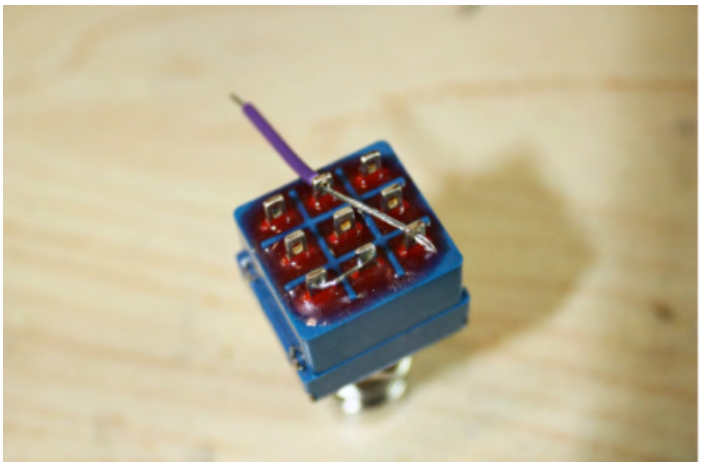




Step 3: Wiring the foot switch.

- Make a jumper between lugs 3 & 6 from clippings from the resistors. Simply use your needle nose pliers to bend a piece of clipping into a U shape & insert into lugs 3 & 6, then solder.

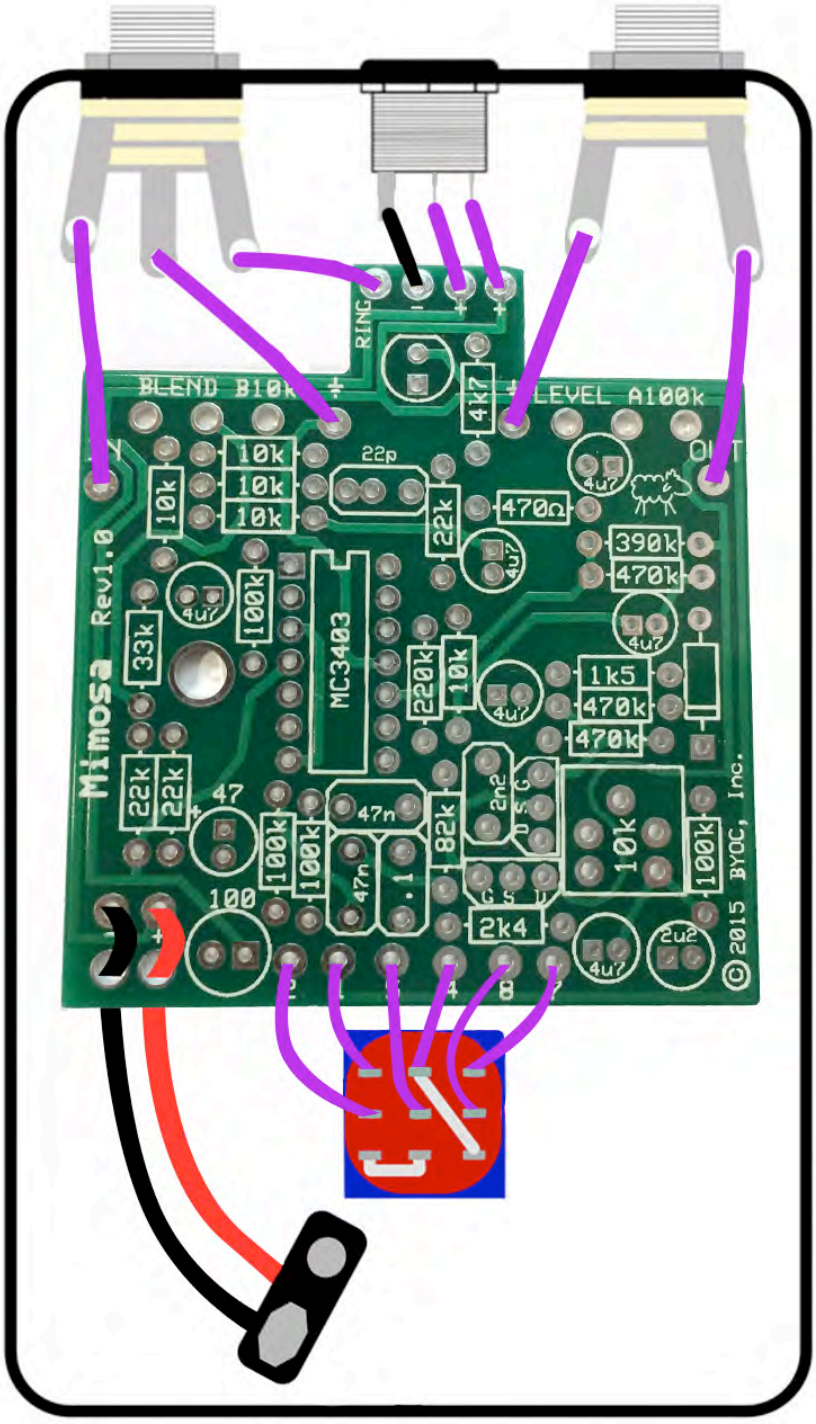




- Cut a two 2” piece of wire. Strip 1/8” off each end and tin.
- Strip ¾” off the other end and very carefully tin. You do not want too much solder on this end.
- Thread the longer stripped end into LUG4 and through to LUG9.
- Solder both lugs 4 and 9.
- This step can be tricky. If it is too difficult or frustrating for you, you can simply insert two pieces of wire into LUG4 and solder. Then connect the other end of one of the wires to LUG9. It won’t look as pretty, but it will accomplish the exact same thing.

Step 4: Install the foot switch into the enclosure if it isn’t already. Insert the foot switch wires into their respective eyelets on the PCB. You can insert them into the top side and solder on the top side as well. The solder pads should be large enough (if you are using a soldering iron that isn’t too big) to allow you to do this without burning the PVC coating on the wires if you are careful. If you do singe the plastic on the wires, it’s OK. It’s not going to hurt anything. It’s purely aesthetic. See diagram below.

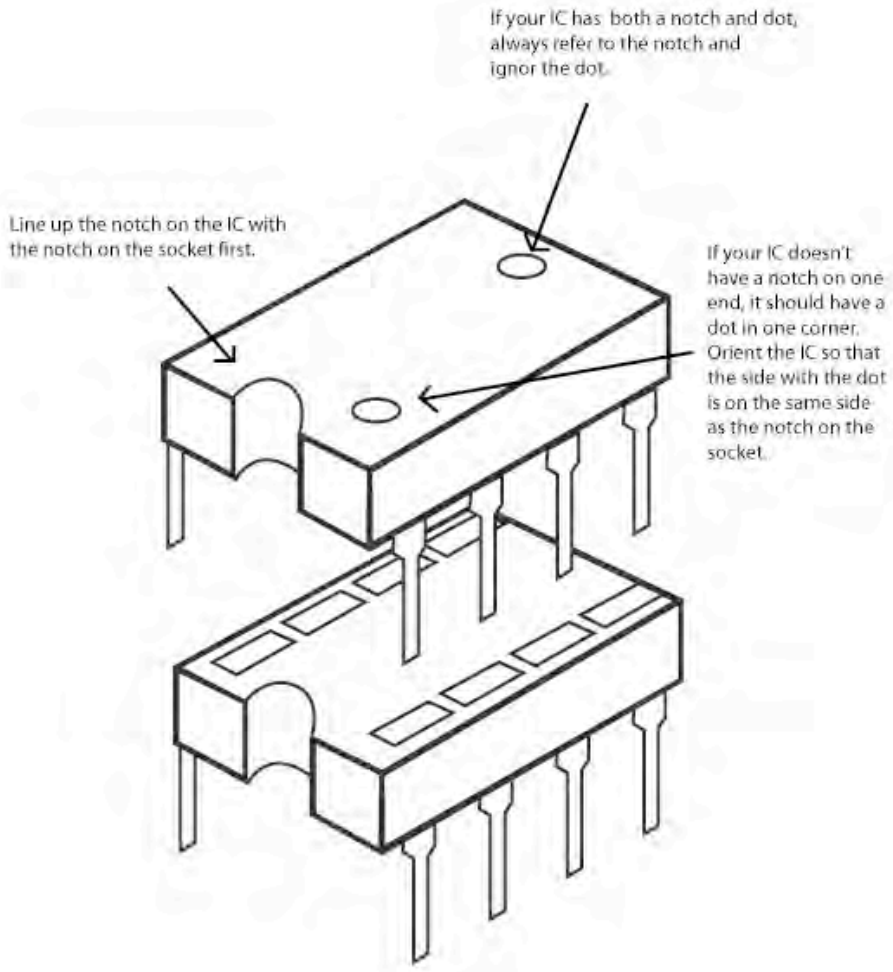
Step 5: Connect the wires at the top end of the PCB to the IN and OUT jacks. The “out” eyelet will go to the tip of the OUT jack and the “in” eyelet will go to the tip of the IN jack (refer to page 20). Connect the ground eyelet on the left to the sleeve of the IN jack and the ground eyelet on the right to the sleeve of the OUT jack. Connect the “ring” eyelet to the ring of the IN jack. See diagram below.



Installing IC/Finish up

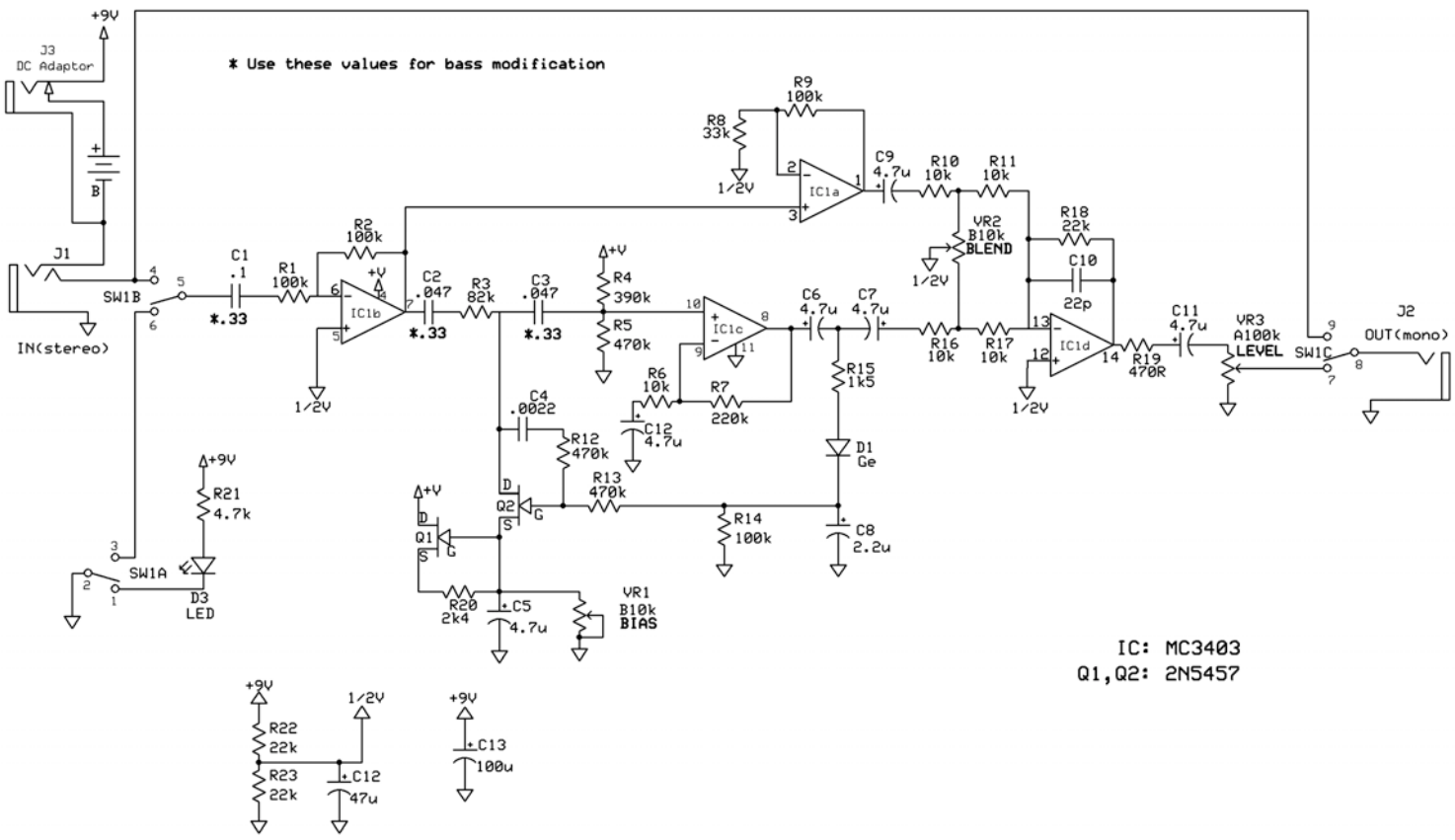
Don't forget to add the knobs, put the cover on the enclosure, and apply the bumpers to the cover.

Remember that you are not soldering the IC directly to the board, you will use the provided sockets. (Shown is an eight pin IC, it is the same for the 14 pin MC3403 IC.)



Operating Overview

Level: This controls the overall output volume.
Blend: This is a blend control between the dry and compressed signal. All the way clockwise is the fully compressed signal, and all the way counter-clockwise is the dry, uncompressed signal.
Power supply: 9V battery or 2.1mm negative tip
Current Draw: 5.5mA
Input Impedance: 100K ohms
Output Impedance: 100k ohms



For hi-res schematic visit http://www.byocelectronics.com/mimosaschematic.pdf

Please visit
http://byocelectronics.com/board
For any technical support