Build Your Own Clone
Phase Royal
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:

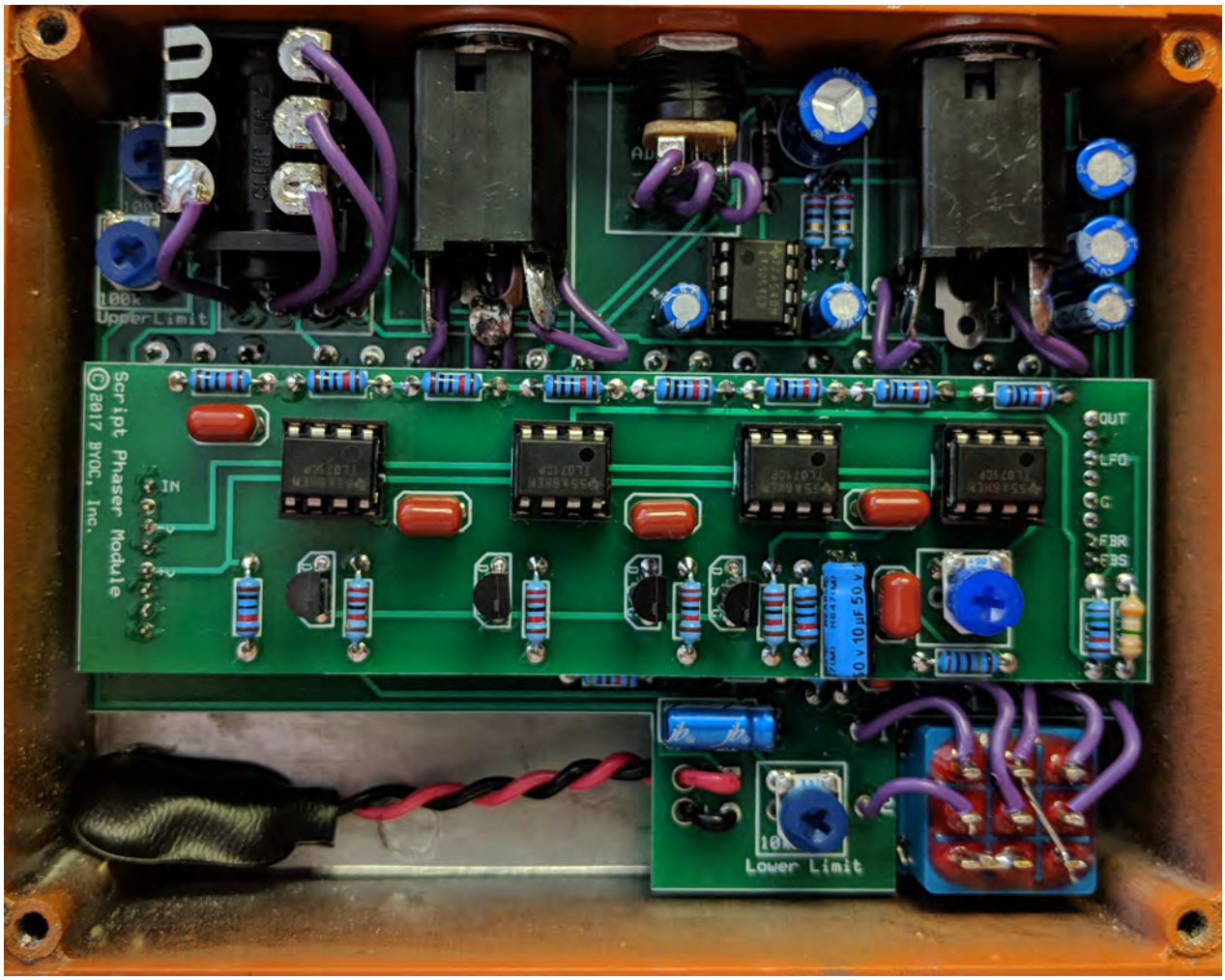
1. A detailed explanation of what the problem is. (more than, "It doesn't work, help")
2. Pic of the topside of your PCB.
3. Pic of the underside of your PCB.
4. Pic that clearly shows your footswitch/jack wiring and the wires going to the PCB
5. A pic that clearly shows your wiring going from the PCB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

Phase Royal
Kit Instruction Index

This is what your kit should look like when it's complete. Your kit may come with different color capacitors, switches ect. Don't be alarmed by this. They all still do the exact same thing.



Parts Checklist for the Phase Royal Kit

Resistors: (Metal Film (5 Bands) / Carbon Film (4 Bands))

2 - 22R (Red/Red/Black/Gold/Brown) / (Red/Red/Brown/Gold)
1 - 220R (Red/Red/Black/Black/Brown) / (Red/Red/Brown/Gold)
2 - 470R (Yellow/Purple/Black/Black/Brown) / (Yellow/Purple/Brown/Gold)
2 - 4k7 (Yellow/Purple/Black/Brown/Brown) / (Yellow/Purple/Red/Gold
3 - 10k (Brown/Black/Black/Red/Brown) / (Brown/Black/Orange/Gold)
6 - 47k (Yellow/Purple/Black/Red/Brown) / (Yellow/Purple/Orange/Gold)
1 - 100k (Brown/Black/Black/Orange/Brown) / (Brown/Black/Yellow/Gold)
2 - 150k (Brown/Green/Black/Orange/Brown) / (Brown/Green/Yellow/Gold)
2 - 470k (Yellow/Purple/Black/Orange/Brown) / (Yellow/Purple/Yellow/Gold)
1 - 1M (Brown/Black/Black/Yellow/Brown) / (Brown/Black/Green/Gold)
1 - 2M2 (Red/Red/Black/Yellow/Brown) / (Red/Red/Green/Gold)

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

Capacitors:

2 - 47p ceramic disc cap (may say “33” on the body)
1 - .01u/10n film cap (may say “103” on the body)
1 - .1u/100n film cap (may say “104” on the body)
8 - 10uf Aluminum Electrolytic
2 - 15uf Aluminum Electrolytic
2 - 100uf Aluminum Electrolytic
1 - 220uf Aluminum Electrolytic

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

Diodes:

1 - 1N4001 diode

IC's:

3 - TL072
1 - LT1054 Charge Pump
4 - DIP8 sockets

Transistors:

1 - 2N3904
1 - 2N3906

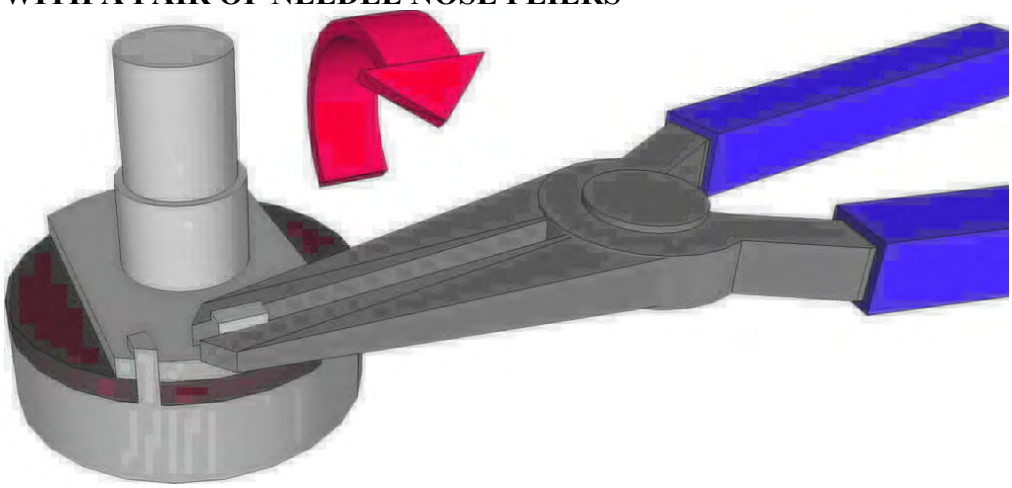
Optocouplers:

1 - Single Element optocoupler VTL5C4

Trimpots:

2 - 250k
1 - 25k

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

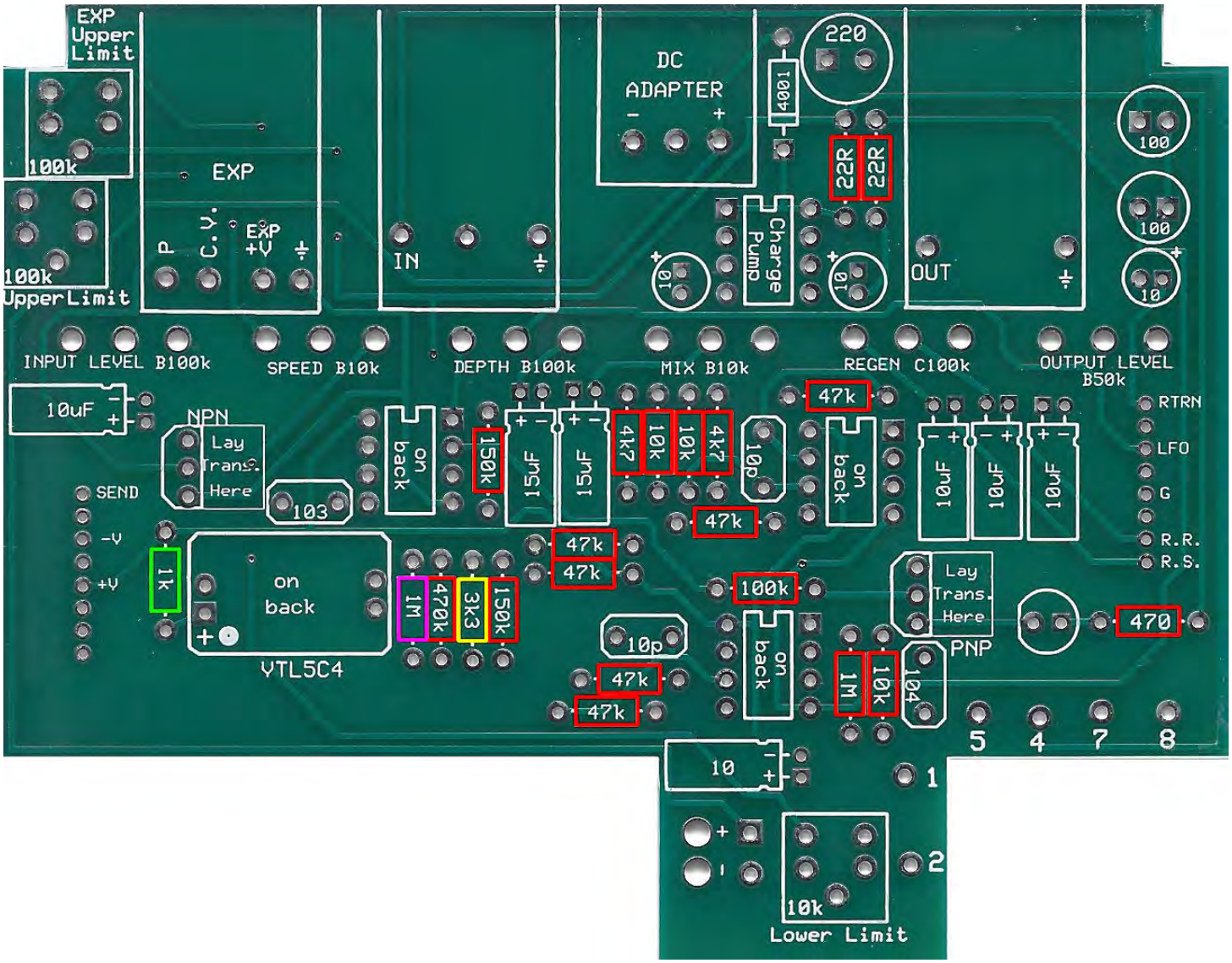


2 - B10k (SPEED, MIX)
1 - B50k (OUTPUT LEVEL)
2 - B100k (INPUT LEVEL, DEPTH)
1 - C100k (REGEN)

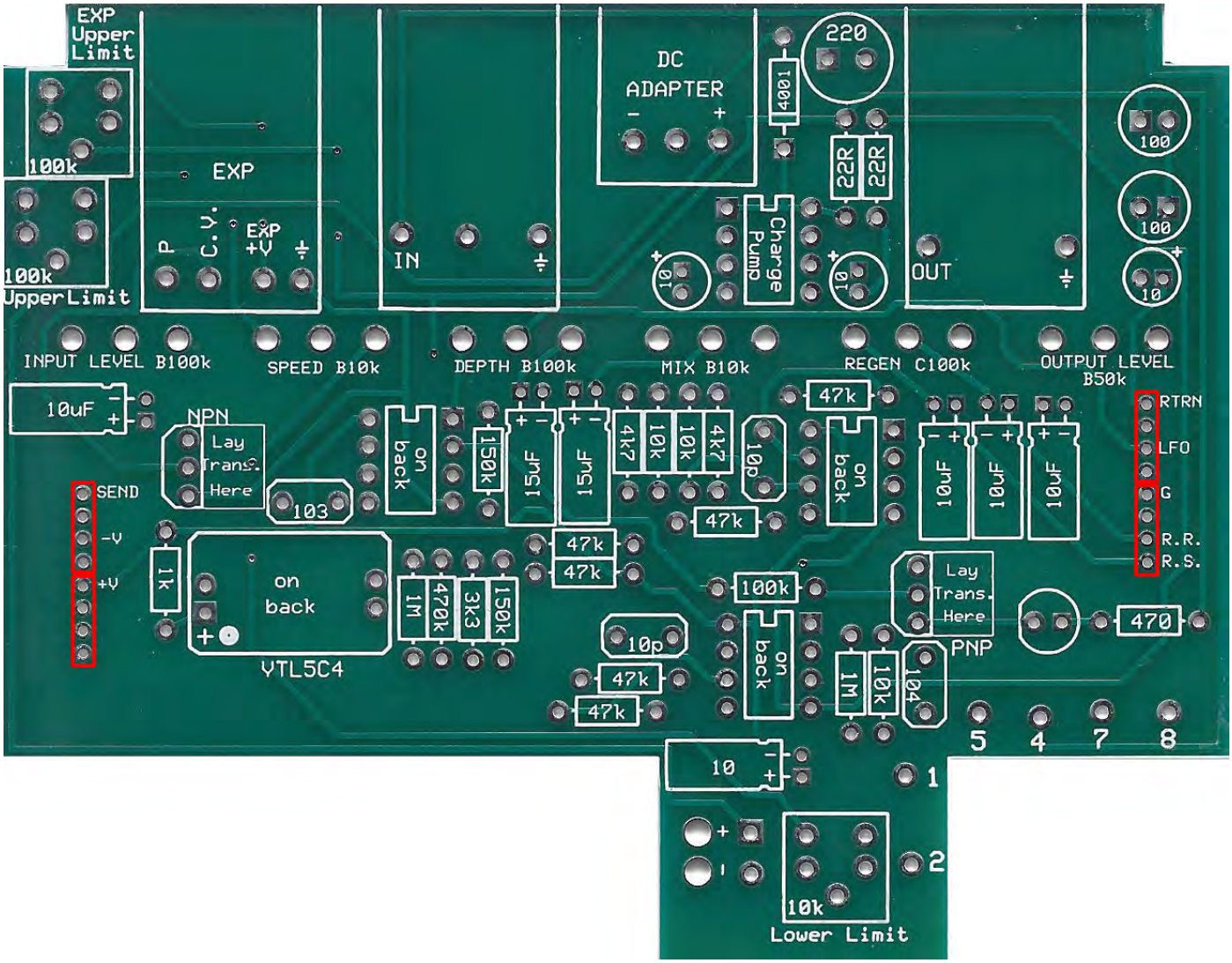
Hardware:
1 - predrilled enclosure w/ 4 screws
1 - Phase Royal Printed Circuit Board
1 - 3PDT Footswitch
4 - 1X4 Header socket
2 - Enclosed Jacks
1 - Switching Jack
4 - rubber bumpers
2 - lock washers (for in and out jacks)
hook-up wire

Populating the Circuit Board

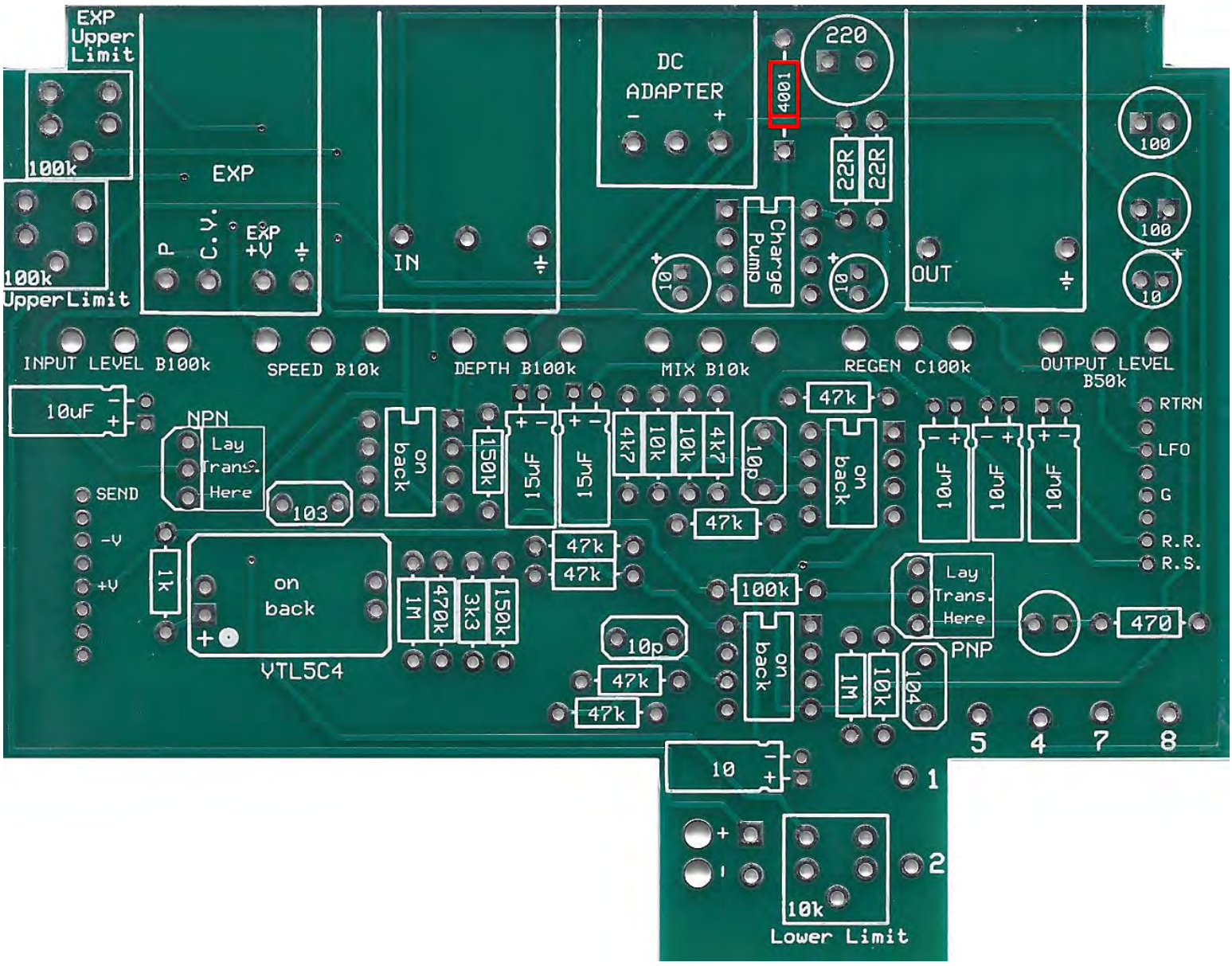
Step 1: Add all the resistors. Resistors are not polarized and can be inserted into the PCB in either direction, meaning you don't have to worry about orienting them.
NOTE: The resistor highlighted in **green** is 220Ohm, the resistor highlighted in **pink** is 2M2, and the resistor highlighted in **yellow** is 470Ohm.



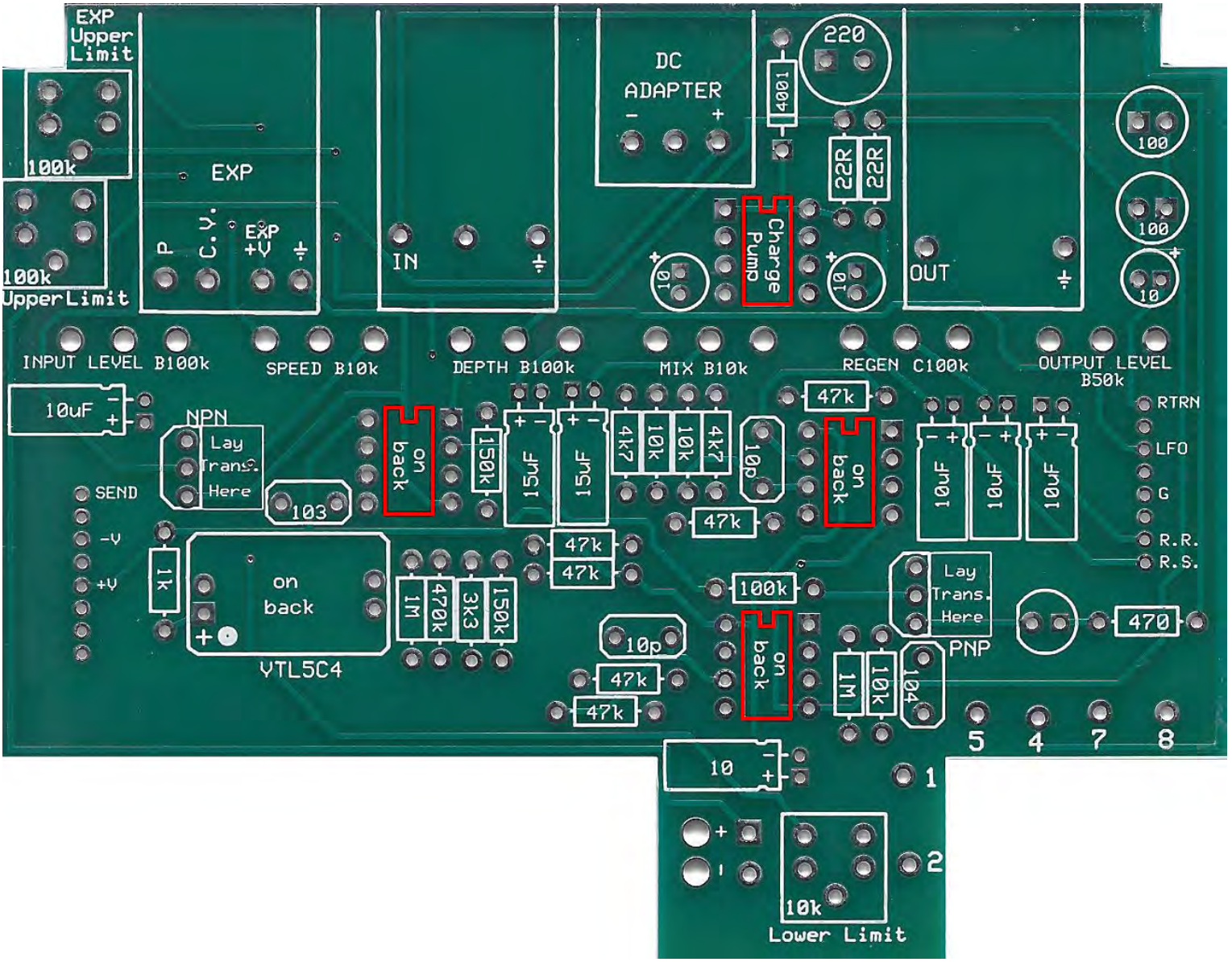
Step 2: Add the header sockets.



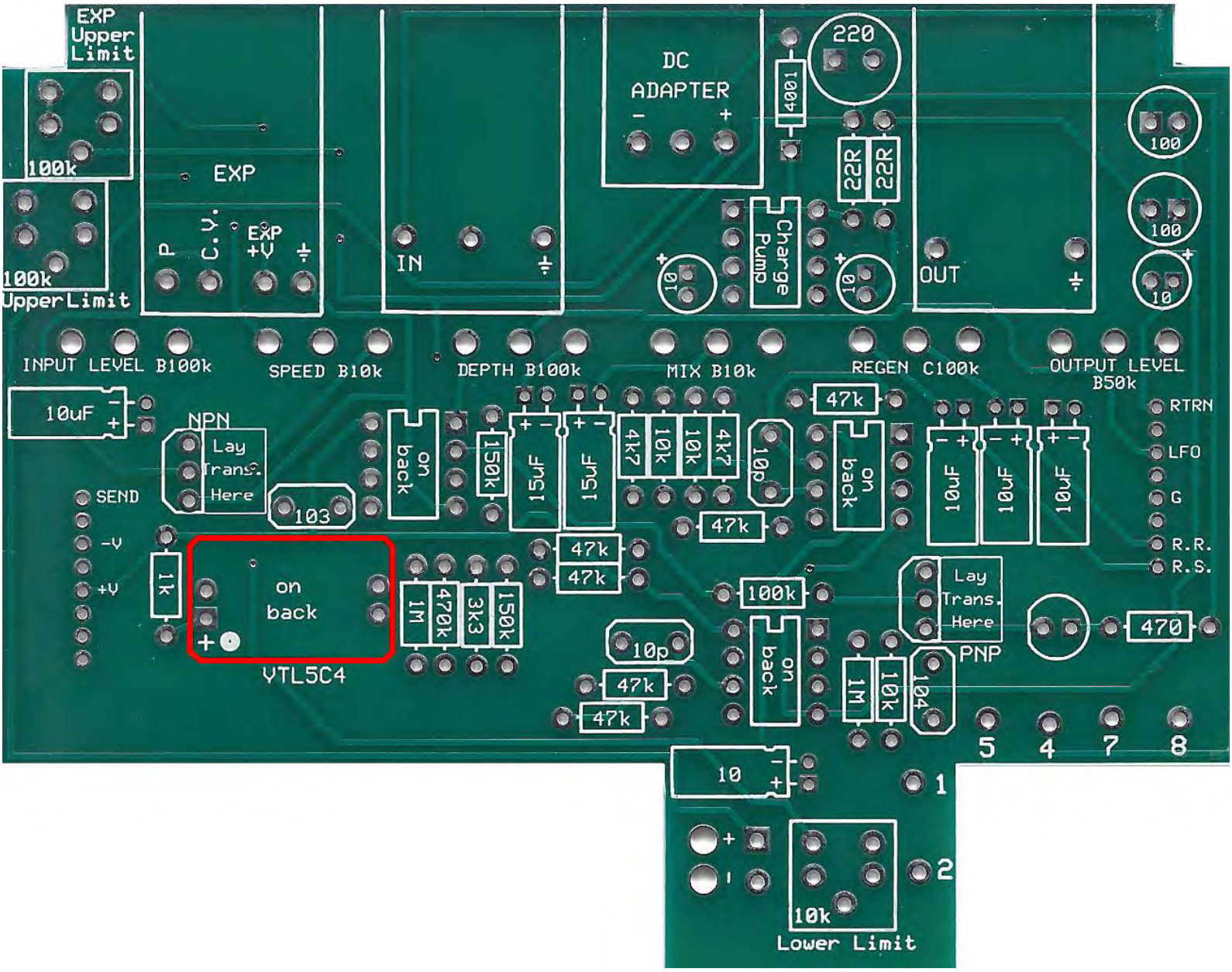
Step 3: Add the diode. Be sure to match the end of the diode with the stripe to the layout on the PCB. The striped end should go in the square solder pad.



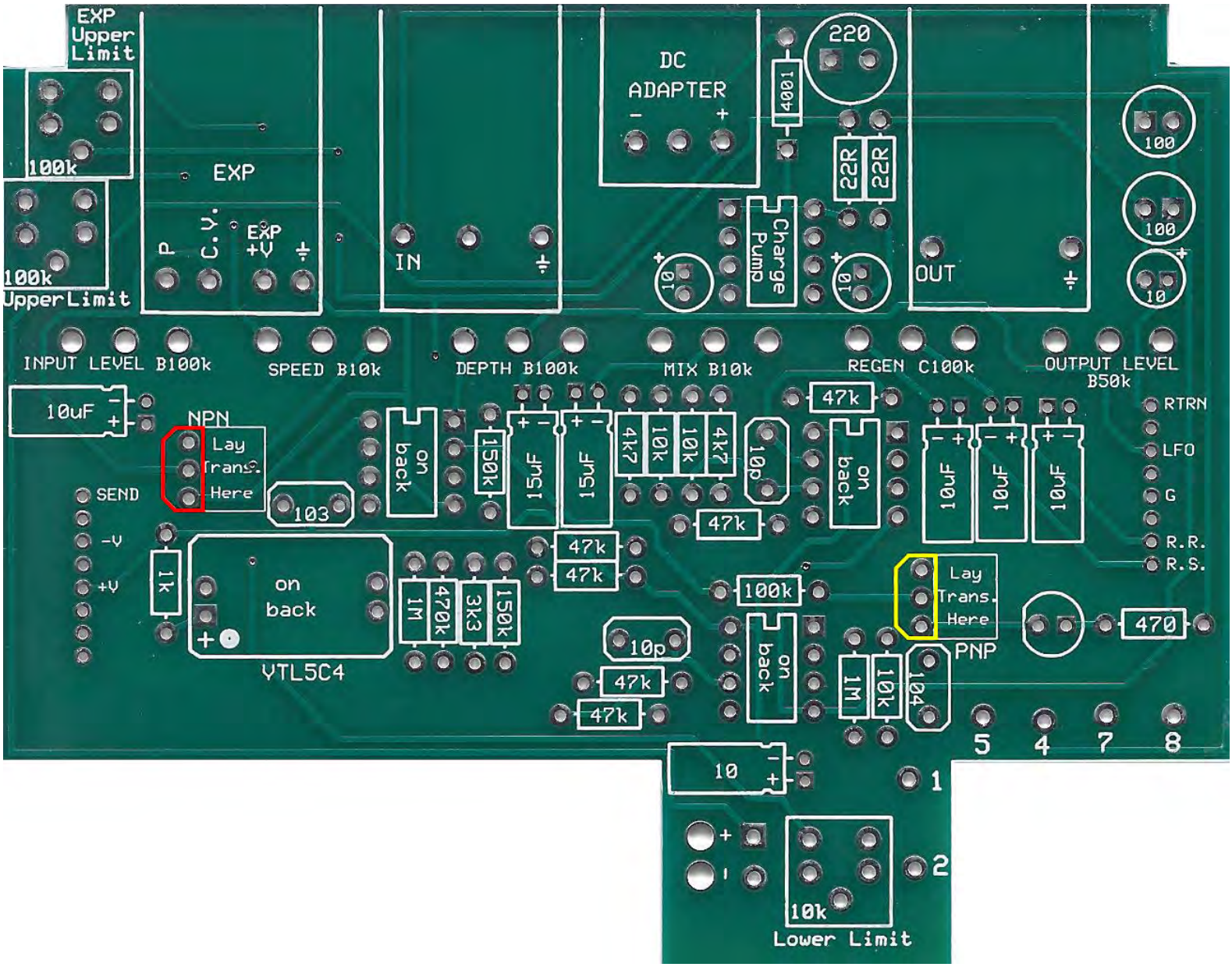
Step 4: Add the IC sockets. Be sure to align the notch on the IC sockets with the notch on the PCB screenprint. **For the TL072 ICs that say 'on back', place the sockets on the back of the PCB. This is required to allow the module board to sit in the headers properly.**



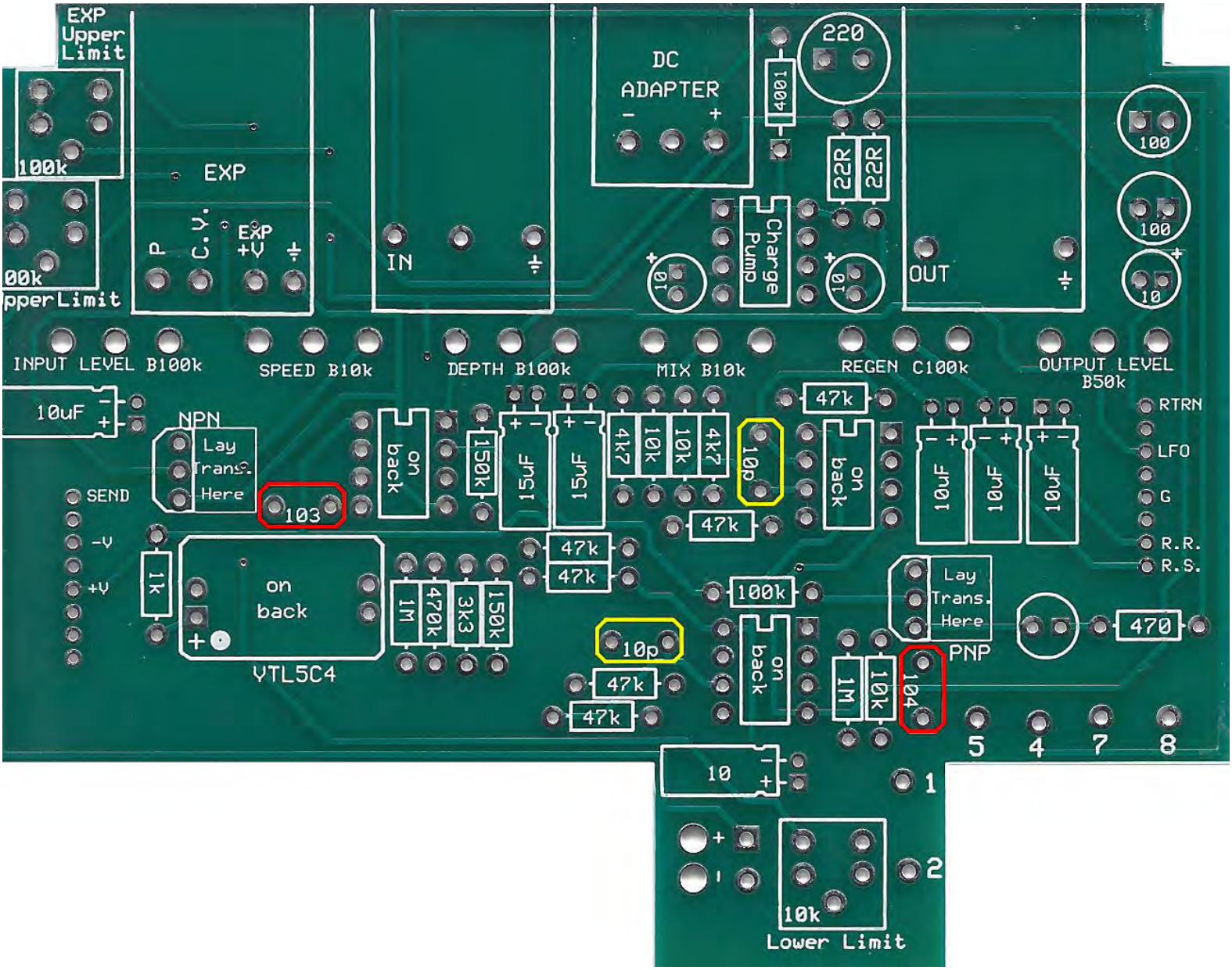
Step 6: Add the optocoupler. **Be sure to orient it correctly, and mount it on the underside of the PCB**. The longer lead will go into the square hole. There will also be a painted dot on the body of the optocoupler to indicate the LED Anode, which goes into the square hole.



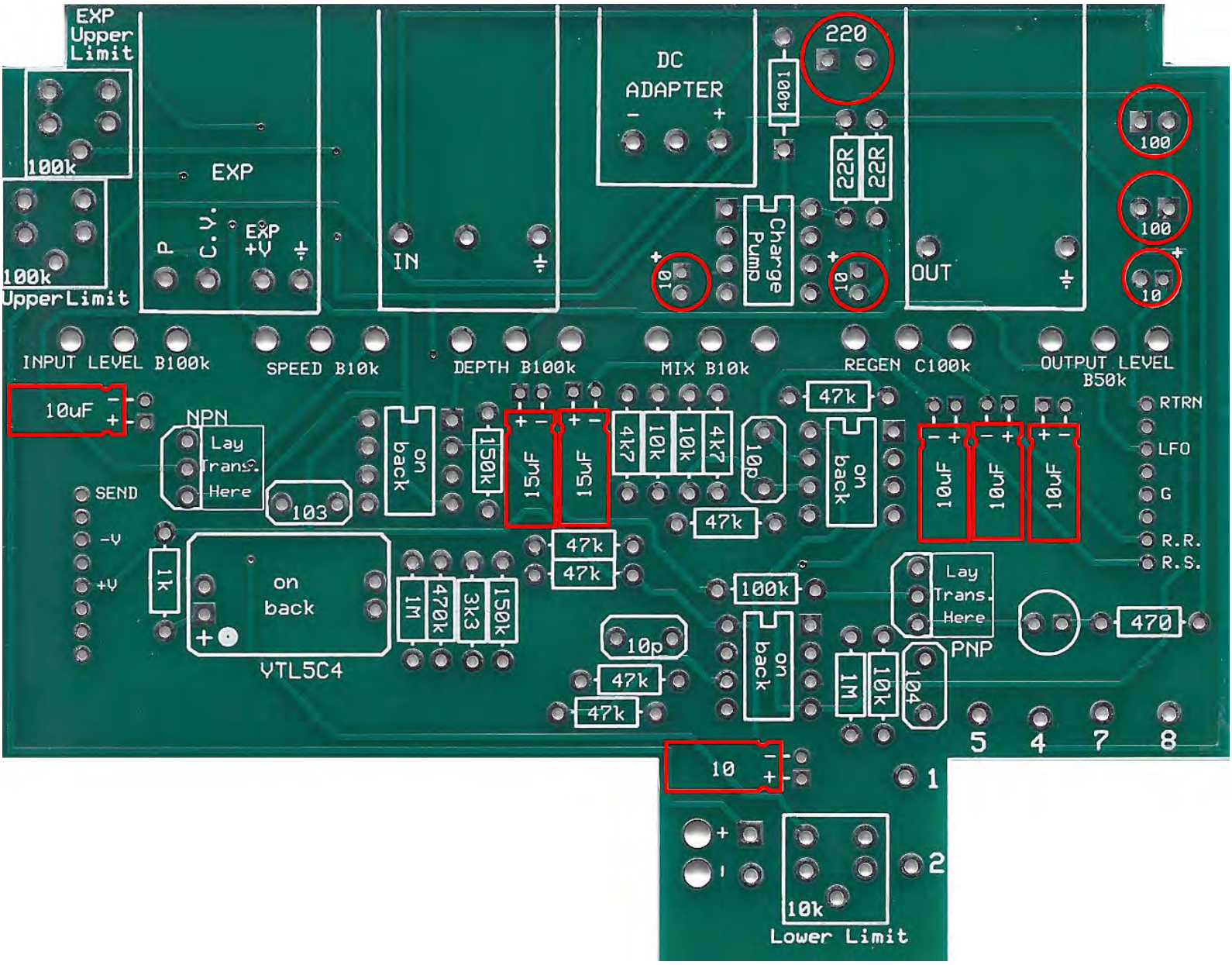
Step 7: Add the transistors. Be sure to match the flat side of the transistors with the flat side on PCB layout. Allow enough room between the PCB and transistor bodies to lay them flat against the screenprinted area. The transistor highlighted in **RED** is the 2N3904. The transistor highlighted in **YELLOW** is the 2N3906.



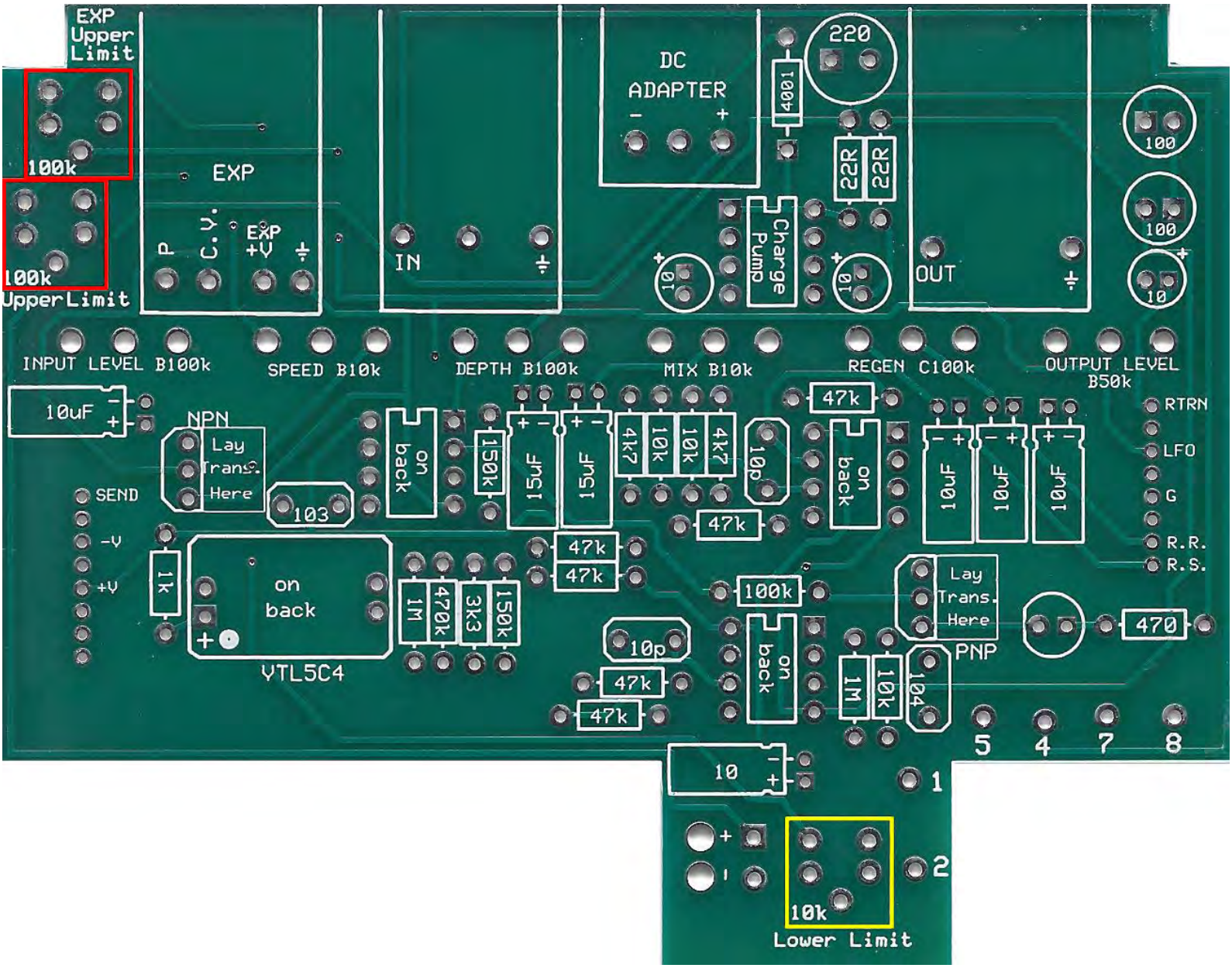
Step 8: Add the film and ceramic disc capacitors. These are non-polarized so they can go in either direction. The ceramic disc capacitors are highlighted in **yellow (The PCB says '10p' but install 47pF ceramic caps instead)**, and are also non-polarized.



Step 9: 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. For the capacitors in the middle of the board, be sure to lay them along the screenprint outline on the PCB. Get them as close to the board as possible. This is required to allow the module boards to sit in the headers properly.

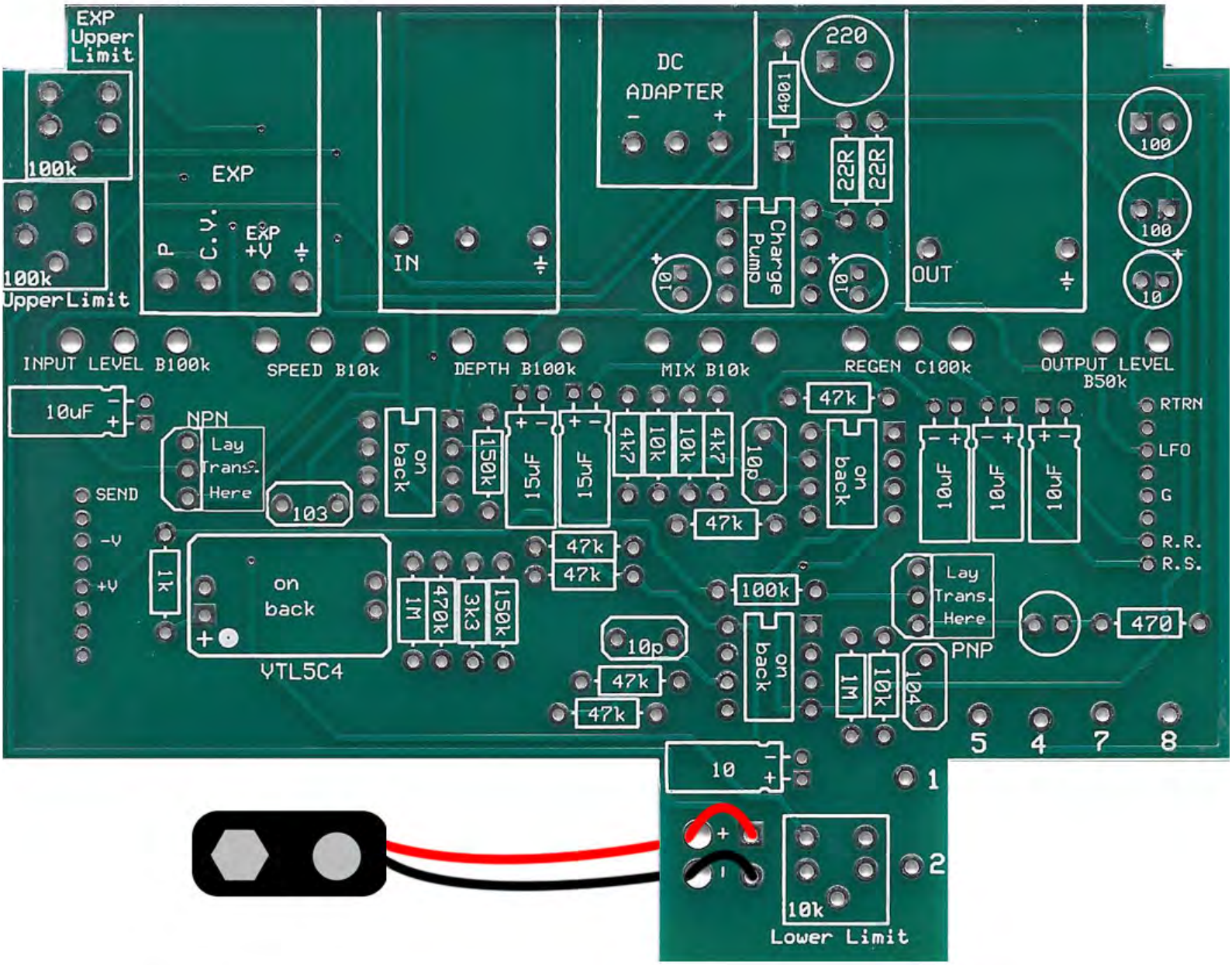


Step 10: Add the trimpots. There are 5 holes, but only three legs on the trimpots. This is normal. Your trimpot will only fit into three holes; the additional holes are to accommodate various sizes of trimpots. NOTE: The 'UPPER LIMIT' trimpots are both 250k, and the 'LOWER LIMIT' trimpot highlighted in yellow is 25k.

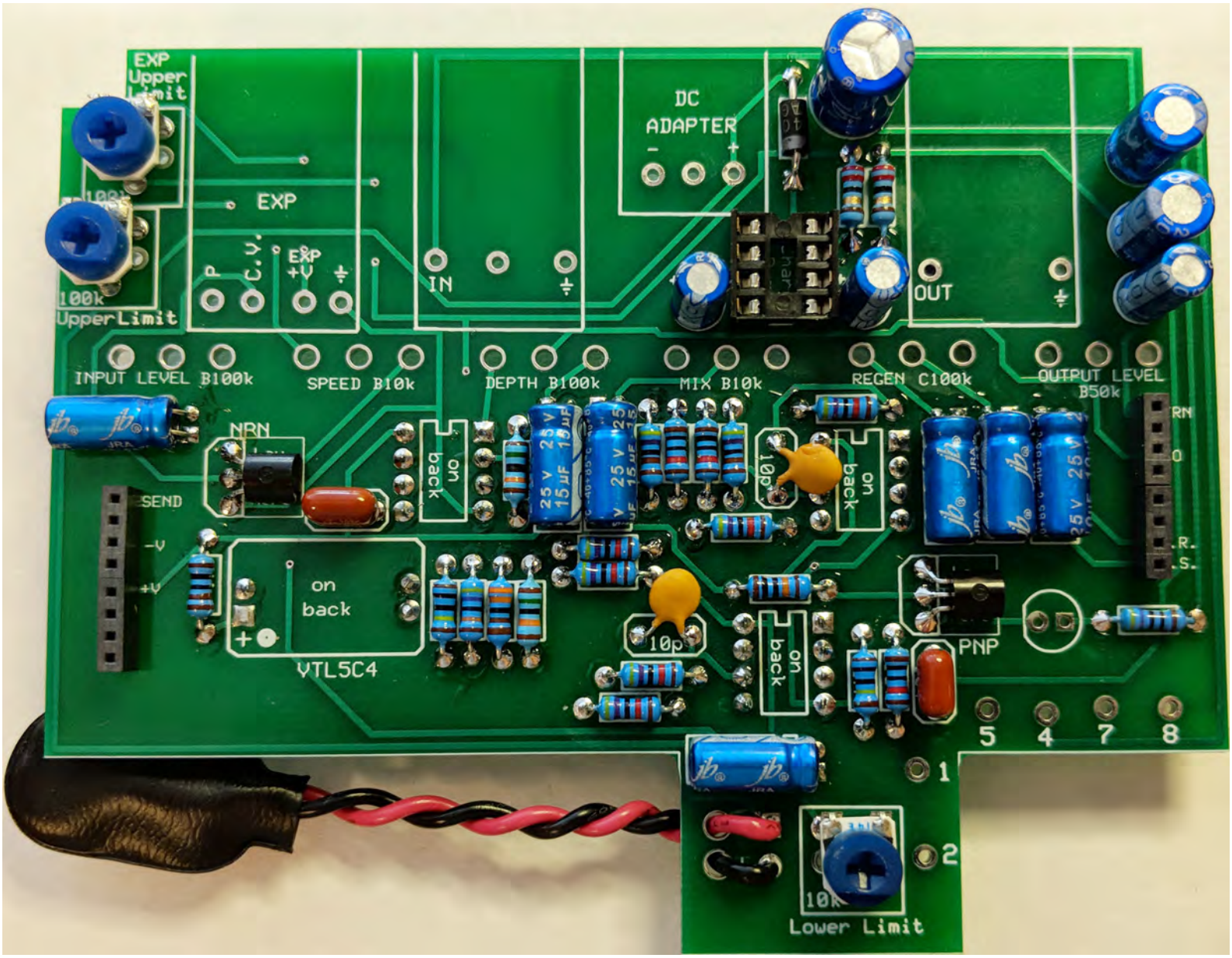


Setting the trimpots: When you are done with your build, and ready to use the pedal, you will use these to control the slowest and fastest speeds. Use your ear and your personal preference. The EXP UPPER LIMIT sets the fastest speed available when using the expression pedal input. The UPPER LIMIT and LOWER LIMIT control the fast and slow settings of the SPEED pot on the pedal itself, respectively. There is no EXP LOWER LIMIT because most expression pedals already have a lower limit control, sometimes called 'minimum' or similar.

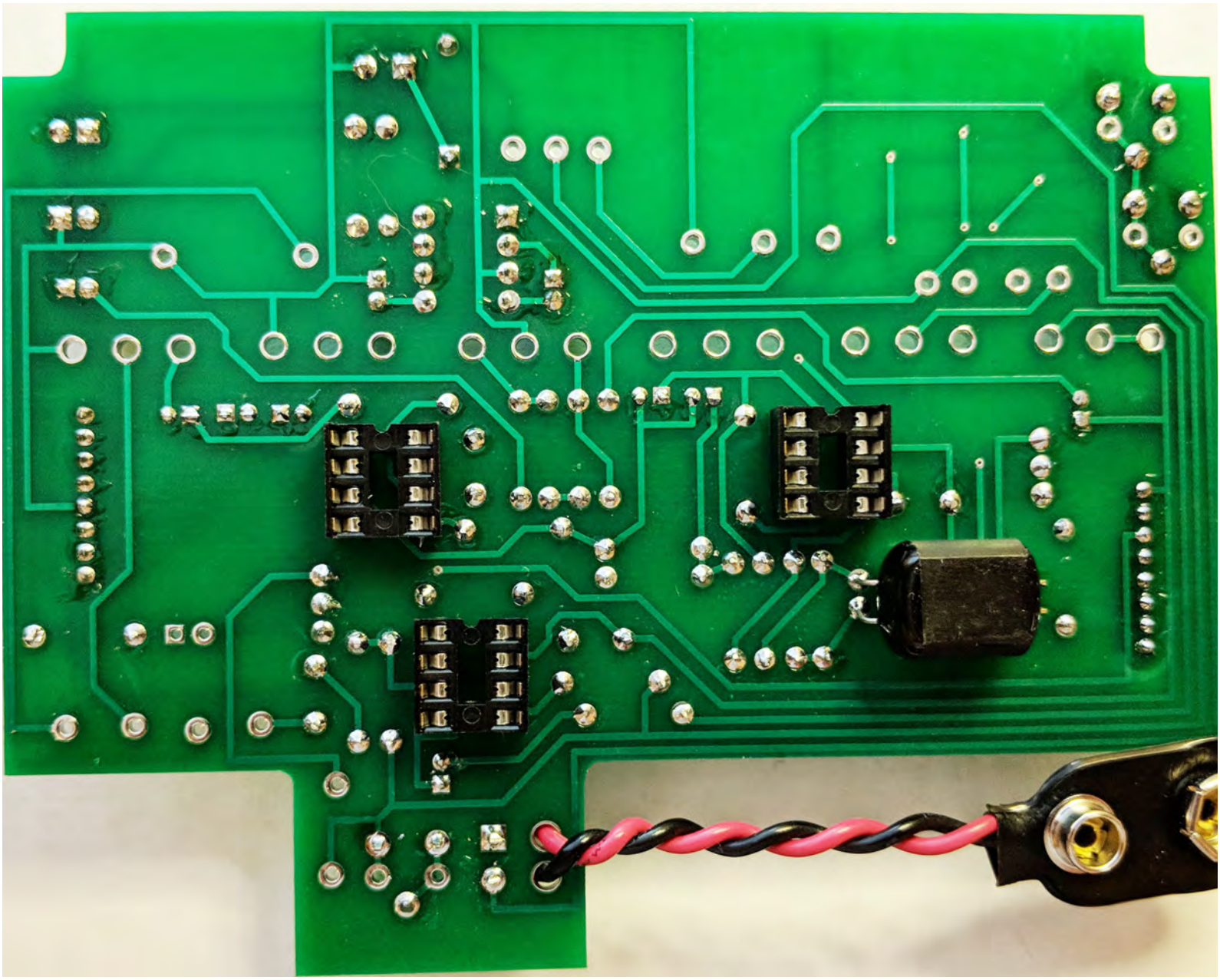
Step 11: Add the battery snap. Thread the wires into the strain relief holes first through the bottom side of the PCB. Then insert the leads into their respective solder pad hole on the top-side of the PCB. The red lead goes in the + hole and the black lead goes in the – hole.



At this point, your build will look like this:



Front

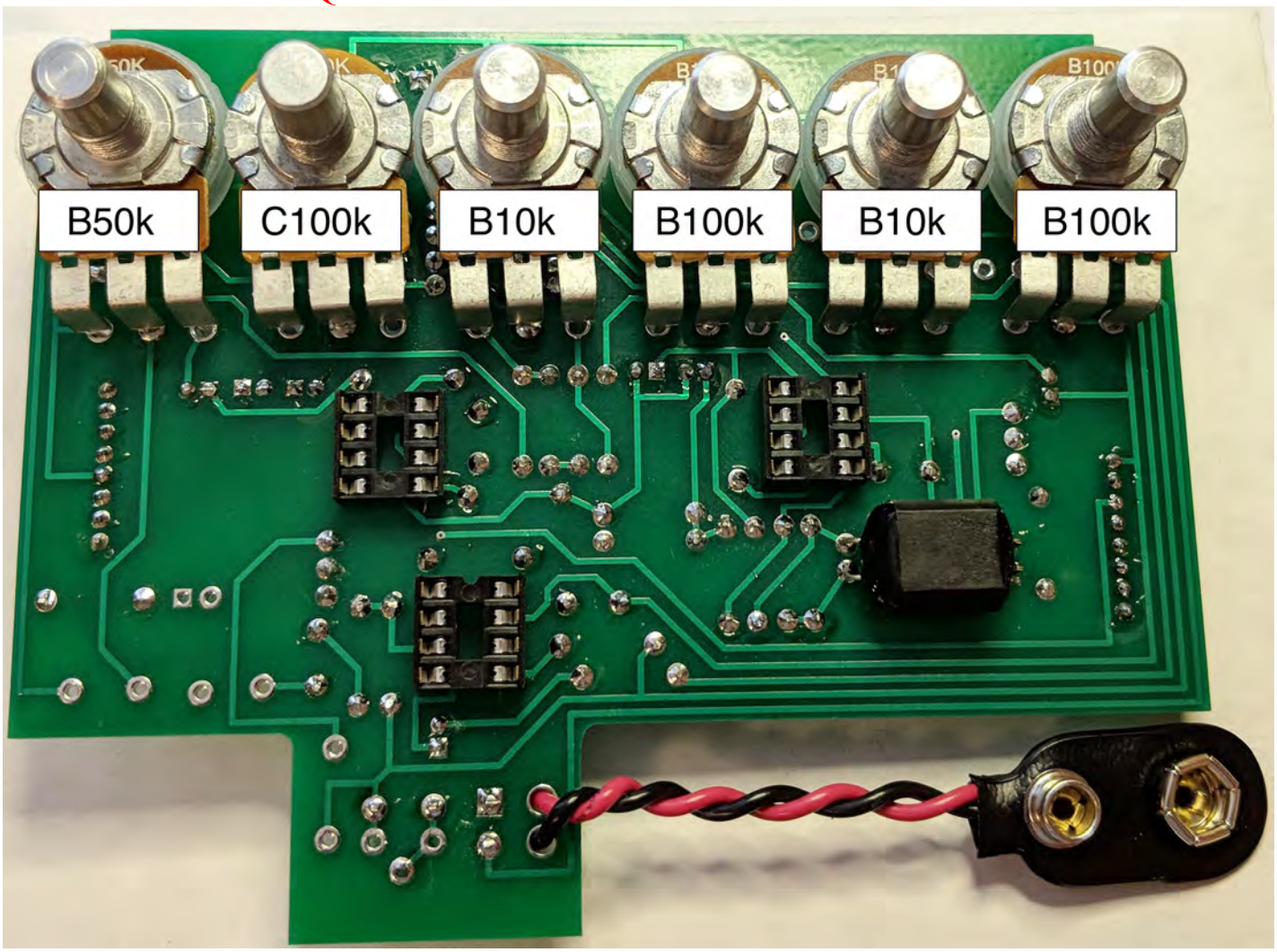


Back

Main PCB Assembly

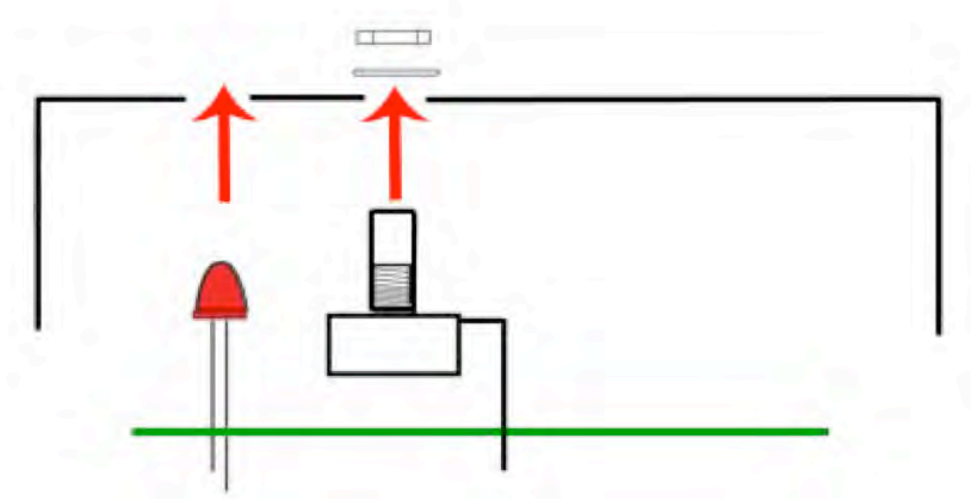
Step 1: Flip the PCB over so that the bottom or solder side is up. Insert the potentiometers, toggle switches, and the LEDs into the bottom side of the PCB. DO NOT SOLDER ANYTHING YET!!! (See below for advanced tip)

The LEDs will have one lead that is longer than the other. THIS WILL GO INTO THE SQUARE SOLDER HOLE.



When placing the potentiometers for enclosure installment, it sometimes helps to solder the middle lug of the pots, allowing about a 1mm gap between the PCB and thick part of the potentiometer lug. This allows rigidity of the pots, as well as giving a little “wiggle room”. After you have your board installed, hand tighten the potentiometers and switches, and **carefully** reflow the middle lugs of the pots to allow the board to settle. After the board has settled, you can solder the rest of the pot and switch lugs.

Remember to install the opamps before the next step!

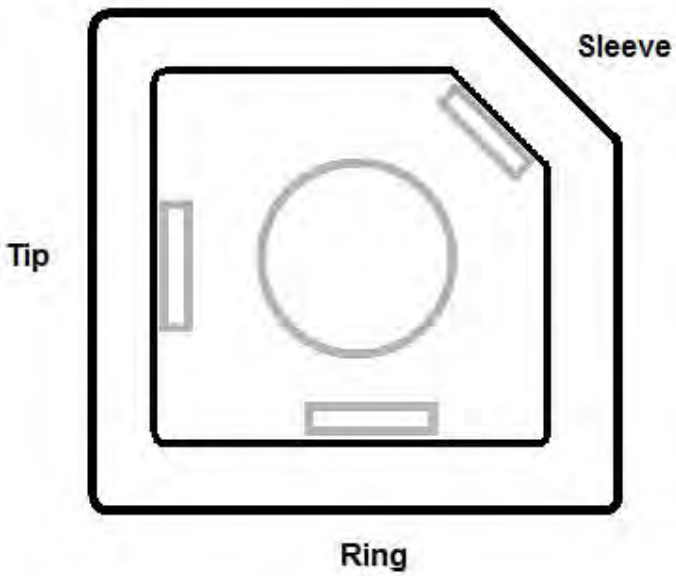


Step 2: Hold the PCB in one hand so that the component side of the PCB is in the palm of your hand and the bottom side with the pots, toggle switch and LED is facing up. Now use your other hand to guide the predrilled enclosure onto the PCB assembly so that the pots, toggle switch and LED all go into 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 3: 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.

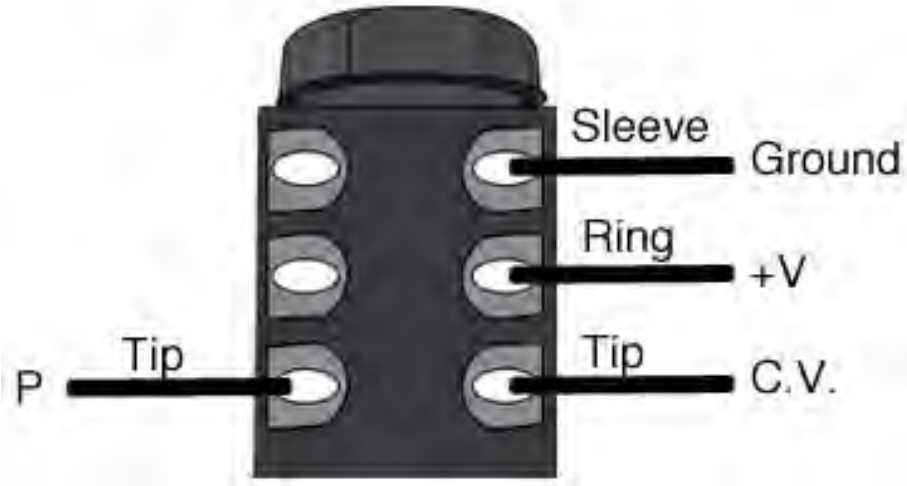
Step 4: Solder the pots, toggle switch and LEDs. You will be soldering on the component side (top) of the PCB. After you have soldered them in place, be sure to tighten up their nuts. TIP: only solder one lug of each component at first. This will secure everything in place and still allow you to wiggle things around if you need to adjust the fit of anything. Once you have everything perfect, go ahead and solder everything else.

Enclosed Jack

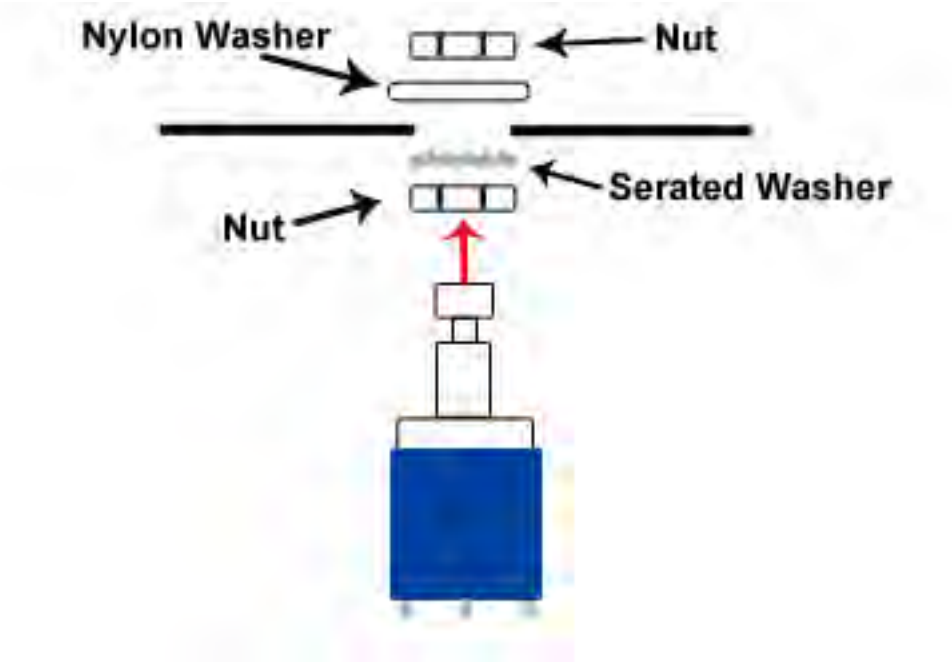


You will want to place the jacks into the enclosure so the sleeve terminal is facing the right like the picture above. Be sure to remember the lock washers so the jacks don't spin on their own.

Switching Jack

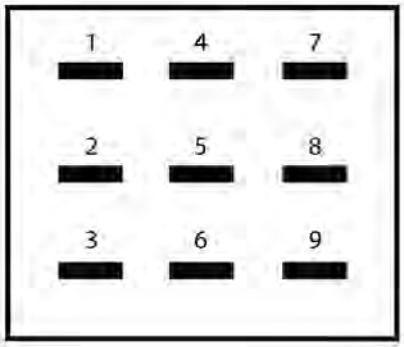


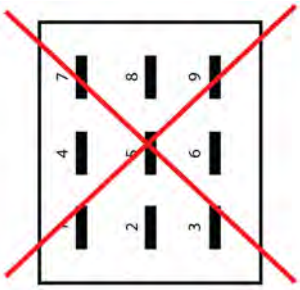
Install the switching jack in the enclosure with the lugs facing up, or towards the back plate of the enclosure.



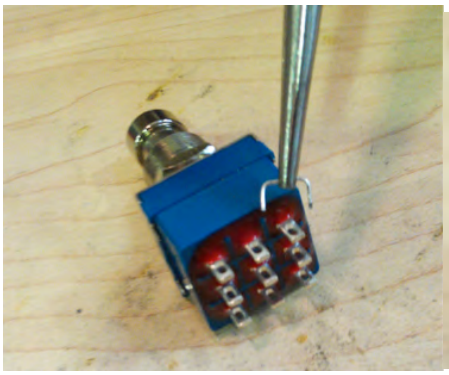
Step 5: Install the footswitch. Orient the footswitch so that the flat sides of the solder lugs are like the diagram below.
NOTE: There are no actual number markings on the footswitch. There are two correct ways you can orient the footswitch. They are both 180 degrees of each other. Either way is fine. It does not matter as long as the flat sides of the solder lugs are running horizontal, not vertical.

FOOT SWITCH SOLDER LUG DESIGNATIONS

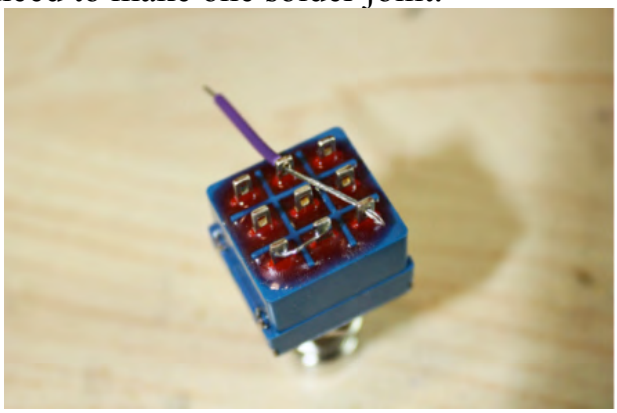




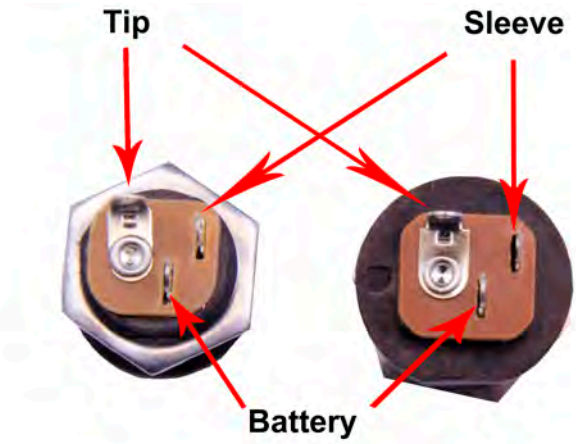
Step 5a: Make a jumper between lugs 3 & 6 from clippings from the resistors. Simply use your needle nose pliers to make a U shape & insert into lugs 3 & 6, then solder.



Step 5b: Connect a wire to LUG 4 that also jumpers to LUG9. Strip about 1" off one end. Make sure there is enough insulated wire to make the connection to the TIP of the in jack. Carefully tin the stripped end. You may want to twist the wire strands together tightly before tinning. Thread the stripped end through LUGs 4 and 9. This can be a little tricky. If this part is too frustrating for you, you can just run a separate wire to connect LUGs 4 and 9. Just be sure to solder the two wires at LUG 4 at the same time so you only need to make one solder joint.



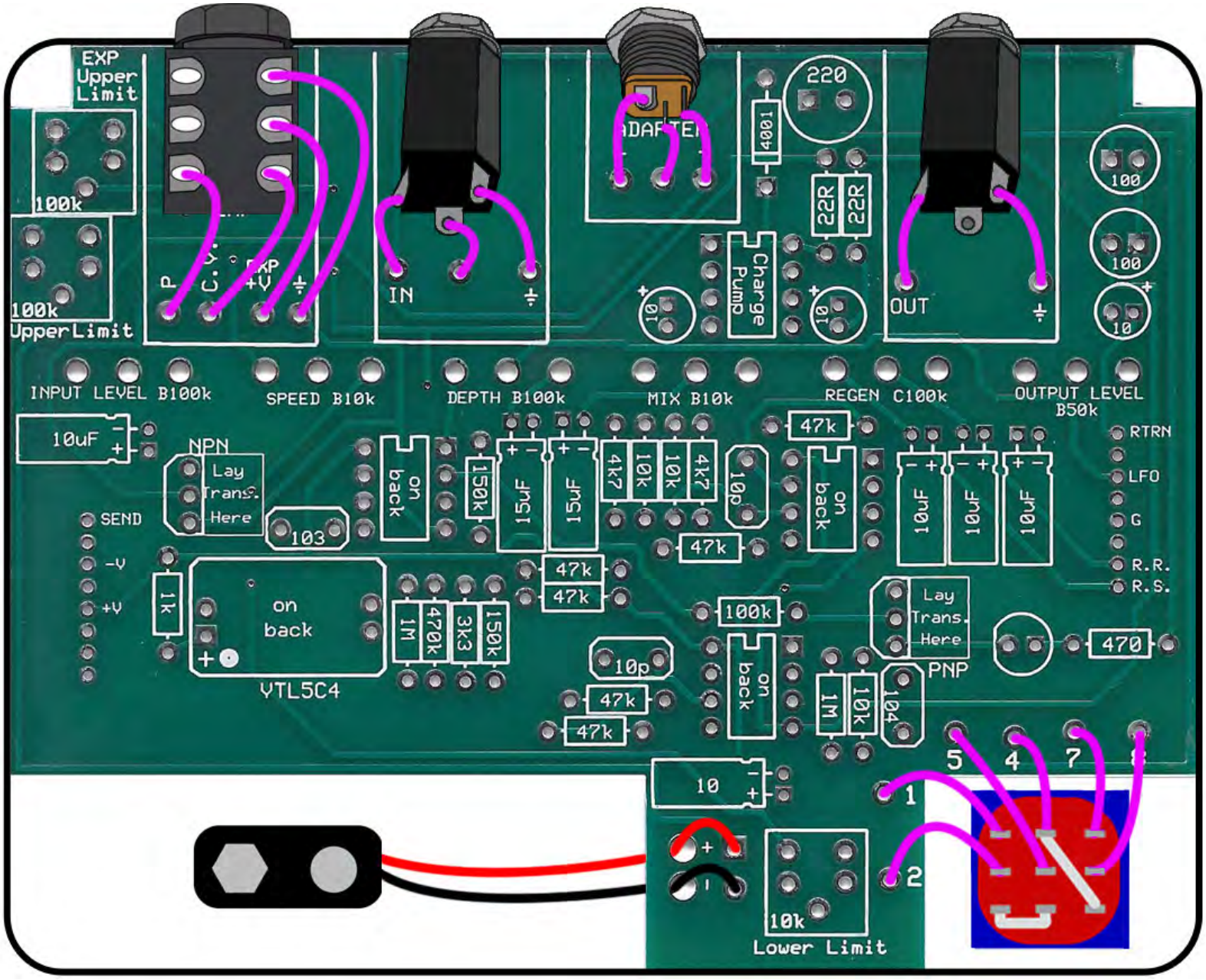
WIRING



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 lug to the eyelet in-between the "-" and "+" eyelets.

Also connect the enclosed audio jacks and the switching jack to the PCB. The SLEEVE connection on the enclosed jacks will go into the ground spots, and the TIP connection will connect to the 'in' and 'out' eyelets.

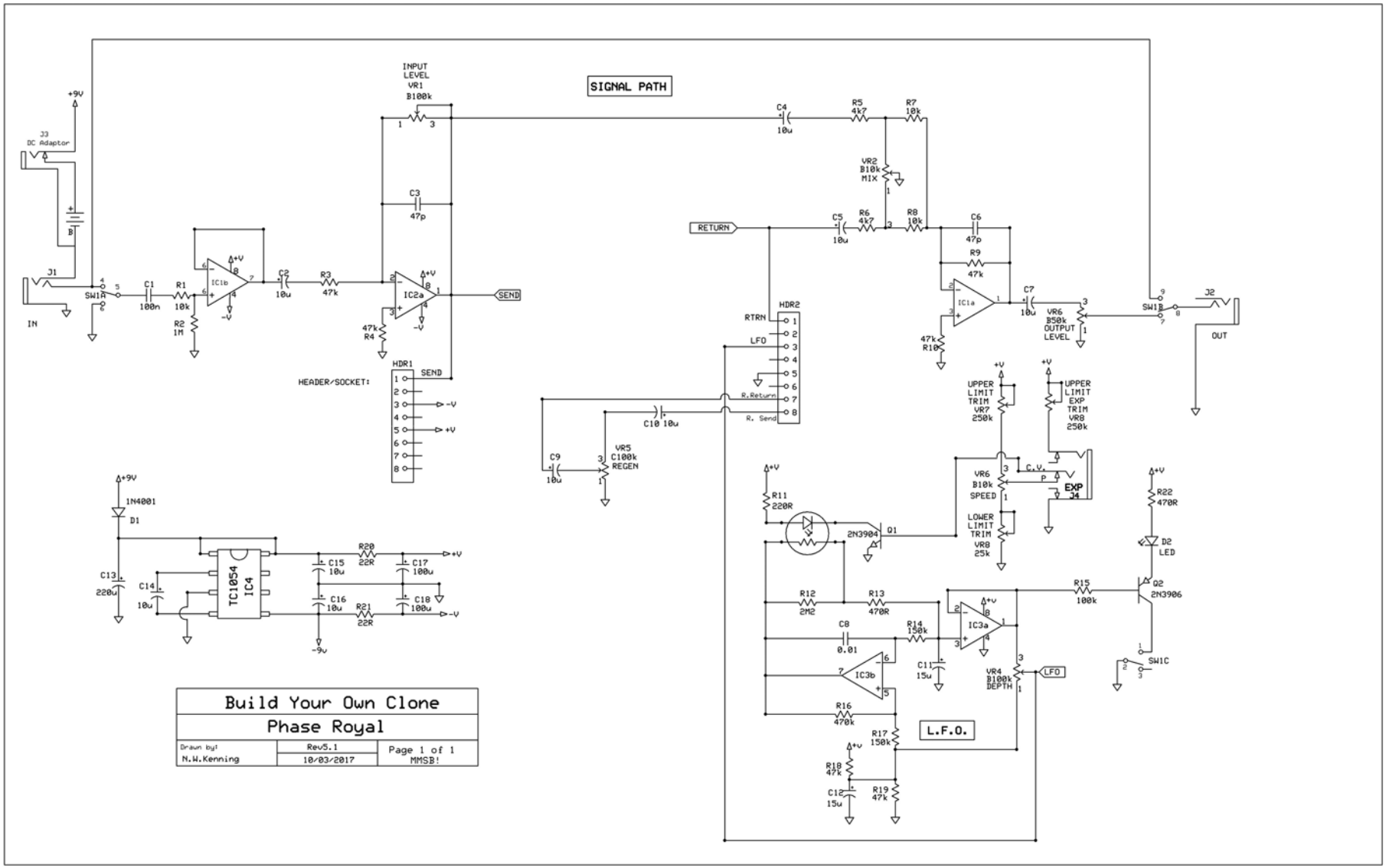
The SLEEVE connection of the switching jack will go to the GROUND eyelet, the RING connection of the switching jack will go to the +V eyelet, the right TIP connection of the switching jack will go into the C.V. eyelet, and the left TIP connection will go into the P eyelet. All of this is shown below.



Finished Wiring for Main board

Operating Overview

OUTPUT: Controls the overall output level.
REGEN: Controls the amount of feedback in the phase circuit.
MIX: Blends between the dry and wet signals -- CW = fully wet, CCW = Fully dry.
DEPTH: Controls the rise and fall depth of the LFO.
RATE: controls the speed of the LFO signal.
INPUT: Controls the input signal level.
IN: Guitar input.
OUT: Output jack to amplifier.
EXP: Expression pedal input.
Power supply: 9v 2.1mm negative tip or 9v battery
Current Draw: 80mA
Input Impedance: 1M Ohm



Build Your Own Clone		
Phase Royal		
Drawn by: N.W.Kenning	Rev5.1 10/03/2017	Page 1 of 1 MMSB!

For hi-res schematic visit:
http://byocelectronics.com/phaseroyal2schematic.pdf

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