Build Your Own Clone
Soaring Skillet
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.buildyourownclone.com/board

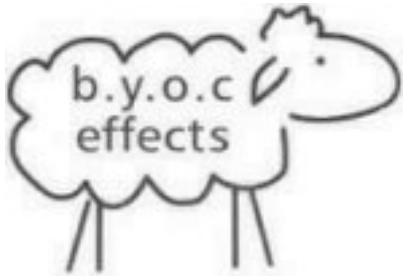
When posting a tech support thread on the BYOC forum, please post it in the correct lounge, and please title your thread appropriately. If everyone titles their threads "HELP!" then it makes it impossible for the people who are helping you to keep track of your progress. A very brief description of your 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.

Revision Notes: Rev1.0 – 100k pan bias trimpots are mislabeled. PCB is labeled "1M". 100k trimpots should go in these spaces. 220n feedback cap space is mislabeled. PCB is labeled 22n. 220n cap should go in this space.

Copyrights: All material in this document is copyrighted 2012 by BYOC, Inc.



SOARING SKILLET KIT
INSTRUCTION INDEX

Parts Checklist…….....…….....page 5 - 7

Populating the Circuit Board…….....page 8 – 24

Main PCB Assembly…….....page 25 - 30

Wiring.....page 31 - 37

Installing the IC's and Finishing Up.....page 38

Operation Overview.....page 39

Schematic.....page 40

Completed Build

BYOC Soaring Skillet
Parts Checklist

Resistors:
1 – 10Ω (brown/black/black/gold/brown)
1 – 1k (brown/black/black/brown/brown)
1 – 2k2 (red/red/black/brown/brown)
4 - 4k7 (yellow/purple/black/brown/brown)
2 – 6k2 (blue/red/black/brown/brown)
1 – 6k8 (blue/gray/black/brown/brown)
21 – 10k (brown/black/black/red/brown)
1 – 18k (brown/grey/black/red/brown)
1 – 24k (red/yellow/black/red/brown)
5 – 30k (orange/black/black/red/brown)
3 – 47k (yellow/purple/black/red/brown)
2 – 56k (green/blue/black/red/brown)
8 – 100k (brown/black/black/orange/brown)
1 – 120k (brown/red/black/orange/brown)
5 – 150k (brown/green/black/orange/brown)
3 – 220k (red/red/black/orange/brown)
1 – 270k (red/purple/black/orange/brown)
1 – 390k (orange/white/black/orange/brown)
7 – 470k (yellow/purple/black/orange/brown)
4 – 1M (brown/black/black/yellow/brown)
1 – 2M2 (red/red/black/yellow/brown)
go to www.buildyourownclone.com/resistorcodes for more info on resistors and how to determine resistor values.

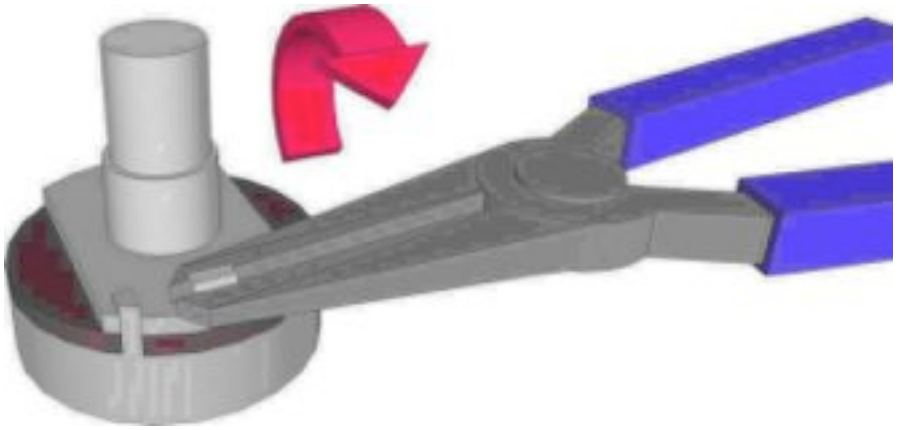
Capacitors:
1 – 3n3 or 0.0033µ film (332)
6 - 10n or 0.01µ film (103)
10 – 47n or .047µ film (473)
1 – 220n or .22µ film (224)
1 - 1µ film (105)
2 - 15µf aluminum electrolytic
2 - 33µf aluminum electrolytic
1 - 100µf aluminum electrolytic
go to www.buildyourownclone.com/capcodes for more info on capacitors and how to determine capacitor values.

Diodes:
1 - 1N4001 (larger black plastic with silver stripe)
1 – 5.1v Zener (small orange glass with black stripe)

IC's:
5 – TL022
5 - 8 pin sockets
2 – Dual element Opto Couplers

Transistors:
7 – 2N3904
2 – 2N5952

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



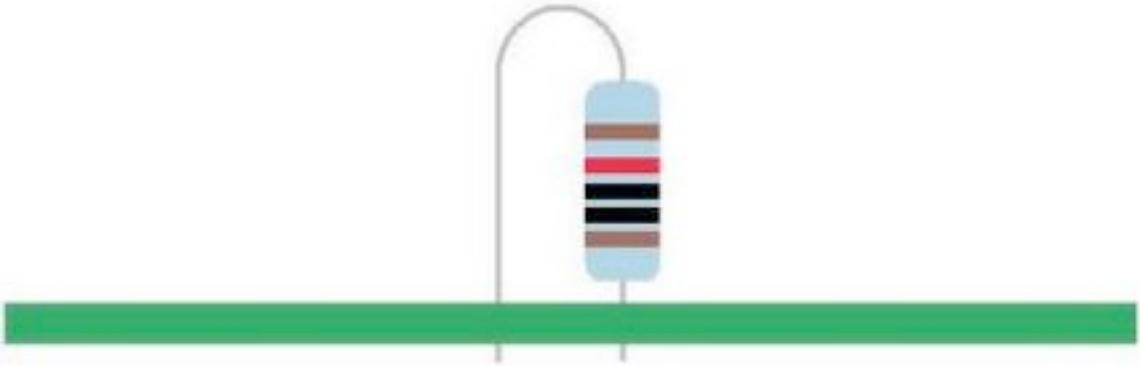
1 – C500k reverse audio (PHASE RATE)
1 – C10k reverse audio (FEEDBACK)
1 – C1M reverse audio (PAN RATE)
1 – 1M trim pot (will be labeled ‘105’)
3 – 100k trim pot (will be labeled ‘104’)

Hardware:
1 - drilled enclosure w/ 4 screws
1 - BYOC Soaring Skillet PCB
2 - 3PDT footswitch
1 – 4 pole, 3 position rotary switch
4 - knobs
1 - DC adaptor jack
3 - ¼”mono jack
3 – lock washers
4 - bumpers
hook-up wire
1 – red LED
1 – green LED

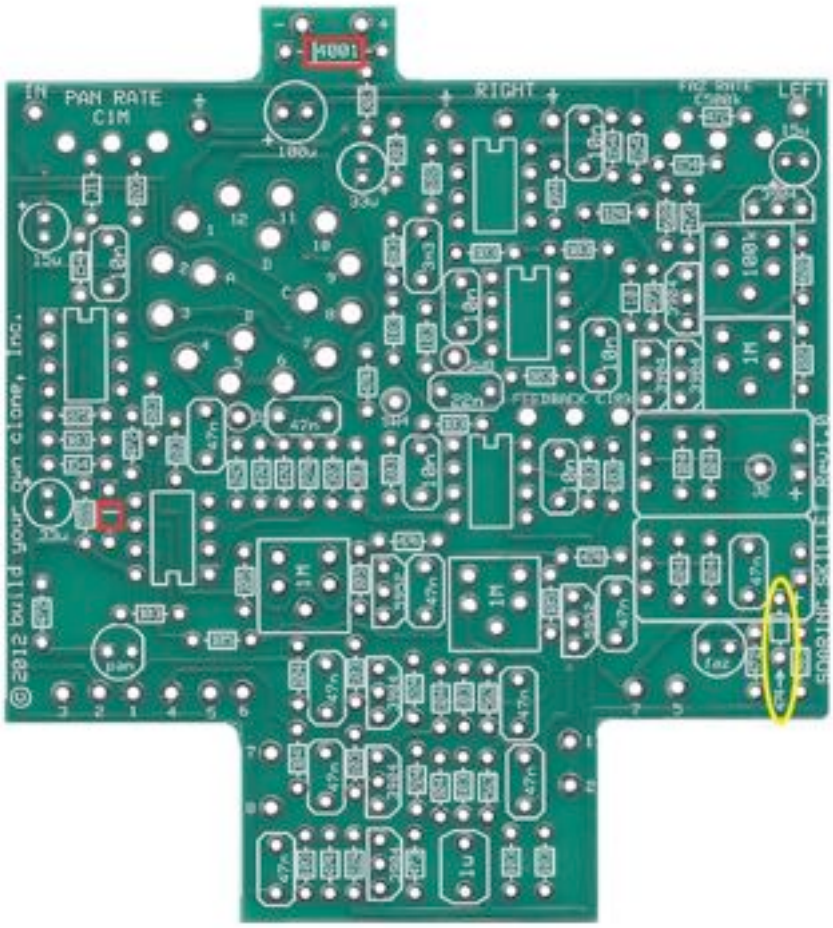
IMPORTANT BUILDING NOTES

Please make sure to follow these instructions step by step. There are components that must be soldered into place before the opto couplers are to be mounted onto the PCB. There will not be access to these afterward, so avoid unnecessary de-soldering by following these instructions carefully.

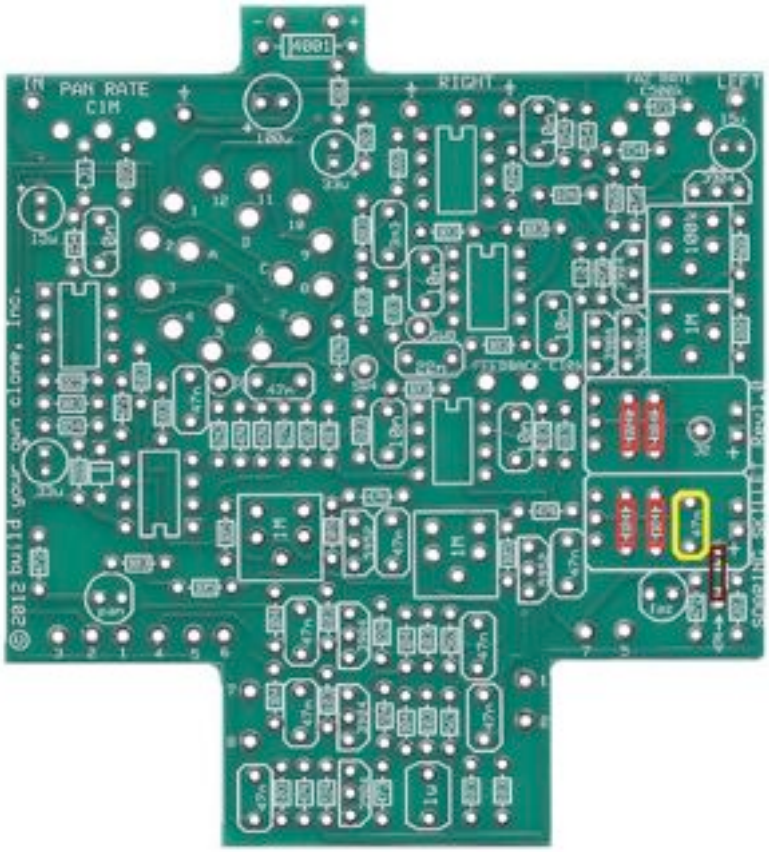
Note about the resistors: Most of the resistors in your kit will be 1/8 watt (small resistors), but your kit may contain some ¼ watt resistors (regular sized resistors). The resistor spaces on your Soaring Skillet PCB will be for 1/8 watt resistors. If your kit contains any ¼ watt resistors, you should bend the lead on one side (either side. Resistors are not polarized, so it does not matter) and stand them on end as shown in the diagram below.



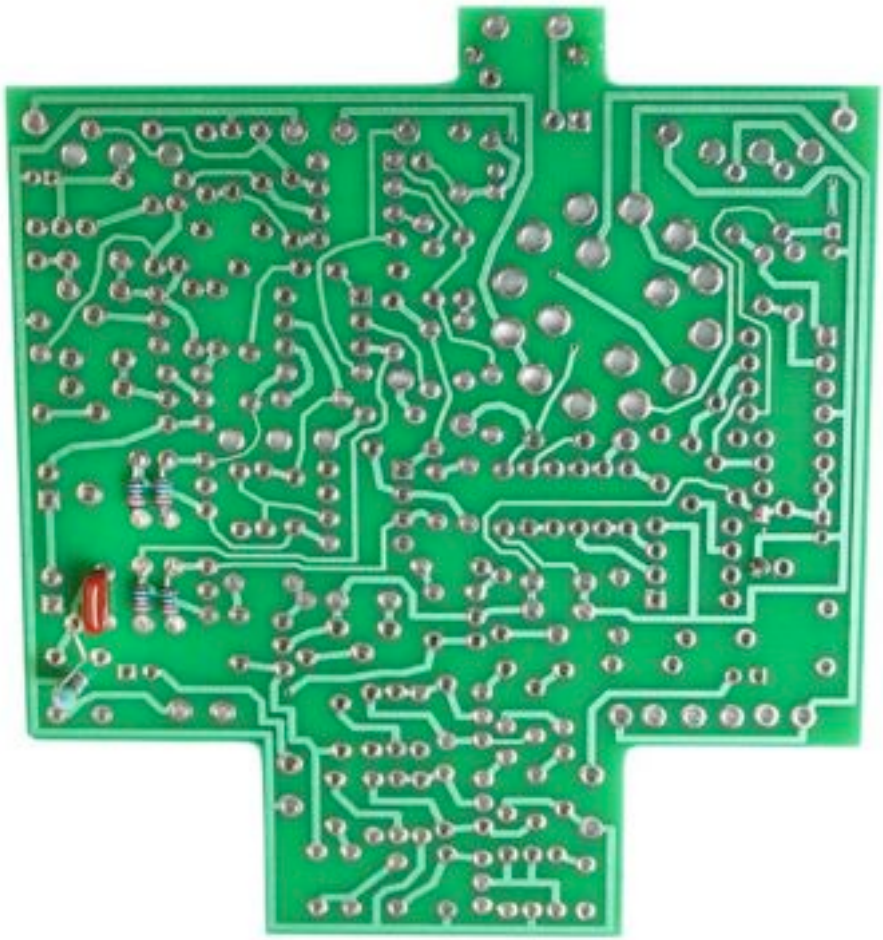
Populating the Circuit Board

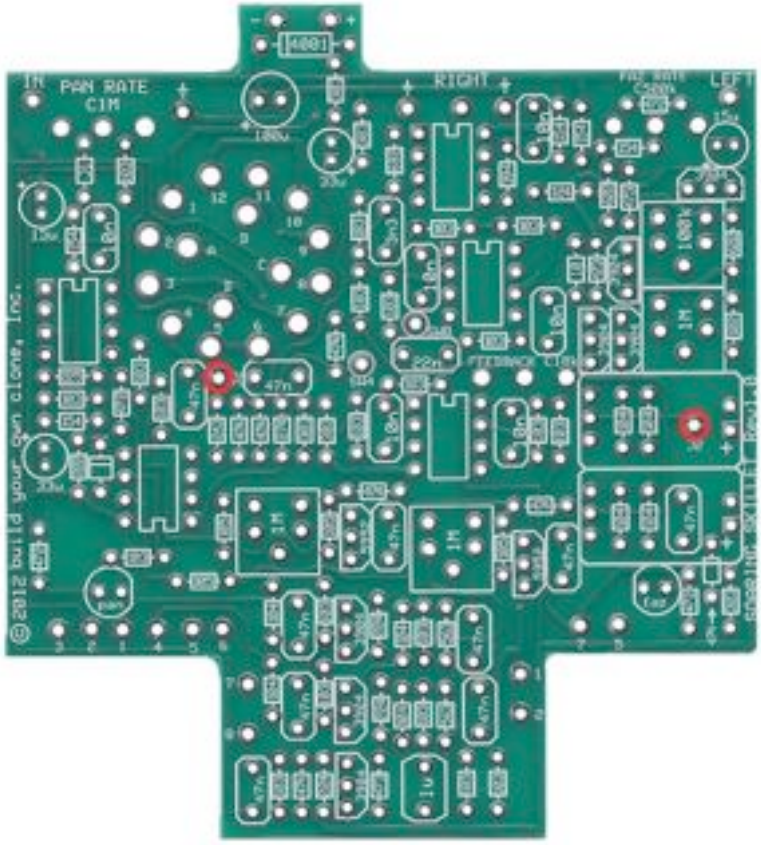


STEP 1: Add the diodes. Be sure to match the end of the diode with the stripe to the layout on the PCB. The stripped end should go in the square solder pad. NOTE: Make sure not to confuse the print on the PCB (circled in yellow) as a diode. It's a 470k resistor & gets mounted on the backside (see Step 2).

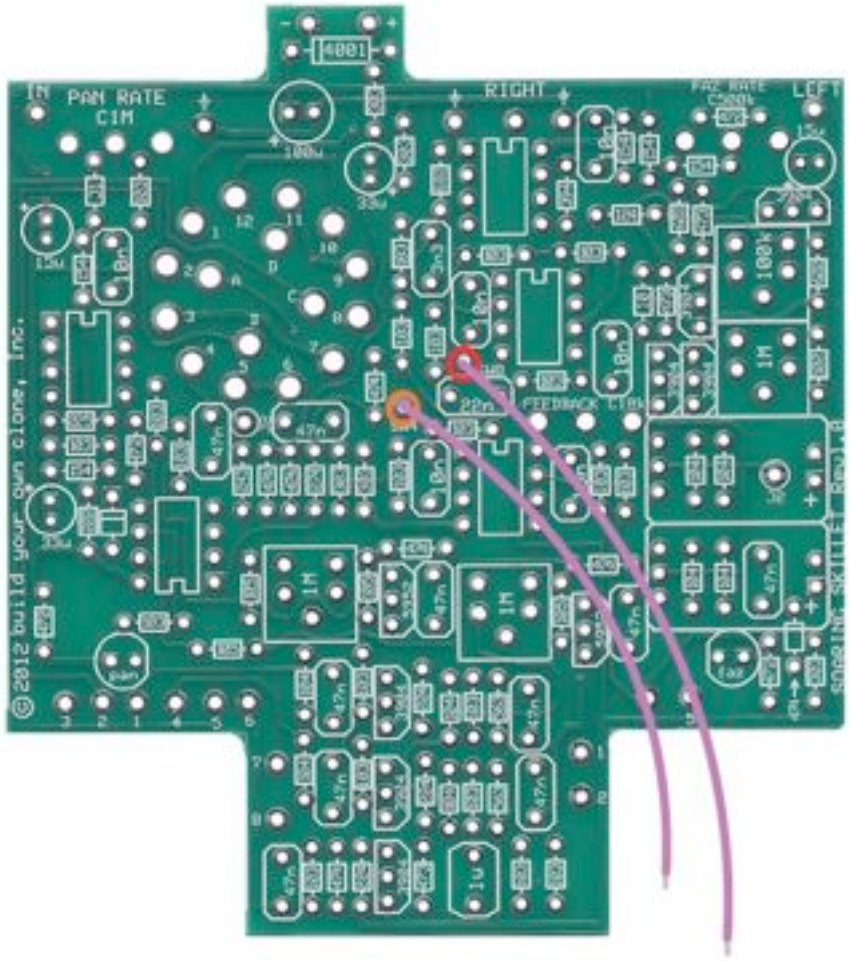


STEP 2a: Add four 100k (highlighted in red), one 470k (highlighted in brown) resistor and one 47n capacitor (highlighted in yellow) to the backside of the PCB. This is to allow the opto couplers to be mounted flat to the front side. See below for proper mounting.

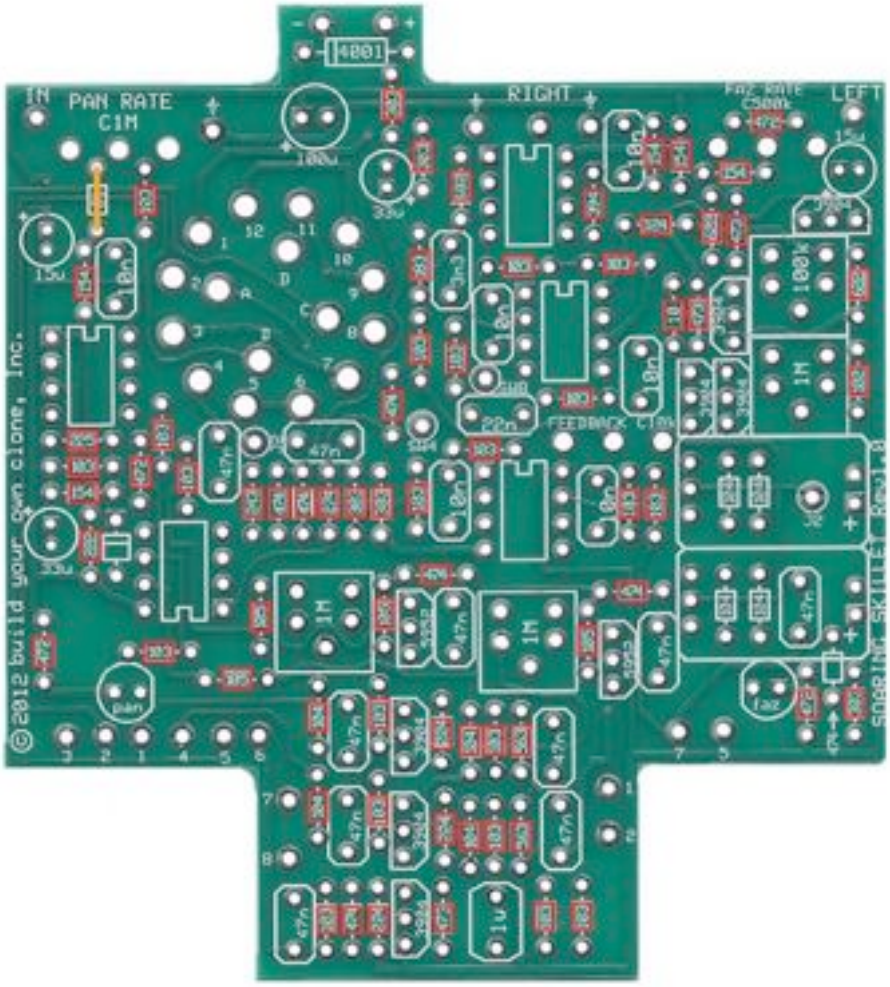




STEP 2b: Cut a 2.25” piece of wire & strip 1/8” off the tips and tin the ends. Tinning means to apply some solder to the stripped ends of the wires. This keeps the strands from fraying and primes the wire for soldering. Find the holes labeled ‘J2’ on the topside of the PCB (highlighted in red). Now, flip the PCB over and insert the tinned ends of the wire into the holes from the backside. Solder these in place on the topside of the PCB as you did the resistors & capacitor in Step2.

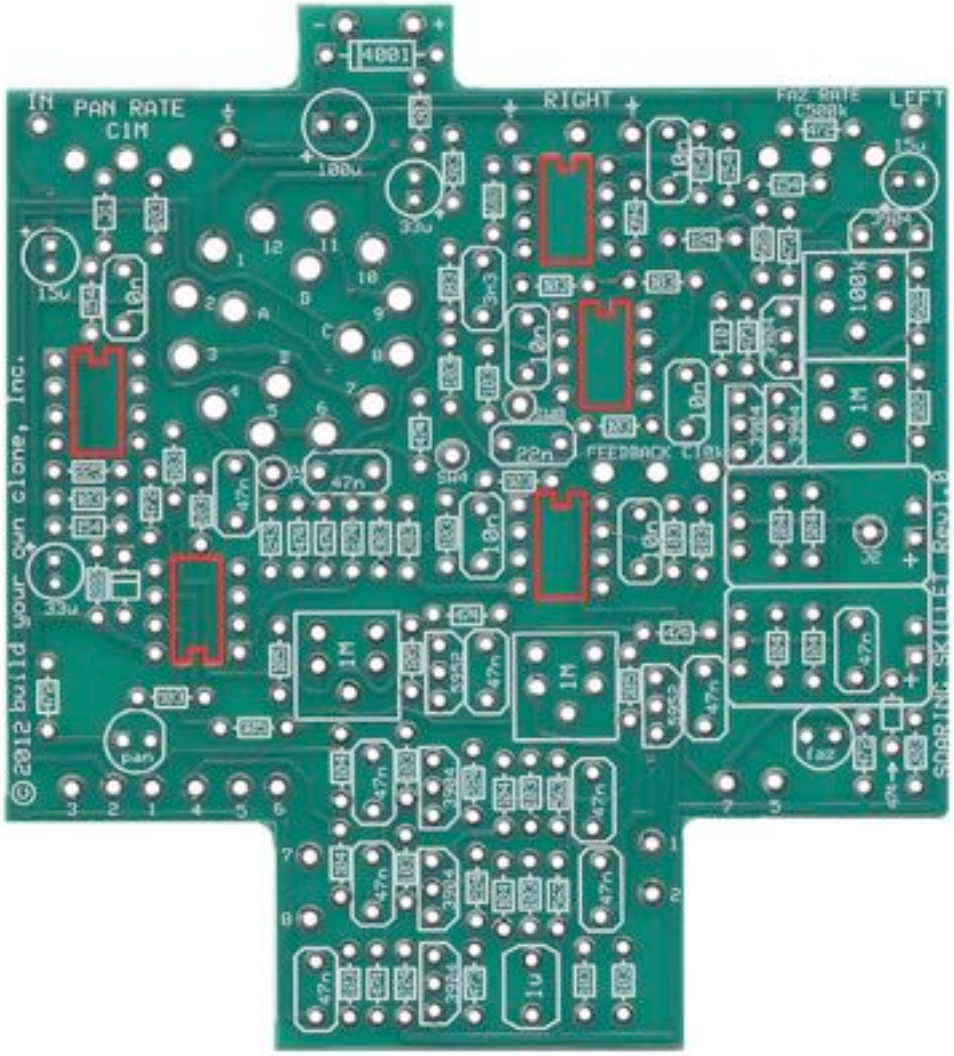


Cut a 3” piece & a 2.5” piece of wire. Strip an 1/8” off each end of both wires & tin them. On the topside of the PCB, find the hole labeled ‘SW8’ (highlighted in red). Take the 3” piece of wire & insert one end into the hole on the topside. Solder on the backside of the PCB. Then, find the hole labeled ‘SW4’ (highlighted in orange). Take the 2.5” wire & insert into hole. Once again, solder in place on the backside.



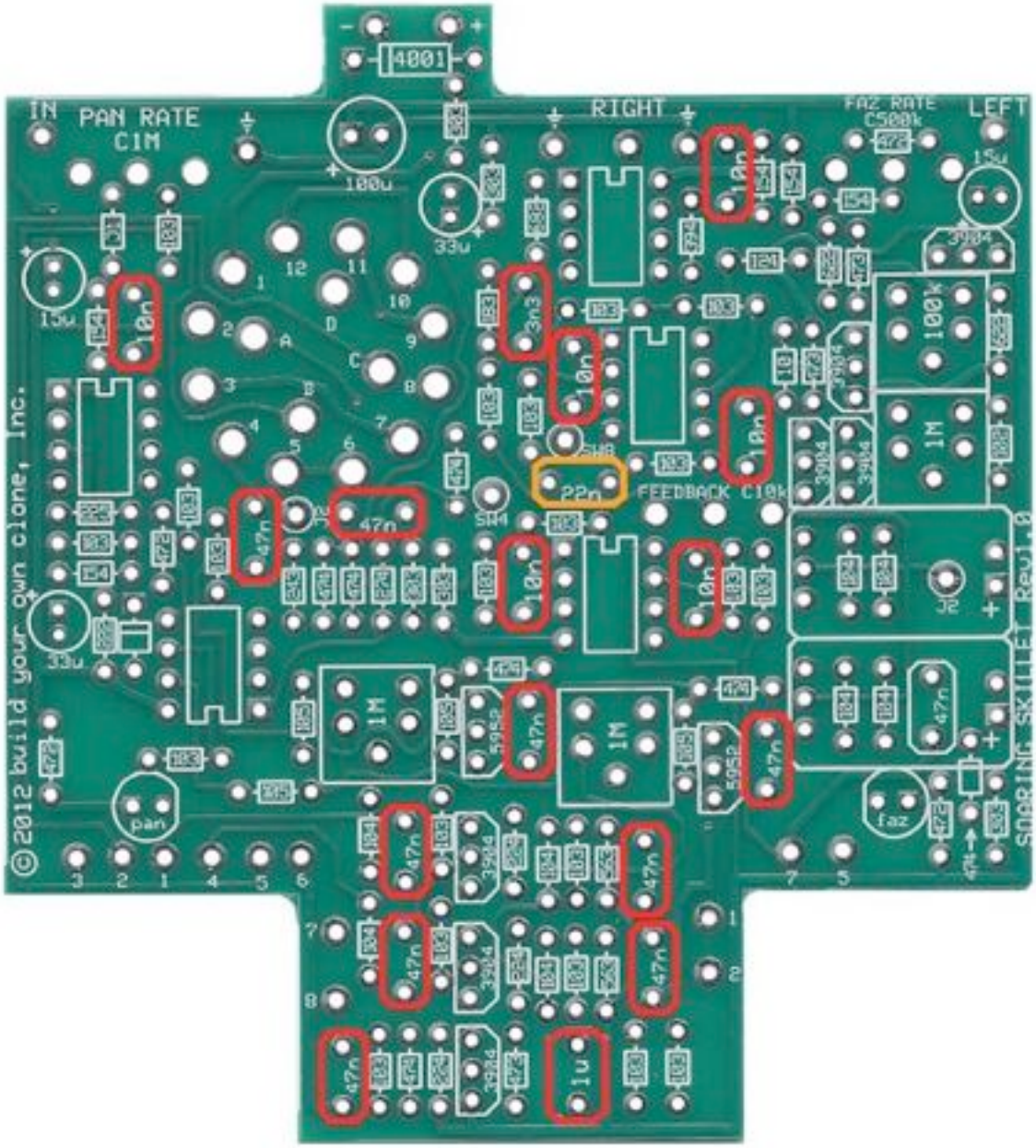
STEP 3: Add the resistors. Resistors are not polarized, so it does not matter which end goes in which solder pad. Install a jumper (highlighted in orange) using left over clipping.

NOTE: Some of the kits have the 1/4 watt resistors. Should your kit contain these, you will need to stand them on end when installing on the PCB. See the diagram on page 8.

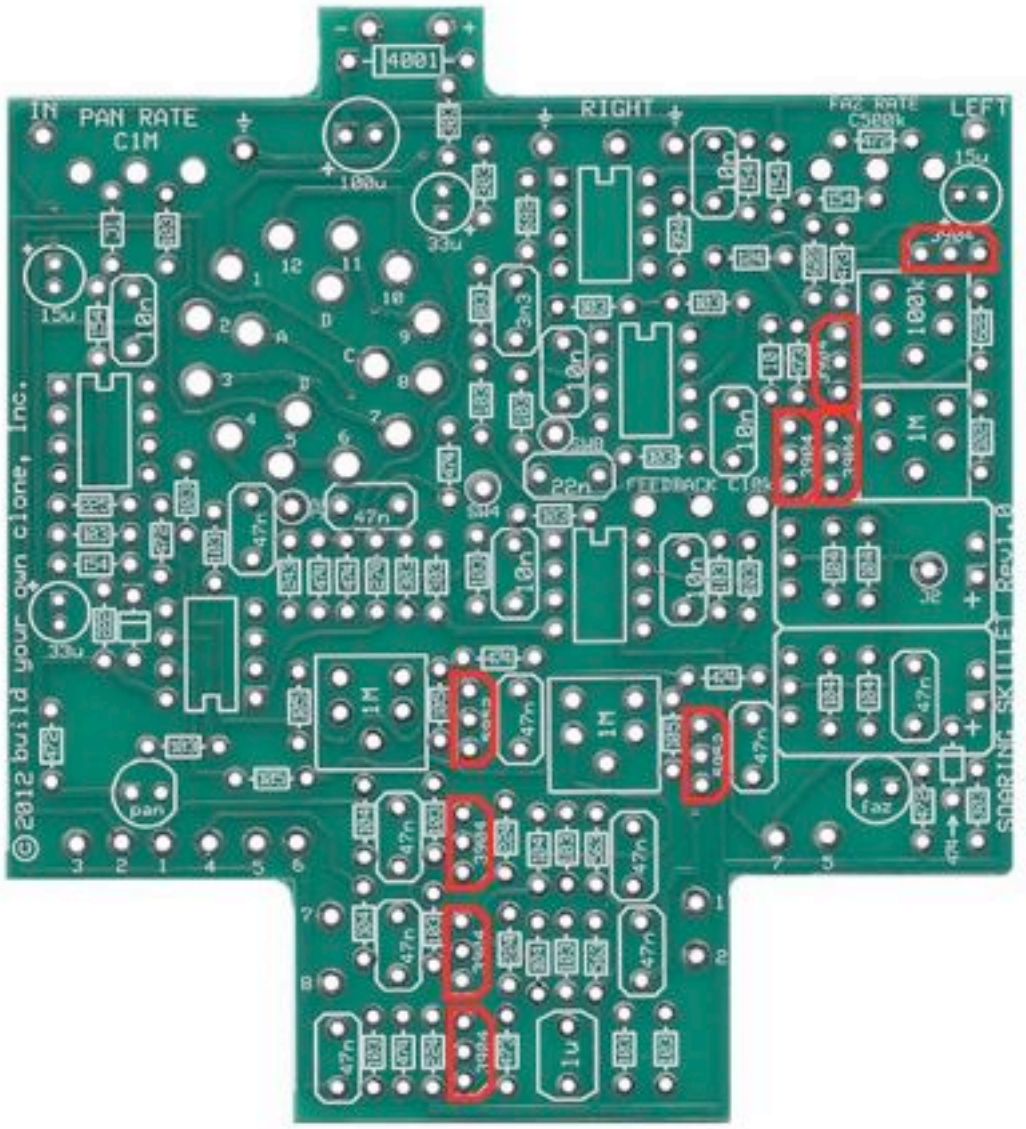


Step 4: Add the 8 pin IC socket. **ONLY SOLDER THE SOCKET! NOT THE ACTUAL IC!** This is a socket. The sockets get soldered to the PCB. The ICs get inserted into the sockets. The actual IC chip itself, never gets soldered. You will insert the IC into the socket after the entire pedal has been built. Orient the socket so that the side with the notch lines up with the layout on the PCB.

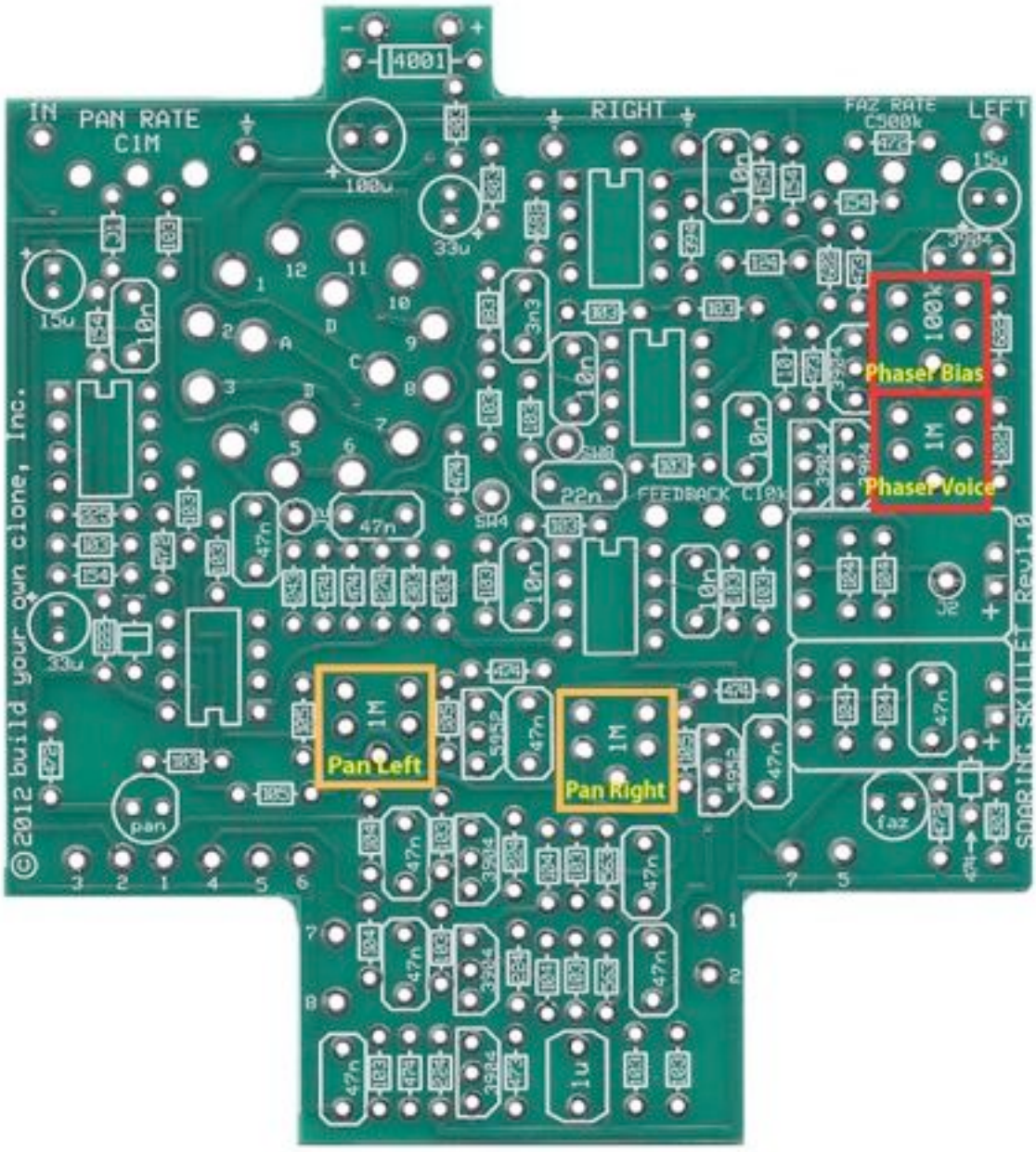
See page 28 for instructions on how to install the IC.



Step 5: Add the film capacitors. Film caps are not polarized and can be inserted into the PCB in either direction. If you have a Rev1.0 PCB, be sure to use a 220n in the 22n space highlighted in yellow.



STEP 6: Add the transistors. Insert these components into the PCB so that the flat side of the component matches up with the flat side of the PCB layout. Make sure to not mix up the 2N5952 and 2N3904. These are two very different types of transistors.



STEP 7: NOTE!!!! If you have a Rev1.0 PCB, use 100k trimpots for the pan left and pan right bias highlighted in orange. The PCB is labeled 1M. This is incorrect.

Add the internal trimpots. Many people get confused by these because the trimpot itself only has 3 legs, but the PCB has 5 holes. The PCB has 5 holes so that it can accommodate a variety of different trimpot brands and models. There should only be one way each trimpot will fit into the PCB without having to bend the legs.

When you have completed your build, do not forget to adjust the trimpots. The pedal will not work until you do. You DO NOT need to use a multimeter or oscilloscope to set any of the trimmers. The best tool for setting the trimmers is your ear.

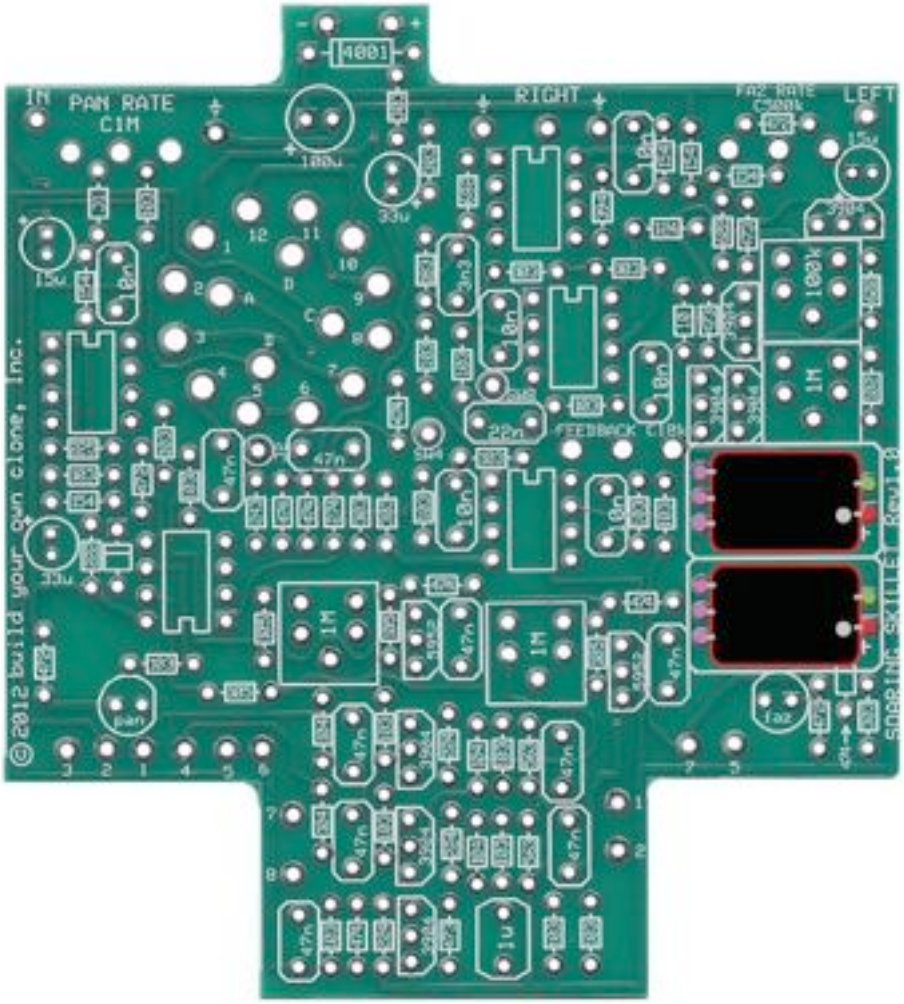
PAN LEFT and RIGHT trimmer(100k): Plug just the left output into only one amp. Do not attempt "stereo" yet. Adjust the PAN LEFT trimmer till you hear a tremolo effect (panning in mono is nothing more than tremolo). Play around with the rate knob. You want to adjust the PAN LEFT trimmer till you get the deepest tremolo effect from the left output.

Now repeat this for the PAN RIGHT trimmer using only the right output and one amp.

Once you have both sides dialed in to produce the deepest tremolo effect, you can plug into two different amps.

PHASER BIAS trimmer(100k): This trimmer sets the bias of the phaser signal. Adjust this trimpot till you get the deepest phase effect.

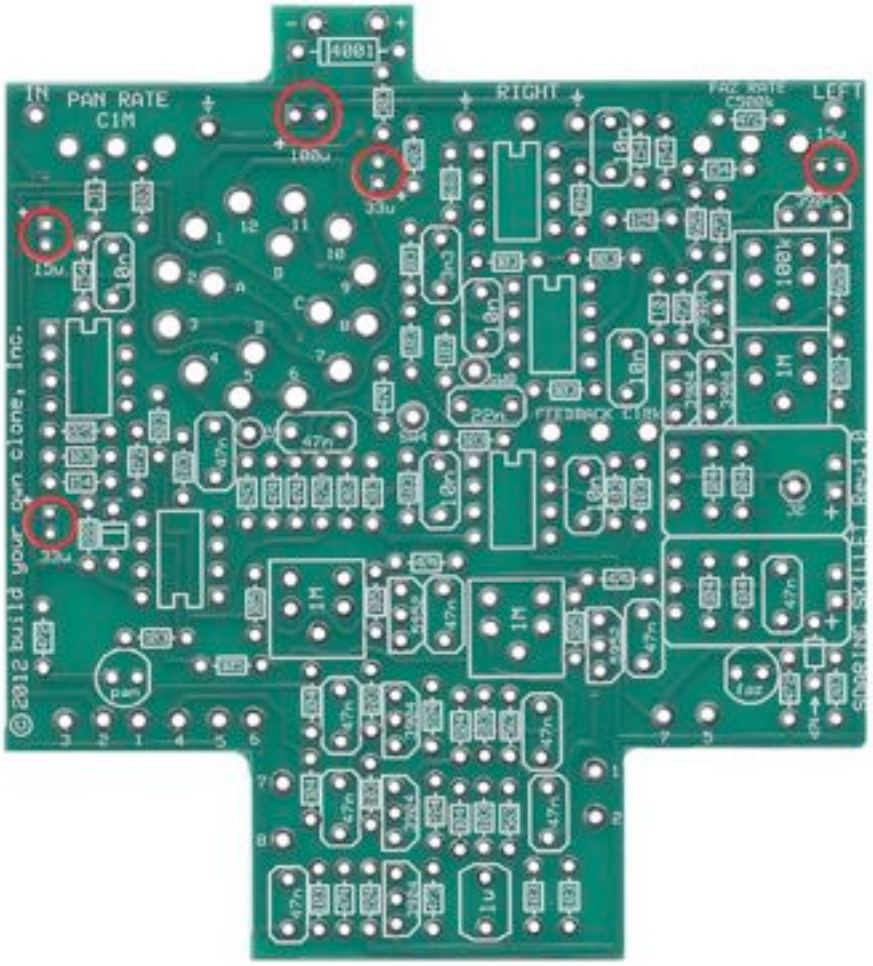
PHASER Voice trimmer(1M): This trimmer affects the voice of the phaser effect. Turning it one way will give you more of an "EW" vowel sound. Turning the other way will give you more of an "AH" vowel sound". There's no right or wrong setting for this trimmer. It's really a matter of preference.



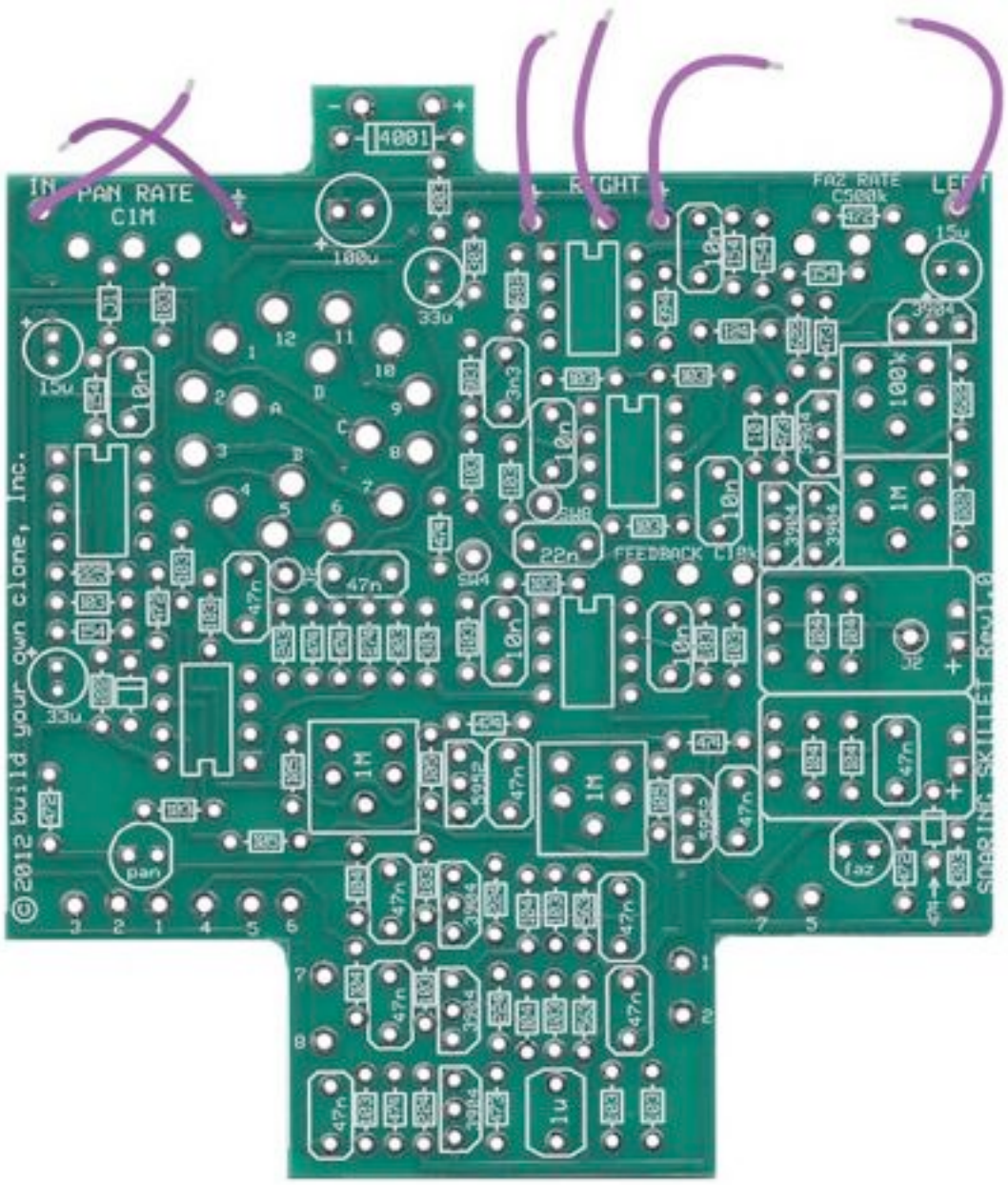
STEP 8: Add the optocouplers.

Insert the optocouplers into the PCB so that the side with the gray dot is facing up. Orient the optocoupler so that the gray dot is pointing towards the positive (+) eyelet. The lead that goes in the positive (+) eyelet highlighted in red, should be slightly longer. The shorter lead goes in the eyelet highlighted in green. Insert the 3 element leads on the opposite side of the optocoupler into the eyelets highlighted in purple. Make sure that the center element lead goes in the center element eyelet. Remember that once the

optocouplers are soldered in place, you will not have easy access to desolder the components on the backside of the PCB. Ensure everything is correct before soldering the couplers in place.

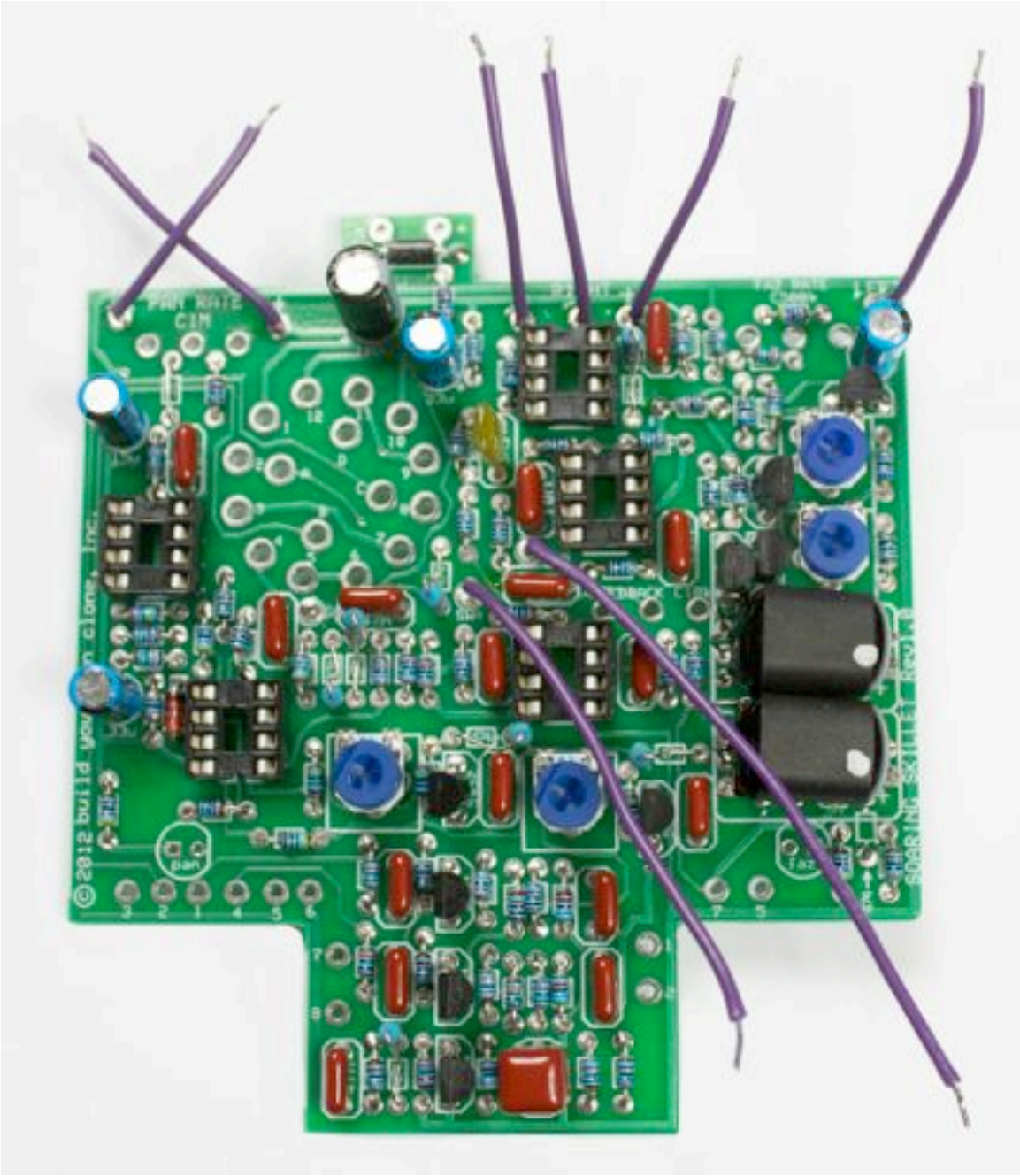


STEP 9: Add the aluminum electrolytic capacitors. These are polarized. The positive end will have a longer lead and should go in the square solder pad. The negative end will have a shorter lead with a black or white stripe running down the body of the capacitor.

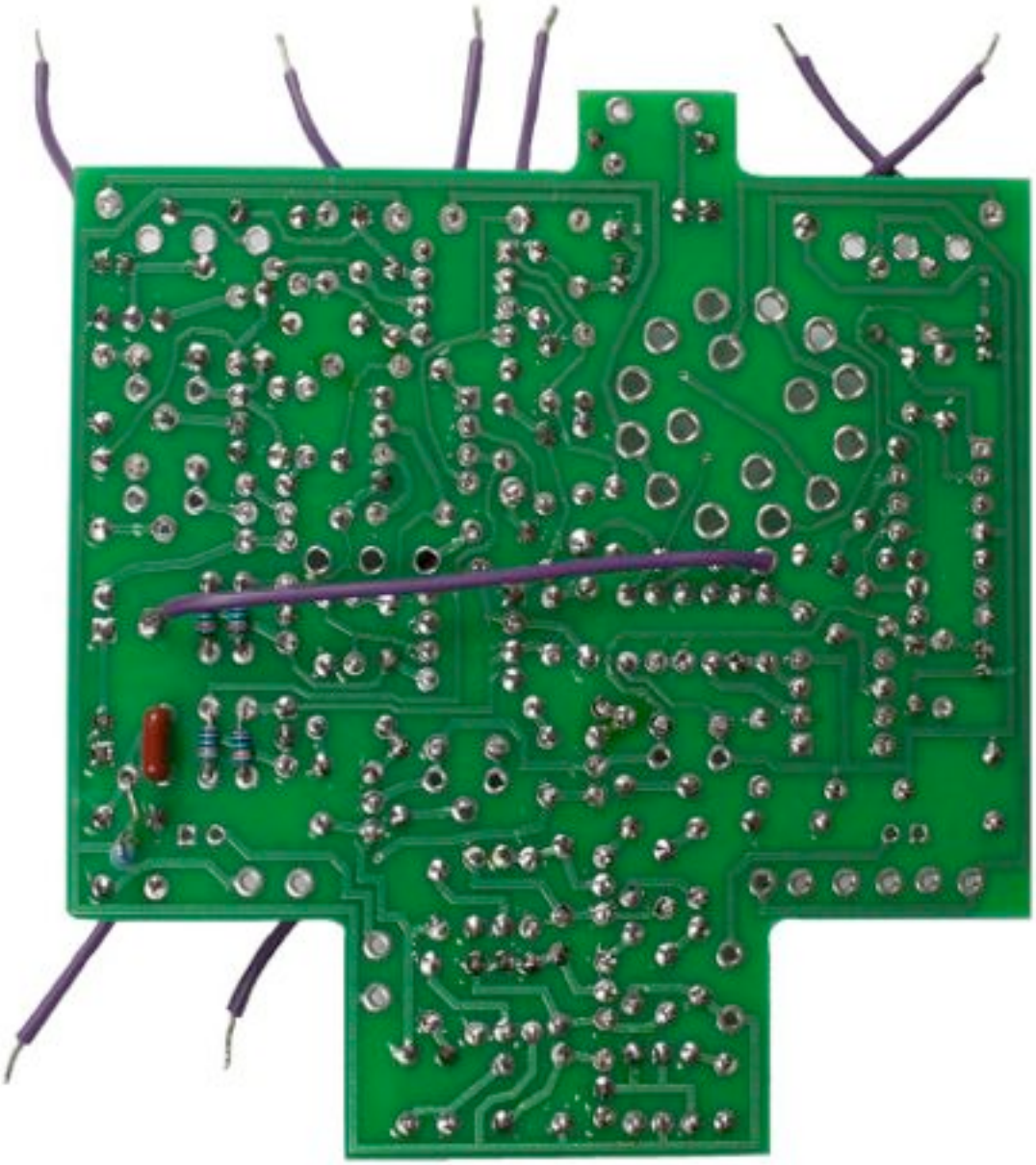


Step 10: Add the IN, the two OUT, and three GROUND wires. Start by cutting six 1.25" pieces of wire. Strip 1/4" off each end and tin the ends. Load the wires in from the top and solder on the bottom of the PCB.

By this point, your PCB should look like this:



Top (front) side



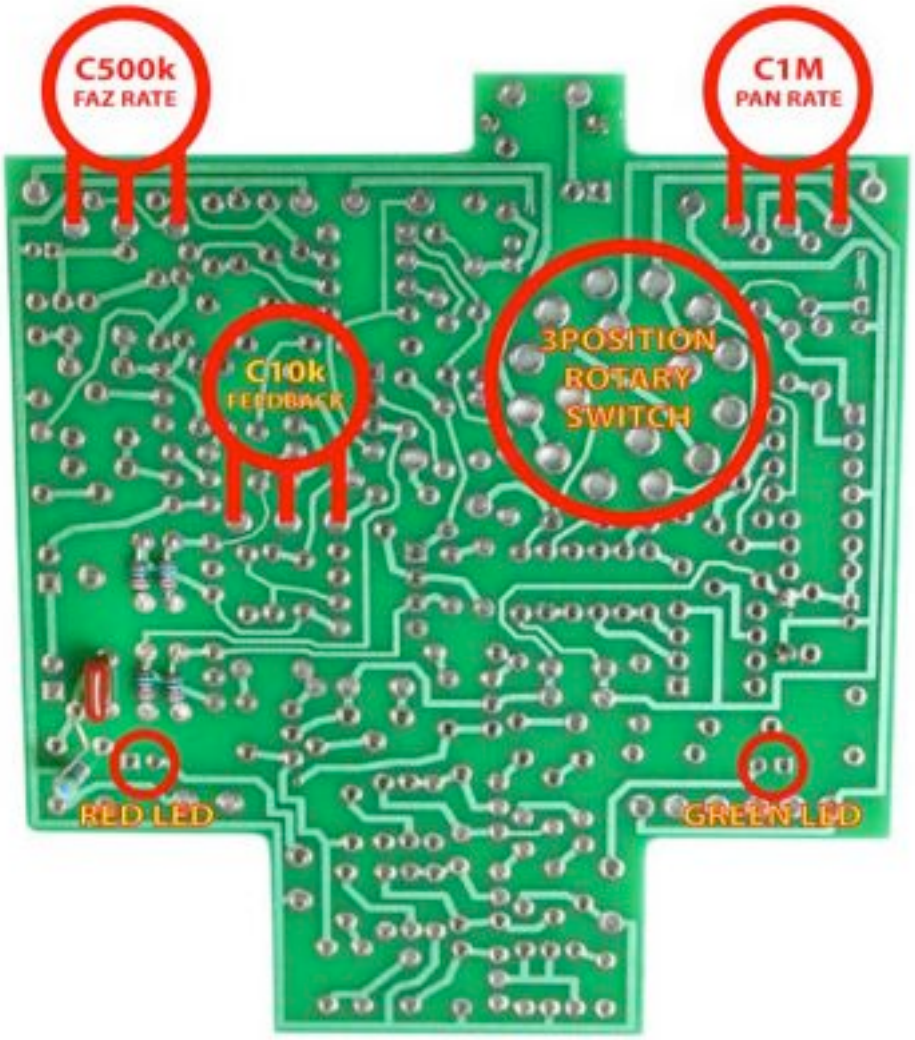
Back (solder) side

STEP 11: TAKE A BREAK!! Then come back and check your work. Are you absolutely certain you have all the resistors and capacitors in their correct spots? Did you remember to make all the jumpers? How do your solder joints look? It's better to take 15 minutes now to double check your work than spend several hours later trouble shooting.

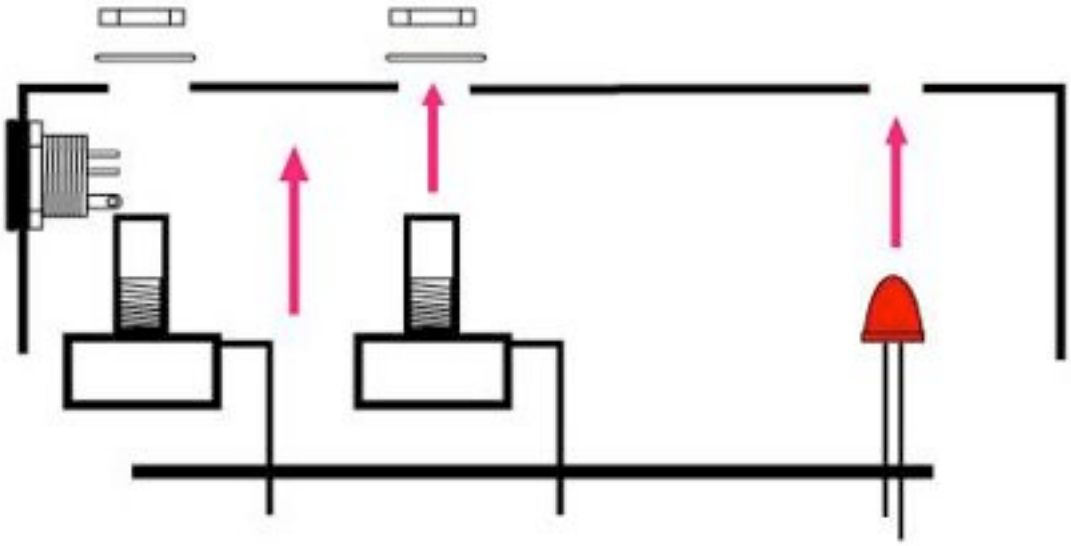
Main PCB Assembly

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

NOTE: Since there is no battery snap, you will not be connecting wire to the center tab on the adapter.



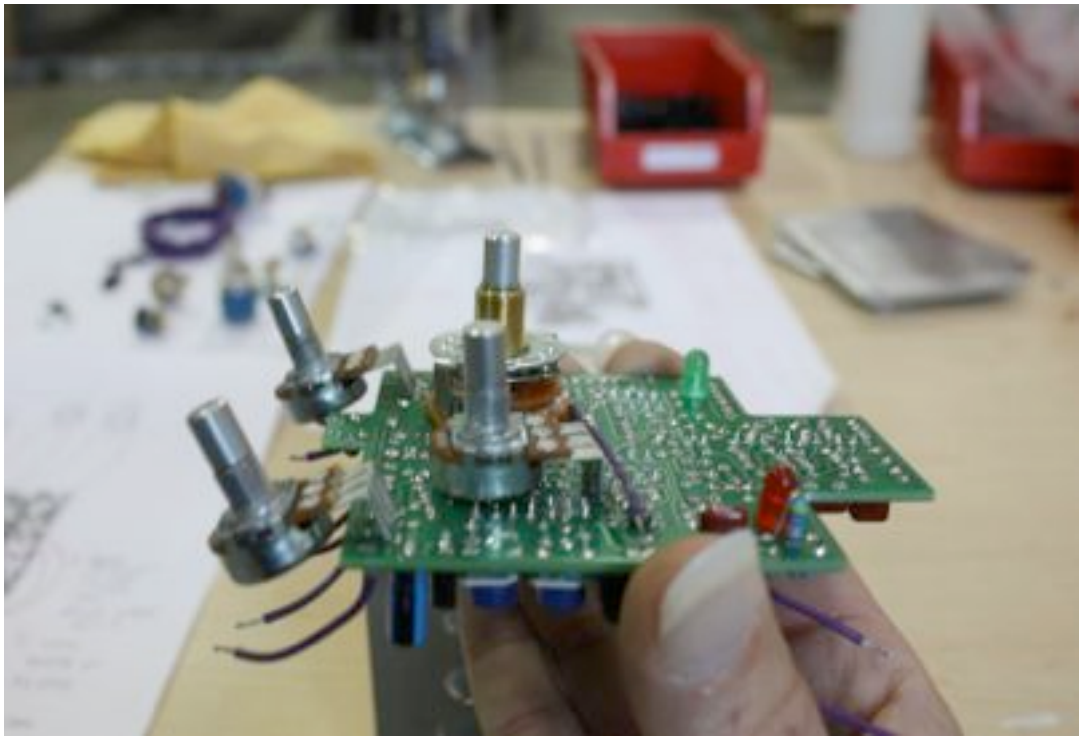
Step 2: Flip the PCB over so that the bottom or solder side is up. Insert the C10k (FEEDBACK), C500k (FAZ RATE), C1M (PAN RATE) potentiometers, the 3 position rotary switch, and the LEDs into the bottom side of the PCB. DO NOT SOLDER ANYTHING YET!!! The LEDs will have one lead that is longer than the other. The longer lead goes in the hole with the square solder pad. FYI, it does not matter which color LED goes where. If you want to switch them around (or use entirely different colors all together) go ahead.

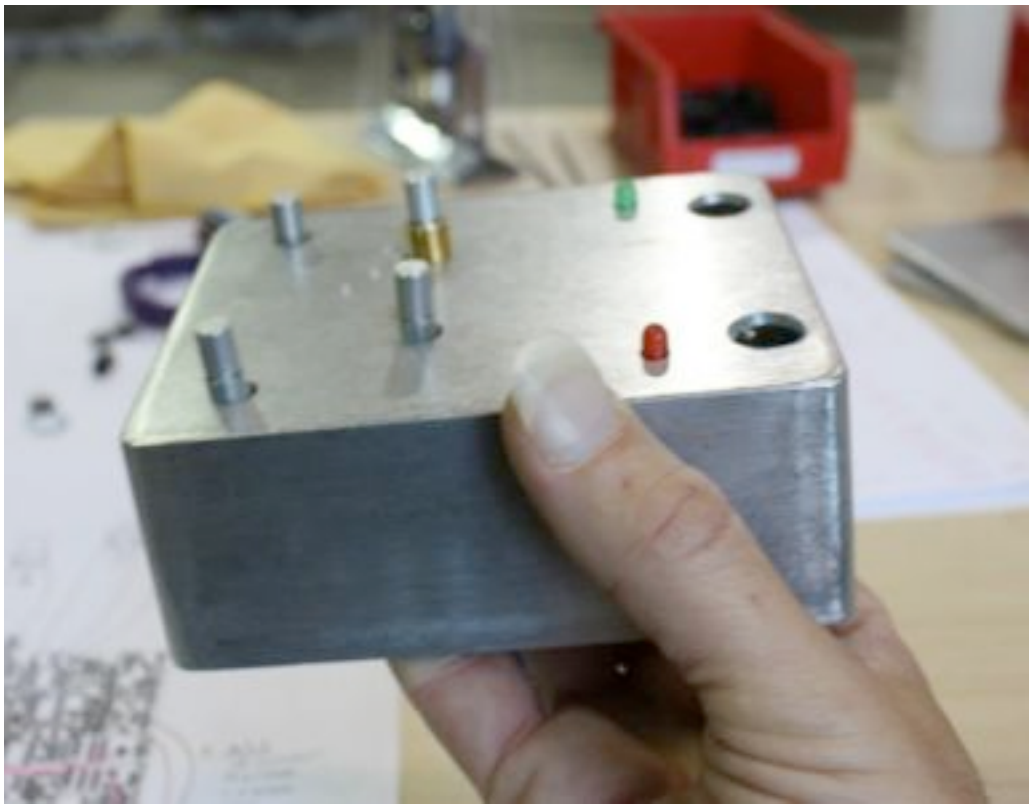


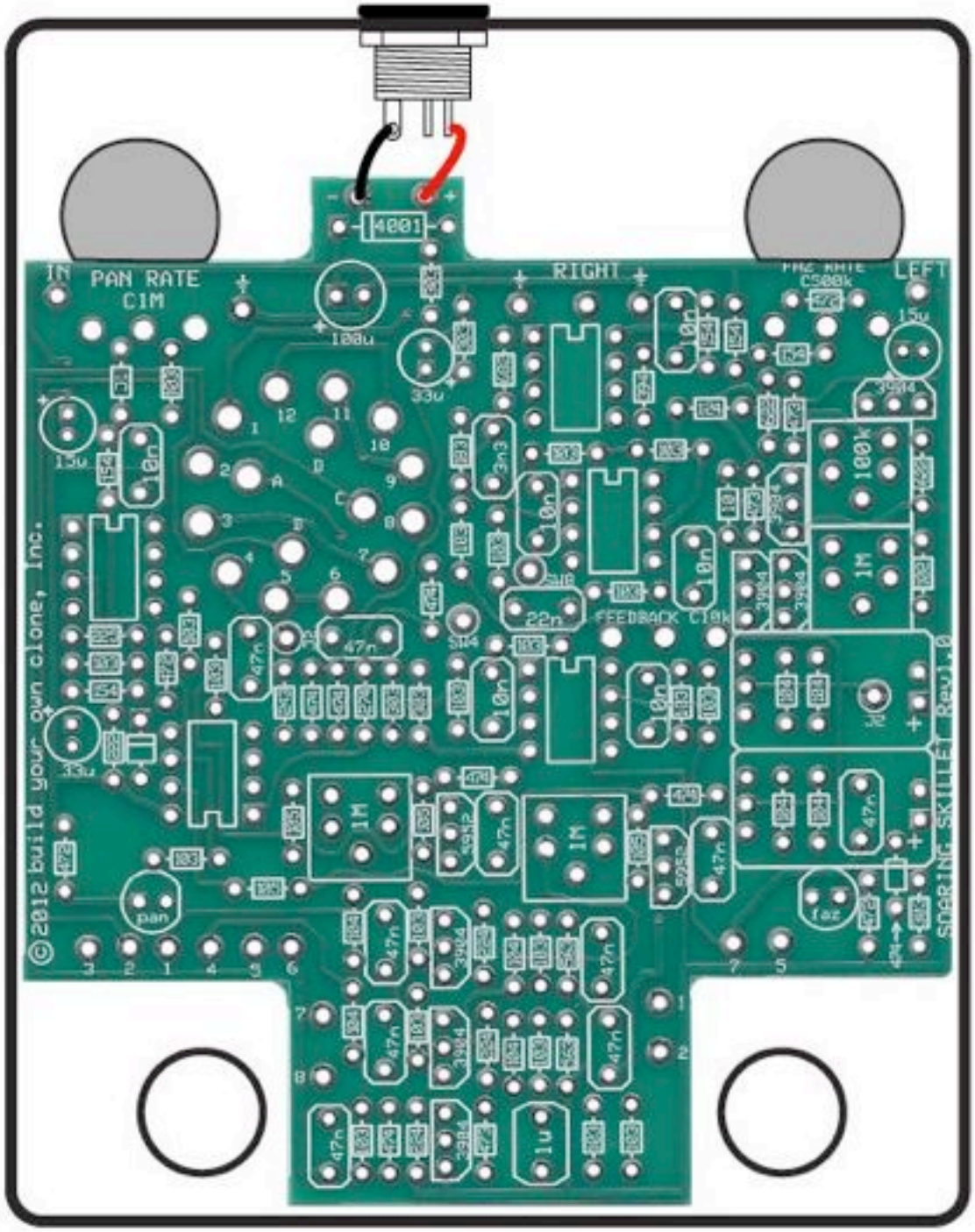
Step 3: Hold the PCB in one hand so that the component side of the PCB is in the palm of your hand and the bottom side with the pots, toggle switch and LED is facing up. Now use your other hand to guide the predrilled enclosure onto the PCB assembly so that the pots and LED all go into their respective 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 4: 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 5: 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.





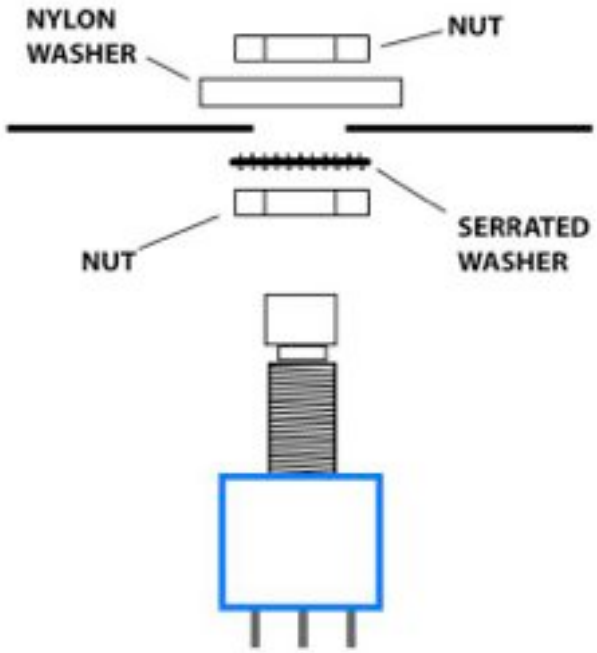


Step 6: Connect the TIP (negative) terminal of the DC adapter jack to the "-" eyelet on the PCB with 3/4 inches of hook up wire. Connect the SLEEVE of the DC adapter jack to the "+" eyelet on the far right side of the PCB with 3/4 inches of hook up wire. Remember, since there is no battery snap, you will not be connecting wire to the center tab on the adapter.

Wiring

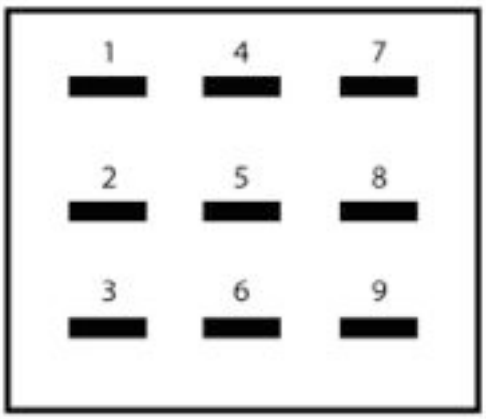
Mono Jack
Input & both Outputs

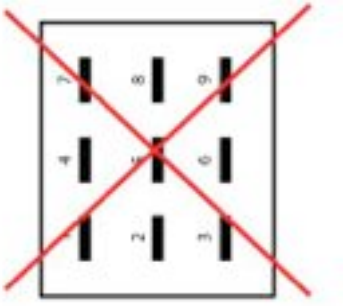
Step 1: Install the 1/4” jacks to the enclosure.



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. It may be helpful to wire both switches outside of the housing before mounting into the enclosure.

FOOT SWITCH SOLDER LUD DESIGNATIONS

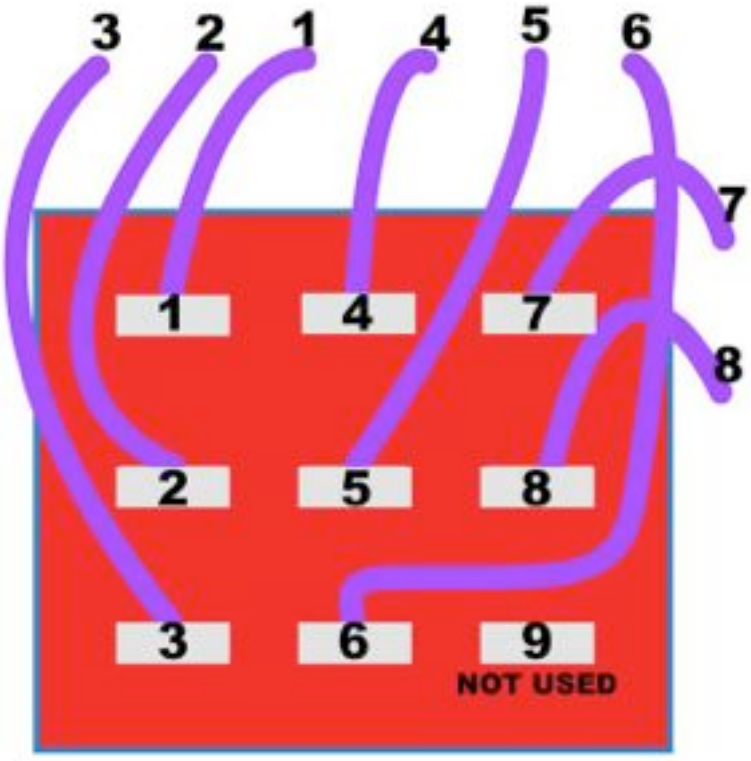




Step 3: Wiring the foot switches. NOTE: If you have built another one of our kits, the wiring of these switches is different. This is not your typical "true bypass footswitch" wiring. Follow the instructions carefully.

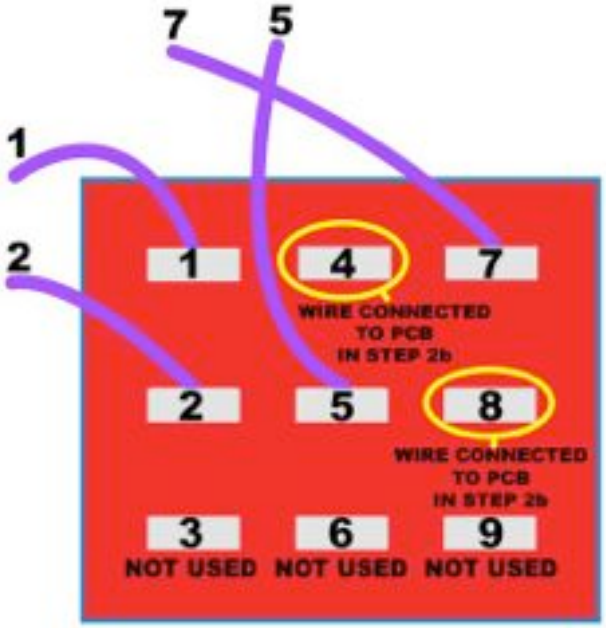
PAN FOOTSWITCH

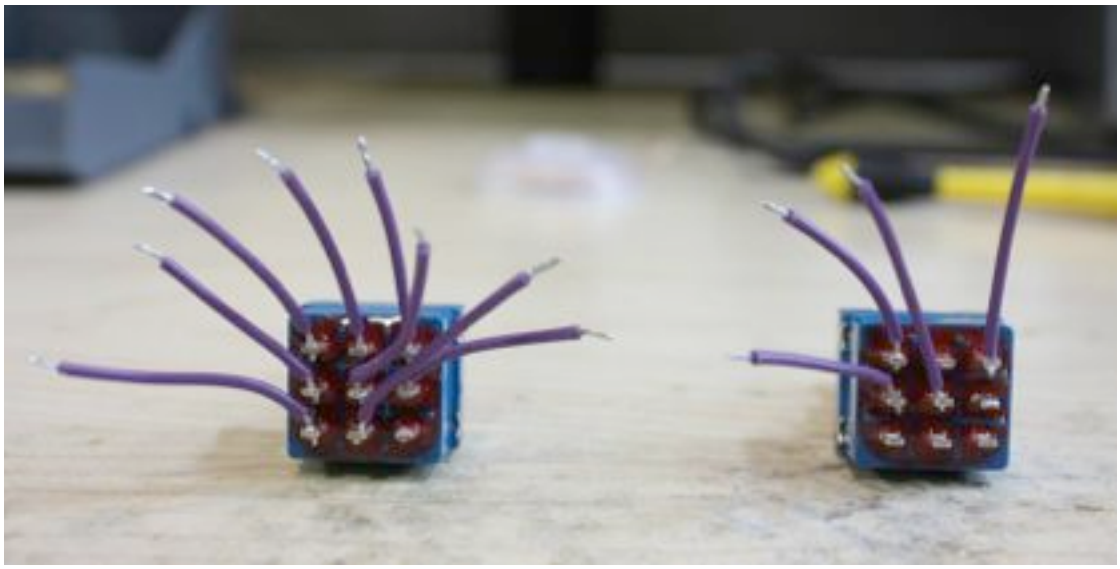
- Cut two pieces of 1.25" wire. Strip 1/8" of both ends. These will be used to connect lugs/eyelets 3 & 6.
- Cut six 1" pieces of wire. Strip 1/8" of both ends. These will be used to connect lugs/eyelets 1, 2, 4, 5,7, & 8.
- Make sure to tin both ends of each wire.
- Solder one end of the pre-cut and pre-stripped wires to the footswitch.



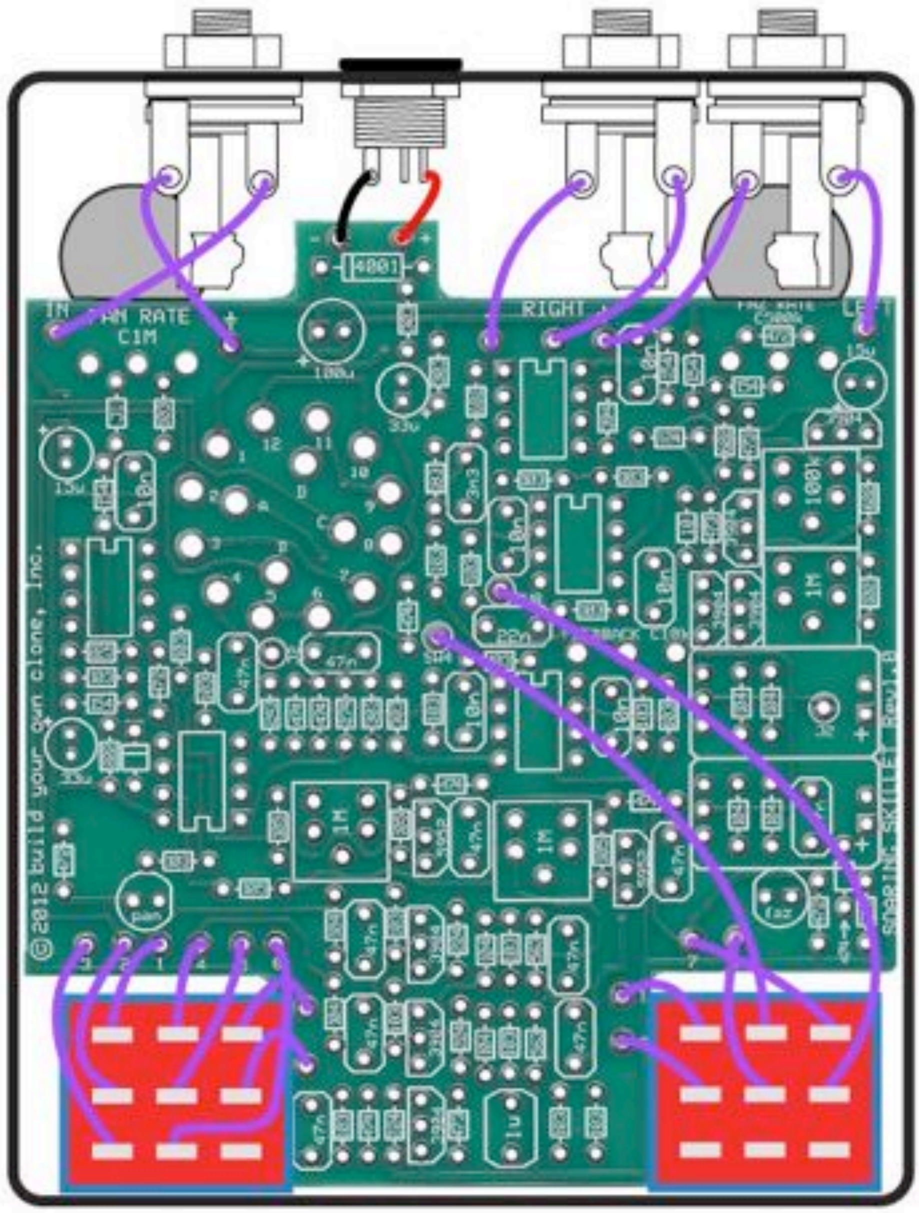
PHASE (FAZ) FOOTSWITCH

- Cut two pieces of 1.25” wire. Strip 1/8” of both ends. These will be used to connect lugs/eyelets 5 & 7.
- Cut two 1” pieces of wire. Strip 1/8” of both ends. These will be used to connect lugs/eyelets 1 & 2.
- Make sure to tin both ends of each wire.
- The wires for lugs/eyelets 4 & 8 leave open, the wires you connected to the PCB in STEP 2b – Populating the Board.
- Lugs 3, 6, & 9 are unused.
- Solder one end of the pre-cut and pre-stripped wires to the footswitch.





PAN Footswitch **FAZ Footswitch**



Step 5: Connect the pre stripped and tinned wires from the PCB to the 1/4” jacks according to the wiring diagram above.

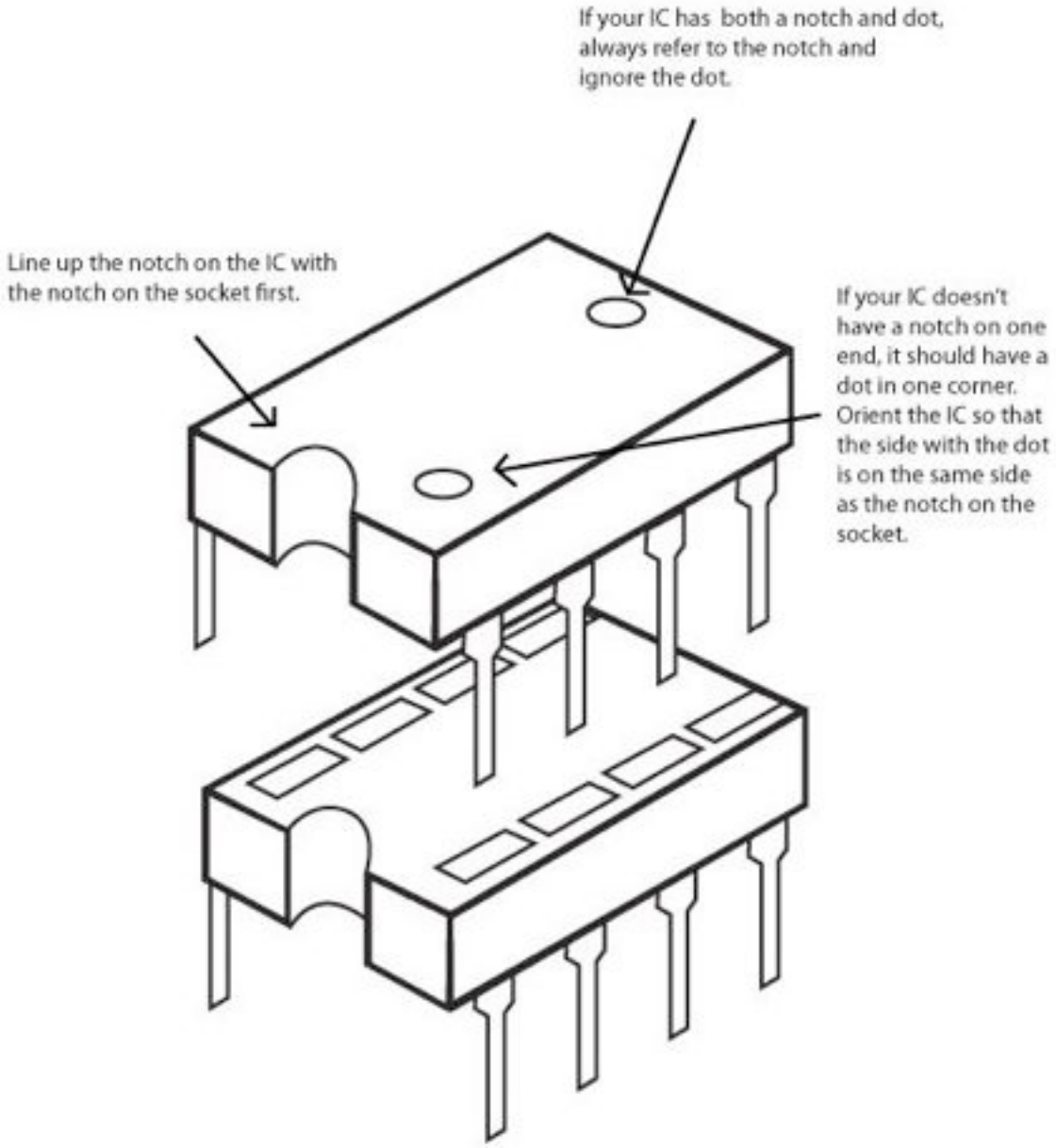
Step 6: Connect the pre stripped and tinned wires from the footswitches to

the PCB according to the wiring diagram above.

Step 7: Connect the SW4 and SW8 wires to the Phase footswitch according to the wiring diagram above.

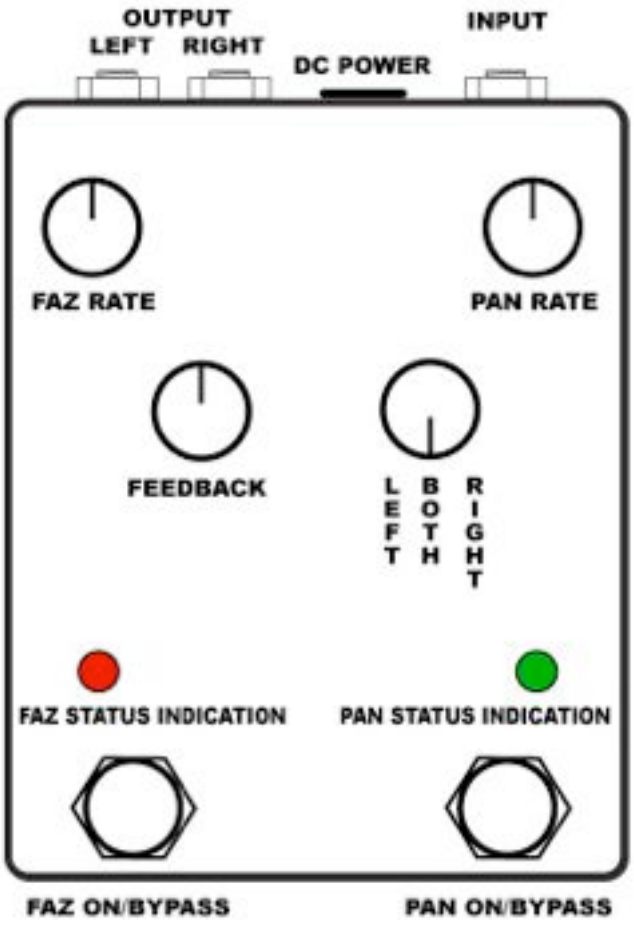
Step 8: Rotate the INPUT JACK so that the “IN” and “Ground” wires are not crossed. See the picture of the completed build on page 4.

Installing the IC's and Finishing Up



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

Operating Overview

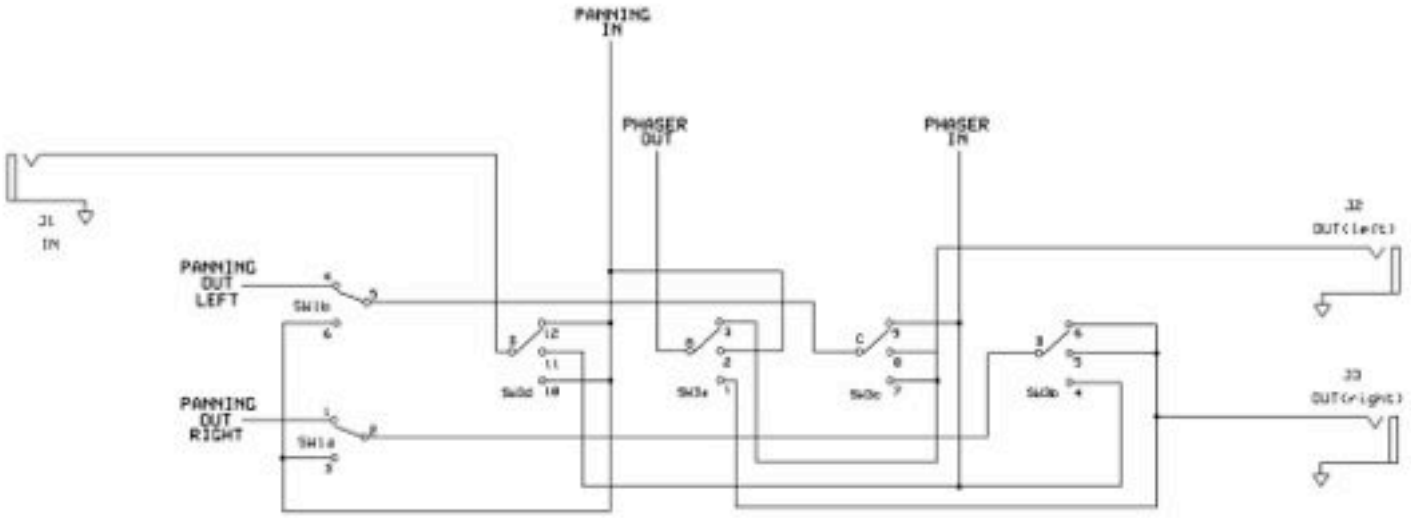


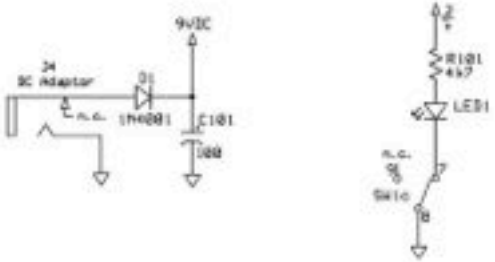
PHASE (FAZ) RATE: Controls the rate of phaser signal
PAN RATE: Controls the rate of panning
FEEDBACK: Controls how much phase shifted signal is looped back into phase stages
OUTPUT SELECTOR: Controls whether the phase signal is sent to just the LEFT OUTPUT, RIGHT OUTPUT, or BOTH.

DC power supply - Use a 2.5mm negative tip 9VDC adapter (this is your standard guitar fx adapter).

Current Draw – 8.5mA
Input Impedance - 470k ohms

Output Impedance – 28k (when only phase) – 100k ohms (when panning is on)

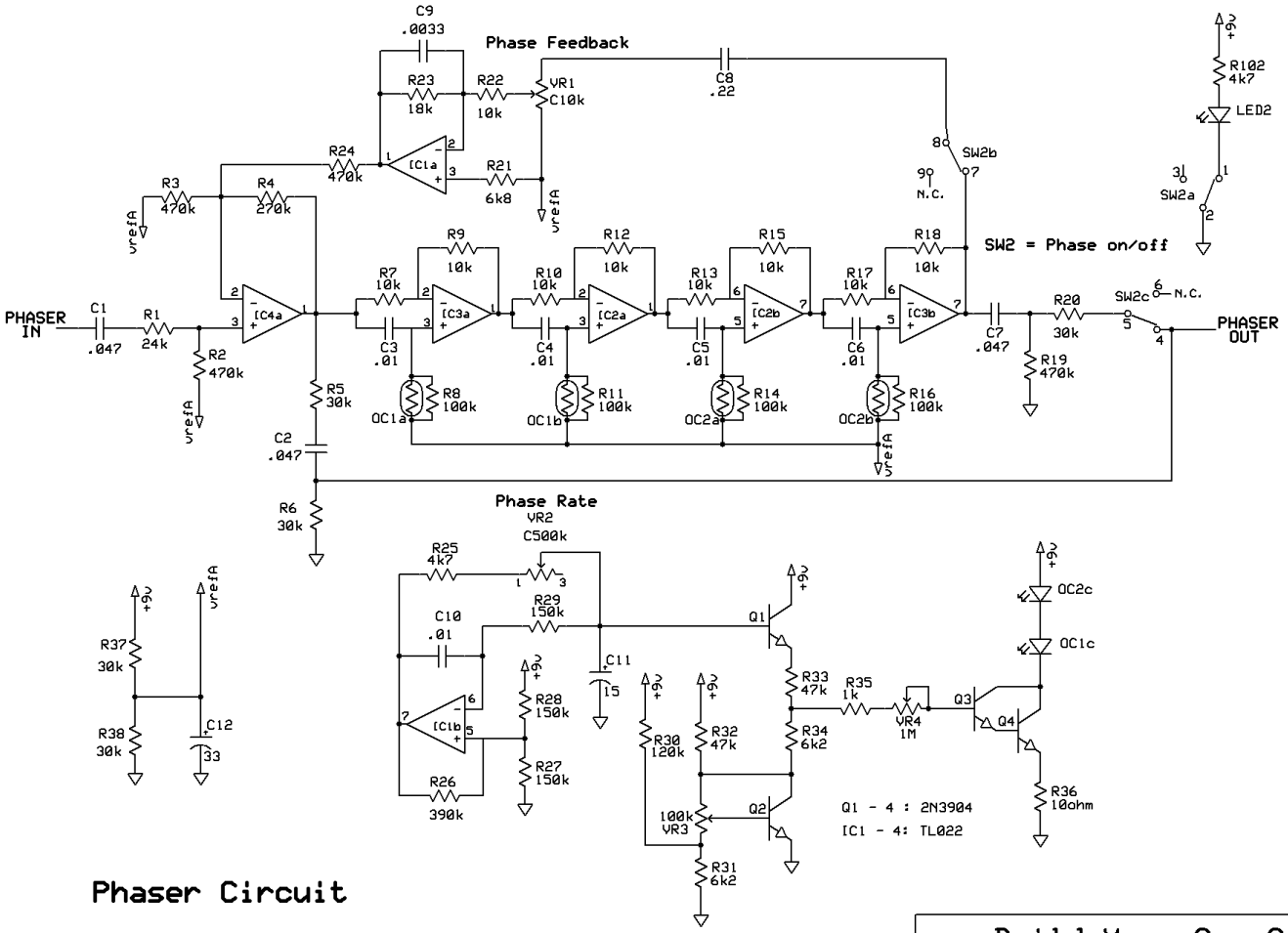




SW1 = Panning On/Off
SW3 Position 1 = Phase Right
SW3 Position 2 = Stereo
SW3 Position 3 = Phase Left

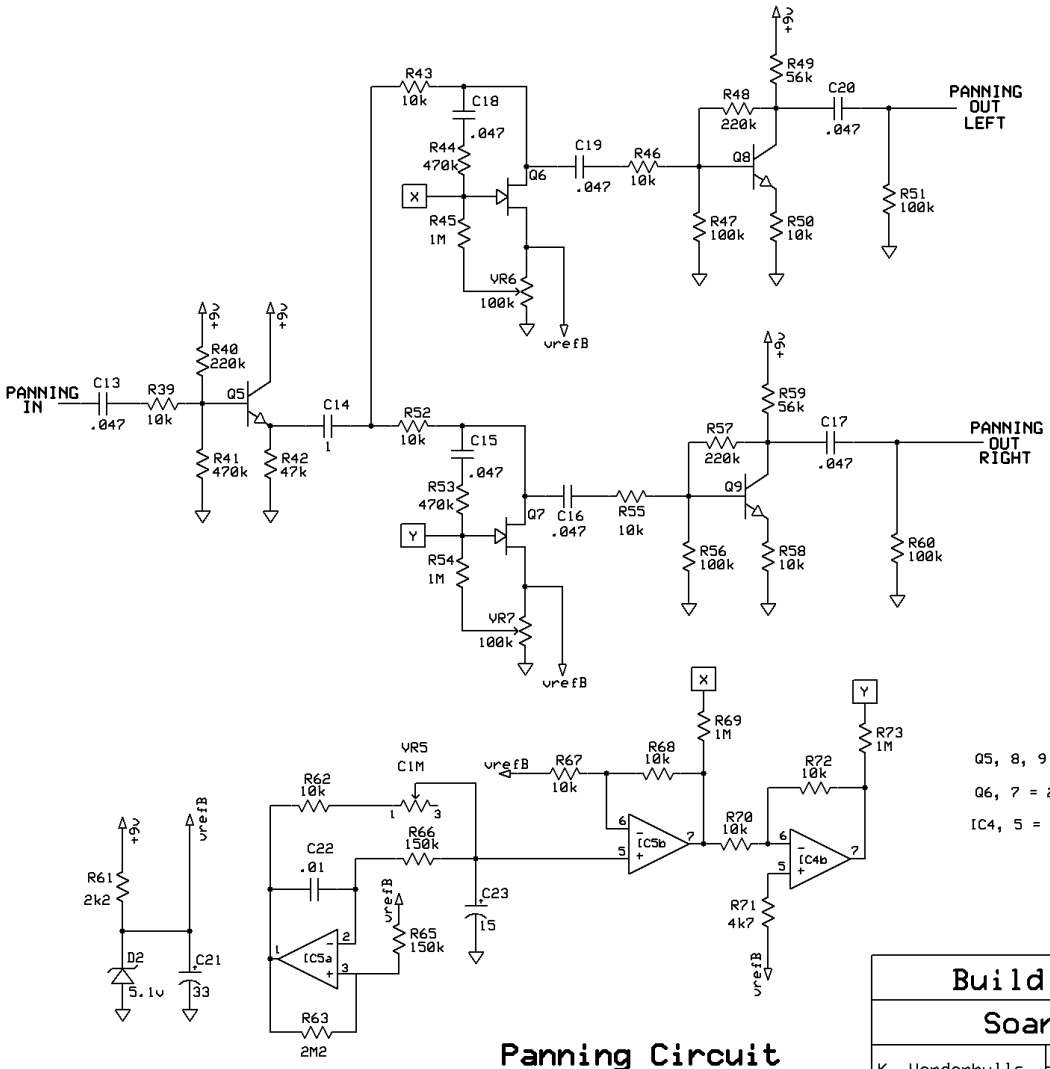
Switching/Routing Circuit

Build Your Own Clone		
Soaring Skillet		
K. Vonderhulls	Rev 1.0 7/13/2012	Page 1 of 3



Phaser Circuit

Build Your Own Clone		
Soaring Skillet		
K. Vonderhulls	Rev 1.0 7/13/2012	Page 2 of 3



Panning Circuit

Build Your Own Clone		
Soaring Skillet		
K. Vonderhulls	Rev 1.0 7/13/2012	Page 3 of 3

Click here for hi res schematic or go to http://www.buildyourownclone.com/soaringskilletscheme1.pdf
Click here for hi res schematic or go to http://www.buildyourownclone.com/soaringskilletscheme2.pdf
Click here for hi res schematic or go to http://www.buildyourownclone.com/soaringskilletscheme3.pdf

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

Copyright 2012
BYOC, Inc.